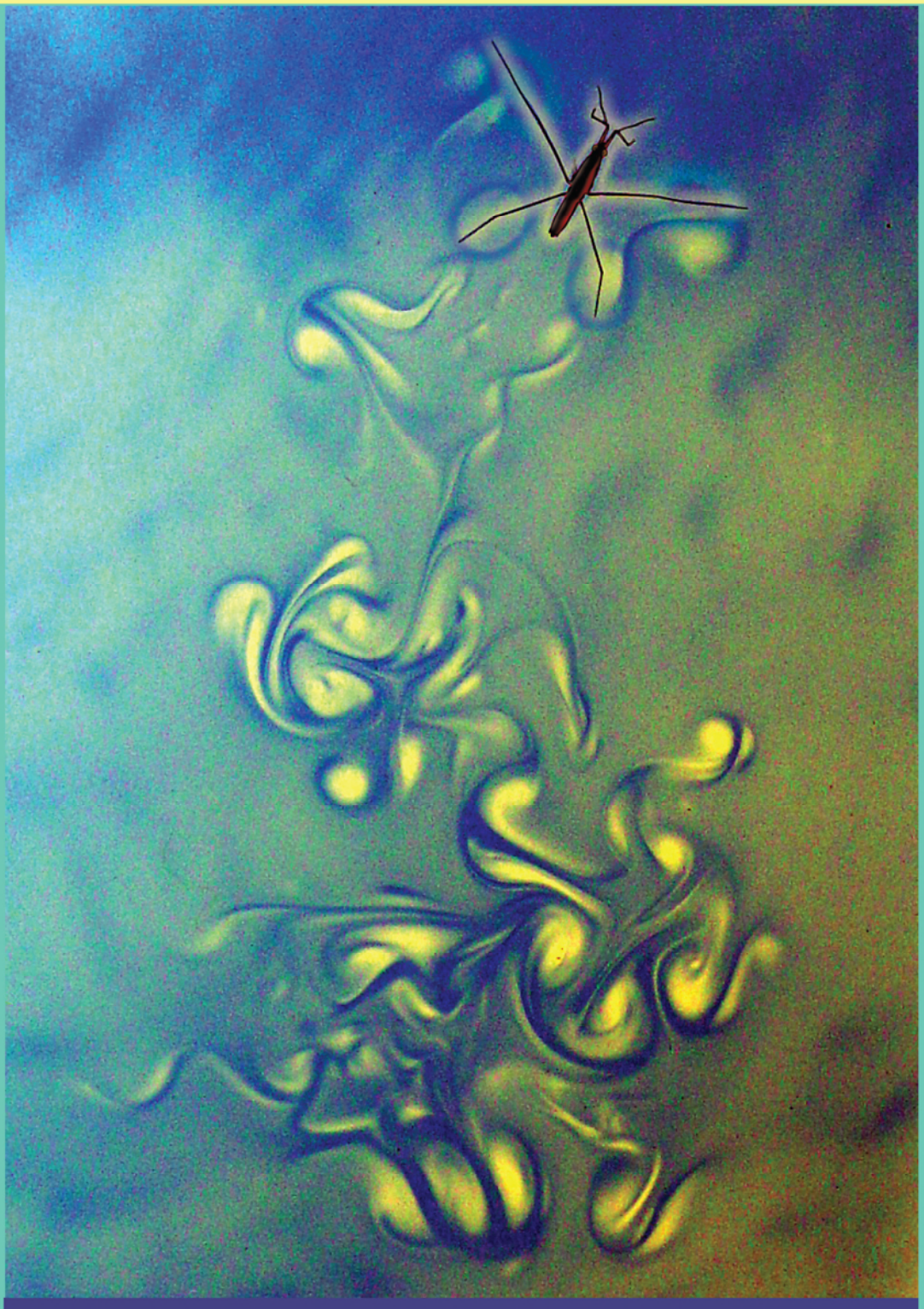
Munson • Young • Okiishi • Huebsch

# FUNDAMENTALS OF
# FLUID MECHANICS



SIXTH EDITION

CMS Cepcor EX 1046 Page 1
CMS Cepcor v. Sandvik IPR2024-01380

To Erik and all others who possess the curiosity,
patience, and desire to learn

Publisher: Don Fowley

Editorial Assistant: Mark Owens

Acquistions Editor: Jennifer Welter

Media Editor: Lauren Sapira

Marketing Manager: Christopher Ruel

Production Manager: Dorothy Sinclair

Production Editor: Sandra Dumas

Senior Designer: Madelyn Lesure

Production Management Services: Ingrao Associates/Suzanne Ingrao

**Cover**: photo shows the development of vortices which the water strider uses for moving on the surface of water by the transfer of momentum from the movement of its legs to the shed vortices (i.e. rowing). The fluid dynamics are visualized by adding thymol blue to the water surface. For more on water striders, see the following in this text: Problem 1.103 in Chapter 1; Fluids in the News titled "Walking on water" in Sect. 1.9; and Video V10.3 "Water strider" in Chapter 10. Photo courtesy of David L. Hu and John W. M. Bush, MIT.

This book was typeset in 10/12 Times Roman by Aptara®, Inc., and printed and bound by R. R. Donnelley/Jefferson City. The cover was printed by R. R. Donnelley/Jefferson City.

The paper in this book was manufactured by a mill whose forest management programs include sustained yield harvesting of its timberlands. Sustained yield harvesting principles ensure that the number of trees cut each year does not exceed the amount of new growth.

This book is printed on acid-free paper. ♾

Copyright © 2009, 2006, 2002, 1999, 1994, 1990 by John Wiley & Sons, Inc. All rights reserved.

No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, scanning or otherwise, except as permitted under Sections 107 or 108 of the 1976 United States Copyright Act, without either the prior written permission of the Publisher or authorization through payment of the appropriate per-copy fee to the Copyright Clearance Center, 222 Rosewood Drive, Danvers, MA 01923, (978) 750-8400, fax (978) 646-8600. Requests to the Publisher for permission should be addressed to the Permission Department, John Wiley & Sons, Inc., 111 River Street, Hoboken, NJ 07030-5774, (201) 748-6011, fax (201) 748-6008. To order books or for customer service call 1-800-CALL-WILEY (225-5945).

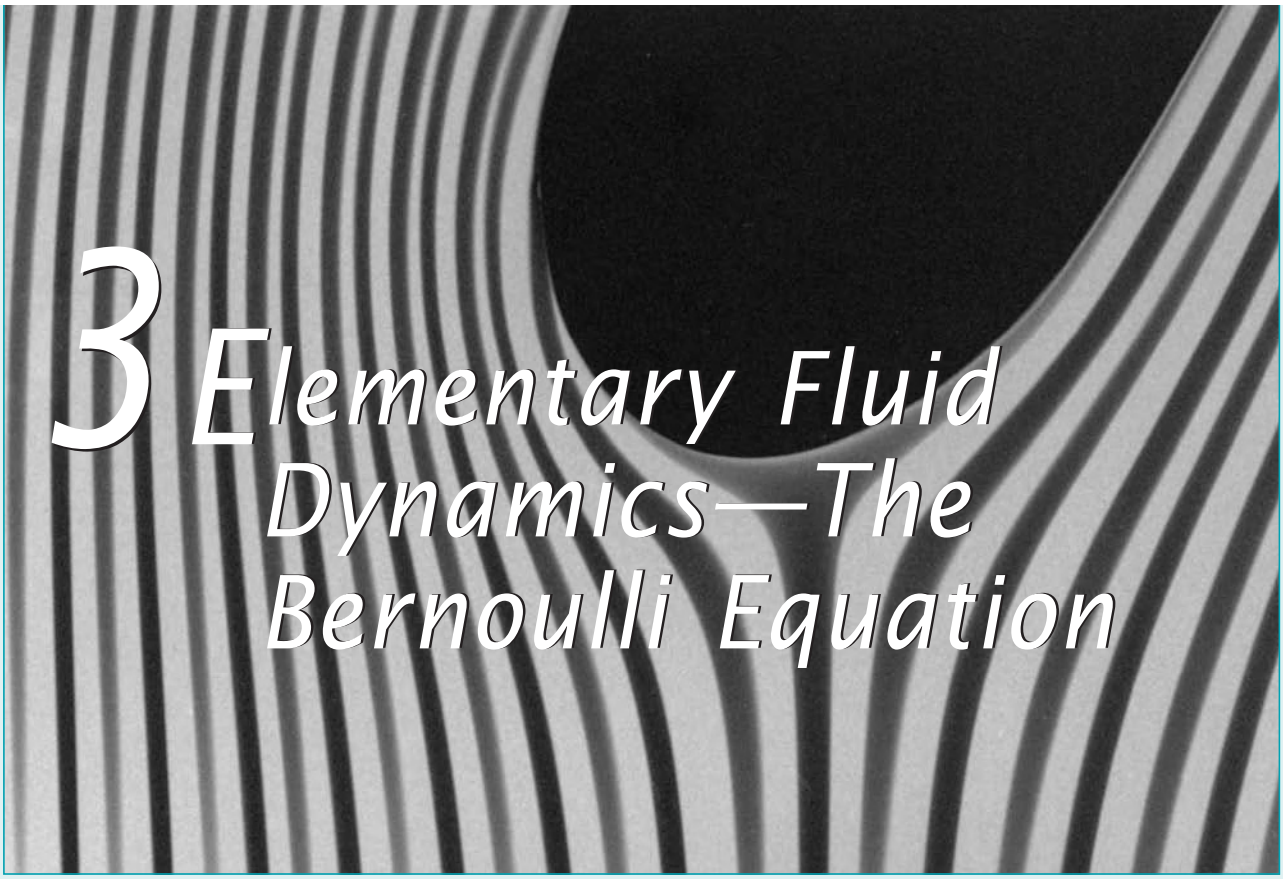
Munson, Bruce R., Young, Donald F., Okiishi, Theodore H., Huebsch, Wade W.
Fundamentals of Fluid Mechanics, Sixth Edition

ISBN 978-0470-26284-9

Printed in the United States of America.

10 9 8 7 6 5 4 3 2 1

CMS Cepcor EX 1046 Page 2
CMS Cepcor v. Sandvik IPR2024-01380



# 3 Elementary Fluid Dynamics—The Bernoulli Equation

CHAPTER OPENING PHOTO: Flow past a blunt body: On any object placed in a moving fluid there is a stagnation point on the front of the object where the velocity is zero. This location has a relatively large pressure and divides the flow field into two portions—one flowing to the left, and one flowing to the right of the body. (Dye in water.) (Photograph by B. R. Munson.)

## Learning Objectives

After completing this chapter, you should be able to:

- discuss the application of Newton's second law to fluid flows.
- explain the development, uses, and limitations of the Bernoulli equation.
- use the Bernoulli equation (stand-alone or in combination with the continuity equation) to solve simple flow problems.
- apply the concepts of static, stagnation, dynamic, and total pressures.
- calculate various flow properties using the energy and hydraulic grade lines.

*The Bernoulli equation may be the most used and abused equation in fluid mechanics.*

In this chapter we investigate some typical fluid motions (fluid dynamics) in an elementary way. We will discuss in some detail the use of Newton's second law ($\mathbf{F} = m\mathbf{a}$) as it is applied to fluid particle motion that is "ideal" in some sense. We will obtain the celebrated Bernoulli equation and apply it to various flows. Although this equation is one of the oldest in fluid mechanics and the assumptions involved in its derivation are numerous, it can be used effectively to predict and analyze a variety of flow situations. However, if the equation is applied without proper respect for its restrictions, serious errors can arise. Indeed, the Bernoulli equation is appropriately called "the most used and the most abused equation in fluid mechanics."

A thorough understanding of the elementary approach to fluid dynamics involved in this chapter will be useful on its own. It also provides a good foundation for the material in the following chapters where some of the present restrictions are removed and "more nearly exact" results are presented.



CMS Cepcor EX 1046 Page 3
CMS Cepcor v. Sandvik IPR2024-01380

## 3.1 Newton's Second Law

As a fluid particle moves from one location to another, it usually experiences an acceleration or deceleration. According to Newton's second law of motion, the net force acting on the fluid particle under consideration must equal its mass times its acceleration,

$$\mathbf{F} = m\mathbf{a}$$

In this chapter we consider the motion of inviscid fluids. That is, the fluid is assumed to have zero viscosity. If the viscosity is zero, then the thermal conductivity of the fluid is also zero and there can be no heat transfer (except by radiation).

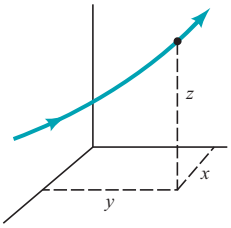
In practice there are no inviscid fluids, since every fluid supports shear stresses when it is subjected to a rate of strain displacement. For many flow situations the viscous effects are relatively small compared with other effects. As a first approximation for such cases it is often possible to ignore viscous effects. For example, often the viscous forces developed in flowing water may be several orders of magnitude smaller than forces due to other influences, such as gravity or pressure differences. For other water flow situations, however, the viscous effects may be the dominant ones. Similarly, the viscous effects associated with the flow of a gas are often negligible, although in some circumstances they are very important.

*Inviscid fluid flow is governed by pressure and gravity forces.*

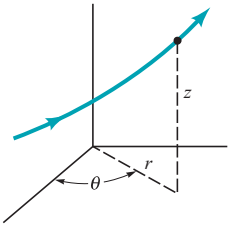
We assume that the fluid motion is governed by pressure and gravity forces only and examine Newton's second law as it applies to a fluid particle in the form:

(Net pressure force on a particle) + (net gravity force on particle) =
(particle mass) × (particle acceleration)

The results of the interaction between the pressure, gravity, and acceleration provide numerous useful applications in fluid mechanics.



Rectangular

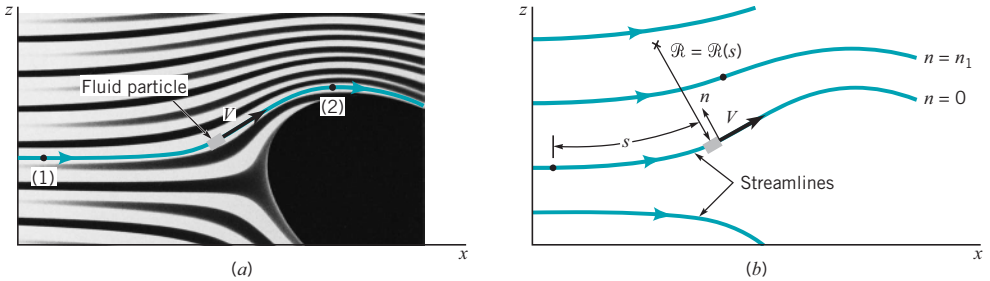


Cylindrical

To apply Newton's second law to a fluid (or any other object), we must define an appropriate coordinate system in which to describe the motion. In general the motion will be three-dimensional and unsteady so that three space coordinates and time are needed to describe it. There are numerous coordinate systems available, including the most often used rectangular $(x, y, z)$ and cylindrical $(r, \theta, z)$ systems shown by the figure in the margin. Usually the specific flow geometry dictates which system would be most appropriate.

In this chapter we will be concerned with two-dimensional motion like that confined to the $x$–$z$ plane as is shown in Fig. 3.1*a*. Clearly we could choose to describe the flow in terms of the components of acceleration and forces in the $x$ and $z$ coordinate directions. The resulting equations are frequently referred to as a two-dimensional form of the *Euler equations* of motion in rectangular Cartesian coordinates. This approach will be discussed in Chapter 6.

As is done in the study of dynamics (Ref. 1), the motion of each fluid particle is described in terms of its velocity vector, **V**, which is defined as the time rate of change of the position of the particle. The particle's velocity is a vector quantity with a magnitude (the speed, $V = |\mathbf{V}|$) and direction. As the particle moves about, it follows a particular path, the shape of which is governed by the velocity of the particle. The location of the particle along the path is a function of where the particle started at the initial time and its velocity along the path. If it is ***steady flow*** (i.e., nothing changes with time at a given location in the flow field), each successive particle that passes through a given point [such as point (1) in Fig. 3.1*a*] will follow the same path. For such cases the

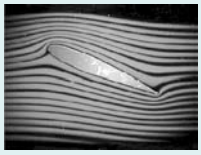


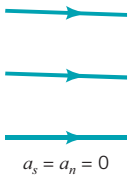

■ **FIGURE 3.1** **(*a*) Flow in the $x$–$z$ plane. (*b*) Flow in terms of streamline and normal coordinates.**

*Fluid particles accelerate normal to and along streamlines.*

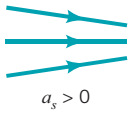


*V3.1 Streamlines past an airfoil*

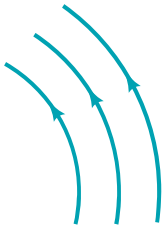




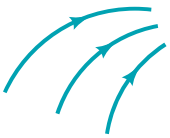
$a_s = a_n = 0$



$a_s > 0$



$a_n > 0$



$a_s > 0,\ a_n > 0$

path is a fixed line in the $x$–$z$ plane. Neighboring particles that pass on either side of point (1) follow their own paths, which may be of a different shape than the one passing through (1). The entire $x$–$z$ plane is filled with such paths.

For steady flows each particle slides along its path, and its velocity vector is everywhere tangent to the path. The lines that are tangent to the velocity vectors throughout the flow field are called ***streamlines.*** For many situations it is easiest to describe the flow in terms of the "streamline" coordinates based on the streamlines as are illustrated in Fig. 3.1*b*. The particle motion is described in terms of its distance, $s = s(t)$, along the streamline from some convenient origin and the local radius of curvature of the streamline, $\mathcal{R} = \mathcal{R}(s)$. The distance along the streamline is related to the particle's speed by $V = ds/dt$, and the radius of curvature is related to the shape of the streamline. In addition to the coordinate along the streamline, $s$, the coordinate normal to the streamline, $n$, as is shown in Fig. 3.1*b*, will be of use.

To apply Newton's second law to a particle flowing along its streamline, we must write the particle acceleration in terms of the streamline coordinates. By definition, the acceleration is the time rate of change of the velocity of the particle, $\mathbf{a} = d\mathbf{V}/dt$. For two-dimensional flow in the $x$–$z$ plane, the acceleration has two components—one along the streamline, $a_s$, the streamwise acceleration, and one normal to the streamline, $a_n$, the normal acceleration.

The streamwise acceleration results from the fact that the speed of the particle generally varies along the streamline, $V = V(s)$. For example, in Fig. 3.1*a* the speed may be 100 ft/s at point (1) and 50 ft/s at point (2). Thus, by use of the chain rule of differentiation, the $s$ component of the acceleration is given by $a_s = dV/dt = (\partial V/\partial s)(ds/dt) = (\partial V/\partial s)V$. We have used the fact that speed is the time rate of change of distance, $V = ds/dt$. Note that the streamwise acceleration is the product of the rate of change of speed with distance along the streamline, $\partial V/\partial s$, and the speed, $V$. Since $\partial V/\partial s$ can be positive, negative, or zero, the streamwise acceleration can, therefore, be positive (acceleration), negative (deceleration), or zero (constant speed).

The normal component of acceleration, the centrifugal acceleration, is given in terms of the particle speed and the radius of curvature of its path. Thus, $a_n = V^2/\mathcal{R}$, where both $V$ and $\mathcal{R}$ may vary along the streamline. These equations for the acceleration should be familiar from the study of particle motion in physics (Ref. 2) or dynamics (Ref. 1). A more complete derivation and discussion of these topics can be found in Chapter 4.

Thus, the components of acceleration in the $s$ and $n$ directions, $a_s$ and $a_n$, are given by

$$a_s = V\frac{\partial V}{\partial s}, \qquad a_n = \frac{V^2}{\mathcal{R}} \tag{3.1}$$

where $\mathcal{R}$ is the local radius of curvature of the streamline, and $s$ is the distance measured along the streamline from some arbitrary initial point. In general there is acceleration along the streamline (because the particle speed changes along its path, $\partial V/\partial s \neq 0$) and acceleration normal to the streamline (because the particle does not flow in a straight line, $\mathcal{R} \neq \infty$). Various flows and the accelerations associated with them are shown in the figure in the margin. As discussed in Section 3.6.2, for incompressible flow the velocity is inversely proportional to the streamline spacing. Hence, converging streamlines produce positive streamwise acceleration. To produce this acceleration there must be a net, nonzero force on the fluid particle.

To determine the forces necessary to produce a given flow (or conversely, what flow results from a given set of forces), we consider the free-body diagram of a small fluid particle as is shown in Fig. 3.2. The particle of interest is removed from its surroundings, and the reactions of the

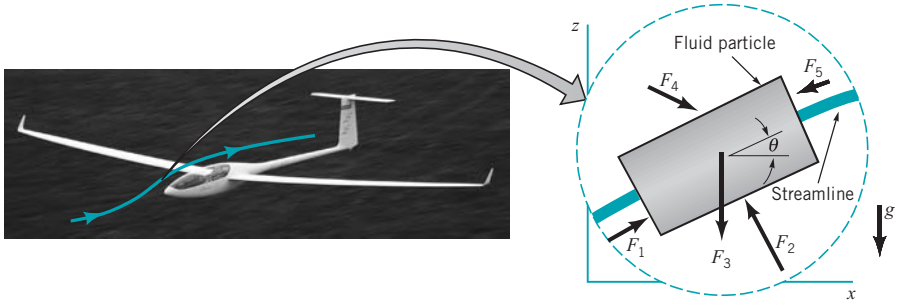



**FIGURE 3.2** **Isolation of a small fluid particle in a flow field. (Photo courtesy of Diana Sailplanes.)**

CMS Cepcor EX 1046 Page 5
CMS Cepcor v. Sandvik IPR2024-01380

surroundings on the particle are indicated by the appropriate forces present, $\mathbf{F}_1$, $\mathbf{F}_2$, and so forth. For the present case, the important forces are assumed to be gravity and pressure. Other forces, such as viscous forces and surface tension effects, are assumed negligible. The acceleration of gravity, $g$, is assumed to be constant and acts vertically, in the negative $z$ direction, at an angle $\theta$ relative to the normal to the streamline.

## 3.2 F = *m*a along a Streamline

Consider the small fluid particle of size $\delta s$ by $\delta n$ in the plane of the figure and $\delta y$ normal to the figure as shown in the free-body diagram of Fig. 3.3. Unit vectors along and normal to the streamline are denoted by $\hat{\mathbf{s}}$ and $\hat{\mathbf{n}}$, respectively. For steady flow, the component of Newton's second law along the streamline direction, $s$, can be written as

$$\sum \delta F_s = \delta m\, a_s = \delta m\, V \frac{\partial V}{\partial s} = \rho\, \delta \forall\, V \frac{\partial V}{\partial s} \tag{3.2}$$

where $\sum \delta F_s$ represents the sum of the $s$ components of all the forces acting on the particle, which has mass $\delta m = \rho\, \delta \forall$, and $V\, \partial V/\partial s$ is the acceleration in the $s$ direction. Here, $\delta \forall = \delta s\, \delta n\, \delta y$ is the particle volume. Equation 3.2 is valid for both compressible and incompressible fluids. That is, the density need not be constant throughout the flow field.

The gravity force (weight) on the particle can be written as $\delta \mathcal{W} = \gamma\, \delta \forall$, where $\gamma = \rho g$ is the specific weight of the fluid (lb/ft$^3$ or N/m$^3$). Hence, the component of the weight force in the direction of the streamline is

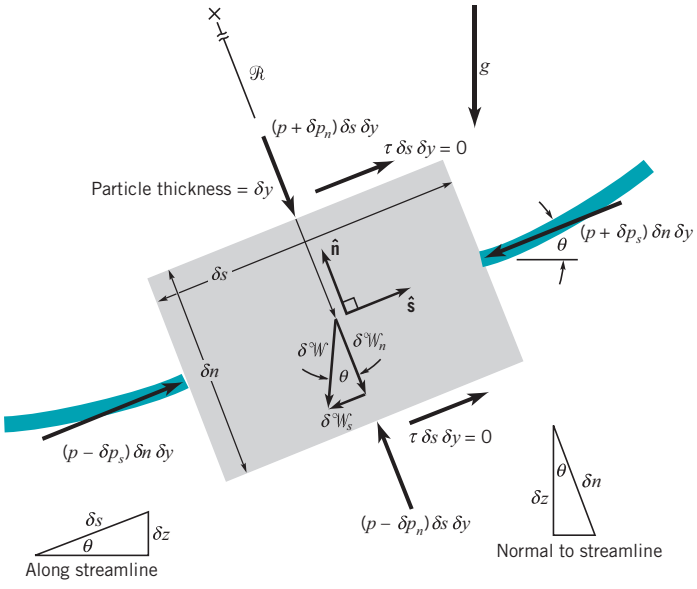
$$\delta \mathcal{W}_s = -\delta \mathcal{W} \sin\theta = -\gamma\, \delta \forall \sin\theta$$

*In a flowing fluid the pressure varies from one location to another.*

If the streamline is horizontal at the point of interest, then $\theta = 0$, and there is no component of particle weight along the streamline to contribute to its acceleration in that direction.

As is indicated in Chapter 2, the pressure is not constant throughout a stationary fluid $(\nabla p \neq 0)$ because of the fluid weight. Likewise, in a flowing fluid the pressure is usually not constant. In general, for steady flow, $p = p(s, n)$. If the pressure at the center of the particle shown in Fig. 3.3 is denoted as $p$, then its average value on the two end faces that are perpendicular to the streamline are $p + \delta p_s$ and $p - \delta p_s$. Since the particle is "small," we can use a one-term Taylor series expansion for the pressure field (as was done in Chapter 2 for the pressure forces in static fluids) to obtain

$$\delta p_s \approx \frac{\partial p}{\partial s}\frac{\delta s}{2}$$




**■ FIGURE 3.3** **Free-body diagram of a fluid particle for which the important forces are those due to pressure and gravity.**

*The net pressure force on a particle is determined by the pressure gradient.*

Thus, if $\delta F_{ps}$ is the net pressure force on the particle in the streamline direction, it follows that

$$\delta F_{ps} = (p - \delta p_s)\,\delta n\,\delta y - (p + \delta p_s)\,\delta n\,\delta y = -2\,\delta p_s\,\delta n\,\delta y$$
$$= -\frac{\partial p}{\partial s}\delta s\,\delta n\,\delta y = -\frac{\partial p}{\partial s}\delta \forall$$

Note that the actual level of the pressure, $p$, is not important. What produces a net pressure force is the fact that the pressure is not constant throughout the fluid. The nonzero pressure gradient, $\nabla p = \partial p/\partial s\,\hat{\mathbf{s}} + \partial p/\partial n\,\hat{\mathbf{n}}$, is what provides a net pressure force on the particle. Viscous forces, represented by $\tau\,\delta s\,\delta y$, are zero, since the fluid is inviscid.

Thus, the net force acting in the streamline direction on the particle shown in Fig. 3.3 is given by

$$\sum \delta F_s = \delta \mathcal{W}_s + \delta F_{ps} = \left(-\gamma \sin\theta - \frac{\partial p}{\partial s}\right)\delta \forall \tag{3.3}$$

By combining Eqs. 3.2 and 3.3, we obtain the following equation of motion along the streamline direction:

$$-\gamma \sin\theta - \frac{\partial p}{\partial s} = \rho V \frac{\partial V}{\partial s} = \rho a_s \tag{3.4}$$

We have divided out the common particle volume factor, $\delta \forall$, that appears in both the force and the acceleration portions of the equation. This is a representation of the fact that it is the fluid density (mass per unit volume), not the mass, per se, of the fluid particle that is important.

The physical interpretation of Eq. 3.4 is that a change in fluid particle speed is accomplished by the appropriate combination of pressure gradient and particle weight along the streamline. For fluid static situations this balance between pressure and gravity forces is such that no change in particle speed is produced—the right-hand side of Eq. 3.4 is zero, and the particle remains stationary. In a flowing fluid the pressure and weight forces do not necessarily balance—the force unbalance provides the appropriate acceleration and, hence, particle motion.

## EXAMPLE 3.1 Pressure Variation along a Streamline

**GIVEN** Consider the inviscid, incompressible, steady flow along the horizontal streamline $A$–$B$ in front of the sphere of radius $a$, as shown in Fig. E3.1*a*. From a more advanced theory of flow past a sphere, the fluid velocity along this streamline is

$$V = V_0\left(1 + \frac{a^3}{x^3}\right)$$

as shown in Fig. E3.1*b*.

**FIND** Determine the pressure variation along the streamline from point $A$ far in front of the sphere ($x_A = -\infty$ and $V_A = V_0$) to point $B$ on the sphere ($x_B = -a$ and $V_B = 0$).

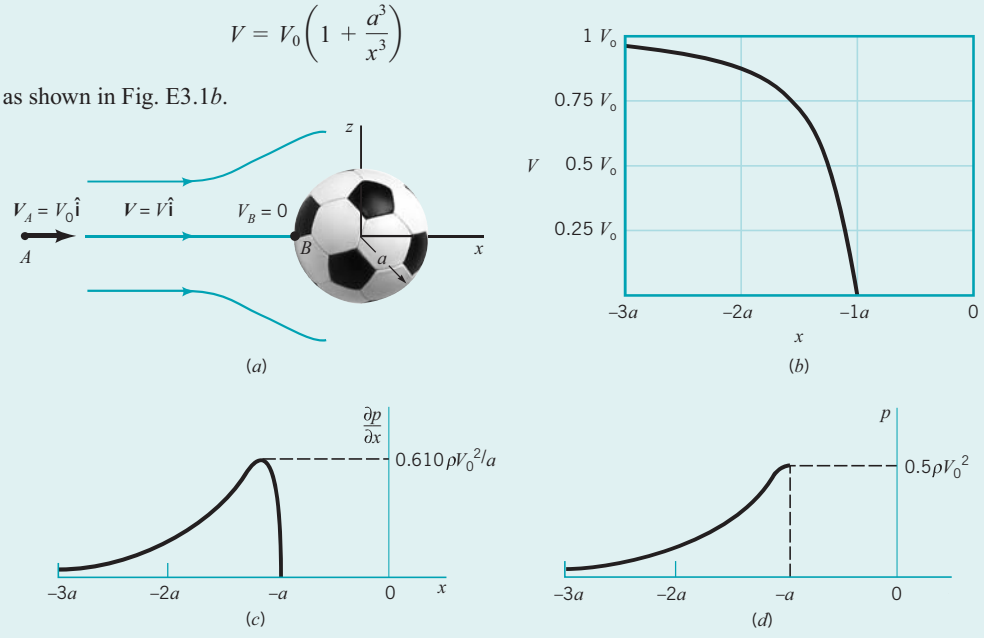



**FIGURE E3.1**

## SOLUTION

Since the flow is steady and inviscid, Eq. 3.4 is valid. In addition, since the streamline is horizontal, $\sin\theta = \sin 0° = 0$ and the equation of motion along the streamline reduces to

$$\frac{\partial p}{\partial s} = -\rho V \frac{\partial V}{\partial s} \tag{1}$$

With the given velocity variation along the streamline, the acceleration term is

$$V\frac{\partial V}{\partial s} = V\frac{\partial V}{\partial x} = V_0\left(1 + \frac{a^3}{x^3}\right)\left(-\frac{3V_0a^3}{x^4}\right)$$
$$= -3V_0^2\left(1 + \frac{a^3}{x^3}\right)\frac{a^3}{x^4}$$

where we have replaced $s$ by $x$ since the two coordinates are identical (within an additive constant) along streamline $A$–$B$. It follows that $V\,\partial V/\partial s < 0$ along the streamline. The fluid slows down from $V_0$ far ahead of the sphere to zero velocity on the "nose" of the sphere ($x = -a$).

Thus, according to Eq. 1, to produce the given motion the pressure gradient along the streamline is

$$\frac{\partial p}{\partial x} = \frac{3\rho a^3 V_0^2(1 + a^3/x^3)}{x^4} \tag{2}$$

This variation is indicated in Fig. E3.1*c*. It is seen that the pressure increases in the direction of flow ($\partial p/\partial x > 0$) from point $A$ to point $B$. The maximum pressure gradient ($0.610\,\rho V_0^2/a$) occurs just slightly ahead of the sphere ($x = -1.205a$). It is the pressure gradient that slows the fluid down from $V_A = V_0$ to $V_B = 0$ as shown in Fig. E3.1*b*.

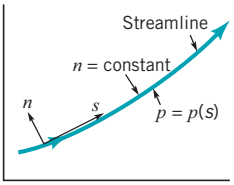
The pressure distribution along the streamline can be obtained by integrating Eq. 2 from $p = 0$ (gage) at $x = -\infty$ to pressure $p$ at location $x$. The result, plotted in Fig. E3.1*d*, is

$$p = -\rho V_0^2\left[\left(\frac{a}{x}\right)^3 + \frac{(a/x)^6}{2}\right] \qquad \textbf{(Ans)}$$

**COMMENT** The pressure at $B$, a stagnation point since $V_B = 0$, is the highest pressure along the streamline ($p_B = \rho V_0^2/2$). As shown in Chapter 9, this excess pressure on the front of the sphere (i.e., $p_B > 0$) contributes to the net drag force on the sphere. Note that the pressure gradient and pressure are directly proportional to the density of the fluid, a representation of the fact that the fluid inertia is proportional to its mass.

## Fluids in the News

**Incorrect raindrop shape** The incorrect representation that raindrops are teardrop shaped is found nearly everywhere—from children's books, to weather maps on the *Weather Channel*. About the only time raindrops possess the typical teardrop shape is when they run down a windowpane. The actual shape of a falling raindrop is a function of the size of the drop and results from a balance between surface tension forces and the air pressure exerted on the falling drop. Small drops with a radius less than about 0.5 mm are spherical shaped because the surface tension effect (which is inversely proportional to drop size) wins over the increased pressure, $\rho V_0^2/2$, caused by the motion of the drop and exerted on its bottom. With increasing size, the drops fall faster and the increased pressure causes the drops to flatten. A 2-mm drop, for example, is flattened into a hamburger bun shape. Slightly larger drops are actually concave on the bottom. When the radius is greater than about 4 mm, the depression of the bottom increases and the drop takes on the form of an inverted bag with an annular ring of water around its base. This ring finally breaks up into smaller drops. (See Problem 3.28.)




Equation 3.4 can be rearranged and integrated as follows. First, we note from Fig. 3.3 that along the streamline $\sin\theta = dz/ds$. Also, we can write $V\,dV/ds = \frac{1}{2}d(V^2)/ds$. Finally, along the streamline the value of $n$ is constant ($dn = 0$) so that $dp = (\partial p/\partial s)\,ds + (\partial p/\partial n)\,dn = (\partial p/\partial s)\,ds$. Hence, as indicated by the figure in the margin, along a given streamline $p(s, n) = p(s)$ and $\partial p/\partial s = dp/ds$. These ideas combined with Eq. 3.4 give the following result valid along a streamline

$$-\gamma\frac{dz}{ds} - \frac{dp}{ds} = \frac{1}{2}\rho\frac{d(V^2)}{ds}$$

This simplifies to

$$dp + \frac{1}{2}\rho d(V^2) + \gamma\,dz = 0 \qquad \text{(along a streamline)} \tag{3.5}$$

*For steady, inviscid flow the sum of certain pressure, velocity, and elevation effects is constant along a streamline.*

which, for constant acceleration of gravity, can be integrated to give

$$\int\frac{dp}{\rho} + \frac{1}{2}V^2 + gz = C \qquad \text{(along a streamline)} \tag{3.6}$$

where $C$ is a constant of integration to be determined by the conditions at some point on the streamline.

*V3.2 Balancing ball*



In general it is not possible to integrate the pressure term because the density may not be constant and, therefore, cannot be removed from under the integral sign. To carry out this integration we must know specifically how the density varies with pressure. This is not always easily determined. For example, for a perfect gas the density, pressure, and temperature are related according to $\rho = p/RT$, where $R$ is the gas constant. To know how the density varies with pressure, we must also know the temperature variation. For now we will assume that the density and specific weight are constant (incompressible flow). The justification for this assumption and the consequences of compressibility will be considered further in Section 3.8.1 and more fully in Chapter 11.

With the additional assumption that the density remains constant (a very good assumption for liquids and also for gases if the speed is "not too high"), Eq. 3.6 assumes the following simple representation for steady, inviscid, incompressible flow.

$$p + \tfrac{1}{2}\rho V^2 + \gamma z = \text{constant along streamline} \qquad \textbf{(3.7)}$$

*V3.3 Flow past a biker*



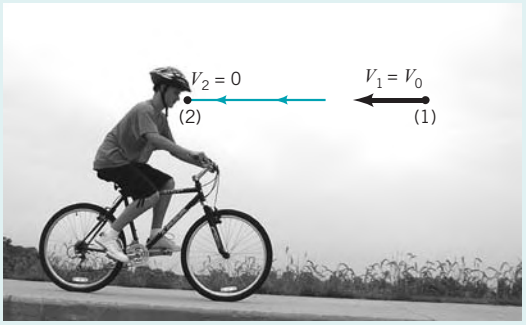
This is the celebrated ***Bernoulli equation***—a very powerful tool in fluid mechanics. In 1738 Daniel Bernoulli (1700–1782) published his *Hydrodynamics* in which an equivalent of this famous equation first appeared. To use it correctly we must constantly remember the basic assumptions used in its derivation: (1) viscous effects are assumed negligible, (2) the flow is assumed to be steady, (3) the flow is assumed to be incompressible, (4) the equation is applicable along a streamline. In the derivation of Eq. 3.7, we assume that the flow takes place in a plane (the $x$–$z$ plane). In general, this equation is valid for both planar and nonplanar (three-dimensional) flows, provided it is applied along the streamline.

We will provide many examples to illustrate the correct use of the Bernoulli equation and will show how a violation of the basic assumptions used in the derivation of this equation can lead to erroneous conclusions. The constant of integration in the Bernoulli equation can be evaluated if sufficient information about the flow is known at one location along the streamline.

## EXAMPLE 3.2 The Bernoulli Equation

**GIVEN** Consider the flow of air around a bicyclist moving through still air with velocity $V_0$, as is shown in Fig. E3.2.

**FIND** Determine the difference in the pressure between points (1) and (2).




■ FIGURE E3.2

### SOLUTION

In a coordinate fixed to the ground, the flow is unsteady as the bicyclist rides by. However, in a coordinate system fixed to the bike, it appears as though the air is flowing steadily toward the bicyclist with speed $V_0$. Since use of the Bernoulli equation is restricted to steady flows, we select the coordinate system fixed to the bike. If the assumptions of Bernoulli's equation are valid (steady, incompressible, inviscid flow), Eq. 3.7 can be applied as follows along the streamline that passes through (1) and (2)

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2$$

We consider (1) to be in the free stream so that $V_1 = V_0$ and (2) to be at the tip of the bicyclist's nose and assume that $z_1 = z_2$ and $V_2 = 0$ (both of which, as is discussed in Section 3.4, are reasonable assumptions). It follows that the pressure at (2) is greater than that at (1) by an amount

$$p_2 - p_1 = \tfrac{1}{2}\rho V_1^2 = \tfrac{1}{2}\rho V_0^2 \qquad \textbf{(Ans)}$$

**COMMENTS** A similar result was obtained in Example 3.1 by integrating the pressure gradient, which was known because the velocity distribution along the streamline, $V(s)$, was known. The Bernoulli equation is a general integration of $\mathbf{F} = m\mathbf{a}$. To determine $p_2 - p_1$, knowledge of the detailed velocity distribution is not needed—only the "boundary conditions" at (1) and (2) are required. Of course, knowledge of the value of $V$ along the streamline is needed to determine the pressure at points between (1) and (2). Note that if we measure $p_2 - p_1$ we can determine the speed, $V_0$. As discussed in Section 3.5, this is the principle upon which many velocity measuring devices are based.

If the bicyclist were accelerating or decelerating, the flow would be unsteady (i.e., $V_0 \neq$ constant) and the above analysis would be incorrect since Eq. 3.7 is restricted to steady flow.

CMS Cepcor EX 1046 Page 9
CMS Cepcor v. Sandvik IPR2024-01380

The difference in fluid velocity between two points in a flow field, $V_1$ and $V_2$, can often be controlled by appropriate geometric constraints of the fluid. For example, a garden hose nozzle is designed to give a much higher velocity at the exit of the nozzle than at its entrance where it is attached to the hose. As is shown by the Bernoulli equation, the pressure within the hose must be larger than that at the exit (for constant elevation, an increase in velocity requires a decrease in pressure if Eq. 3.7 is valid). It is this pressure drop that accelerates the water through the nozzle. Similarly, an airfoil is designed so that the fluid velocity over its upper surface is greater (on the average) than that along its lower surface. From the Bernoulli equation, therefore, the average pressure on the lower surface is greater than that on the upper surface. A net upward force, the lift, results.

## 3.3 F = *m*a Normal to a Streamline

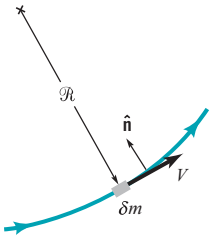


*V3.4 Hydrocyclone separator*



In this section we will consider application of Newton's second law in a direction normal to the streamline. In many flows the streamlines are relatively straight, the flow is essentially one-dimensional, and variations in parameters across streamlines (in the normal direction) can often be neglected when compared to the variations along the streamline. However, in numerous other situations valuable information can be obtained from considering $\mathbf{F} = m\mathbf{a}$ normal to the streamlines. For example, the devastating low-pressure region at the center of a tornado can be explained by applying Newton's second law across the nearly circular streamlines of the tornado.

We again consider the force balance on the fluid particle shown in Fig. 3.3 and the figure in the margin. This time, however, we consider components in the normal direction, $\hat{\mathbf{n}}$, and write Newton's second law in this direction as




$$\sum \delta F_n = \frac{\delta m\, V^2}{\mathcal{R}} = \frac{\rho\, \delta \forall\, V^2}{\mathcal{R}} \tag{3.8}$$

where $\sum \delta F_n$ represents the sum of $n$ components of all the forces acting on the particle and $\delta m$ is particle mass. We assume the flow is steady with a normal acceleration $a_n = V^2/\mathcal{R}$, where $\mathcal{R}$ is the local radius of curvature of the streamlines. This acceleration is produced by the change in direction of the particle's velocity as it moves along a curved path.

We again assume that the only forces of importance are pressure and gravity. The component of the weight (gravity force) in the normal direction is

$$\delta \mathcal{W}_n = -\delta \mathcal{W} \cos\theta = -\gamma\, \delta \forall \cos\theta$$

*To apply* $\mathbf{F} = m\mathbf{a}$ *normal to streamlines, the normal components of force are needed.*

If the streamline is vertical at the point of interest, $\theta = 90°$, and there is no component of the particle weight normal to the direction of flow to contribute to its acceleration in that direction.

If the pressure at the center of the particle is $p$, then its values on the top and bottom of the particle are $p + \delta p_n$ and $p - \delta p_n$, where $\delta p_n = (\partial p/\partial n)(\delta n/2)$. Thus, if $\delta F_{pn}$ is the net pressure force on the particle in the normal direction, it follows that

$$\delta F_{pn} = (p - \delta p_n)\,\delta s\,\delta y - (p + \delta p_n)\,\delta s\,\delta y = -2\,\delta p_n\,\delta s\,\delta y$$

$$= -\frac{\partial p}{\partial n}\delta s\,\delta n\,\delta y = -\frac{\partial p}{\partial n}\delta \forall$$



*V3.5 Aircraft wing tip vortex*



Hence, the net force acting in the normal direction on the particle shown in Fig 3.3 is given by

$$\sum \delta F_n = \delta \mathcal{W}_n + \delta F_{pn} = \left(-\gamma \cos\theta - \frac{\partial p}{\partial n}\right)\delta \forall \tag{3.9}$$

By combining Eqs. 3.8 and 3.9 and using the fact that along a line normal to the streamline $\cos\theta = dz/dn$ (see Fig. 3.3), we obtain the following equation of motion along the normal direction

$$-\gamma \frac{dz}{dn} - \frac{\partial p}{\partial n} = \frac{\rho V^2}{\mathcal{R}} \tag{3.10a}$$

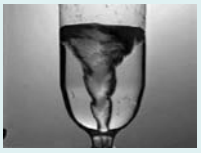
*Weight and/or pressure can produce curved streamlines.*

The physical interpretation of Eq. 3.10 is that a change in the direction of flow of a fluid particle (i.e., a curved path, $\mathcal{R} < \infty$) is accomplished by the appropriate combination of pressure gradient and particle weight normal to the streamline. A larger speed or density or a smaller radius of curvature of the motion requires a larger force unbalance to produce the motion. For example, if gravity is neglected (as is commonly done for gas flows) or if the flow is in a horizontal ($dz/dn = 0$) plane, Eq. 3.10 becomes

$$\frac{\partial p}{\partial n} = -\frac{\rho V^2}{\mathcal{R}} \qquad \textbf{(3.10b)}$$



*V3.6 Free vortex*



This indicates that the pressure increases with distance away from the center of curvature ($\partial p/\partial n$ is negative since $\rho V^2/\mathcal{R}$ is positive—the positive $n$ direction points toward the "inside" of the curved streamline). Thus, the pressure outside a tornado (typical atmospheric pressure) is larger than it is near the center of the tornado (where an often dangerously low partial vacuum may occur). This pressure difference is needed to balance the centrifugal acceleration associated with the curved streamlines of the fluid motion. (See Fig. E6.6a in Section 6.5.3.)

## EXAMPLE 3.3 Pressure Variation Normal to a Streamline

**GIVEN** Shown in Figs. E3.3*a,b* are two flow fields with circular streamlines. The velocity distributions are

$$V(r) = (V_0/r_0)r \quad \text{for case } (a)$$

and

$$V(r) = \frac{(V_0 r_0)}{r} \quad \text{for case } (b)$$

where $V_0$ is the velocity at $r = r_0$.

**FIND** Determine the pressure distributions, $p = p(r)$, for each, given that $p = p_0$ at $r = r_0$.

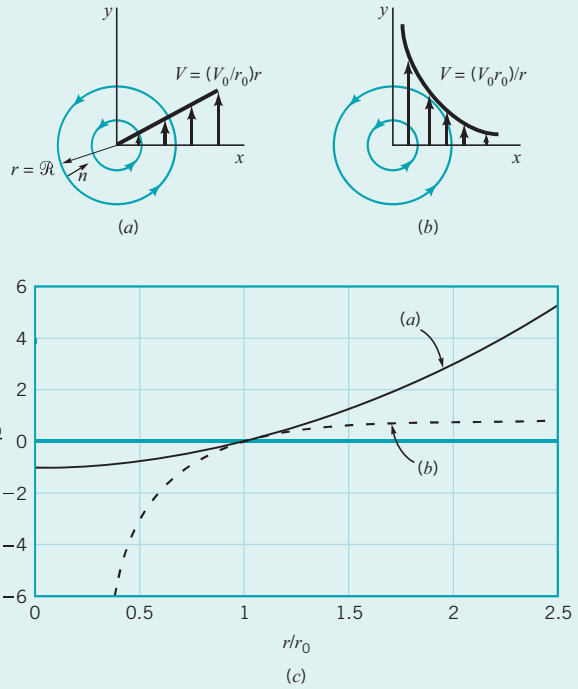



**FIGURE E3.3**

## SOLUTION

We assume the flows are steady, inviscid, and incompressible with streamlines in the horizontal plane ($dz/dn = 0$). Because the streamlines are circles, the coordinate $n$ points in a direction opposite that of the radial coordinate, $\partial/\partial n = -\partial/\partial r$, and the radius of curvature is given by $\mathcal{R} = r$. Hence, Eq. 3.9 becomes

$$\frac{\partial p}{\partial r} = \frac{\rho V^2}{r}$$

For case $(a)$ this gives

$$\frac{\partial p}{\partial r} = \rho(V_0/r_0)^2 r$$

whereas for case $(b)$ it gives

$$\frac{\partial p}{\partial r} = \frac{\rho(V_0\, r_0)^2}{r^3}$$

For either case the pressure increases as $r$ increases since $\partial p/\partial r > 0$. Integration of these equations with respect to $r$, starting with a known pressure $p = p_0$ at $r = r_0$, gives

$$p - p_0 = (\rho V_0^2/2)[(r/r_0)^2 - 1] \qquad \textbf{(Ans)}$$

for case $(a)$ and

$$p - p_0 = (\rho V_0^2/2)[1 - (r_0/r)^2] \qquad \textbf{(Ans)}$$

for case $(b)$. These pressure distributions are shown in Fig. E3.3*c*.

**COMMENT** The pressure distributions needed to balance the centrifugal accelerations in cases $(a)$ and $(b)$ are not the same because the velocity distributions are different. In fact, for case $(a)$ the

pressure increases without bound as $r \to \infty$, whereas for case (*b*) the pressure approaches a finite value as $r \to \infty$. The streamline patterns are the same for each case, however.

Physically, case (*a*) represents rigid body rotation (as obtained in a can of water on a turntable after it has been "spun up") and case (*b*) represents a free vortex (an approximation to a tornado, a hurricane, or the swirl of water in a drain, the "bathtub vortex"). See Fig. E6.6 for an approximation of this type of flow.

*The sum of pressure, elevation, and velocity effects is constant across streamlines.*

If we multiply Eq. 3.10 by $dn$, use the fact that $\partial p/\partial n = dp/dn$ if $s$ is constant, and integrate across the streamline (in the $n$ direction) we obtain

$$\int \frac{dp}{\rho} + \int \frac{V^2}{\mathcal{R}} dn + gz = \text{constant across the streamline} \qquad \textbf{(3.11)}$$

To complete the indicated integrations, we must know how the density varies with pressure and how the fluid speed and radius of curvature vary with $n$. For incompressible flow the density is constant and the integration involving the pressure term gives simply $p/\rho$. We are still left, however, with the integration of the second term in Eq. 3.11. Without knowing the $n$ dependence in $V = V(s, n)$ and $\mathcal{R} = \mathcal{R}(s, n)$ this integration cannot be completed.

Thus, the final form of Newton's second law applied across the streamlines for steady, inviscid, incompressible flow is

$$p + \rho \int \frac{V^2}{\mathcal{R}} dn + \gamma z = \text{constant across the streamline} \qquad \textbf{(3.12)}$$

As with the Bernoulli equation, we must be careful that the assumptions involved in the derivation of this equation are not violated when it is used.

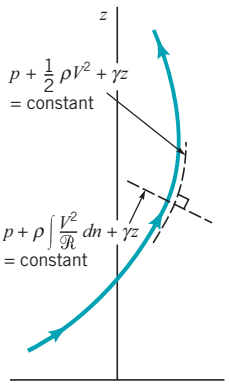
## 3.4 Physical Interpretation

In the previous two sections, we developed the basic equations governing fluid motion under a fairly stringent set of restrictions. In spite of the numerous assumptions imposed on these flows, a variety of flows can be readily analyzed with them. A physical interpretation of the equations will be of help in understanding the processes involved. To this end, we rewrite Eqs. 3.7 and 3.12 here and interpret them physically. Application of $\mathbf{F} = m\mathbf{a}$ along and normal to the streamline results in

$$p + \tfrac{1}{2}\rho V^2 + \gamma z = \text{constant along the streamline} \qquad \textbf{(3.13)}$$

and

$$p + \rho \int \frac{V^2}{\mathcal{R}} dn + \gamma z = \text{constant across the streamline} \qquad \textbf{(3.14)}$$




as indicated by the figure in the margin.

The following basic assumptions were made to obtain these equations: The flow is steady and the fluid is inviscid and incompressible. In practice none of these assumptions is exactly true.

A violation of one or more of the above assumptions is a common cause for obtaining an incorrect match between the "real world" and solutions obtained by use of the Bernoulli equation. Fortunately, many "real-world" situations are adequately modeled by the use of Eqs. 3.13 and 3.14 because the flow is nearly steady and incompressible and the fluid behaves as if it were nearly inviscid.

The Bernoulli equation was obtained by integration of the equation of motion along the "natural" coordinate direction of the streamline. To produce an acceleration, there must be an unbalance of the resultant forces, of which only pressure and gravity were considered to be important. Thus,

there are three processes involved in the flow—mass times acceleration (the $\rho V^2/2$ term), pressure (the $p$ term), and weight (the $\gamma z$ term).

Integration of the equation of motion to give Eq. 3.13 actually corresponds to the work-energy principle often used in the study of dynamics [see any standard dynamics text (Ref. 1)]. This principle results from a general integration of the equations of motion for an object in a way very similar to that done for the fluid particle in Section 3.2. With certain assumptions, a statement of the work-energy principle may be written as follows:

> The work done on a particle by all forces acting on the particle is equal to the change of the kinetic energy of the particle.

The Bernoulli equation is a mathematical statement of this principle.

As the fluid particle moves, both gravity and pressure forces do work on the particle. Recall that the work done by a force is equal to the product of the distance the particle travels times the component of force in the direction of travel (i.e., work $= \mathbf{F} \cdot \mathbf{d}$). The terms $\gamma z$ and $p$ in Eq. 3.13 are related to the work done by the weight and pressure forces, respectively. The remaining term, $\rho V^2/2$, is obviously related to the kinetic energy of the particle. In fact, an alternate method of deriving the Bernoulli equation is to use the first and second laws of thermodynamics (the energy and entropy equations), rather than Newton's second law. With the appropriate restrictions, the general energy equation reduces to the Bernoulli equation. This approach is discussed in Section 5.4.

*The Bernoulli equation can be written in terms of heights called heads.*

An alternate but equivalent form of the Bernoulli equation is obtained by dividing each term of Eq. 3.7 by the specific weight, $\gamma$, to obtain

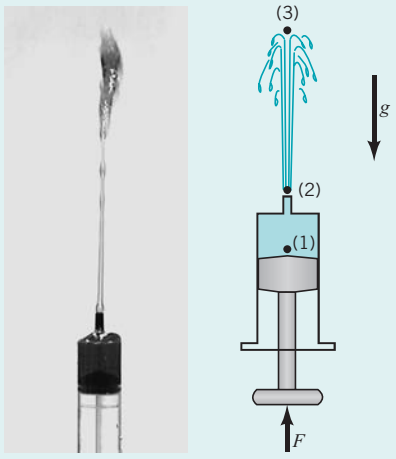
$$\frac{p}{\gamma} + \frac{V^2}{2g} + z = \text{constant on a streamline}$$

Each of the terms in this equation has the units of energy per weight ($LF/F = L$) or length (feet, meters) and represents a certain type of head.

The elevation term, $z$, is related to the potential energy of the particle and is called the ***elevation head.*** The pressure term, $p/\gamma$, is called the ***pressure head*** and represents the height of a column of the fluid that is needed to produce the pressure $p$. The velocity term, $V^2/2g$, is the ***velocity head*** and represents the vertical distance needed for the fluid to fall freely (neglecting friction) if it is to reach velocity $V$ from rest. The Bernoulli equation states that the sum of the pressure head, the velocity head, and the elevation head is constant along a streamline.

## EXAMPLE 3.4 Kinetic, Potential, and Pressure Energy

**GIVEN** Consider the flow of water from the syringe shown in Fig. E3.4(*a*). As indicated in Fig. E3.4*b*, a force, $F$, applied to the plunger will produce a pressure greater than atmospheric at point (1) within the syringe. The water flows from the needle, point (2), with relatively high velocity and coasts up to point (3) at the top of its trajectory.




■ FIGURE E3.4

**FIND** Discuss the energy of the fluid at points (1), (2), and (3) by using the Bernoulli equation.

| | Energy Type | | |
|---|---|---|---|
| Point | Kinetic $\rho V^2/2$ | Potential $\gamma z$ | Pressure $p$ |
| 1 | Small | Zero | Large |
| 2 | Large | Small | Zero |
| 3 | Zero | Large | Zero |

CMS Cepcor EX 1046 Page 13
CMS Cepcor v. Sandvik IPR2024-01380

## SOLUTION

If the assumptions (steady, inviscid, incompressible flow) of the Bernoulli equation are approximately valid, it then follows that the flow can be explained in terms of the partition of the total energy of the water. According to Eq. 3.13 the sum of the three types of energy (kinetic, potential, and pressure) or heads (velocity, elevation, and pressure) must remain constant. The table above indicates the relative magnitude of each of these energies at the three points shown in the figure.

The motion results in (or is due to) a change in the magnitude of each type of energy as the fluid flows from one location to another. An alternate way to consider this flow is as follows. The pressure gradient between (1) and (2) produces an acceleration to eject the water from the needle. Gravity acting on the particle between (2) and (3) produces a deceleration to cause the water to come to a momentary stop at the top of its flight.

**COMMENT** If friction (viscous) effects were important, there would be an energy loss between (1) and (3) and for the given $p_1$ the water would not be able to reach the height indicated in the figure. Such friction may arise in the needle (see Chapter 8 on pipe flow) or between the water stream and the surrounding air (see Chapter 9 on external flow).

## Fluids in the News

**Armed with a water jet for hunting** Archerfish, known for their ability to shoot down insects resting on foliage, are like submarine water pistols. With their snout sticking out of the water, they eject a high-speed water jet at their prey, knocking it onto the water surface where they snare it for their meal. The barrel of their water pistol is formed by placing their tongue against a groove in the roof of their mouth to form a tube. By snapping shut their gills, water is forced through the tube and directed with the tip of their tongue. The archerfish can produce a *pressure head* within their gills large enough so that the jet can reach 2 to 3 m. However, it is accurate to only about 1 m. Recent research has shown that archerfish are very adept at calculating where their prey will fall. Within 100 milliseconds (a reaction time twice as fast as a human's), the fish has extracted all the information needed to predict the point where the prey will hit the water. Without further visual cues it charges directly to that point. (See Problem 3.41.)

A net force is required to accelerate any mass. For steady flow the acceleration can be interpreted as arising from two distinct occurrences—a change in speed along the streamline and a change in direction if the streamline is not straight. Integration of the equation of motion along the streamline accounts for the change in speed (kinetic energy change) and results in the Bernoulli equation. Integration of the equation of motion normal to the streamline accounts for the centrifugal acceleration ($V^2/\mathcal{R}$) and results in Eq. 3.14.

*The pressure variation across straight streamlines is hydrostatic.*

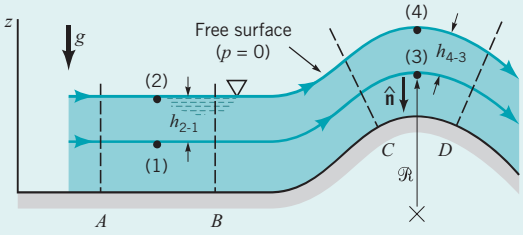
When a fluid particle travels along a curved path, a net force directed toward the center of curvature is required. Under the assumptions valid for Eq. 3.14, this force may be either gravity or pressure, or a combination of both. In many instances the streamlines are nearly straight ($\mathcal{R} = \infty$) so that centrifugal effects are negligible and the pressure variation across the streamlines is merely hydrostatic (because of gravity alone), even though the fluid is in motion.

## EXAMPLE 3.5 Pressure Variation in a Flowing Stream

**GIVEN** Water flows in a curved, undulating waterslide as shown in Fig. E3.5*a*. As an approximation to this flow, consider the inviscid, incompressible, steady flow shown in Fig. E3.5*b*. From section *A* to *B* the streamlines are straight, while from *C* to *D* they follow circular paths.



■ FIGURE E3.5*a* (Photo courtesy of Schlitterbahn® Waterparks.)




■ FIGURE E3.5*b*

**FIND** Describe the pressure variation between points (1) and (2) and points (3) and (4).

## SOLUTION

With the above assumptions and the fact that $\mathcal{R} = \infty$ for the portion from $A$ to $B$, Eq. 3.14 becomes

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

The constant can be determined by evaluating the known variables at the two locations using $p_2 = 0$ (gage), $z_1 = 0$, and $z_2 = h_{2-1}$ to give

$$p_1 = p_2 + \gamma(z_2 - z_1) = p_2 + \gamma h_{2-1} \qquad \textbf{(Ans)}$$

Note that since the radius of curvature of the streamline is infinite, the pressure variation in the vertical direction is the same as if the fluid were stationary.

However, if we apply Eq. 3.14 between points (3) and (4) we obtain (using $dn = -dz$)

$$p_4 + \rho \int_{z_3}^{z_4} \frac{V^2}{\mathcal{R}}(-dz) + \gamma z_4 = p_3 + \gamma z_3$$

With $p_4 = 0$ and $z_4 - z_3 = h_{4-3}$ this becomes

$$p_3 = \gamma h_{4-3} - \rho \int_{z_3}^{z_4} \frac{V^2}{\mathcal{R}}\, dz \qquad \textbf{(Ans)}$$

To evaluate the integral, we must know the variation of $V$ and $\mathcal{R}$ with $z$. Even without this detailed information we note that the integral has a positive value. Thus, the pressure at (3) is less than the hydrostatic value, $\gamma h_{4-3}$, by an amount equal to $\rho \int_{z_3}^{z_4} (V^2/\mathcal{R})\, dz$. This lower pressure, caused by the curved streamline, is necessary to accelerate the fluid around the curved path.

**COMMENT** Note that we did not apply the Bernoulli equation (Eq. 3.13) across the streamlines from (1) to (2) or (3) to (4). Rather we used Eq. 3.14. As is discussed in Section 3.8, application of the Bernoulli equation across streamlines (rather than along them) may lead to serious errors.

## 3.5 Static, Stagnation, Dynamic, and Total Pressure

*Each term in the Bernoulli equation can be interpreted as a form of pressure.*

A useful concept associated with the Bernoulli equation deals with the stagnation and dynamic pressures. These pressures arise from the conversion of kinetic energy in a flowing fluid into a "pressure rise" as the fluid is brought to rest (as in Example 3.2). In this section we explore various results of this process. Each term of the Bernoulli equation, Eq. 3.13, has the dimensions of force per unit area—psi, lb/ft$^2$, N/m$^2$. The first term, $p$, is the actual thermodynamic pressure of the fluid as it flows. To measure its value, one could move along with the fluid, thus being "static" relative to the moving fluid. Hence, it is normally termed the ***static pressure.*** Another way to measure the static pressure would be to drill a hole in a flat surface and fasten a piezometer tube as indicated by the location of point (3) in Fig. 3.4. As we saw in Example 3.5, the pressure in the flowing fluid at (1) is $p_1 = \gamma h_{3-1} + p_3$, the same as if the fluid were static. From the manometer considerations of Chapter 2, we know that $p_3 = \gamma h_{4-3}$. Thus, since $h_{3-1} + h_{4-3} = h$ it follows that $p_1 = \gamma h$.

The third term in Eq. 3.13, $\gamma z$, is termed the *hydrostatic pressure,* in obvious regard to the hydrostatic pressure variation discussed in Chapter 2. It is not actually a pressure but does represent the change in pressure possible due to potential energy variations of the fluid as a result of elevation changes.

The second term in the Bernoulli equation, $\rho V^2/2$, is termed the ***dynamic pressure.*** Its interpretation can be seen in Fig. 3.4 by considering the pressure at the end of a small tube inserted into the flow and pointing upstream. After the initial transient motion has died out, the liquid will fill the tube to a height of $H$ as shown. The fluid in the tube, including that at its tip, (2), will be stationary. That is, $V_2 = 0$, or point (2) is a ***stagnation point.***

If we apply the Bernoulli equation between points (1) and (2), using $V_2 = 0$ and assuming that $z_1 = z_2$, we find that

$$p_2 = p_1 + \tfrac{1}{2}\rho V_1^2$$

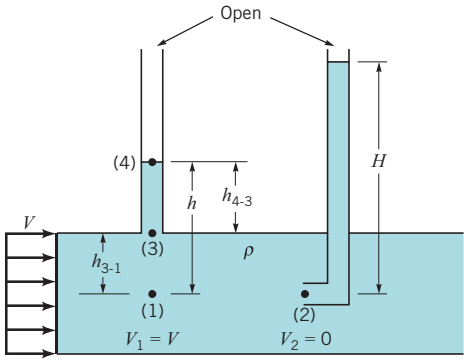


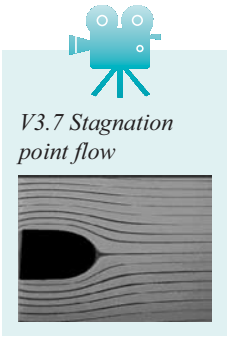

**FIGURE 3.4** Measurement of static and stagnation pressures.

CMS Cepcor EX 1046 Page 15
CMS Cepcor v. Sandvik IPR2024-01380




(*a*) (*b*)

■ **FIGURE 3.5** **Stagnation points.**




Hence, the pressure at the stagnation point is greater than the static pressure, $p_1$, by an amount $\rho V_1^2/2$, the dynamic pressure.

It can be shown that there is a stagnation point on any stationary body that is placed into a flowing fluid. Some of the fluid flows "over" and some "under" the object. The dividing line (or surface for two-dimensional flows) is termed the *stagnation streamline* and terminates at the stagnation point on the body. (See the photograph at the beginning of Chapter 3.) For symmetrical objects (such as a baseball) the stagnation point is clearly at the tip or front of the object as shown in Fig. 3.5*a*. For other flows such as a water jet against a car as shown in Fig. 3.5*b*, there is also a stagnation point on the car.

If elevation effects are neglected, the ***stagnation pressure,*** $p + \rho V^2/2$, is the largest pressure obtainable along a given streamline. It represents the conversion of all of the kinetic energy into a pressure rise. The sum of the static pressure, hydrostatic pressure, and dynamic pressure is termed the ***total pressure,*** $p_T$. The Bernoulli equation is a statement that the total pressure remains constant along a streamline. That is,

$$p + \tfrac{1}{2}\rho V^2 + \gamma z = p_T = \text{constant along a streamline} \qquad \textbf{(3.15)}$$

Again, we must be careful that the assumptions used in the derivation of this equation are appropriate for the flow being considered.

## Fluids in the News

**Pressurized eyes** Our eyes need a certain amount of internal pressure in order to work properly, with the normal range being between 10 and 20 mm of mercury. The pressure is determined by a balance between the fluid entering and leaving the eye. If the pressure is above the normal level, damage may occur to the optic nerve where it leaves the eye, leading to a loss of the visual field termed glaucoma. Measurement of the pressure within the eye can be done by several different noninvasive types of instruments, all of which measure the slight deformation of the eyeball when a force is put on it. Some methods use a physical probe that makes contact with the front of the eye, applies a known force, and measures the deformation. One noncontact method uses a calibrated "puff" of air that is blown against the eye. The *stagnation pressure* resulting from the air blowing against the eyeball causes a slight deformation, the magnitude of which is correlated with the pressure within the eyeball. (See Problem 3.29.)

Knowledge of the values of the static and stagnation pressures in a fluid implies that the fluid speed can be calculated. This is the principle on which the ***Pitot-static tube*** is based [H. de Pitot (1695–1771)]. As shown in Fig. 3.6, two concentric tubes are attached to two pressure gages (or a differential gage) so that the values of $p_3$ and $p_4$ (or the difference $p_3 - p_4$) can be determined. The center tube measures the stagnation pressure at its open tip. If elevation changes are negligible,

$$p_3 = p + \tfrac{1}{2}\rho V^2$$

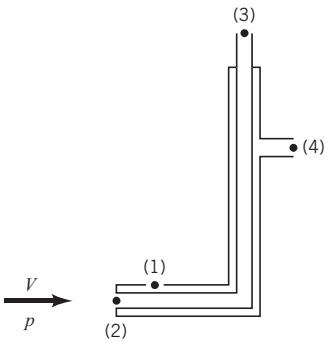



**FIGURE 3.6** The Pitot-static tube.

where $p$ and $V$ are the pressure and velocity of the fluid upstream of point (2). The outer tube is made with several small holes at an appropriate distance from the tip so that they measure the static pressure. If the effect of the elevation difference between (1) and (4) is negligible, then

$$p_4 = p_1 = p$$

*Pitot-static tubes measure fluid velocity by converting velocity into pressure.*

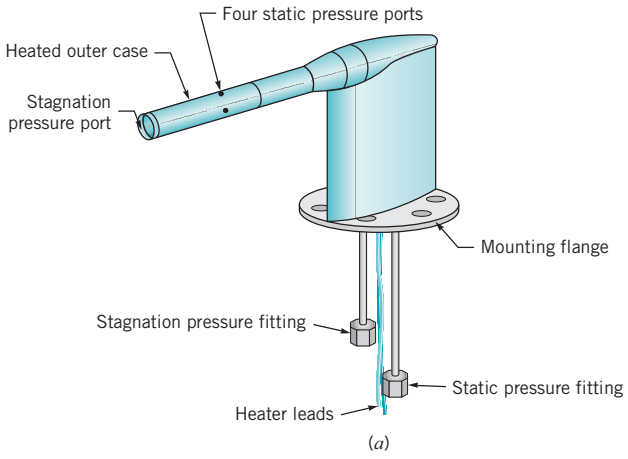
By combining these two equations we see that

$$p_3 - p_4 = \tfrac{1}{2}\rho V^2$$

which can be rearranged to give

$$V = \sqrt{2(p_3 - p_4)/\rho} \tag{3.16}$$

The actual shape and size of Pitot-static tubes vary considerably. A typical Pitot-static probe used to determine aircraft airspeed is shown in Fig. 3.7. (See Fig. E3.6*a* also.)



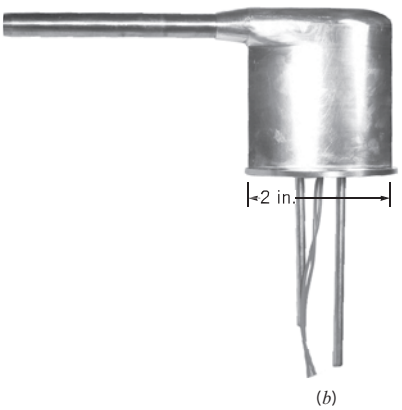




**FIGURE 3.7** Airplane Pitot-static probe. (*a*) Schematic, (*b*) Photograph, (Photograph courtesy of SpaceAge Control, Inc.)

*V3.8 Airspeed indicator*



CMS Cepcor EX 1046 Page 17
CMS Cepcor v. Sandvik IPR2024-01380

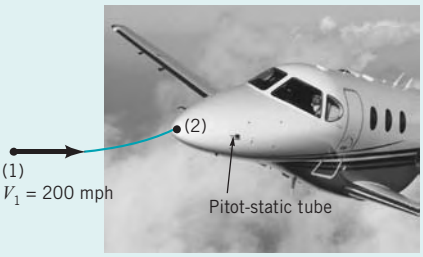
## Fluids in the News

**Bugged and plugged Pitot tubes** Although a *Pitot tube* is a simple device for measuring aircraft airspeed, many airplane accidents have been caused by inaccurate Pitot tube readings. Most of these accidents are the result of having one or more of the holes blocked and, therefore, not indicating the correct pressure (speed). Usually this is discovered during takeoff when time to resolve the issue is short. The two most common causes for such a blockage are either that the pilot (or ground crew) has forgotten to remove the protective Pitot tube cover, or that insects have built their nest within the tube where the standard visual check cannot detect it. The most serious accident (in terms of number of fatalities) caused by a blocked Pitot tube involved a Boeing 757 and occurred shortly after takeoff from Puerto Plata in the Dominican Republic. The incorrect airspeed data was automatically fed to the computer, causing the autopilot to change the angle of attack and the engine power. The flight crew became confused by the false indications, the aircraft stalled, and then plunged into the Caribbean Sea killing all aboard. (See Problem 3.30.)

## EXAMPLE 3.6 Pitot-Static Tube

**GIVEN** An airplane flies 200 mi/hr at an elevation of 10,000 ft in a standard atmosphere as shown in Fig. E3.6*a*.

**FIND** Determine the pressure at point (1) far ahead of the airplane, the pressure at the stagnation point on the nose of the airplane, point (2), and the pressure difference indicated by a Pitot-static probe attached to the fuselage.




■ FIGURE E3.6*a* (Photo courtesy of Hawker Beechcraft.)

### SOLUTION

From Table C.1 we find that the static pressure at the altitude given is

$$p_1 = 1456 \text{ lb/ft}^2 \text{ (abs)} = 10.11 \text{ psia} \qquad \textbf{(Ans)}$$

Also, the density is $\rho = 0.001756$ slug/ft$^3$.

If the flow is steady, inviscid, and incompressible and elevation changes are neglected, Eq. 3.13 becomes

$$p_2 = p_1 + \frac{\rho V_1^2}{2}$$

With $V_1 = 200$ mi/hr $= 293$ ft/s and $V_2 = 0$ (since the coordinate system is fixed to the airplane) we obtain

$$p_2 = 1456 \text{ lb/ft}^2 + (0.001756 \text{ slugs/ft}^3)(293^2 \text{ ft}^2/\text{s}^2)/2$$
$$= (1456 + 75.4) \text{ lb/ft}^2 \text{ (abs)}$$

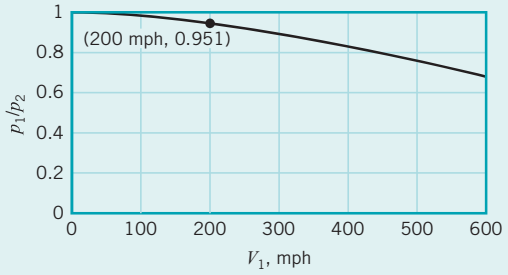
Hence, in terms of gage pressure

$$p_2 = 75.4 \text{ lb/ft}^2 = 0.524 \text{ psi} \qquad \textbf{(Ans)}$$

Thus, the pressure difference indicated by the Pitot-static tube is

$$p_2 - p_1 = \frac{\rho V_1^2}{2} = 0.524 \text{ psi} \qquad \textbf{(Ans)}$$

**COMMENTS** Note that it is very easy to obtain incorrect results by using improper units. Do not add lb/in.$^2$ and lb/ft$^2$. Recall that (slug/ft$^3$)(ft$^2$/s$^2$) = (slug · ft/s$^2$)/(ft$^2$) = lb/ft$^2$.




■ FIGURE E3.6*b*

It was assumed that the flow is incompressible—the density remains constant from (1) to (2). However, since $\rho = p/RT$, a change in pressure (or temperature) will cause a change in density. For this relatively low speed, the ratio of the absolute pressures is nearly unity [i.e., $p_1/p_2 = (10.11 \text{ psia})/(10.11 + 0.524 \text{ psia}) = 0.951$], so that the density change is negligible. However, by repeating the calculations for various values of the speed, $V_1$, the results shown in Fig. E3.6*b* are obtained. Clearly at the 500 to 600 mph speeds normally flown by commercial airliners, the pressure ratio is such that density changes are important. In such situations it is necessary to use compressible flow concepts to obtain accurate results. (See Section 3.8.1 and Chapter 11.)

The Pitot-static tube provides a simple, relatively inexpensive way to measure fluid speed. Its use depends on the ability to measure the static and stagnation pressures. Care is needed to obtain these values accurately. For example, an accurate measurement of static pressure requires that none of the fluid's kinetic energy be converted into a pressure rise at the point of

CMS Cepcor EX 1046 Page 18
CMS Cepcor v. Sandvik IPR2024-01380

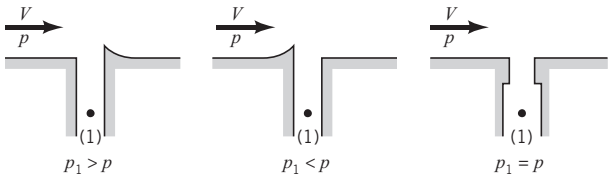



**FIGURE 3.8** Incorrect and correct design of static pressure taps.

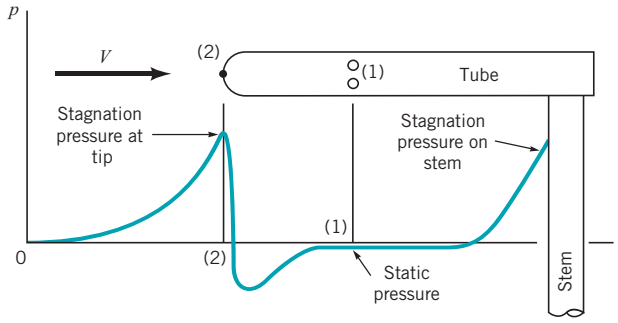



**FIGURE 3.9** Typical pressure distribution along a Pitot-static tube.

*Accurate measurement of static pressure requires great care.*

measurement. This requires a smooth hole with no burrs or imperfections. As indicated in Fig. 3.8, such imperfections can cause the measured pressure to be greater or less than the actual static pressure.

Also, the pressure along the surface of an object varies from the stagnation pressure at its stagnation point to values that may be less than the free stream static pressure. A typical pressure variation for a Pitot-static tube is indicated in Fig. 3.9. Clearly it is important that the pressure taps be properly located to ensure that the pressure measured is actually the static pressure.

In practice it is often difficult to align the Pitot-static tube directly into the flow direction. Any misalignment will produce a nonsymmetrical flow field that may introduce errors. Typically, yaw angles up to 12 to 20° (depending on the particular probe design) give results that are less than 1% in error from the perfectly aligned results. Generally it is more difficult to measure static pressure than stagnation pressure.

One method of determining the flow direction and its speed (thus the velocity) is to use a directional-finding Pitot tube as is illustrated in Fig. 3.10. Three pressure taps are drilled into a small circular cylinder, fitted with small tubes, and connected to three pressure transducers. The cylinder is rotated until the pressures in the two side holes are equal, thus indicating that the center hole points directly upstream. The center tap then measures the stagnation pressure. The two side holes are located at a specific angle ($\beta = 29.5°$) so that they measure the static pressure. The speed is then obtained from $V = [2(p_2 - p_1)/\rho]^{1/2}$.

The above discussion is valid for incompressible flows. At high speeds, compressibility becomes important (the density is not constant) and other phenomena occur. Some of these ideas are discussed in Section 3.8, while others (such as shockwaves for supersonic Pitot-tube applications) are discussed in Chapter 11.

The concepts of static, dynamic, stagnation, and total pressure are useful in a variety of flow problems. These ideas are used more fully in the remainder of the book.

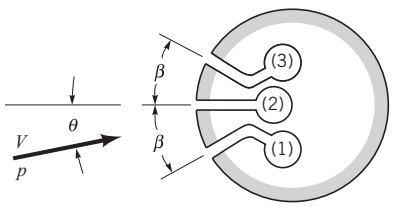



**FIGURE 3.10** Cross section of a directional-finding Pitot-static tube.

CMS Cepcor EX 1046 Page 19
CMS Cepcor v. Sandvik IPR2024-01380

## 3.6 Examples of Use of the Bernoulli Equation

In this section we illustrate various additional applications of the Bernoulli equation. Between any two points, (1) and (2), on a streamline in steady, inviscid, incompressible flow the Bernoulli equation can be applied in the form

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2 \tag{3.17}$$

Obviously if five of the six variables are known, the remaining one can be determined. In many instances it is necessary to introduce other equations, such as the conservation of mass. Such considerations will be discussed briefly in this section and in more detail in Chapter 5.

### 3.6.1 Free Jets



One of the oldest equations in fluid mechanics deals with the flow of a liquid from a large reservoir. A modern version of this type of flow involves the flow of coffee from a coffee urn as indicated by the figure in the margin. The basic principles of this type of flow are shown in Fig. 3.11 where a jet of liquid of diameter $d$ flows from the nozzle with velocity $V$. (A nozzle is a device shaped to accelerate a fluid.) Application of Eq. 3.17 between points (1) and (2) on the streamline shown gives

$$\gamma h = \tfrac{1}{2}\rho V^2$$

We have used the facts that $z_1 = h$, $z_2 = 0$, the reservoir is large ($V_1 \cong 0$) and open to the atmosphere ($p_1 = 0$ gage), and the fluid leaves as a ***"free jet"*** ($p_2 = 0$). Thus, we obtain

$$V = \sqrt{2\frac{\gamma h}{\rho}} = \sqrt{2gh} \tag{3.18}$$

which is the modern version of a result obtained in 1643 by Torricelli (1608–1647), an Italian physicist.

*The exit pressure for an incompressible fluid jet is equal to the surrounding pressure.*

The fact that the exit pressure equals the surrounding pressure ($p_2 = 0$) can be seen by applying $\mathbf{F} = m\mathbf{a}$, as given by Eq. 3.14, across the streamlines between (2) and (4). If the streamlines at the tip of the nozzle are straight ($\mathcal{R} = \infty$), it follows that $p_2 = p_4$. Since (4) is on the surface of the jet, in contact with the atmosphere, we have $p_4 = 0$. Thus, $p_2 = 0$ also. Since (2) is an arbitrary point in the exit plane of the nozzle, it follows that the pressure is atmospheric across this plane. Physically, since there is no component of the weight force or acceleration in the normal (horizontal) direction, the pressure is constant in that direction.

Once outside the nozzle, the stream continues to fall as a free jet with zero pressure throughout ($p_5 = 0$) and as seen by applying Eq. 3.17 between points (1) and (5), the speed increases according to

$$V = \sqrt{2g\,(h + H)}$$

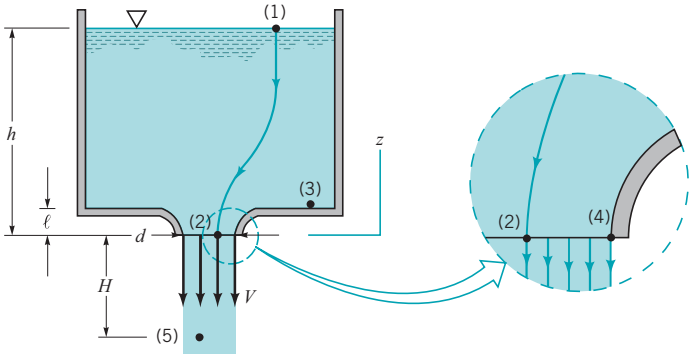
where $H$ is the distance the fluid has fallen outside the nozzle.

Equation 3.18 could also be obtained by writing the Bernoulli equation between points (3) and (4) using the fact that $z_4 = 0, z_3 = \ell$. Also, $V_3 = 0$ since it is far from the nozzle, and from hydrostatics, $p_3 = \gamma(h - \ell)$.

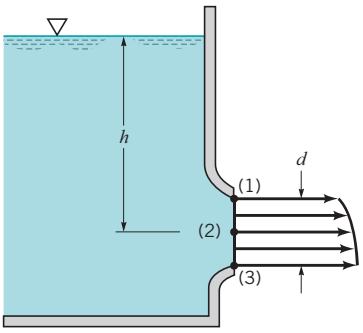


*V3.9 Flow from a tank*

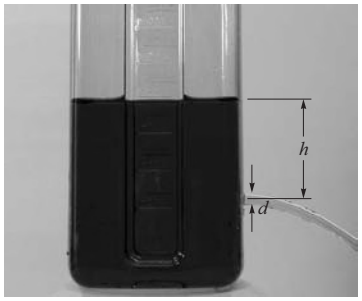





■ **FIGURE 3.11**
**Vertical flow from a tank.**




(a)




(b)

**FIGURE 3.12** Horizontal flow from a tank.

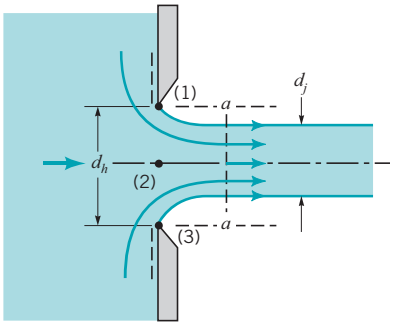
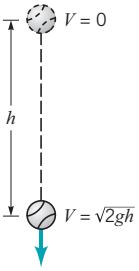
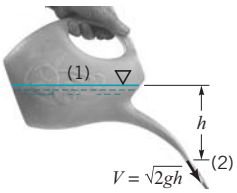



**FIGURE 3.13** Vena contracta effect for a sharp-edged orifice.




*The diameter of a fluid jet is often smaller than that of the hole from which it flows.*

As learned in physics or dynamics and illustrated in the figure in the margin, any object dropped from rest that falls through a distance $h$ in a vacuum will obtain the speed $V = \sqrt{2gh}$, the same as the water leaving the spout of the watering can shown in the figure in the margin. This is consistent with the fact that all of the particle's potential energy is converted to kinetic energy, provided viscous (friction) effects are negligible. In terms of heads, the elevation head at point (1) is converted into the velocity head at point (2). Recall that for the case shown in Fig. 3.11 the pressure is the same (atmospheric) at points (1) and (2).

For the horizontal nozzle of Fig. 3.12*a*, the velocity of the fluid at the centerline, $V_2$,will be slightly greater than that at the top, $V_1$, and slightly less than that at the bottom, $V_3$, due to the differences in elevation. In general, $d \ll h$ as shown in Fig. 3.12*b* and we can safely use the centerline velocity as a reasonable "average velocity."

If the exit is not a smooth, well-contoured nozzle, but rather a flat plate as shown in Fig. 3.13, the diameter of the jet, $d_j$, will be less than the diameter of the hole, $d_h$. This phenomenon, called a *vena contracta* effect, is a result of the inability of the fluid to turn the sharp 90° corner indicated by the dotted lines in the figure.

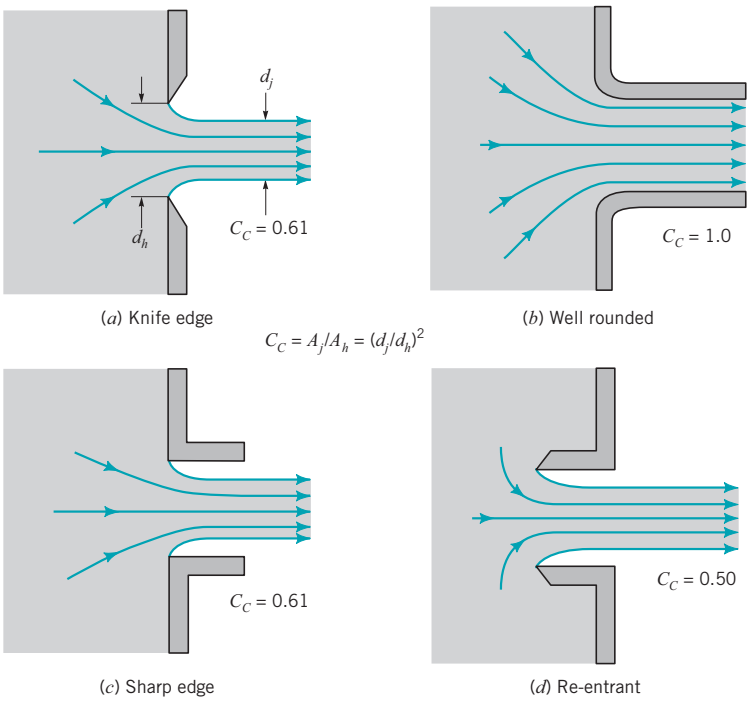
Since the streamlines in the exit plane are curved ($\mathcal{R} < \infty$), the pressure across them is not constant. It would take an infinite pressure gradient across the streamlines to cause the fluid to turn a "sharp" corner ($\mathcal{R} = 0$). The highest pressure occurs along the centerline at (2) and the lowest pressure, $p_1 = p_3 = 0$, is at the edge of the jet. Thus, the assumption of uniform velocity with straight streamlines and constant pressure is not valid at the exit plane. It is valid, however, in the plane of the vena contracta, section $a$–$a$. The uniform velocity assumption is valid at this section provided $d_j \ll h$, as is discussed for the flow from the nozzle shown in Fig. 3.12.

The vena contracta effect is a function of the geometry of the outlet. Some typical configurations are shown in Fig. 3.14 along with typical values of the experimentally obtained *contraction coefficient*, $C_c = A_j/A_h$, where $A_j$ and $A_h$ are the areas of the jet at the vena contracta and the area of the hole, respectively.

## Fluids in the News

**Cotton candy, glass wool, and steel wool** Although cotton candy and glass wool insulation are made of entirely different materials and have entirely different uses, they are made by similar processes. Cotton candy, invented in 1897, consists of sugar fibers. Glass wool, invented in 1938, consists of glass fibers. In a cotton candy machine, sugar is melted and then forced by centrifugal action to flow through numerous tiny *orifices* in a spinning "bowl." Upon emerging, the thin streams of liquid sugar cool very quickly and become solid threads that are collected on a stick or cone. Making glass wool insulation is somewhat more complex, but the basic process is similar. Liquid glass is forced through tiny orifices and emerges as very fine glass streams that quickly solidify. The resulting intertwined flexible fibers, glass wool, form an effective insulation material because the tiny air "cavities" between the fibers inhibit air motion. Although steel wool looks similar to cotton candy or glass wool, it is made by an entirely different process. Solid steel wires are drawn over special cutting blades which have grooves cut into them so that long, thin threads of steel are peeled off to form the matted steel wool.

CMS Cepcor EX 1046 Page 21
CMS Cepcor v. Sandvik IPR2024-01380

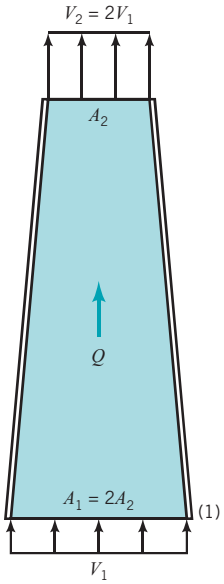



■ **FIGURE 3.14** **Typical flow patterns and contraction coefficients for various round exit configurations. (*a*) Knife edge, (*b*) Well rounded, (*c*) Sharp edge, (*d*) Re-entrant.**

### 3.6.2 Confined Flows

*The continuity equation states that mass cannot be created or destroyed.*

In many cases the fluid is physically constrained within a device so that its pressure cannot be prescribed a priori as was done for the free jet examples above. Such cases include nozzles and pipes of variable diameter for which the fluid velocity changes because the flow area is different from one section to another. For these situations it is necessary to use the concept of conservation of mass (the continuity equation) along with the Bernoulli equation. The derivation and use of this equation are discussed in detail in Chapters 4 and 5. For the needs of this chapter we can use a simplified form of the continuity equation obtained from the following intuitive arguments. Consider a fluid flowing through a fixed volume (such as a syringe) that has one inlet and one outlet as shown in Fig. 3.15*a*. If the flow is steady so that there is no additional accumulation of fluid within the volume, the rate at which the fluid flows into the volume must equal the rate at which it flows out of the volume (otherwise, mass would not be conserved).




The *mass flowrate* from an outlet, $\dot{m}$ (slugs/s or kg/s), is given by $\dot{m} = \rho Q$, where $Q$ (ft³/s or m³/s) is the ***volume flowrate.*** If the outlet area is $A$ and the fluid flows across this area (normal to the area) with an average velocity $V$, then the volume of the fluid crossing this area in a time interval $\delta t$ is $VA\,\delta t$, equal to that in a volume of length $V\,\delta t$ and cross-sectional area $A$ (see Fig. 3.15*b*). Hence, the volume flowrate (volume per unit time) is $Q = VA$. Thus, $\dot{m} = \rho VA$. To conserve mass, the inflow rate must equal the outflow rate. If the inlet is designated as (1) and the outlet as (2), it follows that $\dot{m}_1 = \dot{m}_2$. Thus, conservation of mass requires

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

If the density remains constant, then $\rho_1 = \rho_2$, and the above becomes the ***continuity equation*** for incompressible flow

$$\boxed{A_1 V_1 = A_2 V_2, \text{ or } Q_1 = Q_2} \qquad \textbf{(3.19)}$$

For example, if as shown by the figure in the margin the outlet flow area is one-half the size of the inlet flow area, it follows that the outlet velocity is twice that of the inlet velocity, since

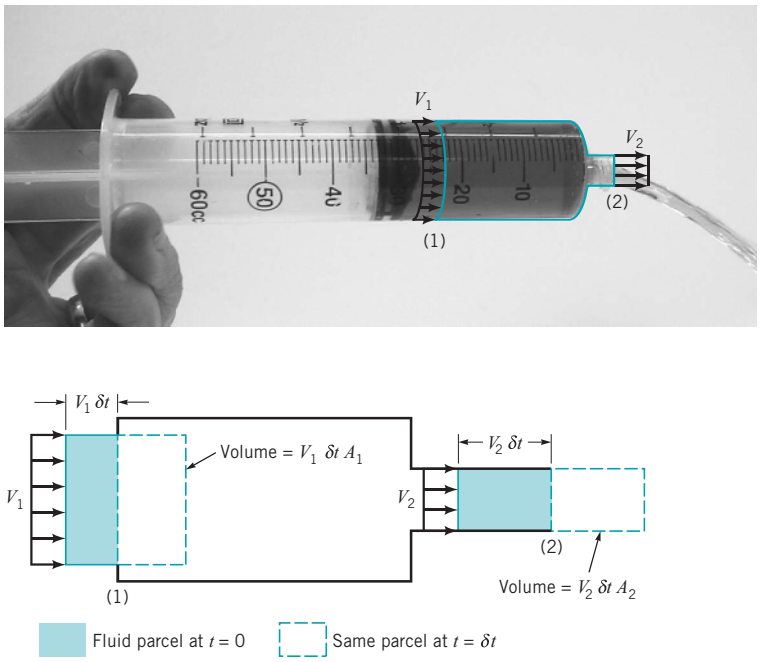



**FIGURE 3.15** **(*a*) Flow through a syringe. (*b*) Steady flow into and out of a volume.**

$V_2 = A_1V_1/A_2 = 2V_1$. The use of the Bernoulli equation and the flowrate equation (continuity equation) is demonstrated by Example 3.7.

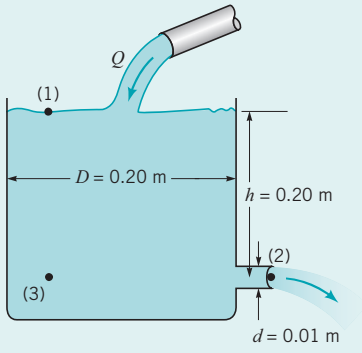
## EXAMPLE 3.7 Flow from a Tank—Gravity

**GIVEN** A stream of refreshing beverage of diameter $d = 0.01$ m flows steadily from the cooler of diameter $D = 0.20$ m as shown in Figs. E3.7*a* and *b*.

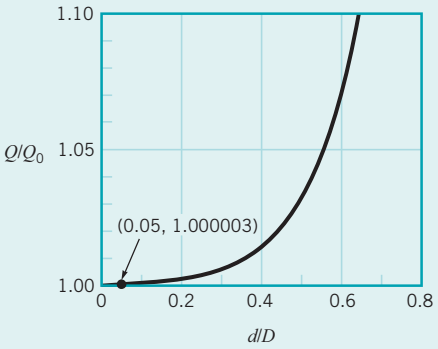
**FIND** Determine the flowrate, $Q$, from the bottle into the cooler if the depth of beverage in the cooler is to remain constant at $h = 0.20$ m



(*a*)




(*b*)




(*c*)

**FIGURE E3.7**

## SOLUTION

For steady, inviscid, incompressible flow, the Bernoulli equation applied between points (1) and (2) is

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2 \qquad \textbf{(1)}$$

With the assumptions that $p_1 = p_2 = 0$, $z_1 = h$, and $z_2 = 0$, Eq. 1 becomes

$$\tfrac{1}{2}V_1^2 + gh = \tfrac{1}{2}V_2^2 \qquad \textbf{(2)}$$

Although the liquid level remains constant ($h$ = constant), there is an average velocity, $V_1$, across section (1) because of the flow from the tank. From Eq. 3.19 for steady incompressible flow, conservation of mass requires $Q_1 = Q_2$, where $Q = AV$. Thus, $A_1V_1 = A_2V_2$, or

$$\frac{\pi}{4}D^2V_1 = \frac{\pi}{4}d^2V_2$$

Hence,

$$V_1 = \left(\frac{d}{D}\right)^2 V_2 \qquad \textbf{(3)}$$

Equations 1 and 3 can be combined to give

$$V_2 = \sqrt{\frac{2gh}{1 - (d/D)^4}} = \sqrt{\frac{2(9.81\ \text{m/s}^2)(0.20\ \text{m})}{1 - (0.01\ \text{m}/0.20\ \text{m})^4}} = 1.98\ \text{m/s}$$

Thus,

$$Q = A_1V_1 = A_2V_2 = \frac{\pi}{4}(0.01\ \text{m})^2(1.98\ \text{m/s})$$
$$= 1.56 \times 10^{-4}\ \text{m}^3/\text{s} \qquad \textbf{(Ans)}$$

**COMMENTS** Note that this problem was solved using points (1) and (2) located at the free surface and the exit of the pipe, respectively. Although this was convenient (because most of the variables are known at those points), other points could be selected and the same result would be obtained. For example, consider points (1) and (3) as indicated in Fig. E3.7*b*. At (3), located sufficiently far from the tank exit, $V_3 = 0$ and $z_3 = z_2 = 0$. Also, $p_3 = \gamma h$ since the pressure is hydrostatic sufficiently far from the exit. Use of this information in the Bernoulli equation applied between (1) and (3) gives the exact same result as obtained using it between (1) and (2). The only difference is that the elevation head, $z_1 = h$, has been interchanged with the pressure head at (3), $p_3/\gamma = h$.

In this example we have not neglected the kinetic energy of the water in the tank ($V_1 \neq 0$). If the tank diameter is large compared to the jet diameter ($D \gg d$), Eq. 3 indicates that $V_1 \ll V_2$ and the assumption that $V_1 \approx 0$ would be reasonable. The error associated with this assumption can be seen by calculating the ratio of the flowrate assuming $V_1 \neq 0$, denoted $Q$, to that assuming $V_1 = 0$, denoted $Q_0$. This ratio, written as

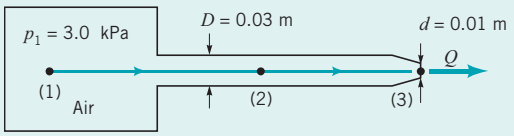
$$\frac{Q}{Q_0} = \frac{V_2}{V_2|_{D=\infty}} = \frac{\sqrt{2gh/[1 - (d/D)^4]}}{\sqrt{2gh}} = \frac{1}{\sqrt{1 - (d/D)^4}}$$

is plotted in Fig. E3.7*c*. With $0 < d/D < 0.4$ it follows that $1 < Q/Q_0 \lesssim 1.01$, and the error in assuming $V_1 = 0$ is less than 1%. For this example with $d/D = 0.01\ \text{m}/0.20\ \text{m} = 0.05$, it follows that $Q/Q_0 = 1.000003$. Thus, it is often reasonable to assume $V_1 = 0$.

The fact that a kinetic energy change is often accompanied by a change in pressure is shown by Example 3.8.

# EXAMPLE 3.8 Flow from a Tank—Pressure

**GIVEN** Air flows steadily from a tank, through a hose of diameter $D = 0.03$ m, and exits to the atmosphere from a nozzle of diameter $d = 0.01$ m as shown in Fig. E3.8. The pressure in the tank remains constant at 3.0 kPa (gage) and the atmospheric conditions are standard temperature and pressure.




■ FIGURE E3.8

**FIND** Determine the flowrate and the pressure in the hose.

## SOLUTION

If the flow is assumed steady, inviscid, and incompressible, we can apply the Bernoulli equation along the streamline from (1) to (2) to (3) as

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2$$
$$= p_3 + \tfrac{1}{2}\rho V_3^2 + \gamma z_3$$

With the assumption that $z_1 = z_2 = z_3$ (horizontal hose), $V_1 = 0$ (large tank), and $p_3 = 0$ (free jet), this becomes

$$V_3 = \sqrt{\frac{2p_1}{\rho}}$$

and

$$p_2 = p_1 - \tfrac{1}{2}\rho V_2^2 \qquad (1)$$

The density of the air in the tank is obtained from the perfect gas law, using standard absolute pressure and temperature, as

$$\begin{aligned}\rho &= \frac{p_1}{RT_1} \\ &= [(3.0 + 101)\ \text{kN/m}^2] \\ &\quad \times \frac{10^3\ \text{N/kN}}{(286.9\ \text{N} \cdot \text{m/kg} \cdot \text{K})(15 + 273)\text{K}} \\ &= 1.26\ \text{kg/m}^3\end{aligned}$$

Thus, we find that

$$V_3 = \sqrt{\frac{2(3.0 \times 10^3\ \text{N/m}^2)}{1.26\ \text{kg/m}^3}} = 69.0\ \text{m/s}$$

or

$$\begin{aligned}Q = A_3 V_3 &= \frac{\pi}{4} d^2 V_3 = \frac{\pi}{4}(0.01\ \text{m})^2 (69.0\ \text{m/s}) \\ &= 0.00542\ \text{m}^3/\text{s} \qquad \textbf{(Ans)}\end{aligned}$$

The pressure within the hose can be obtained from Eq. 1 and the continuity equation (Eq. 3.19)

$$A_2 V_2 = A_3 V_3$$

Hence,

$$\begin{aligned}V_2 = A_3 V_3 / A_2 &= \left(\frac{d}{D}\right)^2 V_3 \\ &= \left(\frac{0.01\ \text{m}}{0.03\ \text{m}}\right)^2 (69.0\ \text{m/s}) = 7.67\ \text{m/s}\end{aligned}$$

and from Eq. 1

$$\begin{aligned}p_2 &= 3.0 \times 10^3\ \text{N/m}^2 - \tfrac{1}{2}(1.26\ \text{kg/m}^3)(7.67\ \text{m/s})^2 \\ &= (3000 - 37.1)\text{N/m}^2 = 2963\ \text{N/m}^2 \qquad \textbf{(Ans)}\end{aligned}$$

**COMMENTS** Note that the value of $V_3$ is determined strictly by the value of $p_1$ (and the assumptions involved in the Bernoulli equation), independent of the "shape" of the nozzle. The pressure head within the tank, $p_1/\gamma = (3.0\ \text{kPa})/(9.81\ \text{m/s}^2)(1.26\ \text{kg/m}^3) = 243$ m, is converted to the velocity head at the exit, $V_2^2/2g = (69.0\ \text{m/s})^2/(2 \times 9.81\ \text{m/s}^2) = 243$ m. Although we used gage pressure in the Bernoulli equation ($p_3 = 0$), we had to use absolute pressure in the perfect gas law when calculating the density.

In the absence of viscous effects the pressure throughout the hose is constant and equal to $p_2$. Physically, the decreases in pressure from $p_1$ to $p_2$ to $p_3$ accelerate the air and increase its kinetic energy from zero in the tank to an intermediate value in the hose and finally to its maximum value at the nozzle exit. Since the air velocity in the nozzle exit is nine times that in the hose, most of the pressure drop occurs across the nozzle ($p_1 = 3000\ \text{N/m}^2$, $p_2 = 2963\ \text{N/m}^2$, and $p_3 = 0$).

Since the pressure change from (1) to (3) is not too great [i.e., in terms of absolute pressure $(p_1 - p_3)/p_1 = 3.0/101 = 0.03$], it follows from the perfect gas law that the density change is also not significant. Hence, the incompressibility assumption is reasonable for this problem. If the tank pressure were considerably larger or if viscous effects were important, the above results would be incorrect.

## Fluids in the News

**Hi-tech inhaler** The term inhaler often brings to mind a treatment for asthma or bronchitis. Work is underway to develop a family of inhalation devices that can do more than treat respiratory ailments. They will be able to deliver medication for diabetes and other conditions by spraying it to reach the bloodstream through the lungs. The concept is to make the spray droplets fine enough to penetrate to the lungs' tiny sacs, the alveoli, where exchanges between blood and the outside world take place. This is accomplished by use of a laser-machined *nozzle* containing an array of very fine holes that cause the liquid to divide into a mist of micron-scale droplets. The device fits the hand and accepts a disposable strip that contains the medicine solution sealed inside a blister of laminated plastic and the nozzle. An electrically actuated piston drives the liquid from its reservoir through the nozzle array and into the respiratory system. To take the medicine, the patient breathes through the device and a differential pressure transducer in the inhaler senses when the patient's breathing has reached the best condition for receiving the medication. At that point, the piston is automatically triggered.

In many situations the combined effects of kinetic energy, pressure, and gravity are important. Example 3.9 illustrates this.

## EXAMPLE 3.9 Flow in a Variable Area Pipe

**GIVEN** Water flows through a pipe reducer as is shown in Fig. E3.9. The static pressures at (1) and (2) are measured by the inverted U-tube manometer containing oil of specific gravity, $SG$, less than one.

**FIND** Determine the manometer reading, $h$.

CMS Cepcor EX 1046 Page 25
CMS Cepcor v. Sandvik IPR2024-01380

## SOLUTION

With the assumptions of steady, inviscid, incompressible flow, the Bernoulli equation can be written as

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2$$

The continuity equation (Eq. 3.19) provides a second relationship between $V_1$ and $V_2$ if we assume the velocity profiles are uniform at those two locations and the fluid incompressible:

$$Q = A_1 V_1 = A_2 V_2$$

By combining these two equations we obtain

$$p_1 - p_2 = \gamma(z_2 - z_1) + \tfrac{1}{2}\rho V_2^2[1 - (A_2/A_1)^2] \qquad \textbf{(1)}$$

This pressure difference is measured by the manometer and can be determined by using the pressure–depth ideas developed in Chapter 2. Thus,

$$p_1 - \gamma(z_2 - z_1) - \gamma\ell - \gamma h + SG\,\gamma h + \gamma\ell = p_2$$

or

$$p_1 - p_2 = \gamma(z_2 - z_1) + (1 - SG)\gamma h \qquad \textbf{(2)}$$

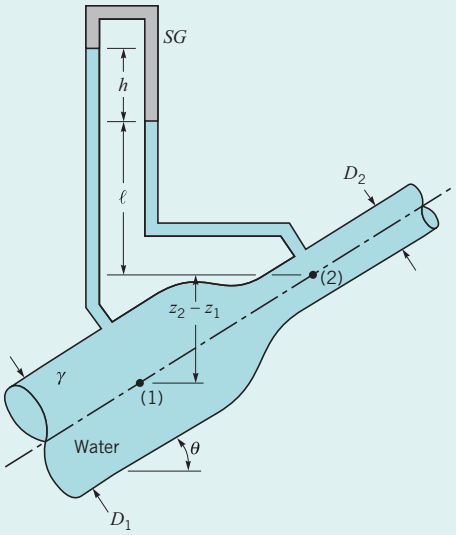
As discussed in Chapter 2, this pressure difference is neither merely $\gamma h$ nor $\gamma(h + z_1 - z_2)$.

Equations 1 and 2 can be combined to give the desired result as follows:

$$(1 - SG)\gamma h = \frac{1}{2}\rho V_2^2\left[1 - \left(\frac{A_2}{A_1}\right)^2\right]$$

or since $V_2 = Q/A_2$

$$h = (Q/A_2)^2 \frac{1 - (A_2/A_1)^2}{2g(1 - SG)} \qquad \textbf{(Ans)}$$

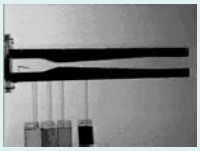



■ FIGURE E3.9

**COMMENT** The difference in elevation, $z_1 - z_2$, was not needed because the change in elevation term in the Bernoulli equation exactly cancels the elevation term in the manometer equation. However, the pressure difference, $p_1 - p_2$, depends on the angle $\theta$, because of the elevation, $z_1 - z_2$, in Eq. 1. Thus, for a given flowrate, the pressure difference, $p_1 - p_2$, as measured by a pressure gage would vary with $\theta$, but the manometer reading, $h$, would be independent of $\theta$.



*V3.10 Venturi channel*



*Cavitation occurs when the pressure is reduced to the vapor pressure.*

In general, an increase in velocity is accompanied by a decrease in pressure. For example, the velocity of the air flowing over the top surface of an airplane wing is, on the average, faster than that flowing under the bottom surface. Thus, the net pressure force is greater on the bottom than on the top—the wing generates a lift.

If the differences in velocity are considerable, the differences in pressure can also be considerable. For flows of gases, this may introduce compressibility effects as discussed in Section 3.8 and Chapter 11. For flows of liquids, this may result in ***cavitation***, a potentially dangerous situation that results when the liquid pressure is reduced to the vapor pressure and the liquid "boils."

As discussed in Chapter 1, the vapor pressure, $p_v$, is the pressure at which vapor bubbles form in a liquid. It is the pressure at which the liquid starts to boil. Obviously this pressure depends on the type of liquid and its temperature. For example, water, which boils at 212 °F at standard atmospheric pressure, 14.7 psia, boils at 80 °F if the pressure is 0.507 psia. That is, $p_v = 0.507$ psia at 80 °F and $p_v = 14.7$ psia at 212 °F. (See Tables B.1 and B.2.)

One way to produce cavitation in a flowing liquid is noted from the Bernoulli equation. If the fluid velocity is increased (for example, by a reduction in flow area as shown in Fig. 3.16) the pressure will decrease. This pressure decrease (needed to accelerate the fluid through the constriction) can be large enough so that the pressure in the liquid is reduced to its vapor pressure. A simple example of cavitation can be demonstrated with an ordinary garden hose. If the hose is "kinked," a restriction in the flow area in some ways analogous to that shown in Fig. 3.16 will result. The water velocity through this restriction will be relatively large. With a sufficient amount of restriction the sound of the flowing water will change—a definite "hissing" sound is produced. This sound is a result of cavitation.

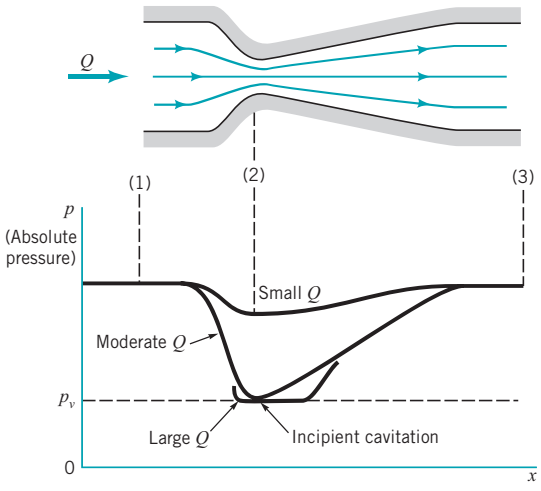



**FIGURE 3.16** Pressure variation and cavitation in a variable area pipe.

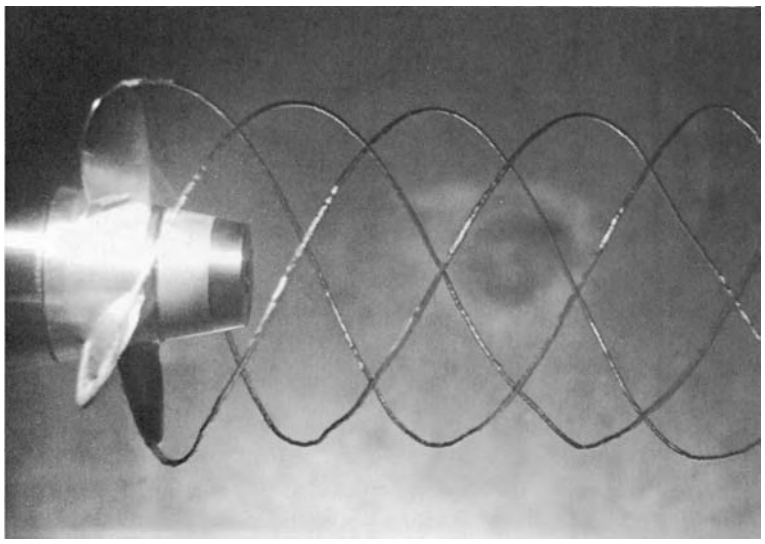


**FIGURE 3.17** Tip cavitation from a propeller. (Photograph courtesy of Garfield Thomas Water Tunnel, Pennsylvania State University.)

*Cavitation can cause damage to equipment.*

In such situations boiling occurs (though the temperature need not be high), vapor bubbles form, and then they collapse as the fluid moves into a region of higher pressure (lower velocity). This process can produce dynamic effects (imploding) that cause very large pressure transients in the vicinity of the bubbles. Pressures as large as 100,000 psi (690 MPa) are believed to occur. If the bubbles collapse close to a physical boundary they can, over a period of time, cause damage to the surface in the cavitation area. Tip cavitation from a propeller is shown in Fig. 3.17. In this case the high-speed rotation of the propeller produced a corresponding low pressure on the propeller. Obviously, proper design and use of equipment are needed to eliminate cavitation damage.

## EXAMPLE 3.10 Siphon and Cavitation

**GIVEN** A liquid can be siphoned from a container as shown in Fig. E3.10*a* provided the end of the tube, point (3), is below the free surface in the container, point (1), and the maximum elevation of the tube, point (2), is "not too great." Consider water at 60° F being siphoned from a large tank through a constant diameter hose as shown in Fig. E3.10*b*. The end of the siphon is 5 ft below the bottom of the tank, and the atmospheric pressure is 14.7 psia.

**FIND** Determine the maximum height of the hill, $H$, over which the water can be siphoned without cavitation occurring.

## SOLUTION

If the flow is steady, inviscid, and incompressible we can apply the Bernoulli equation along the streamline from (1) to (2) to (3) as follows:

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2 = p_3 + \tfrac{1}{2}\rho V_3^2 + \gamma z_3 \qquad \textbf{(1)}$$

With the tank bottom as the datum, we have $z_1 = 15$ ft, $z_2 = H$, and $z_3 = -5$ ft. Also, $V_1 = 0$ (large tank), $p_1 = 0$ (open tank), $p_3 = 0$ (free jet), and from the continuity equation $A_2V_2 = A_3V_3$, or because the hose is constant diameter, $V_2 = V_3$. Thus, the speed of the fluid in the hose is determined from Eq. 1 to be

$$V_3 = \sqrt{2g(z_1 - z_3)} = \sqrt{2(32.2 \text{ ft/s}^2)[15 - (-5)] \text{ ft}}$$
$$= 35.9 \text{ ft/s} = V_2$$

Use of Eq. 1 between points (1) and (2) then gives the pressure $p_2$ at the top of the hill as

$$p_2 = p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 - \tfrac{1}{2}\rho V_2^2 - \gamma z_2$$
$$= \gamma(z_1 - z_2) - \tfrac{1}{2}\rho V_2^2 \qquad \textbf{(2)}$$

From Table B.1, the vapor pressure of water at 60 °F is 0.256 psia. Hence, for incipient cavitation the lowest pressure in the system will be $p = 0.256$ psia. Careful consideration of Eq. 2 and Fig. E3.10*b* will show that this lowest pressure will occur at the top of the hill. Since we have used gage pressure at point (1) ($p_1 = 0$), we must use gage pressure at point (2) also. Thus, $p_2 = 0.256 - 14.7 = -14.4$ psi and Eq. 2 gives

$$(-14.4 \text{ lb/in.}^2)(144 \text{ in.}^2/\text{ft}^2) = (62.4 \text{ lb/ft}^3)(15 - H)\text{ft} - \tfrac{1}{2}(1.94 \text{ slugs/ft}^3)(35.9 \text{ ft/s})^2$$

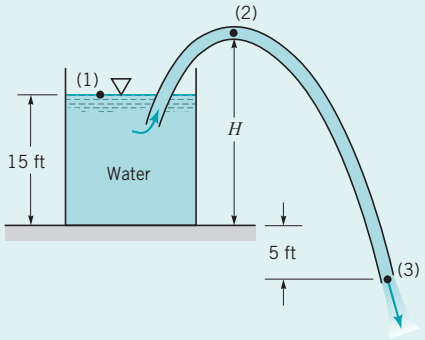
or

$$H = 28.2 \text{ ft} \qquad \textbf{(Ans)}$$

For larger values of $H$, vapor bubbles will form at point (2) and the siphon action may stop.




■ FIGURE E3.10*a*



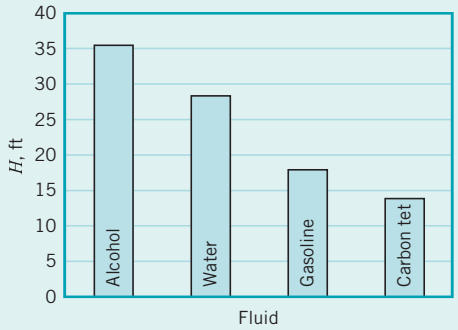

■ FIGURE E3.10*b*

**COMMENTS** Note that we could have used absolute pressure throughout ($p_2 = 0.256$ psia and $p_1 = 14.7$ psia) and obtained the same result. The lower the elevation of point (3), the larger the flowrate and, therefore, the smaller the value of $H$ allowed.

We could also have used the Bernoulli equation between (2) and (3), with $V_2 = V_3$, to obtain the same value of $H$. In this case it would not have been necessary to determine $V_2$ by use of the Bernoulli equation between (1) and (3).

The above results are independent of the diameter and length of the hose (provided viscous effects are not important). Proper design of the hose (or pipe) is needed to ensure that it will not collapse due to the large pressure difference (vacuum) between the inside and outside of the hose.

By using the fluid properties listed in Table 1.5 and repeating the calculations for various fluids, the results shown in Fig. E3.10*c* are obtained. The value of $H$ is a function of both the specific weight of the fluid, $\gamma$, and its vapor pressure, $p_v$.




■ FIGURE E3.10*c*

### 3.6.3 Flowrate Measurement

Many types of devices using principles involved in the Bernoulli equation have been developed to measure fluid velocities and flowrates. The Pitot-static tube discussed in Section 3.5 is an example. Other examples discussed below include devices to measure flowrates in pipes and

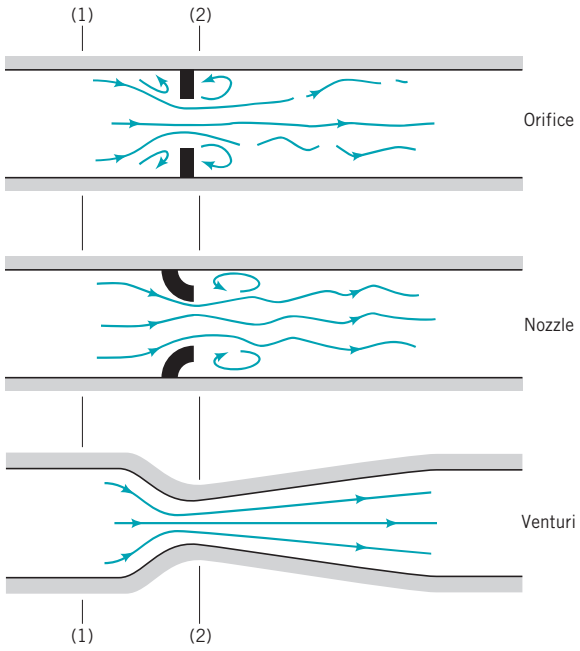



**FIGURE 3.18** Typical devices for measuring flowrate in pipes.

conduits and devices to measure flowrates in open channels. In this chapter we will consider "ideal" ***flow meters***—those devoid of viscous, compressibility, and other "real-world" effects. Corrections for these effects are discussed in Chapters 8 and 10. Our goal here is to understand the basic operating principles of these simple flow meters.

An effective way to measure the flowrate through a pipe is to place some type of restriction within the pipe as shown in Fig. 3.18 and to measure the pressure difference between the low-velocity, high-pressure upstream section (1), and the high-velocity, low-pressure downstream section (2). Three commonly used types of flow meters are illustrated: the *orifice meter*, the *nozzle meter*, and the *Venturi meter*. The operation of each is based on the same physical principles—an increase in velocity causes a decrease in pressure. The difference between them is a matter of cost, accuracy, and how closely their actual operation obeys the idealized flow assumptions.

We assume the flow is horizontal ($z_1 = z_2$), steady, inviscid, and incompressible between points (1) and (2). The Bernoulli equation becomes

$$p_1 + \tfrac{1}{2}\rho V_1^2 = p_2 + \tfrac{1}{2}\rho V_2^2$$

(The effect of nonhorizontal flow can be incorporated easily by including the change in elevation, $z_1 - z_2$, in the Bernoulli equation.)

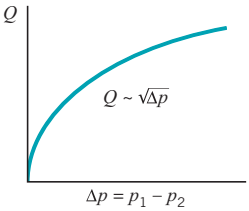
If we assume the velocity profiles are uniform at sections (1) and (2), the continuity equation (Eq. 3.19) can be written as

$$Q = A_1 V_1 = A_2 V_2$$

*The flowrate varies as the square root of the pressure difference across the flow meter.*

where $A_2$ is the small ($A_2 < A_1$) flow area at section (2). Combination of these two equations results in the following theoretical flowrate

$$Q = A_2 \sqrt{\frac{2(p_1 - p_2)}{\rho[1 - (A_2/A_1)^2]}} \tag{3.20}$$



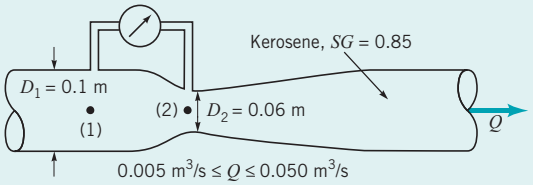

Thus, as shown by the figure in the margin, for a given flow geometry ($A_1$ and $A_2$) the flowrate can be determined if the pressure difference, $p_1 - p_2$, is measured. The actual measured flowrate, $Q_{\text{actual}}$, will be smaller than this theoretical result because of various differences between the "real world" and the assumptions used in the derivation of Eq. 3.20. These differences (which are quite consistent and may be as small as 1 to 2% or as large as 40%, depending on the geometry used) can be accounted for by using an empirically obtained discharge coefficient as discussed in Section 8.6.1.

CMS Cepcor EX 1046 Page 29
CMS Cepcor v. Sandvik IPR2024-01380

## EXAMPLE 3.11 Venturi Meter

**GIVEN** Kerosene ($SG = 0.85$) flows through the Venturi meter shown in Fig. E3.11*a* with flowrates between 0.005 and 0.050 $m^3/s$.

**FIND** Determine the range in pressure difference, $p_1 - p_2$, needed to measure these flowrates.




■ FIGURE E3.11*a*

### SOLUTION

If the flow is assumed to be steady, inviscid, and incompressible, the relationship between flowrate and pressure is given by Eq. 3.20. This can be rearranged to give

$$p_1 - p_2 = \frac{Q^2\rho[1 - (A_2/A_1)^2]}{2A_2^2}$$

With the density of the flowing fluid

$$\rho = SG\,\rho_{H_2O} = 0.85(1000\ \text{kg/m}^3) = 850\ \text{kg/m}^3$$

and the area ratio

$$A_2/A_1 = (D_2/D_1)^2 = (0.06\ \text{m}/0.10\ \text{m})^2 = 0.36$$

the pressure difference for the smallest flowrate is

$$p_1 - p_2 = (0.005\ \text{m}^3/\text{s})^2(850\ \text{kg/m}^3)\frac{(1 - 0.36^2)}{2\,[(\pi/4)(0.06\ \text{m})^2]^2}$$
$$= 1160\ \text{N/m}^2 = 1.16\ \text{kPa}$$

Likewise, the pressure difference for the largest flowrate is

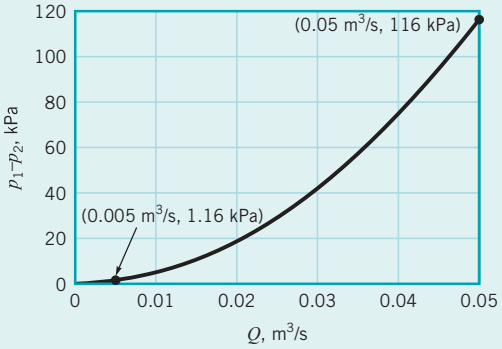
$$p_1 - p_2 = (0.05)^2(850)\frac{(1 - 0.36^2)}{2[(\pi/4)(0.06)^2]^2}$$
$$= 1.16 \times 10^5\ \text{N/m}^2 = 116\ \text{kPa}$$

Thus,

$$1.16\ \text{kPa} \le p_1 - p_2 \le 116\ \text{kPa} \qquad \textbf{(Ans)}$$

**COMMENTS** These values represent the pressure differences for inviscid, steady, incompressible conditions. The ideal results presented here are independent of the particular flow meter geometry—an orifice, nozzle, or Venturi meter (see Fig. 3.18).

It is seen from Eq. 3.20 that the flowrate varies as the square root of the pressure difference. Hence, as indicated by the numerical results and shown in Fig. E3.11*b*, a 10-fold increase in flowrate requires a 100-fold increase in pressure difference. This nonlinear relationship can cause difficulties when measuring flowrates over a wide range of values. Such measurements would require pressure transducers with a wide range of operation. An alternative is to use two flow meters in parallel—one for the larger and one for the smaller flowrate ranges.




■ FIGURE E3.11*b*

Other flow meters based on the Bernoulli equation are used to measure flowrates in open channels such as flumes and irrigation ditches. Two of these devices, the *sluice gate* and the *sharp-crested weir*, are discussed below under the assumption of steady, inviscid, incompressible flow. These and other open-channel flow devices are discussed in more detail in Chapter 10.

Sluice gates like those shown in Fig. 3.19*a* are often used to regulate and measure the flowrate in open channels. As indicated in Fig. 3.19*b*, the flowrate, $Q$, is a function of the water depth upstream, $z_1$, the width of the gate, $b$, and the gate opening, $a$. Application of the Bernoulli equation and continuity equation between points (1) and (2) can provide a good approximation to the actual flowrate obtained. We assume the velocity profiles are uniform sufficiently far upstream and downstream of the gate.

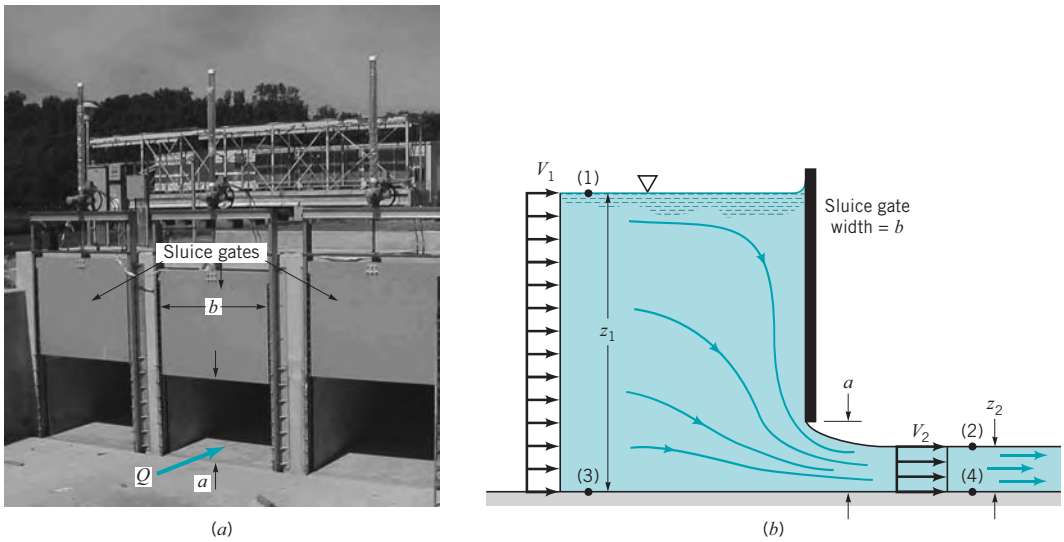



**FIGURE 3.19** Sluice gate geometry. (Photograph courtesy of Plasti-Fab, Inc.)

Thus, we apply the Bernoulli equation between points on the free surfaces at (1) and (2) to give

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2$$

Also, if the gate is the same width as the channel so that $A_1 = bz_1$ and $A_2 = bz_2$, the continuity equation gives

$$Q = A_1 V_1 = bV_1 z_1 = A_2 V_2 = bV_2 z_2$$

*The flowrate under a sluice gate depends on the water depths on either side of the gate.*

With the fact that $p_1 = p_2 = 0$, these equations can be combined and rearranged to give the flowrate as

$$Q = z_2 b \sqrt{\frac{2g(z_1 - z_2)}{1 - (z_2/z_1)^2}} \tag{3.21}$$

In the limit of $z_1 \gg z_2$ this result simply becomes

$$Q = z_2 b \sqrt{2gz_1}$$

This limiting result represents the fact that if the depth ratio, $z_1/z_2$, is large, the kinetic energy of the fluid upstream of the gate is negligible and the fluid velocity after it has fallen a distance $(z_1 - z_2) \approx z_1$ is approximately $V_2 = \sqrt{2gz_1}$.

The results of Eq. 3.21 could also be obtained by using the Bernoulli equation between points (3) and (4) and the fact that $p_3 = \gamma z_1$ and $p_4 = \gamma z_2$ since the streamlines at these sections are straight. In this formulation, rather than the potential energies at (1) and (2), we have the pressure contributions at (3) and (4).

The downstream depth, $z_2$, not the gate opening, $a$, was used to obtain the result of Eq. 3.21. As was discussed relative to flow from an orifice (Fig. 3.14), the fluid cannot turn a sharp 90° corner. A vena contracta results with a contraction coefficient, $C_c = z_2/a$, less than 1. Typically $C_c$ is approximately 0.61 over the depth ratio range of $0 < a/z_1 < 0.2$. For larger values of $a/z_1$ the value of $C_c$ increases rapidly.

## EXAMPLE 3.12 Sluice Gate

**GIVEN** Water flows under the sluice gate shown in Fig. E3.12*a*.

**FIND** Determine the approximate flowrate per unit width of the channel.

## SOLUTION

Under the assumptions of steady, inviscid, incompressible flow, we can apply Eq. 3.21 to obtain $Q/b$, the flowrate per unit width, as

$$\frac{Q}{b} = z_2\sqrt{\frac{2g(z_1 - z_2)}{1 - (z_2/z_1)^2}}$$

In this instance $z_1 = 5.0$ m and $a = 0.80$ m so the ratio $a/z_1 = 0.16 < 0.20$, and we can assume that the contraction coefficient is approximately $C_c = 0.61$. Thus, $z_2 = C_c a = 0.61\,(0.80\text{ m}) = 0.488$ m and we obtain the flowrate

$$\frac{Q}{b} = (0.488\text{ m})\sqrt{\frac{2(9.81\text{ m/s}^2)(5.0\text{ m} - 0.488\text{ m})}{1 - (0.488\text{ m}/5.0\text{ m})^2}}$$

$$= 4.61\text{ m}^2/\text{s} \qquad \textbf{(Ans)}$$

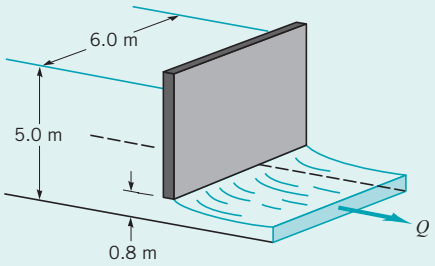
**COMMENT** If we consider $z_1 \gg z_2$ and neglect the kinetic energy of the upstream fluid, we would have

$$\frac{Q}{b} = z_2\sqrt{2gz_1} = 0.488\text{ m}\sqrt{2(9.81\text{ m/s}^2)(5.0\text{ m})}$$

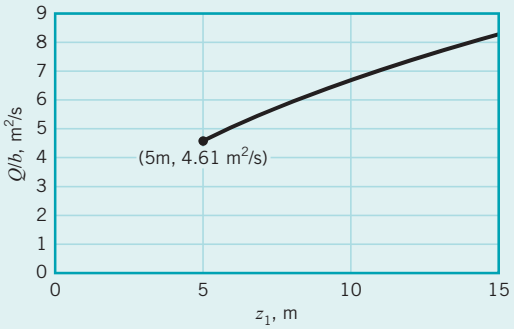
$$= 4.83\text{ m}^2/\text{s}$$

In this case the difference in $Q$ with or without including $V_1$ is not too significant because the depth ratio is fairly large ($z_1/z_2 = 5.0/0.488 = 10.2$). Thus, it is often reasonable to neglect the kinetic energy upstream from the gate compared to that downstream of it.

By repeating the calculations for various flow depths, $z_1$, the results shown in Fig. E3.12*b* are obtained. Note that the flowrate is not directly proportional to the flow depth. Thus, for example, if during flood conditions the upstream depth doubled from $z_1 = 5$ m to $z_1 = 10$ m, the flowrate per unit width of the channel would not double, but would increase only from 4.61 m$^2$/s to 6.67 m$^2$/s.



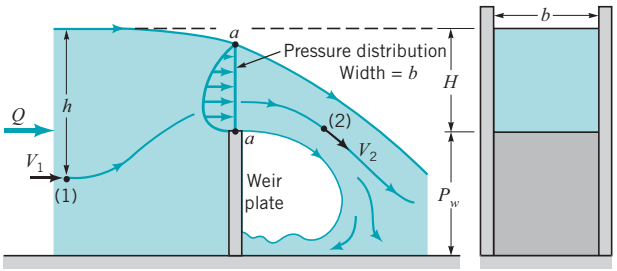

■ FIGURE E3.12*a*




■ FIGURE E3.12*b*

Another device used to measure flow in an open channel is a *weir*. A typical rectangular, sharp-crested weir is shown in Fig. 3.20. For such devices the flowrate of liquid over the top of the weir plate is dependent on the weir height, $P_w$, the width of the channel, $b$, and the head, $H$, of the water above the top of the weir. Application of the Bernoulli equation can provide a simple approximation of the flowrate expected for these situations, even though the actual flow is quite complex.

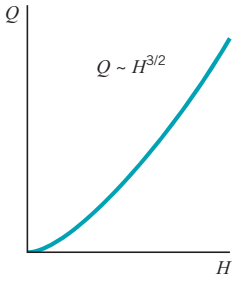
Between points (1) and (2) the pressure and gravitational fields cause the fluid to accelerate from velocity $V_1$ to velocity $V_2$. At (1) the pressure is $p_1 = \gamma h$, while at (2) the pressure is essentially atmospheric, $p_2 = 0$. Across the curved streamlines directly above the top of the weir plate (section $a$–$a$), the pressure changes from atmospheric on the top surface to some maximum value within the fluid stream and then to atmospheric again at the bottom surface. This distribution is indicated in Fig. 3.20. Such a pressure distribution, combined with the streamline curvature and gravity, produces a rather nonuniform velocity profile across this section. This velocity distribution can be obtained from experiments or a more advanced theory.




■ FIGURE 3.20
**Rectangular, sharp-crested weir geometry.**

For now, we will take a very simple approach and assume that the weir flow is similar in many respects to an orifice-type flow with a free streamline. In this instance we would expect the average velocity across the top of the weir to be proportional to $\sqrt{2gH}$ and the flow area for this rectangular weir to be proportional to $Hb$. Hence, it follows that

$$Q = C_1 Hb\sqrt{2gH} = C_1 b\sqrt{2g}\,H^{3/2}$$




where $C_1$ is a constant to be determined.

Simple use of the Bernoulli equation has provided a method to analyze the relatively complex flow over a weir. The correct functional dependence of $Q$ on $H$ has been obtained ($Q \sim H^{3/2}$, as indicated by the figure in the margin), but the value of the coefficient $C_1$ is unknown. Even a more advanced analysis cannot predict its value accurately. As is discussed in Chapter 10, experiments are used to determine the value of $C_1$.

## EXAMPLE 3.13 Weir

**GIVEN** Water flows over a triangular weir, as is shown in Fig. E3.13.

**FIND** Based on a simple analysis using the Bernoulli equation, determine the dependence of the flowrate on the depth $H$. If the flowrate is $Q_0$ when $H = H_0$, estimate the flowrate when the depth is increased to $H = 3H_0$.

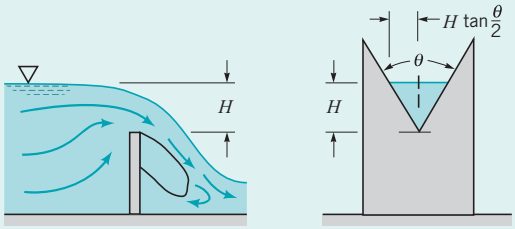



FIGURE E3.13

### SOLUTION

With the assumption that the flow is steady, inviscid, and incompressible, it is reasonable to assume from Eq. 3.18 that the average speed of the fluid over the triangular notch in the weir plate is proportional to $\sqrt{2gH}$. Also, the flow area for a depth of $H$ is $H[H\tan(\theta/2)]$. The combination of these two ideas gives

$$Q = AV = H^2\tan\frac{\theta}{2}\left(C_2\sqrt{2gH}\right) = C_2\tan\frac{\theta}{2}\sqrt{2g}\,H^{5/2} \qquad \textbf{(Ans)}$$

where $C_2$ is an unknown constant to be determined experimentally.

Thus, an increase in the depth by a factor of three (from $H_0$ to $3H_0$) results in an increase of the flowrate by a factor of

$$\frac{Q_{3H_0}}{Q_{H_0}} = \frac{C_2\tan(\theta/2)\sqrt{2g}\,(3H_0)^{5/2}}{C_2\tan(\theta/2)\sqrt{2g}\,(H_0)^{5/2}} = 15.6 \qquad \textbf{(Ans)}$$

**COMMENT** Note that for a triangular weir the flowrate is proportional to $H^{5/2}$, whereas for the rectangular weir discussed above, it is proportional to $H^{3/2}$. The triangular weir can be accurately used over a wide range of flowrates.

## 3.7 The Energy Line and the Hydraulic Grade Line

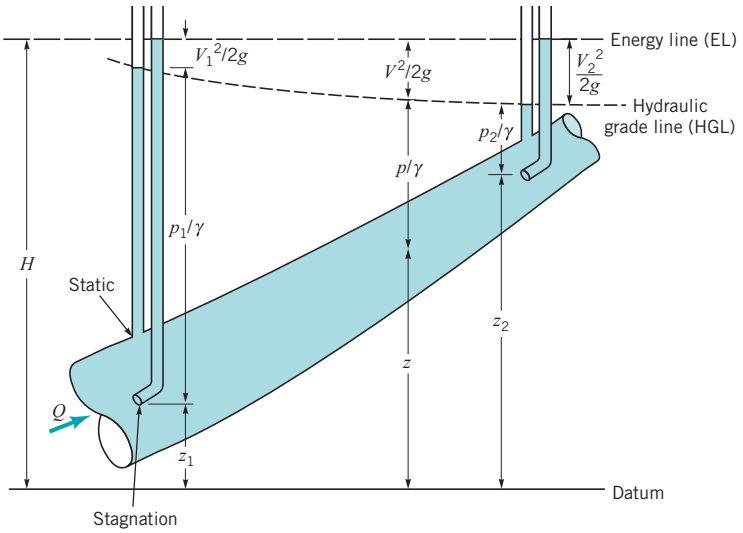
*The hydraulic grade line and energy line are graphical forms of the Bernoulli equation.*

As was discussed in Section 3.4, the Bernoulli equation is actually an energy equation representing the partitioning of energy for an inviscid, incompressible, steady flow. The sum of the various energies of the fluid remains constant as the fluid flows from one section to another. A useful interpretation of the Bernoulli equation can be obtained through the use of the concepts of the ***hydraulic grade line*** (HGL) and the ***energy line*** (EL). These ideas represent a geometrical interpretation of a flow and can often be effectively used to better grasp the fundamental processes involved.

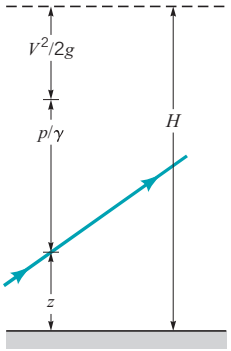
For steady, inviscid, incompressible flow the total energy remains constant along a streamline. The concept of "head" was introduced by dividing each term in Eq. 3.7 by the specific weight, $\gamma = \rho g$, to give the Bernoulli equation in the following form

$$\frac{p}{\gamma} + \frac{V^2}{2g} + z = \text{constant on a streamline} = H \qquad \textbf{(3.22)}$$

CMS Cepcor EX 1046 Page 33
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 3.21** **Representation of the energy line and the hydraulic grade line.**




Each of the terms in this equation has the units of length (feet or meters) and represents a certain type of head. The Bernoulli equation states that the sum of the pressure head, the velocity head, and the elevation head is constant along a streamline. This constant is called the *total head*, $H$.

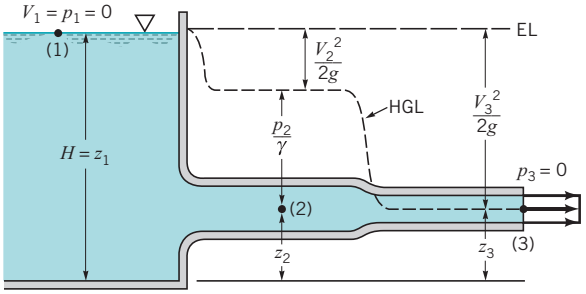
The energy line is a line that represents the total head available to the fluid. As shown in Fig. 3.21, the elevation of the energy line can be obtained by measuring the stagnation pressure with a Pitot tube. (A Pitot tube is the portion of a Pitot-static tube that measures the stagnation pressure. See Section 3.5.) The stagnation point at the end of the Pitot tube provides a measurement of the total head (or energy) of the flow. The static pressure tap connected to the piezometer tube shown, on the other hand, measures the sum of the pressure head and the elevation head, $p/\gamma + z$. This sum is often called the *piezometric head.* The static pressure tap does not measure the velocity head.

According to Eq. 3.22, the total head remains constant along the streamline (provided the assumptions of the Bernoulli equation are valid). Thus, a Pitot tube at any other location in the flow will measure the same total head, as is shown in the figure. The elevation head, velocity head, and pressure head may vary along the streamline, however.

The locus of elevations provided by a series of Pitot tubes is termed the energy line, EL. The locus provided by a series of piezometer taps is termed the hydraulic grade line, HGL. Under the assumptions of the Bernoulli equation, the energy line is horizontal. If the fluid velocity changes along the streamline, the hydraulic grade line will not be horizontal. If viscous effects are important (as they often are in pipe flows), the total head does not remain constant due to a loss in energy as the fluid flows along its streamline. This means that the energy line is no longer horizontal. Such viscous effects are discussed in Chapters 5 and 8.

*Under the assumptions of the Bernoulli equation, the energy line is horizontal.*

The energy line and hydraulic grade line for flow from a large tank are shown in Fig. 3.22. If the flow is steady, incompressible, and inviscid, the energy line is horizontal and at the elevation of the liquid in the tank (since the fluid velocity in the tank and the pressure on the surface




■ **FIGURE 3.22** **The energy line and hydraulic grade line for flow from a tank.**

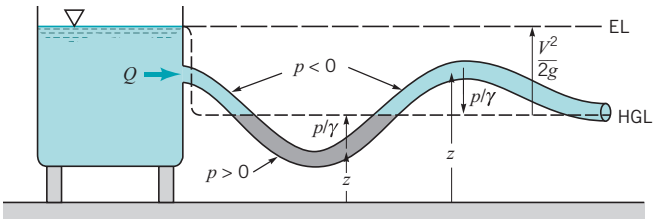



**FIGURE 3.23**
**Use of the energy line and the hydraulic grade line.**

are zero). The hydraulic grade line lies a distance of one velocity head, $V^2/2g$, below the energy line. Thus, a change in fluid velocity due to a change in the pipe diameter results in a change in the elevation of the hydraulic grade line. At the pipe outlet the pressure head is zero (gage) so the pipe elevation and the hydraulic grade line coincide.

*For flow below (above) the hydraulic grade line, the pressure is positive (negative).*

The distance from the pipe to the hydraulic grade line indicates the pressure within the pipe, as is shown in Fig. 3.23. If the pipe lies below the hydraulic grade line, the pressure within the pipe is positive (above atmospheric). If the pipe lies above the hydraulic grade line, the pressure is negative (below atmospheric). Thus, a scale drawing of a pipeline and the hydraulic grade line can be used to readily indicate regions of positive or negative pressure within a pipe.

## EXAMPLE 3.14 Energy Line and Hydraulic Grade Line

**GIVEN** Water is siphoned from the tank shown in Fig. E3.14 through a hose of constant diameter. A small hole is found in the hose at location (1) as indicated.

**FIND** When the siphon is used, will water leak out of the hose, or will air leak into the hose, thereby possibly causing the siphon to malfunction?

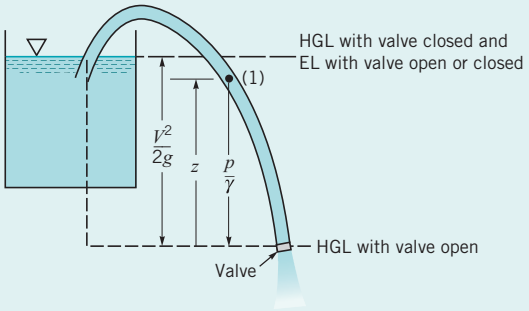



**FIGURE E3.14**

### SOLUTION

Whether air will leak into or water will leak out of the hose depends on whether the pressure within the hose at (1) is less than or greater than atmospheric. Which happens can be easily determined by using the energy line and hydraulic grade line concepts. With the assumption of steady, incompressible, inviscid flow it follows that the total head is constant—thus, the energy line is horizontal.

Since the hose diameter is constant, it follows from the continuity equation ($AV$ = constant) that the water velocity in the hose is constant throughout. Thus, the hydraulic grade line is a constant distance, $V^2/2g$, below the energy line as shown in Fig. E3.14. Since the pressure at the end of the hose is atmospheric, it follows that the hydraulic grade line is at the same elevation as the end of the hose outlet. The fluid within the hose at any point above the hydraulic grade line will be at less than atmospheric pressure.

Thus, air will leak into the hose through the hole at point (1). **(Ans)**

**COMMENT** In practice, viscous effects may be quite important, making this simple analysis (horizontal energy line) incorrect. However, if the hose is "not too small diameter," "not too long," the fluid "not too viscous," and the flowrate "not too large," the above result may be very accurate. If any of these assumptions are relaxed, a more detailed analysis is required (see Chapter 8). If the end of the hose were closed so that the flowrate were zero, the hydraulic grade line would coincide with the energy line ($V^2/2g = 0$ throughout), the pressure at (1) would be greater than atmospheric, and water would leak through the hole at (1).

The above discussion of the hydraulic grade line and the energy line is restricted to ideal situations involving inviscid, incompressible flows. Another restriction is that there are no "sources" or "sinks" of energy within the flow field. That is, there are no pumps or turbines involved. Alterations in the energy line and hydraulic grade line concepts due to these devices are discussed in Chapters 5 and 8.

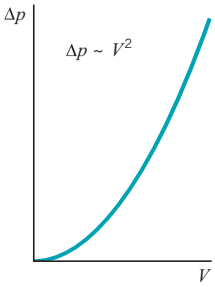
CMS Cepcor EX 1046 Page 35
CMS Cepcor v. Sandvik IPR2024-01380

## 3.8 Restrictions on Use of the Bernoulli Equation

Proper use of the Bernoulli equation requires close attention to the assumptions used in its derivation. In this section we review some of these assumptions and consider the consequences of incorrect use of the equation.

### 3.8.1 Compressibility Effects

One of the main assumptions is that the fluid is incompressible. Although this is reasonable for most liquid flows, it can, in certain instances, introduce considerable errors for gases.




In the previous section, we saw that the stagnation pressure, $p_{stag}$, is greater than the static pressure, $p_{static}$, by an amount $\Delta p = p_{stag} - p_{static} = \rho V^2/2$, provided that the density remains constant. If this dynamic pressure is not too large compared with the static pressure, the density change between two points is not very large and the flow can be considered incompressible. However, since the dynamic pressure varies as $V^2$, the error associated with the assumption that a fluid is incompressible increases with the square of the velocity of the fluid, as indicated by the figure in the margin. To account for compressibility effects we must return to Eq. 3.6 and properly integrate the term $\int dp/\rho$ when $\rho$ is not constant.

A simple, although specialized, case of compressible flow occurs when the temperature of a perfect gas remains constant along the streamline—isothermal flow. Thus, we consider $p = \rho RT$, where $T$ is constant. (In general, $p$, $\rho$, and $T$ will vary.) For steady, inviscid, isothermal flow, Eq. 3.6 becomes

$$RT \int \frac{dp}{p} + \frac{1}{2}V^2 + gz = \text{constant}$$

where we have used $\rho = p/RT$. The pressure term is easily integrated and the constant of integration evaluated if $z_1$, $p_1$, and $V_1$ are known at some location on the streamline. The result is

$$\frac{V_1^2}{2g} + z_1 + \frac{RT}{g}\ln\left(\frac{p_1}{p_2}\right) = \frac{V_2^2}{2g} + z_2 \tag{3.23}$$

*The Bernoulli equation can be modified for compressible flows.*

Equation 3.23 is the inviscid, isothermal analog of the incompressible Bernoulli equation. In the limit of small pressure difference, $p_1/p_2 = 1 + (p_1 - p_2)/p_2 = 1 + \varepsilon$, with $\varepsilon \ll 1$ and Eq. 3.23 reduces to the standard incompressible Bernoulli equation. This can be shown by use of the approximation $\ln(1 + \varepsilon) \approx \varepsilon$ for small $\varepsilon$. The use of Eq. 3.23 in practical applications is restricted by the inviscid flow assumption, since (as is discussed in Section 11.5) most isothermal flows are accompanied by viscous effects.

A much more common compressible flow condition is that of isentropic (constant entropy) flow of a perfect gas. Such flows are reversible adiabatic processes—"no friction or heat transfer"—and are closely approximated in many physical situations. As discussed fully in Chapter 11, for isentropic flow of a perfect gas the density and pressure are related by $p/\rho^k = C$, where $k$ is the specific heat ratio and $C$ is a constant. Hence, the $\int dp/\rho$ integral of Eq. 3.6 can be evaluated as follows. The density can be written in terms of the pressure as $\rho = p^{1/k}C^{-1/k}$ so that Eq. 3.6 becomes

$$C^{1/k} \int p^{-1/k}\, dp + \frac{1}{2}V^2 + gz = \text{constant}$$

The pressure term can be integrated between points (1) and (2) on the streamline and the constant $C$ evaluated at either point ($C^{1/k} = p_1^{1/k}/\rho_1$ or $C^{1/k} = p_2^{1/k}/\rho_2$) to give the following:

$$C^{1/k}\int_{p1}^{p2} p^{-1/k}\, dp = C^{1/k}\left(\frac{k}{k-1}\right)\left[p_2^{(k-1)/k} - p_1^{(k-1)/k}\right]$$

$$= \left(\frac{k}{k-1}\right)\left(\frac{p_2}{\rho_2} - \frac{p_1}{\rho_1}\right)$$

Thus, the final form of Eq. 3.6 for compressible, isentropic, steady flow of a perfect gas is

$$\left(\frac{k}{k-1}\right)\frac{p_1}{\rho_1}+\frac{V_1^2}{2}+gz_1=\left(\frac{k}{k-1}\right)\frac{p_2}{\rho_2}+\frac{V_2^2}{2}+gz_2 \tag{3.24}$$

The similarities between the results for compressible isentropic flow (Eq. 3.24) and incompressible isentropic flow (the Bernoulli equation, Eq. 3.7) are apparent. The only differences are the factors of $[k/(k-1)]$ that multiply the pressure terms and the fact that the densities are different ($\rho_1 \neq \rho_2$). In the limit of "low-speed flow" the two results are exactly the same, as is seen by the following.

We consider the stagnation point flow of Section 3.5 to illustrate the difference between the incompressible and compressible results. As is shown in Chapter 11, Eq. 3.24 can be written in dimensionless form as

$$\frac{p_2-p_1}{p_1}=\left[\left(1+\frac{k-1}{2}\,\mathrm{Ma}_1^2\right)^{k/k-1}-1\right] \qquad \text{(compressible)} \tag{3.25}$$

where (1) denotes the upstream conditions and (2) the stagnation conditions. We have assumed $z_1 = z_2$, $V_2 = 0$, and have denoted $\mathrm{Ma}_1 = V_1/c_1$ as the upstream *Mach number*—the ratio of the fluid velocity to the speed of sound, $c_1 = \sqrt{kRT_1}$.

A comparison between this compressible result and the incompressible result is perhaps most easily seen if we write the incompressible flow result in terms of the pressure ratio and the Mach number. Thus, we divide each term in the Bernoulli equation, $\rho V_1^2/2 + p_1 = p_2$, by $p_1$ and use the perfect gas law, $p_1 = \rho RT_1$, to obtain

$$\frac{p_2-p_1}{p_1}=\frac{V_1^2}{2RT_1}$$

Since $\mathrm{Ma}_1 = V_1/\sqrt{kRT_1}$ this can be written as

$$\frac{p_2-p_1}{p_1}=\frac{k\mathrm{Ma}_1^2}{2} \qquad \text{(incompressible)} \tag{3.26}$$

Equations 3.25 and 3.26 are plotted in Fig. 3.24. In the low-speed limit of $\mathrm{Ma}_1 \to 0$, both of the results are the same. This can be seen by denoting $(k-1)\mathrm{Ma}_1^2/2 = \tilde{\varepsilon}$ and using the binomial expansion, $(1+\tilde{\varepsilon})^n = 1 + n\tilde{\varepsilon} + n(n-1)\,\tilde{\varepsilon}^2/2 + \cdots$, where $n = k/(k-1)$, to write Eq. 3.25 as

*For small Mach numbers the compressible and incompressible results are nearly the same.*

$$\frac{p_2-p_1}{p_1}=\frac{k\mathrm{Ma}_1^2}{2}\left(1+\frac{1}{4}\,\mathrm{Ma}_1^2+\frac{2-k}{24}\,\mathrm{Ma}_1^4+\cdots\right) \qquad \text{(compressible)}$$

For $\mathrm{Ma}_1 \ll 1$ this compressible flow result agrees with Eq. 3.26. The incompressible and compressible equations agree to within about 2% up to a Mach number of approximately $\mathrm{Ma}_1 = 0.3$. For larger Mach numbers the disagreement between the two results increases.

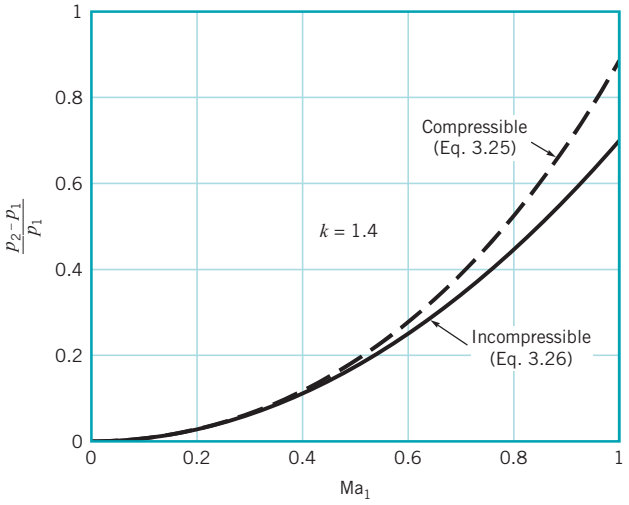



**FIGURE 3.24** Pressure ratio as a function of Mach number for incompressible and compressible (isentropic) flow.

CMS Cepcor EX 1046 Page 37
CMS Cepcor v. Sandvik IPR2024-01380

Thus, a "rule of thumb" is that the flow of a perfect gas may be considered as incompressible provided the Mach number is less than about 0.3. In standard air ($T_1 = 59\ ^\circ\text{F}$, $c_1 = \sqrt{kRT_1} = 1117$ ft/s) this corresponds to a speed of $V_1 = \text{Ma}_1 c_1 = 0.3(1117\ \text{ft/s}) = 335\ \text{ft/s} = 228$ mi/hr. At higher speeds, compressibility may become important.

## EXAMPLE 3.15 Compressible Flow—Mach Number

**GIVEN** The jet shown in Fig. E3.15 flies at Mach 0.82 at an altitude of 10 km in a standard atmosphere.

**FIND** Determine the stagnation pressure on the leading edge of its wing if the flow is incompressible; and if the flow is compressible isentropic.



■ FIGURE E3.15 (Photograph courtesy of Pure stock/superstock.)

### SOLUTION

From Tables 1.8 and C.2 we find that $p_1 = 26.5$ kPa (abs), $T_1 = -49.9\ ^\circ\text{C}$, $\rho = 0.414\ \text{kg/m}^3$, and $k = 1.4$. Thus, if we assume incompressible flow, Eq. 3.26 gives

$$\frac{p_2 - p_1}{p_1} = \frac{k\text{Ma}_1^2}{2} = 1.4\,\frac{(0.82)^2}{2} = 0.471$$

or

$$p_2 - p_1 = 0.471(26.5\ \text{kPa}) = 12.5\ \text{kPa} \qquad \textbf{(Ans)}$$

On the other hand, if we assume isentropic flow, Eq. 3.25 gives

$$\frac{p_2 - p_1}{p_1} = \left\{\left[1 + \frac{(1.4-1)}{2}(0.82)^2\right]^{1.4/(1.4-1)} - 1\right\} = 0.555$$

or

$$p_2 - p_1 = 0.555(26.5\ \text{kPa}) = 14.7\ \text{kPa} \qquad \textbf{(Ans)}$$

**COMMENT** We see that at Mach 0.82 compressibility effects are of importance. The pressure (and, to a first approximation, the lift and drag on the airplane; see Chapter 9) is approximately 14.7/12.5 = 1.18 times greater according to the compressible flow calculations. This may be very significant. As discussed in Chapter 11, for Mach numbers greater than 1 (supersonic flow) the differences between incompressible and compressible results are often not only quantitative but also qualitative.

Note that if the airplane were flying at Mach 0.30 (rather than 0.82) the corresponding values would be $p_2 - p_1 = 1.670$ kPa for incompressible flow and $p_2 - p_1 = 1.707$ kPa for compressible flow. The difference between these two results is about 2%.

### 3.8.2 Unsteady Effects

*The Bernoulli equation can be modified for unsteady flows.*

Another restriction of the Bernoulli equation (Eq. 3.7) is the assumption that the flow is steady. For such flows, on a given streamline the velocity is a function of only $s$, the location along the streamline. That is, along a streamline $V = V(s)$. For unsteady flows the velocity is also a function of time, so that along a streamline $V = V(s, t)$. Thus when taking the time derivative of the velocity to obtain the streamwise acceleration, we obtain $a_s = \partial V/\partial t + V\,\partial V/\partial s$ rather than just $a_s = V\,\partial V/\partial s$ as is true for steady flow. For steady flows the acceleration is due to the change in velocity resulting from a change in position of the particle (the $V\,\partial V/\partial s$ term), whereas for unsteady flow there is an additional contribution to the acceleration resulting from a change in velocity with time at a fixed location (the $\partial V/\partial t$ term). These effects are discussed in detail in Chapter 4. The net effect is that the inclusion of the unsteady term, $\partial V/\partial t$, does not allow the equation of motion to be easily integrated (as was done to obtain the Bernoulli equation) unless additional assumptions are made.

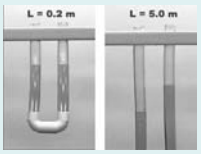
The Bernoulli equation was obtained by integrating the component of Newton's second law (Eq. 3.5) along the streamline. When integrated, the acceleration contribution to this equation, the

$\frac{1}{2}\rho d(V^2)$ term, gave rise to the kinetic energy term in the Bernoulli equation. If the steps leading to Eq. 3.5 are repeated with the inclusion of the unsteady effect ($\partial V/\partial t \neq 0$) the following is obtained:

$$\rho \frac{\partial V}{\partial t} ds + dp + \frac{1}{2}\rho d(V^2) + \gamma\, dz = 0 \qquad \text{(along a streamline)}$$



*V3.11 Oscillations in a U-tube*




For incompressible flow this can be easily integrated between points (1) and (2) to give

$$p_1 + \frac{1}{2}\rho V_1^2 + \gamma z_1 = \rho \int_{s_1}^{s_2} \frac{\partial V}{\partial t} ds + p_2 + \frac{1}{2}\rho V_2^2 + \gamma z_2 \qquad \text{(along a streamline)} \qquad \textbf{(3.27)}$$

Equation 3.27 is an unsteady form of the Bernoulli equation valid for unsteady, incompressible, inviscid flow. Except for the integral involving the local acceleration, $\partial V/\partial t$, it is identical to the steady Bernoulli equation. In general, it is not easy to evaluate this integral because the variation of $\partial V/\partial t$ along the streamline is not known. In some situations the concepts of "irrotational flow" and the "velocity potential" can be used to simplify this integral. These topics are discussed in Chapter 6.

## EXAMPLE 3.16 Unsteady Flow—U-Tube

**GIVEN** An incompressible, inviscid liquid is placed in a vertical, constant diameter U-tube as indicated in Fig. E3.16. When released from the nonequilibrium position shown, the liquid column will oscillate at a specific frequency.

**FIND** Determine this frequency.

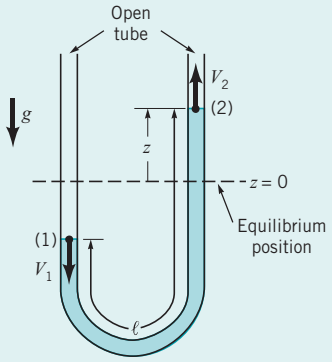



**FIGURE E3.16**

### SOLUTION

The frequency of oscillation can be calculated by use of Eq. 3.27 as follows. Let points (1) and (2) be at the air–water interfaces of the two columns of the tube and $z = 0$ correspond to the equilibrium position of these interfaces. Hence, $p_1 = p_2 = 0$ and if $z_2 = z$, then $z_1 = -z$. In general, $z$ is a function of time, $z = z(t)$. For a constant diameter tube, at any instant in time the fluid speed is constant throughout the tube, $V_1 = V_2 = V$, and the integral representing the unsteady effect in Eq. 3.27 can be written as

$$\int_{s_1}^{s_2} \frac{\partial V}{\partial t} ds = \frac{dV}{dt}\int_{s_1}^{s_2} ds = \ell \frac{dV}{dt}$$

where $\ell$ is the total length of the liquid column as shown in the figure. Thus, Eq. 3.27 can be written as

$$\gamma(-z) = \rho\ell \frac{dV}{dt} + \gamma z$$

Since $V = dz/dt$ and $\gamma = \rho g$, this can be written as the second-order differential equation describing simple harmonic motion

$$\frac{d^2 z}{dt^2} + \frac{2g}{\ell} z = 0$$

which has the solution $z(t) = C_1 \sin(\sqrt{2g/\ell}\, t) + C_2 \cos(\sqrt{2g/\ell}\, t)$. The values of the constants $C_1$ and $C_2$ depend on the initial state (velocity and position) of the liquid at $t = 0$. Thus, the liquid oscillates in the tube with a frequency

$$\omega = \sqrt{2g/\ell} \qquad \textbf{(Ans)}$$

**COMMENT** This frequency depends on the length of the column and the acceleration of gravity (in a manner very similar to the oscillation of a pendulum). The period of this oscillation (the time required to complete an oscillation) is $t_0 = 2\pi\sqrt{\ell/2g}$.

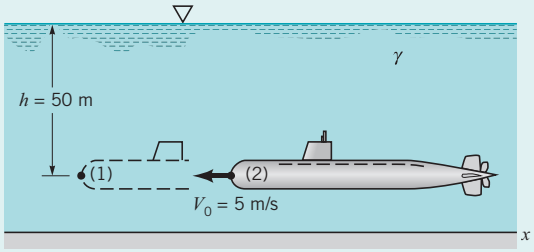
In a few unsteady flow cases, the flow can be made steady by an appropriate selection of the coordinate system. Example 3.17 illustrates this.

CMS Cepcor EX 1046 Page 39
CMS Cepcor v. Sandvik IPR2024-01380

## EXAMPLE 3.17 Unsteady or Steady Flow

**GIVEN** A submarine moves through seawater ($SG = 1.03$) at a depth of 50 m with velocity $V_0 = 5.0$ m/s as shown in Fig. E3.17.

**FIND** Determine the pressure at the stagnation point (2).




■ FIGURE E3.17

### SOLUTION

In a coordinate system fixed to the ground, the flow is unsteady. For example, the water velocity at (1) is zero with the submarine in its initial position, but at the instant when the nose, (2), reaches point (1) the velocity there becomes $\mathbf{V}_1 = -V_0\hat{\mathbf{i}}$. Thus, $\partial \mathbf{V}_1/\partial t \neq 0$ and the flow is unsteady. Application of the steady Bernoulli equation between (1) and (2) would give the incorrect result that "$p_1 = p_2 + \rho V_0^2/2$." According to this result the static pressure is greater than the stagnation pressure—an incorrect use of the Bernoulli equation.

We can either use an unsteady analysis for the flow (which is outside the scope of this text) or redefine the coordinate system so that it is fixed on the submarine, giving steady flow with respect to this system. The correct method would be

$$p_2 = \frac{\rho V_1^2}{2} + \gamma h = [(1.03)(1000)\ \text{kg/m}^3]\,(5.0\ \text{m/s})^2/2 + (9.80 \times 10^3\ \text{N/m}^3)(1.03)(50\ \text{m})$$

$$= (12{,}900 + 505{,}000)\ \text{N/m}^2$$

$$= 518\ \text{kPa} \qquad \textbf{(Ans)}$$

similar to that discussed in Example 3.2.

**COMMENT** If the submarine were accelerating, $\partial V_0/\partial t \neq 0$, the flow would be unsteady in either of the above coordinate systems and we would be forced to use an unsteady form of the Bernoulli equation.

Some unsteady flows may be treated as "quasisteady" and solved approximately by using the steady Bernoulli equation. In these cases the unsteadiness is "not too great" (in some sense), and the steady flow results can be applied at each instant in time as though the flow were steady. The slow draining of a tank filled with liquid provides an example of this type of flow.

### 3.8.3 Rotational Effects

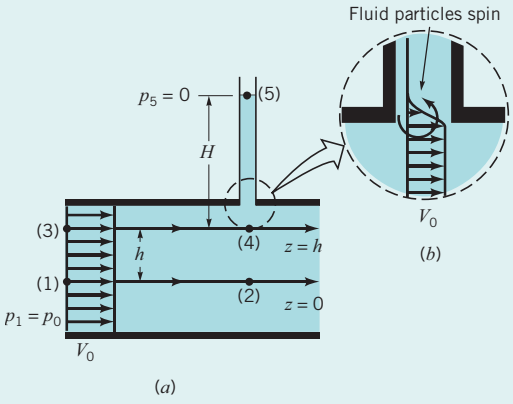
*Care must be used in applying the Bernoulli equation across streamlines.*

Another of the restrictions of the Bernoulli equation is that it is applicable along the streamline. Application of the Bernoulli equation across streamlines (i.e., from a point on one streamline to a point on another streamline) can lead to considerable errors, depending on the particular flow conditions involved. In general, the Bernoulli constant varies from streamline to streamline. However, under certain restrictions this constant is the same throughout the entire flow field. Example 3.18 illustrates this fact.

## EXAMPLE 3.18 Use of Bernoulli Equation across Streamlines

**GIVEN** Consider the uniform flow in the channel shown in Fig. E3.18*a*. The liquid in the vertical piezometer tube is stationary.

**FIND** Discuss the use of the Bernoulli equation between points (1) and (2), points (3) and (4), and points (4) and (5).




■ FIGURE E3.18

### SOLUTION

If the flow is steady, inviscid, and incompressible, Eq. 3.7 written between points (1) and (2) gives

$$p_1 + \tfrac{1}{2}\rho V_1^2 + \gamma z_1 = p_2 + \tfrac{1}{2}\rho V_2^2 + \gamma z_2 = \text{constant} = C_{12}$$

Since $V_1 = V_2 = V_0$ and $z_1 = z_2 = 0$, it follows that $p_1 = p_2 = p_0$ and the Bernoulli constant for this streamline, $C_{12}$, is given by

$$C_{12} = \tfrac{1}{2}\rho V_0^2 + p_0$$

Along the streamline from (3) to (4) we note that $V_3 = V_4 = V_0$ and $z_3 = z_4 = h$. As was shown in Example 3.5, application of $\mathbf{F} = m\mathbf{a}$ across the streamline (Eq. 3.12) gives $p_3 = p_1 - \gamma h$ because the streamlines are straight and horizontal. The above facts combined with the Bernoulli equation applied between (3) and (4) show that $p_3 = p_4$ and that the Bernoulli constant along this streamline is the same as that along the streamline between (1) and (2). That is, $C_{34} = C_{12}$, or

$$p_3 + \tfrac{1}{2}\rho V_3^2 + \gamma z_3 = p_4 + \tfrac{1}{2}\rho V_4^2 + \gamma z_4 = C_{34} = C_{12}$$

Similar reasoning shows that the Bernoulli constant is the same for any streamline in Fig. E3.18. Hence,

$$p + \tfrac{1}{2}\rho V^2 + \gamma z = \text{constant throughout the flow}$$

Again from Example 3.5 we recall that

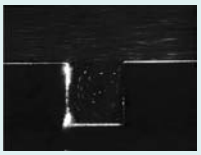
$$p_4 = p_5 + \gamma H = \gamma H$$

If we apply the Bernoulli equation across streamlines from (4) to (5), we obtain the incorrect result "$H = p_4/\gamma + V_4^2/2g$." The correct result is $H = p_4/\gamma$.

From the above we see that we can apply the Bernoulli equation across streamlines (1)–(2) and (3)–(4) (i.e., $C_{12} = C_{34}$) but not across streamlines from (4) to (5). The reason for this is that while the flow in the channel is "irrotational," it is "rotational" between the flowing fluid in the channel and the stationary fluid in the piezometer tube. Because of the uniform velocity profile across the channel, it is seen that the fluid particles do not rotate or "spin" as they move. The flow is "irrotational." However, as seen in Fig. E3.18*b*, there is a very thin shear layer between (4) and (5) in which adjacent fluid particles interact and rotate or "spin." This produces a "rotational" flow. A more complete analysis would show that the Bernoulli equation cannot be applied across streamlines if the flow is "rotational" (see Chapter 6).



*V3.12 Flow over a cavity*



As is suggested by Example 3.18, if the flow is "irrotational" (i.e., the fluid particles do not "spin" as they move), it is appropriate to use the Bernoulli equation across streamlines. However, if the flow is "rotational" (fluid particles "spin"), use of the Bernoulli equation is restricted to flow along a streamline. The distinction between irrotational and rotational flow is often a very subtle and confusing one. These topics are discussed in more detail in Chapter 6. A thorough discussion can be found in more advanced texts (Ref. 3).

### 3.8.4 Other Restrictions

Another restriction on the Bernoulli equation is that the flow is inviscid. As is discussed in Section 3.4, the Bernoulli equation is actually a first integral of Newton's second law along a streamline. This general integration was possible because, in the absence of viscous effects, the fluid system considered was a conservative system. The total energy of the system remains constant. If viscous effects are important the system is nonconservative (dissipative) and energy losses occur. A more detailed analysis is needed for these cases. Such material is presented in Chapter 5.

*The Bernoulli equation is not valid for flows that involve pumps or turbines.*

The final basic restriction on use of the Bernoulli equation is that there are no mechanical devices (pumps or turbines) in the system between the two points along the streamline for which the equation is applied. These devices represent sources or sinks of energy. Since the Bernoulli equation is actually one form of the energy equation, it must be altered to include pumps or turbines, if these are present. The inclusion of pumps and turbines is covered in Chapters 5 and 12.

In this chapter we have spent considerable time investigating fluid dynamic situations governed by a relatively simple analysis for steady, inviscid, incompressible flows. Many flows can be adequately analyzed by use of these ideas. However, because of the rather severe restrictions imposed, many others cannot. An understanding of these basic ideas will provide a firm foundation for the remainder of the topics in this book.

## 3.9 Chapter Summary and Study Guide

In this chapter, several aspects of the steady flow of an inviscid, incompressible fluid are discussed. Newton's second law, $\mathbf{F} = m\mathbf{a}$, is applied to flows for which the only important forces are those due to pressure and gravity (weight)—viscous effects are assumed negligible. The result is the often-used Bernoulli equation, which provides a simple relationship among pressure, elevation, and velocity variations along a streamline. A similar but less often used equation is also obtained to describe the variations in these parameters normal to a streamline.

The concept of a stagnation point and the corresponding stagnation pressure is introduced as are the concepts of static, dynamic, and total pressure and their related heads.

CMS Cepcor EX 1046 Page 41
CMS Cepcor v. Sandvik IPR2024-01380

*steady flow*
*streamline*
*Bernoulli equation*
*elevation head*
*pressure head*
*velocity head*
*static pressure*
*dynamic pressure*
*stagnation point*
*stagnation pressure*
*total pressure*
*Pitot-static tube*
*free jet*
*volume flowrate*
*continuity equation*
*cavitation*
*flow meter*
*hydraulic grade line*
*energy line*

Several applications of the Bernoulli equation are discussed. In some flow situations, such as the use of a Pitot-static tube to measure fluid velocity or the flow of a liquid as a free jet from a tank, a Bernoulli equation alone is sufficient for the analysis. In other instances, such as confined flows in tubes and flow meters, it is necessary to use both the Bernoulli equation and the continuity equation, which is a statement of the fact that mass is conserved as fluid flows.

The following checklist provides a study guide for this chapter. When your study of the entire chapter and end-of-chapter exercises has been completed, you should be able to

- write out meanings of the terms listed here in the margin and understand each of the related concepts. These terms are particularly important and are set in ***italic, bold, and color*** type in the text.
- explain the origin of the pressure, elevation, and velocity terms in the Bernoulli equation and how they are related to Newton's second law of motion.
- apply the Bernoulli equation to simple flow situations, including Pitot-static tubes, free jet flows, confined flows, and flow meters.
- use the concept of conservation of mass (the continuity equation) in conjunction with the Bernoulli equation to solve simple flow problems.
- apply Newton's second law across streamlines for appropriate steady, inviscid, incompressible flows.
- use the concepts of pressure, elevation, velocity, and total heads to solve various flow problems.
- explain and use the concepts of static, stagnation, dynamic, and total pressures.
- use the energy line and the hydraulic grade line concepts to solve various flow problems.
- explain the various restrictions on use of the Bernoulli equation.

Some of the important equations in this chapter are:

| | | |
|---|---|---|
| Streamwise and normal acceleration | $a_s = V\dfrac{\partial V}{\partial s}, \qquad a_n = \dfrac{V^2}{\mathcal{R}}$ | **(3.1)** |
| Force balance along a streamline for steady inviscid flow | $\displaystyle\int \frac{dp}{\rho} + \frac{1}{2}V^2 + gz = C$ (along a streamline) | **(3.6)** |
| The Bernoulli equation | $p + \frac{1}{2}\rho V^2 + \gamma z =$ constant along streamline | **(3.7)** |
| Pressure gradient normal to streamline for inviscid flow in absence of gravity | $\dfrac{\partial p}{\partial n} = -\dfrac{\rho V^2}{\mathcal{R}}$ | **(3.10b)** |
| Force balance normal to a streamline for steady, inviscid, incompressible flow | $\displaystyle p + \rho\int \frac{V^2}{\mathcal{R}}\,dn + \gamma z =$ constant across the streamline | **(3.12)** |
| Velocity measurement for a Pitot-static tube | $V = \sqrt{2\,(p_3 - p_4)/\rho}$ | **(3.16)** |
| Free jet | $\displaystyle V = \sqrt{2\frac{\gamma h}{\rho}} = \sqrt{2gh}$ | **(3.18)** |
| Continuity equation | $A_1V_1 = A_2V_2$, or $Q_1 = Q_2$ | **(3.19)** |
| Flow meter equation | $\displaystyle Q = A_2\sqrt{\frac{2(p_1 - p_2)}{\rho[1 - (A_2/A_1)^2]}}$ | **(3.20)** |
| Sluice gate equation | $\displaystyle Q = z_2 b\sqrt{\frac{2g(z_1 - z_2)}{1 - (z_2/z_1)^2}}$ | **(3.21)** |
| Total head | $\dfrac{p}{\gamma} + \dfrac{V^2}{2g} + z =$ constant on a streamline $= H$ | **(3.22)** |

## References


1. Riley, W. F., and Sturges, L. D., *Engineering Mechanics: Dynamics,* 2nd Ed., Wiley, New York, 1996.
2. Tipler, P. A., *Physics,* Worth, New York, 1982.
3. Panton, R. L., *Incompressible Flow,* Wiley, New York, 1984.


## Review Problems

Go to Appendix G for a set of review problems with answers. Detailed solutions can be found in *Student Solution Manual and Study Guide for Fundamentals of Fluid Mechanics*, by Munson et al. (© 2009 John Wiley and Sons, Inc.).

## Problems

**Note: Unless otherwise indicated, use the values of fluid properties found in the tables on the inside of the front cover. Problems designated with an (*) are intended to be solved with the aid of a programmable calculator or a computer. Problems designated with a (†) are "open-ended" problems and require critical thinking in that to work them one must make various assumptions and provide the necessary data. There is not a unique answer to these problems.**

**Answers to the even-numbered problems are listed at the end of the book. Access to the videos that accompany problems can be obtained through the book's web site, www.wiley.com/college/munson. The lab-type problems can also be accessed on this web site.**

### Section 3.2 F = *m*a along a Streamline

**3.1** Obtain a photograph/image of a situation which can be analyzed by use of the Bernoulli equation. Print this photo and write a brief paragraph that describes the situation involved.

**3.2** Air flows steadily along a streamline from point (1) to point (2) with negligible viscous effects. The following conditions are measured: At point (1) $z_1 = 2$ m and $p_1 = 0$ kPa; at point (2) $z_2 = 10$ m, $p_2 = 20$ N/m$^2$, and $V_2 = 0$. Determine the velocity at point (1).

**3.3** Water flows steadily through the variable area horizontal pipe shown in Fig. P3.3. The centerline velocity is given by $\mathbf{V} = 10(1 + x)\,\hat{\mathbf{i}}$ ft/s, where $x$ is in feet. Viscous effects are neglected. **(a)** Determine the pressure gradient, $\partial p/\partial x$, (as a function of $x$) needed to produce this flow. **(b)** If the pressure at section (1) is 50 psi, determine the pressure at (2) by (i) integration of the pressure gradient obtained in **(a)**, (ii) application of the Bernoulli equation.

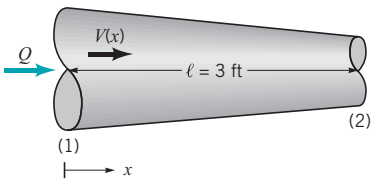



**FIGURE P3.3**

**3.4** Repeat Problem 3.3 if the pipe is vertical with the flow down.

**3.5** An incompressible fluid with density $\rho$ flows steadily past the object shown in Video V3.7 and Fig. P3.5. The fluid velocity along the horizontal dividing streamline $(-\infty \le x \le -a)$ is found to be $V = V_0(1 + a/x)$, where $a$ is the radius of curvature of the front of the object and $V_0$ is the upstream velocity. **(a)** Determine the pressure gradient along this streamline. **(b)** If the upstream pressure is $p_0$, integrate the pressure gradient to obtain the pressure $p(x)$ for $-\infty \le x \le -a$. **(c)** Show from the result of part **(b)** that the pressure at the stagnation point $(x = -a)$ is $p_0 + \rho V_0^2/2$, as expected from the Bernoulli equation.

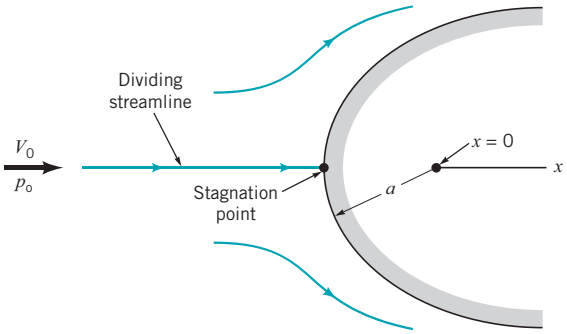



**FIGURE P3.5**

**3.6** What pressure gradient along the streamline, $dp/ds$, is required to accelerate water in a horizontal pipe at a rate of 30 m/s$^2$?

**3.7** A fluid with a specific weight of 100 lb/ft$^3$ and negligible viscous effects flows in the pipe shown in Fig. P3.7. The pressures at points (1) and (2) are 400 lb/ft$^2$ and 900 lb/ft$^2$, respectively. The velocities at points (1) and (2) are equal. Is the fluid accelerating uphill, downhill, or not accelerating? Explain.

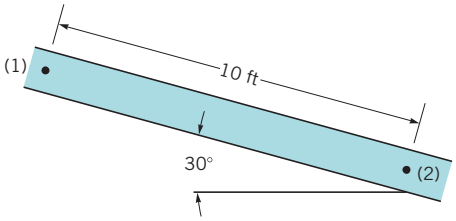


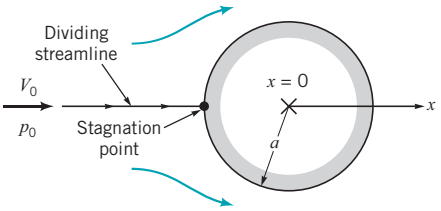

**FIGURE P3.7**

**3.8** What pressure gradient along the streamline, $dp/ds$, is required to accelerate water upward in a vertical pipe at a rate of 30 ft/s$^2$? What is the answer if the flow is downward?

**3.9** Consider a compressible fluid for which the pressure and density are related by $p/\rho^n = C_0$, where $n$ and $C_0$ are constants. Integrate the equation of motion along the streamline, Eq. 3.6, to

obtain the "Bernoulli equation" for this compressible flow as $[n/(n-1)]p/\rho + V^2/2 + gz = \text{constant}$.

**3.10** An incompressible fluid flows steadily past a circular cylinder as shown in Fig. P3.10. The fluid velocity along the dividing streamline $(-\infty \le x \le -a)$ is found to be $V = V_0(1 - a^2/x^2)$, where $a$ is the radius of the cylinder and $V_0$ is the upstream velocity. **(a)** Determine the pressure gradient along this streamline. **(b)** If the upstream pressure is $p_0$, integrate the pressure gradient to obtain the pressure $p(x)$ for $-\infty \le x \le -a$. **(c)** Show from the result of part **(b)** that the pressure at the stagnation point $(x = -a)$ is $p_0 + \rho V_0^2/2$, as expected from the Bernoulli equation.




■ FIGURE P3.10

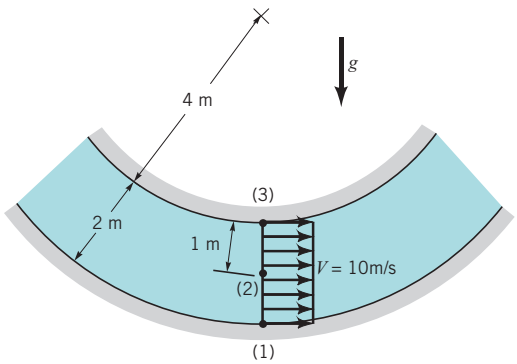
**3.11** Consider a compressible liquid that has a constant bulk modulus. Integrate "**F** = *m***a**" along a streamline to obtain the equivalent of the Bernoulli equation for this flow. Assume steady, inviscid flow.

## Section 3.3 F = *m*a Normal to a Streamline

**3.12** Obtain a photograph/image of a situation in which Newton's second law applied across the streamlines (as given by Eq. 3.12) is important. Print this photo and write a brief paragrph that describes the situation involved.

**3.13** Air flows along a horizontal, curved streamline with a 20 ft radius with a speed of 100 ft/s. Determine the pressure gradient normal to the streamline.

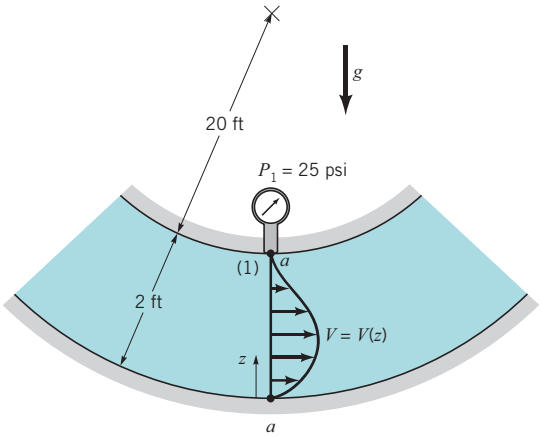
**3.14** Water flows around the vertical two-dimensional bend with circular streamlines and constant velocity as shown in Fig. P3.14. If the pressure is 40 kPa at point (1), determine the pressures at points (2) and (3). Assume that the velocity profile is uniform as indicated.




■ FIGURE P3.14

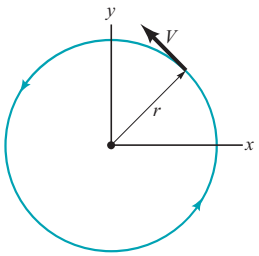
***3.15** Water flows around the vertical two-dimensional bend with circular streamlines as is shown in Fig. P3.15. The pressure at point (1) is measured to be $p_1 = 25$ psi and the velocity across section $a$–$a$ is as indicated in the table. Calculate and plot the pressure across section $a$–$a$ of the channel [$p = p(z)$ for $0 \le z \le 2$ ft].

| $z$ (ft) | $V$ (ft/s) |
|---|---|
| 0 | 0 |
| 0.2 | 8.0 |
| 0.4 | 14.3 |
| 0.6 | 20.0 |
| 0.8 | 19.5 |
| 1.0 | 15.6 |
| 1.2 | 8.3 |
| 1.4 | 6.2 |
| 1.6 | 3.7 |
| 1.8 | 2.0 |
| 2.0 | 0 |




■ FIGURE P3.15

**3.16** Water in a container and air in a tornado flow in horizontal circular streamlines of radius $r$ and speed $V$ as shown in **Video V3.6** and Fig. P3.16. Determine the radial pressure gradient, $\partial p/\partial r$, needed for the following situations: **(a)** The fluid is water with $r = 3$ in. and $V = 0.8$ ft/s. **(b)** The fluid is air with $r = 300$ ft and $V = 200$ mph.




■ FIGURE P3.16

**3.17** Air flows smoothly over the hood of your car and up past the windshield. However, a bug in the air does not follow the same path; it becomes splattered against the windshield. Explain why this is so.

## Section 3.5 Static, Stagnation, Dynamic, and Total Pressure

**3.18** Obtain a photograph/image of a situation in which the concept of the stagnation pressure is important. Print this photo and write a brief paragraph that describes the situation involved.

**3.19** At a given point on a horizontal streamline in flowing air, the static pressure is −2.0 psi (i.e., a vacuum) and the velocity is 150 ft/s. Determine the pressure at a stagnation point on that streamline.

†3.20 Estimate the maximum pressure on the surface of your car when you wash it using a garden hose connected to your outside faucet. List all assumptions and show calculations.

3.21 When an airplane is flying 200 mph at 5000-ft altitude in a standard atmosphere, the air velocity at a certain point on the wing is 273 mph relative to the airplane. **(a)** What suction pressure is developed on the wing at that point? **(b)** What is the pressure at the leading edge (a stagnation point) of the wing?

3.22 Some animals have learned to take advantage of Bernoulli effect without having read a fluid mechanics book. For example, a typical prairie dog burrow contains two entrances—a flat front door, and a mounded back door as shown in Fig. P3.22. When the wind blows with velocity $V_0$ across the front door, the average velocity across the back door is greater than $V_0$ because of the mound. Assume the air velocity across the back door is $1.07V_0$. For a wind velocity of 6 m/s, what pressure differences, $p_1 - p_2$, are generated to provide a fresh air flow within the burrow?

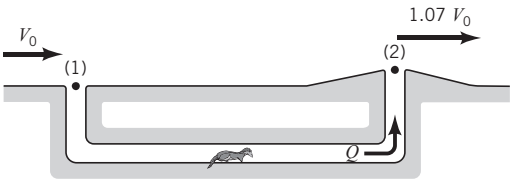



FIGURE P3.22

3.23 A loon is a diving bird equally at home "flying" in the air or water. What swimming velocity under water will produce a dynamic pressure equal to that when it flies in the air at 40 mph?

3.24 A person thrusts his hand into the water while traveling 3 m/s in a motorboat. What is the maximum pressure on his hand?

3.25 A Pitot-static tube is used to measure the velocity of helium in a pipe. The temperature and pressure are 40 °F and 25 psia. A water manometer connected to the Pitot-static tube indicates a reading of 2.3 in. Determine the helium velocity. Is it reasonable to consider the flow as incompressible? Explain.

3.26 An inviscid fluid flows steadily along the stagnation streamline shown in Fig. P3.26 and Video V3.7, starting with speed $V_0$ far upstream of the object. Upon leaving the stagnation point, point (1), the fluid speed along the surface of the object is assumed to be given by $V = 2\ V_0 \sin\theta$, where $\theta$ is the angle indicated. At what angular position, $\theta_2$, should a hole be drilled to give a pressure difference of $p_1 - p_2 = \rho V_0^2/2$? Gravity is negligible.

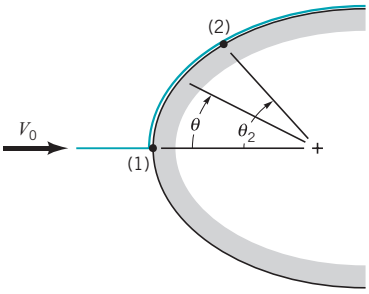



FIGURE P3.26

3.27 A water-filled manometer is connected to a Pitot-static tube to measure a nominal airspeed of 50 ft/s. It is assumed that a change in the manometer reading of 0.002 in. can be detected. What is the minimum deviation from the 50 ft/s airspeed that can be detected by this system? Repeat the problem if the nominal airspeed is 5 ft/s.

3.28 (See Fluids in the News article titled "Incorrect raindrop shape," Section 3.2.) The speed, $V$, at which a raindrop falls is a function of its diameter, $D$, as shown in Fig. P3.28. For what sized raindrop will the stagnation pressure be equal to half the internal pressure caused by surface tension? Recall from Section 1.9 that the pressure inside a drop is $\Delta p = 4\sigma/D$ greater than the surrounding pressure, where $\sigma$ is the surface tension.

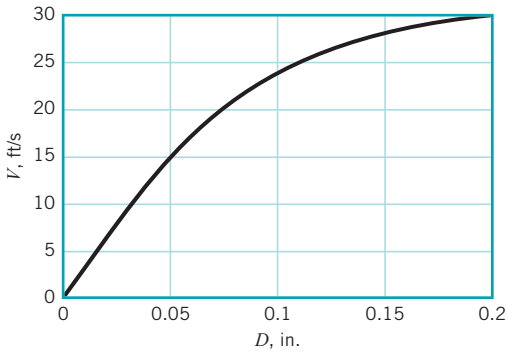



FIGURE P3.28

3.29 (See Fluids in the News article titled "Pressurized eyes," Section 3.5.) Determine the air velocity needed to produce a stagnation pressure equal to 10 mm of mercury.

3.30 (See Fluids in the News article titled "Bugged and plugged Pitot tubes," Section 3.5.) An airplane's Pitot tube used to indicate airspeed is partially plugged by an insect nest so that it measures 60% of the stagnation pressure rather than the actual stagnation pressure. If the airspeed indicator indicates that the plane is flying 150 mph, what is the actual airspeed?

## Section 3.6.1 Free Jets

3.31 Obtain a photograph/image of a situation in which the concept of a free jet is important. Print this photo and write a brief paragraph that describes the situation involved.

3.32 Water flows through a hole in the bottom of a large, open tank with a speed of 8 m/s. Determine the depth of water in the tank. Viscous effects are negligible.

3.33 Water flows from the faucet on the first floor of the building shown in Fig. P3.33 with a maximum velocity of 20 ft/s. For steady

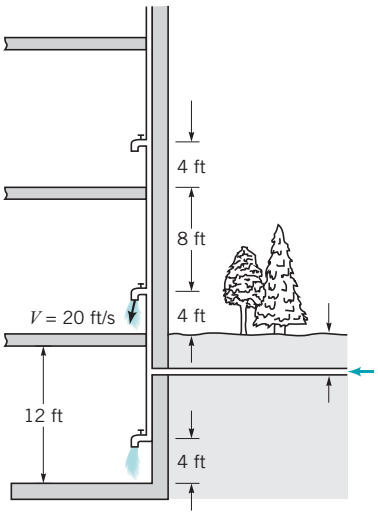
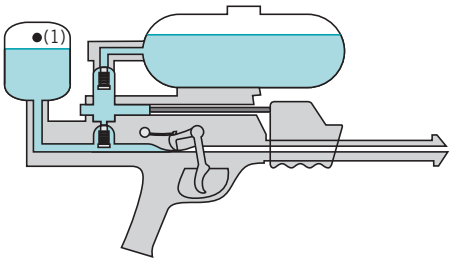



FIGURE P3.33

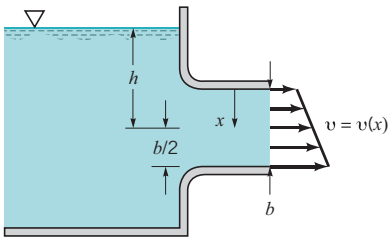
inviscid flow, determine the maximum water velocity from the basement faucet and from the faucet on the second floor (assume each floor is 12 ft tall).

†3.34 The "super soaker" water gun shown in Fig. P3.34 can shoot more than 30 ft in the horizontal direction. Estimate the minimum pressure, $p_1$, needed in the chamber in order to accomplish this. List all assumptions and show all calculations.



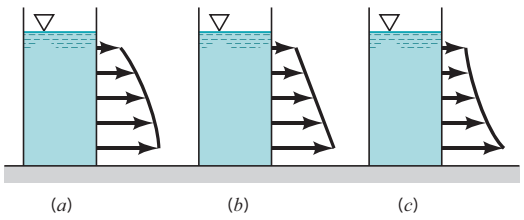

■ FIGURE P3.34

3.35* An inviscid liquid drains from a large tank through a square duct of width $b$ as shown in Fig. P3.35. The velocity of the fluid at the outlet is not precisely uniform because of the difference in elevation across the outlet. If $b \ll h$, this difference in velocity is negligible. For given $b$ and $h$, determine $v$ as a function of $x$ and integrate the results to determine the average velocity, $V = Q/b^2$. Plot the velocity distribution, $v = v(x)$, across the outlet if $h = 1$ and $b = 0.1$, 0.2, 0.4, 0.6, 0.8, and 1.0 m. How small must $b$ be if the centerline velocity, $v$ at $x = b/2$, is to be within 3% of the average velocity?




■ FIGURE P3.35

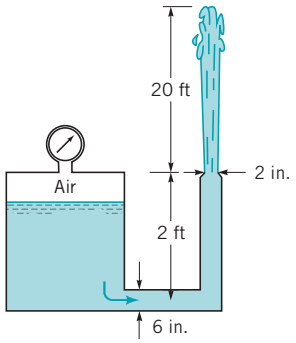
3.36 Several holes are punched into a tin can as shown in Fig. P3.36. Which of the figures represents the variation of the water velocity as it leaves the holes? Justify your choice.




■ FIGURE P3.36

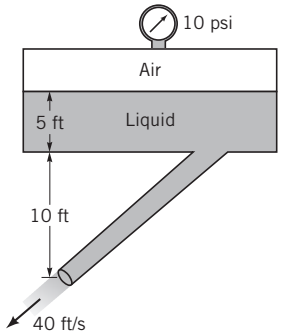
3.37 Water flows from a garden hose nozzle with a velocity of 15 m/s. What is the maximum height that it can reach above the nozzle?

3.38 Water flows from a pressurized tank, through a 6-in.-diameter pipe, exits from a 2-in.-diameter nozzle, and rises 20 ft above the nozzle as shown in Fig. P3.38. Determine the pressure in the tank if the flow is steady, frictionless, and incompressible.



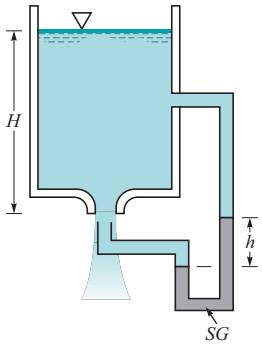

■ FIGURE P3.38

3.39 An inviscid, incompressible liquid flows steadily from the large pressurized tank shown in Fig. P.3.39. The velocity at the exit is 40 ft/s. Determine the specific gravity of the liquid in the tank.




■ FIGURE P3.39

3.40 Water flows from the tank shown in Fig. P3.40. If viscous effects are negligible, determine the value of $h$ in terms of $H$ and the specific gravity, $SG$, of the manometer fluid.




■ FIGURE P3.40

3.41 (See Fluids in the News article titled **"Armed with a water jet for hunting,"** Section 3.4.) Determine the pressure needed in the gills of an archerfish if it can shoot a jet of water 1 m vertically upward. Assume steady, inviscid flow.

## Section 3.6.2 Confined Flows (Also see Lab Problems 3.118 and 3.120.)

3.42 Obtain a photograph/image of a situation that involves a confined flow for which the Bernoulli and continuity equations are important. Print this photo and write a brief paragraph that describes the situation involved.

**3.43** Air flows steadily through a horizontal 4-in.-diameter pipe and exits into the atmosphere through a 3-in.-diameter nozzle. The velocity at the nozzle exit is 150 ft/s. Determine the pressure in the pipe if viscous effects are negligible.

**3.44** A fire hose nozzle has a diameter of $1\frac{1}{8}$ in. According to some fire codes, the nozzle must be capable of delivering at least 250 gal/min. If the nozzle is attached to a 3-in.-diameter hose, what pressure must be maintained just upstream of the nozzle to deliver this flowrate?

**3.45** Water flowing from the 0.75-in.-diameter outlet shown in **Video V8.14** and Fig. P3.45 rises 2.8 in. above the outlet. Determine the flowrate.

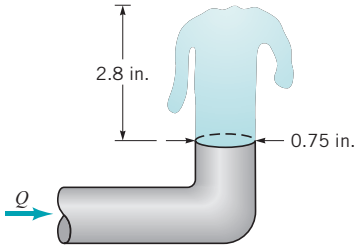



**FIGURE P3.45**

**3.46** Pop (with the same properties as water) flows from a 4-in.-diameter pop container that contains three holes as shown in Fig. P3.46 (see **Video 3.9**). The diameter of each fluid stream is 0.15 in., and the distance between holes is 2 in. If viscous effects are negligible and quasi-steady conditions are assumed, determine the time at which the pop stops draining from the top hole. Assume the pop surface is 2 in. above the top hole when $t = 0$. Compare your results with the time you measure from the video.

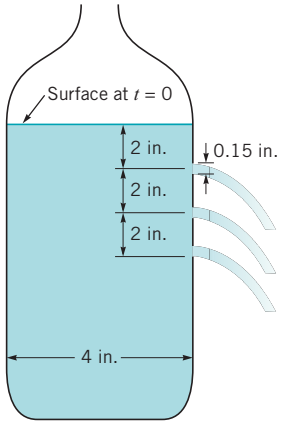



**FIGURE P3.46**

**3.47** Water (assumed inviscid and incompressible) flows steadily in the vertical variable-area pipe shown in Fig. P3.47. Determine the flowrate if the pressure in each of the gages reads 50 kPa..

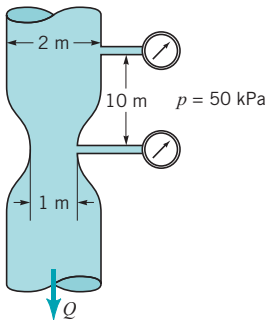



**FIGURE P3.47**

**3.48** Air is drawn into a wind tunnel used for testing automobiles as shown in Fig. P3.48. **(a)** Determine the manometer reading, $h$, when the velocity in the test section is 60 mph. Note that there is a 1-in. column of oil on the water in the manometer. **(b)** Determine the difference between the stagnation pressure on the front of the automobile and the pressure in the test section.

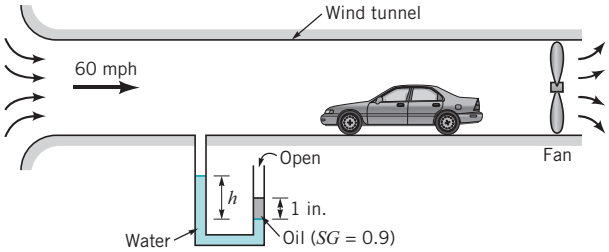



**FIGURE P3.48**

**3.49** Small-diameter, high-pressure liquid jets can be used to cut various materials as shown in Fig. P3.49. If viscous effects are negligible, estimate the pressure needed to produce a 0.10-mm-diameter water jet with a speed of 700 m/s. Determine the flowrate.

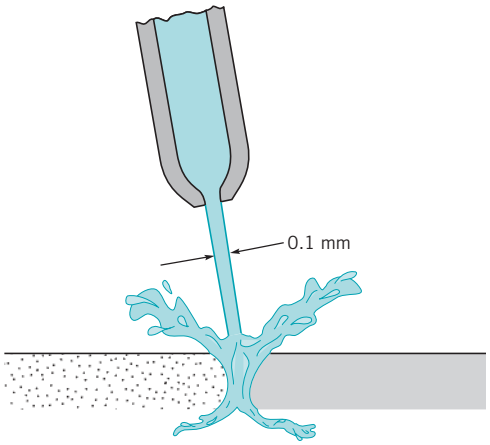



**FIGURE P3.49**

**3.50** Water (assumed inviscid and incompressible) flows steadily with a speed of 10 ft/s from the large tank shown in Fig. P3.50. Determine the depth, $H$, of the layer of light liquid (specific weight = 50 lb/ft$^3$) that covers the water in the tank.

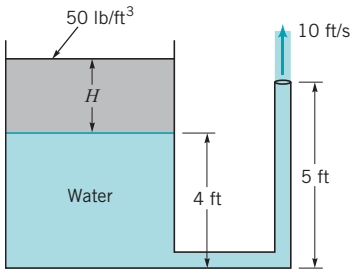
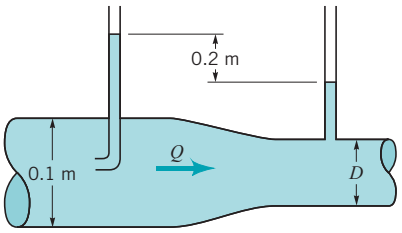


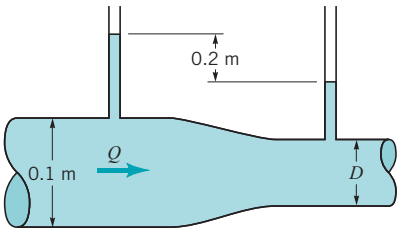

**FIGURE P3.50**

**3.51** Water flows through the pipe contraction shown in Fig. P3.51. For the given 0.2-m difference in manometer level, determine the flowrate as a function of the diameter of the small pipe, $D$.



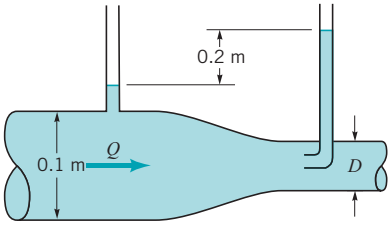

■ FIGURE P3.51

**3.52** Water flows through the pipe contraction shown in Fig. P3.52. For the given 0.2-m difference in the manometer level, determine the flowrate as a function of the diameter of the small pipe, $D$.




■ FIGURE P3.52

**3.53** Water flows through the pipe contraction shown in Fig. P3.53. For the given 0.2-m difference in the manometer level, determine the flowrate as a function of the diameter of the small pipe, $D$.




■ FIGURE P3.53

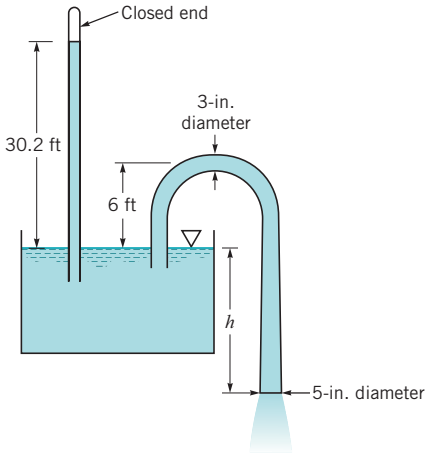
**3.54** A 0.15-m-diameter pipe discharges into a 0.10-m-diameter pipe. Determine the velocity head in each pipe if they are carrying 0.12 $m^3/s$ of kerosene.

**3.55** Carbon tetrachloride flows in a pipe of variable diameter with negligible viscous effects. At point $A$ in the pipe the pressure and velocity are 20 psi and 30 ft/s, respectively. At location $B$ the pressure and velocity are 23 psi and 14 ft/s. Which point is at the higher elevation and by how much?

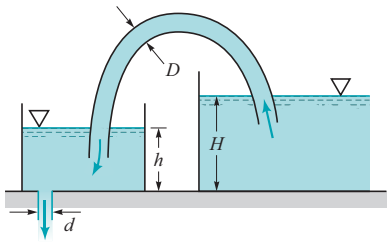
**3.56** The circular stream of water from a faucet is observed to taper from a diameter of 20 mm to 10 mm in a distance of 50 cm. Determine the flowrate.

**3.57** Water is siphoned from the tank shown in Fig. P3.57. The water barometer indicates a reading of 30.2 ft. Determine the maximum value of $h$ allowed without cavitation occurring. Note that the pressure of the vapor in the closed end of the barometer equals the vapor pressure.

**3.58** As shown in Fig. P3.58, water from a large reservoir flows without viscous effects through a siphon of diameter $D$ and into a tank. It exits from a hole in the bottom of the tank as a stream of diameter $d$. The surface of the reservoir remains $H$ above the bottom of the tank. For steady-state conditions, the water depth in the tank, $h$, is constant. Plot a graph of the depth ratio $h/H$ as a function of the diameter ratio $d/D$.

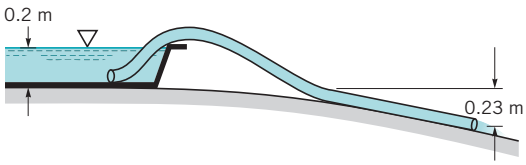



■ FIGURE P3.57



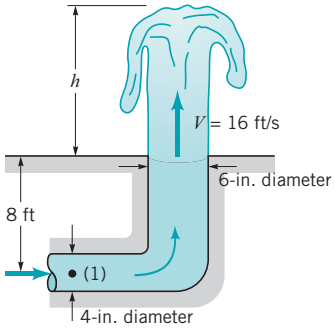

■ FIGURE P3.58

**3.59** A smooth plastic, 10-m-long garden hose with an inside diameter of 20 mm is used to drain a wading pool as is shown in Fig. P3.59. If viscous effects are neglected, what is the flowrate from the pool?




■ FIGURE P3.59

**3.60** Water exits a pipe as a free jet and flows to a height $h$ above the exit plane as shown in Fig. P3.60. The flow is steady, incompressible, and frictionless. **(a)** Determine the height $h$. **(b)** Determine the velocity and pressure at section (1).




■ FIGURE P3.60

CMS Cepcor EX 1046 Page 48
CMS Cepcor v. Sandvik IPR2024-01380

**3.61** Water flows steadily from a large, closed tank as shown in Fig. P3.61. The deflection in the mercury manometer is 1 in. and viscous effects are negligible. **(a)** Determine the volume flowrate. **(b)** Determine the air pressure in the space above the surface of the water in the tank.

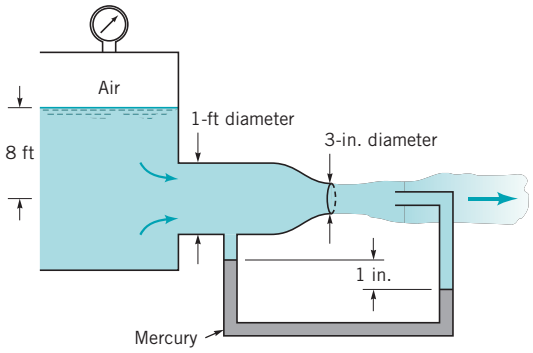



**FIGURE P3.61**

**3.62** Blood ($SG = 1$) flows with a velocity of 0.5 m/s in an artery. It then enters an aneurysm in the artery (i.e., an area of weakened and stretched artery walls that cause a ballooning of the vessel) whose cross-sectional area is 1.8 times that of the artery. Determine the pressure difference between the blood in the aneurysm and that in the artery. Assume the flow is steady and inviscid.

**3.63** Water flows steadily through the variable area pipe shown in Fig. P3.63 with negligible viscous effects. Determine the manometer reading, $H$, if the flowrate is 0.5 $m^3/s$ and the density of the manometer fluid is 600 $kg/m^3$.

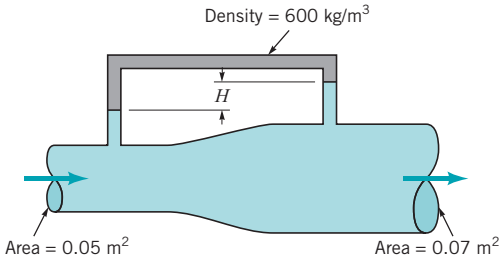



**FIGURE P3.63**

**3.64** Water flows steadily with negligible viscous effects through the pipe shown in Fig. P3.64. It is known that the 4-in.-diameter section of thin-walled tubing will collapse if the pressure within it becomes less than 10 psi below atmospheric pressure. Determine the maximum value that $h$ can have without causing collapse of the tubing.

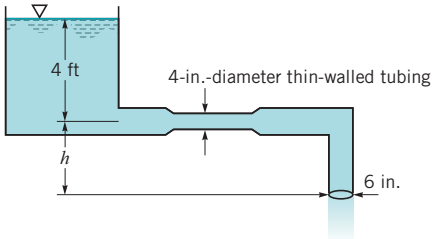



**FIGURE P3.64**

**3.65** Helium flows through a 0.30-m-diameter horizontal pipe with a temperature of 20 °C and a pressure of 200 kPa (abs) at a rate of 0.30 kg/s. If the pipe reduces to 0.25-m-diameter determine the pressure difference between these two sections. Assume incompressible, inviscid flow.

**3.66** Water is pumped from a lake through an 8-in. pipe at a rate of 10 $ft^3/s$. If viscous effects are negligible, what is the pressure in the suction pipe (the pipe between the lake and the pump) at an elevation 6 ft above the lake?

**3.67** Air flows through a Venturi channel of rectangular cross section as shown in **Video V3.10** and Fig. P3.67. The constant width of the channel is 0.06 m and the height at the exit is 0.04 m. Compressibility and viscous effects are negligible. **(a)** Determine the flowrate when water is drawn up 0.10 m in a small tube attached to the static pressure tap at the throat where the channel height is 0.02 m. **(b)** Determine the channel height, $h_2$, at section (2) where, for the same flowrate as in part **(a)**, the water is drawn up 0.05 m. **(c)** Determine the pressure needed at section (1) to produce this flow.

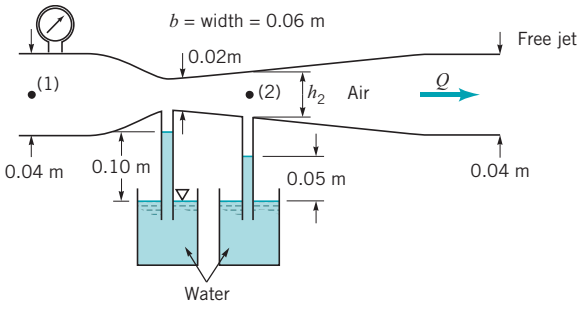



**FIGURE P3.67**

**3.68** Water flows steadily from the large open tank shown in Fig. P3.68. If viscous effects are negligible, determine **(a)** the flowrate, $Q$, and **(b)** the manometer reading, $h$.

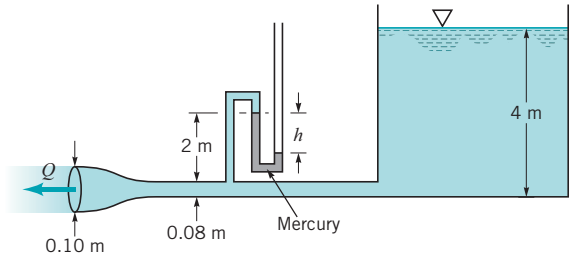
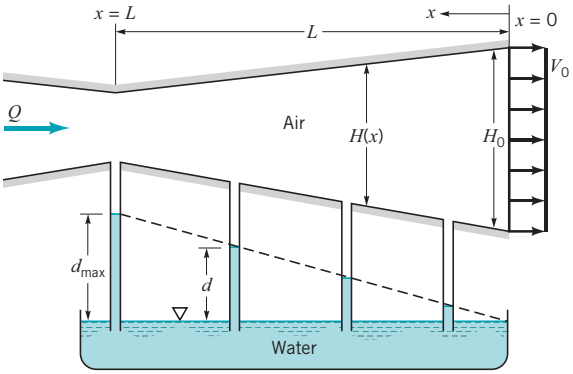



**FIGURE P3.68**

**3.69** Water from a faucet fills a 16-oz glass (volume = 28.9 $in.^3$) in 20 s. If the diameter of the jet leaving the faucet is 0.60 in., what is the diameter of the jet when it strikes the water surface in the glass which is positioned 14 in. below the faucet?

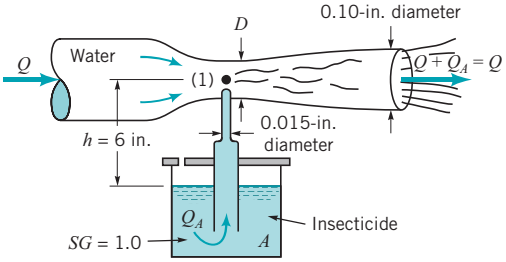
**3.70** Air flows steadily through a converging–diverging rectangular channel of constant width as shown in Fig. P3.70 and **Video V3.10**. The height of the channel at the exit and the exit velocity are $H_0$ and $V_0$, respectively. The channel is to be shaped so that the distance, $d$, that water is drawn up into tubes attached to static pressure taps along the channel wall is linear with distance along the channel. That is, $d = (d_{max}/L)\,x$, where $L$ is the channel length and $d_{max}$ is the maximum water depth (at the minimum channel height; $x = L$). Determine the height, $H(x)$, as a function of $x$ and the other important parameters.

CMS Cepcor EX 1046 Page 49
CMS Cepcor v. Sandvik IPR2024-01380



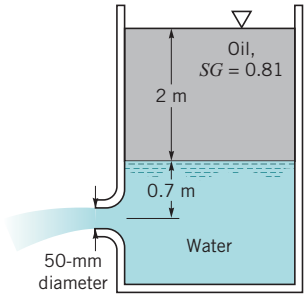

■ FIGURE P3.70

*3.71 The device shown in Fig. P3.71 is used to spray an appropriate mixture of water and insecticide. The flowrate from tank $A$ is to be $Q_A = 0.02$ gal/min when the water flowrate through the hose is $Q = 1$ gal/min. Determine the pressure needed at point (1) and the diameter, $D$, of the device For the diameter determined above, plot the ratio of insecticide flowrate to water flowrate as a function of water flowrate, $Q$, for $0.1 \le Q \le 1$ gal/min. Can this device be used to provide a reasonably constant ratio of insecticide to water regardless of the water flowrate? Explain.



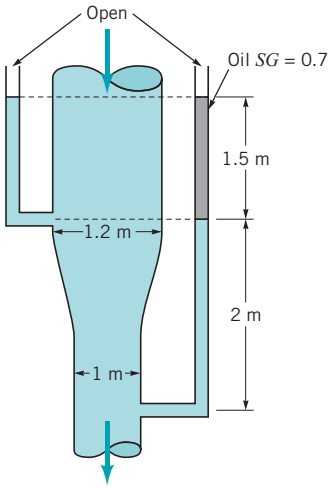

■ FIGURE P3.71

3.72 If viscous effects are neglected and the tank is large, determine the flowrate from the tank shown in Fig. P3.72.



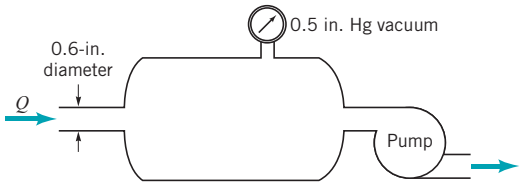

■ FIGURE P3.72

3.73 Water flows steadily downward in the pipe shown in Fig. 3.73 with negligible losses. Determine the flowrate.



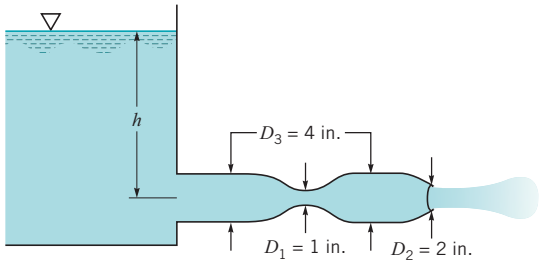

■ FIGURE P3.73

3.74 Air at 80 °F and 14.7 psia flows into the tank shown in Fig. P3.74. Determine the flowrate in ft$^3$/s, lb/s, and slugs/s. Assume incompressible flow.




■ FIGURE P3.74

3.75 Water flows from a large tank as shown in Fig. P3.75. Atmospheric pressure is 14.5 psia, and the vapor pressure is 1.60 psia. If viscous effects are neglected, at what height, $h$, will cavitation begin? To avoid cavitation, should the value of $D_1$ be increased or decreased? To avoid cavitation, should the value of $D_2$ be increased or decreased? Explain.




■ FIGURE P3.75

3.76 Water flows into the sink shown in Fig. P3.76 and Video V5.1 at a rate of 2 gal/min. If the drain is closed, the water will eventually flow through the overflow drain holes rather than over the edge of the sink. How many 0.4-in.-diameter drain holes are needed to ensure that the water does not overflow the sink? Neglect viscous effects.

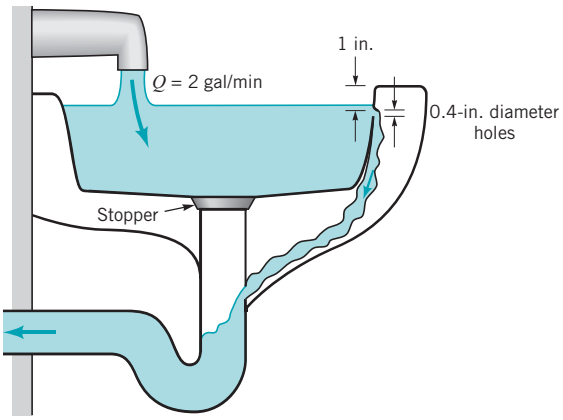



FIGURE P3.76

**3.77** What pressure, $p_1$, is needed to produce a flowrate of 0.09 $ft^3/s$ from the tank shown in Fig. P3.77?

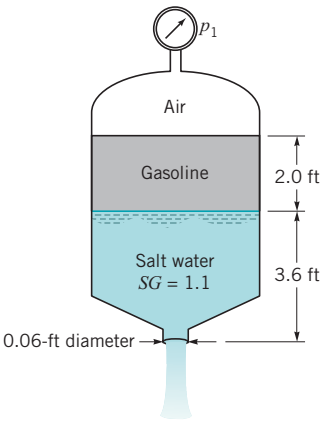



FIGURE P3.77

**3.78** Water is siphoned from the tank shown in Fig. P3.78. Determine the flowrate from the tank and the pressures at points (1), (2), and (3) if viscous effects are negligible.

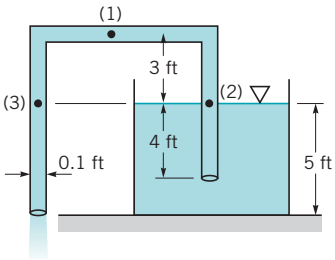



FIGURE P3.78

**3.79** Water is siphoned from a large tank and discharges into the atmosphere through a 2-in.-diameter tube as shown in Fig. P3.79. The end of the tube is 3 ft below the tank bottom, and viscous effects are negligible. **(a)** Determine the volume flowrate from the tank. **(b)** Determine the maximum height, $H$, over which the water can be siphoned without cavitation occurring. Atmospheric pressure is 14.7 psia, and the water vapor pressure is 0.26 psia.

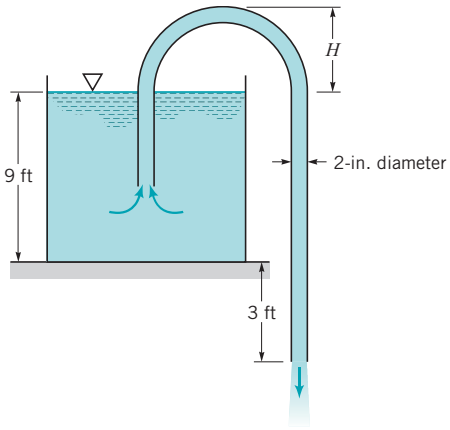



FIGURE P3.79

**3.80** Determine the manometer reading, $h$, for the flow shown in Fig. P3.80.

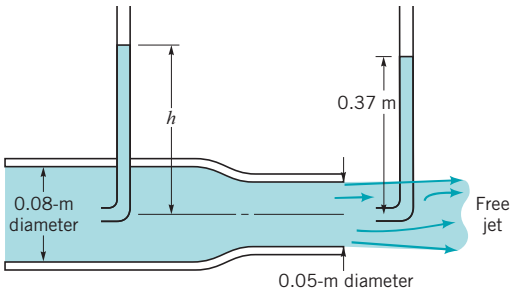



FIGURE P3.80

**3.81** Air flows steadily through the variable area pipe shown in Fig. P3.81. Determine the flowrate if viscous and compressibility effects are negligible.

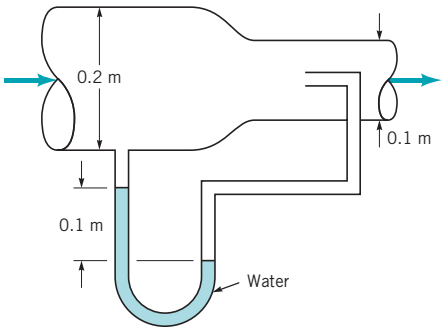
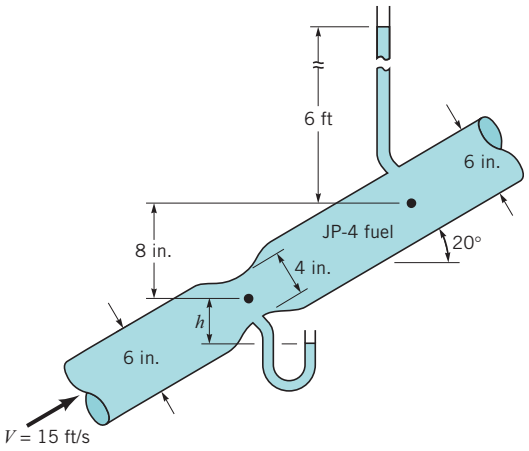



FIGURE P3.81

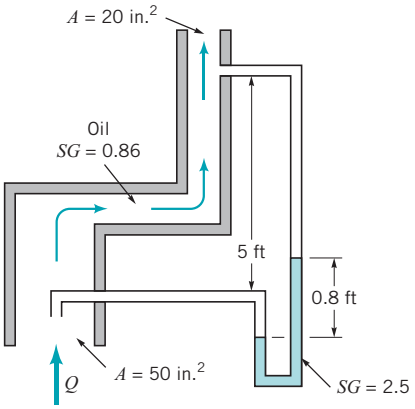
**3.82** JP-4 fuel ($SG$ = 0.77) flows through the Venturi meter shown in Fig. P3.82 with a velocity of 15 ft/s in the 6-in. pipe. If viscous effects are negligible, determine the elevation, $h$, of the fuel in the open tube connected to the throat of the Venturi meter.




■ FIGURE P3.82

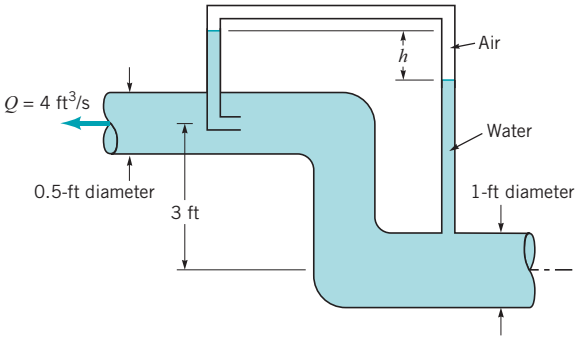
**3.83** Repeat Problem 3.82 if the flowing fluid is water rather than JP-4 fuel.

**3.84** Oil flows through the system shown in Fig. P3.84 with negligible losses. Determine the flowrate.



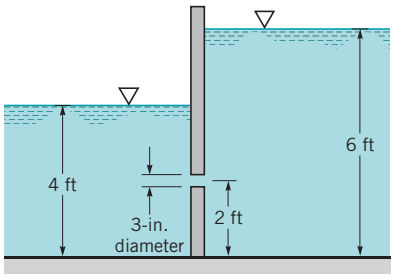

■ FIGURE P3.84

**3.85** Water, considered an inviscid, incompressible fluid, flows steadily as shown in Fig. P3.85. Determine $h$.



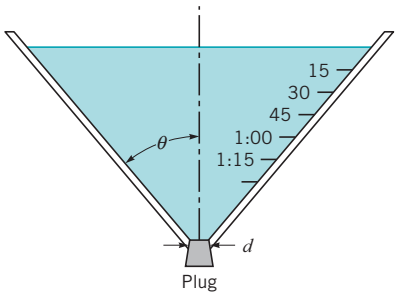

■ FIGURE P3.85

**3.86** Determine the flowrate through the submerged orifice shown in Fig. P3.86 if the contraction coefficient is $C_c = 0.63$.



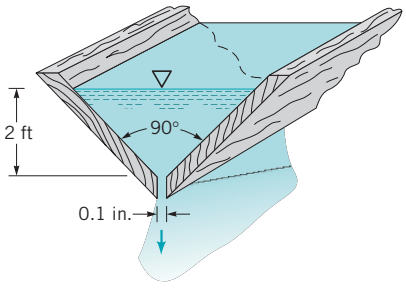

■ FIGURE P3.86

***3.87** An inexpensive timer is to be made from a funnel as indicated in Fig. P3.87. The funnel is filled to the top with water and the plug is removed at time $t = 0$ to allow the water to run out. Marks are to be placed on the wall of the funnel indicating the time in 15-s intervals, from 0 to 3 min (at which time the funnel becomes empty). If the funnel outlet has a diameter of $d = 0.1$ in., draw to scale the funnel with the timing marks for funnels with angles of $\theta = 30$, 45, and 60°. Repeat the problem if the diameter is changed to 0.05 in.




■ FIGURE P3.87

**3.88** A long water trough of triangular cross section is formed from two planks as is shown in Fig. P3.88. A gap of 0.1 in. remains at the junction of the two planks. If the water depth initially was 2 ft, how long a time does it take for the water depth to reduce to 1 ft?




■ FIGURE P3.88

***3.89** A spherical tank of diameter $D$ has a drain hole of diameter $d$ at its bottom. A vent at the top of the tank maintains atmospheric pressure at the liquid surface within the tank. The flow is quasi-steady and inviscid and the tank is full of water initially. Determine the water depth as a function of time, $h = h(t)$, and plot graphs of $h(t)$ for tank diameters of 1, 5, 10, and 20 ft if $d = 1$ in.

**3.90** When the drain plug is pulled, water flows from a hole in the bottom of a large, open cylindrical tank. Show that if viscous effects are negligible and if the flow is assumed to be quasisteady, then it takes 3.41 times longer to empty the entire tank than it does to empty the first half of the tank. Explain why this is so.

***3.91** The surface area, $A$, of the pond shown in Fig. P3.91 varies with the water depth, $h$, as shown in the table. At time $t = 0$ a valve is

opened and the pond is allowed to drain through a pipe of diameter $D$. If viscous effects are negligible and quasisteady conditions are assumed, plot the water depth as a function of time from when the valve is opened ($t = 0$) until the pond is drained for pipe diameters of $D = 0.5, 1.0, 1.5, 2.0, 2.5$, and 3.0 ft. Assume $h = 18$ ft at $t = 0$.

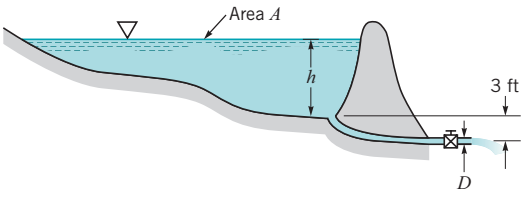



FIGURE P3.91

| $h$ (ft) | $A$ [acres (1 acre = 43,560 ft$^2$)] |
|---|---|
| 0 | 0 |
| 2 | 0.3 |
| 4 | 0.5 |
| 6 | 0.8 |
| 8 | 0.9 |
| 10 | 1.1 |
| 12 | 1.5 |
| 14 | 1.8 |
| 16 | 2.4 |
| 18 | 2.8 |

**3.92** Water flows through a horizontal branching pipe as shown in Fig. P3.92. Determine the pressure at section (3).

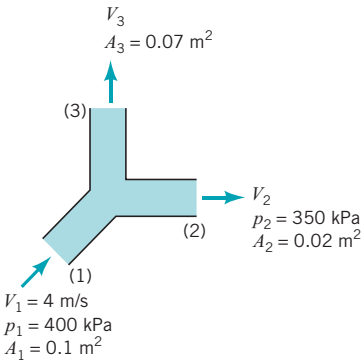



FIGURE P3.92

**3.93** Water flows through the horizontal branching pipe shown in Fig. P3.93 at a rate of 10 ft$^3$/s. If viscous effects are negligible, determine the water speed at section (2), the pressure at section (3), and the flowrate at section (4).

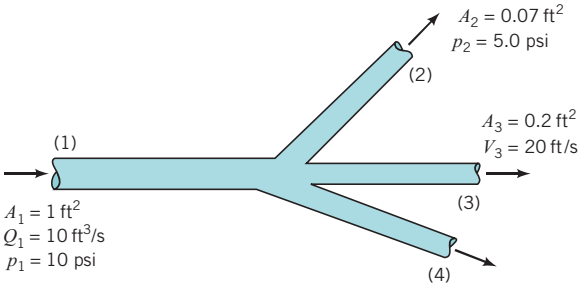



FIGURE P3.93

**3.94** Water flows from a large tank through a large pipe that splits into two smaller pipes as shown in Fig. P3.94. If viscous effects are negligible, determine the flowrate from the tank and the pressure at point (1).

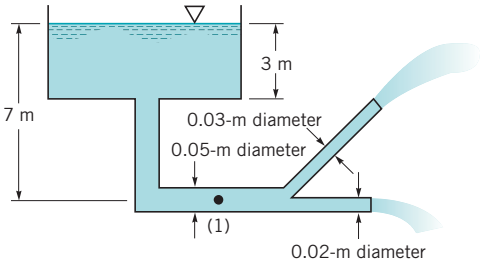



FIGURE P3.94

**3.95** An air cushion vehicle is supported by forcing air into the chamber created by a skirt around the periphery of the vehicle as shown in Fig. P3.95. The air escapes through the 3-in. clearance between the lower end of the skirt and the ground (or water). Assume the vehicle weighs 10,000 lb and is essentially rectangular in shape, 30 by 65 ft. The volume of the chamber is large enough so that the kinetic energy of the air within the chamber is negligible. Determine the flowrate, $Q$, needed to support the vehicle. If the ground clearance were reduced to 2 in., what flowrate would be needed? If the vehicle weight were reduced to 5000 lb and the ground clearance maintained at 3 in., what flowrate would be needed?

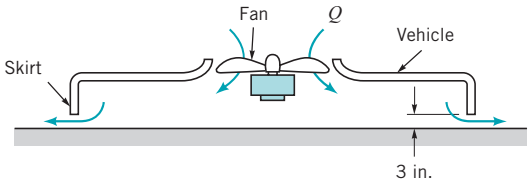



FIGURE P3.95

**3.96** Water flows from the pipe shown in Fig. P3.96 as a free jet and strikes a circular flat plate. The flow geometry shown is axisymmetrical. Determine the flowrate and the manometer reading, $H$.

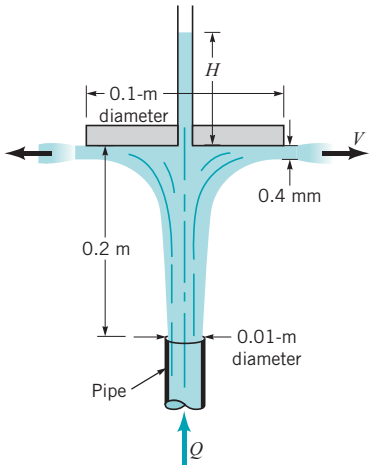
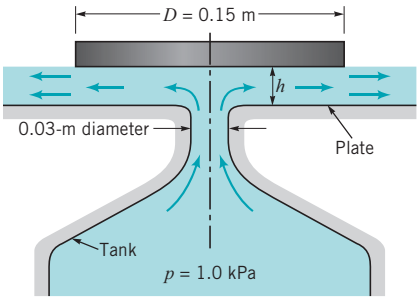



FIGURE P3.96

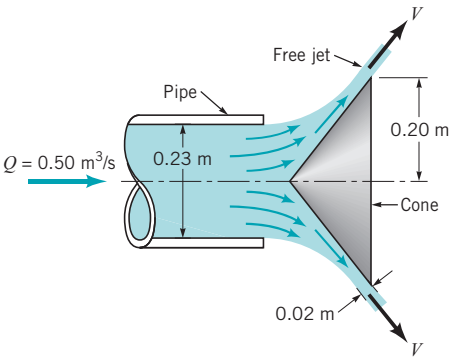
**3.97** Air flows from a hole of diameter 0.03 m in a flat plate as shown in Fig. P3.97. A circular disk of diameter $D$ is placed a distance $h$ from the lower plate. The pressure in the tank is maintained at 1 kPa. Determine the flowrate as a function of $h$ if viscous




■ FIGURE P3.97

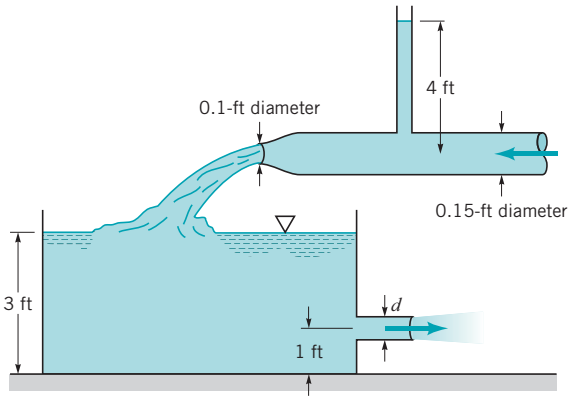
effects and elevation changes are assumed negligible and the flow exits radially from the circumference of the circular disk with uniform velocity.

**3.98** A conical plug is used to regulate the air flow from the pipe shown in Fig. P3.98. The air leaves the edge of the cone with a uniform thickness of 0.02 m. If viscous effects are negligible and the flowrate is 0.50 $m^3/s$, determine the pressure within the pipe.



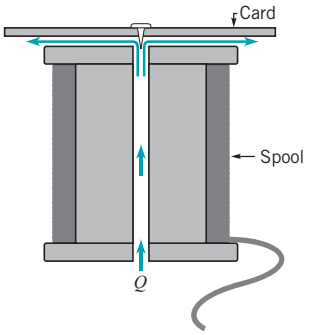

■ FIGURE P3.98

**3.99** Water flows steadily from a nozzle into a large tank as shown in Fig. P3.99. The water then flows from the tank as a jet of diameter $d$. Determine the value of $d$ if the water level in the tank remains constant. Viscous effects are negligible.




■ FIGURE P3.99

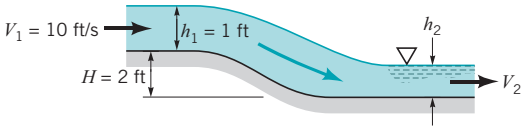
**3.100** A small card is placed on top of a spool as shown in Fig. P3.100. It is not possible to blow the card off the spool by blowing air through the hole in the center of the spool. The harder one blows, the harder the card "sticks" to the spool. In fact, by blowing hard enough it is possible to keep the card against the




■ FIGURE P3.100

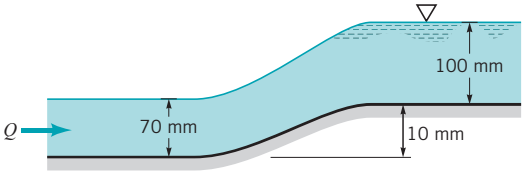
spool with the spool turned upside down. (*Note:* It may be necessary to use a thumb tack to prevent the card from sliding from the spool.) Explain this phenomenon.

**3.101** Water flows down the sloping ramp shown in Fig. P3.101 with negligible viscous effects. The flow is uniform at sections (1) and (2). For the conditions given, show that three solutions for the downstream depth, $h_2$, are obtained by use of the Bernoulli and continuity equations. However, show that only two of these solutions are realistic. Determine these values.



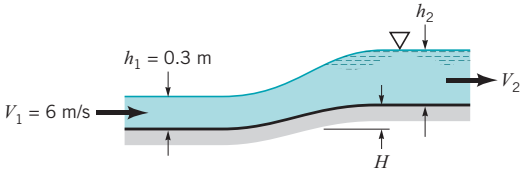

■ FIGURE P3.101

**3.102** Water flows in a rectangular channel that is 2.0 m wide as shown in Fig. P3.102. The upstream depth is 70 mm. The water surface rises 40 mm as it passes over a portion where the channel bottom rises 10 mm. If viscous effects are negligible, what is the flowrate?




■ FIGURE P3.102

***3.103** Water flows up the ramp shown in Fig. P3.103 with negligible viscous losses. The upstream depth and velocity are maintained at $h_1 = 0.3$ m and $V_1 = 6$ m/s. Plot a graph of the downstream depth, $h_2$, as a function of the ramp height, $H$, for $0 \le H \le 2$ m. Note that for each value of $H$ there are three solutions, not all of which are realistic.




■ FIGURE P3.103

### Section 3.6.3 Flowrate Measurement (Also see Lab Problems 3.119 and 3.121.)

**3.104** Obtain a photograph/image of a situation that involves some type of flow meter. Print this photo and write a brief paragraph that describes the situation involved.

**3.105** A Venturi meter with a minimum diameter of 3 in. is to be used to measure the flowrate of water through a 4-in.-diameter pipe. Determine the pressure difference indicated by the pressure gage attached to the flow meter if the flowrate is 0.5 $ft^3/s$ and viscous effects are negligible.

**3.106** Determine the flowrate through the Venturi meter shown in Fig. P3.106 if ideal conditions exist.

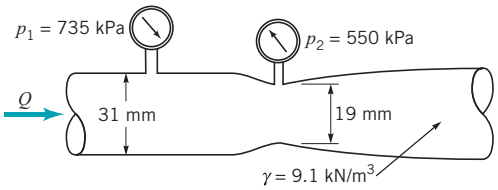



FIGURE P3.106

**3.107** For what flowrate through the Venturi meter of Problem 3.106 will cavitation begin if $p_1 = 275$ kPa gage, atmospheric pressure is 101 kPa (abs), and the vapor pressure is 3.6 kPa (abs)?

**3.108** What diameter orifice hole, $d$, is needed if under ideal conditions the flowrate through the orifice meter of Fig. P3.108 is to be 30 gal/min of seawater with $p_1 - p_2 = 2.37$ lb/in.$^2$? The contraction coefficient is assumed to be 0.63.

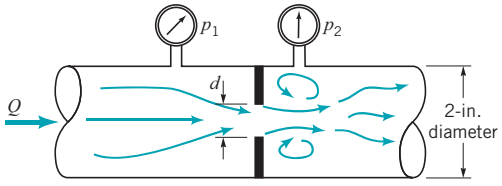



FIGURE P3.108

**3.109** Water flows over a weir plate (see Video V10.13) which has a parabolic opening as shown in Fig. P3.109. That is, the opening in the weir plate has a width $CH^{1/2}$, where $C$ is a constant. Determine the functional dependence of the flowrate on the head, $Q = Q(H)$.

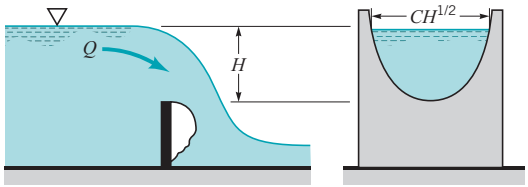



FIGURE P3.109

**3.110** A weir (see Video V10.13) of trapezoidal cross section is used to measure the flowrate in a channel as shown in Fig. P3.110. If the flowrate is $Q_0$ when $H = \ell/2$, what flowrate is expected when $H = \ell$?

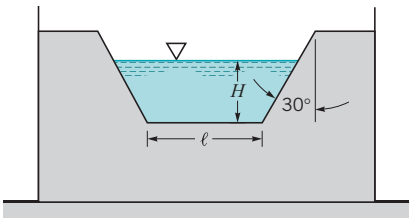



FIGURE P3.110

**3.111** The flowrate in a water channel is sometimes determined by use of a device called a Venturi flume. As shown in Fig. P3.111, this device consists simply of a hump on the bottom of the channel. If the water surface dips a distance of 0.07 m for the conditions shown, what is the flowrate per width of the channel? Assume the velocity is uniform and viscous effects are negligible.

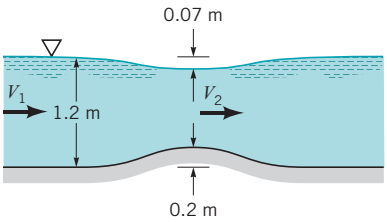



FIGURE P3.111

**3.112** Water flows under the inclined sluice gate shown in Fig. P3.112. Determine the flowrate if the gate is 8 ft wide.

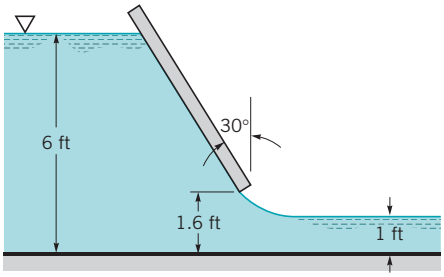



FIGURE P3.112

### Section 3.7 The Energy Line and the Hydraulic Grade Line

**3.113** Water flows in a vertical pipe of 0.15-m diameter at a rate of 0.2 $m^3/s$ and a pressure of 200 kPa at an elevation of 25 m. Determine the velocity head and pressure head at elevations of 20 and 55 m.

**3.114** Draw the energy line and the hydraulic grade line for the flow shown in Problem 3.78.

**3.115** Draw the energy line and the hydraulic grade line for the flow of Problem 3.75.

**3.116** Draw the energy line and hydraulic grade line for the flow shown in Problem 3.64.

### Section 3.8 Restrictions on the Use of the Bernoulli Equation

**3.117** Obtain a photograph/image of a flow in which it would not be appropriate to use the Bernoulli equation. Print this photo and write a brief paragraph that describes the situation involved.

## ■ Lab Problems

**3.118** This problem involves the pressure distribution between two parallel circular plates. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

**3.119** This problem involves the calibration of a nozzle-type flow meter. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

**3.120** This problem involves the pressure distribution in a two-dimensional channel. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

**3.121** This problem involves the determination of the flowrate under a sluice gate as a function of the water depth. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

## ■ Life Long Learning Problems

**3.122** The concept of the use of a Pitot-static tube to measure the airspeed of an airplane is rather straightforward. However, the design and manufacture of reliable, accurate, inexpensive Pitot-static tube airspeed indicators is not necessarily simple. Obtain information about the design and construction of modern Pitot-static tubes. Summarize your findings in a brief report.

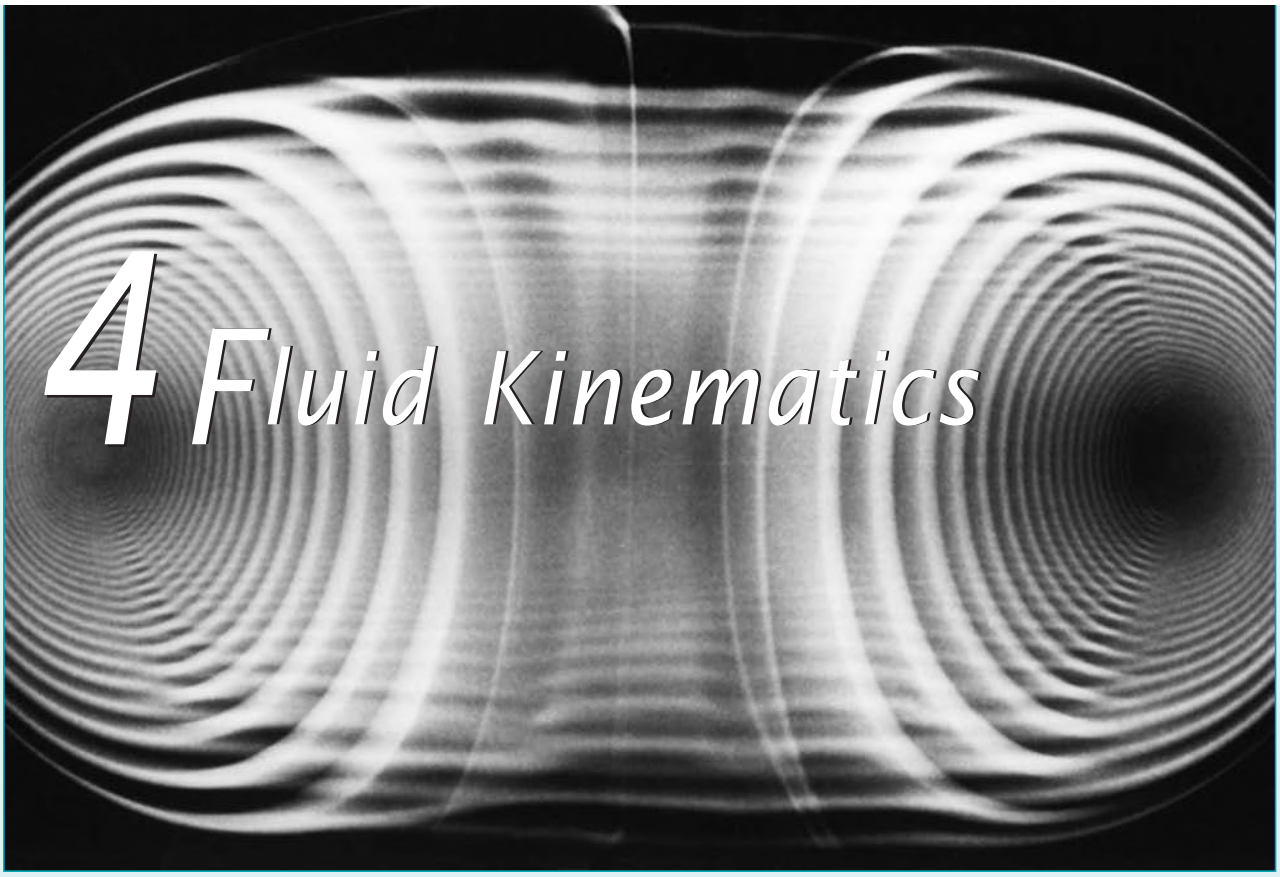
**3.123** In recent years damage due to hurricanes has been significant, particularly in the southeastern United States. The low barometric pressure, high winds, and high tides generated by hurricanes can combine to cause considerable damage. According to some experts, in the coming years hurricane frequency may increase because of global warming. Obtain information about the fluid mechanics of hurricanes. Summarize your findings in a brief report.

**3.124** Orifice, nozzle, or Venturi flow meters have been used for a long time to predict accurately the flowrate in pipes. However, recently there have been several new concepts suggested or used for such flowrate measurements. Obtain information about new methods to obtain pipe flowrate information. Summarize your findings in a brief report.

**3.125** Ultra-high-pressure, thin jets of liquids can be used to cut various materials ranging from leather to steel and beyond. Obtain information about new methods and techniques proposed for liquid jet cutting and investigate how they may alter various manufacturing processes. Summarize your findings in a brief report.

## ■ FE Exam Problems

Sample FE (Fundamentals of Engineering) exam questions for fluid mechanics are provided on the book's web site, www.wiley.com/college/munson.



# 4 Fluid Kinematics

CHAPTER OPENING PHOTO: A vortex ring: The complex, three-dimensional structure of a smoke ring is indicated in this cross-sectional view. (Smoke in air.) [Photograph courtesy of R. H. Magarvey and C. S. MacLatchy (Ref. 4).]

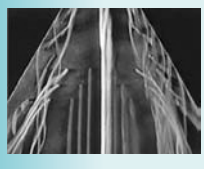
## Learning Objectives

After completing this chapter, you should be able to:

- discuss the differences between the Eulerian and Lagrangian descriptions of fluid motion.
- identify various flow characteristics based on the velocity field.
- determine the streamline pattern and acceleration field given a velocity field.
- discuss the differences between a system and control volume.
- apply the Reynolds transport theorem and the material derivative.



*V4.1 Streaklines*



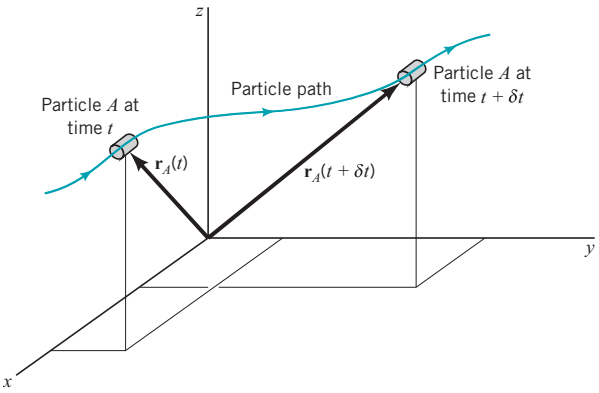
In this chapter we will discuss various aspects of fluid motion without being concerned with the actual forces necessary to produce the motion. That is, we will consider the *kinematics* of the motion—the velocity and acceleration of the fluid, and the description and visualization of its motion. The analysis of the specific forces necessary to produce the motion (the *dynamics* of the motion) will be discussed in detail in the following chapters. A wide variety of useful information can be gained from a thorough understanding of fluid kinematics. Such an understanding of how to describe and observe fluid motion is an essential step to the complete understanding of fluid dynamics.

## 4.1 The Velocity Field

In general, fluids flow. That is, there is a net motion of molecules from one point in space to another point as a function of time. As is discussed in Chapter 1, a typical portion of fluid contains so many molecules that it becomes totally unrealistic (except in special cases) for us to attempt to



CMS Cepcor EX 1046 Page 57
CMS Cepcor v. Sandvik IPR2024-01380

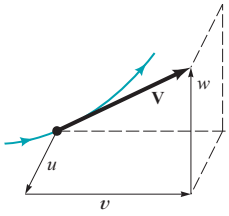


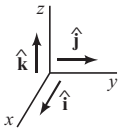

■ **FIGURE 4.1** **Particle location in terms of its position vector.**



*V4.2 Velocity field*

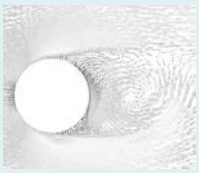







*V4.3 Cylinder-velocity vectors*



account for the motion of individual molecules. Rather, we employ the continuum hypothesis and consider fluids to be made up of fluid particles that interact with each other and with their surroundings. Each particle contains numerous molecules. Thus, we can describe the flow of a fluid in terms of the motion of fluid particles rather than individual molecules. This motion can be described in terms of the velocity and acceleration of the fluid particles.

The infinitesimal particles of a fluid are tightly packed together (as is implied by the continuum assumption). Thus, at a given instant in time, a description of any fluid property (such as density, pressure, velocity, and acceleration) may be given as a function of the fluid's location. This representation of fluid parameters as functions of the spatial coordinates is termed a ***field representation*** of the flow. Of course, the specific field representation may be different at different times, so that to describe a fluid flow we must determine the various parameters not only as a function of the spatial coordinates ($x$, $y$, $z$, for example) but also as a function of time, $t$. Thus, to completely specify the temperature, $T$, in a room we must specify the temperature field, $T = T(x, y, z, t)$, throughout the room (from floor to ceiling and wall to wall) at any time of the day or night.

Shown in the margin figure is one of the most important fluid variables, the ***velocity field,***

$$\mathbf{V} = u(x, y, z, t)\hat{\mathbf{i}} + v(x, y, z, t)\hat{\mathbf{j}} + w(x, y, z, t)\hat{\mathbf{k}}$$

where $u$, $v$, and $w$ are the $x$, $y$, and $z$ components of the velocity vector. By definition, the velocity of a particle is the time rate of change of the position vector for that particle. As is illustrated in Fig. 4.1, the position of particle $A$ relative to the coordinate system is given by its *position vector*, $\mathbf{r}_A$, which (if the particle is moving) is a function of time. The time derivative of this position gives the *velocity* of the particle, $d\mathbf{r}_A/dt = \mathbf{V}_A$. By writing the velocity for all of the particles we can obtain the field description of the velocity vector $\mathbf{V} = \mathbf{V}(x, y, z, t)$.

Since the velocity is a vector, it has both a direction and a magnitude. The magnitude of $\mathbf{V}$, denoted $V = |\mathbf{V}| = (u^2 + v^2 + w^2)^{1/2}$, is the speed of the fluid. (It is very common in practical situations to call $V$ velocity rather than speed, i.e., "the velocity of the fluid is 12 m/s.") As is discussed in the next section, a change in velocity results in an acceleration. This acceleration may be due to a change in speed and/or direction.

## Fluids in the News

**Follow those particles** Superimpose two photographs of a bouncing ball taken a short time apart and draw an arrow between the two images of the ball. This arrow represents an approximation of the velocity (displacement/time) of the ball. The particle image velocimeter (PIV) uses this technique to provide the instantaneous *velocity field* for a given cross section of a flow. The flow being studied is seeded with numerous micron-sized particles which are small enough to follow the flow yet big enough to reflect enough light to be captured by the camera. The flow is illuminated with a light sheet from a double-pulsed laser. A digital camera captures both light pulses on the same image frame, allowing the movement of the particles to be tracked. By using appropriate computer software to carry out a pixel-by-pixel interrogation of the double image, it is possible to track the motion of the particles and determine the two components of velocity in the given cross section of the flow. By using two cameras in a stereoscopic arrangement it is possible to determine all three components of velocity. (See Problem 4.62.)

CMS Cepcor EX 1046 Page 58
CMS Cepcor v. Sandvik IPR2024-01380

## EXAMPLE 4.1 Velocity Field Representation

**GIVEN** A velocity field is given by $\mathbf{V} = (V_0/\ell)(-x\hat{\mathbf{i}} + y\hat{\mathbf{j}})$ where $V_0$ and $\ell$ are constants.

**FIND** At what location in the flow field is the speed equal to $V_0$? Make a sketch of the velocity field for $x \geq 0$ by drawing arrows representing the fluid velocity at representative locations.

## SOLUTION

The $x$, $y$, and $z$ components of the velocity are given by $u = -V_0 x/\ell$, $v = V_0 y/\ell$, and $w = 0$ so that the fluid speed, $V$, is

$$V = (u^2 + v^2 + w^2)^{1/2} = \frac{V_0}{\ell}(x^2 + y^2)^{1/2} \qquad (1)$$

The speed is $V = V_0$ at any location on the circle of radius $\ell$ centered at the origin $[(x^2 + y^2)^{1/2} = \ell]$ as shown in Fig. E4.1*a*. **(Ans)**

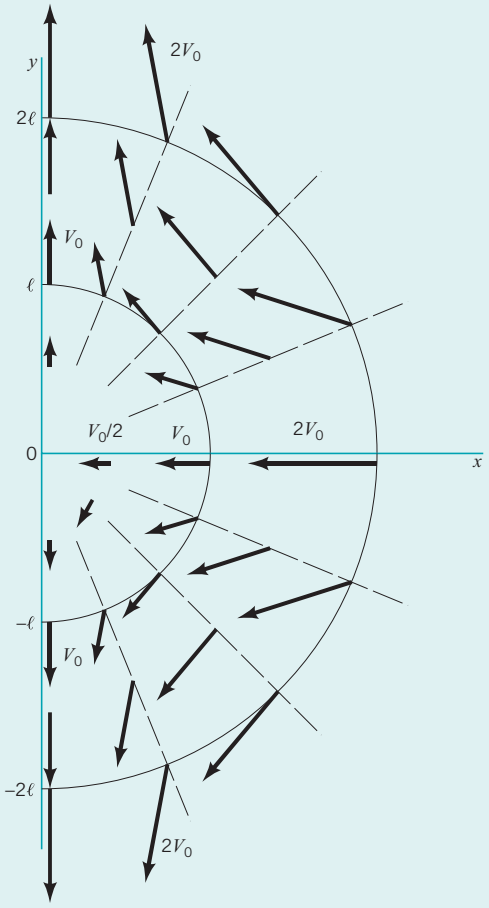
The direction of the fluid velocity relative to the $x$ axis is given in terms of $\theta = \arctan(v/u)$ as shown in Fig. E4.1*b*. For this flow

$$\tan\theta = \frac{v}{u} = \frac{V_0 y/\ell}{-V_0 x/\ell} = \frac{y}{-x}$$

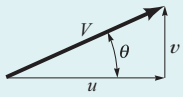
Thus, along the $x$ axis ($y = 0$) we see that $\tan\theta = 0$, so that $\theta = 0°$ or $\theta = 180°$. Similarly, along the $y$ axis ($x = 0$) we obtain $\tan\theta = \pm\infty$ so that $\theta = 90°$ or $\theta = 270°$. Also, for $y = 0$ we find $\mathbf{V} = (-V_0 x/\ell)\hat{\mathbf{i}}$, while for $x = 0$ we have $\mathbf{V} = (V_0 y/\ell)\hat{\mathbf{j}}$, indicating (if $V_0 > 0$) that the flow is directed away from the origin along the $y$ axis and toward the origin along the $x$ axis as shown in Fig. E4.1*a*.

By determining $\mathbf{V}$ and $\theta$ for other locations in the $x$–$y$ plane, the velocity field can be sketched as shown in the figure. For example, on the line $y = x$ the velocity is at a 45° angle relative to the $x$ axis ($\tan\theta = v/u = -y/x = -1$). At the origin $x = y = 0$ so that $\mathbf{V} = 0$. This point is a stagnation point. The farther from the origin the fluid is, the faster it is flowing (as seen from Eq. 1). By careful consideration of the velocity field it is possible to determine considerable information about the flow.

**COMMENT** The velocity field given in this example approximates the flow in the vicinity of the center of the sign shown in Fig. E4.1*c*. When wind blows against the sign, some air flows over the sign, some under it, producing a stagnation point as indicated.




(*a*)

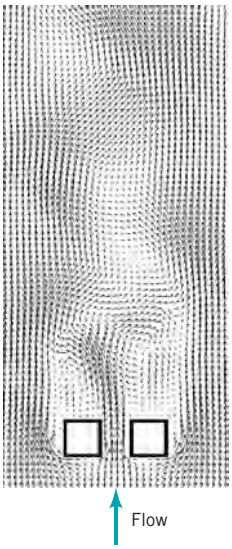



(*b*)




(*c*)

**FIGURE E4.1**

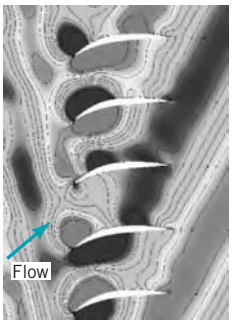



The figure in the margin shows the velocity field (i.e., velocity vectors) for flow past two square bars. It is possible to obtain much qualitative and quantitative information for complex flows by using plots such as this.

### 4.1.1 Eulerian and Lagrangian Flow Descriptions

There are two general approaches in analyzing fluid mechanics problems (or problems in other branches of the physical sciences, for that matter). The first method, called the ***Eulerian method,*** uses the field concept introduced above. In this case, the fluid motion is given by completely prescribing the necessary properties (pressure, density, velocity, etc.) as functions of space and time. From this method we obtain information about the flow in terms of what happens at fixed points in space as the fluid flows through those points.

A typical Eulerian representation of the flow is shown by the figure in the margin which involves flow past a row of turbine blades as occurs in a jet engine. The pressure field is indicated by using a contour plot showing lines of constant pressure, with grey shading indicating the intensity of the pressure.




*Either Eulerian or Lagrangian methods can be used to describe flow fields.*

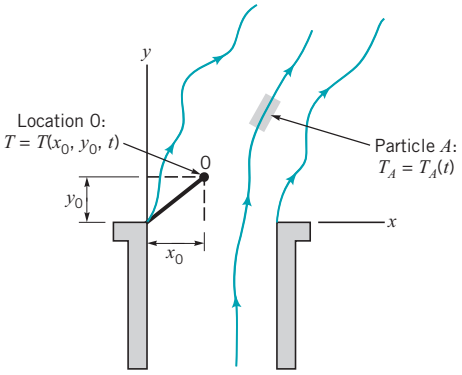
The second method, called the ***Lagrangian method,*** involves following individual fluid particles as they move about and determining how the fluid properties associated with these particles change as a function of time. That is, the fluid particles are "tagged" or identified, and their properties determined as they move.

The difference between the two methods of analyzing fluid flow problems can be seen in the example of smoke discharging from a chimney, as is shown in Fig. 4.2. In the Eulerian method one may attach a temperature-measuring device to the top of the chimney (point 0) and record the temperature at that point as a function of time. At different times there are different fluid particles passing by the stationary device. Thus, one would obtain the temperature, $T$, for that location ($x = x_0$, $y = y_0$, and $z = z_0$) as a function of time. That is, $T = T(x_0, y_0, z_0, t)$. The use of numerous temperature-measuring devices fixed at various locations would provide the temperature field, $T = T(x, y, z, t)$. The temperature of a particle as a function of time would not be known unless the location of the particle were known as a function of time.

In the Lagrangian method, one would attach the temperature-measuring device to a particular fluid particle (particle $A$) and record that particle's temperature as it moves about. Thus, one would obtain that particle's temperature as a function of time, $T_A = T_A(t)$. The use of many such measuring devices moving with various fluid particles would provide the temperature of these fluid particles as a function of time. The temperature would not be known as a function of position unless the location of each particle were known as a function of time. If enough information in Eulerian form is available, Lagrangian information can be derived from the Eulerian data—and vice versa.

Example 4.1 provides an Eulerian description of the flow. For a Lagrangian description we would need to determine the velocity as a function of time for each particle as it flows along from one point to another.

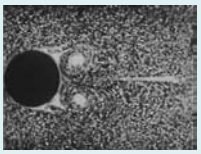
In fluid mechanics it is usually easier to use the Eulerian method to describe a flow—in either experimental or analytical investigations. There are, however, certain instances in which the Lagrangian method is more convenient. For example, some numerical fluid mechanics calculations are based on determining the motion of individual fluid particles (based on the appropriate interactions among the particles), thereby describing the motion in Lagrangian terms. Similarly, in




**■ FIGURE 4.2** **Eulerian and Lagrangian descriptions of temperature of a flowing fluid.**

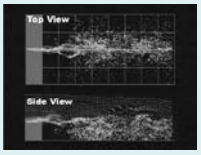


*V4.4 Follow the particles (experiment)*





*V4.5 Follow the particles (computer)*




*Most flow fields are actually three-dimensional.*



*V4.6 Flow past a wing*



some experiments individual fluid particles are "tagged" and are followed throughout their motion, providing a Lagrangian description. Oceanographic measurements obtained from devices that flow with the ocean currents provide this information. Similarly, by using X-ray opaque dyes it is possible to trace blood flow in arteries and to obtain a Lagrangian description of the fluid motion. A Lagrangian description may also be useful in describing fluid machinery (such as pumps and turbines) in which fluid particles gain or lose energy as they move along their flow paths.

Another illustration of the difference between the Eulerian and Lagrangian descriptions can be seen in the following biological example. Each year thousands of birds migrate between their summer and winter habitats. Ornithologists study these migrations to obtain various types of important information. One set of data obtained is the rate at which birds pass a certain location on their migration route (birds per hour). This corresponds to an Eulerian description—"flowrate" at a given location as a function of time. Individual birds need not be followed to obtain this information. Another type of information is obtained by "tagging" certain birds with radio transmitters and following their motion along the migration route. This corresponds to a Lagrangian description—"position" of a given particle as a function of time.

### 4.1.2 One-, Two-, and Three-Dimensional Flows

Generally, a fluid flow is a rather complex three-dimensional, time-dependent phenomenon—$\mathbf{V} = \mathbf{V}(x, y, z, t) = u\hat{\mathbf{i}} + v\hat{\mathbf{j}} + w\hat{\mathbf{k}}$. In many situations, however, it is possible to make simplifying assumptions that allow a much easier understanding of the problem without sacrificing needed accuracy. One of these simplifications involves approximating a real flow as a simpler one- or two-dimensional flow.

In almost any flow situation, the velocity field actually contains all three velocity components ($u$, $v$, and $w$, for example). In many situations the ***three-dimensional flow*** characteristics are important in terms of the physical effects they produce. (See the photograph at the beginning of Chapter 4.) For these situations it is necessary to analyze the flow in its complete three-dimensional character. Neglect of one or two of the velocity components in these cases would lead to considerable misrepresentation of the effects produced by the actual flow.

The flow of air past an airplane wing provides an example of a complex three-dimensional flow. A feel for the three-dimensional structure of such flows can be obtained by studying Fig. 4.3, which is a photograph of the flow past a model wing; the flow has been made visible by using a flow visualization technique.

In many situations one of the velocity components may be small (in some sense) relative to the two other components. In situations of this kind it may be reasonable to neglect the smaller component and assume ***two-dimensional flow.*** That is, $\mathbf{V} = u\hat{\mathbf{i}} + v\hat{\mathbf{j}}$, where $u$ and $v$ are functions of $x$ and $y$ (and possibly time, $t$).

It is sometimes possible to further simplify a flow analysis by assuming that two of the velocity components are negligible, leaving the velocity field to be approximated as a ***one-dimensional flow*** field. That is, $\mathbf{V} = u\hat{\mathbf{i}}$. As we will learn from examples throughout the remainder of the book, although there are very few, if any, flows that are truly one-dimensional, there are

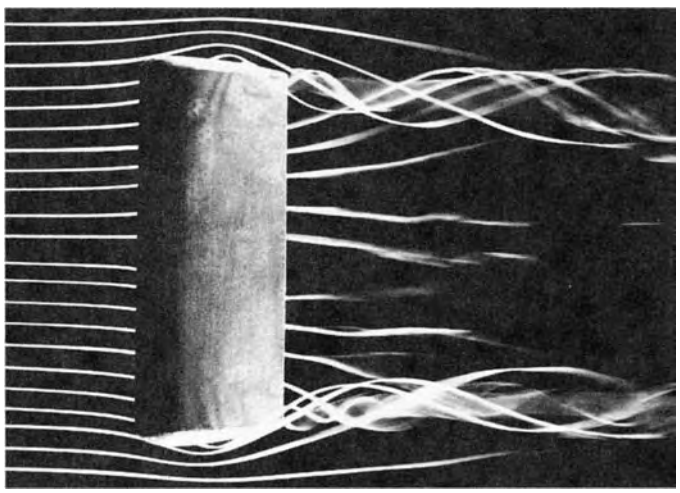
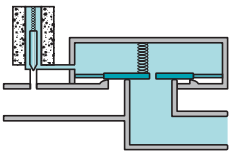
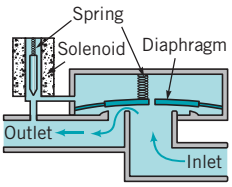


**FIGURE 4.3**
**Flow visualization of the complex three-dimensional flow past a model wing. (Photograph by M. R. Head.)**

CMS Cepcor EX 1046 Page 61
CMS Cepcor v. Sandvik IPR2024-01380



Solenoid off, valve closed




Solenoid on, valve open

many flow fields for which the one-dimensional flow assumption provides a reasonable approximation. There are also many flow situations for which use of a one-dimensional flow field assumption will give completely erroneous results.

### 4.1.3 Steady and Unsteady Flows

In the previous discussion we have assumed ***steady flow***—the velocity at a given point in space does not vary with time, $\partial \mathbf{V}/\partial t = 0$. In reality, almost all flows are unsteady in some sense. That is, the velocity does vary with time. It is not difficult to believe that ***unsteady flows*** are usually more difficult to analyze (and to investigate experimentally) than are steady flows. Hence, considerable simplicity often results if one can make the assumption of steady flow without compromising the usefulness of the results. Among the various types of unsteady flows are nonperiodic flow, periodic flow, and truly random flow. Whether or not unsteadiness of one or more of these types must be included in an analysis is not always immediately obvious.

An example of a nonperiodic, unsteady flow is that produced by turning off a faucet to stop the flow of water. Usually this unsteady flow process is quite mundane and the forces developed as a result of the unsteady effects need not be considered. However, if the water is turned off suddenly (as with the electrically operated valve in a dishwasher shown in the figure in the margin), the unsteady effects can become important [as in the "water hammer" effects made apparent by the loud banging of the pipes under such conditions (Ref. 1)].

In other flows the unsteady effects may be periodic, occurring time after time in basically the same manner. The periodic injection of the air–gasoline mixture into the cylinder of an automobile engine is such an example. The unsteady effects are quite regular and repeatable in a regular sequence. They are very important in the operation of the engine.



*V4.7 Flow types*



## Fluids in the News

**New pulsed liquid-jet scalpel** High-speed liquid-jet cutters are used for cutting a wide variety of materials such as leather goods, jigsaw puzzles, plastic, ceramic, and metal. Typically, compressed air is used to produce a continuous stream of water that is ejected from a tiny nozzle. As this stream impacts the material to be cut, a high pressure (the stagnation pressure) is produced on the surface of the material, thereby cutting the material. Such liquid-jet cutters work well in air, but are difficult to control if the jet must pass through a liquid as often happens in surgery. Researchers have developed a new pulsed jet cutting tool that may allow surgeons to perform microsurgery on tissues that are immersed in water. Rather than using a steady water jet, the system uses *unsteady flow.* A high-energy electrical discharge inside the nozzle momentarily raises the temperature of the microjet to approximately 10,000 °C. This creates a rapidly expanding vapor bubble in the nozzle and expels a tiny fluid jet from the nozzle. Each electrical discharge creates a single, brief jet, which makes a small cut in the material.

In many situations the unsteady character of a flow is quite random. That is, there is no repeatable sequence or regular variation to the unsteadiness. This behavior occurs in *turbulent flow* and is absent from *laminar flow.* The "smooth" flow of highly viscous syrup onto a pancake represents a "deterministic" laminar flow. It is quite different from the turbulent flow observed in the "irregular" splashing of water from a faucet onto the sink below it. The "irregular" gustiness of the wind represents another random turbulent flow. The differences between these types of flows are discussed in considerable detail in Chapters 8 and 9.

It must be understood that the definition of steady or unsteady flow pertains to the behavior of a fluid property as observed at a fixed point in space. For steady flow, the values of all fluid properties (velocity, temperature, density, etc.) at any fixed point are independent of time. However, the value of those properties for a given fluid particle may change with time as the particle flows along, even in steady flow. Thus, the temperature of the exhaust at the exit of a car's exhaust pipe may be constant for several hours, but the temperature of a fluid particle that left the exhaust pipe five minutes ago is lower now than it was when it left the pipe, even though the flow is steady.



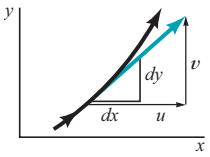
*V4.8 Jupiter red spot*




### 4.1.4 Streamlines, Streaklines, and Pathlines

Although fluid motion can be quite complicated, there are various concepts that can be used to help in the visualization and analysis of flow fields. To this end we discuss the use of streamlines,

CMS Cepcor EX 1046 Page 62
CMS Cepcor v. Sandvik IPR2024-01380




streaklines, and pathlines in flow analysis. The streamline is often used in analytical work while the streakline and pathline are often used in experimental work.

A ***streamline*** is a line that is everywhere tangent to the velocity field. If the flow is steady, nothing at a fixed point (including the velocity direction) changes with time, so the streamlines are fixed lines in space. (See the photograph at the beginning of Chapter 6.) For unsteady flows the streamlines may change shape with time. Streamlines are obtained analytically by integrating the equations defining lines tangent to the velocity field. As illustrated in the margin figure, for two-dimensional flows the slope of the streamline, $dy/dx$, must be equal to the tangent of the angle that the velocity vector makes with the $x$ axis or

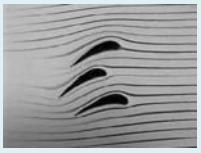
$$\frac{dy}{dx} = \frac{v}{u} \tag{4.1}$$

If the velocity field is known as a function of $x$ and $y$ (and $t$ if the flow is unsteady), this equation can be integrated to give the equation of the streamlines.

For unsteady flow there is no easy way to produce streamlines experimentally in the laboratory. As discussed below, the observation of dye, smoke, or some other tracer injected into a flow can provide useful information, but for unsteady flows it is not necessarily information about the streamlines.



*V4.9 Streamlines*



## EXAMPLE 4.2 Streamlines for a Given Velocity Field

**GIVEN** Consider the two-dimensional steady flow discussed in Example 4.1, $\mathbf{V} = (V_0/\ell)(-x\hat{\mathbf{i}} + y\hat{\mathbf{j}})$.

**FIND** Determine the streamlines for this flow.

### SOLUTION

Since

$$u = (-V_0/\ell)x \text{ and } v = (V_0/\ell)y \tag{1}$$

it follows that streamlines are given by solution of the equation

$$\frac{dy}{dx} = \frac{v}{u} = \frac{(V_0/\ell)y}{-(V_0/\ell)x} = -\frac{y}{x}$$

in which variables can be separated and the equation integrated to give

$$\int \frac{dy}{y} = -\int \frac{dx}{x}$$

or

$$\ln y = -\ln x + \text{constant}$$

Thus, along the streamline

$$xy = C, \qquad \text{where } C \text{ is a constant} \tag{Ans}$$

By using different values of the constant $C$, we can plot various lines in the $x$–$y$ plane—the streamlines. The streamlines for $x \geq 0$ are plotted in Fig. E4.2. A comparison of this figure with Fig. E4.1*a* illustrates the fact that streamlines are lines tangent to the velocity field.

**COMMENT** Note that a flow is not completely specified by the shape of the streamlines alone. For example, the streamlines for the flow with $V_0/\ell = 10$ have the same shape as those for the flow with $V_0/\ell = -10$. However, the direction of the flow is opposite for these two cases. The arrows in Fig. E4.2 representing the flow direction are correct for $V_0/\ell = 10$ since, from Eq. 1, $u = -10x$ and $v = 10y$. That is, the flow is from right to left. For $V_0/\ell = -10$ the arrows are reversed. The flow is from left to right.

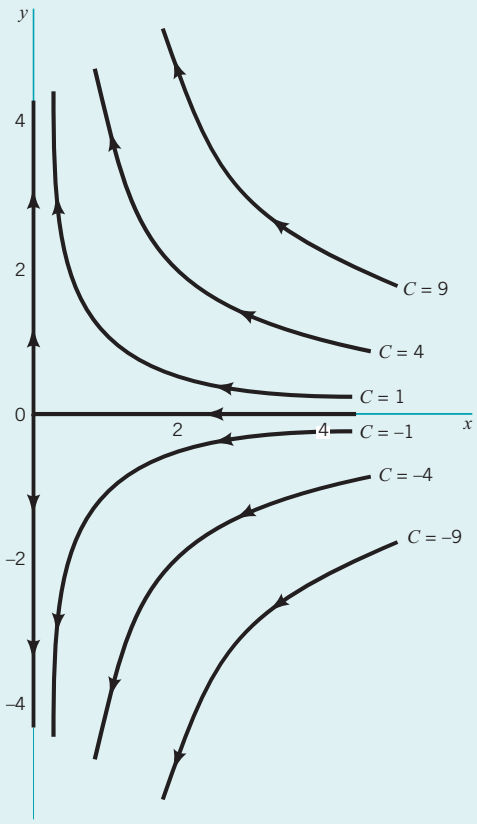
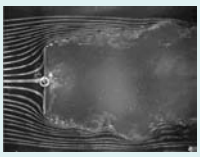



FIGURE E4.2



*V4.10 Streaklines*



A ***streakline*** consists of all particles in a flow that have previously passed through a common point. Streaklines are more of a laboratory tool than an analytical tool. They can be obtained by taking instantaneous photographs of marked particles that all passed through a given location in the flow field at some earlier time. Such a line can be produced by continuously injecting marked fluid (neutrally buoyant smoke in air, or dye in water) at a given location (Ref. 2). (See Fig. 9.1.) If the flow is steady, each successively injected particle follows precisely behind the previous one, forming a steady streakline that is exactly the same as the streamline through the injection point.

For unsteady flows, particles injected at the same point at different times need not follow the same path. An instantaneous photograph of the marked fluid would show the streakline at that instant, but it would not necessarily coincide with the streamline through the point of injection at that particular time nor with the streamline through the same injection point at a different time (see Example 4.3).

The third method used for visualizing and describing flows involves the use of ***pathlines.*** A pathline is the line traced out by a given particle as it flows from one point to another. The pathline is a Lagrangian concept that can be produced in the laboratory by marking a fluid particle (dying a small fluid element) and taking a time exposure photograph of its motion. (See the photograph at the beginning of Chapter 7)

## Fluids in the News

**Air bridge spanning the oceans** It has long been known that large quantities of material are transported from one location to another by airborne dust particles. It is estimated that 2 billion metric tons of dust are lifted into the atmosphere each year. Most of these particles settle out fairly rapidly, but significant amounts travel large distances. Scientists are beginning to understand the full impact of this phenomena—it is not only the tonnage transported, but the type of material transported that is significant. In addition to the mundane inert material we all term "dust," it is now known that a wide variety of hazardous materials and organisms are also carried along these literal *particle paths*. Satellite images reveal the amazing rate by which desert soils and other materials are transformed into airborne particles as a result of storms that produce strong winds. Once the tiny particles are aloft, they may travel thousands of miles, crossing the oceans and eventually being deposited on other continents. For the health and safety of all, it is important that we obtain a better understanding of the air bridges that span the oceans and also understand the ramification of such material transport.

*For steady flow, streamlines, streaklines, and pathlines are the same.*

If the flow is steady, the path taken by a marked particle (a pathline) will be the same as the line formed by all other particles that previously passed through the point of injection (a streakline). For such cases these lines are tangent to the velocity field. Hence, pathlines, streamlines, and streaklines are the same for steady flows. For unsteady flows none of these three types of lines need be the same (Ref. 3). Often one sees pictures of "streamlines" made visible by the injection of smoke or dye into a flow as is shown in Fig. 4.3. Actually, such pictures show streaklines rather than streamlines. However, for steady flows the two are identical; only the nomenclature is incorrectly used.

## EXAMPLE 4.3 Comparison of Streamlines, Pathlines, and Streaklines

**GIVEN** Water flowing from the oscillating slit shown in Fig. E4.3*a* produces a velocity field given by $\mathbf{V} = u_0 \sin[\omega(t - y/v_0)]\hat{\mathbf{i}} + v_0\hat{\mathbf{j}}$, where $u_0$, $v_0$, and $\omega$ are constants. Thus, the $y$ component of velocity remains constant ($v = v_0$) and the $x$ component of velocity at $y = 0$ coincides with the velocity of the oscillating sprinkler head [$u = u_0 \sin(\omega t)$ at $y = 0$].

**FIND** (a) Determine the streamline that passes through the origin at $t = 0$; at $t = \pi/2\omega$. (b) Determine the pathline of the particle that was at the origin at $t = 0$; at $t = \pi/2$. (c) Discuss the shape of the streakline that passes through the origin.

### SOLUTION

**(a)** Since $u = u_0 \sin[\omega(t - y/v_0)]$ and $v = v_0$ it follows from Eq. 4.1 that streamlines are given by the solution of

$$\frac{dy}{dx} = \frac{v}{u} = \frac{v_0}{u_0 \sin[\omega(t - y/v_0)]}$$

in which the variables can be separated and the equation integrated (for any given time $t$) to give

$$u_0 \int \sin\left[\omega\left(t - \frac{y}{v_0}\right)\right] dy = v_0 \int dx,$$

or

$$u_0(v_0/\omega)\cos\left[\omega\left(t-\frac{y}{v_0}\right)\right]=v_0x+C \qquad (1)$$

where $C$ is a constant. For the streamline at $t=0$ that passes through the origin ($x=y=0$), the value of $C$ is obtained from Eq. 1 as $C=u_0v_0/\omega$. Hence, the equation for this streamline is

$$x=\frac{u_0}{\omega}\left[\cos\left(\frac{\omega y}{v_0}\right)-1\right] \qquad (2)\ \textbf{(Ans)}$$

Similarly, for the streamline at $t=\pi/2\omega$ that passes through the origin, Eq. 1 gives $C=0$. Thus, the equation for this streamline is

$$x=\frac{u_0}{\omega}\cos\left[\omega\left(\frac{\pi}{2\omega}-\frac{y}{v_0}\right)\right]=\frac{u_0}{\omega}\cos\left(\frac{\pi}{2}-\frac{\omega y}{v_0}\right)$$

or

$$x=\frac{u_0}{\omega}\sin\left(\frac{\omega y}{v_0}\right) \qquad (3)\ \textbf{(Ans)}$$

**COMMENT** These two streamlines, plotted in Fig. E4.3*b*, are not the same because the flow is unsteady. For example, at the origin ($x=y=0$) the velocity is $\mathbf{V}=v_0\hat{\mathbf{j}}$ at $t=0$ and $\mathbf{V}=u_0\hat{\mathbf{i}}+v_0\hat{\mathbf{j}}$ at $t=\pi/2\omega$. Thus, the angle of the streamline passing through the origin changes with time. Similarly, the shape of the entire streamline is a function of time.

**(b)** The pathline of a particle (the location of the particle as a function of time) can be obtained from the velocity field and the definition of the velocity. Since $u=dx/dt$ and $v=dy/dt$ we obtain

$$\frac{dx}{dt}=u_0\sin\left[\omega\left(t-\frac{y}{v_0}\right)\right] \quad \text{and} \quad \frac{dy}{dt}=v_0$$

The $y$ equation can be integrated (since $v_0=$ constant) to give the $y$ coordinate of the pathline as

$$y=v_0t+C_1 \qquad (4)$$

where $C_1$ is a constant. With this known $y=y(t)$ dependence, the $x$ equation for the pathline becomes

$$\frac{dx}{dt}=u_0\sin\left[\omega\left(t-\frac{v_0t+C_1}{v_0}\right)\right]=-u_0\sin\left(\frac{C_1\omega}{v_0}\right)$$

This can be integrated to give the $x$ component of the pathline as

$$x=-\left[u_0\sin\left(\frac{C_1\omega}{v_0}\right)\right]t+C_2 \qquad (5)$$

where $C_2$ is a constant. For the particle that was at the origin ($x=y=0$) at time $t=0$, Eqs. 4 and 5 give $C_1=C_2=0$. Thus, the pathline is

$$x=0 \quad \text{and} \quad y=v_0t \qquad (6)\ \textbf{(Ans)}$$

Similarly, for the particle that was at the origin at $t=\pi/2\omega$, Eqs. 4 and 5 give $C_1=-\pi v_0/2\omega$ and $C_2=-\pi u_0/2\omega$. Thus, the pathline for this particle is

$$x=u_0\left(t-\frac{\pi}{2\omega}\right) \quad \text{and} \quad y=v_0\left(t-\frac{\pi}{2\omega}\right) \qquad (7)$$

The pathline can be drawn by plotting the locus of $x(t)$, $y(t)$ values for $t\geq 0$ or by eliminating the parameter $t$ from Eq. 7 to give

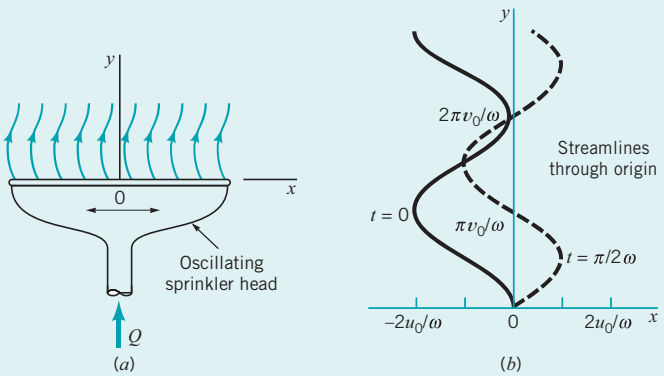
$$y=\frac{v_0}{u_0}x \qquad (8)\ \textbf{(Ans)}$$

**COMMENT** The pathlines given by Eqs. 6 and 8, shown in Fig. E4.3*c*, are straight lines from the origin (rays). The pathlines and streamlines do not coincide because the flow is unsteady.

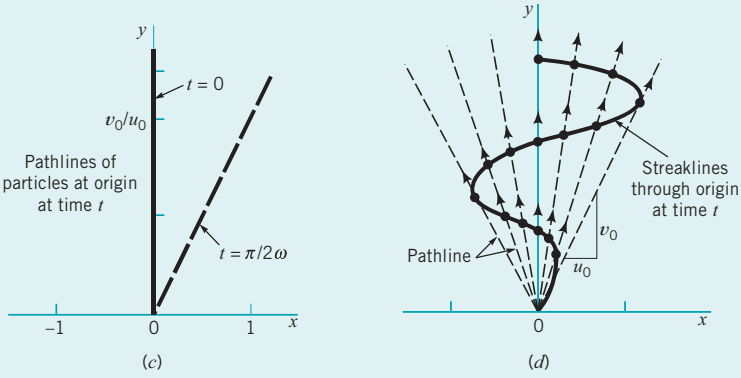
**(c)** The streakline through the origin at time $t=0$ is the locus of particles at $t=0$ that previously ($t<0$) passed through the origin. The general shape of the streaklines can be seen as follows. Each particle that flows through the origin travels in a straight line (pathlines are rays from the origin), the slope of which lies between $\pm v_0/u_0$ as shown in Fig. E4.3*d*. Particles passing through the origin at different times are located on different rays from the origin and at different distances from the origin. The net result is that a stream of dye continually injected at the origin (a streakline) would have the shape shown in Fig. E4.3*d*. Because of the unsteadiness, the streakline will vary with time, although it will always have the oscillating, sinuous character shown.

**COMMENT** Similar streaklines are given by the stream of water from a garden hose nozzle that oscillates back and forth in a direction normal to the axis of the nozzle.

In this example neither the streamlines, pathlines, nor streaklines coincide. If the flow were steady, all of these lines would be the same.




■ **FIGURE E4.3(a), (b)**

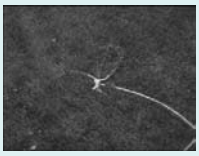



■ FIGURE E4.3(c), (d)

## 4.2 The Acceleration Field



*V4.11 Pathlines*



*Acceleration is the time rate of change of velocity for a given particle.*

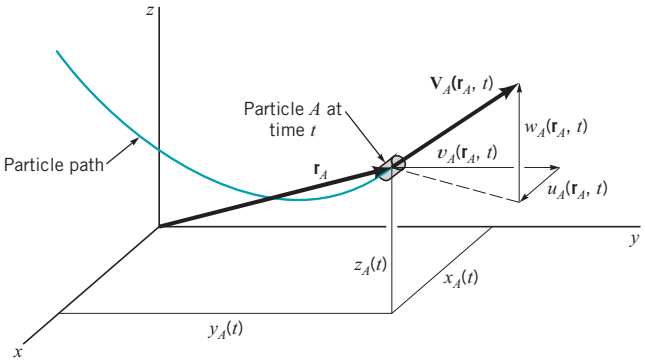
As indicated in the previous section, we can describe fluid motion by either (1) following individual particles (Lagrangian description) or (2) remaining fixed in space and observing different particles as they pass by (Eulerian description). In either case, to apply Newton's second law ($\mathbf{F} = m\mathbf{a}$) we must be able to describe the particle acceleration in an appropriate fashion. For the infrequently used Lagrangian method, we describe the fluid acceleration just as is done in solid body dynamics—$\mathbf{a} = \mathbf{a}(t)$ for each particle. For the Eulerian description we describe the ***acceleration field*** as a function of position and time without actually following any particular particle. This is analogous to describing the flow in terms of the velocity field, $\mathbf{V} = \mathbf{V}(x, y, z, t)$, rather than the velocity for particular particles. In this section we will discuss how to obtain the acceleration field if the velocity field is known.

The acceleration of a particle is the time rate of change of its velocity. For unsteady flows the velocity at a given point in space (occupied by different particles) may vary with time, giving rise to a portion of the fluid acceleration. In addition, a fluid particle may experience an acceleration because its velocity changes as it flows from one point to another in space. For example, water flowing through a garden hose nozzle under steady conditions (constant number of gallons per minute from the hose) will experience an acceleration as it changes from its relatively low velocity in the hose to its relatively high velocity at the tip of the nozzle.

### 4.2.1 The Material Derivative

Consider a fluid particle moving along its pathline as is shown in Fig. 4.4. In general, the particle's velocity, denoted $\mathbf{V}_A$ for particle $A$, is a function of its location and the time. That is,

$$\mathbf{V}_A = \mathbf{V}_A(\mathbf{r}_A, t) = \mathbf{V}_A[x_A(t), y_A(t), z_A(t), t]$$




■ FIGURE 4.4
**Velocity and position of particle *A* at time *t*.**

where $x_A = x_A(t), y_A = y_A(t)$, and $z_A = z_A(t)$ define the location of the moving particle. By definition, the acceleration of a particle is the time rate of change of its velocity. Since the velocity may be a function of both position and time, its value may change because of the change in time as well as a change in the particle's position. Thus, we use the chain rule of differentiation to obtain the acceleration of particle $A$, denoted $\mathbf{a}_A$, as

$$\mathbf{a}_A(t) = \frac{d\mathbf{V}_A}{dt} = \frac{\partial \mathbf{V}_A}{\partial t} + \frac{\partial \mathbf{V}_A}{\partial x}\frac{dx_A}{dt} + \frac{\partial \mathbf{V}_A}{\partial y}\frac{dy_A}{dt} + \frac{\partial \mathbf{V}_A}{\partial z}\frac{dz_A}{dt} \tag{4.2}$$

Using the fact that the particle velocity components are given by $u_A = dx_A/dt$, $v_A = dy_A/dt$, and $w_A = dz_A/dt$, Eq. 4.2 becomes

$$\mathbf{a}_A = \frac{\partial \mathbf{V}_A}{\partial t} + u_A\frac{\partial \mathbf{V}_A}{\partial x} + v_A\frac{\partial \mathbf{V}_A}{\partial y} + w_A\frac{\partial \mathbf{V}_A}{\partial z}$$

Since the above is valid for any particle, we can drop the reference to particle $A$ and obtain the acceleration field from the velocity field as

$$\mathbf{a} = \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z} \tag{4.3}$$

This is a vector result whose scalar components can be written as

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}$$

$$a_y = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \tag{4.4}$$

and

$$a_z = \frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}$$

where $a_x$, $a_y$, and $a_z$ are the $x$, $y$, and $z$ components of the acceleration.

The above result is often written in shorthand notation as

$$\mathbf{a} = \frac{D\mathbf{V}}{Dt}$$

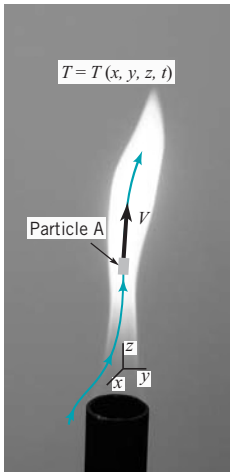
*The material derivative is used to describe time rates of change for a given particle.*

where the operator

$$\frac{D(\ )}{Dt} \equiv \frac{\partial(\ )}{\partial t} + u\frac{\partial(\ )}{\partial x} + v\frac{\partial(\ )}{\partial y} + w\frac{\partial(\ )}{\partial z} \tag{4.5}$$

is termed the ***material derivative*** or *substantial derivative*. An often-used shorthand notation for the material derivative operator is

$$\frac{D(\ )}{Dt} = \frac{\partial(\ )}{\partial t} + (\mathbf{V}\cdot\nabla)(\ ) \tag{4.6}$$




The dot product of the velocity vector, $\mathbf{V}$, and the gradient operator, $\nabla(\ ) = \partial(\ )/\partial x\,\hat{\mathbf{i}} + \partial(\ )/\partial y\,\hat{\mathbf{j}} + \partial(\ )/\partial z\,\hat{\mathbf{k}}$ (a vector operator) provides a convenient notation for the spatial derivative terms appearing in the Cartesian coordinate representation of the material derivative. Note that the notation $\mathbf{V}\cdot\nabla$ represents the operator $\mathbf{V}\cdot\nabla(\ ) = u\partial(\ )/\partial x + v\partial(\ )/\partial y + w\partial(\ )/\partial z$.

The material derivative concept is very useful in analysis involving various fluid parameters, not just the acceleration. The material derivative of any variable is the rate at which that variable changes with time for a given particle (as seen by one moving along with the fluid—the Lagrangian description). For example, consider a temperature field $T = T(x, y, z, t)$ associated with a given flow, like the flame shown in the figure in the margin. It may be of interest to determine the time rate of change of temperature of a fluid particle (particle $A$) as it moves through this temperature

CMS Cepcor EX 1046 Page 67
CMS Cepcor v. Sandvik IPR2024-01380

field. If the velocity, $\mathbf{V} = \mathbf{V}(x, y, z, t)$, is known, we can apply the chain rule to determine the rate of change of temperature as

$$\frac{dT_A}{dt} = \frac{\partial T_A}{\partial t} + \frac{\partial T_A}{\partial x}\frac{dx_A}{dt} + \frac{\partial T_A}{\partial y}\frac{dy_A}{dt} + \frac{\partial T_A}{\partial z}\frac{dz_A}{dt}$$

This can be written as

$$\frac{DT}{Dt} = \frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} = \frac{\partial T}{\partial t} + \mathbf{V}\cdot\nabla T$$

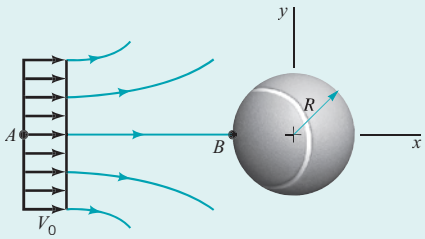
As in the determination of the acceleration, the material derivative operator, $D(\ )/Dt$, appears.

## EXAMPLE 4.4 Acceleration along a Streamline

**GIVEN** An incompressible, inviscid fluid flows steadily past a ball of radius $R$, as shown in Fig. E4.4*a*. According to a more advanced analysis of the flow, the fluid velocity along streamline $A$–$B$ is given by

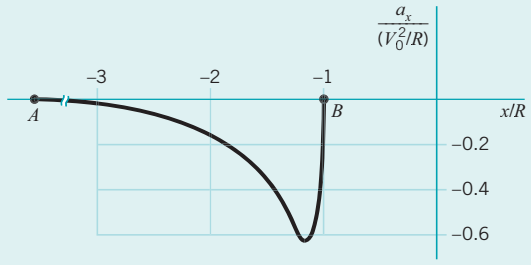
$$\mathbf{V} = u(x)\hat{\mathbf{i}} = V_0\left(1 + \frac{R^3}{x^3}\right)\hat{\mathbf{i}}$$

where $V_0$ is the upstream velocity far ahead of the sphere.




(*a*)

**FIND** Determine the acceleration experienced by fluid particles as they flow along this streamline.




(*b*)

■ FIGURE E4.4

### SOLUTION

Along streamline $A$–$B$ there is only one component of velocity ($v = w = 0$) so that from Eq. 4.3

$$\mathbf{a} = \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} = \left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x}\right)\hat{\mathbf{i}}$$

or

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x}, \qquad a_y = 0, \qquad a_z = 0$$

Since the flow is steady the velocity at a given point in space does not change with time. Thus, $\partial u/\partial t = 0$. With the given velocity distribution along the streamline, the acceleration becomes

$$a_x = u\frac{\partial u}{\partial x} = V_0\left(1 + \frac{R^3}{x^3}\right)V_0[R^3(-3x^{-4})]$$

or

$$a_x = -3(V_0^2/R)\frac{1 + (R/x)^3}{(x/R)^4} \qquad \textbf{(Ans)}$$

**COMMENTS** Along streamline $A$–$B$ ($-\infty \le x \le -R$ and $y = 0$) the acceleration has only an $x$ component and it is negative (a deceleration). Thus, the fluid slows down from its upstream velocity of $\mathbf{V} = V_0\hat{\mathbf{i}}$ at $x = -\infty$ to its stagnation point velocity of $\mathbf{V} = 0$ at $x = -R$, the "nose" of the ball. The variation of $a_x$ along streamline $A$–$B$ is shown in Fig. E4.4*b*. It is the same result as is obtained in Example 3.1 by using the streamwise component of the acceleration, $a_x = V\,\partial V/\partial s$. The maximum deceleration occurs at $x = -1.205R$ and has a value of $a_{x,max} = -0.610\,V_0^2/R$. Note that this maximum deceleration increases with increasing velocity and decreasing size. As indicated in the following table, typical values of this deceleration can be quite large. For example, the $a_{x,max} = -4.08 \times 10^4$ ft/s$^2$ value for a pitched baseball is a deceleration approximately 1500 times that of gravity.

CMS Cepcor EX 1046 Page 68
CMS Cepcor v. Sandvik IPR2024-01380

| Object | $V_0$ (ft/s) | $R$ (ft) | $a_{x,max}$ (ft/s$^2$) |
|---|---|---|---|
| Rising weather balloon | 1 | 4.0 | $-0.153$ |
| Soccer ball | 20 | 0.80 | $-305$ |
| Baseball | 90 | 0.121 | $-4.08 \times 10^4$ |
| Tennis ball | 100 | 0.104 | $-5.87 \times 10^4$ |
| Golf ball | 200 | 0.070 | $-3.49 \times 10^5$ |

In general, for fluid particles on streamlines other than *A*–*B*, all three components of the acceleration ($a_x$, $a_y$, and $a_z$) will be nonzero.

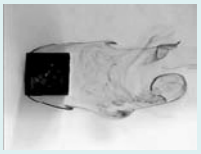
### 4.2.2 Unsteady Effects

*The local derivative is a result of the unsteadiness of the flow.*

As is seen from Eq. 4.5, the material derivative formula contains two types of terms—those involving the time derivative $[\partial(\ )/\partial t]$ and those involving spatial derivatives $[\partial(\ )/\partial x$, $\partial(\ )/\partial y$, and $\partial(\ )/\partial z]$. The time derivative portions are denoted as the *local derivative*. They represent effects of the unsteadiness of the flow. If the parameter involved is the acceleration, that portion given by $\partial \mathbf{V}/\partial t$ is termed the ***local acceleration.*** For steady flow the time derivative is zero throughout the flow field $[\partial(\ )/\partial t \equiv 0]$, and the local effect vanishes. Physically, there is no change in flow parameters at a fixed point in space if the flow is steady. There may be a change of those parameters for a fluid particle as it moves about, however.



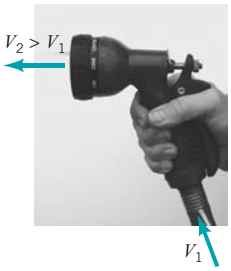
*V4.12 Unsteady flow*



If a flow is unsteady, its parameter values (velocity, temperature, density, etc.) at any location may change with time. For example, an unstirred ($\mathbf{V} = 0$) cup of coffee will cool down in time because of heat transfer to its surroundings. That is, $DT/Dt = \partial T/\partial t + \mathbf{V} \cdot \nabla T = \partial T/\partial t < 0$. Similarly, a fluid particle may have nonzero acceleration as a result of the unsteady effect of the flow. Consider flow in a constant diameter pipe as is shown in Fig. 4.5. The flow is assumed to be spatially uniform throughout the pipe. That is, $\mathbf{V} = V_0(t)\,\hat{\mathbf{i}}$ at all points in the pipe. The value of the acceleration depends on whether $V_0$ is being increased, $\partial V_0/\partial t > 0$, or decreased, $\partial V_0/\partial t < 0$. Unless $V_0$ is independent of time ($V_0 \equiv$ constant) there will be an acceleration, the local acceleration term. Thus, the acceleration field, $\mathbf{a} = \partial V_0/\partial t\,\hat{\mathbf{i}}$, is uniform throughout the entire flow, although it may vary with time ($\partial V_0/\partial t$ need not be constant). The acceleration due to the spatial variations of velocity ($u\,\partial u/\partial x$, $v\,\partial v/\partial y$, etc.) vanishes automatically for this flow, since $\partial u/\partial x = 0$ and $v = w = 0$. That is,

$$\mathbf{a} = \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z} = \frac{\partial \mathbf{V}}{\partial t} = \frac{\partial V_0}{\partial t}\hat{\mathbf{i}}$$

### 4.2.3 Convective Effects




The portion of the material derivative (Eq. 4.5) represented by the spatial derivatives is termed the *convective derivative*. It represents the fact that a flow property associated with a fluid particle may vary because of the motion of the particle from one point in space where the parameter has one value to another point in space where its value is different. For example, the water velocity at the inlet of the garden hose nozzle shown in the figure in the margin is different (both in direction and speed) than it is at the exit. This contribution to the time rate of change of the parameter for the particle can occur whether the flow is steady or unsteady.

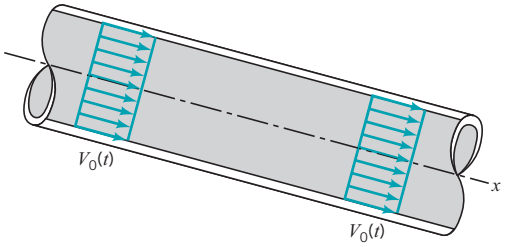
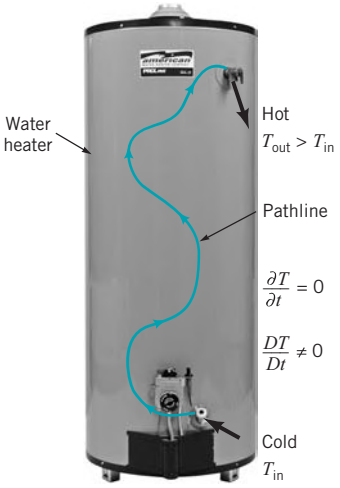



**FIGURE 4.5** Uniform, unsteady flow in a constant diameter pipe.

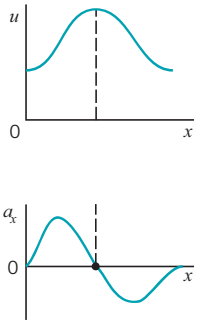



■ **FIGURE 4.6** **Steady-state operation of a water heater. (Photo courtesy of American Water Heater Company.)**

*The convective derivative is a result of the spatial variation of the flow.*

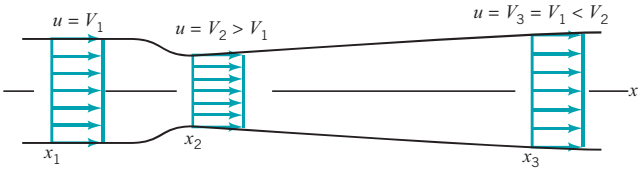
It is due to the convection, or motion, of the particle through space in which there is a gradient $[\nabla(\ ) = \partial(\ )/\partial x\,\hat{\mathbf{i}} + \partial(\ )/\partial y\,\hat{\mathbf{j}} + \partial(\ )/\partial z\,\hat{\mathbf{k}}]$ in the parameter value. That portion of the acceleration given by the term $(\mathbf{V}\cdot\nabla)\mathbf{V}$ is termed the ***convective acceleration.***

As is illustrated in Fig. 4.6, the temperature of a water particle changes as it flows through a water heater. The water entering the heater is always the same cold temperature and the water leaving the heater is always the same hot temperature. The flow is steady. However, the temperature, $T$, of each water particle increases as it passes through the heater—$T_{out} > T_{in}$. Thus, $DT/Dt \neq 0$ because of the convective term in the total derivative of the temperature. That is, $\partial T/\partial t = 0$, but $u\,\partial T/\partial x \neq 0$ (where $x$ is directed along the streamline), since there is a nonzero temperature gradient along the streamline. A fluid particle traveling along this nonconstant temperature path ($\partial T/\partial x \neq 0$) at a specified speed ($u$) will have its temperature change with time at a rate of $DT/Dt = u\,\partial T/\partial x$ even though the flow is steady ($\partial T/\partial t = 0$).




The same types of processes are involved with fluid accelerations. Consider flow in a variable area pipe as shown in Fig. 4.7. It is assumed that the flow is steady and one-dimensional with velocity that increases and decreases in the flow direction as indicated. As the fluid flows from section (1) to section (2), its velocity increases from $V_1$ to $V_2$. Thus, even though $\partial\mathbf{V}/\partial t = 0$ (steady flow), fluid particles experience an acceleration given by $a_x = u\,\partial u/\partial x$ (convective acceleration). For $x_1 < x < x_2$, it is seen that $\partial u/\partial x > 0$ so that $a_x > 0$—the fluid accelerates. For $x_2 < x < x_3$, it is seen that $\partial u/\partial x < 0$ so that $a_x < 0$—the fluid decelerates. This acceleration and deceleration are shown in the figure in the margin. If $V_1 = V_3$, the amount of acceleration precisely balances the amount of deceleration even though the distances between $x_2$ and $x_1$ and $x_3$ and $x_2$ are not the same.

The concept of the material derivative can be used to determine the time rate of change of any parameter associated with a particle as it moves about. Its use is not restricted to fluid mechanics alone. The basic ingredients needed to use the material derivative concept are the field description of the parameter, $P = P(x, y, z, t)$, and the rate at which the particle moves through that field, $\mathbf{V} = \mathbf{V}(x, y, z, t)$.




■ **FIGURE 4.7** **Uniform, steady flow in a variable area pipe.**

## EXAMPLE 4.5 Acceleration from a Given Velocity Field

**GIVEN** Consider the steady, two-dimensional flow field discussed in Example 4.2.

**FIND** Determine the acceleration field for this flow.

### SOLUTION

In general, the acceleration is given by

$$\mathbf{a} = \frac{D\mathbf{V}}{Dt} = \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V}\cdot\nabla)(\mathbf{V})$$
$$= \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z} \qquad (1)$$

where the velocity is given by $\mathbf{V} = (V_0/\ell)(-x\hat{\mathbf{i}} + y\hat{\mathbf{j}})$ so that $u = -(V_0/\ell)x$ and $v = (V_0/\ell)y$. For steady $[\partial(\ )/\partial t = 0]$, two-dimensional $[w = 0$ and $\partial(\ )/\partial z = 0]$ flow, Eq. 1 becomes

$$\mathbf{a} = u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y}$$
$$= \left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right)\hat{\mathbf{i}} + \left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right)\hat{\mathbf{j}}$$

Hence, for this flow the acceleration is given by

$$\mathbf{a} = \left[\left(-\frac{V_0}{\ell}\right)(x)\left(-\frac{V_0}{\ell}\right) + \left(\frac{V_0}{\ell}\right)(y)(0)\right]\hat{\mathbf{i}}$$
$$+ \left[\left(-\frac{V_0}{\ell}\right)(x)(0) + \left(\frac{V_0}{\ell}\right)(y)\left(\frac{V_0}{\ell}\right)\right]\hat{\mathbf{j}}$$

or

$$a_x = \frac{V_0^2 x}{\ell^2}, \qquad a_y = \frac{V_0^2 y}{\ell^2} \qquad \textbf{(Ans)}$$

**COMMENTS** The fluid experiences an acceleration in both the $x$ and $y$ directions. Since the flow is steady, there is no local acceleration—the fluid velocity at any given point is constant in time. However, there is a convective acceleration due to the change in velocity from one point on the particle's pathline to another. Recall that the velocity is a vector—it has both a magnitude and a direction. In this flow both the fluid speed (magnitude) and flow direction change with location (see Fig. E4.1*a*).

For this flow the magnitude of the acceleration is constant on circles centered at the origin, as is seen from the fact that

$$|\mathbf{a}| = (a_x^2 + a_y^2 + a_z^2)^{1/2} = \left(\frac{V_0}{\ell}\right)^2 (x^2 + y^2)^{1/2} \qquad (2)$$

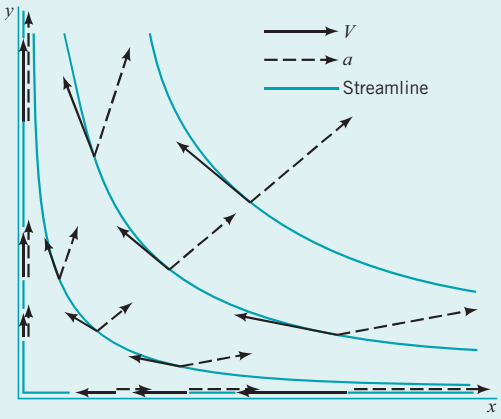



FIGURE E4.5

Also, the acceleration vector is oriented at an angle $\theta$ from the $x$ axis, where

$$\tan\theta = \frac{a_y}{a_x} = \frac{y}{x}$$

This is the same angle as that formed by a ray from the origin to point $(x, y)$. Thus, the acceleration is directed along rays from the origin and has a magnitude proportional to the distance from the origin. Typical acceleration vectors (from Eq. 2) and velocity vectors (from Example 4.1) are shown in Fig. E4.5 for the flow in the first quadrant. Note that **a** and **V** are not parallel except along the $x$ and $y$ axes (a fact that is responsible for the curved pathlines of the flow), and that both the acceleration and velocity are zero at the origin $(x = y = 0)$. An infinitesimal fluid particle placed precisely at the origin will remain there, but its neighbors (no matter how close they are to the origin) will drift away.

## EXAMPLE 4.6 The Material Derivative

**GIVEN** A fluid flows steadily through a two-dimensional nozzle of length $\ell$ as shown in Fig. E4.6*a*. The nozzle shape is given by

$$y/\ell = \pm 0.5/[1 + (x/\ell)]$$

If viscous and gravitational effects are negligible, the velocity field is approximately

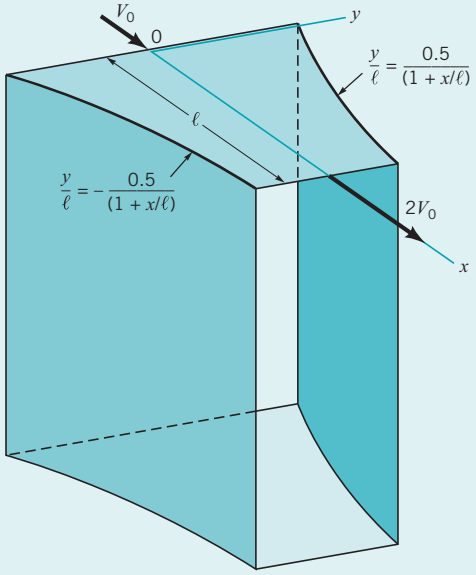
$$u = V_0[1 + x/\ell], \quad v = -V_0 y/\ell \qquad (1)$$

and the pressure field is

$$p - p_0 = -(\rho V_0^2/2)[(x^2 + y^2)/\ell^2 + 2x/\ell]$$

where $V_0$ and $p_0$ are the velocity and pressure at the origin, $x = y = 0$. Note that the fluid speed increases as it flows through the nozzle. For example, along the center line ($y = 0$), $V = V_0$ at $x = 0$ and $V = 2V_0$ at $x = \ell$.

**FIND** Determine, as a function of $x$ and $y$, the time rate of change of pressure felt by a fluid particle as it flows through the nozzle.




■ FIGURE E4.6a

## SOLUTION

The time rate of change of pressure at any given, fixed point in this steady flow is zero. However, the time rate of change of pressure felt by a particle flowing through the nozzle is given by the material derivative of the pressure and is not zero. Thus,

$$\frac{Dp}{Dt} = \frac{\partial p}{\partial t} + u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} = u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} \qquad (2)$$

where the $x$- and $y$-components of the pressure gradient can be written as

$$\frac{\partial p}{\partial x} = -\frac{\rho V_0^2}{\ell}\left(\frac{x}{\ell} + 1\right) \qquad (3)$$

and

$$\frac{\partial p}{\partial y} = -\frac{\rho V_0^2}{\ell}\left(\frac{y}{\ell}\right) \qquad (4)$$

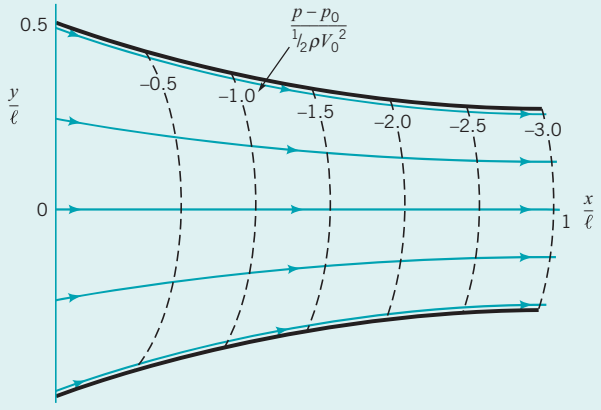
Therefore, by combining Eqs. (1), (2), (3), and (4) we obtain

$$\frac{Dp}{Dt} = V_0\left(1 + \frac{x}{\ell}\right)\left(-\frac{\rho V_0^2}{\ell}\right)\left(\frac{x}{\ell} + 1\right) + \left(-V_0\frac{y}{\ell}\right)\left(-\frac{\rho V_0^2}{\ell}\right)\left(\frac{y}{\ell}\right)$$

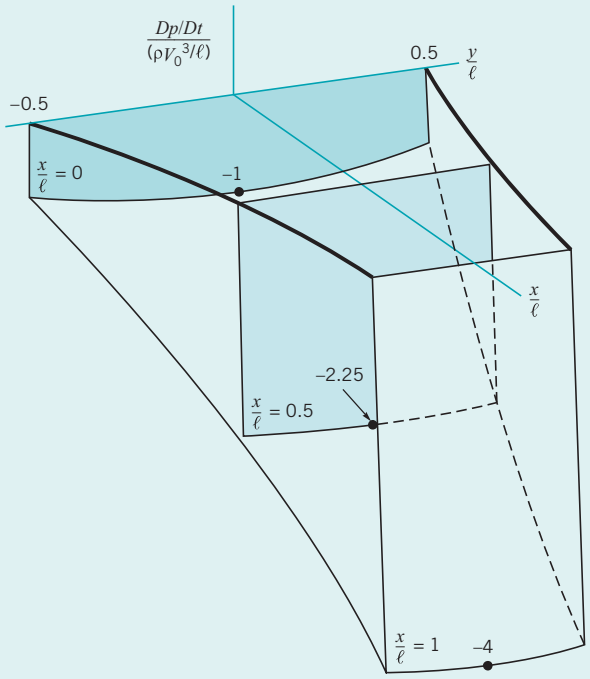
or

$$\frac{Dp}{Dt} = -\frac{\rho V_0^3}{\ell}\left[\left(\frac{x}{\ell} + 1\right)^2 - \left(\frac{y}{\ell}\right)^2\right] \qquad (5) \quad \textbf{(Ans)}$$

**COMMENT** Lines of constant pressure within the nozzle are indicated in Fig. E4.6*b*, along with some representative streamlines of the flow. Note that as a fluid particle flows along its streamline, it moves into areas of lower and lower pressure. Hence, even though the flow is steady, the time rate of change of the pressure for any given particle is negative. This can be verified from Eq. (5) which, when plotted in Fig. E4.6*c*, shows that for any point within the nozzle $Dp/Dt < 0$.




■ FIGURE E4.6b




■ FIGURE E4.6c

### 4.2.4 Streamline Coordinates



*V4.13 Streamline coordinates*



In many flow situations it is convenient to use a coordinate system defined in terms of the streamlines of the flow. An example for steady, two-dimensional flows is illustrated in Fig. 4.8. Such flows can be described either in terms of the usual $x$, $y$ Cartesian coordinate system (or some other system such as the $r$, $\theta$ polar coordinate system) or the streamline coordinate system. In the streamline coordinate system the flow is described in terms of one coordinate along the streamlines, denoted $s$, and the second coordinate normal to the streamlines, denoted $n$. Unit vectors in these two directions are denoted by $\hat{\mathbf{s}}$ and $\hat{\mathbf{n}}$, as shown in the figure. Care is needed not to confuse the coordinate distance $s$ (a scalar) with the unit vector along the streamline direction, $\hat{\mathbf{s}}$.

The flow plane is therefore covered by an orthogonal curved net of coordinate lines. At any point the $s$ and $n$ directions are perpendicular, but the lines of constant $s$ or constant $n$ are not necessarily straight. Without knowing the actual velocity field (hence, the streamlines) it is not possible to construct this flow net. In many situations appropriate simplifying assumptions can be made so that this lack of information does not present an insurmountable difficulty. One of the major advantages of using the streamline coordinate system is that the velocity is always tangent to the $s$ direction. That is,

$$\mathbf{V} = V\hat{\mathbf{s}}$$

This allows simplifications in describing the fluid particle acceleration and in solving the equations governing the flow.

For steady, two-dimensional flow we can determine the acceleration as

$$\mathbf{a} = \frac{D\mathbf{V}}{Dt} = a_s\hat{\mathbf{s}} + a_n\hat{\mathbf{n}}$$

where $a_s$ and $a_n$ are the streamline and normal components of acceleration, respectively, as indicated by the figure in the margin. We use the material derivative because by definition the acceleration is the time rate of change of the velocity of a given particle as it moves about. If the streamlines

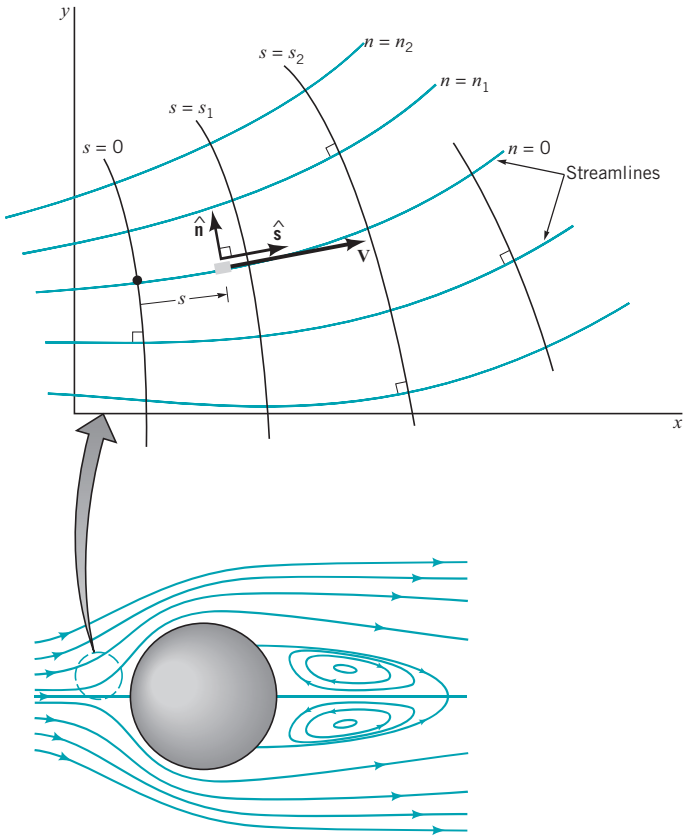



**FIGURE 4.8**
**Streamline coordinate system for two-dimensional flow.**

CMS Cepcor EX 1046 Page 73
CMS Cepcor v. Sandvik IPR2024-01380

are curved, both the speed of the particle and its direction of flow may change from one point to another. In general, for steady flow both the speed and the flow direction are a function of location—$V = V(s, n)$ and $\hat{\mathbf{s}} = \hat{\mathbf{s}}(s, n)$. For a given particle, the value of $s$ changes with time, but the value of $n$ remains fixed because the particle flows along a streamline defined by $n$ = constant. (Recall that streamlines and pathlines coincide in steady flow.) Thus, application of the chain rule gives

$$\mathbf{a} = \frac{D(V\hat{\mathbf{s}})}{Dt} = \frac{DV}{Dt}\hat{\mathbf{s}} + V\frac{D\hat{\mathbf{s}}}{Dt}$$

or

$$\mathbf{a} = \left(\frac{\partial V}{\partial t} + \frac{\partial V}{\partial s}\frac{ds}{dt} + \frac{\partial V}{\partial n}\frac{dn}{dt}\right)\hat{\mathbf{s}} + V\left(\frac{\partial \hat{\mathbf{s}}}{\partial t} + \frac{\partial \hat{\mathbf{s}}}{\partial s}\frac{ds}{dt} + \frac{\partial \hat{\mathbf{s}}}{\partial n}\frac{dn}{dt}\right)$$

This can be simplified by using the fact that for steady flow nothing changes with time at a given point so that both $\partial V/\partial t$ and $\partial\hat{\mathbf{s}}/\partial t$ are zero. Also, the velocity along the streamline is $V = ds/dt$ and the particle remains on its streamline ($n$ = constant) so that $dn/dt = 0$. Hence,

$$\mathbf{a} = \left(V\frac{\partial V}{\partial s}\right)\hat{\mathbf{s}} + V\left(V\frac{\partial \hat{\mathbf{s}}}{\partial s}\right)$$

*The orientation of the unit vector along the streamline changes with distance along the streamline.*

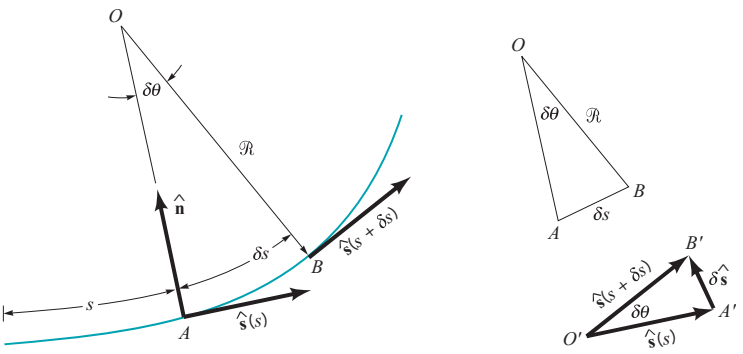
The quantity $\partial\hat{\mathbf{s}}/\partial s$ represents the limit as $\delta s \to 0$ of the change in the unit vector along the streamline, $\delta\hat{\mathbf{s}}$, per change in distance along the streamline, $\delta s$. The magnitude of $\hat{\mathbf{s}}$ is constant ($|\hat{\mathbf{s}}| = 1$; it is a unit vector), but its direction is variable if the streamlines are curved. From Fig. 4.9 it is seen that the magnitude of $\partial\hat{\mathbf{s}}/\partial s$ is equal to the inverse of the radius of curvature of the streamline, $\mathcal{R}$, at the point in question. This follows because the two triangles shown ($AOB$ and $A'O'B'$) are similar triangles so that $\delta s/\mathcal{R} = |\delta\hat{\mathbf{s}}|/|\hat{\mathbf{s}}| = |\delta\hat{\mathbf{s}}|$, or $|\delta\hat{\mathbf{s}}/\delta s| = 1/\mathcal{R}$. Similarly, in the limit $\delta s \to 0$, the direction of $\delta\hat{\mathbf{s}}/\delta s$ is seen to be normal to the streamline. That is,

$$\frac{\partial \hat{\mathbf{s}}}{\partial s} = \lim_{\delta s \to 0}\frac{\delta\hat{\mathbf{s}}}{\delta s} = \frac{\hat{\mathbf{n}}}{\mathcal{R}}$$

Hence, the acceleration for steady, two-dimensional flow can be written in terms of its streamwise and normal components in the form

$$\mathbf{a} = V\frac{\partial V}{\partial s}\hat{\mathbf{s}} + \frac{V^2}{\mathcal{R}}\hat{\mathbf{n}} \quad \text{or} \quad a_s = V\frac{\partial V}{\partial s}, \qquad a_n = \frac{V^2}{\mathcal{R}} \tag{4.7}$$

The first term, $a_s = V\,\partial V/\partial s$, represents the convective acceleration along the streamline and the second term, $a_n = V^2/\mathcal{R}$, represents centrifugal acceleration (one type of convective acceleration) normal to the fluid motion. These components can be noted in Fig. E4.5 by resolving the acceleration vector into its components along and normal to the velocity vector. Note that the unit vector $\hat{\mathbf{n}}$ is directed from the streamline toward the center of curvature. These forms of the acceleration were used in Chapter 3 and are probably familiar from previous dynamics or physics considerations.




**■ FIGURE 4.9** **Relationship between the unit vector along the streamline, $\hat{\mathbf{s}}$, and the radius of curvature of the streamline, $\mathcal{R}$.**

## 4.3 Control Volume and System Representations

As is discussed in Chapter 1, a fluid is a type of matter that is relatively free to move and interact with its surroundings. As with any matter, a fluid's behavior is governed by fundamental physical laws which are approximated by an appropriate set of equations. The application of laws such as the conservation of mass, Newton's laws of motion, and the laws of thermodynamics form the foundation of fluid mechanics analyses. There are various ways that these governing laws can be applied to a fluid, including the system approach and the control volume approach. By definition, a ***system*** is a collection of matter of fixed identity (always the same atoms or fluid particles), which may move, flow, and interact with its surroundings. A ***control volume,*** on the other hand, is a volume in space (a geometric entity, independent of mass) through which fluid may flow.

*Both control volume and system concepts can be used to describe fluid flow.*

A system is a specific, identifiable quantity of matter. It may consist of a relatively large amount of mass (such as all of the air in the earth's atmosphere), or it may be an infinitesimal size (such as a single fluid particle). In any case, the molecules making up the system are "tagged" in some fashion (dyed red, either actually or only in your mind) so that they can be continually identified as they move about. The system may interact with its surroundings by various means (by the transfer of heat or the exertion of a pressure force, for example). It may continually change size and shape, but it always contains the same mass.

A mass of air drawn into an air compressor can be considered as a system. It changes shape and size (it is compressed), its temperature may change, and it is eventually expelled through the outlet of the compressor. The matter associated with the original air drawn into the compressor remains as a system, however. The behavior of this material could be investigated by applying the appropriate governing equations to this system.

One of the important concepts used in the study of statics and dynamics is that of the free-body diagram. That is, we identify an object, isolate it from its surroundings, replace its surroundings by the equivalent actions that they put on the object, and apply Newton's laws of motion. The body in such cases is our system—an identified portion of matter that we follow during its interactions with its surroundings. In fluid mechanics, it is often quite difficult to identify and keep track of a specific quantity of matter. A finite portion of a fluid contains an uncountable number of fluid particles that move about quite freely, unlike a solid that may deform but usually remains relatively easy to identify. For example, we cannot as easily follow a specific portion of water flowing in a river as we can follow a branch floating on its surface.



**(Photograph courtesy of NASA.)**

We may often be more interested in determining the forces put on a fan, airplane, or automobile by air flowing past the object than we are in the information obtained by following a given portion of the air (a system) as it flows along. Similarly, for the Space Shuttle launch vehicle shown in the margin, we may be more interested in determining the thrust produced than we are in the information obtained by following the highly complex, irregular path of the exhaust plume from the rocket engine nozzle. For these situations we often use the control volume approach. We identify a specific volume in space (a volume associated with the fan, airplane, or automobile, for example) and analyze the fluid flow within, through, or around that volume. In general, the control volume can be a moving volume, although for most situations considered in this book we will use only fixed, nondeformable control volumes. The matter within a control volume may change with time as the fluid flows through it. Similarly, the amount of mass within the volume may change with time. The control volume itself is a specific geometric entity, independent of the flowing fluid.

Examples of control volumes and *control surfaces* (the surface of the control volume) are shown in Fig. 4.10. For case (*a*), fluid flows through a pipe. The fixed control surface consists of the inside surface of the pipe, the outlet end at section (2), and a section across the pipe at (1). One portion of the control surface is a physical surface (the pipe), while the remainder is simply a surface in space (across the pipe). Fluid flows across part of the control surface, but not across all of it.

Another control volume is the rectangular volume surrounding the jet engine shown in Fig. 4.10*b*. If the airplane to which the engine is attached is sitting still on the runway, air flows through this control volume because of the action of the engine within it. The air that was within the engine itself at time $t = t_1$ (a system) has passed through the engine and is outside of the control volume at a later time $t = t_2$ as indicated. At this later time other air (a different system) is within the engine. If the airplane is moving, the control volume is fixed relative to an observer on the airplane, but it

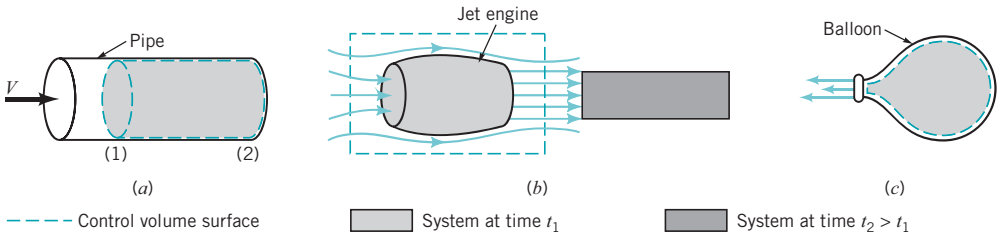



**FIGURE 4.10** **Typical control volumes: (*a*) fixed control volume, (*b*) fixed or moving control volume, (*c*) deforming control volume.**

is a moving control volume relative to an observer on the ground. In either situation air flows through and around the engine as indicated.

The deflating balloon shown in Fig. 4.10*c* provides an example of a deforming control volume. As time increases, the control volume (whose surface is the inner surface of the balloon) decreases in size. If we do not hold onto the balloon, it becomes a moving, deforming control volume as it darts about the room. The majority of the problems we will analyze can be solved by using a fixed, nondeforming control volume. In some instances, however, it will be advantageous, in fact necessary, to use a moving, deforming control volume.

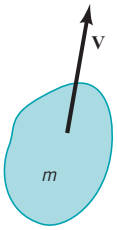
In many ways the relationship between a system and a control volume is similar to the relationship between the Lagrangian and Eulerian flow description introduced in Section 4.1.1. In the system or Lagrangian description, we follow the fluid and observe its behavior as it moves about. In the control volume or Eulerian description we remain stationary and observe the fluid's behavior at a fixed location. (If a moving control volume is used, it virtually never moves with the system—the system flows through the control volume.) These ideas are discussed in more detail in the next section.

*The governing laws of fluid motion are stated in terms of fluid systems, not control volumes.*

All of the laws governing the motion of a fluid are stated in their basic form in terms of a system approach. For example, "the mass of a system remains constant," or "the time rate of change of momentum of a system is equal to the sum of all the forces acting on the system." Note the word system, not control volume, in these statements. To use the governing equations in a control volume approach to problem solving, we must rephrase the laws in an appropriate manner. To this end we introduce the Reynolds transport theorem in the following section.

## 4.4 The Reynolds Transport Theorem

We are sometimes interested in what happens to a particular part of the fluid as it moves about. Other times we may be interested in what effect the fluid has on a particular object or volume in space as fluid interacts with it. Thus, we need to describe the laws governing fluid motion using both system concepts (consider a given mass of the fluid) and control volume concepts (consider a given volume). To do this we need an analytical tool to shift from one representation to the other. The ***Reynolds transport theorem*** provides this tool.




All physical laws are stated in terms of various physical parameters. Velocity, acceleration, mass, temperature, and momentum are but a few of the more common parameters. Let $B$ represent any of these (or other) fluid parameters and $b$ represent the amount of that parameter per unit mass. That is,

$$B = mb$$

where $m$ is the mass of the portion of fluid of interest. For example, as shown by the figure in the margin, if $B = m$, the mass, it follows that $b = 1$. The mass per unit mass is unity. If $B = mV^2/2$, the kinetic energy of the mass, then $b = V^2/2$, the kinetic energy per unit mass. The parameters $B$ and $b$ may be scalars or vectors. Thus, if $\mathbf{B} = m\mathbf{V}$, the momentum of the mass, then $\mathbf{b} = \mathbf{V}$. (The momentum per unit mass is the velocity.)

| $B$ | $b = B/m$ |
|---|---|
| $m$ | 1 |
| $m\mathbf{V}$ | $\mathbf{V}$ |
| $\frac{1}{2}mV^2$ | $\frac{1}{2}V^2$ |

The parameter $B$ is termed an *extensive property* and the parameter $b$ is termed an *intensive property*. The value of $B$ is directly proportional to the amount of the mass being considered, whereas the value of $b$ is independent of the amount of mass. The amount of an extensive property that a system possesses at a given instant, $B_{sys}$, can be determined by adding up the amount associated with each fluid particle in the system. For infinitesimal fluid particles of size $\delta\forall$ and mass $\rho\,\delta\forall$,

this summation (in the limit of $\delta \forall \to 0$) takes the form of an integration over all the particles in the system and can be written as

$$B_{sys} = \lim_{\delta \forall \to 0} \sum_i b_i (\rho_i \, \delta \forall_i) = \int_{sys} \rho b \, d\forall$$

The limits of integration cover the entire system—a (usually) moving volume. We have used the fact that the amount of $B$ in a fluid particle of mass $\rho \, \delta \forall$ is given in terms of $b$ by $\delta B = b\rho \, \delta \forall$.

Most of the laws governing fluid motion involve the time rate of change of an extensive property of a fluid system—the rate at which the momentum of a system changes with time, the rate at which the mass of a system changes with time, and so on. Thus, we often encounter terms such as

$$\frac{dB_{sys}}{dt} = \frac{d\left( \int_{sys} \rho b \, d\forall \right)}{dt} \tag{4.8}$$

To formulate the laws into a control volume approach, we must obtain an expression for the time rate of change of an extensive property within a control volume, $B_{cv}$, not within a system. This can be written as

$$\frac{dB_{cv}}{dt} = \frac{d\left( \int_{cv} \rho b \, d\forall \right)}{dt} \tag{4.9}$$

*Differences between control volume and system concepts are subtle but very important.*

where the limits of integration, denoted by cv, cover the control volume of interest. Although Eqs. 4.8 and 4.9 may look very similar, the physical interpretation of each is quite different. Mathematically, the difference is represented by the difference in the limits of integration. Recall that the control volume is a volume in space (in most cases stationary, although if it moves it need not move with the system). On the other hand, the system is an identifiable collection of mass that moves with the fluid (indeed it is a specified portion of the fluid). We will learn that even for those instances when the control volume and the system momentarily occupy the same volume in space, the two quantities $dB_{sys}/dt$ and $dB_{cv}/dt$ need not be the same. The Reynolds transport theorem provides the relationship between the time rate of change of an extensive property for a system and that for a control volume—the relationship between Eqs. 4.8 and 4.9.

## EXAMPLE 4.7 Time Rate of Change for a System and a Control Volume

**GIVEN** Fluid flows from the fire extinguisher tank shown in Fig. E4.7*a*.

**FIND** Discuss the differences between $dB_{sys}/dt$ and $dB_{cv}/dt$ if $B$ represents mass.

### SOLUTION

With $B = m$, the system mass, it follows that $b = 1$ and Eqs. 4.8 and 4.9 can be written as

$$\frac{dB_{sys}}{dt} \equiv \frac{dm_{sys}}{dt} = \frac{d\left( \int_{sys} \rho \, d\forall \right)}{dt}$$

and

$$\frac{dB_{cv}}{dt} \equiv \frac{dm_{cv}}{dt} = \frac{d\left( \int_{cv} \rho \, d\forall \right)}{dt}$$




(*a*)

■ FIGURE E4.7

Physically these represent the time rate of change of mass within the system and the time rate of change of mass within the control volume, respectively. We choose our system to be the fluid within the tank at the time the valve was opened ($t = 0$) and the control volume to be the tank itself as shown in Fig. E4.7*b*. A short time after the valve is opened, part of the system has moved outside of the control volume as is shown in Fig. E4.7*c*. The control volume remains fixed. The limits of integration are fixed for the control volume; they are a function of time for the system.

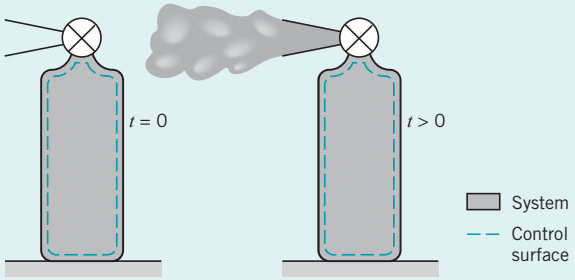
Clearly, if mass is to be conserved (one of the basic laws governing fluid motion), the mass of the fluid in the system is constant, so that

$$\frac{d\left(\int_{\text{sys}} \rho \, d\forall\right)}{dt} = 0$$

On the other hand, it is equally clear that some of the fluid has left the control volume through the nozzle on the tank. Hence, the amount of mass within the tank (the control volume) decreases with time, or

$$\frac{d\left(\int_{\text{cv}} \rho \, d\forall\right)}{dt} < 0$$

The actual numerical value of the rate at which the mass in the control volume decreases will depend on the rate at which the fluid flows through the nozzle (i.e., the size of the nozzle and the speed and density of the fluid). Clearly the meanings of $dB_{\text{sys}}/dt$ and $dB_{\text{cv}}/dt$ are different. For this example, $dB_{\text{cv}}/dt < dB_{\text{sys}}/dt$. Other situations may have $dB_{\text{cv}}/dt \geq dB_{\text{sys}}/dt$.



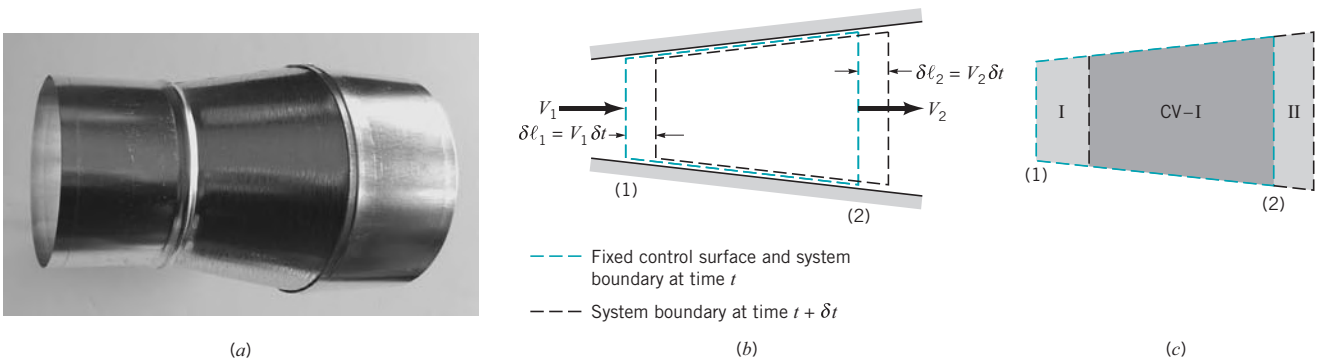

■ FIGURE E4.7

### 4.4.1 Derivation of the Reynolds Transport Theorem

*The moving system flows through the fixed control volume.*

A simple version of the Reynolds transport theorem relating system concepts to control volume concepts can be obtained easily for the one-dimensional flow through a fixed control volume such as the variable area duct section shown in Fig. 4.11*a*. We consider the control volume to be that stationary volume within the duct between sections (1) and (2) as indicated in Fig. 4.11*b*. The system that we consider is that fluid occupying the control volume at some initial time $t$. A short time later, at time $t + \delta t$, the system has moved slightly to the right. The fluid particles that coincided with section (2) of the control surface at time $t$ have moved a distance $\delta\ell_2 = V_2\,\delta t$ to the right, where $V_2$ is the velocity of the fluid as it passes section (2). Similarly, the fluid initially at section (1) has moved a distance $\delta\ell_1 = V_1\,\delta t$, where $V_1$ is the fluid velocity at section (1). We assume the fluid flows across sections (1) and (2) in a direction normal to these surfaces and that $V_1$ and $V_2$ are constant across sections (1) and (2).

As is shown in Fig. 4.11*c*, the outflow from the control volume from time $t$ to $t + \delta t$ is denoted as volume II, the inflow as volume I, and the control volume itself as CV. Thus, the system at time $t$ consists of the fluid in section CV; that is, "SYS = CV" at time $t$. At time $t + \delta t$ the system consists of the same fluid that now occupies sections (CV − I) + II. That is, "SYS = CV − I + II" at time $t + \delta t$. The control volume remains as section CV for all time.




■ FIGURE 4.11 **Control volume and system for flow through a variable area pipe.**

If $B$ is an extensive parameter of the system, then the value of it for the system at time $t$ is

$$B_{sys}(t) = B_{cv}(t)$$

since the system and the fluid within the control volume coincide at this time. Its value at time $t + \delta t$ is

$$B_{sys}(t + \delta t) = B_{cv}(t + \delta t) - B_I(t + \delta t) + B_{II}(t + \delta t)$$

Thus, the change in the amount of $B$ in the system in the time interval $\delta t$ divided by this time interval is given by

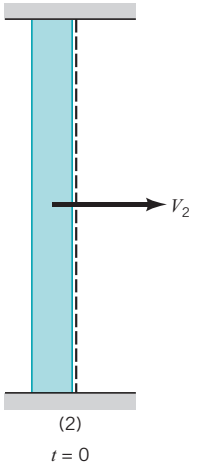
$$\frac{\delta B_{sys}}{\delta t} = \frac{B_{sys}(t + \delta t) - B_{sys}(t)}{\delta t} = \frac{B_{cv}(t + \delta t) - B_I(t + \delta t) + B_{II}(t + \delta t) - B_{sys}(t)}{\delta t}$$

By using the fact that at the initial time $t$ we have $B_{sys}(t) = B_{cv}(t)$, this ungainly expression may be rearranged as follows.

*The time rate of change of a system property is a Lagrangian concept.*

$$\frac{\delta B_{sys}}{\delta t} = \frac{B_{cv}(t + \delta t) - B_{cv}(t)}{\delta t} - \frac{B_I(t + \delta t)}{\delta t} + \frac{B_{II}(t + \delta t)}{\delta t} \tag{4.10}$$

In the limit $\delta t \to 0$, the left-hand side of Eq. 4.10 is equal to the time rate of change of $B$ for the system and is denoted as $DB_{sys}/Dt$. We use the material derivative notation, $D(\ )/Dt$, to denote this time rate of change to emphasize the Lagrangian character of this term. (Recall from Section 4.2.1 that the material derivative, $DP/Dt$, of any quantity $P$ represents the time rate of change of that quantity associated with a given fluid particle as it moves along.) Similarly, the quantity $DB_{sys}/Dt$ represents the time rate of change of property $B$ associated with a system (a given portion of fluid) as it moves along.

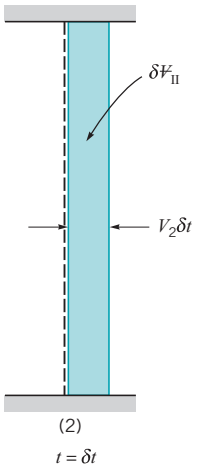



In the limit $\delta t \to 0$, the first term on the right-hand side of Eq. 4.10 is seen to be the time rate of change of the amount of $B$ within the control volume

$$\lim_{\delta t \to 0} \frac{B_{cv}(t + \delta t) - B_{cv}(t)}{\delta t} = \frac{\partial B_{cv}}{\partial t} = \frac{\partial \left( \int_{cv} \rho b \, d\forall \right)}{\partial t} \tag{4.11}$$

The third term on the right-hand side of Eq. 4.10 represents the rate at which the extensive parameter $B$ flows from the control volume, across the control surface. As indicated by the figure in the margin, during the time interval from $t = 0$ to $t = \delta t$ the volume of fluid that flows across section (2) is given by $\delta\forall_{II} = A_2 \delta\ell_2 = A_2 (V_2 \delta t)$. Thus, the amount of $B$ within region II, the outflow region, is its amount per unit volume, $\rho b$, times the volume

$$B_{II}(t + \delta t) = (\rho_2 b_2)(\delta\forall_{II}) = \rho_2 b_2 A_2 V_2 \, \delta t$$




where $b_2$ and $\rho_2$ are the constant values of $b$ and $\rho$ across section (2). Thus, the rate at which this property flows from the control volume, $\dot{B}_{out}$, is given by

$$\dot{B}_{out} = \lim_{\delta t \to 0} \frac{B_{II}(t + \delta t)}{\delta t} = \rho_2 A_2 V_2 \, b_2 \tag{4.12}$$

Similarly, the inflow of $B$ into the control volume across section (1) during the time interval $\delta t$ corresponds to that in region I and is given by the amount per unit volume times the volume, $\delta\forall_1 = A_1 \, \delta\ell_1 = A_1(V_1 \, \delta t)$. Hence,

$$B_I(t + \delta t) = (\rho_1 b_1)(\delta\forall_1) = \rho_1 b_1 A_1 V_1 \, \delta t$$

where $b_1$ and $\rho_1$ are the constant values of $b$ and $\rho$ across section (1). Thus, the rate of inflow of the property $B$ into the control volume, $\dot{B}_{in}$, is given by

$$\dot{B}_{in} = \lim_{\delta t \to 0} \frac{B_I(t + \delta t)}{\delta t} = \rho_1 A_1 V_1 b_1 \tag{4.13}$$

If we combine Eqs. 4.10, 4.11, 4.12, and 4.13 we see that the relationship between the time rate of change of $B$ for the system and that for the control volume is given by

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\delta t} + \dot{B}_{out} - \dot{B}_{in} \tag{4.14}$$

or

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\partial t} + \rho_2 A_2 V_2 b_2 - \rho_1 A_1 V_1 b_1 \tag{4.15}$$

*The time derivative associated with a system may be different from that for a control volume.*

This is a version of the Reynolds transport theorem valid under the restrictive assumptions associated with the flow shown in Fig. 4.11—fixed control volume with one inlet and one outlet having uniform properties (density, velocity, and the parameter $b$) across the inlet and outlet with the velocity normal to sections (1) and (2). Note that the time rate of change of $B$ for the system (the left-hand side of Eq. 4.15 or the quantity in Eq. 4.8) is not necessarily the same as the rate of change of $B$ within the control volume (the first term on the right-hand side of Eq. 4.15 or the quantity in Eq. 4.9). This is true because the inflow rate ($b_1\rho_1 V_1 A_1$) and the outflow rate ($b_2\rho_2 V_2 A_2$) of the property $B$ for the control volume need not be the same.

## EXAMPLE 4.8 Use of the Reynolds Transport Theorem

**GIVEN** Consider again the flow from the fire extinguisher shown in Fig. E4.7. Let the extensive property of interest be the system mass ($B = m$, the system mass, or $b = 1$).

**FIND** Write the appropriate form of the Reynolds transport theorem for this flow.

### SOLUTION

Again we take the control volume to be the fire extinguisher, and the system to be the fluid within it at time $t = 0$. For this case there is no inlet, section (1), across which the fluid flows into the control volume ($A_1 = 0$). There is, however, an outlet, section (2). Thus, the Reynolds transport theorem, Eq. 4.15, along with Eq. 4.9 with $b = 1$ can be written as

$$\frac{Dm_{sys}}{Dt} = \frac{\partial\left(\int_{cv} \rho\, d\forall\right)}{\partial t} + \rho_2 A_2 V_2 \qquad (1) \quad \text{(Ans)}$$

**COMMENT** If we proceed one step further and use the basic law of conservation of mass, we may set the left-hand side of this equation equal to zero (the amount of mass in a system is constant) and rewrite Eq. 1 in the form

$$\frac{\partial\left(\int_{cv} \rho\, d\forall\right)}{\partial t} = -\rho_2 A_2 V_2 \tag{2}$$

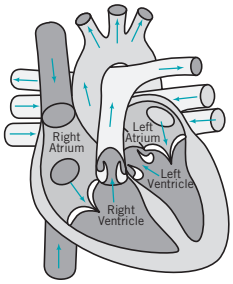
The physical interpretation of this result is that the rate at which the mass in the tank decreases in time is equal in magnitude but opposite to the rate of flow of mass from the exit, $\rho_2 A_2 V_2$. Note the units for the two terms of Eq. 2 (kg/s or slugs/s). Note that if there were both an inlet and an outlet to the control volume shown in Fig. E4.7, Eq. 2 would become

$$\frac{\partial\left(\int_{cv} \rho\, d\forall\right)}{\partial t} = \rho_1 A_1 V_1 - \rho_2 A_2 V_2 \tag{3}$$

In addition, if the flow were steady, the left-hand side of Eq. 3 would be zero (the amount of mass in the control would be constant in time) and Eq. 3 would become

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

This is one form of the conservation of mass principle discussed in Sect. 3.6.2—the mass flowrates into and out of the control volume are equal. Other more general forms are discussed in Chapter 5.




Equation 4.15 is a simplified version of the Reynolds transport theorem. We will now derive it for much more general conditions. A general, fixed control volume with fluid flowing through it is shown in Fig. 4.12. The flow field may be quite simple (as in the above one-dimensional flow considerations), or it may involve a quite complex, unsteady, three-dimensional situation such as the flow through a human heart as illustrated by the figure in the margin. In any case we again consider the system to be the fluid within the control volume at the initial time $t$. A short time later a portion of the fluid (region II) has exited from the control volume and additional fluid (region I, not part of the original system) has entered the control volume.

CMS Cepcor EX 1046 Page 80
CMS Cepcor v. Sandvik IPR2024-01380

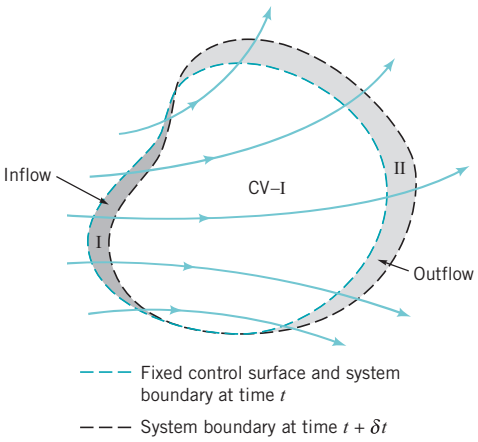



**FIGURE 4.12** Control volume and system for flow through an arbitrary, fixed control volume.

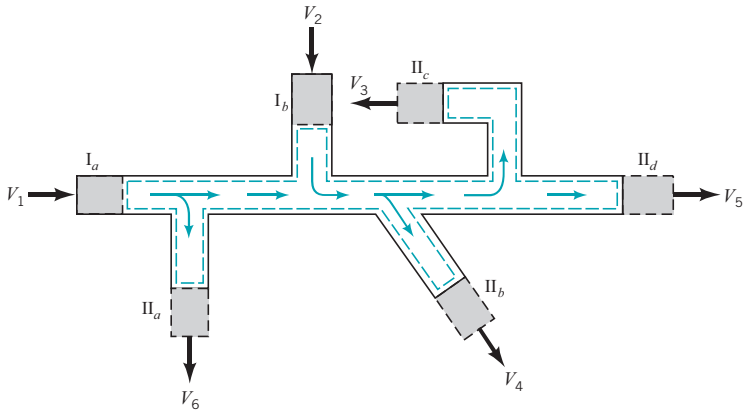



**FIGURE 4.13** Typical control volume with more than one inlet and outlet.

*The simplified Reynolds transport theorem can be easily generalized.*

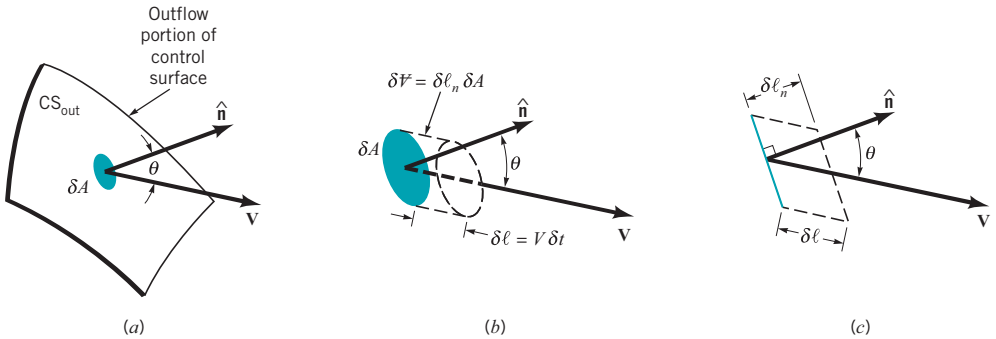
We consider an extensive fluid property $B$ and seek to determine how the rate of change of $B$ associated with the system is related to the rate of change of $B$ within the control volume at any instant. By repeating the exact steps that we did for the simplified control volume shown in Fig. 4.11, we see that Eq. 4.14 is valid for the general case also, provided that we give the correct interpretation to the terms $\dot{B}_{\text{out}}$ and $\dot{B}_{\text{in}}$. In general, the control volume may contain more (or less) than one inlet and one outlet. A typical pipe system may contain several inlets and outlets as are shown in Fig. 4.13. In such instances we think of all inlets grouped together ($\text{I} = \text{I}_a + \text{I}_b + \text{I}_c + \cdots$) and all outlets grouped together ($\text{II} = \text{II}_a + \text{II}_b + \text{II}_c + \cdots$), at least conceptually.

The term $\dot{B}_{\text{out}}$ represents the net flowrate of the property $B$ from the control volume. Its value can be thought of as arising from the addition (integration) of the contributions through each infinitesimal area element of size $\delta A$ on the portion of the control surface dividing region II and the control volume. This surface is denoted $\text{CS}_{\text{out}}$. As is indicated in Fig. 4.14, in time $\delta t$ the volume of fluid that passes across each area element is given by $\delta \mathcal{V} = \delta \ell_n \, \delta A$, where $\delta \ell_n = \delta \ell \cos \theta$ is the height (normal to the base, $\delta A$) of the small volume element, and $\theta$ is the angle between the velocity vector and the outward pointing normal to the surface, $\hat{\mathbf{n}}$. Thus, since $\delta \ell = V \, \delta t$, the amount of the property $B$ carried across the area element $\delta A$ in the time interval $\delta t$ is given by

$$\delta B = b\rho \, \delta \mathcal{V} = b\rho (V \cos \theta \, \delta t) \, \delta A$$

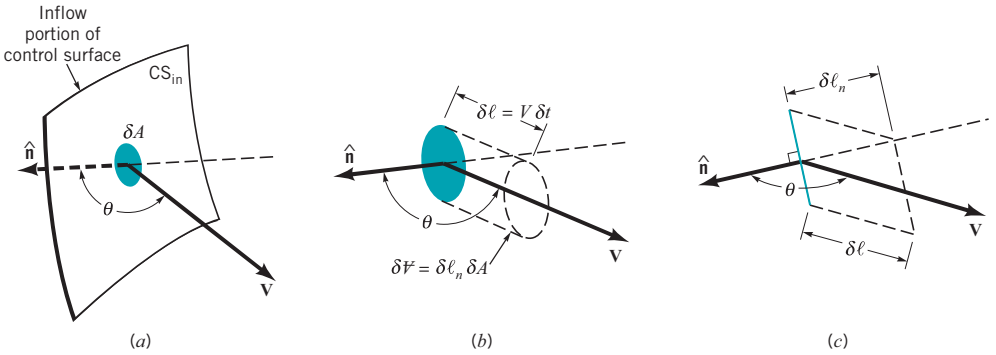
The rate at which $B$ is carried out of the control volume across the small area element $\delta A$, denoted $\delta \dot{B}_{\text{out}}$, is

$$\delta \dot{B}_{\text{out}} = \lim_{\delta t \to 0} \frac{\rho b \, \delta \mathcal{V}}{\delta t} = \lim_{\delta t \to 0} \frac{(\rho b V \cos \theta \, \delta t) \, \delta A}{\delta t} = \rho b V \cos \theta \, \delta A$$




(a) (b) (c)

**FIGURE 4.14** Outflow across a typical portion of the control surface.




■ **FIGURE 4.15** **Inflow across a typical portion of the control surface.**

By integrating over the entire outflow portion of the control surface, $CS_{out}$, we obtain

$$\dot{B}_{out} = \int_{CS_{out}} d\dot{B}_{out} = \int_{CS_{out}} \rho b V \cos\theta \, dA$$

The quantity $V \cos\theta$ is the component of the velocity normal to the area element $\delta A$. From the definition of the dot product, this can be written as $V \cos\theta = \mathbf{V} \cdot \hat{\mathbf{n}}$. Hence, an alternate form of the outflow rate is

*The flowrate of a parameter across the control surface is written in terms of a surface integral.*

$$\dot{B}_{out} = \int_{CS_{out}} \rho b \mathbf{V} \cdot \hat{\mathbf{n}} \, dA \tag{4.16}$$

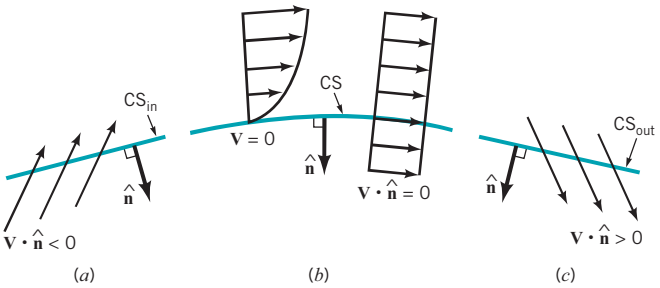
In a similar fashion, by considering the inflow portion of the control surface, $CS_{in}$, as shown in Fig. 4.15, we find that the inflow rate of $B$ into the control volume is

$$\dot{B}_{in} = -\int_{CS_{in}} \rho b V \cos\theta \, dA = -\int_{CS_{in}} \rho b \mathbf{V} \cdot \hat{\mathbf{n}} \, dA \tag{4.17}$$

We use the standard notation that the unit normal vector to the control surface, $\hat{\mathbf{n}}$, points out from the control volume. Thus, as is shown in Fig. 4.16, $-90° < \theta < 90°$ for outflow regions (the normal component of $\mathbf{V}$ is positive; $\mathbf{V} \cdot \hat{\mathbf{n}} > 0$). For inflow regions $90° < \theta < 270°$ (the normal component of $\mathbf{V}$ is negative; $\mathbf{V} \cdot \hat{\mathbf{n}} < 0$). The value of $\cos\theta$ is, therefore, positive on the $CV_{out}$ portions of the control surface and negative on the $CV_{in}$ portions. Over the remainder of the control surface, there is no inflow or outflow, leading to $\mathbf{V} \cdot \hat{\mathbf{n}} = V \cos\theta = 0$ on those portions. On such portions either $V = 0$ (the fluid "sticks" to the surface) or $\cos\theta = 0$ (the fluid "slides"along the surface without crossing it) (see Fig. 4.16). Therefore, the net flux (flowrate) of parameter $B$ across the entire control surface is

$$\begin{aligned} \dot{B}_{out} - \dot{B}_{in} &= \int_{CS_{out}} \rho b \mathbf{V} \cdot \hat{\mathbf{n}} \, dA - \left( -\int_{CS_{in}} \rho b \mathbf{V} \cdot \hat{\mathbf{n}} \, dA \right) \\ &= \int_{CS} \rho b \mathbf{V} \cdot \hat{\mathbf{n}} \, dA \end{aligned} \tag{4.18}$$

where the integration is over the entire control surface.




■ **FIGURE 4.16**
**Possible velocity configurations on portions of the control surface: (*a*) inflow, (*b*) no flow across the surface, (*c*) outflow.**

By combining Eqs. 4.14 and 4.18 we obtain

$$\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\partial t} + \int_{cs} \rho b \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$$

This can be written in a slightly different form by using $B_{cv} = \int_{cv} \rho b\, d\forall$ so that

$$\boxed{\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{cv} \rho b\, d\forall + \int_{cs} \rho b\, \mathbf{V} \cdot \hat{\mathbf{n}}\, dA} \qquad (4.19)$$

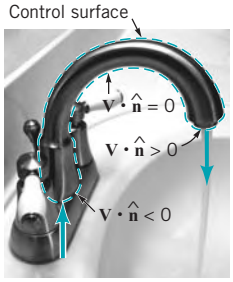
Equation 4.19 is the general form of the Reynolds transport theorem for a fixed, nondeforming control volume. Its interpretation and use are discussed in the following sections.

### 4.4.2 Physical Interpretation

The Reynolds transport theorem as given in Eq. 4.19 is widely used in fluid mechanics (and other areas as well). At first it appears to be a rather formidable mathematical expression—perhaps one to be steered clear of if possible. However, a physical understanding of the concepts involved will show that it is a rather straightforward, relatively easy-to-use tool. Its purpose is to provide a link between control volume ideas and system ideas.

The left side of Eq. 4.19 is the time rate of change of an arbitrary extensive parameter of a system. This may represent the rate of change of mass, momentum, energy, or angular momentum of the system, depending on the choice of the parameter $B$.

Because the system is moving and the control volume is stationary, the time rate of change of the amount of $B$ within the control volume is not necessarily equal to that of the system. The first term on the right side of Eq. 4.19 represents the rate of change of $B$ within the control volume as the fluid flows through it. Recall that $b$ is the amount of $B$ per unit mass, so that $\rho b\, d\forall$ is the amount of $B$ in a small volume $d\forall$. Thus, the time derivative of the integral of $\rho b$ throughout the control volume is the time rate of change of $B$ within the control volume at a given time.




The last term in Eq. 4.19 (an integral over the control surface) represents the net flowrate of the parameter $B$ across the entire control surface. As illustrated by the figure in the margin, over a portion of the control surface this property is being carried out of the control volume ($\mathbf{V} \cdot \hat{\mathbf{n}} > 0$); over other portions it is being carried into the control volume ($\mathbf{V} \cdot \hat{\mathbf{n}} < 0$). Over the remainder of the control surface there is no transport of $B$ across the surface since $b\mathbf{V} \cdot \hat{\mathbf{n}} = 0$, because either $b = 0$, $\mathbf{V} = 0$, or $\mathbf{V}$ is parallel to the surface at those locations. The mass flowrate through area element $\delta A$, given by $\rho \mathbf{V} \cdot \hat{\mathbf{n}}\, \delta A$, is positive for outflow (efflux) and negative for inflow (influx). Each fluid particle or fluid mass carries a certain amount of $B$ with it, as given by the product of $B$ per unit mass, $b$, and the mass. The rate at which this $B$ is carried across the control surface is given by the area integral term of Eq. 4.19. This net rate across the entire control surface may be negative, zero, or positive depending on the particular situation involved.

### 4.4.3 Relationship to Material Derivative

In Section 4.2.1 we discussed the concept of the material derivative $D(\ )/Dt = \partial(\ )/\partial t + \mathbf{V} \cdot \nabla(\ ) = \partial(\ )\partial t + u\, \partial(\ )\partial x + v\, \partial(\ )/\partial y + w\, \partial(\ )/\partial z$. The physical interpretation of this derivative is that it provides the time rate of change of a fluid property (temperature, velocity, etc.) associated with a particular fluid particle as it flows. The value of that parameter for that particle may change because of unsteady effects [the $\partial(\ )/\partial t$ term] or because of effects associated with the particle's motion [the $\mathbf{V} \cdot \nabla(\ )$ term].

*The Reynolds transport theorem is the integral counterpart of the material derivative.*

Careful consideration of Eq. 4.19 indicates the same type of physical interpretation for the Reynolds transport theorem. The term involving the time derivative of the control volume integral represents unsteady effects associated with the fact that values of the parameter within the control volume may change with time. For steady flow this effect vanishes—fluid flows through the control volume but the amount of any property, $B$, within the control volume is constant in time. The term involving the control surface integral represents the convective effects associated with the flow of the system across the fixed control surface. The sum of these two terms gives the rate of change of the parameter $B$ for the system. This corresponds to the interpretation of the material derivative,

CMS Cepcor EX 1046 Page 83
CMS Cepcor v. Sandvik IPR2024-01380

$D(\ )/Dt = \partial(\ )/\partial t + \mathbf{V} \cdot \nabla(\ )$, in which the sum of the unsteady effect and the convective effect gives the rate of change of a parameter for a fluid particle. As is discussed in Section 4.2, the material derivative operator may be applied to scalars (such as temperature) or vectors (such as velocity). This is also true for the Reynolds transport theorem. The particular parameters of interest, $B$ and $b$, may be scalars or vectors.

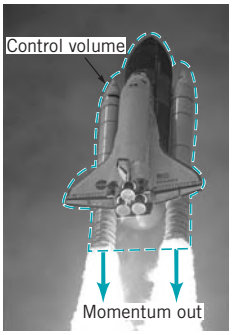
Thus, both the material derivative and the Reynolds transport theorem equations represent ways to transfer from the Lagrangian viewpoint (follow a particle or follow a system) to the Eulerian viewpoint (observe the fluid at a given location in space or observe what happens in the fixed control volume). The material derivative (Eq. 4.5) is essentially the infinitesimal (or derivative) equivalent of the finite size (or integral) Reynolds transport theorem (Eq. 4.19).

### 4.4.4 Steady Effects

Consider a steady flow $[\partial(\ )/\partial t \equiv 0]$ so that Eq. 4.19 reduces to

$$\frac{DB_{sys}}{Dt} = \int_{cs} \rho b \mathbf{V} \cdot \hat{\mathbf{n}}\, dA \tag{4.20}$$

In such cases if there is to be a change in the amount of $B$ associated with the system (nonzero left-hand side), there must be a net difference in the rate that $B$ flows into the control volume compared with the rate that it flows out of the control volume. That is, the integral of $\rho b \mathbf{V} \cdot \hat{\mathbf{n}}$ over the inflow portions of the control surface would not be equal and opposite to that over the outflow portions of the surface.




**(Photograph courtesy of NASA.)**

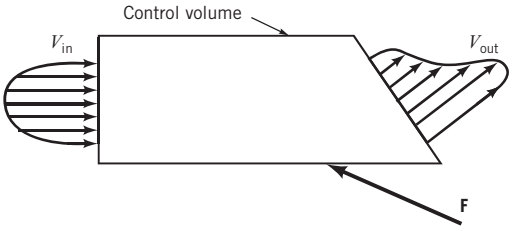
Consider steady flow through the "black box" control volume that is shown in Fig. 4.17. If the parameter $B$ is the mass of the system, the left-hand side of Eq. 4.20 is zero (conservation of mass for the system as discussed in detail in Section 5.1). Hence, the flowrate of mass into the box must be the same as the flowrate of mass out of the box because the right-hand side of Eq. 4.20 represents the net flowrate through the control surface. On the other hand, assume the parameter $B$ is the momentum of the system. The momentum of the system need not be constant. In fact, according to Newton's second law the time rate of change of the system momentum equals the net force, $\mathbf{F}$, acting on the system. In general, the left-hand side of Eq. 4.20 will therefore be nonzero. Thus, the right-hand side, which then represents the net flux of momentum across the control surface, will be nonzero. The flowrate of momentum into the control volume need not be the same as the flux of momentum from the control volume. We will investigate these concepts much more fully in Chapter 5. They are the basic principles describing the operation of such devices as jet or rocket engines like the one shown in the figure in the margin.

For steady flows the amount of the property $B$ within the control volume does not change with time. The amount of the property associated with the system may or may not change with time, depending on the particular property considered and the flow situation involved. The difference between that associated with the control volume and that associated with the system is determined by the rate at which $B$ is carried across the control surface—the term $\int_{cs} \rho b \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$.

*The Reynolds transport theorem involves both steady and unsteady effects.*

### 4.4.5 Unsteady Effects

Consider unsteady flow $[\partial(\ )/\partial t \neq 0]$ so that all terms in Eq. 4.19 must be retained. When they are viewed from a control volume standpoint, the amount of parameter $B$ within the system may change because the amount of $B$ within the fixed control volume may change with time




■ **FIGURE 4.17** **Steady flow through a control volume.**

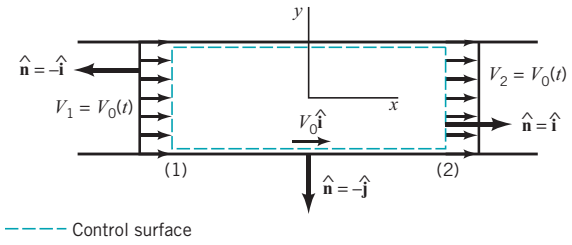



**FIGURE 4.18** Unsteady flow through a constant diameter pipe.

*For some flow situations, certain portions of the Reynolds transport theorem are automatically zero.*

[the $\partial(\int_{cv} \rho b\, d\forall)/\partial t$ term] and because there may be a net nonzero flow of that parameter across the control surface (the $\int_{cs} \rho b \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$ term).

For the special unsteady situations in which the rate of inflow of parameter $B$ is exactly balanced by its rate of outflow, it follows that $\int_{cs} \rho b \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = 0$, and Eq. 4.19 reduces to

$$\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{cv} \rho b\, d\forall \tag{4.21}$$

For such cases, any rate of change in the amount of $B$ associated with the system is equal to the rate of change of $B$ within the control volume. This can be illustrated by considering flow through a constant diameter pipe as is shown in Fig. 4.18. The control volume is as shown, and the system is the fluid within this volume at time $t_0$. We assume the flow is one-dimensional with $\mathbf{V} = V_0\hat{\mathbf{i}}$, where $V_0(t)$ is a function of time, and that the density is constant. At any instant in time, all particles in the system have the same velocity. We let $\mathbf{B}$ = system momentum = $m\mathbf{V} = mV_0\hat{\mathbf{i}}$, where $m$ is the system mass, so that $\mathbf{b} = \mathbf{B}/m = \mathbf{V} = V_0\hat{\mathbf{i}}$, the fluid velocity. The magnitude of the momentum efflux across the outlet [section (2)] is the same as the magnitude of the momentum influx across the inlet [section (1)]. However, the sign of the efflux is opposite to that of the influx since $\mathbf{V} \cdot \hat{\mathbf{n}} > 0$ for the outflow and $\mathbf{V} \cdot \hat{\mathbf{n}} < 0$ for the inflow. Note that $\mathbf{V} \cdot \hat{\mathbf{n}} = 0$ along the sides of the control volume. Thus, with $\mathbf{V} \cdot \hat{\mathbf{n}} = -V_0$ on section (1), $\mathbf{V} \cdot \hat{\mathbf{n}} = V_0$ on section (2), and $A_1 = A_2$, we obtain

$$\begin{aligned}\int_{cs} \rho b \mathbf{V} \cdot \hat{\mathbf{n}}\, dA &= \int_{cs} \rho(V_0\hat{\mathbf{i}})(\mathbf{V} \cdot \hat{\mathbf{n}})\, dA \\ &= \int_{(1)} \rho(V_0\hat{\mathbf{i}})(-V_0)\, dA + \int_{(2)} \rho(V_0\hat{\mathbf{i}})(V_0)\, dA \\ &= -\rho V_0^2 A_1\hat{\mathbf{i}} + \rho V_0^2 A_2\hat{\mathbf{i}} = 0\end{aligned}$$

It is seen that for this special case Eq. 4.21 is valid. The rate at which the momentum of the system changes with time is the same as the rate of change of momentum within the control volume. If $V_0$ is constant in time, there is no rate of change of momentum of the system and for this special case each of the terms in the Reynolds transport theorem is zero by itself.

Consider the flow through a variable area pipe shown in Fig. 4.19. In such cases the fluid velocity is not the same at section (1) as it is at (2). Hence, the efflux of momentum from the control volume is not equal to the influx of momentum, so that the convective term in Eq. 4.20 [the integral of $\rho\mathbf{V}(\mathbf{V} \cdot \hat{\mathbf{n}})$ over the control surface] is not zero. These topics will be discussed in considerably more detail in Chapter 5.

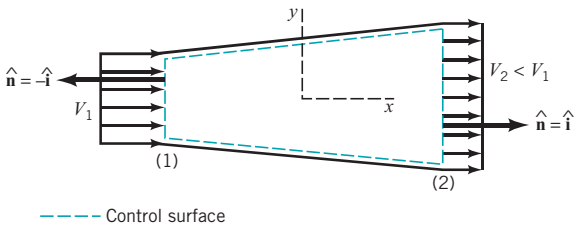
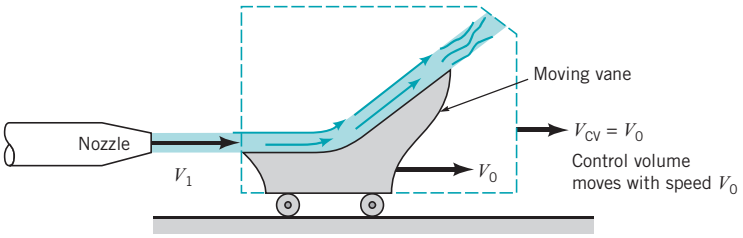



**FIGURE 4.19** Flow through a variable area pipe.

CMS Cepcor EX 1046 Page 85
CMS Cepcor v. Sandvik IPR2024-01380




■ FIGURE 4.20 **Example of a moving control volume.**

### 4.4.6 Moving Control Volumes

For most problems in fluid mechanics, the control volume may be considered as a fixed volume through which the fluid flows. There are, however, situations for which the analysis is simplified if the control volume is allowed to move or deform. The most general situation would involve a control volume that moves, accelerates, and deforms. As one might expect, the use of these control volumes can become fairly complex.

A number of important problems can be most easily analyzed by using a nondeforming control volume that moves with a constant velocity. Such an example is shown in Fig. 4.20 in which a stream of water with velocity $\mathbf{V}_1$ strikes a vane that is moving with constant velocity $\mathbf{V}_0$. It may be of interest to determine the force, $\mathbf{F}$, that the water puts on the vane. Such problems frequently occur in turbines where a stream of fluid (water or steam, for example) strikes a series of blades that move past the nozzle. To analyze such problems it is advantageous to use a moving control volume. We will obtain the Reynolds transport theorem for such control volumes.

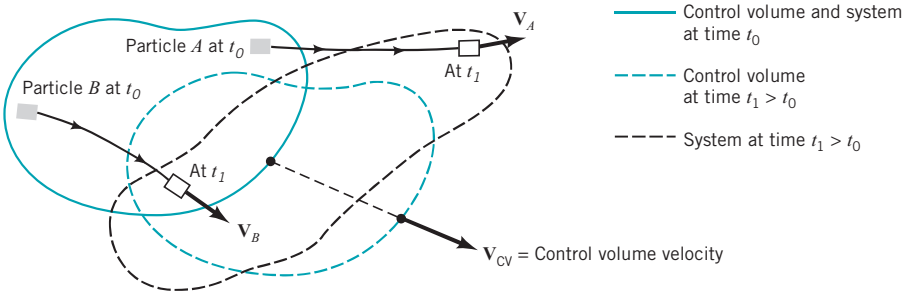
We consider a control volume that moves with a constant velocity as is shown in Fig. 4.21. The shape, size, and orientation of the control volume do not change with time. The control volume merely translates with a constant velocity, $\mathbf{V}_{cv}$, as shown. In general, the velocity of the control volume and the fluid are not the same, so that there is a flow of fluid through the moving control volume just as in the stationary control volume cases discussed in Section 4.4.2. The main difference between the fixed and the moving control volume cases is that it is the *relative velocity*, $\mathbf{W}$, that carries fluid across the moving control surface, whereas it is the *absolute velocity*, $\mathbf{V}$, that carries the fluid across the fixed control surface. The relative velocity is the fluid velocity relative to the moving control volume—the fluid velocity seen by an observer riding along on the control volume. The absolute velocity is the fluid velocity as seen by a stationary observer in a fixed coordinate system.

*The absolute and relative velocities differ by an amount equal to the control volume velocity.*

The difference between the absolute and relative velocities is the velocity of the control volume, $\mathbf{V}_{cv} = \mathbf{V} - \mathbf{W}$, or

$$\mathbf{V} = \mathbf{W} + \mathbf{V}_{cv} \tag{4.22}$$

Since the velocity is a vector, we must use vector addition as is shown in Fig. 4.22 to obtain the relative velocity if we know the absolute velocity and the velocity of the control volume. Thus, if the water leaves the nozzle in Fig. 4.20 with a velocity of $\mathbf{V}_1 = 100\hat{\mathbf{i}}$ ft/s and the vane has a velocity of $\mathbf{V}_0 = 20\hat{\mathbf{i}}$ ft/s (the same as the control volume), it appears to an observer riding on the vane that the water approaches the vane with a velocity of $\mathbf{W} = \mathbf{V} - \mathbf{V}_{cv} = 80\hat{\mathbf{i}}$ ft/s. In general, the absolute




■ FIGURE 4.21 **Typical moving control volume and system.**

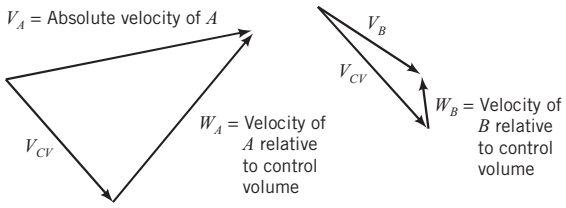



**FIGURE 4.22**
**Relationship between absolute and relative velocities.**

velocity, $\mathbf{V}$, and the control volume velocity, $\mathbf{V}_{cv}$, will not be in the same direction so that the relative and absolute velocities will have different directions (see Fig. 4.22).

The Reynolds transport theorem for a moving, nondeforming control volume can be derived in the same manner that it was obtained for a fixed control volume. As is indicated in Fig. 4.23, the only difference that needs be considered is the fact that relative to the moving control volume the fluid velocity observed is the relative velocity, not the absolute velocity. An observer fixed to the moving control volume may or may not even know that he or she is moving relative to some fixed coordinate system. If we follow the derivation that led to Eq. 4.19 (the Reynolds transport theorem for a fixed control volume), we note that the corresponding result for a moving control volume can be obtained by simply replacing the absolute velocity, $\mathbf{V}$, in that equation by the relative velocity, $\mathbf{W}$. Thus, the Reynolds transport theorem for a control volume moving with constant velocity is given by

*The Reynolds transport theorem for a moving control volume involves the relative velocity.*

$$\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{cv} \rho b \, d\forall + \int_{cs} \rho b \, \mathbf{W} \cdot \hat{\mathbf{n}} \, dA \qquad \textbf{(4.23)}$$

where the relative velocity is given by Eq. 4.22.

### 4.4.7 Selection of a Control Volume

Any volume in space can be considered as a control volume. It may be of finite size or it may be infinitesimal in size, depending on the type of analysis to be carried out. In most of our cases, the control volume will be a fixed, nondeforming volume. In some situations we will consider control volumes that move with constant velocity. In either case it is important that considerable thought go into the selection of the specific control volume to be used.

The selection of an appropriate control volume in fluid mechanics is very similar to the selection of an appropriate free-body diagram in dynamics or statics. In dynamics, we select the body in which we are interested, represent the object in a free-body diagram, and then apply the appropriate governing laws to that body. The ease of solving a given dynamics problem is often very dependent on the specific object that we select for use in our free-body diagram. Similarly, the ease of solving a given fluid mechanics problem is often very dependent on the choice of the control volume used. Only by practice can we develop skill at selecting the "best" control volume. None are "wrong," but some are "much better" than others.

Solution of a typical problem will involve determining parameters such as velocity, pressure, and force at some point in the flow field. It is usually best to ensure that this point is located on the control surface, not "buried" within the control volume. The unknown will then appear in the convective term (the surface integral) of the Reynolds transport theorem. If possible, the control

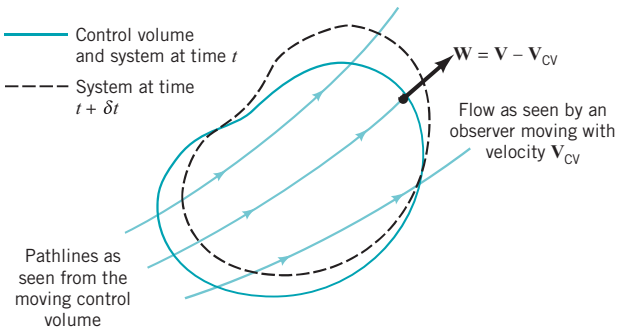
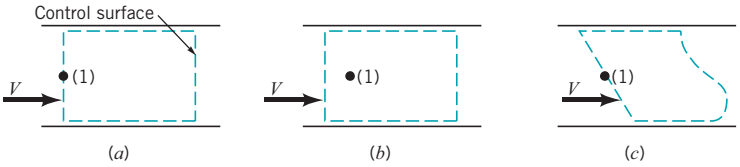
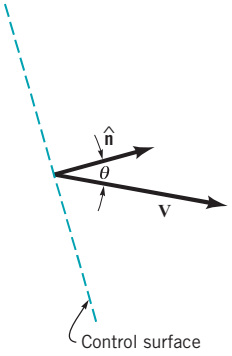



**FIGURE 4.23**
**Control volume and system as seen by an observer moving with the control volume.**

CMS Cepcor EX 1046 Page 87
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 4.24** **Various control volumes for flow through a pipe.**




surface should be normal to the fluid velocity so that the angle $\theta$ ($\mathbf{V} \cdot \hat{\mathbf{n}} = V \cos\theta$ as shown by the figure in the margin) in the flux terms of Eq. 4.19 will be 0 or 180°. This will usually simplify the solution process.

Figure 4.24 illustrates three possible control volumes associated with flow through a pipe. If the problem is to determine the pressure at point (1), the selection of the control volume (*a*) is better than that of (*b*) because point (1) lies on the control surface. Similarly, control volume (*a*) is better than (*c*) because the flow is normal to the inlet and exit portions of the control volume. None of these control volumes are wrong—(*a*) will be easier to use. Proper control volume selection will become much clearer in Chapter 5 where the Reynolds transport theorem is used to transform the governing equations from the system formulation into the control volume formulation, and numerous examples using control volume ideas are discussed.

## 4.5 Chapter Summary and Study Guide

*field representation*
*velocity field*
*Eulerian method*
*Lagrangian method*
*one-, two-, and three-dimensional flow*
*steady and unsteady flow*
*streamline*
*streakline*
*pathline*
*acceleration field*
*material derivative*
*local acceleration*
*convective acceleration*
*system*
*control volume*
*Reynolds transport theorem*

This chapter considered several fundamental concepts of fluid kinematics. That is, various aspects of fluid motion are discussed without regard to the forces needed to produce this motion. The concepts of a field representation of a flow and the Eulerian and Lagrangian approaches to describing a flow are introduced, as are the concepts of velocity and acceleration fields.

The properties of one-, two-, or three-dimensional flows and steady or unsteady flows are introduced along with the concepts of streamlines, streaklines, and pathlines. Streamlines, which are lines tangent to the velocity field, are identical to streaklines and pathlines if the flow is steady. For unsteady flows, they need not be identical.

As a fluid particle moves about, its properties (i.e., velocity, density, temperature) may change. The rate of change of these properties can be obtained by using the material derivative, which involves both unsteady effects (time rate of change at a fixed location) and convective effects (time rate of change due to the motion of the particle from one location to another).

The concepts of a control volume and a system are introduced, and the Reynolds transport theorem is developed. By using these ideas, the analysis of flows can be carried out using a control volume (a volume, usually fixed, through which the fluid flows), whereas the governing principles are stated in terms of a system (a flowing portion of fluid).

The following checklist provides a study guide for this chapter. When your study of the entire chapter and end-of-chapter exercises has been completed you should be able to

- write out meaning of the terms listed here in the margin and understand each of the related concepts. These terms are particularly important and are set in ***italic, bold, and color*** type in the text.
- understand the concept of the field representation of a flow and the difference between Eulerian and Lagrangian methods of describing a flow.
- explain the differences among streamlines, streaklines, and pathlines.
- calculate and plot streamlines for flows with given velocity fields.
- use the concept of the material derivative, with its unsteady and convective effects, to determine time rate of change of a fluid property.
- determine the acceleration field for a flow with a given velocity field.
- understand the properties of and differences between a system and a control volume.
- interpret, physically and mathematically, the concepts involved in the Reynolds transport theorem.

Some of the important equations in this chapter are:

| | | |
|---|---|---|
| Equation for streamlines | $\frac{dy}{dx} = \frac{v}{u}$ | (4.1) |
| Acceleration | $\mathbf{a} = \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z}$ | (4.3) |
| Material derivative | $\frac{D(\ )}{Dt} = \frac{\partial(\ )}{\partial t} + (\mathbf{V}\cdot\nabla)(\ )$ | (4.6) |
| Streamwise and normal components of acceleration | $a_s = V\frac{\partial V}{\partial s}, \qquad a_n = \frac{V^2}{\mathcal{R}}$ | (4.7) |
| Reynolds transport theorem (restricted form) | $\frac{DB_{sys}}{Dt} = \frac{\partial B_{cv}}{\partial t} + \rho_2 A_2 V_2 b_2 - \rho_1 A_1 V_1 b_1$ | (4.15) |
| Reynolds transport theorem (general form) | $\frac{DB_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{cv} \rho b\, d\forall + \int_{cs} \rho b\, \mathbf{V}\cdot\hat{\mathbf{n}}\, dA$ | (4.19) |
| Relative and absolute velocities | $\mathbf{V} = \mathbf{W} + \mathbf{V}_{cv}$ | (4.22) |

## References


1. Streeter, V. L., and Wylie, E. B., *Fluid Mechanics*, 8th Ed., McGraw-Hill, New York, 1985.
2. Goldstein, R. J., *Fluid Mechanics Measurements*, Hemisphere, New York, 1983.
3. Homsy, G. M., et al., *Multimedia Fluid Mechanics* CD-ROM, 2nd Ed., Cambridge University Press, New York, 2007.
4. Magarvey, R. H., and MacLatchy, C. S., The Formation and Structure of Vortex Rings, *Canadian Journal of Physics*, Vol. 42, 1964.


## Review Problems

Go to Appendix G for a set of review problems with answers. Detailed solutions can be found in *Student Solution Manual and Study Guide for Fundamentals of Fluid Mechanics*, by Munson et al. (© 2009 John Wiley and Sons, Inc.).

## Problems

**Note: Unless otherwise indicated, use the values of fluid properties found in the tables on the inside of the front cover. Problems designated with an (*) are intended to be solved with the aid of a programmable calculator or a computer. Problems designated with a (†) are "open-ended" problems and require critical thinking in that to work them one must make various assumptions and provide the necessary data. There is not a unique answer to these problems.**

**Answers to the even-numbered problems are listed at the end of the book. Access to the videos that accompany problems can be obtained through the book's web site, www.wiley.com/college/munson. The lab-type problems can also be accessed on this web site.**

### Section 4.1 The Velocity Field

**4.1** Obtain a photograph/image that shows a flowing fluid. Print this photo and write a brief paragraph that describes the flow in terms of an Eulerian description; a Lagrangian description.

**4.2** Obtain a photograph/image of a situation in which the unsteadiness of the flow is important. Print this photo and write a brief paragraph that describes the situation involved.

**4.3** Obtain a photograph/image of a situation in which a fluid is flowing. Print this photo and draw in some lines to represent how you think some streamlines may look. Write a brief paragraph to describe the acceleration of a fluid particle as it flows along one of these streamlines.

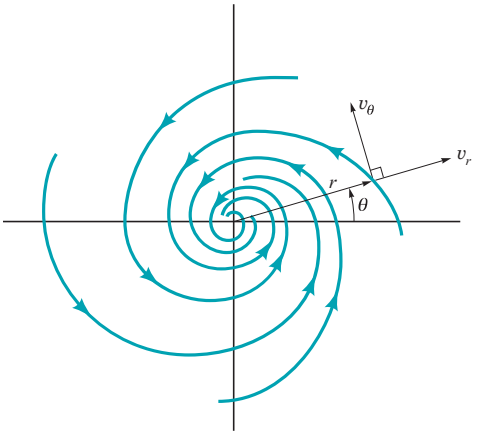
**4.4** The $x$- and $y$-components of a velocity field are given by $u = -(V_0/\ell)\,x$ and $v = -(V_0/\ell)\,y$, where $V_0$ and $\ell$ are constants. Make a sketch of the velocity field in the first quadrant ($x > 0, y > 0$) by drawing arrows representing the fluid velocity at representative locations.

**4.5** A two-dimensional velocity field is given by $u = 1 + y$ and $v = 1$. Determine the equation of the streamline that passes through the origin. On a graph, plot this streamline.

**4.6** The velocity field of a flow is given by $\mathbf{V} = (5z - 3)\hat{\mathbf{i}} + (x + 4)\hat{\mathbf{j}} + 4y\hat{\mathbf{k}}$ ft/s, where $x$, $y$, and $z$ are in feet. Determine the fluid speed at the origin ($x = y = z = 0$) and on the $x$ axis ($y = z = 0$).

**4.7** A flow can be visualized by plotting the velocity field as velocity vectors at representative locations in the flow as shown in Video V4.2 and Fig. E4.1. Consider the velocity field given in

polar coordinates by $v_r = -10/r$, and $v_\theta = 10/r$. This flow approximates a fluid swirling into a sink as shown in Fig. P4.7. Plot the velocity field at locations given by $r = 1$, 2, and 3 with $\theta = 0$, 30, 60, and 90°.




■ FIGURE P4.7

**4.8** The velocity field of a flow is given by $\mathbf{V} = 20y/(x^2 + y^2)^{1/2}\hat{\mathbf{i}} - 20x/(x^2 + y^2)^{1/2}\hat{\mathbf{j}}$ ft/s, where $x$ and $y$ are in feet. Determine the fluid speed at points along the $x$ axis; along the $y$ axis. What is the angle between the velocity vector and the $x$ axis at points $(x, y) = (5, 0)$, $(5, 5)$, and $(0, 5)$?

**4.9** The components of a velocity field are given by $u = x + y$, $v = xy^3 + 16$, and $w = 0$. Determine the location of any stagnation points ($V = 0$) in the flow field.

**4.10** The $x$ and $y$ components of velocity for a two-dimensional flow are $u = 6y$ ft/s and $v = 3$ ft/s, where $y$ is in feet. Determine the equation for the streamlines and sketch representative streamlines in the upper half plane.

**4.11** Show that the streamlines for a flow whose velocity components are $u = c(x^2 - y^2)$ and $v = -2cxy$, where $c$ is a constant, are given by the equation $x^2y - y^3/3 =$ constant. At which point (points) is the flow parallel to the $y$ axis? At which point (points) is the fluid stationary?

**4.12** A velocity field is given by $\mathbf{V} = x\hat{\mathbf{i}} + x(x - 1)(y + 1)\hat{\mathbf{j}}$, where $u$ and $v$ are in ft/s and $x$ and $y$ are in feet. Plot the streamline that passes through $x = 0$ and $y = 0$. Compare this streamline with the streakline through the origin.

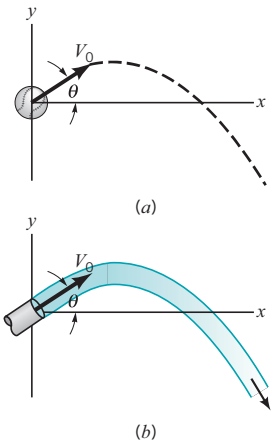
**4.13** From time $t = 0$ to $t = 5$ hr radioactive steam is released from a nuclear power plant accident located at $x = -1$ mile and $y = 3$ miles. The following wind conditions are expected: $\mathbf{V} = 10\hat{\mathbf{i}} - 5\hat{\mathbf{j}}$ mph for $0 < t < 3$ hr, $\mathbf{V} = 15\hat{\mathbf{i}} + 8\hat{\mathbf{j}}$ mph for $3 < t < 10$ hr, and $\mathbf{V} = 5\hat{\mathbf{i}}$ mph for $t > 10$ hr. Draw to scale the expected streakline of the steam for $t = 3$, 10, and 15 hr.

***4.14** Consider a ball thrown with initial speed $V_0$ at an angle of $\theta$ as shown in Fig. P4.14*a*. As discussed in beginning physics, if friction is negligible the path that the ball takes is given by

$$y = (\tan\theta)x - [g/(2\,V_0^2\cos^2\theta)]x^2$$

That is, $y = c_1x + c_2x^2$, where $c_1$ and $c_2$ are constants. The path is a parabola. The pathline for a stream of water leaving a small nozzle is shown in Fig. P4.14*b* and Video V4.12. The coordinates for this water stream are given in the following table. **(a)** Use the given data to determine appropriate values for $c_1$ and $c_2$ in the above equation and, thus, show that these water particles also follow a parabolic pathline. **(b)** Use your values of $c_1$ and $c_2$ to determine the speed of the water, $V_0$, leaving the nozzle.

| $x$ (in.) | $y$ (in.) |
|---|---|
| 0 | 0 |
| 0.25 | 0.13 |
| 0.50 | 0.16 |
| 0.75 | 0.13 |
| 1.0 | 0.00 |
| 1.25 | −0.20 |
| 1.50 | −0.53 |
| 1.75 | −0.90 |
| 2.00 | −1.43 |

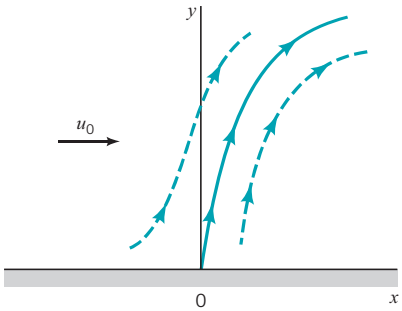



■ FIGURE P4.14

**4.15** The $x$ and $y$ components of a velocity field are given by $u = x^2y$ and $v = -xy^2$. Determine the equation for the streamlines of this flow and compare it with those in Example 4.2. Is the flow in this problem the same as that in Example 4.2? Explain.

**4.16** A flow in the $x$–$y$ plane is given by the following velocity field: $u = 3$ and $v = 6$ m/s for $0 < t < 20$ s; $u = -4$ and $v = 0$ m/s for $20 < t < 40$ s. Dye is released at the origin ($x = y = 0$) for $t \geq 0$. **(a)** Draw the pathlines at $t = 30$ s for two particles that were released from the origin—one released at $t = 0$ and the other released at $t = 20$ s. **(b)** On the same graph draw the streamlines at times $t = 10$ s and $t = 30$ s.

**4.17** In addition to the customary horizontal velocity components of the air in the atmosphere (the "wind"), there often are vertical air currents (thermals) caused by buoyant effects due to uneven heating of the air as indicated in Fig. P4.17. Assume that the velocity field in a certain region is approximated by $u = u_0$, $v = v_0\,(1 - y/h)$ for $0 < y < h$, and $u = u_0$, $v = 0$ for $y > h$. Plot the shape of the streamline that passes through the origin for values of $u_0/v_0 = 0.5$, 1, and 2.




■ FIGURE P4.17

***4.18** Repeat Problem 4.17 using the same information except that $u = u_0y/h$ for $0 \leq y \leq h$ rather than $u = u_0$. Use values of $u_0/v_0 = 0$, 0.1, 0.2, 0.4, 0.6, 0.8, and 1.0.

**4.19** As shown in Video V4.6 and Fig. P4.19, a flying airplane produces swirling flow near the end of its wings. In certain circumstances this flow can be approximated by the velocity field $u = -Ky/(x^2 + y^2)$ and $v = Kx/(x^2 + y^2)$, where $K$ is a constant depending on various parameters associated with the airplane (i.e., its weight, speed) and $x$ and $y$ are measured from the center of the swirl. **(a)** Show that for this flow the velocity is inversely proportional to the distance from the origin. That is, $V = K/(x^2 + y^2)^{1/2}$. **(b)** Show that the streamlines are circles.

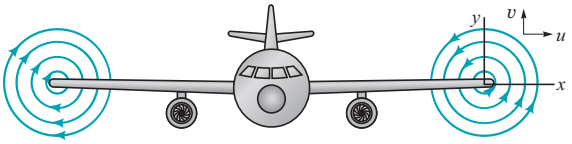



FIGURE P4.19

**4.20** (See Fluids in the News article titled **"Follow those particles,"** Section 4.1.) Two photographs of four particles in a flow past a sphere are superposed as shown in Fig. P4.20. The time interval between the photos is $\Delta t = 0.002$ s. The locations of the particles, as determined from the photos, are shown in the table. **(a)** Determine the fluid velocity for these particles. **(b)** Plot a graph to compare the results of part **(a)** with the theoretical velocity which is given by $V = V_0(1 + a^3/x^3)$, where $a$ is the sphere radius and $V_0$ is the fluid speed far from the sphere.

| Particle | $x$ at $t = 0$ s (ft) | $x$ at $t = 0.002$ s (ft) |
|---|---|---|
| 1 | −0.500 | −0.480 |
| 2 | −0.250 | −0.232 |
| 3 | −0.140 | −0.128 |
| 4 | −0.120 | −0.112 |

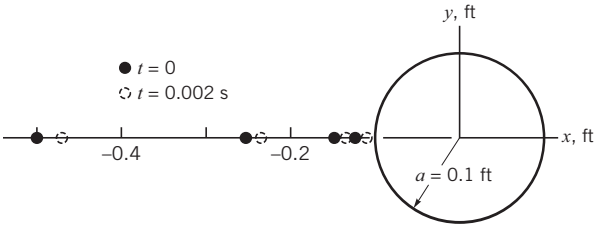



FIGURE P4.20

**4.21** (See Fluids in the News article titled **"Winds on Earth and Mars,"** Section 4.1.4.) A 10-ft-diameter dust devil that rotates one revolution per second travels across the Martian surface (in the $x$-direction) with a speed of 5 ft/s. Plot the pathline etched on the surface by a fluid particle 10 ft from the center of the dust devil for time $0 \le t \le 3$ s. The particle position is given by the sum of that for a stationary swirl $[x = 10\ \cos(2\pi t), y = 10\ \sin(2\pi t)]$ and that for a uniform velocity ($x = 5t$, $y =$ constant), where $x$ and $y$ are in feet and $t$ is in seconds.

## Section 4.2 The Acceleration Field

**4.22** The $x$- and $y$-components of a velocity field are given by $u = (V_0/\ell)x$ and $v = -(V_0/\ell)y$, where $V_0$ and $\ell$ are constants. Plot the streamlines for this flow and determine the acceleration field.

**4.23** A velocity field is given by $u = cx^2$ and $v = cy^2$, where $c$ is a constant. Determine the $x$ and $y$ components of the acceleration. At what point (points) in the flow field is the acceleration zero?

**4.24** Determine the acceleration field for a three-dimensional flow with velocity components $u = -x$, $v = 4x^2y^2$, and $w = x - y$.

**†4.25** Estimate the deceleration of a water particle in a raindrop as it strikes the sidewalk. List all assumptions and show all calculations.

**4.26** The velocity of air in the diverging pipe shown in Fig. P4.26 is given by $V_1 = 4t$ ft/s and $V_2 = 2t$ ft/s, where $t$ is in seconds. **(a)** Determine the local acceleration at points (1) and (2). **(b)** Is the average convective acceleration between these two points negative, zero, or positive? Explain.

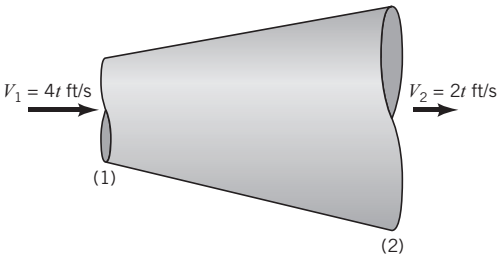



FIGURE P4.26

**4.27** Water flows in a pipe so that its velocity triples every 20 s. At $t = 0$ it has $u = 5$ ft/s. That is, $\mathbf{V} = u(t)\hat{\mathbf{i}} = 5\,(3^{t/20})\hat{\mathbf{i}}$ ft/s. Determine the acceleration when $t = 0$, 10, and 20 s.

**4.28** When a valve is opened, the velocity of water in a certain pipe is given by $u = 10(1 - e^{-t})$, $v = 0$, and $w = 0$, where $u$ is in ft/s and $t$ is in seconds. Determine the maximum velocity and maximum acceleration of the water.

**4.29** The velocity of the water in the pipe shown in Fig. P4.29 is given by $V_1 = 0.50t$ m/s and $V_2 = 1.0t$ m/s, where $t$ is in seconds. Determine the local acceleration at points (1) and (2). Is the average convective acceleration between these two points negative, zero, or positive? Explain.

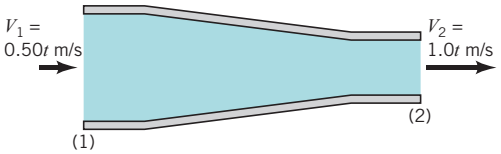



FIGURE P4.29

**4.30** A shock wave is a very thin layer (thickness = $\ell$) in a high-speed (supersonic) gas flow across which the flow properties (velocity, density, pressure, etc.) change from state (1) to state (2) as shown in Fig. P4.30. If $V_1 = 1800$ fps, $V_2 = 700$ fps, and $\ell = 10^{-4}$ in., estimate the average deceleration of the gas as it flows across the shock wave. How many $g$'s deceleration does this represent?

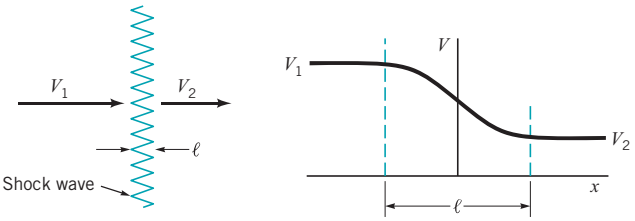
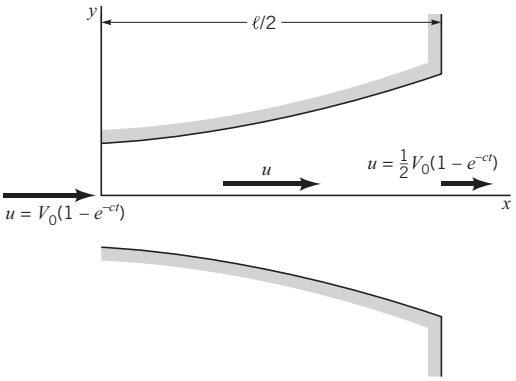



FIGURE P4.30

**†4.31** Estimate the average acceleration of water as it travels through the nozzle on your garden hose. List all assumptions and show all calculations.

**4.32** As a valve is opened, water flows through the diffuser shown in Fig. P4.32 at an increasing flowrate so that the velocity along the centerline is given by $\mathbf{V} = u\hat{\mathbf{i}} = V_0(1 - e^{-ct})\,(1 - x/\ell)\,\hat{\mathbf{i}}$, where $u_0$, $c$, and $\ell$ are constants. Determine the acceleration as a function of $x$ and $t$. If $V_0 = 10$ ft/s and $\ell = 5$ ft, what value of $c$ (other than $c = 0$) is needed to make the acceleration zero for any $x$ at $t = 1$ s? Explain how the acceleration can be zero if the flowrate is increasing with time.

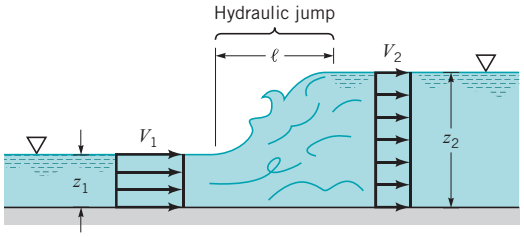
CMS Cepcor EX 1046 Page 91
CMS Cepcor v. Sandvik IPR2024-01380




■ FIGURE P4.32

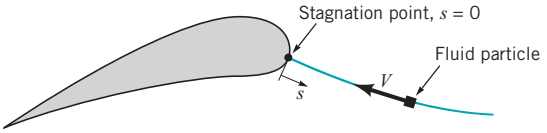
**4.33** A fluid flows along the $x$ axis with a velocity given by $\mathbf{V} = (x/t)\,\hat{\mathbf{i}}$, where $x$ is in feet and $t$ in seconds. **(a)** Plot the speed for $0 \le x \le 10$ ft and $t = 3$ s. **(b)** Plot the speed for $x = 7$ ft and $2 \le t \le 4$ s. **(c)** Determine the local and convective acceleration. **(d)** Show that the acceleration of any fluid particle in the flow is zero. **(e)** Explain physically how the velocity of a particle in this unsteady flow remains constant throughout its motion.

**4.34** A hydraulic jump is a rather sudden change in depth of a liquid layer as it flows in an open channel as shown in Fig. P4.34 and **Video V10.12**. In a relatively short distance (thickness = $\ell$) the liquid depth changes from $z_1$ to $z_2$, with a corresponding change in velocity from $V_1$ to $V_2$. If $V_1 = 1.20$ ft/s, $V_2 = 0.30$ ft/s, and $\ell = 0.02$ ft, estimate the average deceleration of the liquid as it flows across the hydraulic jump. How many $g$'s deceleration does this represent?




■ FIGURE P4.34

**4.35** A fluid particle flowing along a stagnation streamline, as shown in **Video V4.9** and Fig. P4.35, slows down as it approaches the stagnation point. Measurements of the dye flow in the video indicate that the location of a particle starting on the stagnation streamline a distance $s = 0.6$ ft upstream of the stagnation point at $t = 0$ is given approximately by $s = 0.6e^{-0.5t}$, where $t$ is in seconds and $s$ is in feet. **(a)** Determine the speed of a fluid particle as a function of time, $V_{\text{particle}}(t)$, as it flows along the streamline. **(b)** Determine the speed of the fluid as a function of position along the streamline, $V = V(s)$. **(c)** Determine the fluid acceleration along the streamline as a function of position, $a_s = a_s(s)$.

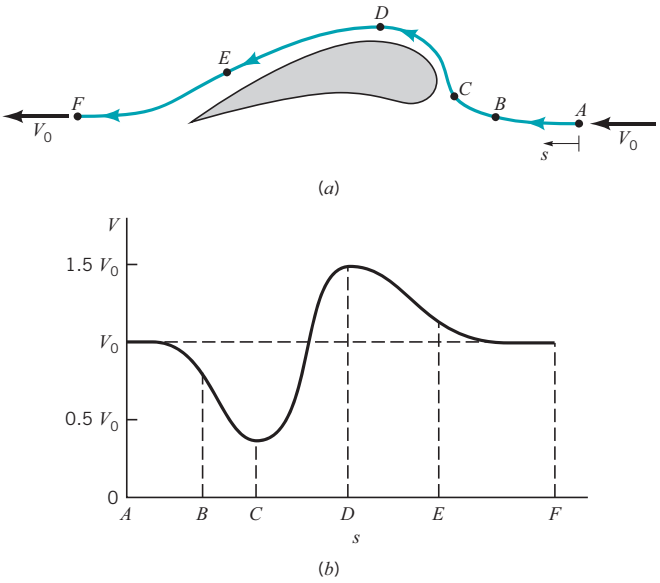



■ FIGURE P4.35

**4.36** A nozzle is designed to accelerate the fluid from $V_1$ to $V_2$ in a linear fashion. That is, $V = ax + b$, where $a$ and $b$ are constants. If the flow is constant with $V_1 = 10$ m/s at $x_1 = 0$ and $V_2 = 25$ m/s at $x_2 = 1$ m, determine the local acceleration, the convective acceleration, and the acceleration of the fluid at points (1) and (2).

**4.37** Repeat Problem 4.36 with the assumption that the flow is not steady, but at the time when $V_1 = 10$ m/s and $V_2 = 25$ m/s, it is known that $\partial V_1/\partial t = 20$ m/s$^2$ and $\partial V_2/\partial t = 60$ m/s$^2$.

**4.38** An incompressible fluid flows past a turbine blade as shown in Fig. P4.38*a* and **Video V4.9**. Far upstream and downstream of the blade the velocity is $V_0$. Measurements show that the velocity of the fluid along streamline $A$–$F$ near the blade is as indicated in Fig. P4.38*b*. Sketch the streamwise component of acceleration, $a_s$, as a function of distance, $s$, along the streamline. Discuss the important characteristics of your result.




■ FIGURE P4.38

***4.39** Air flows steadily through a variable area pipe with a velocity of $\mathbf{V} = u(x)\hat{\mathbf{i}}$ ft/s, where the approximate measured values of $u(x)$ are given in the table. Plot the acceleration as a function of $x$ for $0 \le x \le 12$ in. Plot the acceleration if the flowrate is increased by a factor of $N$ (i.e., the values of $u$ are increased by a factor of $N$) for $N = 2, 4, 10$.

| $x$ (in.) | $u$ (ft/s) | $x$ (in.) | $u$ (ft/s) |
|---|---|---|---|
| 0 | 10.0 | 7 | 20.1 |
| 1 | 10.2 | 8 | 17.4 |
| 2 | 13.0 | 9 | 13.5 |
| 3 | 20.1 | 10 | 11.9 |
| 4 | 28.3 | 11 | 10.3 |
| 5 | 28.4 | 12 | 10.0 |
| 6 | 25.8 | 13 | 10.0 |

***4.40** As is indicated in Fig. P4.40, the speed of exhaust in a car's exhaust pipe varies in time and distance because of the periodic nature of the engine's operation and the damping effect with distance from the engine. Assume that the speed is given by $V = V_0[1 + ae^{-bx}\sin(\omega t)]$, where $V_0 = 8$ fps, $a = 0.05$, $b = 0.2$ ft$^{-1}$, and $\omega = 50$ rad/s. Calculate and plot the fluid acceleration at $x = 0$, 1, 2, 3, 4, and 5 ft for $0 \le t \le \pi/25$ s.

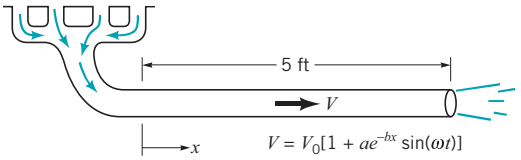



**FIGURE P4.40**

**4.41** Water flows over the crest of a dam with speed $V$ as shown in Fig. P4.41. Determine the speed if the magnitude of the normal acceleration at point (1) is to equal the acceleration of gravity, $g$.

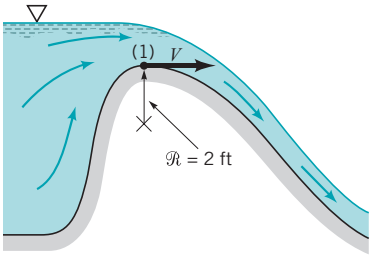



**FIGURE P4.41**

**4.42** Assume that the streamlines for the wingtip vortices from an airplane (see Fig. P4.19 and **Video V4.6**) can be approximated by circles of radius $r$ and that the speed is $V = K/r$, where $K$ is a constant. Determine the streamline acceleration, $a_s$, and the normal acceleration, $a_n$, for this flow.

**4.43** A fluid flows past a sphere with an upstream velocity of $V_0 = 40$ m/s as shown in Fig. P4.43. From a more advanced theory it is found that the speed of the fluid along the front part of the sphere is $V = \frac{3}{2}V_0 \sin\theta$. Determine the streamwise and normal components of acceleration at point $A$ if the radius of the sphere is $a = 0.20$ m.

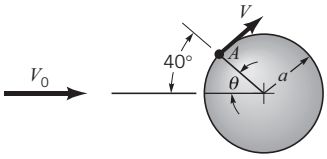



**FIGURE P4.43**

***4.44** For flow past a sphere as discussed in Problem 4.43, plot a graph of the streamwise acceleration, $a_s$, the normal acceleration, $a_n$, and the magnitude of the acceleration as a function of $\theta$ for $0 \le \theta \le 90°$ with $V_0 = 50$ ft/s and $a = 0.1$, 1.0, and 10 ft. Repeat for $V_0 = 5$ ft/s. At what point is the acceleration a maximum; a minimum?

***4.45** The velocity components for steady flow through the nozzle shown in Fig. P4.45 are $u = -V_0\, x/\ell$ and $v = V_0\,[1 + (y/\ell)]$, where $V_0$ and $\ell$ are constants. Determine the ratio of the magnitude of the acceleration at point (1) to that at point (2).

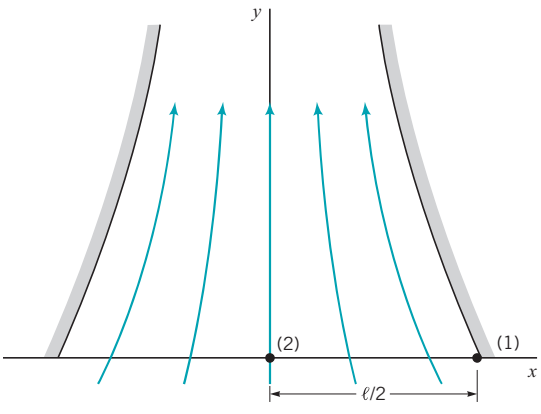



**FIGURE P4.45**

***4.46** A fluid flows past a circular cylinder of radius $a$ with an upstream speed of $V_0$ as shown in Fig. P4.46. A more advanced theory indicates that if viscous effects are negligible, the velocity of the fluid along the surface of the cylinder is given by $V = 2V_0 \sin\theta$. Determine the streamline and normal components of acceleration on the surface of the cylinder as a function of $V_0$, $a$, and $\theta$ and plot graphs of $a_s$ and $a_n$ for $0 \le \theta \le 90°$ with $V_0 = 10$ m/s and $a = 0.01$, 0.10, 1.0, and 10.0 m.

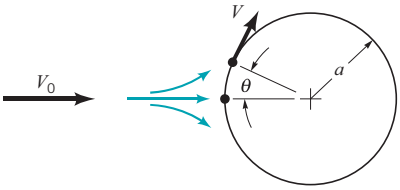



**FIGURE P4.46**

**4.47** Determine the $x$ and $y$ components of acceleration for the flow given in Problem 4.11. If $c > 0$, is the particle at point $x = x_0 > 0$ and $y = 0$ accelerating or decelerating? Explain. Repeat if $x_0 < 0$.

**4.48** When flood gates in a channel are opened, water flows along the channel downstream of the gates with an increasing speed given by $V = 4(1 + 0.1t)$ ft/s, for $0 \le t \le 20$ s, where $t$ is in seconds. For $t > 20$ s the speed is a constant $V = 12$ ft/s. Consider a location in the curved channel where the radius of curvature of the streamlines is 50 ft. For $t = 10$ s determine **(a)** the component of acceleration along the streamline, **(b)** the component of acceleration normal to the streamline, and **(c)** the net acceleration (magnitude and direction). Repeat for $t = 30$ s.

**4.49** Water flows steadily through the funnel shown in Fig. P4.49. Throughout most of the funnel the flow is approximately radial (along rays from $O$) with a velocity of $V = c/r^2$, where $r$ is the radial coordinate and $c$ is a constant. If the velocity is 0.4 m/s when $r = 0.1$ m, determine the acceleration at points $A$ and $B$.

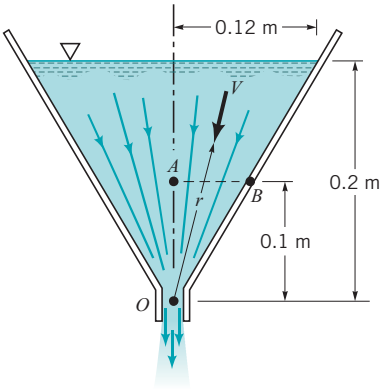
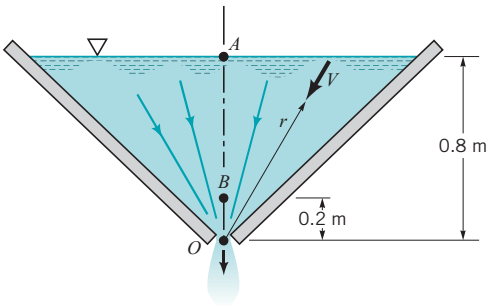



**FIGURE P4.49**

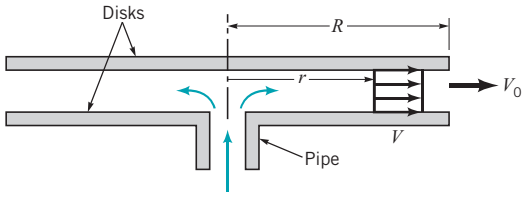
**4.50** Water flows though the slit at the bottom of a two-dimensional water trough as shown in Fig. P4.50. Throughout most of the trough the flow is approximately radial (along rays from $O$) with a velocity of $V = c/r$, where $r$ is the radial coordinate and $c$ is a constant. If the velocity is 0.04 m/s when $r = 0.1$ m, determine the acceleration at points $A$ and $B$.

CMS Cepcor EX 1046 Page 93
CMS Cepcor v. Sandvik IPR2024-01380



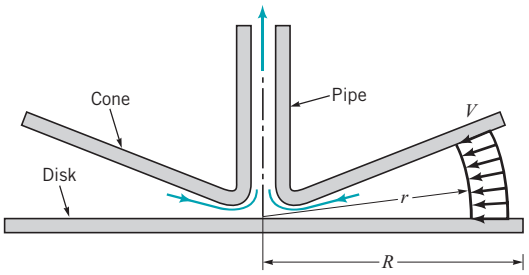

■ FIGURE P4.50

**4.51** Air flows from a pipe into the region between two parallel circular disks as shown in Fig. P4.51. The fluid velocity in the gap between the disks is closely approximated by $V = V_0R/r$, where $R$ is the radius of the disk, $r$ is the radial coordinate, and $V_0$ is the fluid velocity at the edge of the disk. Determine the acceleration for $r = 1$, 2, or 3 ft if $V_0 = 5$ ft/s and $R = 3$ ft.




■ FIGURE P4.51

**4.52** Air flows into a pipe from the region between a circular disk and a cone as shown in Fig. P4.52. The fluid velocity in the gap between the disk and the cone is closely approximated by $V = V_0R^2/r^2$, where $R$ is the radius of the disk, $r$ is the radial coordinate, and $V_0$ is the fluid velocity at the edge of the disk. Determine the acceleration for $r = 0.5$ and 2 ft if $V_0 = 5$ ft/s and $R = 2$ ft.




■ FIGURE P4.52

## Section 4.2.1 The Material Derivative

**4.53** Air flows steadily through a long pipe with a speed of $u = 50 + 0.5x$, where $x$ is the distance along the pipe in feet, and $u$ is in ft/s. Due to heat transfer into the pipe, the air temperature, $T$, within the pipe is $T = 300 + 10x$ °F. Determine the rate of change of the temperature of air particles as they flow past the section at $x = 5$ ft.

**4.54** A company produces a perishable product in a factory located at $x = 0$ and sells the product along the distribution route $x > 0$. The selling price of the product, $P$, is a function of the length of time after it was produced, $t$, and the location at which it is sold, $x$. That is, $P = P(x, t)$. At a given location the price of the product decreases in time (it is perishable) according to $\partial P/\partial t = -8$ dollars/hr. In addition, because of shipping costs the price increases with distance from the factory according to $\partial P/\partial x = 0.2$ dollars/mi. If the manufacturer wishes to sell the product for the same 100-dollar price anywhere along the distribution route, determine how fast he must travel along the route.

**4.55** Assume the temperature of the exhaust in an exhaust pipe can be approximated by $T = T_0(1 + ae^{-bx})\,[1 + c\cos(\omega t)]$, where $T_0 = 100$ °C, $a = 3$, $b = 0.03$ m$^{-1}$, $c = 0.05$, and $\omega = 100$ rad/s. If the exhaust speed is a constant 3 m/s, determine the time rate of change of temperature of the fluid particles at $x = 0$ and $x = 4$ m when $t = 0$.

**4.56** A bicyclist leaves from her home at 9 A.M. and rides to a beach 40 mi away. Because of a breeze off the ocean, the temperature at the beach remains 60 °F throughout the day. At the cyclist's home the temperature increases linearly with time, going from 60 °F at 9 A.M. to 80 °F by 1 P.M. The temperature is assumed to vary linearly as a function of position between the cyclist's home and the beach. Determine the rate of change of temperature observed by the cyclist for the following conditions: **(a)** as she pedals 10 mph through a town 10 mi from her home at 10 A.M.; **(b)** as she eats lunch at a rest stop 30 mi from her home at noon; **(c)** as she arrives enthusiastically at the beach at 1 P.M., pedaling 20 mph.

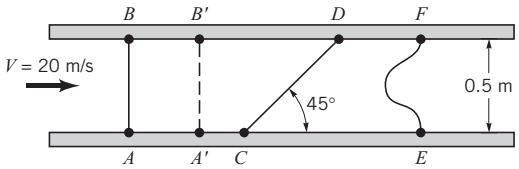
**4.57** The temperature distribution in a fluid is given by $T = 10x + 5y$, where $x$ and $y$ are the horizontal and vertical coordinates in meters and $T$ is in degrees centigrade. Determine the time rate of change of temperature of a fluid particle traveling **(a)** horizontally with $u = 20$ m/s, $v = 0$ or **(b)** vertically with $u = 0$, $v = 20$ m/s.

## Section 4.4 The Reynolds Transport Theorem

**4.58** Obtain a photograph/image of a situation in which a fluid is flowing. Print this photo and draw a control volume through which the fluid flows. Write a brief paragraph that describes how the fluid flows into and out of this control volume.

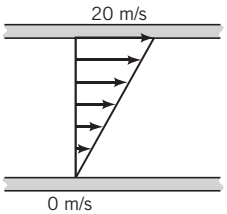
**4.59** The wind blows through the front door of a house with a speed of 2 m/s and exits with a speed of 1 m/s through two windows on the back of the house. Consider the system of interest for this flow to be the air within the house at time $t = 0$. Draw a simple sketch of the house and show an appropriate control volume for this flow. On the sketch, show the position of the system at time $t = 1$ s.

**4.60** Water flows through a duct of square cross section as shown in Fig. P4.60 with a constant, uniform velocity of $V = 20$ m/s. Consider fluid particles that lie along line $A$–$B$ at time $t = 0$. Determine the position of these particles, denoted by line $A'$–$B'$, when $t = 0.20$ s. Use the volume of fluid in the region between lines $A$–$B$ and $A'$–$B'$ to determine the flowrate in the duct. Repeat the problem for fluid particles originally along line $C$–$D$; along line $E$–$F$. Compare your three answers.




■ FIGURE P4.60

**4.61** Repeat Problem 4.60 if the velocity profile is linear from 0 to 20 m/s across the duct as shown in Fig. P4.61.




■ FIGURE P4.61

**4.62** In the region just downstream of a sluice gate, the water may develop a reverse flow region as is indicated in Fig. P4.62 and **Video V10.9**. The velocity profile is assumed to consist of two uniform regions, one with velocity $V_a = 10$ fps and the other with $V_b = 3$ fps. Determine the net flowrate of water across the portion of the control surface at section (2) if the channel is 20 ft wide.

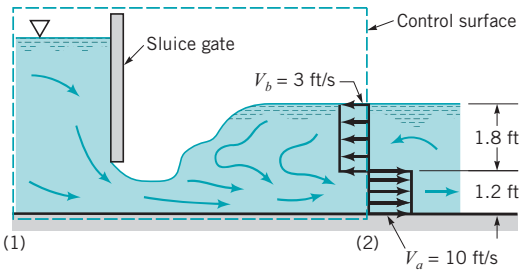



**FIGURE P4.62**

**4.63** At time $t = 0$ the valve on an initially empty (perfect vacuum, $\rho = 0$) tank is opened and air rushes in. If the tank has a volume of $\forall_0$ and the density of air within the tank increases as $\rho = \rho_\infty(1 - e^{-bt})$, where $b$ is a constant, determine the time rate of change of mass within the tank.

**†4.64** From calculus, one obtains the following formula (Leibnitz rule) for the time derivative of an integral that contains time in both the integrand and the limits of the integration:

$$\frac{d}{dt}\int_{x_1(t)}^{x_2(t)} f(x,t)dx = \int_{x_1}^{x_2}\frac{\partial f}{\partial t}dx + f(x_2,t)\frac{dx_2}{dt} - f(x_1,t)\frac{dx_1}{dt}$$

Discuss how this formula is related to the time derivative of the total amount of a property in a system and to the Reynolds transport theorem.

**4.65** Water enters the bend of a river with the uniform velocity profile shown in Fig. P4.65. At the end of the bend there is a region of separation or reverse flow. The fixed control volume *ABCD* coincides with the system at time $t = 0$. Make a sketch to indicate **(a)** the system at time $t = 5$ s and **(b)** the fluid that has entered and exited the control volume in that time period.

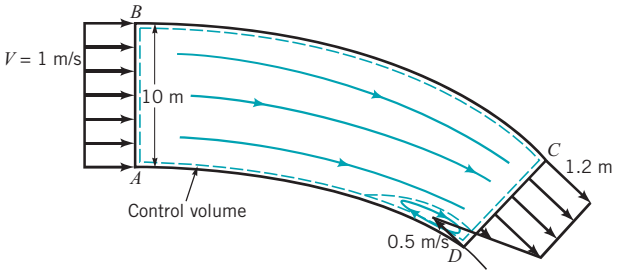



**FIGURE P4.65**

**4.66** A layer of oil flows down a vertical plate as shown in Fig. P4.66 with a velocity of $\mathbf{V} = (V_0/h^2)(2hx - x^2)\,\hat{\mathbf{j}}$ where $V_0$ and $h$ are constants. **(a)** Show that the fluid sticks to the plate and that the shear stress at the edge of the layer ($x = h$) is zero. **(b)** Determine the flowrate across surface *AB*. Assume the width of the plate is $b$. (*Note:* The velocity profile for laminar flow in a pipe has a similar shape. See **Video V6.13**.)

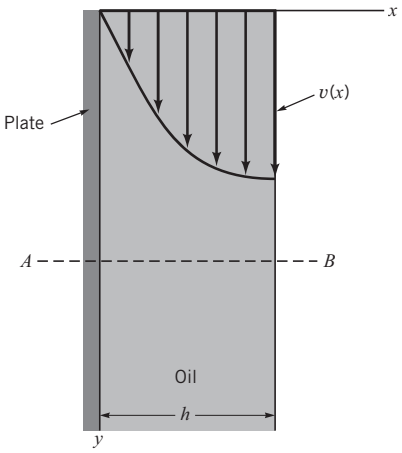



**FIGURE P4.66**

**4.67** Water flows in the branching pipe shown in Fig. P4.67 with uniform velocity at each inlet and outlet. The fixed control volume indicated coincides with the system at time $t = 20$ s. Make a sketch to indicate **(a)** the boundary of the system at time $t = 20.1$ s, **(b)** the fluid that left the control volume during that 0.1-s interval, and **(c)** the fluid that entered the control volume during that time interval.

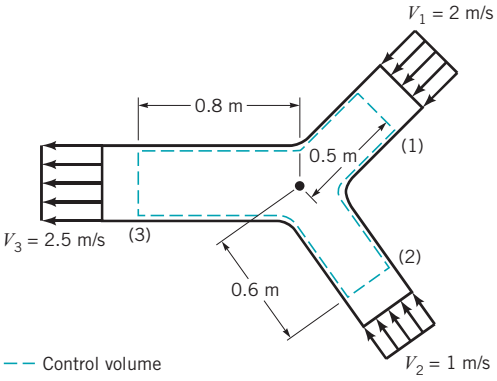



**FIGURE P4.67**

**4.68** Two plates are pulled in opposite directions with speeds of 1.0 ft/s as shown in Fig. P4.68. The oil between the plates moves with a velocity given by $\mathbf{V} = 10\,y\,\hat{\mathbf{i}}$ ft/s, where $y$ is in feet. The fixed control volume *ABCD* coincides with the system at time $t = 0$. Make a sketch to indicate **(a)** the system at time $t = 0.2$ s and **(b)** the fluid that has entered and exited the control volume in that time period.

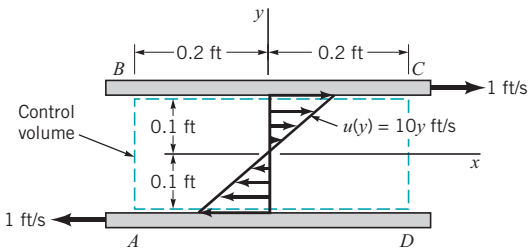
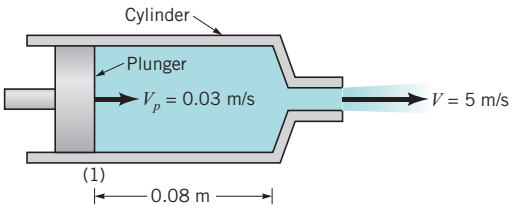



**FIGURE P4.68**

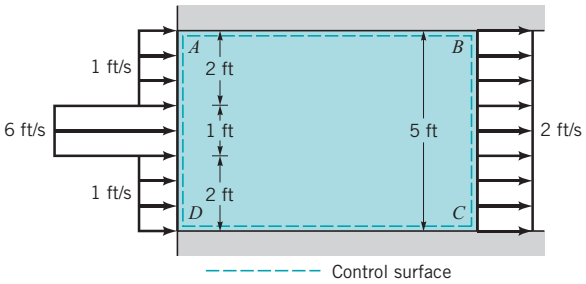
**4.69** Water is squirted from a syringe with a speed of $V = 5$ m/s by pushing in the plunger with a speed of $V_p = 0.03$ m/s as shown in Fig. P4.69. The surface of the deforming control volume consists of the sides and end of the cylinder and the end of the plunger. The system consists of the water in the syringe at $t = 0$ when the plunger is at section (1) as shown. Make a sketch to indicate the control surface and the system when $t = 0.5$ s.

CMS Cepcor EX 1046 Page 95
CMS Cepcor v. Sandvik IPR2024-01380



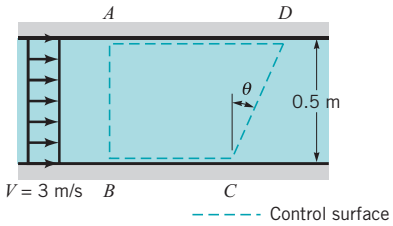

■ FIGURE P4.69

**4.70** Water enters a 5-ft-wide, 1-ft-deep channel as shown in Fig. P4.70. Across the inlet the water velocity is 6 ft/s in the center portion of the channel and 1 ft/s in the remainder of it. Farther downstream the water flows at a uniform 2 ft/s velocity across the entire channel. The fixed control volume *ABCD* coincides with the system at time $t = 0$. Make a sketch to indicate **(a)** the system at time $t = 0.5$ s and **(b)** the fluid that has entered and exited the control volume in that time period.



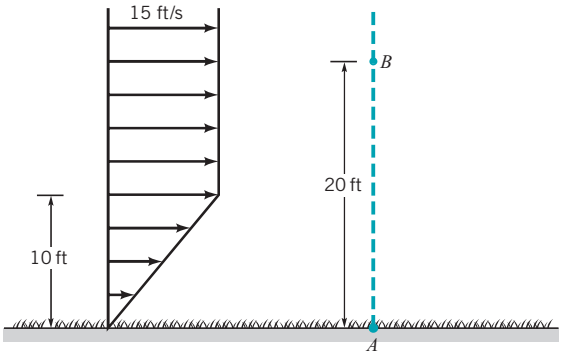

■ FIGURE P4.70

**4.71** Water flows through the 2-m-wide rectangular channel shown in Fig. P4.71 with a uniform velocity of 3 m/s. **(a)** Directly integrate Eq. 4.16 with $b = 1$ to determine the mass flowrate (kg/s) across section *CD* of the control volume. **(b)** Repeat part (a) with $b = 1/\rho$, where $\rho$ is the density. Explain the physical interpretation of the answer to part **(b)**.




■ FIGURE P4.71

**4.72** The wind blows across a field with an approximate velocity profile as shown in Fig. P4.72. Use Eq. 4.16 with the parameter *b* equal to the velocity to determine the momentum flowrate across the vertical surface *A*–*B*, which is of unit depth into the paper.




■ FIGURE P4.72

## ■ Life Long Learning Problems

**4.73** Even for the simplest flows it is often not be easy to visually represent various flow field quantities such as velocity, pressure, or temperature. For more complex flows, such as those involving three-dimensional or unsteady effects, it is extremely difficult to "show the data." However, with the use of computers and appropriate software, novel methods are being devised to more effectively illustrate the structure of a given flow. Obtain information about methods used to present complex flow data. Summarize your findings in a brief report.

**4.74** For centuries people have obtained qualitative and quantitative information about various flow fields by observing the motion of objects or particles in a flow. For example, the speed of the current in a river can be approximated by timing how long it takes a stick to travel a certain distance. The swirling motion of a tornado can be observed by following debris moving within the tornado funnel. Recently various high-tech methods using lasers and minute particles seeded within the flow have been developed to measure velocity fields. Such techniques include the laser doppler anemometer (LDA), the particle image velocimeter (PIV), and others. Obtain information about new laser-based techniques for measuring velocity fields. Summarize your findings in a brief report.

## ■ FE Exam Problems

Sample FE (Fundamentals of Engineering) exam questions for fluid mechanics are provided on the book's web site, www.wiley.com/college/munson.

CMS Cepcor EX 1046 Page 96
CMS Cepcor v. Sandvik IPR2024-01380



# 5 Finite Control Volume Analysis

CHAPTER OPENING PHOTO: Wind turbine farms (this is the Middelgrunden Offshore Wind Farm in Denmark) are becoming more common. Finite control volume analysis can be used to estimate the amount of energy transferred between the moving air and each turbine rotor. *(Photograph courtesy of Siemens Wind Power.)*

## Learning Objectives

After completing this chapter, you should be able to:

- select an appropriate finite control volume to solve a fluid mechanics problem.
- apply conservation of mass and energy and Newton's second law of motion to the contents of a finite control volume to get important answers.
- know how velocity changes and energy transfers in fluid flows are related to forces and torques.
- understand why designing for minimum loss of energy in fluid flows is so important.

*Many fluid mechanics problems can be solved by using control volume analysis.*

To solve many practical problems in fluid mechanics, questions about the behavior of the contents of a finite region in space (a finite control volume) are answered. For example, we may be asked to estimate the maximum anchoring force required to hold a turbojet engine stationary during a test. Or we may be called on to design a propeller to move a boat both forward and backward. Or we may need to determine how much power it would take to move natural gas from one location to another many miles away.

The bases of finite control volume analysis are some fundamental laws of physics, namely, conservation of mass, Newton's second law of motion, and the first and second laws of thermodynamics. While some simplifying approximations are made for practicality, the engineering answers possible with the estimates of this powerful analysis method have proven valuable in numerous instances.

Conservation of mass is the key to tracking flowing fluid. How much enters and leaves a control volume can be ascertained.



CMS Cepcor EX 1046 Page 97
CMS Cepcor v. Sandvik IPR2024-01380

Newton's second law of motion leads to the conclusion that forces can result from or cause changes in a flowing fluid's velocity magnitude and/or direction. Moment of force (torque) can result from or cause changes in a flowing fluid's moment of velocity. These forces and torques can be associated with work and power transfer.

The first law of thermodynamics is a statement of conservation of energy. The second law of thermodynamics identifies the loss of energy associated with every actual process. The mechanical energy equation based on these two laws can be used to analyze a large variety of steady, incompressible flows in terms of changes in pressure, elevation, speed, and of shaft work and loss.

Good judgment is required in defining the finite region in space, the control volume, used in solving a problem. What exactly to leave out of and what to leave in the control volume are important considerations. The formulas resulting from applying the fundamental laws to the contents of the control volume are easy to interpret physically and are not difficult to derive and use.

Because a finite region of space, a control volume, contains many fluid particles and even more molecules that make up each particle, the fluid properties and characteristics are often average values. In Chapter 6 an analysis of fluid flow based on what is happening to the contents of an infinitesimally small region of space or control volume through which numerous molecules simultaneously flow (what we might call a point in space) is considered.

## 5.1 Conservation of Mass—The Continuity Equation

### 5.1.1 Derivation of the Continuity Equation

A system is defined as a collection of unchanging contents, so the ***conservation of mass*** principle for a system is simply stated as

time rate of change of the system mass = 0

*The amount of mass in a system is constant.*

or

$$\frac{DM_{sys}}{Dt} = 0 \tag{5.1}$$

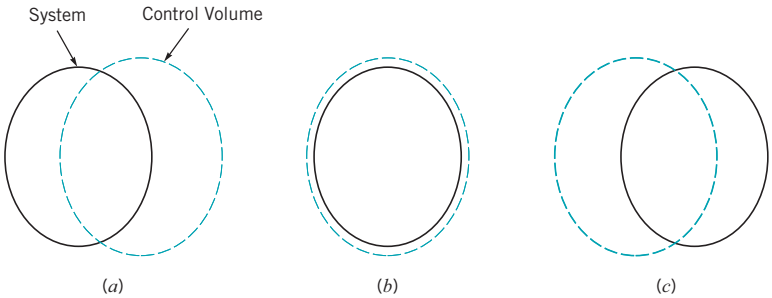
where the system mass, $M_{sys}$, is more generally expressed as

$$M_{sys} = \int_{sys} \rho \, d\forall \tag{5.2}$$

and the integration is over the volume of the system. In words, Eq. 5.2 states that the system mass is equal to the sum of all the density-volume element products for the contents of the system.

For a system and a fixed, nondeforming control volume that are coincident at an instant of time, as illustrated in Fig. 5.1, the Reynolds transport theorem (Eq. 4.19) with $B$ = mass and $b = 1$ allows us to state that

$$\frac{D}{Dt}\int_{sys} \rho \, d\forall = \frac{\partial}{\partial t}\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA \tag{5.3}$$




■ **FIGURE 5.1** **System and control volume at three different instances of time. (*a*) System and control volume at time $t - \delta t$. (*b*) System and control volume at time $t$, coincident condition. (*c*) System and control volume at time $t + \delta t$.**

or

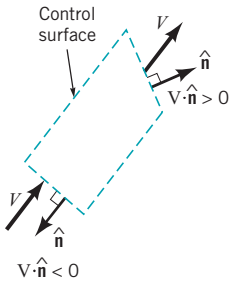
$$\begin{array}{l}\text{time rate of change}\\ \text{of the mass of the}\\ \text{coincident system}\end{array} = \begin{array}{l}\text{time rate of change}\\ \text{of the mass of the}\\ \text{contents of the coin-}\\ \text{cident control volume}\end{array} + \begin{array}{l}\text{net rate of flow}\\ \text{of mass through}\\ \text{the control}\\ \text{surface}\end{array}$$

In Eq. 5.3, we express the time rate of change of the system mass as the sum of two control volume quantities, the time rate of change of the mass of the contents of the control volume,

$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall$$

and the net rate of mass flow through the control surface,

$$\int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA$$




When a flow is steady, all field properties (i.e., properties at any specified point) including density remain constant with time and the time rate of change of the mass of the contents of the control volume is zero. That is,

$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall = 0$$

The integrand, $\mathbf{V} \cdot \hat{\mathbf{n}} \, dA$, in the mass flowrate integral represents the product of the component of velocity, $\mathbf{V}$, perpendicular to the small portion of control surface and the differential area, $dA$. Thus, $\mathbf{V} \cdot \hat{\mathbf{n}} \, dA$ is the volume flowrate through $dA$ and $\rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA$ is the mass flowrate through $dA$. Furthermore, as shown in the sketch in the margin, the sign of the dot product $\mathbf{V} \cdot \hat{\mathbf{n}}$ is "+" for flow *out* of the control volume and "−" for flow *into* the control volume since $\hat{\mathbf{n}}$ is considered positive when it points out of the control volume. When all of the differential quantities, $\rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA$, are summed over the entire control surface, as indicated by the integral

$$\int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA$$

the result is the net mass flowrate through the control surface, or

$$\int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \sum \dot{m}_{out} - \sum \dot{m}_{in} \qquad \textbf{(5.4)}$$

where $\dot{m}$ is the mass flowrate (lbm/s, slug/s or kg/s). If the integral in Eq. 5.4 is positive, the net flow is out of the control volume; if the integral is negative, the net flow is into the control volume.

*The continuity equation is a statement that mass is conserved.*

The control volume expression for conservation of mass, which is commonly called the ***continuity equation***, for a fixed, nondeforming control volume is obtained by combining Eqs. 5.1, 5.2, and 5.3 to obtain

$$\boxed{\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA = 0} \qquad \textbf{(5.5)}$$

In words, Eq. 5.5 states that to conserve mass the time rate of change of the mass of the contents of the control volume plus the net rate of mass flow through the control surface must equal zero. Actually, the same result could have been obtained more directly by equating the rates of mass flow into and out of the control volume to the rates of accumulation and depletion of mass within the control volume (see Section 3.6.2). It is reassuring, however, to see that the Reynolds transport theorem works for this simple-to-understand case. This confidence will serve us well as we develop control volume expressions for other important principles.

An often-used expression for ***mass flowrate***, $\dot{m}$, through a section of control surface having area $A$ is

$$\boxed{\dot{m} = \rho Q = \rho A V} \qquad \textbf{(5.6)}$$

CMS Cepcor EX 1046 Page 99
CMS Cepcor v. Sandvik IPR2024-01380

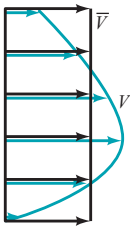
*Mass flowrate equals the product of density and volume flowrate.*

where $\rho$ is the fluid density, $Q$ is the volume flowrate ($ft^3/s$ or $m^3/s$), and $V$ is the component of fluid velocity perpendicular to area $A$. Since

$$\dot{m} = \int_A \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$$

application of Eq. 5.6 involves the use of *representative* or average values of fluid density, $\rho$, and fluid velocity, $V$. For incompressible flows, $\rho$ is uniformly distributed over area $A$. For compressible flows, we will normally consider a uniformly distributed fluid density at each section of flow and allow density changes to occur only from section to section. The appropriate fluid velocity to use in Eq. 5.6 is the average value of the component of velocity normal to the section area involved. This average value, $\overline{V}$, defined as

$$\overline{V} = \frac{\int_A \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA}{\rho A} \tag{5.7}$$




is shown in the figure in the margin.

If the velocity is considered uniformly distributed (one-dimensional flow) over the section area, $A$, then

$$\overline{V} = \frac{\int_A \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA}{\rho A} = V \tag{5.8}$$

and the bar notation is not necessary (as in Example 5.1). When the flow is not uniformly distributed over the flow cross-sectional area, the bar notation reminds us that an average velocity is being used (as in Examples 5.2 and 5.4).



*V5.1 Sink flow*



## 5.1.2 Fixed, Nondeforming Control Volume

In many applications of fluid mechanics, an appropriate control volume to use is fixed and nondeforming. Several example problems that involve the continuity equation for fixed, nondeforming control volumes (Eq. 5.5) follow.

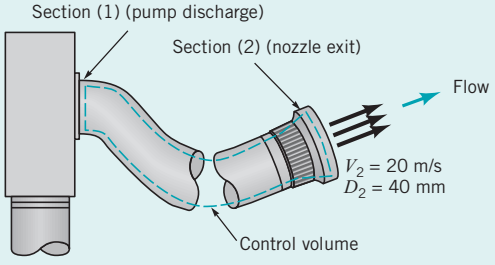
## EXAMPLE 5.1 Conservation of Mass—Steady, Incompressible Flow

**GIVEN** Water flows steadily through a nozzle at the end of a fire hose as illustrated in Fig. E5.1*a*. According to local regulations, the nozzle exit velocity must be at least 20 m/s as shown in Fig. E5.1*b*.

**FIND** Determine the minimum pumping capacity, $Q$, required in $m^3/s$.




■ FIGURE E5.1*a*




■ FIGURE E5.1*b*

## SOLUTION

The pumping capacity sought is the volume flowrate delivered by the fire pump to the hose and nozzle. Since we desire knowledge about the pump discharge flowrate and we have information about the nozzle exit flowrate, we link these two flowrates with the control volume designated with the dashed line in Fig. E5.1*b*. This control volume contains, at any instant, water that is within the hose and nozzle from the pump discharge to the nozzle exit plane.

Equation 5.5 is applied to the contents of this control volume to give

$$\cancel{\frac{\partial}{\partial t}\int_{cv}\rho\, d\forall} + \int_{cs}\rho\mathbf{V}\cdot\hat{\mathbf{n}}\, dA = 0 \qquad \text{(1)}$$

(0 (flow is steady))

The time rate of change of the mass of the contents of this control volume is zero because the flow is steady. Because there is only one inflow [the pump discharge, section (1)] and one outflow [the nozzle exit, section (2)], Eq. (1) becomes

$$\rho_2 A_2 V_2 - \rho_1 A_1 V_1 = 0$$

so that with $\dot{m} = \rho A V$

$$\dot{m}_1 = \dot{m}_2 \qquad \text{(2)}$$

Because the mass flowrate is equal to the product of fluid density, $\rho$, and volume flowrate, $Q$ (see Eq. 5.6), we obtain from Eq. 2

$$\rho_2 Q_2 = \rho_1 Q_1 \qquad \text{(3)}$$

Liquid flow at low speeds, as in this example, may be considered incompressible. Therefore

$$\rho_2 = \rho_1 \qquad \text{(4)}$$

and from Eqs. 3 and 4

$$Q_2 = Q_1 \qquad \text{(5)}$$

The pumping capacity is equal to the volume flowrate at the nozzle exit. If, for simplicity, the velocity distribution at the nozzle exit plane, section (2), is considered uniform (one-dimensional), then from Eq. 5

$$Q_1 = Q_2 = V_2 A_2$$
$$= V_2 \frac{\pi}{4} D_2^2 = (20\ \text{m/s})\frac{\pi}{4}\left(\frac{40\ \text{mm}}{1000\ \text{mm/m}}\right)^2$$
$$= 0.0251\ \text{m}^3/\text{s} \qquad \textbf{(Ans)}$$

**COMMENT** By repeating the calculations for various values of the nozzle exit diameter, $D_2$, the results shown in Fig. E5.1*c* are obtained. The flowrate is proportional to the exit area, which varies as the diameter squared. Hence, if the diameter were doubled, the flowrate would increase by a factor of four, provided the exit velocity remained the same.

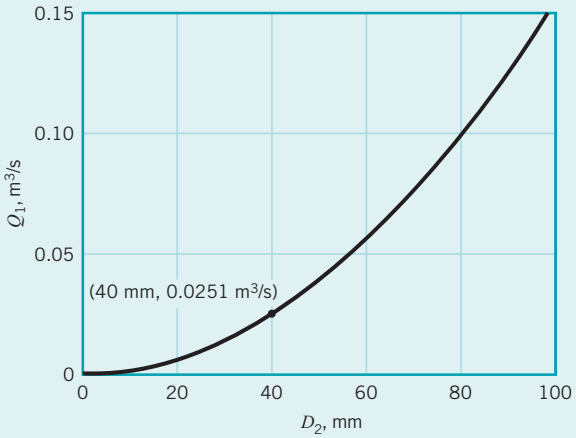



FIGURE E5.1*c*

# EXAMPLE 5.2 Conservation of Mass—Steady, Compressible Flow

**GIVEN** Air flows steadily between two sections in a long, straight portion of 4-in. inside diameter pipe as indicated in Fig. E5.2. The uniformly distributed temperature and pressure at each section are given. The average air velocity (nonuniform velocity distribution) at section (2) is 1000 ft/s.

**FIND** Calculate the average air velocity at section (1).

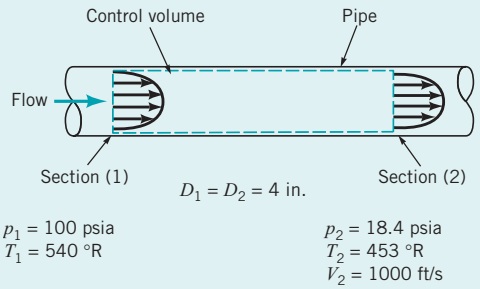



FIGURE E5.2

## SOLUTION

The average fluid velocity at any section is that velocity which yields the section mass flowrate when multiplied by the section average fluid density and section area (Eq. 5.7). We relate the flows at sections (1) and (2) with the control volume designated with a dashed line in Fig. E5.2.

Equation 5.5 is applied to the contents of this control volume to obtain

$$\cancel{\frac{\partial}{\partial t}\int_{cv}\rho\, d\forall} + \int_{cs}\rho\mathbf{V}\cdot\hat{\mathbf{n}}\, dA = 0$$

(0 (flow is steady))

The time rate of change of the mass of the contents of this control volume is zero because the flow is steady. The control surface

integral involves mass flowrates at sections (1) and (2) so that from Eq. 5.4 we get

$$\int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = \dot{m}_2 - \dot{m}_1 = 0$$

or

$$\dot{m}_1 = \dot{m}_2 \tag{1}$$

and from Eqs. 1, 5.6, and 5.7 we obtain

$$\rho_1 A_1 \overline{V}_1 = \rho_2 A_2 \overline{V}_2 \tag{2}$$

or since $A_1 = A_2$

$$\overline{V}_1 = \frac{\rho_2}{\rho_1}\overline{V}_2 \tag{3}$$

Air at the pressures and temperatures involved in this example problem behaves like an ideal gas. The ideal gas equation of state (Eq. 1.8) is

$$\rho = \frac{p}{RT} \tag{4}$$

Thus, combining Eqs. 3 and 4 we obtain

$$\overline{V}_1 = \frac{p_2 T_1 \overline{V}_2}{p_1 T_2}$$

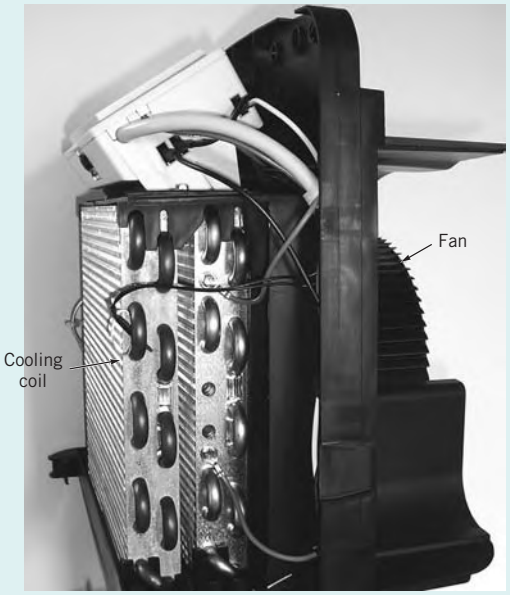
$$= \frac{(18.4 \text{ psia})(540 \text{ °R})(1000 \text{ ft/s})}{(100 \text{ psia})(453 \text{ °R})} = 219 \text{ ft/s} \quad \textbf{(Ans)}$$

**COMMENT** We learn from this example that the continuity equation (Eq. 5.5) is valid for compressible as well as incompressible flows. Also, nonuniform velocity distributions can be handled with the average velocity concept. Significant average velocity changes can occur in pipe flow if the fluid is compressible.

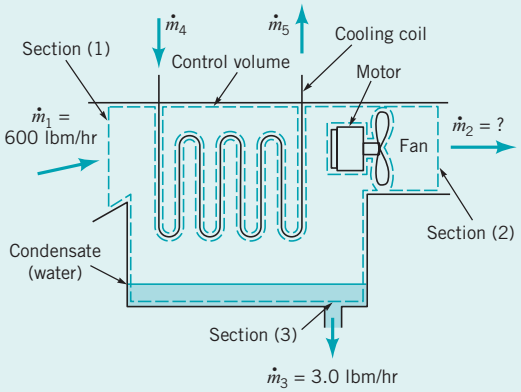
## EXAMPLE 5.3 Conservation of Mass—Two Fluids

**GIVEN** The inner workings of a dehumidifier are shown in Fig. E5.3*a*. Moist air (a mixture of dry air and water vapor) enters the dehumidifier at the rate of 600 lbm/hr. Liquid water drains out of the dehumidifier at a rate of 3.0 lbm/hr. A simplified sketch of the process is provided in Fig. E5.3*b*.

**FIND** Determine the mass flowrate of the dry air and the water vapor leaving the dehumidifier.




■ FIGURE E5.3*a*




■ FIGURE E5.3*b*

## SOLUTION

The unknown mass flowrate at section (2) is linked with the known flowrates at sections (1) and (3) with the control volume designated with a dashed line in Fig. E5.3*b*. The contents of the control volume are the air and water vapor mixture and the condensate (liquid water) in the dehumidifier at any instant.

Not included in the control volume are the fan and its motor, and the condenser coils and refrigerant. Even though the flow in the vicinity of the fan blade is unsteady, it is unsteady in a cyclical way. Thus, the flowrates at sections (1), (2), and (3) appear steady and the time rate of change of the mass of the contents of

the control volume may be considered equal to zero on a time-average basis. The application of Eqs. 5.4 and 5.5 to the control volume contents results in

$$\int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = -\dot{m}_1 + \dot{m}_2 + \dot{m}_3 = 0$$

or

$$\dot{m}_2 = \dot{m}_1 - \dot{m}_3 = 600 \text{ lbm/hr} - 3.0 \text{ lbm/hr} = 597 \text{ lbm/hr} \quad \textbf{(Ans)}$$

**COMMENT** Note that the continuity equation (Eq. 5.5) can be used when there is more than one stream of fluid flowing through the control volume.

The answer is the same with a control volume which includes the cooling coils to be within the control volume. The continuity equation becomes

$$\dot{m}_2 = \dot{m}_1 - \dot{m}_3 + \dot{m}_4 - \dot{m}_5 \qquad \textbf{(1)}$$

where $\dot{m}_4$ is the mass flowrate of the cooling fluid flowing into the control volume, and $\dot{m}_5$ is the flowrate out of the control volume through the cooling coil. Since the flow through the coils is steady, it follows that $\dot{m}_4 = \dot{m}_5$. Hence, Eq. 1 gives the same answer as obtained with the original control volume.

## EXAMPLE 5.4 Conservation of Mass—Nonuniform Velocity Profile

**GIVEN** Incompressible, laminar water flow develops in a straight pipe having radius $R$ as indicated in Fig. E5.4*a*. At section (1), the velocity profile is uniform; the velocity is equal to a constant value $U$ and is parallel to the pipe axis everywhere. At section (2), the velocity profile is axisymmetric and parabolic, with zero velocity at the pipe wall and a maximum value of $u_{max}$ at the centerline.

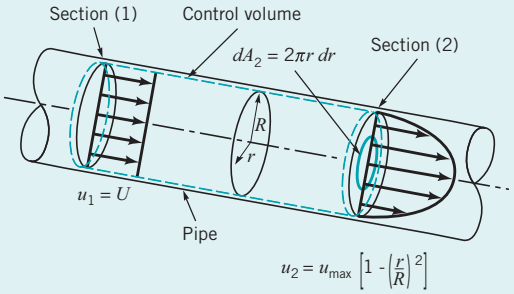



FIGURE E5.4*a*

**FIND**

**(a)** How are $U$ and $u_{max}$ related?

**(b)** How are the average velocity at section (2), $\bar{V}_2$, and $u_{max}$ related?

## SOLUTION

**(a)** An appropriate control volume is sketched (dashed lines) in Fig. E5.4*a*. The application of Eq. 5.5 to the contents of this control volume yields

$$\underbrace{\frac{\partial}{\partial t}\int_{cv} \rho\, d\forall}_{0 \text{ (flow is steady)}} + \int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = 0 \qquad \textbf{(1)}$$

At the inlet, section (1), the velocity is uniform with $V_1 = U$ so that

$$\int_{(1)} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = -\rho_1 A_1 U \qquad \textbf{(2)}$$

At the outlet, section (2), the velocity is not uniform. However, the net flowrate through this section is the sum of flows through numerous small washer-shaped areas of size $dA_2 = 2\pi r\, dr$ as shown by the shaded area element in Fig. E5.4*b*. On each of these infinitesimal areas the fluid velocity is denoted as $u_2$. Thus, in the limit of infinitesimal area elements, the summation is replaced by an integration and the outflow through section (2) is given by

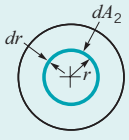



FIGURE E5.4*b*

$$\int_{(2)} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = \rho_2 \int_0^R u_2 2\pi r\, dr \qquad \textbf{(3)}$$

By combining Eqs. 1, 2, and 3 we get

$$\rho_2 \int_0^R u_2 2\pi r\, dr - \rho_1 A_1 U = 0 \qquad \textbf{(4)}$$

Since the flow is considered incompressible, $\rho_1 = \rho_2$. The parabolic velocity relationship for flow through section (2) is used in Eq. 4 to yield

$$2\pi u_{max} \int_0^R \left[1 - \left(\frac{r}{R}\right)^2\right] r\, dr - A_1 U = 0 \qquad \textbf{(5)}$$

Integrating, we get from Eq. 5

$$2\pi u_{max} \left(\frac{r^2}{2} - \frac{r^4}{4R^2}\right)_0^R - \pi R^2 U = 0$$

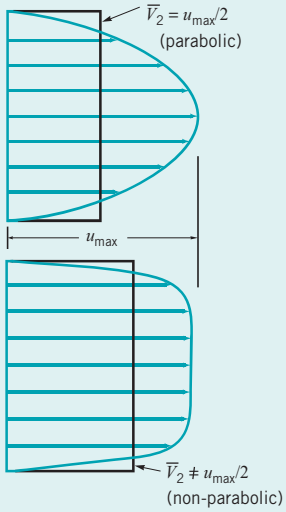
or

$$u_{max} = 2U \qquad \textbf{(Ans)}$$

**(b)** Since this flow is incompressible, we conclude from Eq. 5.7 that $U$ is the average velocity at all sections of the control volume. Thus, the average velocity at section (2), $\overline{V}_2$, is one-half the maximum velocity, $u_{max}$, there or

$$\overline{V}_2 = \frac{u_{max}}{2} \qquad \textbf{(Ans)}$$

**COMMENT** The relationship between the maximum velocity at section (2) and the average velocity is a function of the "shape" of the velocity profile. For the parabolic profile assumed in this example, the average velocity, $u_{max}/2$, is the actual "average" of the maximum velocity at section (2), $u_2 = u_{max}$, and the minimum velocity at that section, $u_2 = 0$. However, as shown in Fig. E5.4*c*, if the velocity profile is a different shape (non-parabolic), the average velocity is not necessarily one half of the maximum velocity.

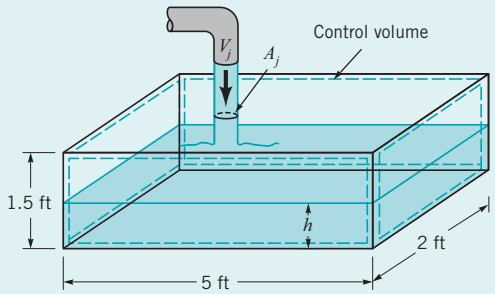



■ FIGURE E5.4*c*

## EXAMPLE 5.5 Conservation of Mass—Unsteady Flow

**GIVEN** A bathtub is being filled with water from a faucet. The rate of flow from the faucet is steady at 9 gal/min. The tub volume is approximated by a rectangular space as indicated in Fig. E5.5*a*.

**FIND** Estimate the time rate of change of the depth of water in the tub, $\partial h/\partial t$, in inches per minute at any instant.




■ FIGURE E5.5*a*

### SOLUTION

We use the fixed, nondeforming control volume outlined with a dashed line in Fig. E5.5*a*. This control volume includes in it, at any instant, the water accumulated in the tub, some of the water flowing from the faucet into the tub, and some air. Application of Eqs. 5.4 and 5.5 to these contents of the control volume results in

$$\frac{\partial}{\partial t}\int_{\substack{\text{air}\\\text{volume}}} \rho_{air}\, d\mathcal{V}_{air} + \frac{\partial}{\partial t}\int_{\substack{\text{water}\\\text{volume}}} \rho_{water}\, d\mathcal{V}_{water} - \dot{m}_{water} + \dot{m}_{air} = 0 \qquad \textbf{(1)}$$

Recall that the mass, $dm$, of fluid contained in a small volume $d\mathcal{V}$ is $dm = \rho\, d\mathcal{V}$. Hence, the two *integrals* in Eq. 1 represent the total amount of air and water in the control volume, and the sum of the first two *terms* is the time rate of change of mass within the control volume.

Note that the time rate of change of air mass and water mass are each not zero. Recognizing, however, that the air mass must be conserved, we know that the time rate of change of the mass of air in the control volume must be equal to the rate of air mass flow out of the control volume. For simplicity, we disregard any water evaporation that occurs. Thus, applying Eqs. 5.4 and 5.5 to the air only and to the water only, we obtain

$$\frac{\partial}{\partial t}\int_{\substack{\text{air}\\\text{volume}}} \rho_{air}\, d\mathcal{V}_{air} + \dot{m}_{air} = 0$$

for air, and

$$\frac{\partial}{\partial t}\int_{\substack{\text{water}\\\text{volume}}} \rho_{water}\, d\mathcal{V}_{water} = \dot{m}_{water} \qquad \textbf{(2)}$$

for water. The volume of water in the control volume is given by

$$\int_{\substack{\text{water}\\\text{volume}}} \rho_{water}\, d\mathcal{V}_{water} = \rho_{water}\,[h(2\text{ ft})(5\text{ ft}) + (1.5\text{ ft} - h)A_j] \qquad \textbf{(3)}$$

where $A_j$ is the cross-sectional area of the water flowing from the faucet into the tub. Combining Eqs. 2 and 3, we obtain

$$\rho_{water}\,(10\text{ ft}^2 - A_j)\frac{\partial h}{\partial t} = \dot{m}_{water}$$

and, thus, since $\dot{m} = \rho Q$,

$$\frac{\partial h}{\partial t} = \frac{Q_{water}}{(10\text{ ft}^2 - A_j)}$$

For $A_j \ll 10$ ft$^2$ we can conclude that

$$\frac{\partial h}{\partial t} = \frac{Q_{\text{water}}}{(10 \text{ ft}^2)}$$

or

$$\frac{\partial h}{\partial t} = \frac{(9 \text{ gal/min})(12 \text{ in./ft})}{(7.48 \text{ gal/ft}^3)(10 \text{ ft}^2)} = 1.44 \text{ in./min} \qquad \textbf{(Ans)}$$

**COMMENT** By repeating the calculations for the same flowrate but with various water jet diameters, $D_j$, the results shown in Fig. E5.5*b* are obtained. With the flowrate held constant, the value of $\partial h/\partial t$ is nearly independent of the jet diameter for values of the diameter less than about 10 in.

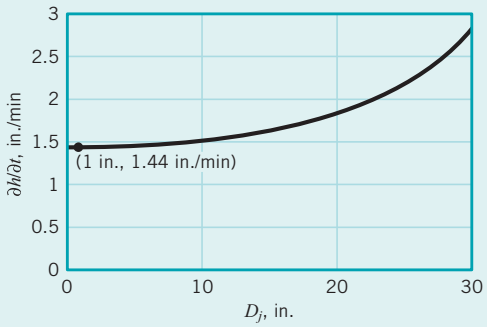



**FIGURE E5.5*b***

The preceding example problems illustrate some important results of applying the conservation of mass principle to the contents of a fixed, nondeforming control volume. The dot product $\mathbf{V} \cdot \hat{\mathbf{n}}$ is "+" for flow out of the control volume and "−" for flow into the control volume. Thus, mass flowrate out of the control volume is "+" and mass flowrate in is "−." When the flow is steady, the time rate of change of the mass of the contents of the control volume

*The appropriate sign convention must be followed.*

$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall$$

is zero and the net amount of mass flowrate, $\dot{m}$, through the control surface is therefore also zero

$$\sum \dot{m}_{\text{out}} - \sum \dot{m}_{\text{in}} = 0 \qquad \textbf{(5.9)}$$



*V5.2 Shop vac filter*



If the steady flow is also incompressible, the net amount of volume flowrate, $Q$, through the control surface is also zero:

$$\sum Q_{\text{out}} - \sum Q_{\text{in}} = 0 \qquad \textbf{(5.10)}$$

An unsteady, but cyclical flow can be considered steady on a time-average basis. When the flow is unsteady, the instantaneous time rate of change of the mass of the contents of the control volume is not necessarily zero and can be an important variable. When the value of

$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall$$

is "+," the mass of the contents of the control volume is increasing. When it is "−," the mass of the contents of the control volume is decreasing.

When the flow is uniformly distributed over the opening in the control surface (one-dimensional flow),

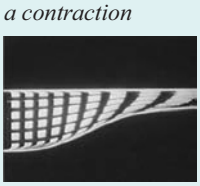
$$\dot{m} = \rho A V$$

where $V$ is the uniform value of the velocity component normal to the section area $A$. When the velocity is nonuniformly distributed over the opening in the control surface,

$$\dot{m} = \rho A \overline{V} \qquad \textbf{(5.11)}$$



*V5.3 Flow through a contraction*



where $\overline{V}$ is the average value of the component of velocity normal to the section area $A$ as defined by Eq. 5.7.

For steady flow involving only one stream of a specific fluid flowing through the control volume at sections (1) and (2),

$$\dot{m} = \rho_1 A_1 \overline{V}_1 = \rho_2 A_2 \overline{V}_2 \qquad \textbf{(5.12)}$$

and for incompressible flow,

$$Q = A_1 \overline{V}_1 = A_2 \overline{V}_2 \qquad \textbf{(5.13)}$$

CMS Cepcor EX 1046 Page 105
CMS Cepcor v. Sandvik IPR2024-01380

For steady flow involving more than one stream of a specific fluid or more than one specific fluid flowing through the control volume,

$$\sum \dot{m}_{in} = \sum \dot{m}_{out}$$

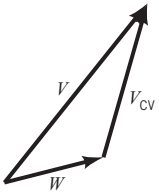
The variety of example problems solved above should give the correct impression that the fixed, nondeforming control volume is versatile and useful.

## Fluids in the News

**New 1.6 GPF standards** Toilets account for approximately 40% of all indoor household water use. To conserve water, the new standard is 1.6 gallons of water per flush (gpf). Old toilets use up to 7 gpf; those manufactured after 1980 use 3.5 gpf. Neither are considered low-flush toilets. A typical 3.2 person household in which each person flushes a 7-gpf toilet 4 times a day uses 32,700 gallons of water each year; with a 3.5-gpf toilet the amount is reduced to 16,400 gallons. Clearly the new 1.6-gpf toilets will save even more water. However, designing a toilet that flushes properly with such a small amount of water is not simple. Today there are two basic types involved: those that are gravity powered and those that are pressure powered. Gravity toilets (typical of most currently in use) have rather long cycle times. The water starts flowing under the action of gravity and the swirling vortex motion initiates the siphon action which builds to a point of discharge. In the newer pressure-assisted models, the *flowrate* is large but the cycle time is short and the amount of water used is relatively small. (See Problem 5.32.)

### 5.1.3 Moving, Nondeforming Control Volume

*Some problems are most easily solved by using a moving control volume.*




It is sometimes necessary to use a nondeforming control volume attached to a moving reference frame. Examples include control volumes containing a gas turbine engine on an aircraft in flight, the exhaust stack of a ship at sea, and the gasoline tank of an automobile passing by.

As discussed in Section 4.4.6, when a moving control volume is used, the fluid velocity relative to the moving control volume (relative velocity) is an important flow field variable. The relative velocity, **W**, is the fluid velocity seen by an observer moving with the control volume. The control volume velocity, $\mathbf{V}_{cv}$, is the velocity of the control volume as seen from a fixed coordinate system. The absolute velocity, **V**, is the fluid velocity seen by a stationary observer in a fixed coordinate system. These velocities are related to each other by the vector equation

$$\mathbf{V} = \mathbf{W} + \mathbf{V}_{cv} \tag{5.14}$$

as illustrated by the figure in the margin. This is the same as Eq. 4.22, introduced earlier.

For a system and a moving, nondeforming control volume that are coincident at an instant of time, the Reynolds transport theorem (Eq. 4.23) for a moving control volume leads to

$$\frac{DM_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{W} \cdot \hat{\mathbf{n}} \, dA \tag{5.15}$$

From Eqs. 5.1 and 5.15, we can get the control volume expression for conservation of mass (the continuity equation) for a moving, nondeforming control volume, namely,

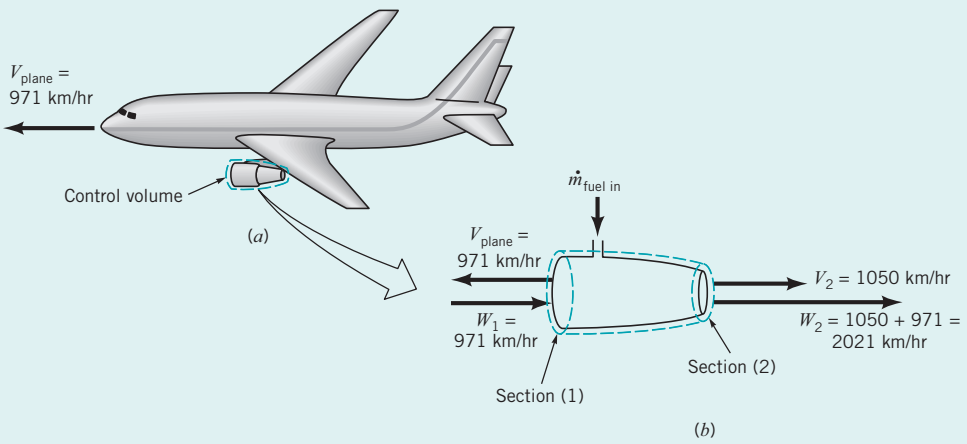
$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{W} \cdot \hat{\mathbf{n}} \, dA = 0 \tag{5.16}$$

Some examples of the application of Eq. 5.16 follow.

## EXAMPLE 5.6 Conservation of Mass—Compressible Flow with a Moving Control Volume

**GIVEN** An airplane moves forward at a speed of 971 km/hr as shown in Fig. E5.6a. The frontal intake area of the jet engine is 0.80 $m^2$ and the entering air density is 0.736 $kg/m^3$. A stationary observer determines that relative to the earth, the jet engine exhaust gases move away from the engine with a speed of 1050 km/hr. The engine exhaust area is 0.558 $m^2$, and the exhaust gas density is 0.515 $kg/m^3$.

**FIND** Estimate the mass flowrate of fuel into the engine in kg/hr.




■ FIGURE E5.6

## SOLUTION

The control volume, which moves with the airplane (see Fig. E5.6b), surrounds the engine and its contents and includes all fluids involved at an instant. The application of Eq. 5.16 to these contents of the control volume yields

0 (flow relative to moving control volume is considered steady on a time-average basis)

$$\frac{\partial}{\partial t}\int_{cv}\rho\, d\forall + \int_{cs}\rho \mathbf{W}\cdot\hat{\mathbf{n}}\, dA = 0 \qquad (1)$$

Assuming one-dimensional flow, we evaluate the surface integral in Eq. 1 and get

$$-\dot{m}_{\substack{fuel\\in}} - \rho_1 A_1 W_1 + \rho_2 A_2 W_2 = 0$$

or

$$\dot{m}_{\substack{fuel\\in}} = \rho_2 A_2 W_2 - \rho_1 A_1 W_1 \qquad (2)$$

We consider the intake velocity, $W_1$, relative to the moving control volume, as being equal in magnitude to the speed of the airplane, 971 km/hr. The exhaust velocity, $W_2$, also needs to be measured relative to the moving control volume. Since a fixed observer noted that the exhaust gases were moving away from the engine at a speed of 1050 km/hr, the speed of the exhaust gases relative to the moving control volume, $W_2$, is determined as follows by using Eq. 5.14

$$V_2 = W_2 + V_{plane}$$

or

$$W_2 = V_2 - V_{plane} = 1050\ \text{km/hr} - (-971\ \text{km/hr}) = 2021\ \text{km/hr}$$

and is shown in Fig. E5.6*b*.

From Eq. 2,

$$\dot{m}_{\substack{fuel\\in}} = (0.515\ \text{kg/m}^3)(0.558\ \text{m}^2)(2021\ \text{km/hr})(1000\ \text{m/km}) - (0.736\ \text{kg/m}^3)(0.80\ \text{m}^2)(971\ \text{km/hr})(1000\ \text{m/km})$$
$$= (580{,}800 - 571{,}700)\ \text{kg/hr}$$
$$\dot{m}_{\substack{fuel\\in}} = 9100\ \text{kg/hr} \qquad \textbf{(Ans)}$$

**COMMENT** Note that the fuel flowrate was obtained as the difference of two large, nearly equal numbers. Precise values of $W_2$ and $W_1$ are needed to obtain a modestly accurate value of $\dot{m}_{\substack{fuel\\in}}$.

# EXAMPLE 5.7 Conservation of Mass—Relative Velocity

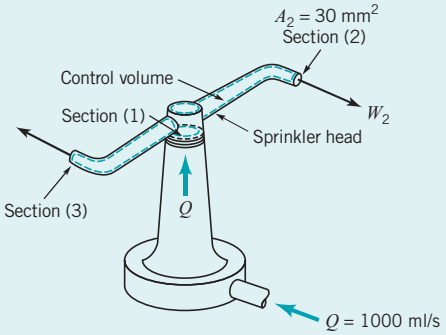
**GIVEN** Water enters a rotating lawn sprinkler through its base at the steady rate of 1000 ml/s as sketched in Fig. E5.7. The exit area of each of the two nozzles is 30 mm$^2$.

**FIND** Determine the average speed of the water leaving the nozzle, relative to the nozzle, if

**(a)** the rotary sprinkler head is stationary,

**(b)** the sprinkler head rotates at 600 rpm, and

**(c)** the sprinkler head accelerates from 0 to 600 rpm.




■ FIGURE E5.7

## SOLUTION

**(a)** We specify a control volume that contains the water in the rotary sprinkler head at any instant. This control volume is nondeforming, but it moves (rotates) with the sprinkler head.

The application of Eq. 5.16 to the contents of this control volume for situation **(a)**, **(b)**, or **(c)** of the problem results in the same expression, namely

$$\cancel{\frac{\partial}{\partial t}}\int_{cv}\rho\, d\forall + \int_{cs}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA = 0$$

(0: flow is steady or the control volume is filled with an incompressible fluid)

or

$$\Sigma\rho_{out} A_{out} W_{out} - \Sigma\rho_{in} A_{in} W_{in} = 0 \qquad \textbf{(1)}$$

The time rate of change of the mass of water in the control volume is zero because the flow is steady and the control volume is filled with water.

Because there is only one inflow [at the base of the rotating arm, section (1)] and two outflows [the two nozzles at the tips of the arm, sections (2) and (3), each have the same area and fluid velocity], Eq. 1 becomes

$$\rho_2 A_2 W_2 + \rho_3 A_3 W_3 - \rho_1 A_1 W_1 = 0 \qquad \textbf{(2)}$$

Hence, for incompressible flow with $\rho_1 = \rho_2 = \rho_3$, Eq. 2 becomes

$$A_2 W_2 + A_3 W_3 - A_1 W_1 = 0$$

With $Q = A_1 W_1$, $A_2 = A_3$, and $W_2 = W_3$ it follows that

$$W_2 = \frac{Q}{2A_2}$$

or

$$W_2 = \frac{(1000\text{ ml/s})(0.001\text{ m}^3/\text{liter})(10^6\text{ mm}^2/\text{m}^2)}{(1000\text{ ml/liter})(2)(30\text{ mm}^2)} = 16.7\text{ m/s} \qquad \textbf{(Ans)}$$

**(b), (c)** The value of $W_2$ is independent of the speed of rotation of the sprinkler head and represents the average velocity of the water exiting from each nozzle with respect to the nozzle for cases **(a)**, **(b)**, and **(c)**.

**COMMENT** The velocity of water discharging from each nozzle, when viewed from a stationary reference (i.e., $V_2$), will vary as the rotation speed of the sprinkler head varies since from Eq. 5.14,

$$V_2 = W_2 - U$$

where $U = \omega R$ is the speed of the nozzle and $\omega$ and $R$ are the angular velocity and radius of the sprinkler head, respectively.

When a moving, nondeforming control volume is used, the dot product sign convention used earlier for fixed, nondeforming control volume applications is still valid. Also, if the flow within the moving control volume is steady, or steady on a time-average basis, the time rate of change of the mass of the contents of the control volume is zero. Velocities seen from the control volume reference frame (relative velocities) must be used in the continuity equation. Relative and absolute velocities are related by a vector equation (Eq. 5.14), which also involves the control volume velocity.

### 5.1.4 Deforming Control Volume

*Care is needed to ensure that absolute and relative velocities are used correctly.*

Occasionally, a deforming control volume can simplify the solution of a problem. A deforming control volume involves changing volume size and control surface movement. Thus, the Reynolds transport theorem for a moving control volume can be used for this case, and Eqs. 4.23 and 5.1 lead to

$$\boxed{\frac{DM_{sys}}{Dt} = \frac{\partial}{\partial t}\int_{cv}\rho\, d\forall + \int_{cs}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA = 0} \qquad \textbf{(5.17)}$$

The time rate of change term in Eq. 5.17,

$$\frac{\partial}{\partial t}\int_{cv}\rho\, d\forall$$

is usually nonzero and must be carefully evaluated because the extent of the control volume varies with time. The mass flowrate term in Eq. 5.17,

$$\int_{cs}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA$$

*The velocity of the surface of a deforming control volume is not the same at all points on the surface.*

must be determined with the relative velocity, **W**, the velocity referenced to the control surface. Since the control volume is deforming, the control surface velocity is not necessarily uniform and identical to the control volume velocity, $\mathbf{V}_{cv}$, as was true for moving, nondeforming control volumes. For the deforming control volume,

$$\mathbf{V} = \mathbf{W} + \mathbf{V}_{cs} \tag{5.18}$$

where $\mathbf{V}_{cs}$ is the velocity of the control surface as seen by a fixed observer. The relative velocity, **W**, must be ascertained with care wherever fluid crosses the control surface. Two example problems that illustrate the use of the continuity equation for a deforming control volume, Eq. 5.17, follow.

## EXAMPLE 5.8 Conservation of Mass—Deforming Control Volume

**GIVEN** A syringe (Fig. E5.8) is used to inoculate a cow. The plunger has a face area of 500 mm$^2$. The liquid in the syringe is to be injected steadily at a rate of 300 cm$^3$/min. The leakage rate past the plunger is 0.10 times the volume flowrate out of the needle.

**FIND** With what speed should the plunger be advanced?

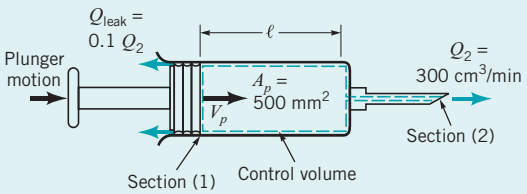



**FIGURE E5.8**

### SOLUTION

The control volume selected for solving this problem is the deforming one illustrated in Fig. E5.8. Section (1) of the control surface moves with the plunger. The surface area of section (1), $A_1$, is considered equal to the circular area of the face of the plunger, $A_p$, although this is not strictly true, since leakage occurs. The difference is small, however. Thus,

$$A_1 = A_p \tag{1}$$

Liquid also leaves the needle through section (2), which involves fixed area $A_2$. The application of Eq. 5.17 to the contents of this control volume gives

$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall + \dot{m}_2 + \rho Q_{leak} = 0 \tag{2}$$

Even though $Q_{leak}$ and the flow through section area $A_2$ are steady, the time rate of change of the mass of liquid in the shrinking control volume is not zero because the control volume is getting smaller. To evaluate the first term of Eq. 2, we note that

$$\int_{cv} \rho \, d\forall = \rho(\ell A_1 + \forall_{needle}) \tag{3}$$

where $\ell$ is the changing length of the control volume (see Fig. E5.8) and $\forall_{needle}$ is the volume of the needle. From Eq. 3, we obtain

$$\frac{\partial}{\partial t}\int_{cv} \rho \, d\forall = \rho A_1 \frac{\partial \ell}{\partial t} \tag{4}$$

Note that

$$-\frac{\partial \ell}{\partial t} = V_p \tag{5}$$

where $V_p$ is the speed of the plunger sought in the problem statement. Combining Eqs. 2, 4, and 5 we obtain

$$-\rho A_1 V_p + \dot{m}_2 + \rho Q_{leak} = 0 \tag{6}$$

However, from Eq. 5.6, we see that

$$\dot{m}_2 = \rho Q_2 \tag{7}$$

and Eq. 6 becomes

$$-\rho A_1 V_p + \rho Q_2 + \rho Q_{leak} = 0 \tag{8}$$

Solving Eq. 8 for $V_p$ yields

$$V_p = \frac{Q_2 + Q_{leak}}{A_1} \tag{9}$$

Since $Q_{leak} = 0.1 Q_2$, Eq. 9 becomes

$$V_p = \frac{Q_2 + 0.1Q_2}{A_1} = \frac{1.1Q_2}{A_1}$$

and

$$V_p = \frac{(1.1)(300 \text{ cm}^3/\text{min})}{(500 \text{ mm}^2)}\left(\frac{1000 \text{ mm}^3}{\text{cm}^3}\right)$$
$$= 660 \text{ mm/min} \quad \textbf{(Ans)}$$

## EXAMPLE 5.9 Conservation of Mass—Deforming Control Volume

**GIVEN** Consider Example 5.5.

**FIND** Solve the problem of Example 5.5 using a deforming control volume that includes only the water accumulating in the bathtub.

## SOLUTION

For this deforming control volume, Eq. 5.17 leads to

$$\frac{\partial}{\partial t}\int_{\substack{\text{water}\\\text{volume}}} \rho\, d\forall + \int_{cs} \rho \mathbf{W}\cdot\hat{\mathbf{n}}\, dA = 0 \qquad (1)$$

The first term of Eq. 1 can be evaluated as

$$\frac{\partial}{\partial t}\int_{\substack{\text{water}\\\text{volume}}} \rho\, d\forall = \frac{\partial}{\partial t}[\rho h(2\text{ ft})(5\text{ ft})]$$

$$= \rho\,(10\text{ ft}^2)\frac{\partial h}{\partial t} \qquad (2)$$

The second term of Eq. 1 can be evaluated as

$$\int_{cs} \rho\,\mathbf{W}\cdot\hat{\mathbf{n}}\, dA = -\rho\left(V_j + \frac{\partial h}{\partial t}\right)A_j \qquad (3)$$

where $A_j$ and $V_j$ are the cross-sectional area and velocity of the water flowing from the faucet into the tube. Thus, from Eqs. 1, 2, and 3 we obtain

$$\frac{\partial h}{\partial t} = \frac{V_j A_j}{(10\text{ ft}^2 - A_j)} = \frac{Q_{\text{water}}}{(10\text{ ft}^2 - A_j)}$$

or for $A_j \ll 10\text{ ft}^2$

$$\frac{\partial h}{\partial t} = \frac{9(\text{gal/min})(12\text{ in./ft})}{(7.48\text{ gal/ft}^3)(10\text{ ft}^2)} = 1.44\text{ in./min} \qquad \textbf{(Ans)}$$

**COMMENT** Note that these results using a deforming control volume are the same as that obtained in Example 5.5 with a fixed control volume.

The conservation of mass principle is easily applied to the contents of a control volume. The appropriate selection of a specific kind of control volume (for example, fixed and nondeforming, moving and nondeforming, or deforming) can make the solution of a particular problem less complicated. In general, where fluid flows through the control surface, it is advisable to make the control surface perpendicular to the flow. In the sections ahead we learn that the conservation of mass principle is primarily used in combination with other important laws to solve problems.

## 5.2 Newton's Second Law—The Linear Momentum and Moment-of-Momentum Equations



*V5.4 Smokestack plume momentum*



### 5.2.1 Derivation of the Linear Momentum Equation

Newton's second law of motion for a system is

time rate of change of the linear momentum of the system = sum of external forces acting on the system

Since momentum is mass times velocity, the momentum of a small particle of mass $\rho d\forall$ is $\mathbf{V}\rho d\forall$. Thus, the momentum of the entire system is $\int_{sys}\mathbf{V}\rho d\forall$ and Newton's law becomes

$$\frac{D}{Dt}\int_{sys}\mathbf{V}\rho\, d\forall = \sum \mathbf{F}_{sys} \qquad (5.19)$$

Any reference or coordinate system for which this statement is true is called *inertial*. A fixed coordinate system is inertial. A coordinate system that moves in a straight line with constant velocity and is thus without acceleration is also inertial. We proceed to develop the control volume formula for this important law. When a control volume is coincident with a system at an instant of time, the forces acting on the system and the forces acting on the contents of the coincident control volume (see Fig. 5.2) are instantaneously identical, that is,

*Forces acting on a flowing fluid can change its velocity magnitude and/or direction.*

$$\sum \mathbf{F}_{sys} = \sum \mathbf{F}_{\substack{\text{contents of the}\\\text{coincident control volume}}} \qquad (5.20)$$

Furthermore, for a system and the contents of a coincident control volume that is fixed and nondeforming, the Reynolds transport theorem [Eq. 4.19 with $b$ set equal to the velocity (i.e., momentum per unit mass), and $B_{sys}$ being the system momentum] allows us to conclude that

$$\frac{D}{Dt}\int_{sys}\mathbf{V}\rho\, d\forall = \frac{\partial}{\partial t}\int_{cv}\mathbf{V}\rho\, d\forall + \int_{cs}\mathbf{V}\rho\mathbf{V}\cdot\hat{\mathbf{n}}\, dA \qquad (5.21)$$

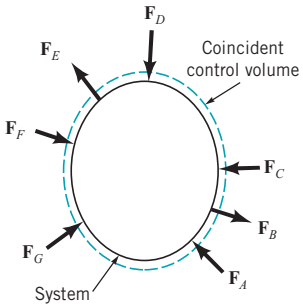



**FIGURE 5.2** External forces acting on system and coincident control volume.

or

$$\begin{array}{l}\text{time rate of change}\\ \text{of the linear}\\ \text{momentum of the}\\ \text{system}\end{array} = \begin{array}{l}\text{time rate of change}\\ \text{of the linear}\\ \text{momentum of the}\\ \text{contents of the}\\ \text{control volume}\end{array} + \begin{array}{l}\text{net rate of flow}\\ \text{of linear momentum}\\ \text{through the}\\ \text{control surface}\end{array}$$



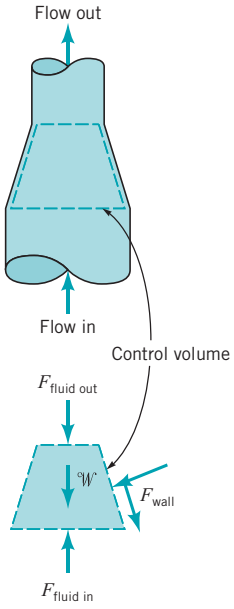
*V5.5 Marine propulsion*



Equation 5.21 states that the time rate of change of system linear momentum is expressed as the sum of the two control volume quantities: the time rate of change of the *linear momentum of the contents of the control volume*, and the net rate of *linear momentum flow through the control surface*. As particles of mass move into or out of a control volume through the control surface, they carry linear momentum in or out. Thus, linear momentum flow should seem no more unusual than mass flow.

For a control volume that is fixed (and thus inertial) and nondeforming, Eqs. 5.19, 5.20, and 5.21 provide an appropriate mathematical statement of Newton's second law of motion as

$$\frac{\partial}{\partial t}\int_{cv} \mathbf{V}\rho\, d\forall + \int_{cs} \mathbf{V}\rho\mathbf{V}\cdot\hat{\mathbf{n}}\, dA = \sum \mathbf{F}_{\substack{\text{contents of the}\\ \text{control volume}}} \tag{5.22}$$




We call Eq. 5.22 the ***linear momentum equation***.

In our application of the linear momentum equation, we initially confine ourselves to fixed, nondeforming control volumes for simplicity. Subsequently, we discuss the use of a moving but inertial, nondeforming control volume. We do not consider deforming control volumes and accelerating (noninertial) control volumes. If a control volume is noninertial, the acceleration components involved (for example, translation acceleration, Coriolis acceleration, and centrifugal acceleration) require consideration.

The forces involved in Eq. 5.22 are body and surface forces that act on what is contained in the control volume as shown in the sketch in the margin. The only body force we consider in this chapter is the one associated with the action of gravity. We experience this body force as weight, $\mathcal{W}$. The surface forces are basically exerted on the contents of the control volume by material just outside the control volume in contact with material just inside the control volume. For example, a wall in contact with fluid can exert a reaction surface force on the fluid it bounds. Similarly, fluid just outside the control volume can push on fluid just inside the control volume at a common interface, usually an opening in the control surface through which fluid flow occurs. An immersed object can resist fluid motion with surface forces.

The linear momentum terms in the momentum equation deserve careful explanation. We clarify their physical significance in the following sections.



*V5.6 Force due to a water jet*



## 5.2.2 Application of the Linear Momentum Equation

The linear momentum equation for an inertial control volume is a vector equation (Eq. 5.22). In engineering applications, components of this vector equation resolved along orthogonal coordinates, for example, $x$, $y$, and $z$ (rectangular coordinate system) or $r$, $\theta$, and $x$ (cylindrical coordinate system), will normally be used. A simple example involving steady, incompressible flow is considered first.

CMS Cepcor EX 1046 Page 111
CMS Cepcor v. Sandvik IPR2024-01380

## EXAMPLE 5.10 Linear Momentum—Change in Flow Direction

**GIVEN** As shown in Fig. E5.10*a*, a horizontal jet of water exits a nozzle with a uniform speed of $V_1 = 10$ ft/s, strikes a vane, and is turned through an angle $\theta$.

**FIND** Determine the anchoring force needed to hold the vane stationary if gravity and viscous effects are negligible.

## SOLUTION

We select a control volume that includes the vane and a portion of the water (see Figs. E5.10*b*, *c*) and apply the linear momentum equation to this fixed control volume. The only portions of the control surface across which fluid flows are section (1) (the entrance) and section (2) (the exit). Hence, the $x$ and $z$ components of Eq. 5.22 become

$$\frac{\partial}{\partial t}\int_{cv} u\,\rho\,d\forall + \int_{cs} u\,\rho\,\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = \sum F_x$$

(0 (flow is steady))

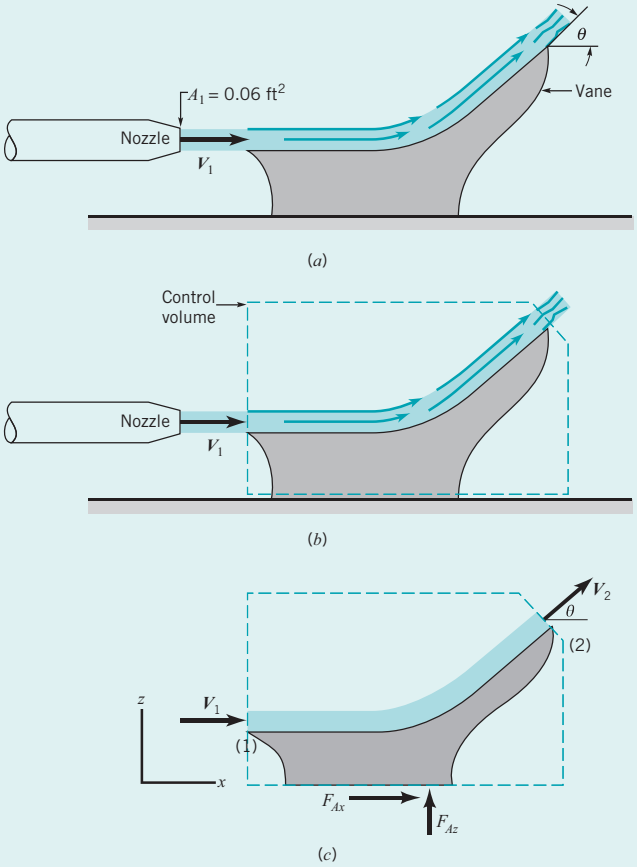
and

$$\frac{\partial}{\partial t}\int_{cv} w\,\rho\,d\forall + \int_{cs} w\,\rho\,\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = \sum F_z$$

(0 (flow is steady))

or

$$u_2\rho A_2 V_2 - u_1\rho A_1 V_1 = \Sigma F_x \qquad (1)$$




■ FIGURE E5.10

and

$$w_2\rho A_2 V_2 - w_1\rho A_1 V_1 = \Sigma F_z \qquad (2)$$

where $\mathbf{V} = u\hat{\mathbf{i}} + w\hat{\mathbf{k}}$, and $\Sigma F_x$ and $\Sigma F_z$ are the net $x$ and $z$ components of force acting on the contents of the control volume. Depending on the particular flow situation being considered and the coordinate system chosen, the $x$ and $z$ components of velocity, $u$ and $w$, can be positive, negative, or zero. In this example the flow is in the positive directions at both the inlet and the outlet.

The water enters and leaves the control volume as a free jet at atmospheric pressure. Hence, there is atmospheric pressure surrounding the entire control volume, and the net pressure force on the control volume surface is zero. If we neglect the weight of the water and vane, the only forces applied to the control volume contents are the horizontal and vertical components of the anchoring force, $F_{Ax}$ and $F_{Az}$, respectively.

With negligible gravity and viscous effects, and since $p_1 = p_2$, the speed of the fluid remains constant so that $V_1 = V_2 = 10$ ft/s (see the Bernoulli equation, Eq. 3.7). Hence, at section (1), $u_1 = V_1$, $w_1 = 0$, and at section (2), $u_2 = V_1\cos\theta$, $w_2 = V_1\sin\theta$.

By using this information, Eqs. 1 and 2 can be written as

$$V_1\cos\theta\,\rho\,A_2 V_1 - V_1\,\rho\,A_1 V_1 = F_{Ax} \qquad (3)$$

and

$$V_1\sin\theta\,\rho\,A_2 V_1 - 0\,\rho\,A_1 V_1 = F_{Az} \qquad (4)$$

Equations 3 and 4 can be simplified by using conservation of mass, which states that for this incompressible flow $A_1V_1 = A_2V_2$, or $A_1 = A_2$ since $V_1 = V_2$. Thus

$$F_{Ax} = -\rho A_1 V_1^2 + \rho A_1 V_1^2\cos\theta = -\rho A_1 V_1^2(1-\cos\theta) \qquad (5)$$

and

$$F_{Az} = \rho A_1 V_1^2\sin\theta \qquad (6)$$

With the given data we obtain

$$\begin{aligned} F_{Ax} &= -(1.94\ \text{slugs/ft}^3)(0.06\ \text{ft}^2)(10\ \text{ft/s})^2(1-\cos\theta) \\ &= -11.64(1-\cos\theta)\ \text{slugs}\cdot\text{ft/s}^2 \\ &= -11.64(1-\cos\theta)\ \text{lb} \end{aligned}$$ **(Ans)**

and

$$\begin{aligned} F_{Az} &= (1.94\ \text{slugs/ft}^3)(0.06\ \text{ft}^2)(10\ \text{ft/s})^2\sin\theta \\ &= 11.64\sin\theta\ \text{lb} \end{aligned}$$ **(Ans)**

**COMMENTS** The values of $F_{Ax}$ and $F_{Az}$ as a function of $\theta$ are shown in Fig. E5.10*d*. Note that if $\theta = 0$ (i.e., the vane does not turn the water), the anchoring force is zero. The inviscid fluid merely slides along the vane without putting any force on it. If $\theta = 90°$, then $F_{Ax} = -11.64$ lb and $F_{Az} = 11.64$ lb. It is necessary to push on the vane (and, hence, for the vane to push on the water)

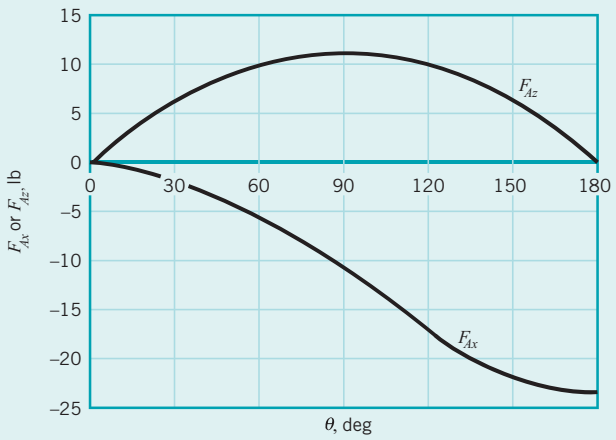



**FIGURE E5.10*d***

to the left ($F_{Ax}$ is negative) and up in order to change the direction of flow of the water from horizontal to vertical. This momentum change requires a force. If $\theta = 180°$, the water jet is turned back on itself. This requires no vertical force ($F_{Az} = 0$), but the horizontal force ($F_{Ax} = -23.3$ lb) is two times that required if $\theta = 90°$. This horizontal fluid momentum change requires a horizontal force only.

Note that the anchoring force (Eqs. 5, 6) can be written in terms of the mass flowrate, $\dot{m} = \rho A_1 V_1$, as

$$F_{Ax} = -\dot{m}V_1(1 - \cos\theta)$$

and

$$F_{Az} = \dot{m}V_1 \sin\theta$$

In this example exerting a force on a fluid flow resulted in a change in its direction only (i.e., change in its linear momentum).

## Fluids in the News

**Where the plume goes** Commercial airliners have wheel brakes very similar to those on highway vehicles. In fact, antilock brakes now found on most new cars were first developed for use on airplanes. However, when landing, the major braking force comes from the engine rather than the wheel brakes. Upon touchdown, a piece of engine cowling translates aft and blocker doors drop down, directing the engine airflow into a honeycomb structure called a cascade. The cascade reverses the direction of the high-speed engine exhausts by nearly 180° so that it flows forward. As predicted by the *momentum equation*, the air passing through the engine produces a substantial braking force—the reverse thrust. Designers must know the flow pattern of the exhaust plumes to eliminate potential problems. For example, the plumes of hot exhaust must be kept away from parts of the aircraft where repeated heating and cooling could cause premature fatigue. Also, the plumes must not re-enter the engine inlet, or blow debris from the runway in front of the engine, or envelop the vertical tail. (See Problem 5.67.)

## EXAMPLE 5.11 Linear Momentum—Weight, Pressure, and Change in Speed

**GIVEN** As shown in Fig. E5.11*a*, water flows through a nozzle attached to the end of a laboratory sink faucet with a flowrate of 0.6 liters/s. The nozzle inlet and exit diameters are 16 and 5 mm, respectively, and the nozzle axis is vertical. The mass of the nozzle is 0.1 kg. The pressure at section (1) is 464 kPa.

**FIND** Determine the anchoring force required to hold the nozzle in place.

### SOLUTION

The anchoring force sought is the reaction force between the faucet and nozzle threads. To evaluate this force we select a control volume that includes the entire nozzle and the water contained in the nozzle at an instant, as is indicated in Figs. E5.11*a* and E5.11*b*. All of the vertical forces acting on the contents of this control volume are identified in Fig. E5.11*b*. The action of atmospheric pressure cancels out in every direction and is not shown. Gage pressure forces do not cancel out in the vertical direction and are shown. Application of the vertical or $z$ direction component of Eq. 5.22 to the contents of this control volume leads to

$$\frac{\partial}{\partial t}\int_{cv} w\rho\, d\forall + \int_{cs} w\rho \mathbf{V}\cdot\hat{\mathbf{n}}\, dA = F_A - \mathcal{W}_n - p_1A_1 - \mathcal{W}_w + p_2A_2 \qquad (1)$$

(0 (flow is steady))

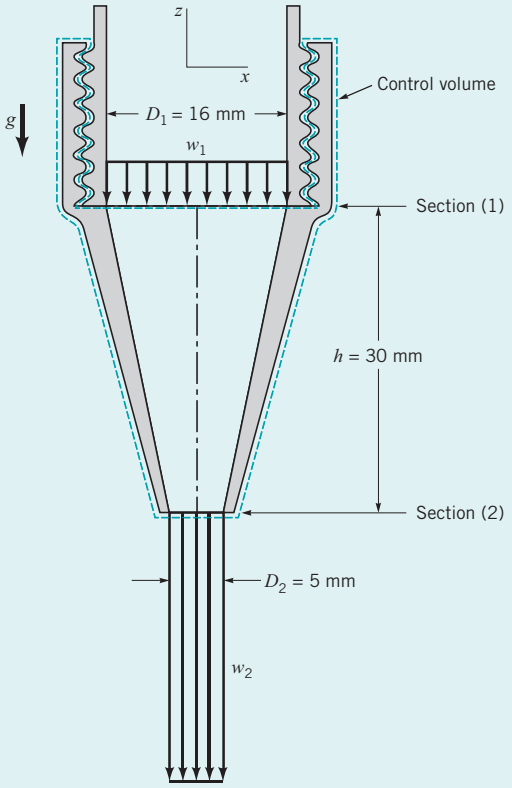
where $w$ is the $z$ direction component of fluid velocity, and the various parameters are identified in the figure.

Note that the positive direction is considered "up" for the forces. We will use this same sign convention for the fluid velocity, $w$, in Eq. 1. In Eq. 1, the dot product, $\mathbf{V}\cdot\hat{\mathbf{n}}$, is "+" for flow out of the control volume and "−" for flow into the control volume. For this particular example

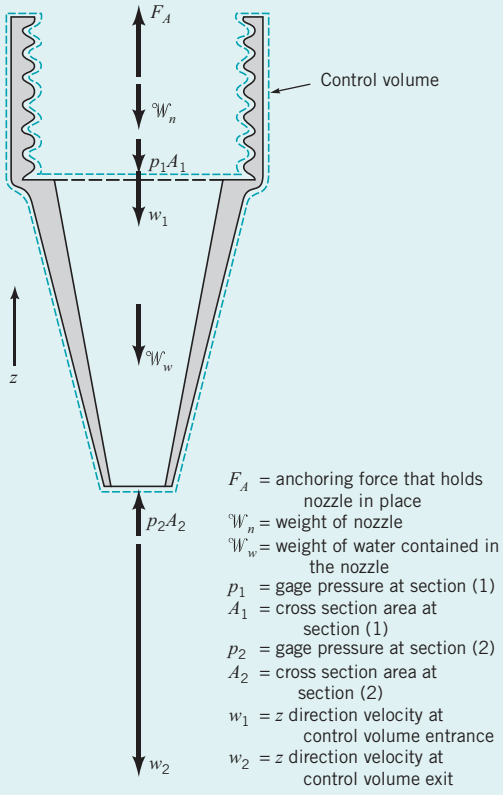
$$\mathbf{V}\cdot\hat{\mathbf{n}}\, dA = \pm|w|\, dA \qquad (2)$$

with the "+" used for flow out of the control volume and "−" used for flow in. To evaluate the control surface integral in Eq. 1, we need to assume a distribution for fluid velocity, $w$, and fluid density, $\rho$. For simplicity, we assume that $w$ is uniformly distributed or constant, with magnitudes of $w_1$ and $w_2$ over cross-sectional areas $A_1$ and $A_2$. Also, this flow is incompressible so the

CMS Cepcor EX 1046 Page 113
CMS Cepcor v. Sandvik IPR2024-01380




■ FIGURE E5.11a




■ FIGURE E5.11b

fluid density, $\rho$, is constant throughout. Proceeding further we obtain for Eq. 1

$$(-\dot{m}_1)(-w_1) + \dot{m}_2(-w_2) = F_A - \mathcal{W}_n - p_1A_1 - \mathcal{W}_w + p_2A_2 \quad \textbf{(3)}$$

where $\dot{m} = \rho AV$ is the mass flowrate.

Note that $-w_1$ and $-w_2$ are used because both of these velocities are "down." Also, $-\dot{m}_1$ is used because it is associated with flow into the control volume. Similarly, $+\dot{m}_2$ is used because it is associated with flow out of the control volume. Solving Eq. 3 for the anchoring force, $F_A$, we obtain

$$F_A = \dot{m}_1w_1 - \dot{m}_2w_2 + \mathcal{W}_n + p_1A_1 + \mathcal{W}_w - p_2A_2 \quad \textbf{(4)}$$

From the conservation of mass equation, Eq. 5.12, we obtain

$$\dot{m}_1 = \dot{m}_2 = \dot{m} \quad \textbf{(5)}$$

which when combined with Eq. 4 gives

$$F_A = \dot{m}(w_1 - w_2) + \mathcal{W}_n + p_1A_1 + \mathcal{W}_w - p_2A_2 \quad \textbf{(6)}$$

It is instructive to note how the anchoring force is affected by the different actions involved. As expected, the nozzle weight, $\mathcal{W}_n$, the water weight, $\mathcal{W}_w$, and gage pressure force at section (1), $p_1A_1$, all increase the anchoring force, while the gage pressure force at section (2), $p_2A_2$, acts to decrease the anchoring force. The change in the vertical momentum flowrate, $\dot{m}(w_1 - w_2)$, will, in this instance, decrease the anchoring force because this change is negative ($w_2 > w_1$).

To complete this example we use quantities given in the problem statement to quantify the terms on the right-hand side of Eq. 6.

From Eq. 5.6,

$$\dot{m} = \rho w_1A_1 = \rho Q = (999 \text{ kg/m}^3)(0.6 \text{ liter/s})(10^{-3} \text{ m}^3\text{/liter}) = 0.599 \text{ kg/s} \quad \textbf{(7)}$$

and

$$w_1 = \frac{Q}{A_1} = \frac{Q}{\pi(D_1^2/4)} = \frac{(0.6 \text{ liter/s})(10^{-3} \text{ m}^3\text{/liter})}{\pi(16 \text{ mm})^2/4(1000^2 \text{ mm}^2/\text{m}^2)} = 2.98 \text{ m/s} \quad \textbf{(8)}$$

Also from Eq. 5.6,

$$w_2 = \frac{Q}{A_2} = \frac{Q}{\pi(D_2^2/4)} = \frac{(0.6 \text{ liter/s})(10^{-3} \text{ m}^3\text{/liter})}{\pi(5 \text{ mm})^2/4(1000^2 \text{ mm}^2/\text{m}^2)} = 30.6 \text{ m/s} \quad \textbf{(9)}$$

The weight of the nozzle, $\mathcal{W}_n$, can be obtained from the nozzle mass, $m_n$, with

$$\mathcal{W}_n = m_ng = (0.1 \text{ kg})(9.81 \text{ m/s}^2) = 0.981 \text{ N} \quad \textbf{(10)}$$

The weight of the water in the control volume, $\mathcal{W}_w$, can be obtained from the water density, $\rho$, and the volume of water, $\forall_w$, in

the truncated cone of height $h$. That is,

$$\mathcal{W}_w = \rho \forall_w g$$

where

$$\begin{aligned}\forall_w &= \tfrac{1}{12}\pi h(D_1^2 + D_2^2 + D_1D_2)\\ &= \frac{1}{12}\pi \frac{(30\text{ mm})}{(1000\text{ mm/m})}\\ &\quad \times \left[\frac{(16\text{ mm})^2 + (5\text{ mm})^2 + (16\text{ mm})(5\text{ mm})}{(1000^2\text{ mm}^2/\text{m}^2)}\right]\\ &= 2.84 \times 10^{-6}\text{ m}^3\end{aligned}$$

Thus,

$$\begin{aligned}\mathcal{W}_w &= (999\text{ kg/m}^3)(2.84 \times 10^{-6}\text{ m}^3)(9.81\text{ m/s}^2)\\ &= 0.0278\text{ N}\end{aligned} \qquad \textbf{(11)}$$

The gage pressure at section (2), $p_2$, is zero since, as discussed in Section 3.6.1, when a subsonic flow discharges to the atmosphere as in the present situation, the discharge pressure is essentially atmospheric. The anchoring force, $F_A$, can now be determined from Eqs. 6 through 11 with

$$\begin{aligned}F_A &= (0.599\text{ kg/s})(2.98\text{ m/s} - 30.6\text{ m/s}) + 0.981\text{ N}\\ &\quad + (464\text{ kPa})(1000\text{ Pa/kPa})\frac{\pi(16\text{ mm})^2}{4(1000^2\text{ mm}^2/\text{m}^2)}\\ &\quad + 0.0278\text{ N} - 0\end{aligned}$$

or

$$\begin{aligned}F_A &= -16.5\text{ N} + 0.981\text{ N} + 93.3\text{ N} + 0.0278\text{ N}\\ &= 77.8\text{ N}\end{aligned} \qquad \textbf{(Ans)}$$

Since the anchoring force, $F_A$, is positive, it acts upward in the $z$ direction. The nozzle would be pushed off the pipe if it were not fastened securely.

**COMMENT** The control volume selected above to solve problems such as these is not unique. The following is an alternate solution that involves two other control volumes—one containing only the nozzle and the other containing only the water in the nozzle. These control volumes are shown in Figs. E5.11*c* and E5.11*d* along with the vertical forces acting on the contents of each control volume. The new force involved, $R_z$, represents the interaction between the water and the conical inside surface of the nozzle. It includes the net pressure and viscous forces at this interface.

Application of Eq. 5.22 to the contents of the control volume of Fig. E5.11*c* leads to

$$F_A = \mathcal{W}_n + R_z - p_{\text{atm}}(A_1 - A_2) \qquad \textbf{(12)}$$

The term $p_{\text{atm}}(A_1 - A_2)$ is the resultant force from the atmospheric pressure acting upon the exterior surface of the nozzle (i.e., that portion of the surface of the nozzle that is not in contact with the water). Recall that the pressure force on a curved surface (such as the exterior surface of the nozzle) is equal to the pressure times the projection of the surface area on a plane perpendicular to the axis of the nozzle. The projection of this area on a plane perpendicular to the $z$ direction is $A_1 - A_2$. The effect of the atmospheric pressure on the internal area (between the nozzle and the water) is already included in $R_z$ which represents the net force on this area.

Similarly, for the control volume of Fig. E5.11*d* we obtain

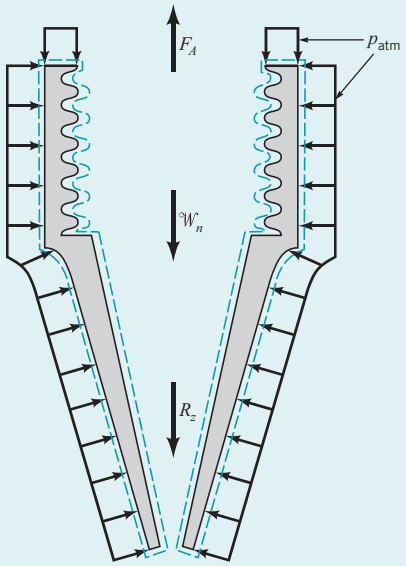
$$\begin{aligned}R_z = \dot{m}(w_1 - w_2) + \mathcal{W}_w + (p_1 + p_{\text{atm}})A_1\\ - (p_2 - p_{\text{atm}})A_2\end{aligned} \qquad \textbf{(13)}$$

where $p_1$ and $p_2$ are gage pressures. From Eq. 13 it is clear that the value of $R_z$ depends on the value of the atmospheric pressure, $p_{\text{atm}}$, since $A_1 \neq A_2$. That is, we must use absolute pressure, not gage pressure, to obtain the correct value of $R_z$. From Eq. 13 we can easily identify which forces acting on the flowing fluid change its velocity magnitude and thus linear momentum.

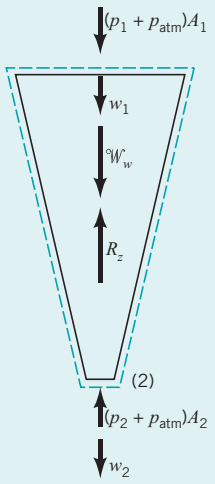
By combining Eqs. 12 and 13 we obtain the same result for $F_A$ as before (Eq. 6):

$$F_A = \dot{m}(w_1 - w_2) + \mathcal{W}_n + p_1A_1 + W_w - p_2A_2$$

Note that although the force between the fluid and the nozzle wall, $R_z$, is a function of $p_{\text{atm}}$, the anchoring force, $F_A$, is not. That is, we were correct in using gage pressure when solving for $F_A$ by means of the original control volume shown in Fig. E5.11*b*.



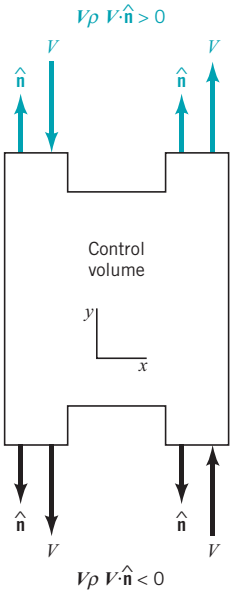

■ FIGURE E5.11*c*




■ FIGURE E5.11*d*

Several important generalities about the application of the linear momentum equation (Eq. 5.22) are apparent in the example just considered.

1. When the flow is uniformly distributed over a section of the control surface where flow into or out of the control volume occurs, the integral operations are simplified. Thus, one-dimensional flows are easier to work with than flows involving nonuniform velocity distributions.
2. Linear momentum is directional; it can have components in as many as three orthogonal coordinate directions. Furthermore, along any one coordinate, the linear momentum of a fluid particle can be in the positive or negative direction and thus be considered as a positive or a negative quantity. In Example 5.11, only the linear momentum in the $z$ direction was considered (all of it was in the negative $z$ direction and was hence treated as being negative).
3. The flow of positive or negative linear momentum *into* a control volume involves a negative $\mathbf{V} \cdot \hat{\mathbf{n}}$ product. Momentum flow *out* of the control volume involves a positive $\mathbf{V} \cdot \hat{\mathbf{n}}$ product. The correct algebraic sign (+ or −) to assign to momentum flow ($\mathbf{V}\rho\mathbf{V} \cdot \hat{\mathbf{n}}\, dA$) will depend on the sense of the velocity (+ in positive coordinate direction, − in negative coordinate direction) and the $\mathbf{V} \cdot \hat{\mathbf{n}}$ product (+ for flow out of the control volume, − for flow into the control volume). This is shown in the figure in the margin. In Example 5.11, the momentum flow into the control volume past section (1) was a positive (+) quantity while the momentum flow out of the control volume at section (2) was a negative (−) quantity.
4. The time rate of change of the linear momentum of the contents of a nondeforming control volume (i.e., $\partial/\partial t \int_{cv} \mathbf{V}\rho\, d\forall$) is zero for steady flow. The momentum problems considered in this text all involve steady flow.
5. If the control surface is selected so that it is perpendicular to the flow where fluid enters or leaves the control volume, the surface force exerted at these locations by fluid outside the control volume on fluid inside will be due to pressure. Furthermore, when subsonic flow exits from a control volume into the atmosphere, atmospheric pressure prevails at the exit cross section. In Example 5.11, the flow was subsonic and so we set the exit flow pressure at the atmospheric level. The continuity equation (Eq. 5.12) allowed us to evaluate the fluid flow velocities $w_1$ and $w_2$ at sections (1) and (2).
6. The forces due to atmospheric pressure acting on the control surface may need consideration as indicated by Eq. 13 in Example 5.11 for the reaction force between the nozzle and the fluid. When calculating the anchoring force, $F_A$, the forces due to atmospheric pressure on the control surface cancel each other (for example, after combining Eqs. 12 and 13 the atmospheric pressure forces are no longer involved) and gage pressures may be used.
7. The external forces have an algebraic sign, positive if the force is in the assigned positive coordinate direction and negative otherwise.
8. Only external forces acting on the contents of the control volume are considered in the linear momentum equation (Eq. 5.22). If the fluid alone is included in a control volume, reaction forces between the fluid and the surface or surfaces in contact with the fluid [wetted surface(s)] will need to be in Eq. 5.22. If the fluid and the wetted surface or surfaces are within the control volume, the reaction forces between fluid and wetted surface(s) do not appear in the linear momentum equation (Eq. 5.22) because they are internal, not external forces. The anchoring force that holds the wetted surface(s) in place is an external force, however, and must therefore be in Eq. 5.22.
9. The force required to anchor an object will generally exist in response to surface pressure and/or shear forces acting on the control surface, to a change in linear momentum flow through the control volume containing the object, and to the weight of the object and the fluid contained in the control volume. In Example 5.11 the nozzle anchoring force was required mainly because of pressure forces and partly because of a change in linear momentum flow associated with accelerating the fluid in the nozzle. The weight of the water and the nozzle contained in the control volume influenced the size of the anchoring force only slightly.




*A control volume diagram is similar to a free-body diagram.*



*V5.7 Running on water*



## Fluids in the News

**Motorized surfboard** When Bob Montgomery, a former professional surfer, started to design his motorized surfboard (called a jet board), he discovered that there were many engineering challenges to the design. The idea is to provide surfing to anyone, no matter where they live, near or far from the ocean. The rider stands on the device like a surfboard and steers it like a surfboard by shifting his/her body weight. A new, sleek, compact 45-horsepower engine and pump was designed to fit within the surfboard hull. Thrust is produced in response to the change in *linear momentum* of the water stream as it enters through the inlet passage and exits through an appropriately designed nozzle. Some of the fluid dynamic problems associated with designing the craft included one-way valves so that water does not get into the engine (at both the intake or exhaust ports), buoyancy, hydrodynamic lift, drag, thrust, and hull stability. (See Problem 5.68.)

To further demonstrate the use of the linear momentum equation (Eq. 5.22), we consider another one-dimensional flow example before moving on to other facets of this important equation.

## EXAMPLE 5.12 Linear Momentum—Pressure and Change in Flow Direction

**GIVEN** Water flows through a horizontal, 180° pipe bend as illustrated in Fig. E5.12*a*. The flow cross-sectional area is constant at a value of 0.1 ft$^2$ through the bend. The magnitude of the flow velocity everywhere in the bend is axial and 50 ft/s. The absolute pressures at the entrance and exit of the bend are 30 psia and 24 psia, respectively.

**FIND** Calculate the horizontal ($x$ and $y$) components of the anchoring force required to hold the bend in place.

### SOLUTION

Since we want to evaluate components of the anchoring force to hold the pipe bend in place, an appropriate control volume (see dashed line in Fig. E5.12*a*) contains the bend and the water in the bend at an instant. The horizontal forces acting on the contents of this control volume are identified in Fig. E5.12*b*. Note that the weight of the water is vertical (in the negative $z$ direction) and does not contribute to the $x$ and $y$ components of the anchoring force. All of the horizontal normal and tangential forces exerted on the fluid and the pipe bend are resolved and combined into the two resultant components, $F_{Ax}$ and $F_{Ay}$. These two forces act on the control volume contents, and thus for the $x$ direction, Eq. 5.22 leads to

$$\int_{cs} u\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = F_{Ax} \qquad (1)$$

At sections (1) and (2), the flow is in the $y$ direction and therefore $u = 0$ at both cross sections. There is no $x$ direction momentum flow into or out of the control volume and we conclude from Eq. 1 that

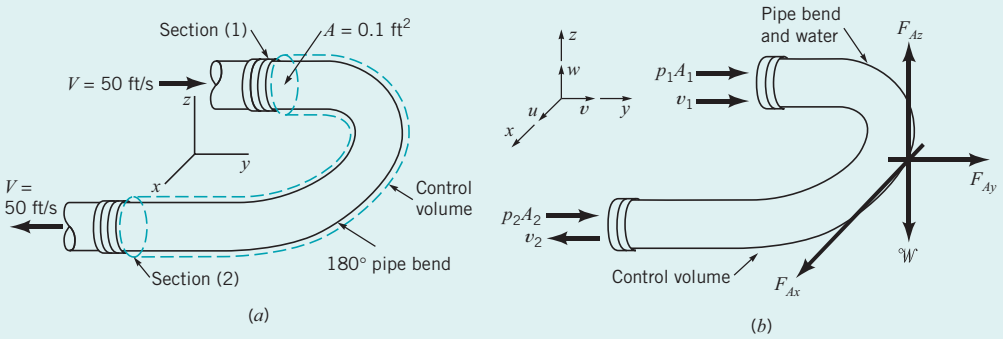
$$F_{Ax} = 0 \qquad \textbf{(Ans)}$$

For the $y$ direction, we get from Eq. 5.22

$$\int_{cs} v\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = F_{Ay} + p_1A_1 + p_2A_2 \qquad (2)$$

For one-dimensional flow, the surface integral in Eq. 2 is easy to evaluate and Eq. 2 becomes

$$(+v_1)(-\dot{m}_1) + (-v_2)(+\dot{m}_2) = F_{Ay} + p_1A_1 + p_2A_2 \qquad (3)$$




■ FIGURE E5.12

Note that the $y$ component of velocity is positive at section (1) but is negative at section (2). Also, the mass flowrate term is negative at section (1) (flow in) and is positive at section (2) (flow out). From the continuity equation (Eq. 5.12), we get

$$\dot{m} = \dot{m}_1 = \dot{m}_2 \tag{4}$$

and thus Eq. 3 can be written as

$$-\dot{m}(v_1 + v_2) = F_{Ay} + p_1A_1 + p_2A_2 \tag{5}$$

Solving Eq. 5 for $F_{Ay}$ we obtain

$$F_{Ay} = -\dot{m}(v_1 + v_2) - p_1A_1 - p_2A_2 \tag{6}$$

From the given data we can calculate the mass flowrate, $\dot{m}$, from Eq. 5.6 as

$$\dot{m} = \rho_1 A_1 v_1 = (1.94 \text{ slugs/ft}^3)(0.1 \text{ ft}^2)(50 \text{ ft/s}) = 9.70 \text{ slugs/s}$$

For determining the anchoring force, $F_{Ay}$, the effects of atmospheric pressure cancel and thus gage pressures for $p_1$ and $p_2$ are appropriate. By substituting numerical values of variables into Eq. 6, and using the fact that 1 lb = 1 slug · ft/s$^2$ we get

$$\begin{aligned} F_{Ay} = &-(9.70 \text{ slugs/s})(50 \text{ ft/s} + 50 \text{ ft/s}) \\ &- (30 \text{ psia} - 14.7 \text{ psia})(144 \text{ in.}^2/\text{ft}^2)(0.1 \text{ ft}^2) \\ &- (24 \text{ psia} - 14.7 \text{ psia})(144 \text{ in.}^2/\text{ft}^2)(0.1 \text{ ft}^2) \\ F_{Ay} = &-970 \text{ lb} - 220 \text{ lb} - 134 \text{ lb} = -1324 \text{ lb} \end{aligned} \quad \textbf{(Ans)}$$

The negative sign for $F_{Ay}$ is interpreted as meaning that the $y$ component of the anchoring force is actually in the negative $y$ direction, not the positive $y$ direction as originally indicated in Fig. E5.12*b*.

**COMMENT** As with Example 5.11, the anchoring force for the pipe bend is independent of the atmospheric pressure. However, the force that the bend puts on the fluid inside of it, $R_y$, depends on the atmospheric pressure. We can see this by using a control volume which surrounds only the fluid within the bend as shown in Fig. E5.12*c*. Application of the momentum equation to this situation gives

$$R_y = -\dot{m}(v_1 + v_2) - p_1A_1 - p_2A_2$$

where $p_1$ and $p_2$ must be in terms of absolute pressure because the force between the fluid and the pipe wall, $R_y$, is the complete pressure effect (i.e., absolute pressure). We see that forces exerted on the flowing fluid result in a change in its velocity direction (a change in linear momentum).

Thus, we obtain

$$\begin{aligned} R_y = &-(9.70 \text{ slugs/s})(50 \text{ ft/s} + 50 \text{ ft/s}) \\ &- (30 \text{ psia})(144 \text{ in.}^2/\text{ft}^2)(0.1 \text{ ft}^2) \\ &- (24 \text{ psia})(144 \text{ in.}^2/\text{ft}^2)(0.1 \text{ ft}^2) \\ = &-1748 \text{ lb} \end{aligned} \tag{7}$$

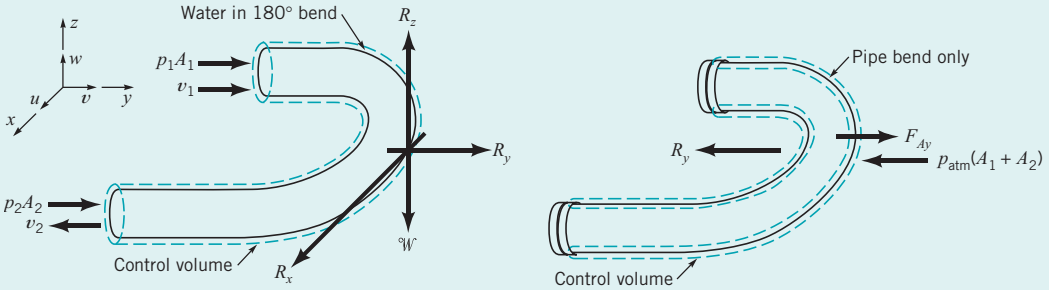
We can use the control volume that includes just the pipe bend (without the fluid inside it) as shown in Fig. E5.12*d* to determine $F_{Ay}$, the anchoring force component in the $y$ direction necessary to hold the bend stationary. The $y$ component of the momentum equation applied to this control volume gives

$$F_{Ay} = R_y + p_{\text{atm}}(A_1 + A_2) \tag{8}$$

where $R_y$ is given by Eq. 7. The $p_{\text{atm}}(A_1 + A_2)$ term represents the net pressure force on the outside portion of the control volume. Recall that the pressure force on the inside of the bend is accounted for by $R_y$. By combining Eqs. 7 and 8 and using the fact that $p_{\text{atm}} = 14.7 \text{ lb/in.}^2 (144 \text{ in.}^2/\text{ft}^2) = 2117 \text{ lb/ft}^2$, we obtain

$$\begin{aligned} F_{Ay} &= -1748 \text{ lb} + 2117 \text{ lb/ft}^2 (0.1 \text{ ft}^2 + 0.1 \text{ ft}^2) \\ &= -1324 \text{ lb} \end{aligned}$$

in agreement with the original answer obtained using the control volume of Fig. E5.12*b*.




■ FIGURE E5.12 cont.



*V5.8 Fire hose*

In Examples 5.10 and 5.12 the force exerted on a flowing fluid resulted in a change in flow direction only. This force was associated with constraining the flow, with a vane in Example 5.10, and with a pipe bend in Example 5.12. In Example 5.11 the force exerted on a flowing fluid resulted in a change in velocity magnitude only. This force was associated with a converging nozzle. Anchoring forces are required to hold a vane or conduit stationary. They are most easily estimated with a control volume that contains the vane or conduit and the flowing fluid involved. Alternately, two separate control volumes can be used, one containing the vane or conduit only and one containing the flowing fluid only.

CMS Cepcor EX 1046 Page 118
CMS Cepcor v. Sandvik IPR2024-01380

## EXAMPLE 5.13 Linear Momentum—Pressure, Change in Speed, and Friction

**GIVEN** Air flows steadily between two cross sections in a long, straight portion of 4-in. inside diameter pipe as indicated in Fig. E5.13, where the uniformly distributed temperature and pressure at each cross section are given. If the average air velocity at section (2) is 1000 ft/s, we found in Example 5.2 that the average air velocity at section (1) must be 219 ft/s. Assume uniform velocity distributions at sections (1) and (2).

**FIND** Determine the frictional force exerted by the pipe wall on the air flow between sections (1) and (2).

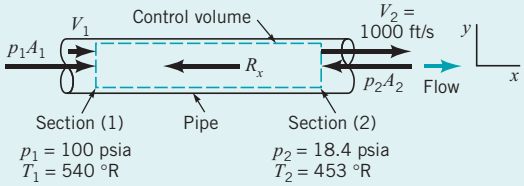



FIGURE E5.13

### SOLUTION

The control volume of Example 5.2 is appropriate for this problem. The forces acting on the air between sections (1) and (2) are identified in Fig. E5.13. The weight of air is considered negligibly small. The reaction force between the wetted wall of the pipe and the flowing air, $R_x$, is the frictional force sought. Application of the axial component of Eq. 5.22 to this control volume yields

$$\int_{cs} u\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = -R_x + p_1A_1 - p_2A_2 \tag{1}$$

The positive $x$ direction is set as being to the right. Furthermore, for uniform velocity distributions (one-dimensional flow), Eq. 1 becomes

$$(+u_1)(-\dot{m}_1) + (+u_2)(+\dot{m}_2) = -R_x + p_1A_1 - p_2A_2 \tag{2}$$

From conservation of mass (Eq. 5.12) we get

$$\dot{m} = \dot{m}_1 = \dot{m}_2 \tag{3}$$

so that Eq. 2 becomes

$$\dot{m}(u_2 - u_1) = -R_x + A_2(p_1 - p_2) \tag{4}$$

Solving Eq. 4 for $R_x$, we get

$$R_x = A_2(p_1 - p_2) - \dot{m}(u_2 - u_1) \tag{5}$$

The equation of state gives

$$\rho_2 = \frac{p_2}{RT_2} \tag{6}$$

and the equation for area $A_2$ is

$$A_2 = \frac{\pi D_2^2}{4} \tag{7}$$

Thus, from Eqs. 3, 6, and 7

$$\dot{m} = \left(\frac{p_2}{RT_2}\right)\left(\frac{\pi D_2^2}{4}\right)u_2$$

English Engineering (EE) units are often used for this kind of flow. The gas constant, R, for air in EE units is

$$\mathrm{R} = \frac{1716(\mathrm{ft}\cdot\mathrm{lb})/(\mathrm{slug}\cdot{}^\circ\mathrm{R})}{32.174(\mathrm{lbm/slug})} = 53.3(\mathrm{ft}\cdot\mathrm{lb})/(\mathrm{lbm}\cdot{}^\circ\mathrm{R})$$

$$\text{Hence, } \dot{m} = \frac{(18.4\ \mathrm{psia})(144\ \mathrm{in.^2/ft^2})}{[53.3(\mathrm{ft}\cdot\mathrm{lb})/(\mathrm{lbm}\cdot{}^\circ\mathrm{R})]\,(453\ {}^\circ\mathrm{R})} \times \frac{\pi(4\ \mathrm{in.})^2}{4(144\ \mathrm{in.^2/ft^2})}(1000\ \mathrm{ft/s}) = 9.57\ \mathrm{lbm/s} \tag{8}$$

Thus, from Eqs. 5 and 8

$$R_x = \frac{\pi(4\ \mathrm{in.})^2}{4}(100\ \mathrm{psia} - 18.4\ \mathrm{psia}) - (9.57\ \mathrm{lbm})(1000\ \mathrm{ft/s} - 219\ \mathrm{ft/s})/32.174(\mathrm{lbm}\cdot\mathrm{ft})/(\mathrm{lb}\cdot\mathrm{s}^2)$$
$$= 1025\ \mathrm{lb} - 232\ \mathrm{lb}$$

or

$$R_x = 793\ \mathrm{lb} \quad \textbf{(Ans)}$$

**COMMENT** For this compressible flow, the pressure difference drives the motion which results in a frictional force, $R_x$, and an acceleration of the fluid (i.e., a velocity magnitude increase). For a similar incompressible pipe flow, a pressure difference results in fluid motion with a frictional force only (i.e., no change in velocity magnitude).

## EXAMPLE 5.14 Linear Momentum—Weight, Pressure, Friction, and Nonuniform Velocity Profile

**GIVEN** Consider the flow of Example 5.4 to be vertically upward.

**FIND** Develop an expression for the fluid pressure drop that occurs between sections (1) and (2).

CMS Cepcor EX 1046 Page 119
CMS Cepcor v. Sandvik IPR2024-01380

## SOLUTION

A control volume (see dashed lines in Fig. E5.14) that includes only fluid from section (1) to section (2) is selected. The forces acting on the fluid in this control volume are identified in Fig. E5.14. The application of the axial component of Eq. 5.22 to the fluid in this control volume results in

$$\int_{cs} w\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = p_1A_1 - R_z - \mathcal{W} - p_2A_2 \qquad (1)$$

where $R_z$ is the resultant force of the wetted pipe wall on the fluid. Further, for uniform flow at section (1), and because the flow at section (2) is out of the control volume, Eq. 1 becomes

$$(+w_1)(-\dot{m}_1) + \int_{A_2}(+w_2)\rho(+w_2\,dA_2) = p_1A_1 - R_z - \mathcal{W} - p_2A_2 \qquad (2)$$

The positive direction is considered up. The surface integral over the cross-sectional area at section (2), $A_2$, is evaluated by using the parabolic velocity profile obtained in Example 5.4, $w_2 = 2w_1[1-(r/R)^2]$, as

$$\int_{A_2} w_2\rho w_2\,dA_2 = \rho\int_0^R w_2^2\,2\pi r\,dr$$

$$= 2\pi\rho\int_0^R (2w_1)^2\left[1-\left(\frac{r}{R}\right)^2\right]^2 r\,dr$$

or

$$\int_{A_2} w_2\rho w_2\,dA_2 = 4\pi\rho w_1^2\frac{R^2}{3} \qquad (3)$$

Combining Eqs. 2 and 3 we obtain

$$-w_1^2\rho\pi R^2 + \tfrac{4}{3}w_1^2\rho\pi R^2 = p_1A_1 - R_z - \mathcal{W} - p_2A_2 \qquad (4)$$

Solving Eq. 4 for the pressure drop from section (1) to section (2), $p_1 - p_2$, we obtain

$$p_1 - p_2 = \frac{\rho w_1^2}{3} + \frac{R_z}{A_1} + \frac{\mathcal{W}}{A_1} \qquad \textbf{(Ans)}$$

**COMMENT** We see that the drop in pressure from section (1) to section (2) occurs because of the following:

1. The change in momentum flow between the two sections associated with going from a uniform velocity profile to a parabolic velocity profile, $\rho w_1^2/3$
2. Pipe wall friction, $R_z$
3. The weight of the water column, $\mathcal{W}$; a hydrostatic pressure effect.

If the velocity profiles had been identically parabolic at sections (1) and (2), the momentum flowrate at each section would have been identical, a condition we call "fully developed" flow. Then, the pressure drop, $p_1 - p_2$, would be due only to pipe wall friction and the weight of the water column. If in addition to being fully developed, the flow involved negligible weight effects (for example, horizontal flow of liquids or the flow of gases in any direction) the drop in pressure between any two sections, $p_1 - p_2$, would be a result of pipe wall friction only.

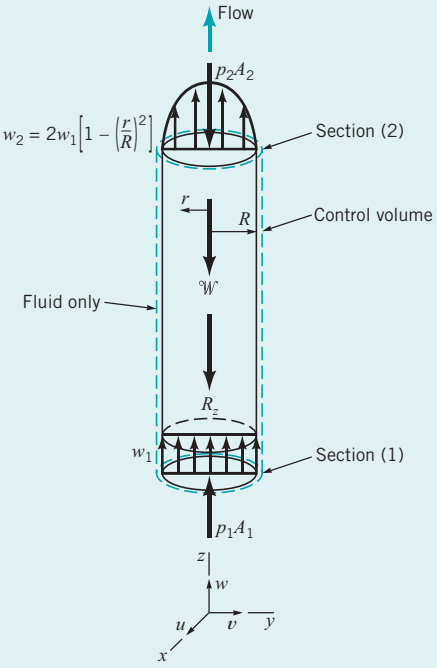
Note that although the average velocity is the same at section (1) as it is at section (2) ($\overline{V}_1 = \overline{V}_2 = w_1$), the momentum flux across section (1) is not the same as it is across section (2). If it were, the left-hand side of Eq. (4) would be zero. For this nonuniform flow the momentum flux can be written in terms of the average velocity, $\overline{V}$, and the *momentum coefficient*, $\beta$, as

$$\beta = \frac{\int w\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA}{\rho\overline{V}^2A}$$

Hence the momentum flux can be written as

$$\int_{cs} w\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = -\beta_1w_1^2\rho\pi R^2 + \beta_2w_1^2\rho\pi R^2$$

where $\beta_1 = 1$ ($\beta = 1$ for uniform flow) and $\beta_2 = 4/3$ ($\beta > 1$ for any nonuniform flow).




■ FIGURE E5.14

## EXAMPLE 5.15 Linear Momentum—Thrust

**GIVEN** A static thrust stand as sketched in Fig. E5.15 is to be designed for testing a jet engine. The following conditions are known for a typical test: Intake air velocity = 200 m/s; exhaust gas velocity = 500 m/s; intake cross-sectional area = 1 m$^2$; intake static pressure = −22.5 kPa = 78.5 kPa (abs); intake static temperature = 268 K; exhaust static pressure = 0 kPa= 101 kPa (abs).

**FIND** Estimate the nominal anchoring force for which to design.

## SOLUTION

The cylindrical control volume outlined with a dashed line in Fig. E5.15 is selected. The external forces acting in the axial direction are also shown. Application of the momentum equation (Eq. 5.22) to the contents of this control volume yields

$$\int_{cs} u\rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = p_1A_1 + F_{th} - p_2A_2 - p_{atm}(A_1 - A_2) \quad (1)$$

where the pressures are absolute. Thus, for one-dimensional flow, Eq. 1 becomes

$$(+u_1)(-\dot{m}_1) + (+u_2)(+\dot{m}_2) = (p_1 - p_{atm})A_1 - (p_2 - p_{atm})A_2 + F_{th} \quad (2)$$

The positive direction is to the right. The conservation of mass equation (Eq. 5.12) leads to

$$\dot{m} = \dot{m}_1 = \rho_1A_1u_1 = \dot{m}_2 = \rho_2A_2u_2 \quad (3)$$

Combining Eqs. 2 and 3 and using gage pressure we obtain

$$\dot{m}(u_2 - u_1) = p_1A_1 - p_2A_2 + F_{th} \quad (4)$$

Solving Eq. 4 for the thrust force, $F_{th}$, we obtain

$$F_{th} = -p_1A_1 + p_2A_2 + \dot{m}(u_2 - u_1) \quad (5)$$

We need to determine the mass flowrate, $\dot{m}$, to calculate $F_{th}$, and to calculate $\dot{m} = \rho_1A_1u_1$, we need $\rho_1$. From the ideal gas equation of state

$$\rho_1 = \frac{p_1}{RT_1} = \frac{(78.5\ \text{kPa})(1000\ \text{Pa/kPa})[1(\text{N/m}^2)/\text{Pa}]}{(286.9\ \text{J/kg}\cdot\text{K})(268\ \text{K})(1\ \text{N}\cdot\text{m/J})} = 1.02\ \text{kg/m}^3$$

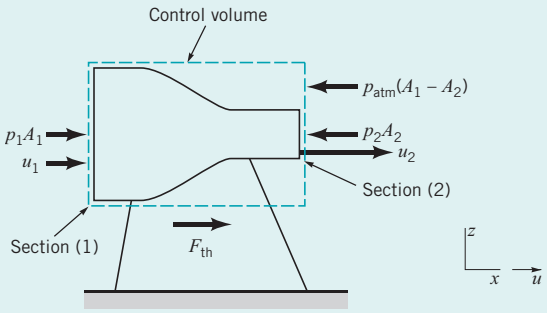



FIGURE E5.15

Thus,

$$\dot{m} = \rho_1A_1u_1 = (1.02\ \text{kg/m}^3)(1\ \text{m}^2)(200\ \text{m/s}) = 204\ \text{kg/s} \quad (6)$$

Finally, combining Eqs. 5 and 6 and substituting given data with $p_2 = 0$, we obtain

$$F_{th} = -(1\ \text{m}^2)(-22.5\ \text{kPa})(1000\ \text{Pa/kPa})[1(\text{N/m}^2)/\text{Pa}] + (204\ \text{kg/s})(500\ \text{m/s} - 200\ \text{m/s})[1\ \text{N}/(\text{kg}\cdot\text{m/s}^2)]$$

or

$$F_{th} = 22{,}500\ \text{N} + 61{,}200\ \text{N} = 83{,}700\ \text{N} \quad \textbf{(Ans)}$$

**COMMENT** The force of the thrust stand on the engine is directed toward the right. Conversely, the engine pushes to the left on the thrust stand (or aircraft).

## Fluids in the News

**Bow thrusters** In the past, large ships required the use of tugboats for precise maneuvering, especially when docking. Nowadays, most large ships (and many moderate to small ones as well) are equipped with bow thrusters to help steer in close quarters. The units consist of a mechanism (usually a ducted propeller mounted at right angles to the fore/aft axis of the ship) that takes water from one side of the bow and ejects it as a water jet on the other side. The *momentum flux* of this jet produces a starboard or port force on the ship for maneuvering. Sometimes a second unit is installed in the stern. Initially used in the bows of ferries, these versatile control devices have became popular in offshore oil servicing boats, fishing vessels, and larger ocean-going craft. They permit unassisted maneuvering alongside of oilrigs, vessels, loading platforms, fishing nets, and docks. They also provide precise control at slow speeds through locks, narrow channels, and bridges, where the rudder becomes very ineffective. (See Problem 5.69.)

## EXAMPLE 5.16 Linear Momentum—Nonuniform Pressure

**GIVEN** A sluice gate across a channel of width $b$ is shown in the closed and open positions in Figs. E5.16*a* and E5.16*b*.

**FIND** Is the anchoring force required to hold the gate in place larger when the gate is closed or when it is open?

## SOLUTION

We will answer this question by comparing expressions for the horizontal reaction force, $R_x$, between the gate and the water when the gate is closed and when the gate is open. The control volume used in each case is indicated with dashed lines in Figs. E5.16*a* and E5.16*b*.

When the gate is closed, the horizontal forces acting on the contents of the control volume are identified in Fig. E5.16*c*. Application of Eq. 5.22 to the contents of this control volume yields

$$\int_{cs} u\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = \tfrac{1}{2}\gamma H^2 b - R_x \qquad (1)$$

(0 (no flow))

Note that the hydrostatic pressure force, $\gamma H^2 b/2$, is used. From Eq. 1, the force exerted on the water by the gate (which is equal to the force necessary to hold the gate stationary) is

$$R_x = \tfrac{1}{2}\gamma H^2 b \qquad (2)$$

which is equal in magnitude to the hydrostatic force exerted on the gate by the water.

When the gate is open, the horizontal forces acting on the contents of the control volume are shown in Fig. E5.16*d*. Application of Eq. 5.22 to the contents of this control volume leads to

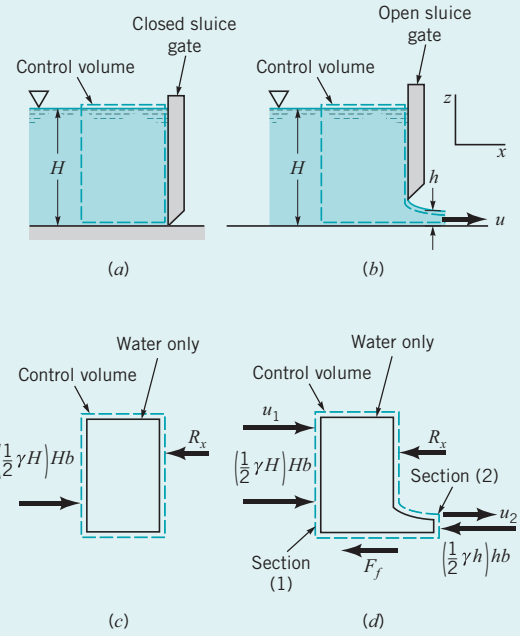
$$\int_{cs} u\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = \tfrac{1}{2}\gamma H^2 b - R_x - \tfrac{1}{2}\gamma h^2 b - F_f \qquad (3)$$

Note that because the water at sections (1) and (2) is flowing along straight, horizontal streamlines, the pressure distribution at those locations is hydrostatic, varying from zero at the free surface to $\gamma$ times the water depth at the bottom of the channel (see Chapter 3, Section 3.4). Thus, the pressure forces at sections (1) and (2) (given by the pressure at the centroid times the area) are $\gamma H^2 b/2$ and $\gamma h^2 b/2$, respectively. Also, the frictional force between the channel bottom and the water is specified as $F_f$. The surface integral in Eq. 3 is nonzero only where there is flow across the control surface. With the assumption of uniform velocity distributions,

$$\int_{cs} u\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA = (u_1)\rho(-u_1)Hb + (+u_2)\rho(+u_2)hb \qquad (4)$$

Thus, Eqs. 3 and 4 combine to form

$$-\rho u_1^2 Hb + \rho u_2^2 hb = \tfrac{1}{2}\gamma H^2 b - R_x - \tfrac{1}{2}\gamma h^2 b - F_f \qquad (5)$$




■ **FIGURE E5.16**

If $H \gg h$, the upstream velocity, $u_1$, is much less than $u_2$ so that the contribution of the incoming momentum flow to the control surface integral can be neglected and from Eq. 5 we obtain

$$R_x = \tfrac{1}{2}\gamma H^2 b - \tfrac{1}{2}\gamma h^2 b - F_f - \rho u_2^2 hb \qquad (6)$$

By using the continuity equation, $\dot{m} = \rho bHu_1 = \rho bhu_2$, Eq. (6) can be rewritten as

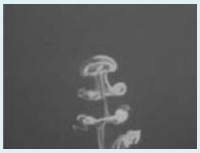
$$R_x = \tfrac{1}{2}\gamma H^2 b - \tfrac{1}{2}\gamma h^2 b - F_f - \dot{m}(u_2 - u_1) \qquad (7)$$

Hence, since $u_2 > u_1$, by comparing the expressions for $R_x$ (Eqs. 2 and 7) we conclude that the reaction force between the gate and the water (and therefore the anchoring force required to hold the gate in place) is smaller when the gate is open than when it is closed. **(Ans)**

*The linear momentum equation can be written for a moving control volume.*



*V5.9 Jelly fish*



All of the linear momentum examples considered thus far have involved stationary and nondeforming control volumes which are thus inertial because there is no acceleration. A nondeforming control volume translating in a straight line at constant speed is also inertial because there is no acceleration. For a system and an inertial, moving, nondeforming control volume that are both coincident at an instant of time, the Reynolds transport theorem (Eq. 4.23) leads to

$$\frac{D}{Dt}\int_{sys}\mathbf{V}\rho\,d\forall = \frac{\partial}{\partial t}\int_{cv}\mathbf{V}\rho\,d\forall + \int_{cs}\mathbf{V}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\,dA \qquad (5.23)$$

When we combine Eq. 5.23 with Eqs. 5.19 and 5.20, we get

$$\frac{\partial}{\partial t}\int_{cv}\mathbf{V}\rho\,d\forall + \int_{cs}\mathbf{V}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\,dA = \sum\mathbf{F}_{\substack{\text{contents of the}\\ \text{control volume}}} \qquad (5.24)$$

When the equation relating absolute, relative, and control volume velocities (Eq. 5.14) is used with Eq. 5.24, the result is

$$\frac{\partial}{\partial t}\int_{cv}(\mathbf{W}+\mathbf{V}_{cv})\rho\,d\forall + \int_{cs}(\mathbf{W}+\mathbf{V}_{cv})\rho\mathbf{W}\cdot\hat{\mathbf{n}}\,dA = \sum\mathbf{F}_{\substack{\text{contents of the}\\ \text{control volume}}} \qquad (5.25)$$

For a constant control volume velocity, $\mathbf{V}_{cv}$, and steady flow in the control volume reference frame,

$$\frac{\partial}{\partial t}\int_{cv}(\mathbf{W}+\mathbf{V}_{cv})\rho\, d\forall = 0 \tag{5.26}$$

Also, for this inertial, nondeforming control volume

$$\int_{cs}(\mathbf{W}+\mathbf{V}_{cv})\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA = \int_{cs}\mathbf{W}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA + \mathbf{V}_{cv}\int_{cs}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA \tag{5.27}$$

For steady flow (on an instantaneous or time-average basis), Eq. 5.15 gives

$$\int_{cs}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA = 0 \tag{5.28}$$

Combining Eqs. 5.25, 5.26, 5.27, and 5.28, we conclude that the linear momentum equation for an inertial, moving, nondeforming control volume that involves steady (instantaneous or time-average) flow is

*The linear momentum equation for a moving control volume involves the relative velocity.*

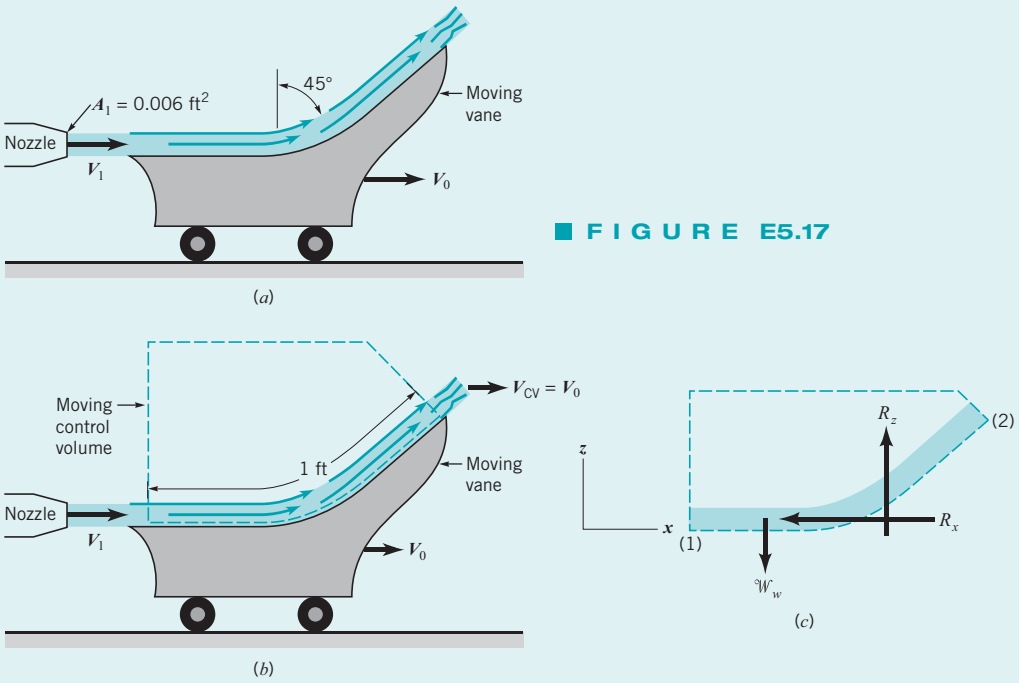
$$\boxed{\int_{cs}\mathbf{W}\rho\mathbf{W}\cdot\hat{\mathbf{n}}\, dA = \sum\mathbf{F}_{\substack{\text{contents of the}\\ \text{control volume}}}} \tag{5.29}$$

Example 5.17 illustrates the use of Eq. 5.29.

## EXAMPLE 5.17 Linear Momentum—Moving Control Volume

**GIVEN** A vane on wheels moves with constant velocity $\mathbf{V}_0$ when a stream of water having a nozzle exit velocity of $\mathbf{V}_1$ is turned 45° by the vane as indicated in Fig. E5.17*a*. Note that this is the same moving vane considered in Section 4.4.6 earlier. The speed of the water jet leaving the nozzle is 100 ft/s, and the vane is moving to the right with a constant speed of 20 ft/s.

**FIND** Determine the magnitude and direction of the force, **F**, exerted by the stream of water on the vane surface.




■ FIGURE E5.17

## SOLUTION

To determine the magnitude and direction of the force, **F**, exerted by the water on the vane, we apply Eq. 5.29 to the contents of the moving control volume shown in Fig. E5.17*b*. The forces acting on the contents of this control volume are indicated in Fig. E5.17*c*. Note that since the ambient pressure is atmospheric, all pressure forces cancel each other out. Equation 5.29 is applied to the contents of the moving control volume in component directions. For the $x$ direction (positive to the right), we get

$$\int_{cs} W_x \rho\, \mathbf{W} \cdot \hat{\mathbf{n}}\, dA = -R_x$$

or

$$(+W_1)(-\dot{m}_1) + (+W_2 \cos 45°)(+\dot{m}_2) = -R_x \qquad \textbf{(1)}$$

where

$$\dot{m}_1 = \rho_1 W_1 A_1 \qquad \text{and} \qquad \dot{m}_2 = \rho_2 W_2 A_2.$$

For the vertical or $z$ direction (positive up) we get

$$\int_{cs} W_z \rho \mathbf{W} \cdot \hat{\mathbf{n}}\, dA = R_z - \mathcal{W}_w$$

or

$$(+W_2 \sin 45°)(+\dot{m}_2) = R_z - \mathcal{W}_w \qquad \textbf{(2)}$$

We assume for simplicity that the water flow is frictionless and that the change in water elevation across the vane is negligible. Thus, from the Bernoulli equation (Eq. 3.7) we conclude that the speed of the water relative to the moving control volume, $W$, is constant or

$$W_1 = W_2$$

The relative speed of the stream of water entering the control volume, $W_1$, is

$$W_1 = V_1 - V_0 = 100 \text{ ft/s} - 20 \text{ ft/s} = 80 \text{ ft/s} = W_2$$

The water density is constant so that

$$\rho_1 = \rho_2 = 1.94 \text{ slugs/ft}^3$$

Application of the conservation of mass principle to the contents of the moving control volume (Eq. 5.16) leads to

$$\dot{m}_1 = \rho_1 W_1 A_1 = \rho_2 W_2 A_2 = \dot{m}_2$$

Combining results we get

$$R_x = \rho W_1^2 A_1 (1 - \cos 45°)$$

or

$$R_x = (1.94 \text{ slugs/ft}^3)(80 \text{ ft/s})^2(0.006 \text{ ft}^2)(1 - \cos 45°)$$
$$= 21.8 \text{ lb}$$

Also,

$$R_z = \rho W_1^2 (\sin 45°) A_1 + \mathcal{W}_w$$

where

$$\mathcal{W}_w = \rho g A_1 \ell$$

Thus,

$$R_z = (1.94 \text{ slugs/ft}^3)(80 \text{ ft/s})^2(\sin 45°)(0.006 \text{ ft}^2) + (62.4 \text{ lb/ft}^3)(0.006 \text{ ft}^2)(1 \text{ ft})$$
$$= 52.6 \text{ lb} + 0.37 \text{ lb} = 53 \text{ lb}$$

Combining the components we get

$$R = \sqrt{R_x^2 + R_z^2} = [(21.8 \text{ lb})^2 + (53 \text{ lb})^2]^{1/2} = 57.3 \text{ lb}$$

The angle of **R** from the $x$ direction, $\alpha$, is

$$\alpha = \tan^{-1}\frac{R_z}{R_x} = \tan^{-1}(53 \text{ lb}/21.8 \text{ lb}) = 67.6°$$

The force of the water on the vane is equal in magnitude but opposite in direction from **R**; thus it points to the right and down at an angle of 67.6° from the $x$ direction and is equal in magnitude to 57.3 lb. **(Ans)**

**COMMENT** The force of the fluid on the vane in the $x$-direction, $R_x = 21.8$ lb, is associated with $x$-direction motion of the vane at a constant speed of 20 ft/s. Since the vane is not accelerating, this $x$-direction force is opposed mainly by a wheel friction force of the same magnitude. From basic physics we recall that the power this situation involves is the product of force and speed. Thus,

$$\mathcal{P} = R_x V_0 = \frac{(21.8 \text{ lb})(20 \text{ ft/s})}{550(\text{ft} \cdot \text{lb})/(\text{hp} \cdot \text{s})} = 0.79 \text{ hp}$$

All of this power is consumed by friction.

It is clear from the preceding examples that a flowing fluid can be forced to

1. change direction
2. speed up or slow down
3. have a velocity profile change
4. do only some or all of the above
5. do none of the above

A net force on the fluid is required for achieving any or all of the first four above. The forces on a flowing fluid balance out with no net force for the fifth.

Typical forces considered in this book include

(a) pressure
(b) friction
(c) weight

and involve some type of constraint such as a vane, channel, or conduit to guide the flowing fluid. A flowing fluid can cause a vane, channel or conduit to move. When this happens, power is produced.

The selection of a control volume is an important matter. For determining anchoring forces, consider including fluid and its constraint in the control volume. For determining force between a fluid and its constraint, consider including only the fluid in the control volume.

### 5.2.3 Derivation of the Moment-of-Momentum Equation[2]

In many engineering problems, the moment of a force with respect to an axis, namely, *torque*, is important. Newton's second law of motion has already led to a useful relationship between forces and linear momentum flow. The linear momentum equation can also be used to solve problems involving torques. However, by forming the moment of the linear momentum and the resultant force associated with each particle of fluid with respect to a point in an inertial coordinate system, we will develop a ***moment-of-momentum equation*** that relates *torques* and *angular momentum flow* for the contents of a control volume. When torques are important, the moment-of-momentum equation is often more convenient to use than the linear momentum equation.

*The angular momentum equation is derived from Newton's second law.*

Application of Newton's second law of motion to a particle of fluid yields

$$\frac{D}{Dt}(\mathbf{V}\rho\,\delta\forall) = \delta\mathbf{F}_{\text{particle}} \tag{5.30}$$

where $\mathbf{V}$ is the particle velocity measured in an inertial reference system, $\rho$ is the particle density, $\delta\forall$ is the infinitesimally small particle volume, and $\delta\mathbf{F}_{\text{particle}}$ is the resultant external force acting on the particle. If we form the moment of each side of Eq. 5.30 with respect to the origin of an inertial coordinate system, we obtain

$$\mathbf{r} \times \frac{D}{Dt}(\mathbf{V}\rho\,\delta\forall) = \mathbf{r} \times \delta\mathbf{F}_{\text{particle}} \tag{5.31}$$

where $\mathbf{r}$ is the position vector from the origin of the inertial coordinate system to the fluid particle (Fig. 5.3). We note that

$$\frac{D}{Dt}[(\mathbf{r} \times \mathbf{V})\rho\,\delta\forall] = \frac{D\mathbf{r}}{Dt} \times \mathbf{V}\rho\,\delta\forall + \mathbf{r} \times \frac{D(\mathbf{V}\rho\,\delta\forall)}{Dt} \tag{5.32}$$

and

$$\frac{D\mathbf{r}}{Dt} = \mathbf{V} \tag{5.33}$$

Thus, since

$$\mathbf{V} \times \mathbf{V} = 0 \tag{5.34}$$

by combining Eqs. 5.31, 5.32, 5.33, and 5.34, we obtain the expression

$$\frac{D}{Dt}[(\mathbf{r} \times \mathbf{V})\rho\,\delta\forall] = \mathbf{r} \times \delta\mathbf{F}_{\text{particle}} \tag{5.35}$$

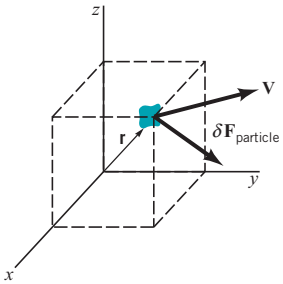



**FIGURE 5.3** Inertial coordinate system.

[2]This section may be omitted, along with Sections 5.2.4 and 5.3.5, without loss of continuity in the text material. However, these sections are recommended for those interested in Chapter 12.

Equation 5.35 is valid for every particle of a system. For a system (collection of fluid particles), we need to use the sum of both sides of Eq. 5.35 to obtain

$$\int_{\text{sys}} \frac{D}{Dt}[(\mathbf{r} \times \mathbf{V})\rho \, d\forall] = \sum (\mathbf{r} \times \mathbf{F})_{\text{sys}} \tag{5.36}$$

where

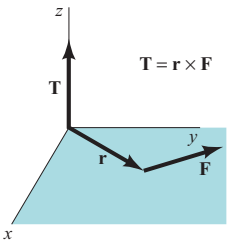
$$\sum \mathbf{r} \times \delta\mathbf{F}_{\text{particle}} = \sum (\mathbf{r} \times \mathbf{F})_{\text{sys}} \tag{5.37}$$

We note that

$$\frac{D}{Dt} \int_{\text{sys}} (\mathbf{r} \times \mathbf{V})\rho \, d\forall = \int_{\text{sys}} \frac{D}{Dt}[(\mathbf{r} \times \mathbf{V})\rho \, d\forall] \tag{5.38}$$

since the sequential order of differentiation and integration can be reversed without consequence. (Recall that the material derivative, $D(\ )/Dt$, denotes the time derivative following a given system; see Section 4.2.1.) Thus, from Eqs. 5.36 and 5.38 we get

$$\frac{D}{Dt} \int_{\text{sys}} (\mathbf{r} \times \mathbf{V})\rho \, d\forall = \sum (\mathbf{r} \times \mathbf{F})_{\text{sys}} \tag{5.39}$$




or

| | | |
|---|---|---|
| the time rate of change of the moment-of-momentum of the system | = | sum of external torques acting on the system |

The sketch in the margin illustrates what torque, $\mathbf{T} = \mathbf{r} \times \mathbf{F}$, is. For a control volume that is instantaneously coincident with the system, the torques acting on the system and on the control volume contents will be identical:

$$\sum (\mathbf{r} \times \mathbf{F})_{\text{sys}} = \sum (\mathbf{r} \times \mathbf{F})_{\text{cv}} \tag{5.40}$$

Further, for the system and the contents of the coincident control volume that is fixed and nondeforming, the Reynolds transport theorem (Eq. 4.19) leads to

$$\frac{D}{Dt} \int_{\text{sys}} (\mathbf{r} \times \mathbf{V})\rho \, d\forall = \frac{\partial}{\partial t} \int_{\text{cv}} (\mathbf{r} \times \mathbf{V})\rho \, d\forall + \int_{\text{cs}} (\mathbf{r} \times \mathbf{V})\rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA \tag{5.41}$$

or

| | | | | |
|---|---|---|---|---|
| time rate of change of the moment-of-momentum of the system | = | time rate of change of the moment-of-momentum of the contents of the control volume | + | net rate of flow of the moment-of-momentum through the control surface |

For a control volume that is fixed (and therefore inertial) and nondeforming, we combine Eqs. 5.39, 5.40, and 5.41 to obtain the moment-of-momentum equation:

*For a system, the rate of change of moment-of-momentum equals the net torque.*

$$\frac{\partial}{\partial t} \int_{\text{cv}} (\mathbf{r} \times \mathbf{V})\rho \, d\forall + \int_{\text{cs}} (\mathbf{r} \times \mathbf{V})\rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \sum (\mathbf{r} \times \mathbf{F})_{\substack{\text{contents of the} \\ \text{control volume}}} \tag{5.42}$$

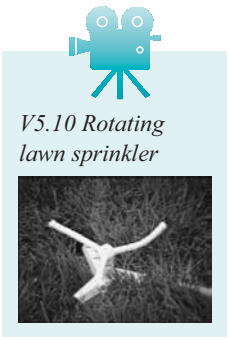
An important category of fluid mechanical problems that is readily solved with the help of the moment-of-momentum equation (Eq. 5.42) involves machines that rotate or tend to rotate around a single axis. Examples of these machines include rotary lawn sprinklers, ceiling fans, lawn mower blades, wind turbines, turbochargers, and gas turbine engines. As a class, these devices are often called turbomachines.

### 5.2.4 Application of the Moment-of-Momentum Equation[3]

We simplify our use of Eq. 5.42 in several ways:

1. We assume that flows considered are one-dimensional (uniform distributions of average velocity at any section).

[3]This section may be omitted, along with Sections 5.2.3 and 5.3.5, without loss of continuity in the text material. However, these sections are recommended for those interested in Chapter 12.



*V5.10 Rotating lawn sprinkler*

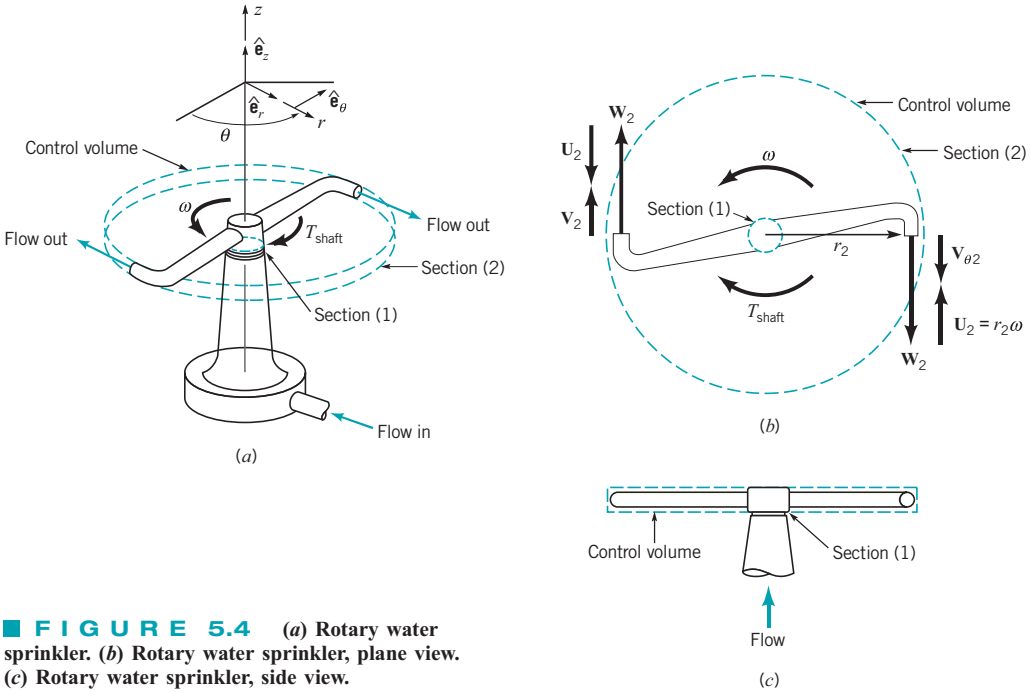



**FIGURE 5.4** **(*a*) Rotary water sprinkler. (*b*) Rotary water sprinkler, plane view. (*c*) Rotary water sprinkler, side view.**

**2.** We confine ourselves to steady or steady-in-the-mean cyclical flows. Thus,

$$\frac{\partial}{\partial t}\int_{cv}(\mathbf{r}\times\mathbf{V})\rho\, d\forall = 0$$

at any instant of time for steady flows or on a time-average basis for cyclical unsteady flows.

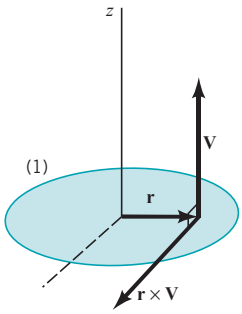
**3.** We work only with the component of Eq. 5.42 resolved along the axis of rotation.

*Change in moment of fluid velocity around an axis can result in torque and rotation around that same axis.*

Consider the rotating sprinkler sketched in Fig. 5.4. Because the direction and magnitude of the flow through the sprinkler from the inlet [section (1)] to the outlet [section (2)] of the arm changes, the water exerts a torque on the sprinkler head causing it to tend to rotate or to actually rotate in the direction shown, much like a turbine rotor. In applying the moment-of-momentum equation (Eq. 5.42) to this flow situation, we elect to use the fixed and nondeforming control volume shown in Fig. 5.4. This disk-shaped control volume contains within its boundaries the spinning or stationary sprinkler head and the portion of the water flowing through the sprinkler contained in the control volume at an instant. The control surface cuts through the sprinkler head's solid material so that the shaft torque that resists motion can be clearly identified. When the sprinkler is rotating, the flow field in the stationary control volume is cyclical and unsteady, but steady in the mean. We proceed to use the axial component of the moment-of-momentum equation (Eq. 5.42) to analyze this flow.

The integrand of the moment-of-momentum flow term in Eq. 5.42,

$$\int_{cs}(\mathbf{r}\times\mathbf{V})\rho\mathbf{V}\cdot\hat{\mathbf{n}}\, dA$$




can be nonzero only where fluid is crossing the control surface. Everywhere else on the control surface this term will be zero because $\mathbf{V}\cdot\hat{\mathbf{n}} = 0$. Water enters the control volume axially through the hollow stem of the sprinkler at section (1). At this portion of the control surface, the component of $\mathbf{r}\times\mathbf{V}$ resolved along the axis of rotation is zero because as illustrated by the figure in the margin, $\mathbf{r}\times\mathbf{V}$ lies in the plane of section (1), perpendicular to the axis of rotation. Thus, there is no axial moment-of-momentum flow in at section (1). Water leaves the control volume through each of the two nozzle openings at section (2). For the exiting flow, the magnitude of the axial component of $\mathbf{r}\times\mathbf{V}$ is $r_2V_{\theta 2}$, where $r_2$ is the radius from the axis of rotation to the nozzle centerline and $V_{\theta 2}$ is the value of the tangential component of the velocity of the flow exiting each nozzle as

CMS Cepcor EX 1046 Page 127
CMS Cepcor v. Sandvik IPR2024-01380

observed from a frame of reference attached to the fixed and nondeforming control volume. The fluid velocity measured relative to a fixed control surface is an absolute velocity, **V**. The velocity of the nozzle exit flow as viewed from the nozzle is called the relative velocity, **W**. The absolute and relative velocities, **V** and **W**, are related by the vector relationship

$$\mathbf{V} = \mathbf{W} + \mathbf{U} \tag{5.43}$$

where **U** is the velocity of the moving nozzle as measured relative to the fixed control surface.

The cross product and the dot product involved in the moment-of-momentum flow term of Eq. 5.42,

$$\int_{cs} (\mathbf{r} \times \mathbf{V})\rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$$

can each result in a positive or negative value. For flow into the control volume, $\mathbf{V} \cdot \hat{\mathbf{n}}$ is negative. For flow out, $\mathbf{V} \cdot \hat{\mathbf{n}}$ is positive. The correct algebraic sign to assign the axis component of $\mathbf{r} \times \mathbf{V}$ can be ascertained by using the right-hand rule. The positive direction along the axis of rotation is the direction the thumb of the right hand points when it is extended and the remaining fingers are curled around the rotation axis in the positive direction of rotation as illustrated in Fig. 5.5. The direction of the axial component of $\mathbf{r} \times \mathbf{V}$ is similarly ascertained by noting the direction of the cross product of the radius from the axis of rotation, $r\hat{\mathbf{e}}_r$, and the tangential component of absolute velocity, $V_\theta \hat{\mathbf{e}}_\theta$. Thus, for the sprinkler of Fig. 5.4, we can state that

*The algebraic sign of* $\mathbf{r} \times \mathbf{V}$ *is obtained by the right-hand rule.*

$$\left[ \int_{cs} (\mathbf{r} \times \mathbf{V})\rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA \right]_{axial} = (-r_2 V_{\theta 2})(+\dot{m}) \tag{5.44}$$

where, because of mass conservation, $\dot{m}$ is the total mass flowrate through both nozzles. As was demonstrated in Example 5.7, the mass flowrate is the same whether the sprinkler rotates or not. The correct algebraic sign of the axial component of $\mathbf{r} \times \mathbf{V}$ can be easily remembered in the following way: if $\mathbf{V}_\theta$ and **U** are in the same direction, use $+$; if $\mathbf{V}_\theta$ and **U** are in opposite directions, use $-$.

The torque term $[\sum (\mathbf{r} \times \mathbf{F})_{\text{contents of the control volume}}]$ of the moment-of-momentum equation (Eq. 5.42) is analyzed next. Confining ourselves to torques acting with respect to the axis of rotation only, we conclude that the shaft torque is important. The net torque with respect to the axis of rotation associated with normal forces exerted on the contents of the control volume will be very small if not zero. The net axial torque due to fluid tangential forces is also negligibly small for the control volume of Fig. 5.4. Thus, for the sprinkler of Fig. 5.4

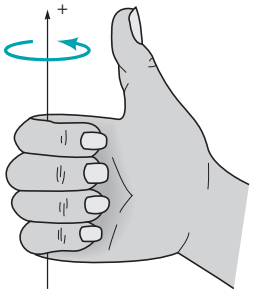
$$\sum \left[ (\mathbf{r} \times \mathbf{F})_{\substack{\text{contents of the} \\ \text{control volume}}} \right]_{axial} = \mathbf{T}_{shaft} \tag{5.45}$$

Note that we have entered $T_{shaft}$ as a positive quantity in Eq. 5.45. This is equivalent to assuming that $T_{shaft}$ is in the same direction as rotation.

For the sprinkler of Fig. 5.4, the axial component of the moment-of-momentum equation (Eq. 5.42) is, from Eqs. 5.44 and 5.45

$$-r_2 V_{\theta 2} \dot{m} = T_{shaft} \tag{5.46}$$

We interpret $T_{shaft}$ being a negative quantity from Eq. 5.46 to mean that the shaft torque actually opposes the rotation of the sprinkler arms as shown in Fig. 5.4. The shaft torque, $T_{shaft}$, opposes rotation in all turbine devices.




■ **FIGURE 5.5** **Right-hand rule convention.**

We could evaluate the ***shaft power***, $\dot{W}_{shaft}$, associated with ***shaft torque***, $T_{shaft}$, by forming the product of $T_{shaft}$ and the rotational speed of the shaft, $\omega$. [We use the notation that $W$ = work, $(\cdot) = d(\ )/dt$, and thus $\dot{W}$ = power.] Thus, from Eq. 5.46 we get

$$\dot{W}_{shaft} = T_{shaft}\,\omega = -r_2 V_{\theta 2}\dot{m}\,\omega \tag{5.47}$$

*Power is equal to angular velocity times torque.*

Since $r_2\omega$ is the speed of each sprinkler nozzle, $U$, we can also state Eq. 5.47 in the form

$$\dot{W}_{shaft} = -U_2 V_{\theta 2}\dot{m} \tag{5.48}$$



*V5.11 Impulse-type lawn sprinkler*



Shaft work per unit mass, $w_{shaft}$, is equal to $\dot{W}_{shaft}/\dot{m}$. Dividing Eq. 5.48 by the mass flowrate, $\dot{m}$, we obtain

$$w_{shaft} = -U_2 V_{\theta 2} \tag{5.49}$$

Negative shaft work as in Eqs. 5.47, 5.48, and 5.49 is work out of the control volume, that is, work done by the fluid on the rotor and thus its shaft.

The principles associated with this sprinkler example can be extended to handle most simplified turbomachine flows. The fundamental technique is not difficult. However, the geometry of some turbomachine flows is quite complicated.

Example 5.18 further illustrates how the axial component of the moment-of-momentum equation (Eq. 5.46) can be used.

## EXAMPLE 5.18 Moment-of-Momentum—Torque

**GIVEN** Water enters a rotating lawn sprinkler through its base at the steady rate of 1000 ml/s as sketched in Fig. E5.18*a*. The exit area of each of the two nozzles is 30 mm$^2$ and the flow leaving each nozzle is in the tangential direction. The radius from the axis of rotation to the centerline of each nozzle is 200 mm.

**FIND** **(a)** Determine the resisting torque required to hold the sprinkler head stationary.

**(b)** Determine the resisting torque associated with the sprinkler rotating with a constant speed of 500 rev/min.

**(c)** Determine the speed of the sprinkler if no resisting torque is applied.

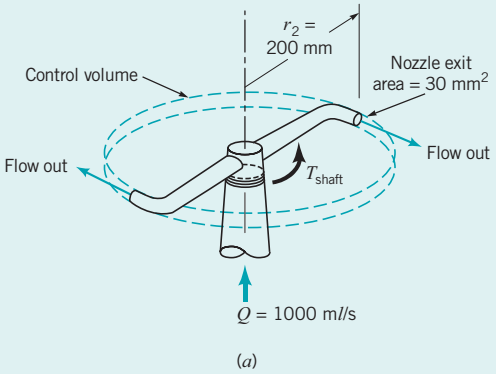



**FIGURE E5.18**

### SOLUTION

To solve parts **(a)**, **(b)**, and **(c)** of this example we can use the same fixed and nondeforming, disk-shaped control volume illustrated in Fig. 5.4. As indicated in Fig. E5.18*a*, the only axial torque considered is the one resisting motion, $T_{shaft}$.

**(a)** When the sprinkler head is held stationary as specified in part **(a)** of this example problem, the velocities of the fluid entering and leaving the control volume are shown in Fig. E5.18*b*. Equation 5.46 applies to the contents of this control volume. Thus,

$$T_{shaft} = -r_2 V_{\theta 2}\dot{m} \tag{1}$$

Since the control volume is fixed and nondeforming and the flow exiting from each nozzle is tangential,

$$V_{\theta 2} = V_2 \tag{2}$$

Equations 1 and 2 give

$$T_{shaft} = -r_2 V_2 \dot{m} \tag{3}$$

In Example 5.7, we ascertained that $V_2 = 16.7$ m/s. Thus, from Eq. 3 with

$$\dot{m} = Q\rho = \frac{(1000 \text{ ml/s})(10^{-3} \text{ m}^3/\text{liter})(999 \text{ kg/m}^3)}{(1000 \text{ ml/liter})}$$
$$= 0.999 \text{ kg/s}$$

we obtain

$$T_{\text{shaft}} = -\frac{(200 \text{ mm})(16.7 \text{ m/s})(0.999 \text{ kg/s})[1 (\text{N/kg})/(\text{m/s}^2)]}{(1000 \text{ mm/m})}$$

or

$$T_{\text{shaft}} = -3.34 \text{ N} \cdot \text{m} \qquad \textbf{(Ans)}$$

**(b)** When the sprinkler is rotating at a constant speed of 500 rpm, the flow field in the control volume is unsteady but cyclical. Thus, the flow field is steady in the mean. The velocities of the flow entering and leaving the control volume are as indicated in Fig. E5.18*c*. The absolute velocity of the fluid leaving each nozzle, $V_2$, is from Eq. 5.43,

$$V_2 = W_2 - U_2 \qquad \textbf{(4)}$$

where

$$W_2 = 16.7 \text{ m/s}$$

as determined in Example 5.7. The speed of the nozzle, $U_2$, is obtained from

$$U_2 = r_2\omega \qquad \textbf{(5)}$$

Application of the axial component of the moment-of-momentum equation (Eq. 5.46) leads again to Eq. 3. From Eqs. 4 and 5,

$$V_2 = 16.7 \text{ m/s} - r_2\omega$$
$$= 16.7 \text{ m/s} - \frac{(200 \text{ mm})(500 \text{ rev/min})(2\pi \text{ rad/rev})}{(1000 \text{ mm/m})(60 \text{ s/min})}$$

or

$$V_2 = 16.7 \text{ m/s} - 10.5 \text{ m/s} = 6.2 \text{ m/s}$$

Thus, using Eq. 3, with $\dot{m} = 0.999$ kg/s (as calculated previously), we get

$$T_{\text{shaft}} = -\frac{(200 \text{ mm})(6.2 \text{ m/s})\ 0.999 \text{ kg/s}\ [1 (\text{N/kg})/(\text{m/s}^2)]}{(1000 \text{ mm/m})}$$

or

$$T_{\text{shaft}} = -1.24 \text{ N} \cdot \text{m} \qquad \textbf{(Ans)}$$

**COMMENT** Note that the resisting torque associated with sprinkler head rotation is much less than the resisting torque that is required to hold the sprinkler stationary.

**(c)** When no resisting torque is applied to the rotating sprinkler head, a maximum constant speed of rotation will occur as demonstrated below. Application of Eqs. 3, 4, and 5 to the contents of the control volume results in

$$T_{\text{shaft}} = -r_2(W_2 - r_2\omega)\dot{m} \qquad \textbf{(6)}$$

For no resisting torque, Eq. 6 yields

$$0 = -r_2(W_2 - r_2\omega)\dot{m}$$

Thus,

$$\omega = \frac{W_2}{r_2} \qquad \textbf{(7)}$$

In Example 5.4, we learned that the relative velocity of the fluid leaving each nozzle, $W_2$, is the same regardless of the speed of rotation of the sprinkler head, $\omega$, as long as the mass flowrate of the fluid, $\dot{m}$, remains constant. Thus, by using Eq. 7 we obtain

$$\omega = \frac{W_2}{r_2} = \frac{(16.7 \text{ m/s})(1000 \text{ mm/m})}{(200 \text{ mm})} = 83.5 \text{ rad/s}$$

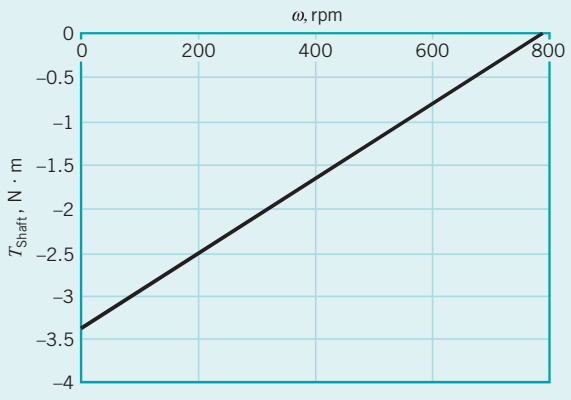
or

$$\omega = \frac{(83.5 \text{ rad/s})(60 \text{ s/min})}{2\ \pi \text{ rad/rev}} = 797 \text{ rpm} \qquad \textbf{(Ans)}$$

For this condition ($T_{\text{shaft}} = 0$), the water both enters and leaves the control volume with zero angular momentum.

**COMMENT** Note that forcing a change in direction of a flowing fluid, in this case with a sprinkler, resulted in rotary motion and a useful "sprinkling" of water over an area.

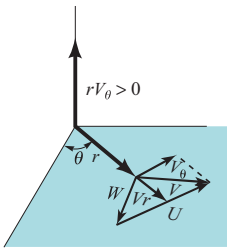
By repeating the calculations for various values of the angular velocity, $\omega$, the results shown in Fig. E5.18*d* are obtained. It is seen that the magnitude of the resisting torque associated with rotation is less than the torque required to hold the rotor stationary. Even in the absence of a resisting torque, the rotor maximum speed is finite.

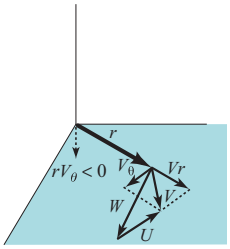



■ FIGURE E5.18*d*

When the moment-of-momentum equation (Eq. 5.42) is applied to a more general, one-dimensional flow through a rotating machine, we obtain

$$\boxed{T_{\text{shaft}} = (-\dot{m}_{\text{in}})(\pm r_{\text{in}} V_{\theta\text{in}}) + \dot{m}_{\text{out}}(\pm r_{\text{out}} V_{\theta\text{out}})} \qquad \textbf{(5.50)}$$




*When shaft torque and shaft rotation are in the same (opposite) direction, power is into (out of) the fluid.*

by applying the same kind of analysis used with the sprinkler of Fig. 5.4. The "−" is used with mass flowrate into the control volume, $\dot{m}_{in}$, and the "+" is used with mass flowrate out of the control volume, $\dot{m}_{out}$, to account for the sign of the dot product, $\mathbf{V} \cdot \hat{\mathbf{n}}$, involved. Whether "+" or "−" is used with the $rV_\theta$ product depends on the direction of $(\mathbf{r} \times \mathbf{V})_{axial}$. A simple way to determine the sign of the $rV_\theta$ product is to compare the direction of $V_\theta$ and the blade speed, $U$. As shown in the margin, if $V_\theta$ and $U$ are in the same direction, then the $rV_\theta$ product is positive. If $V_\theta$ and $U$ are in opposite directions, the $rV_\theta$ product is negative. The sign of the shaft torque is "+" if $T_{shaft}$ is in the same direction along the axis of rotation as $\omega$, and "−" otherwise.

The shaft power, $\dot{W}_{shaft}$, is related to shaft torque, $T_{shaft}$, by

$$\dot{W}_{shaft} = T_{shaft}\,\omega \tag{5.51}$$

Thus, using Eqs. 5.50 and 5.51 with a "+" sign for $T_{shaft}$ in Eq. 5.50, we obtain

$$\dot{W}_{shaft} = (-\dot{m}_{in})(\pm r_{in}\omega V_{\theta in}) + \dot{m}_{out}(\pm r_{out}\omega V_{\theta out}) \tag{5.52}$$

or since $r\omega = U$

$$\boxed{\dot{W}_{shaft} = (-\dot{m}_{in})(\pm U_{in} V_{\theta in}) + \dot{m}_{out}(\pm U_{out} V_{\theta out})} \tag{5.53}$$

The "+" is used for the $UV_\theta$ product when $U$ and $V_\theta$ are in the same direction; the "−" is used when $U$ and $V_\theta$ are in opposite directions. Also, since $+T_{shaft}$ was used to obtain Eq. 5.53, when $\dot{W}_{shaft}$ is positive, power is into the fluid (for example, a pump), and when $\dot{W}_{shaft}$ is negative, power is out of the fluid (for example, a turbine).

The shaft work per unit mass, $w_{shaft}$, can be obtained from the shaft power, $\dot{W}_{shaft}$, by dividing Eq. 5.53 by the mass flowrate, $\dot{m}$. By conservation of mass,

$$\dot{m} = \dot{m}_{in} = \dot{m}_{out}$$

From Eq. 5.53, we obtain

$$\boxed{w_{shaft} = -(\pm U_{in} V_{\theta in}) + (\pm U_{out} V_{\theta out})} \tag{5.54}$$

The application of Eqs. 5.50, 5.53, and 5.54 is demonstrated in Example 5.19. More examples of the application of Eqs. 5.50, 5.53, and 5.54 are included in Chapter 12.

## EXAMPLE 5.19 Moment-of-Momentum—Power

**GIVEN** An air fan has a bladed rotor of 12-in. outside diameter and 10-in. inside diameter as illustrated in Fig. E5.19*a*. The height of each rotor blade is constant at 1 in. from blade inlet to outlet. The flowrate is steady, on a time-average basis, at 230 ft$^3$/min and the absolute velocity of the air at blade inlet, $\mathbf{V}_1$, is radial. The blade discharge angle is 30° from the tangential direction. The rotor rotates at a constant speed of 1725 rpm.

**FIND** Estimate the power required to run the fan.

### SOLUTION

We select a fixed and nondeforming control volume that includes the rotating blades and the fluid within the blade row at an instant, as shown with a dashed line in Fig. E5.19*a*. The flow within this control volume is cyclical, but steady in the mean. The only torque we consider is the driving shaft torque, $T_{shaft}$. This torque is provided by a motor. We assume that the entering and leaving flows are each represented by uniformly distributed velocities and flow properties. Since shaft power is sought, Eq. 5.53 is appropriate. Application of Eq. 5.53 to the contents of the control volume in Fig. E5.19 gives

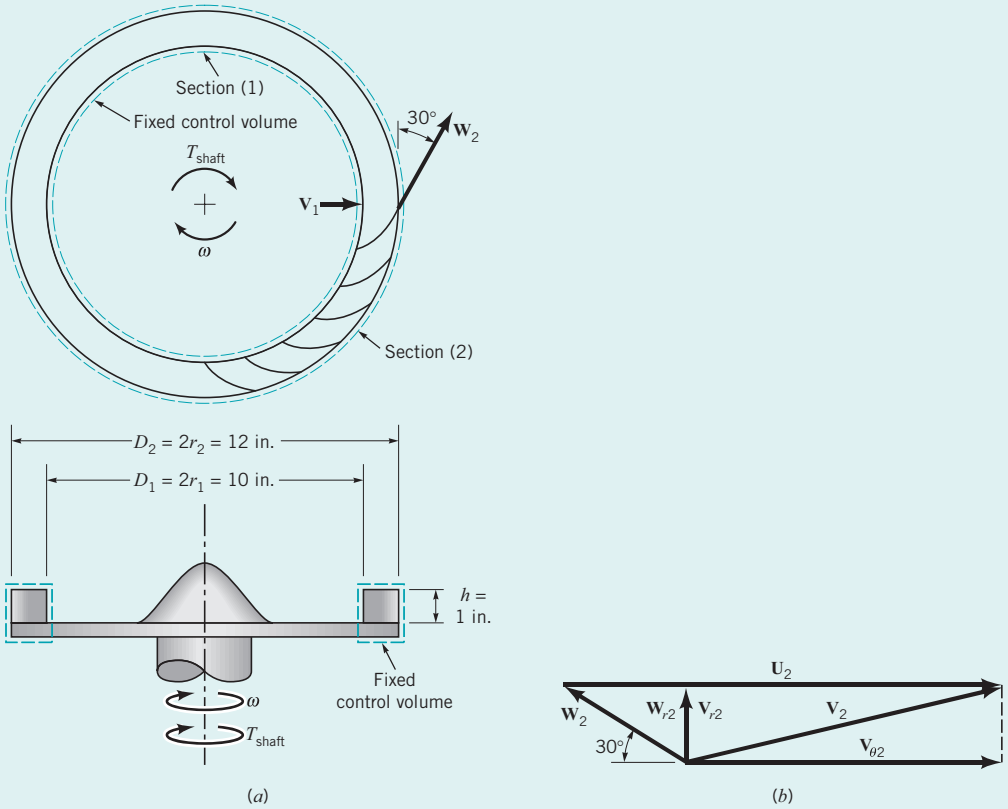
$$\dot{W}_{shaft} = -\dot{m}_1(\pm U_1 \cancel{V_{\theta 1}}^{\,0\ (\mathbf{V}_1 \text{ is radial})}) + \dot{m}_2(\pm U_2 V_{\theta 2}) \tag{1}$$

From Eq. 1 we see that to calculate fan power, we need mass flowrate, $\dot{m}$, rotor exit blade velocity, $U_2$, and fluid tangential velocity at blade exit, $V_{\theta 2}$. The mass flowrate, $\dot{m}$, is easily obtained from Eq. 5.6 as

$$\dot{m} = \rho Q = \frac{(2.38 \times 10^{-3}\ \text{slug/ft}^3)(230\ \text{ft}^3\text{/min})}{(60\ \text{s/min})}$$

$$= 0.00912\ \text{slug/s} \tag{2}$$

Often, problems involving fans are solved using English Engineering units. Since 1slug = 32.174 lbm, we could have used as the density of air $\rho_{air} = (2.38 \times 10^{-3}\ \text{slug/ft}^3)(32.174\text{lbm/slug}) = 0.0766\ \text{lbm/ft}^3$.




■ FIGURE E5.19

Then

$$\dot{m} = \frac{(0.0766 \text{ lbm/ft}^3)(230 \text{ ft}^3/\text{min})}{(60 \text{ s/min})} = 0.294 \text{ lbm/s}$$

The rotor exit blade speed, $U_2$, is

$$U_2 = r_2\omega = \frac{(6 \text{ in.})(1725 \text{ rpm})(2\pi \text{ rad/rev})}{(12 \text{ in./ft})(60 \text{ s/min})} = 90.3 \text{ ft/s} \quad (3)$$

To determine the fluid tangential speed at the fan rotor exit, $V_{\theta 2}$, we use Eq. 5.43 to get

$$\mathbf{V}_2 = \mathbf{W}_2 + \mathbf{U}_2 \quad (4)$$

The vector addition of Eq. 4 is shown in the form of a "velocity triangle" in Fig. E5.19*b*. From Fig. E5.19*b,* we can see that

$$V_{\theta 2} = U_2 - W_2 \cos 30° \quad (5)$$

To solve Eq. 5 for $V_{\theta 2}$ we need a value of $W_2$, in addition to the value of $U_2$ already determined (Eq. 3). To get $W_2$, we recognize that

$$W_2 \sin 30° = V_{r2} \quad (6)$$

where $V_{r2}$ is the radial component of either $\mathbf{W}_2$ or $\mathbf{V}_2$. Also, using Eq. 5.6, we obtain

$$\dot{m} = \rho A_2 V_{r2} \quad (7)$$

or since

$$A_2 = 2\pi r_2 h \quad (8)$$

where $h$ is the blade height, Eqs. 7 and 8 combine to form

$$\dot{m} = \rho 2\pi r_2 h V_{r2} \quad (9)$$

Taking Eqs. 6 and 9 together we get

$$W_2 = \frac{\dot{m}}{\rho 2\pi r_2 h \sin 30°} = \frac{\rho Q}{\rho 2\pi r_2 h \sin 30°} = \frac{Q}{2\pi r_2 h \sin 30°} \quad (10)$$

Substituting known values into Eq. 10, we obtain

$$W_2 = \frac{(230 \text{ ft}^3/\text{min})(12 \text{ in./ft})(12 \text{ in./ft})}{(60 \text{ s/min})2\pi(6 \text{ in.})(1 \text{ in.}) \sin 30°} = 29.3 \text{ ft/s}$$

By using this value of $W_2$ in Eq. 5 we get

$$V_{\theta 2} = U_2 - W_2 \cos 30° = 90.3 \text{ ft/s} - (29.3 \text{ ft/s})(0.866) = 64.9 \text{ ft/s}$$

Equation 1 can now be used to obtain

$$\dot{W}_{shaft} = \dot{m} U_2 V_{\theta 2} = \frac{(0.00912 \text{ slug/s})(90.3 \text{ ft/s})(64.9 \text{ ft/s})}{[1(\text{slug} \cdot \text{ft/s}^2)/\text{lb}][550(\text{ft} \cdot \text{lb})/(\text{hp} \cdot \text{s})]}$$

with BG units.

With EE units

$$\dot{W}_{shaft} = \frac{(0.294 \text{ lbm/s})(90.3 \text{ ft/s})(64.9 \text{ ft/s})}{[32.174 (\text{lbm} \cdot \text{ft})/(\text{lb/s}^2)][550 (\text{ft} \cdot \text{lb})/(\text{hp} \cdot \text{s})]}$$

In either case

$$\dot{W}_{\text{shaft}} = 0.097 \text{ hp} \qquad \textbf{(Ans)}$$

**COMMENT** Note that the "+" was used with the $U_2 V_{\theta 2}$ product because $U_2$ and $V_{\theta 2}$ are in the same direction. This result, 0.097 hp, is the power that needs to be delivered through the fan shaft for the given conditions. Ideally, all of this power would go into the flowing air. However, because of fluid friction, only some of this power will produce useful effects (e.g., movement and pressure rise) on the air. How much useful effect depends on the efficiency of the energy transfer between the fan blades and the fluid.

## 5.3 First Law of Thermodynamics—The Energy Equation

### 5.3.1 Derivation of the Energy Equation

The ***first law of thermodynamics*** for a system is, in words

| time rate of increase of the total stored energy of the system | = | net time rate of energy addition by heat transfer into the system | + | net time rate of energy addition by work transfer into the system |
|---|---|---|---|---|

In symbolic form, this statement is

$$\frac{D}{Dt}\int_{\text{sys}} e\rho \, d\forall = \left(\sum \dot{Q}_{\text{in}} - \sum \dot{Q}_{\text{out}}\right)_{\text{sys}} + \left(\sum \dot{W}_{\text{in}} - \sum \dot{W}_{\text{out}}\right)_{\text{sys}}$$

*The first law of thermodynamics is a statement of conservation of energy.*

or

$$\frac{D}{Dt}\int_{\text{sys}} e\rho \, d\forall = (\dot{Q}_{\substack{\text{net}\\ \text{in}}} + \dot{W}_{\substack{\text{net}\\ \text{in}}})_{\text{sys}} \tag{5.55}$$

Some of these variables deserve a brief explanation before proceeding further. The total stored energy per unit mass for each particle in the system, $e$, is related to the internal energy per unit mass, $\breve{u}$, the kinetic energy per unit mass, $V^2/2$, and the potential energy per unit mass, $gz$, by the equation

$$e = \breve{u} + \frac{V^2}{2} + gz \tag{5.56}$$

The net ***rate of heat transfer*** into the system is denoted with $\dot{Q}_{\text{net in}}$, and the net rate of work transfer into the system is labeled $\dot{W}_{\text{net in}}$. Heat transfer and work transfer are considered "+" going into the system and "−" coming out.

Equation 5.55 is valid for inertial and noninertial reference systems. We proceed to develop the control volume statement of the first law of thermodynamics. For the control volume that is coincident with the system at an instant of time

$$(\dot{Q}_{\substack{\text{net}\\ \text{in}}} + \dot{W}_{\substack{\text{net}\\ \text{in}}})_{\text{sys}} = (\dot{Q}_{\substack{\text{net}\\ \text{in}}} + \dot{W}_{\substack{\text{net}\\ \text{in}}})_{\substack{\text{coincident}\\ \text{control volume}}} \tag{5.57}$$

Furthermore, for the system and the contents of the coincident control volume that is fixed and nondeforming, the Reynolds transport theorem (Eq. 4.19 with the parameter $b$ set equal to $e$) allows us to conclude that

$$\frac{D}{Dt}\int_{\text{sys}} e\rho \, d\forall = \frac{\partial}{\partial t}\int_{\text{cv}} e\rho \, d\forall + \int_{\text{cs}} e\rho \mathbf{V}\cdot\hat{\mathbf{n}}\, dA \tag{5.58}$$

or in words,

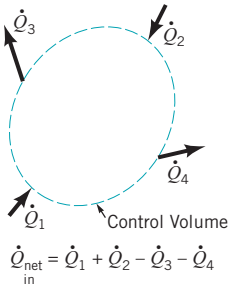
| the time rate of increase of the total stored energy of the system | = | the time rate of increase of the total stored energy of the contents of the control volume | + | the net rate of flow of the total stored energy out of the control volume through the control surface |
|---|---|---|---|---|

*The energy equation involves stored energy and heat and work transfer.*

Combining Eqs. 5.55, 5.57, and 5.58 we get the control volume formula for the first law of thermodynamics:

$$\frac{\partial}{\partial t}\int_{cv} e\rho \, d\forall + \int_{cs} e\rho \mathbf{V}\cdot\hat{\mathbf{n}}\, dA = (\dot{Q}_{\substack{net\\in}} + \dot{W}_{\substack{net\\in}})_{cv} \tag{5.59}$$

The total stored energy per unit mass, $e$, in Eq. 5.59 is for fluid particles entering, leaving, and within the control volume. Further explanation of the heat transfer and work transfer involved in this equation follows.




The heat transfer rate, $\dot{Q}$, represents all of the ways in which energy is exchanged between the control volume contents and surroundings because of a temperature difference. Thus, radiation, conduction, and/or convection are possible. As shown by the figure in the margin, heat transfer into the control volume is considered positive, heat transfer out is negative. In many engineering applications, the process is *adiabatic*; the heat transfer rate, $\dot{Q}$, is zero. The net heat transfer rate, $\dot{Q}_{net\ in}$, can also be zero when $\sum \dot{Q}_{in} - \sum \dot{Q}_{out} = 0$.

The work transfer rate, $\dot{W}$, also called *power*, is positive when work is done on the contents of the control volume by the surroundings. Otherwise, it is considered negative. Work can be transferred across the control surface in several ways. In the following paragraphs, we consider some important forms of work transfer.

In many instances, work is transferred across the control surface by a moving shaft. In rotary devices such as turbines, fans, and propellers, a rotating shaft transfers work across that portion of the control surface that slices through the shaft. Even in reciprocating machines like positive displacement internal combustion engines and compressors that utilize piston-in-cylinder arrangements, a rotating crankshaft is used. Since work is the dot product of force and related displacement, rate of work (or power) is the dot product of force and related displacement per unit time. For a rotating shaft, the power transfer, $\dot{W}_{shaft}$, is related to the shaft torque that causes the rotation, $T_{shaft}$, and the angular velocity of the shaft, $\omega$, by the relationship

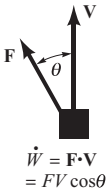
$$\dot{W}_{shaft} = T_{shaft}\omega$$

When the control surface cuts through the shaft material, the shaft torque is exerted by shaft material at the control surface. To allow for consideration of problems involving more than one shaft we use the notation

$$\dot{W}_{\substack{shaft\\net\ in}} = \sum \dot{W}_{\substack{shaft\\in}} - \sum \dot{W}_{\substack{shaft\\out}} \tag{5.60}$$

Work transfer can also occur at the control surface when a force associated with fluid normal stress acts over a distance. Consider the simple pipe flow illustrated in Fig. 5.6 and the control volume shown. For this situation, the fluid normal stress, $\sigma$, is simply equal to the negative of fluid pressure, $p$, in all directions; that is,

$$\sigma = -p \tag{5.61}$$




This relationship can be used with varying amounts of approximation for many engineering problems (see Chapter 6).

The power transfer, $\dot{W}$, associated with a force $\mathbf{F}$ acting on an object moving with velocity $\mathbf{V}$ is given by the dot product $\mathbf{F}\cdot\mathbf{V}$. This is illustrated by the figure in the margin. Hence, the power transfer associated with normal stresses acting on a single fluid particle, $\delta\dot{W}_{normal\ stress}$, can be evaluated as the dot product of the normal stress force, $\delta\mathbf{F}_{normal\ stress}$, and the fluid particle velocity, $\mathbf{V}$, as

$$\delta\dot{W}_{normal\ stress} = \delta\mathbf{F}_{normal\ stress}\cdot\mathbf{V}$$

If the normal stress force is expressed as the product of local normal stress, $\sigma = -p$, and fluid particle surface area, $\hat{\mathbf{n}}\,\delta A$, the result is

$$\delta\dot{W}_{normal\ stress} = \sigma\hat{\mathbf{n}}\,\delta A\cdot\mathbf{V} = -p\hat{\mathbf{n}}\,\delta A\cdot\mathbf{V} = -p\mathbf{V}\cdot\hat{\mathbf{n}}\,\delta A$$

For all fluid particles on the control surface of Fig. 5.6 at the instant considered, power transfer due to fluid normal stress, $\dot{W}_{normal\ stress}$, is

$$\dot{W}_{\substack{normal\\stress}} = \int_{cs}\sigma\mathbf{V}\cdot\hat{\mathbf{n}}\, dA = \int_{cs} -p\mathbf{V}\cdot\hat{\mathbf{n}}\, dA \tag{5.62}$$

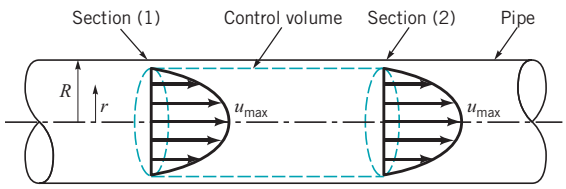



**FIGURE 5.6** Simple, fully developed pipe flow.

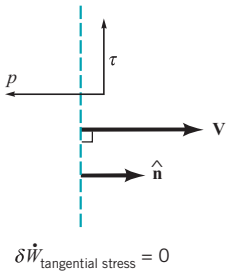
Note that the value of $\dot{W}_{\text{normal stress}}$ for particles on the wetted inside surface of the pipe is zero because $\mathbf{V} \cdot \hat{\mathbf{n}}$ is zero there. Thus, $\dot{W}_{\text{normal stress}}$ can be nonzero only where fluid enters and leaves the control volume. Although only a simple pipe flow was considered, Eq. 5.62 is quite general and the control volume used in this example can serve as a general model for other cases.

*Work is transferred by rotating shafts, normal stresses, and tangential stresses.*

Work transfer can also occur at the control surface because of tangential stress forces. Rotating shaft work is transferred by tangential stresses in the shaft material. For a fluid particle, shear stress force power, $\delta\dot{W}_{\text{tangential stress}}$, can be evaluated as the dot product of tangential stress force, $\delta\mathbf{F}_{\text{tangential stress}}$, and the fluid particle velocity, $\mathbf{V}$. That is,

$$\delta\dot{W}_{\text{tangential stress}} = \delta\mathbf{F}_{\text{tangential stress}} \cdot \mathbf{V}$$

For the control volume of Fig. 5.6, the fluid particle velocity is zero everywhere on the wetted inside surface of the pipe. Thus, no tangential stress work is transferred across that portion of the control surface. Furthermore, if we select the control surface so that it is perpendicular to the fluid particle velocity, then the tangential stress force is also perpendicular to the velocity. Therefore, the tangential stress work transfer is zero on that part of the control surface. This is illustrated in the figure in the margin. Thus, in general, we select control volumes like the one of Fig. 5.6 and consider fluid tangential stress power transfer to be negligibly small.




Using the information we have developed about power, we can express the first law of thermodynamics for the contents of a control volume by combining Eqs. 5.59, 5.60, and 5.62 to obtain

$$\frac{\partial}{\partial t}\int_{cv} e\rho \, d\forall + \int_{cs} e\rho\mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \dot{Q}_{\substack{\text{net}\\\text{in}}} + \dot{W}_{\substack{\text{shaft}\\\text{net in}}} - \int_{cs} p\mathbf{V} \cdot \hat{\mathbf{n}} \, dA \qquad \textbf{(5.63)}$$

When the equation for total stored energy (Eq. 5.56) is considered with Eq. 5.63, we obtain the ***energy equation***:

$$\boxed{\frac{\partial}{\partial t}\int_{cv} e\rho \, d\forall + \int_{cs}\left(\breve{u} + \frac{p}{\rho} + \frac{V^2}{2} + gz\right)\rho\mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \dot{Q}_{\substack{\text{net}\\\text{in}}} + \dot{W}_{\substack{\text{shaft}\\\text{net in}}}} \qquad \textbf{(5.64)}$$

## 5.3.2 Application of the Energy Equation

In Eq. 5.64, the term $\partial/\partial t \int_{cv} e\rho \, d\forall$ represents the time rate of change of the total stored energy, $e$, of the contents of the control volume. This term is zero when the flow is steady. This term is also zero in the mean when the flow is steady in the mean (cyclical).

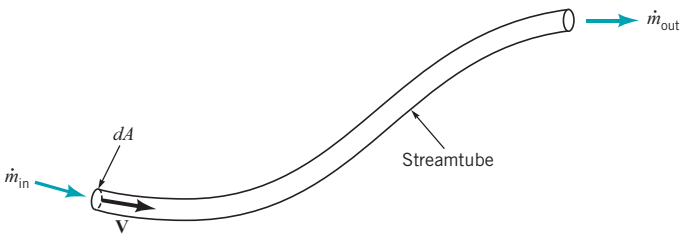
In Eq. 5.64, the integrand of

$$\int_{cs}\left(\breve{u} + \frac{p}{\rho} + \frac{V^2}{2} + gz\right)\rho\mathbf{V} \cdot \hat{\mathbf{n}} \, dA$$

can be nonzero only where fluid crosses the control surface ($\mathbf{V} \cdot \hat{\mathbf{n}} \neq 0$). Otherwise, $\mathbf{V} \cdot \hat{\mathbf{n}}$ is zero and the integrand is zero for that portion of the control surface. If the properties within parentheses, $\breve{u}$, $p/\rho$, $V^2/2$, and $gz$, are all assumed to be uniformly distributed over the flow cross-sectional areas involved, the integration becomes simple and gives

$$\int_{cs}\left(\breve{u} + \frac{p}{\rho} + \frac{V^2}{2} + gz\right)\rho\mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \sum_{\substack{\text{flow}\\\text{out}}}\left(\breve{u} + \frac{p}{\rho} + \frac{V^2}{2} + gz\right)\dot{m} - \sum_{\substack{\text{flow}\\\text{in}}}\left(\breve{u} + \frac{p}{\rho} + \frac{V^2}{2} + gz\right)\dot{m} \qquad \textbf{(5.65)}$$

CMS Cepcor EX 1046 Page 135
CMS Cepcor v. Sandvik IPR2024-01380




■ FIGURE 5.7
Streamtube flow.

Furthermore, if there is only one stream entering and leaving the control volume, then

$$\int_{cs}\left(\breve{u}+\frac{p}{\rho}+\frac{V^2}{2}+gz\right)\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA=\left(\breve{u}+\frac{p}{\rho}+\frac{V^2}{2}+gz\right)_{out}\dot{m}_{out}-\left(\breve{u}+\frac{p}{\rho}+\frac{V^2}{2}+gz\right)_{in}\dot{m}_{in} \tag{5.66}$$

Uniform flow as described above will occur in an infinitesimally small diameter streamtube as illustrated in Fig. 5.7. This kind of streamtube flow is representative of the steady flow of a particle of fluid along a pathline. We can also idealize actual conditions by disregarding nonuniformities in a finite cross section of flow. We call this one-dimensional flow and although such uniform flow rarely occurs in reality, the simplicity achieved with the one-dimensional approximation often justifies its use. More details about the effects of nonuniform distributions of velocities and other fluid flow variables are considered in Section 5.3.4 and in Chapters 8, 9, and 10.

If shaft work is involved, the flow must be unsteady, at least locally (see Refs. 1 and 2). The flow in any fluid machine that involves shaft work is unsteady within that machine. For example, the velocity and pressure at a fixed location near the rotating blades of a fan are unsteady. However, upstream and downstream of the machine, the flow may be steady. Most often shaft work is associated with flow that is unsteady in a recurring or cyclical way. On a time-average basis for flow that is one-dimensional, cyclical, and involves only one stream of fluid entering and leaving the control volume, Eq. 5.64 can be simplified with the help of **Eqs. 5.9** and 5.66 to form

$$\dot{m}\left[\breve{u}_{out}-\breve{u}_{in}+\left(\frac{p}{\rho}\right)_{out}-\left(\frac{p}{\rho}\right)_{in}+\frac{V_{out}^2-V_{in}^2}{2}+g(z_{out}-z_{in})\right]=\dot{Q}_{\substack{net\\in}}+\dot{W}_{\substack{shaft\\net\ in}} \tag{5.67}$$

We call Eq. 5.67 the *one-dimensional energy equation for steady-in-the-mean flow*. Note that Eq. 5.67 is valid for incompressible and compressible flows. Often, the fluid property called *enthalpy*, $\breve{h}$, where

$$\breve{h}=\breve{u}+\frac{p}{\rho} \tag{5.68}$$

*The energy equation is sometimes written in terms of enthalpy.*

is used in Eq. 5.67. With enthalpy, the one-dimensional energy equation for steady-in-the-mean flow (Eq. 5.67) is

$$\dot{m}\left[\breve{h}_{out}-\breve{h}_{in}+\frac{V_{out}^2-V_{in}^2}{2}+g(z_{out}-z_{in})\right]=\dot{Q}_{\substack{net\\in}}+\dot{W}_{\substack{shaft\\net\ in}} \tag{5.69}$$

Equation 5.69 is often used for solving compressible flow problems. Examples 5.20 and 5.21 illustrate how Eqs. 5.67 and 5.69 can be used.

## EXAMPLE 5.20 Energy—Pump Power

**GIVEN** A pump delivers water at a steady rate of 300 gal/min as shown in Fig. E5.20. Just upstream of the pump [section (1)] where the pipe diameter is 3.5 in., the pressure is 18 psi. Just downstream of the pump [section (2)] where the pipe diameter is 1 in., the pressure is 60 psi. The change in water elevation across the pump is zero. The rise in internal energy of water, $\breve{u}_2-\breve{u}_1$, associated with a temperature rise across the pump is 93 ft · lb/lbm. The pumping process is considered to be adiabatic.

**FIND** Determine the power (hp) required by the pump.

## SOLUTION

We include in our control volume the water contained in the pump between its entrance and exit sections. Application of Eq. 5.67 to the contents of this control volume on a time-average basis yields

$$\dot{m}\left[\breve{u}_2 - \breve{u}_1 + \left(\frac{p}{\rho}\right)_2 - \left(\frac{p}{\rho}\right)_1 + \frac{V_2^2 - V_1^2}{2} + g(z_2 - z_1)\right] = \dot{Q}_{\substack{\text{net}\\\text{in}}} + \dot{W}_{\substack{\text{shaft}\\\text{net in}}} \qquad (1)$$

(0 (no elevation change); 0 (adiabatic flow))

We can solve directly for the power required by the pump, $\dot{W}_{\text{shaft net in}}$, from Eq. 1, after we first determine the mass flowrate, $\dot{m}$, the speed of flow into the pump, $V_1$, and the speed of the flow out of the pump, $V_2$. All other quantities in Eq. 1 are given in the problem statement. From Eq. 5.6, we get

$$\dot{m} = \rho Q = \frac{(1.94 \text{ slugs/ft}^3)(300 \text{ gal/min})(32.174 \text{ lbm/slug})}{(7.48 \text{ gal/ft}^3)(60 \text{ s/min})} = 41.8 \text{ lbm/s} \qquad (2)$$

Also from Eq. 5.6,

$$V = \frac{Q}{A} = \frac{Q}{\pi D^2/4}$$

so

$$V_1 = \frac{Q}{A_1} = \frac{(300 \text{ gal/min})4(12 \text{ in./ft})^2}{(7.48 \text{ gal/ft}^3)(60 \text{ s/min})\pi(3.5 \text{ in.})^2} = 10.0 \text{ ft/s} \qquad (3)$$

and

$$V_2 = \frac{Q}{A_2} = \frac{(300 \text{ gal/min})4\,(12 \text{ in./ft})^2}{(7.48 \text{ gal/ft}^3)(60 \text{ s/min})\pi(1 \text{ in.})^2} = 123 \text{ ft/s} \qquad (4)$$

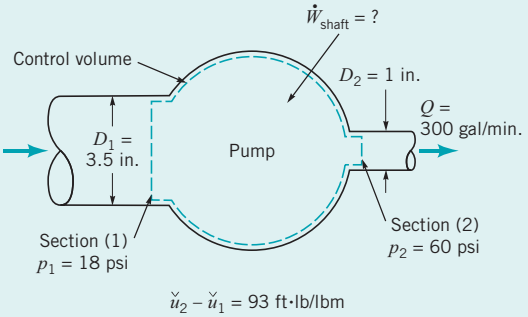



FIGURE E5.20

Substituting the values of Eqs. 2, 3, and 4 and values from the problem statement into Eq. 1 we obtain

$$\dot{W}_{\substack{\text{shaft}\\\text{net in}}} = (41.8 \text{ lbm/s})\left[(93 \text{ ft·lb/lbm}) + \frac{(60 \text{ psi})(144 \text{ in.}^2/\text{ft}^2)}{(1.94 \text{ slugs/ft}^3)(32.174 \text{ lbm/slug})} - \frac{(18 \text{ psi})(144 \text{ in.}^2/\text{ft}^2)}{(1.94 \text{ slugs/ft}^3)(32.174 \text{ lbm/slug})} + \frac{(123 \text{ ft/s})^2 - (10.0 \text{ ft/s})^2}{2[32.174 \text{ (lbm·ft)/(lb·s}^2)]}\right] \times \frac{1}{[550(\text{ft·lb/s})/\text{hp}]} = 32.2 \text{ hp} \qquad \textbf{(Ans)}$$

**COMMENT** Of the total 32.2 hp, internal energy change accounts for 7.09 hp, the pressure rise accounts for 7.37 hp, and the kinetic energy increase accounts for 17.8 hp.

## EXAMPLE 5.21 Energy—Turbine Power per Unit Mass of Flow

**GIVEN** A steam turbine generator unit used to produce electricity is shown in Fig. E5.21*a*. Assume the steam enters a turbine with a velocity of 30 m/s and enthalpy, $\breve{h}_1$, of 3348 kJ/kg (see Fig. E5.21*b*). The steam leaves the turbine as a mixture of vapor and liquid having a velocity of 60 m/s and an enthalpy of 2550 kJ/kg. The flow through the turbine is adiabatic, and changes in elevation are negligible.

**FIND** Determine the work output involved per unit mass of steam through-flow.

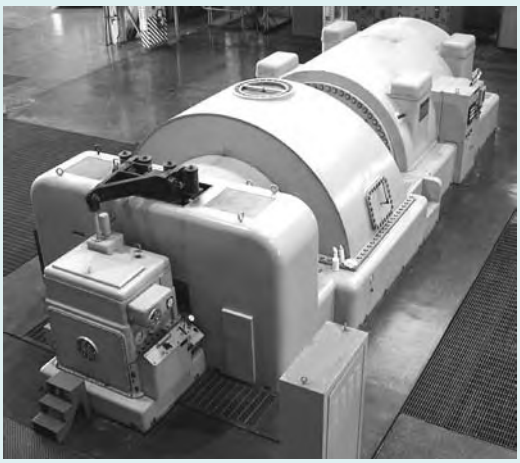


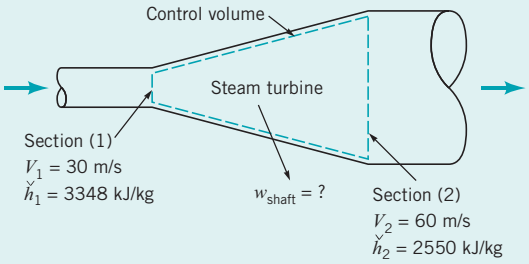
FIGURE E5.21*a*

## SOLUTION

We use a control volume that includes the steam in the turbine from the entrance to the exit as shown in Fig. E5.21*b*. Applying Eq. 5.69 to the steam in this control volume we get

$$\dot{m}\left[\check{h}_2 - \check{h}_1 + \frac{V_2^2 - V_1^2}{2} + g(z_2 - z_1)\right] = \dot{Q}_{\substack{\text{net}\\\text{in}}} + \dot{W}_{\substack{\text{shaft}\\\text{net in}}} \quad \textbf{(1)}$$

(0 (elevation change is negligible); 0 (adiabatic flow))




■ FIGURE E5.21*b*

The work output per unit mass of steam through-flow, $w_{\substack{\text{shaft}\\\text{net in}}}$, can be obtained by dividing Eq. 1 by the mass flow rate, $\dot{m}$, to obtain

$$w_{\substack{\text{shaft}\\\text{net in}}} = \frac{\dot{W}_{\substack{\text{shaft}\\\text{net in}}}}{\dot{m}} = \check{h}_2 - \check{h}_1 + \frac{V_2^2 - V_1^2}{2} \quad \textbf{(2)}$$

Since $w_{\text{shaft net out}} = -w_{\text{shaft net in}}$, we obtain

$$w_{\substack{\text{shaft}\\\text{net out}}} = \check{h}_1 - \check{h}_2 + \frac{V_1^2 - V_2^2}{2}$$

or

$$w_{\substack{\text{shaft}\\\text{net out}}} = 3348 \text{ kJ/kg} - 2550 \text{ kJ/kg} + \frac{[(30 \text{ m/s})^2 - (60 \text{ m/s})^2][1 \text{ J/(N·m)}]}{2[1 \text{ (kg·m)/(N·s}^2)](1000 \text{ J/kJ})}$$

Thus,

$$w_{\substack{\text{shaft}\\\text{net out}}} = 3348 \text{ kJ/kg} - 2550 \text{ kJ/kg} - 1.35 \text{ kJ/kg} = 797 \text{ kJ/kg} \quad \textbf{(Ans)}$$

**COMMENT** Note that in this particular example, the change in kinetic energy is small in comparison to the difference in enthalpy involved. This is often true in applications involving steam turbines. To determine the power output, $\dot{W}_{\text{shaft}}$, we must know the mass flowrate, $\dot{m}$.



*V5.12 Pelton wheel turbine*



If the flow is steady throughout, one-dimensional, and only one fluid stream is involved, then the shaft work is zero and the energy equation is

$$\dot{m}\left[\check{u}_{\text{out}} - \check{u}_{\text{in}} + \left(\frac{p}{\rho}\right)_{\text{out}} - \left(\frac{p}{\rho}\right)_{\text{in}} + \frac{V_{\text{out}}^2 - V_{\text{in}}^2}{2} + g(z_{\text{out}} - z_{\text{in}})\right] = \dot{Q}_{\substack{\text{net}\\\text{in}}} \quad \textbf{(5.70)}$$

We call Eq. 5.70 the *one-dimensional, steady flow energy equation*. This equation is valid for incompressible and compressible flows. For compressible flows, enthalpy is most often used in the one-dimensional, steady flow energy equation and, thus, we have

$$\dot{m}\left[\check{h}_{\text{out}} - \check{h}_{\text{in}} + \frac{V_{\text{out}}^2 - V_{\text{in}}^2}{2} + g(z_{\text{out}} - z_{\text{in}})\right] = \dot{Q}_{\substack{\text{net}\\\text{in}}} \quad \textbf{(5.71)}$$

An example of the application of Eq. 5.70 follows.

## EXAMPLE 5.22 Energy—Temperature Change

**GIVEN** The 420-ft waterfall shown in Fig. E5.22*a* involves steady flow from one large body of water to another.

**FIND** Determine the temperature change associated with this flow.

## SOLUTION

To solve this problem we consider a control volume consisting of a small cross-sectional streamtube from the nearly motionless surface of the upper body of water to the nearly motionless surface of the lower body of water as is sketched in Fig. E5.22*b*. We need to determine $T_2 - T_1$. This temperature change is related to the change of internal energy of the water, $\check{u}_2 - \check{u}_1$, by the relationship

$$T_2 - T_1 = \frac{\check{u}_2 - \check{u}_1}{\check{c}} \quad \textbf{(1)}$$



**FIGURE E5.22a**
**[Photograph of Akaka Falls (Hawaii) courtesy of Scott and Margaret Jones.]**

where $\check{c} = 1$ Btu/(lbm · °R) is the specific heat of water. The application of Eq. 5.70 to the contents of this control volume leads to

$$\dot{m}\left[\check{u}_2 + \check{u}_1 + \left(\frac{p}{\rho}\right)_2 - \left(\frac{p}{\rho}\right)_1 + \frac{V_2^2 - V_1^2}{2} + g(z_2 - z_1)\right] = \dot{Q}_{\substack{\text{net}\\ \text{in}}} \tag{2}$$

We assume that the flow is adiabatic. Thus $\dot{Q}_{\text{net in}} = 0$. Also,

$$\left(\frac{p}{\rho}\right)_1 = \left(\frac{p}{\rho}\right)_2 \tag{3}$$

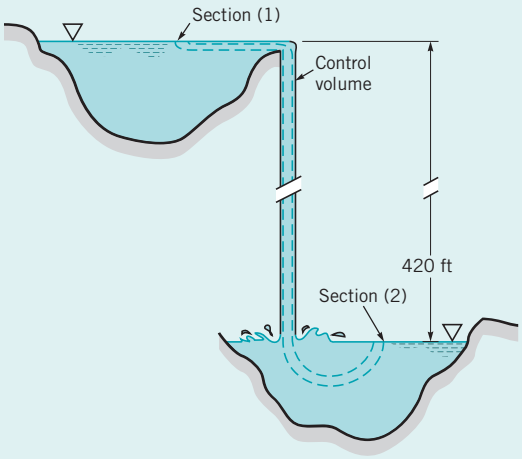



**FIGURE E5.22b**

because the flow is incompressible and atmospheric pressure prevails at sections (1) and (2). Furthermore,

$$V_1 = V_2 = 0 \tag{4}$$

because the surface of each large body of water is considered motionless. Thus, Eqs. 1 through 4 combine to yield

$$T_2 - T_1 = \frac{g(z_1 - z_2)}{\check{c}}$$

so that with

$$\check{c} = [1 \text{ Btu/(lbm} \cdot \text{°R)}]\,(778 \text{ ft} \cdot \text{lb/Btu}) = [778 \text{ ft} \cdot \text{lb/(lbm} \cdot \text{°R)}]$$

$$T_2 - T_1 = \frac{(32.2 \text{ ft/s}^2)(420 \text{ ft})}{[778 \text{ ft} \cdot \text{lb/(lbm} \cdot \text{°R)}][32.2 \text{ (lbm} \cdot \text{ft)/(lb} \cdot \text{s}^2)]} = 0.540 \text{ °R} \quad \textbf{(Ans)}$$

**COMMENT** Note that it takes a considerable change of potential energy to produce even a small increase in temperature.

A form of the energy equation that is most often used to solve incompressible flow problems is developed in the next section.

### 5.3.3 Comparison of the Energy Equation with the Bernoulli Equation

When the one-dimensional energy equation for steady-in-the-mean flow, Eq. 5.67, is applied to a flow that is steady, Eq. 5.67 becomes the one-dimensional, steady-flow energy equation, Eq. 5.70. The only difference between Eq. 5.67 and Eq. 5.70 is that shaft power, $\dot{W}_{\text{shaft net in}}$, is zero if the flow is steady throughout the control volume (fluid machines involve locally unsteady flow). If in addition to being steady, the flow is incompressible, we get from Eq. 5.70

$$\dot{m}\left[\check{u}_{\text{out}} - \check{u}_{\text{in}} + \frac{p_{\text{out}}}{\rho} - \frac{p_{\text{in}}}{\rho} + \frac{V_{\text{out}}^2 - V_{\text{in}}^2}{2} + g(z_{\text{out}} - z_{\text{in}})\right] = \dot{Q}_{\substack{\text{net}\\ \text{in}}} \tag{5.72}$$

Dividing Eq. 5.72 by the mass flowrate, $\dot{m}$, and rearranging terms we obtain

$$\frac{p_{\text{out}}}{\rho} + \frac{V_{\text{out}}^2}{2} + gz_{\text{out}} = \frac{p_{\text{in}}}{\rho} + \frac{V_{\text{in}}^2}{2} + gz_{\text{in}} - (\check{u}_{\text{out}} - \check{u}_{\text{in}} - q_{\substack{\text{net}\\ \text{in}}}) \tag{5.73}$$

where

$$q_{\substack{\text{net}\\\text{in}}} = \frac{\dot{Q}_{\text{net in}}}{\dot{m}}$$

is the heat transfer rate per mass flowrate, or heat transfer per unit mass. Note that Eq. 5.73 involves energy per unit mass and is applicable to one-dimensional flow of a single stream of fluid between two sections or flow along a streamline between two sections.

If the steady, incompressible flow we are considering also involves negligible viscous effects (frictionless flow), then the Bernoulli equation, Eq. 3.7, can be used to describe what happens between two sections in the flow as

$$p_{\text{out}} + \frac{\rho V_{\text{out}}^2}{2} + \gamma z_{\text{out}} = p_{\text{in}} + \frac{\rho V_{\text{in}}^2}{2} + \gamma z_{\text{in}} \tag{5.74}$$

where $\gamma = \rho g$ is the specific weight of the fluid. To get Eq. 5.74 in terms of energy per unit mass, so that it can be compared directly with Eq. 5.73, we divide Eq. 5.74 by density, $\rho$, and obtain

$$\frac{p_{\text{out}}}{\rho} + \frac{V_{\text{out}}^2}{2} + g z_{\text{out}} = \frac{p_{\text{in}}}{\rho} + \frac{V_{\text{in}}^2}{2} + g z_{\text{in}} \tag{5.75}$$

A comparison of Eqs. 5.73 and 5.75 prompts us to conclude that

$$\breve{u}_{\text{out}} - \breve{u}_{\text{in}} - q_{\substack{\text{net}\\\text{in}}} = 0 \tag{5.76}$$

when the steady incompressible flow is frictionless. For steady incompressible flow with friction, we learn from experience (second law of thermodynamics) that

$$\breve{u}_{\text{out}} - \breve{u}_{\text{in}} - q_{\substack{\text{net}\\\text{in}}} > 0 \tag{5.77}$$

In Eqs. 5.73 and 5.75, we can consider the combination of variables

$$\frac{p}{\rho} + \frac{V^2}{2} + gz$$

as equal to *useful* or *available energy*. Thus, from inspection of Eqs. 5.73 and 5.75, we can conclude that $\breve{u}_{\text{out}} - \breve{u}_{\text{in}} - q_{\text{net in}}$ represents the ***loss*** of useful or available energy that occurs in an incompressible fluid flow because of friction. In equation form we have

*Minimizing loss is the central goal of fluid mechanical design.*

$$\breve{u}_{\text{out}} - \breve{u}_{\text{in}} - q_{\substack{\text{net}\\\text{in}}} = \text{loss} \tag{5.78}$$

For a frictionless flow, Eqs. 5.73 and 5.75 tell us that loss equals zero.

It is often convenient to express Eq. 5.73 in terms of loss as

$$\frac{p_{\text{out}}}{\rho} + \frac{V_{\text{out}}^2}{2} + g z_{\text{out}} = \frac{p_{\text{in}}}{\rho} + \frac{V_{\text{in}}^2}{2} + g z_{\text{in}} - \text{loss} \tag{5.79}$$

An example of the application of Eq. 5.79 follows.

## EXAMPLE 5.23 Energy—Effect of Loss of Available Energy

**GIVEN** As shown in Fig. E5.23*a*, air flows from a room through two different vent configurations: a cylindrical hole in the wall having a diameter of 120 mm and the same diameter cylindrical hole in the wall but with a well-rounded entrance. The room pressure is held constant at 1.0 kPa above atmospheric pressure. Both vents exhaust into the atmosphere. As discussed in Section 8.4.2, the loss in available energy associated with flow through the cylindrical vent from the room to the vent exit is $0.5V_2^2/2$ where $V_2$ is the uniformly distributed exit velocity of air. The loss in available energy associated with flow through the rounded entrance vent from the room to the vent exit is $0.05V_2^2/2$, where $V_2$ is the uniformly distributed exit velocity of air.

**FIND** Compare the volume flowrates associated with the two different vent configurations.

## SOLUTION

We use the control volume for each vent sketched in Fig. E5.23*a*. What is sought is the flowrate, $Q = A_2V_2$, where $A_2$ is the vent exit cross-sectional area, and $V_2$ is the uniformly distributed exit velocity. For both vents, application of Eq. 5.79 leads to

$$\underbrace{\frac{p_2}{\rho} + \frac{V_2^2}{2} + g\cancel{z_2}}_{0\ (\text{no elevation change})} = \underbrace{\frac{p_1}{\rho} + \cancel{\frac{V_1^2}{2}} + g\cancel{z_1}}_{0\ (V_1 \approx 0)} - {}_1\text{loss}_2 \tag{1}$$

where ${}_1\text{loss}_2$ is the loss between sections (1) and (2). Solving Eq. 1 for $V_2$ we get

$$V_2 = \sqrt{2\left[\left(\frac{p_1 - p_2}{\rho}\right) - {}_1\text{loss}_2\right]} \tag{2}$$

Since

$${}_1\text{loss}_2 = K_L\frac{V_2^2}{2} \tag{3}$$

where $K_L$ is the loss coefficient ($K_L = 0.5$ and 0.05 for the two vent configurations involved), we can combine Eqs. 2 and 3 to get

$$V_2 = \sqrt{2\left[\left(\frac{p_1 - p_2}{\rho}\right) - K_L\frac{V_2^2}{2}\right]} \tag{4}$$

Solving Eq. 4 for $V_2$ we obtain

$$V_2 = \sqrt{\frac{p_1 - p_2}{\rho[(1 + K_L)/2]}} \tag{5}$$

Therefore, for flowrate, $Q$, we obtain

$$Q = A_2V_2 = \frac{\pi D_2^2}{4}\sqrt{\frac{p_1 - p_2}{\rho[(1 + K_L)/2]}} \tag{6}$$

For the rounded entrance cylindrical vent, Eq. 6 gives

$$Q = \frac{\pi(120\text{ mm})^2}{4(1000\text{ mm/m})^2} \times \sqrt{\frac{(1.0\text{ kPa})(1000\text{ Pa/kPa})[1(\text{N/m}^2)/(\text{Pa})]}{(1.23\text{ kg/m}^3)[(1 + 0.05)/2][1(\text{N}\cdot\text{s}^2)/(\text{kg}\cdot\text{m})]}}$$

or

$$Q = 0.445\text{ m}^3/\text{s} \quad \textbf{(Ans)}$$

For the cylindrical vent, Eq. 6 gives us

$$Q = \frac{\pi(120\text{ mm})^2}{4(1000\text{ mm/m})^2} \times \sqrt{\frac{(1.0\text{ kPa})(1000\text{ Pa/kPa})[1(\text{N/m}^2)/(\text{Pa})]}{(1.23\text{ kg/m}^3)[(1 + 0.5)/2][1(\text{N}\cdot\text{s}^2)/(\text{kg}\cdot\text{m})]}}$$

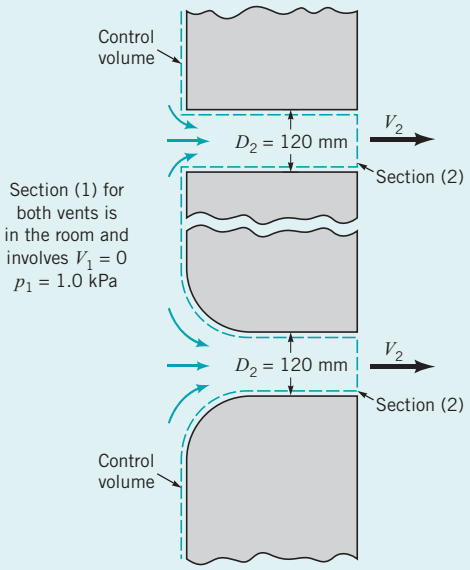



**FIGURE E5.23*a***

or

$$Q = 0.372\text{ m}^3/\text{s} \quad \textbf{(Ans)}$$

**COMMENT** By repeating the calculations for various values of the loss coefficient, $K_L$, the results shown in Fig. E5.23*b* are obtained. Note that the rounded entrance vent allows the passage of more air than does the cylindrical vent because the loss associated with the rounded entrance vent is less than that for the cylindrical one. For this flow the pressure drop, $p_1 - p_2$, has two purposes: (1) overcome the loss associated with the flow, and (2) produce the kinetic energy at the exit. Even if there were no loss (i.e., $K_L = 0$), a pressure drop would be needed to accelerate the fluid through the vent.

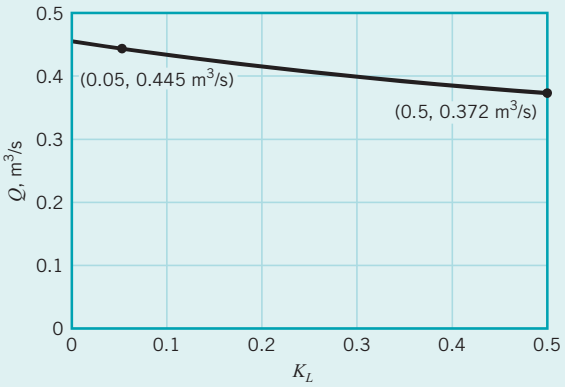



**FIGURE E5.23*b***

An important group of fluid mechanics problems involves one-dimensional, incompressible, steady-in-the-mean flow with friction and shaft work. Included in this category are constant density flows through pumps, blowers, fans, and turbines. For this kind of flow, Eq. 5.67 becomes

$$\dot{m}\left[\check{u}_{\text{out}} - \check{u}_{\text{in}} + \frac{p_{\text{out}}}{\rho} - \frac{p_{\text{in}}}{\rho} + \frac{V_{\text{out}}^2 - V_{\text{in}}^2}{2} + g(z_{\text{out}} - z_{\text{in}})\right] = \dot{Q}_{\substack{\text{net}\\\text{in}}} + \dot{W}_{\substack{\text{shaft}\\\text{net in}}} \tag{5.80}$$

CMS Cepcor EX 1046 Page 141
CMS Cepcor v. Sandvik IPR2024-01380

Dividing Eq. 5.80 by mass flowrate and using the work per unit mass, $w_{\substack{\text{shaft}\\\text{net in}}} = \dot{W}_{\substack{\text{shaft}\\\text{net in}}}/\dot{m}$, we obtain

$$\frac{p_{out}}{\rho} + \frac{V_{out}^2}{2} + gz_{out} = \frac{p_{in}}{\rho} + \frac{V_{in}^2}{2} + gz_{in} + w_{\substack{\text{shaft}\\\text{net in}}} - (\breve{u}_{out} - \breve{u}_{in} - q_{\substack{\text{net}\\\text{in}}}) \tag{5.81}$$

*The mechanical energy equation can be written in terms of energy per unit mass.*

If the flow is steady throughout, Eq. 5.81 becomes identical to Eq. 5.73, and the previous observation that $\breve{u}_{out} - \breve{u}_{in} - q_{net\ in}$ equals the loss of available energy is valid. Thus, we conclude that Eq. 5.81 can be expressed as

$$\frac{p_{out}}{\rho} + \frac{V_{out}^2}{2} + gz_{out} = \frac{p_{in}}{\rho} + \frac{V_{in}^2}{2} + gz_{in} + w_{\substack{\text{shaft}\\\text{net in}}} - \text{loss} \tag{5.82}$$



*V5.13 Energy transfer*



This is a form of the energy equation for steady-in-the-mean flow that is often used for incompressible flow problems. It is sometimes called the *mechanical energy* equation or the *extended Bernoulli* equation. Note that Eq. 5.82 involves energy per unit mass ($ft \cdot lb/slug = ft^2/s^2$ or $N \cdot m = m^2/s^2$).

According to Eq. 5.82, when the shaft work is into the control volume, as for example with a pump, a larger amount of loss will result in more shaft work being required for the same rise in available energy. Similarly, when the shaft work is out of the control volume (for example, a turbine), a larger loss will result in less shaft work out for the same drop in available energy. Designers spend a great deal of effort on minimizing losses in fluid flow components. The following examples demonstrate why losses should be kept as small as possible in fluid systems.

## EXAMPLE 5.24 Energy—Fan Work and Efficiency

**GIVEN** An axial-flow ventilating fan driven by a motor that delivers 0.4 kW of power to the fan blades produces a 0.6-m-diameter axial stream of air having a speed of 12 m/s. The flow upstream of the fan involves negligible speed.

**FIND** Determine how much of the work to the air actually produces useful effects, that is, fluid motion and a rise in available energy. Estimate the fluid mechanical efficiency of this fan.

### SOLUTION

We select a fixed and nondeforming control volume as is illustrated in Fig. E5.24. The application of Eq. 5.82 to the contents of this control volume leads to

$$w_{\substack{\text{shaft}\\\text{net in}}} - \text{loss} = \left(\frac{p_2}{\rho} + \frac{V_2^2}{2} + gz_2\right) - \left(\frac{p_1}{\rho} + \frac{V_1^2}{2} + gz_1\right) \tag{1}$$

(atmospheric pressures cancel); ($V_1 \approx 0$); (no elevation change)

where $w_{\substack{\text{shaft}\\\text{net in}}} - \text{loss}$ is the amount of work added to the air that produces a useful effect. Equation 1 leads to

$$w_{\substack{\text{shaft}\\\text{net in}}} - \text{loss} = \frac{V_2^2}{2} = \frac{(12\ m/s)^2}{2[1(kg\cdot m)/(N\cdot s^2)]}$$

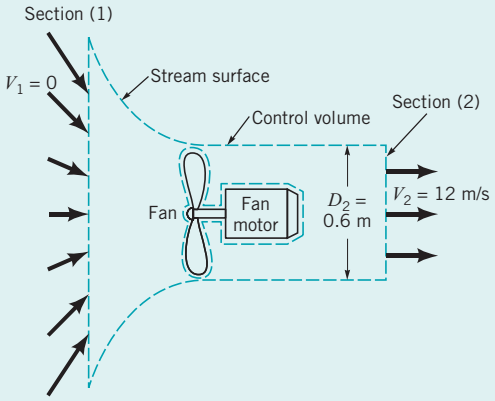
$$= 72.0\ N\cdot m/kg \tag{2}$$ **(Ans)**

A reasonable estimate of *efficiency*, $\eta$, would be the ratio of amount of work that produces a useful effect, Eq. 2, to the amount of work delivered to the fan blades. That is

$$\eta = \frac{w_{\substack{\text{shaft}\\\text{net in}}} - \text{loss}}{w_{\substack{\text{shaft}\\\text{net in}}}} \tag{3}$$

To calculate the efficiency, we need a value of $w_{\substack{\text{shaft}\\\text{net in}}}$, which is related to the power delivered to the blades, $\dot{W}_{\substack{\text{shaft}\\\text{net in}}}$. We note that

$$w_{\substack{\text{shaft}\\\text{net in}}} = \frac{\dot{W}_{\substack{\text{shaft}\\\text{net in}}}}{\dot{m}} \tag{4}$$




■ FIGURE E5.24

where the mass flowrate, $\dot{m}$, is (from Eq. 5.6)

$$\dot{m} = \rho A V = \rho \frac{\pi D_2^2}{4} V_2 \qquad (5)$$

For fluid density, $\rho$, we use 1.23 kg/m$^3$ (standard air) and, thus, from Eqs. 4 and 5 we obtain

$$w_{\substack{\text{shaft}\\\text{net in}}} = \frac{\dot{W}_{\substack{\text{shaft}\\\text{net in}}}}{(\rho\pi D_2^2/4)V_2} = \frac{(0.4\text{ kW})[1000\text{ (Nm)/(skW)}]}{(1.23\text{ kg/m}^3)[(\pi)(0.6\text{ m})^2/4](12\text{ m/s})}$$

or

$$w_{\substack{\text{shaft}\\\text{net in}}} = 95.8\text{ N·m/kg} \qquad (6)$$

From Eqs. 2, 3, and 6 we obtain

$$\eta = \frac{72.0\text{ N·m/kg}}{95.8\text{ N·m/kg}} = 0.752 \qquad \textbf{(Ans)}$$

**COMMENT** Note that only 75% of the power that was delivered to the air resulted in useful effects, and, thus, 25% of the shaft power is lost to air friction.

## Fluids in the News

**Curtain of air** An air curtain is produced by blowing air through a long rectangular nozzle to produce a high-velocity sheet of air, or a "curtain of air." This air curtain is typically directed over a doorway or opening as a replacement for a conventional door. The air curtain can be used for such things as keeping warm air from infiltrating dedicated cold spaces, preventing dust and other contaminates from entering a clean environment, and even just keeping insects out of the workplace, still allowing people to enter or exit. A disadvantage over conventional doors is the added *power requirements* to operate the air curtain, although the advantages can outweigh the disadvantage for various industrial applications. New applications for current air curtain designs continue to be developed. For example, the use of air curtains as a means of road tunnel fire security is currently being investigated. In such an application, the air curtain would act to isolate a portion of the tunnel where fire has broken out and not allow smoke and fumes to infiltrate the entire tunnel system. (See Problem 5.123.)

If Eq. 5.82, which involves energy per unit mass, is multiplied by fluid density, $\rho$, we obtain

$$p_{\text{out}} + \frac{\rho V_{\text{out}}^2}{2} + \gamma z_{\text{out}} = p_{\text{in}} + \frac{\rho V_{\text{in}}^2}{2} + \gamma z_{\text{in}} + \rho w_{\substack{\text{shaft}\\\text{net in}}} - \rho(\text{loss}) \qquad \textbf{(5.83)}$$



*V5.14 Water plant aerator*



where $\gamma = \rho g$ is the specific weight of the fluid. Equation 5.83 involves *energy per unit volume* and the units involved are identical with those used for pressure (ft · lb/ft$^3$ = lb/ft$^2$ or N · m/m$^3$ = N/m$^2$).

If Eq. 5.82 is divided by the acceleration of gravity, $g$, we get

$$\frac{p_{\text{out}}}{\gamma} + \frac{V_{\text{out}}^2}{2g} + z_{\text{out}} = \frac{p_{\text{in}}}{\gamma} + \frac{V_{\text{in}}^2}{2g} + z_{\text{in}} + h_s - h_L \qquad \textbf{(5.84)}$$

where

$$h_s = w_{\text{shaft net in}}/g = \frac{\dot{W}_{\substack{\text{shaft}\\\text{net in}}}}{\dot{m}g} = \frac{\dot{W}_{\substack{\text{shaft}\\\text{net in}}}}{\gamma Q} \qquad \textbf{(5.85)}$$

*The energy equation written in terms of energy per unit weight involves heads.*

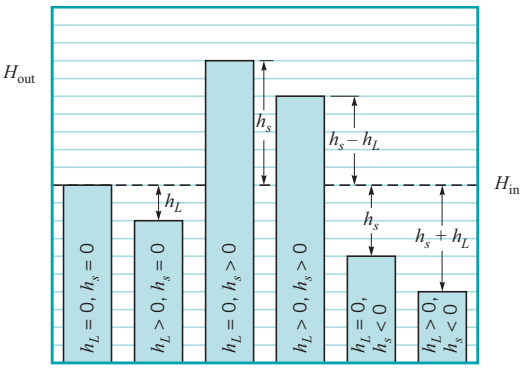
is the ***shaft work head*** and $h_L$ = loss/$g$ is the ***head loss***. Equation 5.84 involves *energy per unit weight* (ft · lb/lb = ft or N · m/N = m). In Section 3.7, we introduced the notion of "head," which is energy per unit weight. Units of length (for example, ft, m) are used to quantify the amount of head involved. If a turbine is in the control volume, $h_s$ is negative because it is associated with shaft work out of the control volume. For a pump in the control volume, $h_s$ is positive because it is associated with shaft work into the control volume.

We can define a total head, $H$, as follows

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

Then Eq. 5.84 can be expressed as

$$H_{\text{out}} = H_{\text{in}} + h_s - h_L$$




■ **FIGURE 5.8** **Total-head change in fluid flows.**

Some important possible values of $H_{out}$ in comparison to $H_{in}$ are shown in Fig. 5.8. Note that $h_L$ (head loss) always reduces the value of $H_{out}$, except in the ideal case when it is zero. Note also that $h_L$ lessens the effect of shaft work that can be extracted from a fluid. When $h_L = 0$ (ideal condition) the shaft work head, $h_s$, and the change in total head are the same. This head change is sometimes called "ideal head change." The corresponding ideal shaft work head is the minimum required to achieve a desired effect. For work out, it is the maximum possible. Designers usually strive to minimize loss. In Chapter 12 we learn of one instance when minimum loss is sacrificed for survivability of fish coursing through a turbine rotor.

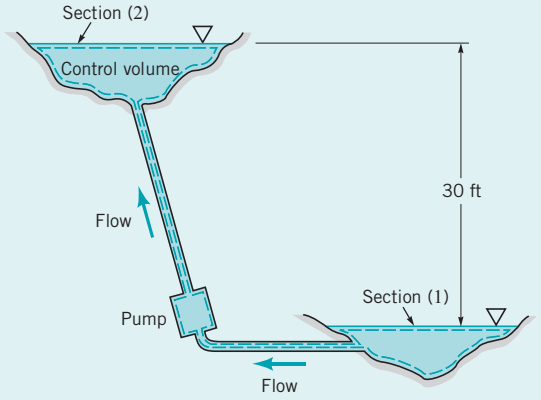
## EXAMPLE 5.25 Energy—Head Loss and Power Loss

**GIVEN** The pump shown in Fig. E5.25*a* adds 10 horsepower to the water as it pumps water from the lower lake to the upper lake. The elevation difference between the lake surfaces is 30 ft and the head loss is 15 ft.

**FIND** Determine

**(a)** the flowrate and

**(b)** the power loss associated with this flow.




■ **FIGURE E5.25*a***

### SOLUTION

**(a)** The energy equation (Eq. 5.84) for this flow is

$$\frac{p_2}{\gamma} + \frac{V_2^2}{2g} + z_2 = \frac{p_1}{\gamma} + \frac{V_1^2}{2g} + z_1 + h_s - h_L \qquad (1)$$

where points 2 and 1 (corresponding to "out" and "in" in Eq. 5.84) are located on the lake surfaces. Thus, $p_2 = p_1 = 0$ and $V_2 = V_1 = 0$ so that Eq. 1 becomes

$$h_s = h_L + z_2 - z_1 \qquad (2)$$

where $z_2 = 30$ ft, $z_1 = 0$, and $h_L = 15$ ft. The pump head is obtained from Eq. 5.85 as

$$h_s = \dot{W}_{\text{shaft net in}}/\gamma Q$$
$$= (10 \text{ hp})(550 \text{ ft·lb/s/hp})/(62.4 \text{ lb/ft}^3)\, Q$$
$$= 88.1/Q$$

where $h_s$ is in ft when $Q$ is in ft$^3$/s.

Hence, from Eq. 2,

$$88.1/Q = 15 \text{ ft} + 30 \text{ ft}$$

or

$$Q = 1.96 \text{ ft}^3/\text{s} \qquad \textbf{(Ans)}$$

**COMMENT** Note that in this example the purpose of the pump is to lift the water (a 30-ft head) and overcome the head loss (a 15-ft head); it does not, overall, alter the water's pressure or velocity.

**(b)** The power lost due to friction can be obtained from Eq. 5.85 as

$$\dot{W}_{\text{loss}} = \gamma Q h_L = (62.4 \text{ lb/ft}^3)(1.96 \text{ ft}^3\text{/s})(15 \text{ ft})$$
$$= 1830 \text{ ft·lb/s } (1 \text{ hp}/550 \text{ ft·lb/s})$$
$$= 3.33 \text{ hp} \qquad \textbf{(Ans)}$$

**COMMENTS** The remaining 10 hp − 3.33 hp = 6.67 hp that the pump adds to the water is used to lift the water from the lower to the upper lake. This energy is not "lost," but it is stored as potential energy.

By repeating the calculations for various head losses, $h_L$, the results shown in Fig. E5.25*b* are obtained. Note that as the head loss increases, the flowrate decreases because an increasing portion of the 10 hp supplied by the pump is lost and, therefore, not available to lift the fluid to the higher elevation.

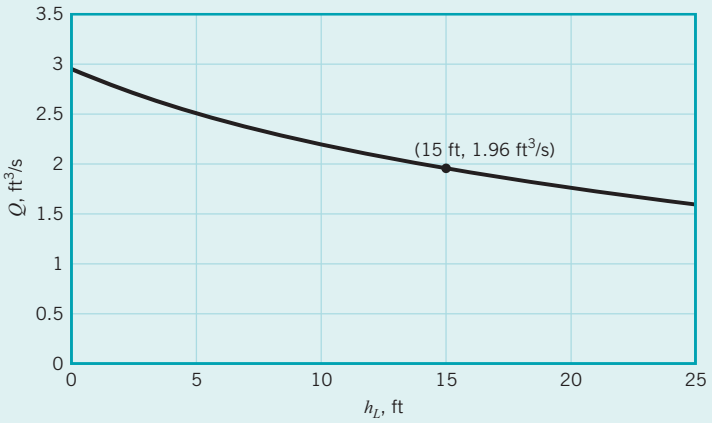



**FIGURE E5.25*b***

A comparison of the energy equation and the Bernoulli equation has led to the concept of loss of available energy in incompressible fluid flows with friction. In Chapter 8, we discuss in detail some methods for estimating loss in incompressible flows with friction. In Section 5.4 and Chapter 11, we demonstrate that loss of available energy is also an important factor to consider in compressible flows with friction.

## Fluids in the News

**Smart shocks** Vehicle shock absorbers are dampers used to provide a smooth, controllable ride. When going over a bump, the relative motion between the tires and the vehicle body displaces a piston in the shock and forces a viscous fluid through a small orifice or channel. The viscosity of the fluid produces a *head loss* that dissipates energy to dampen the vertical motion. Current shocks use a fluid with fixed viscosity. However, recent technology has been developed that uses a synthetic oil with millions of tiny iron balls suspended in it. These tiny balls react to a magnetic field generated by an electric coil on the shock piston in a manner that changes the fluid viscosity, going anywhere from essentially no damping to a solid almost instantly. A computer adjusts the current to the coil to select the proper viscosity for the given conditions (i.e., wheel speed, vehicle speed, steering-wheel angle, lateral acceleration, brake application, and temperature). The goal of these adjustments is an optimally tuned shock that keeps the vehicle on a smooth, even keel while maximizing the contact of the tires with the pavement for any road conditions. (See Problem 5.107.)

### 5.3.4 Application of the Energy Equation to Nonuniform Flows

The forms of the energy equation discussed in Sections 5.3.2 and 5.3.3 are applicable to one-dimensional flows, flows that are approximated with uniform velocity distributions where fluid crosses the control surface.

CMS Cepcor EX 1046 Page 145
CMS Cepcor v. Sandvik IPR2024-01380

If the velocity profile at any section where flow crosses the control surface is not uniform, inspection of the energy equation for a control volume, Eq. 5.64, suggests that the integral

$$\int_{cs} \frac{V^2}{2} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$$

*The kinetic energy coefficient is used to account for nonuniform flows.*

will require special attention. The other terms of Eq. 5.64 can be accounted for as already discussed in Sections 5.3.2 and 5.3.3.

For one stream of fluid entering and leaving the control volume, we can define the relationship

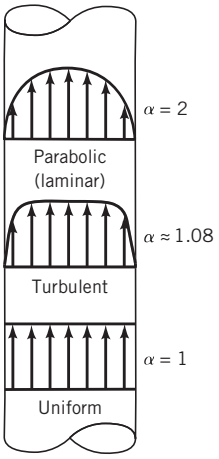
$$\int_{cs} \frac{V^2}{2} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA = \dot{m}\left(\frac{\alpha_{out}\overline{V}_{out}^2}{2} - \frac{\alpha_{in}\overline{V}_{in}^2}{2}\right)$$

where $\alpha$ is the ***kinetic energy coefficient*** and $\overline{V}$ is the average velocity defined earlier in Eq. 5.7. From the above we can conclude that

$$\frac{\dot{m}\alpha\overline{V}^2}{2} = \int_A \frac{V^2}{2} \rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA$$

for flow through surface area $A$ of the control surface. Thus,

$$\alpha = \frac{\displaystyle\int_A (V^2/2)\rho \mathbf{V} \cdot \hat{\mathbf{n}}\, dA}{\dot{m}\overline{V}^2/2} \tag{5.86}$$




It can be shown that for any velocity profile, $\alpha \geq 1$, with $\alpha = 1$ only for uniform flow. Some typical velocity profile examples for flow in a conventional pipe are shown in the sketch in the margin. Therefore, for nonuniform velocity profiles, the energy equation on an energy per unit mass basis for the incompressible flow of one stream of fluid through a control volume that is steady in the mean is

$$\frac{p_{out}}{\rho} + \frac{\alpha_{out}\overline{V}_{out}^2}{2} + gz_{out} = \frac{p_{in}}{\rho} + \frac{\alpha_{in}\overline{V}_{in}^2}{2} + gz_{in} + w_{\substack{shaft\\net\ in}} - \text{loss} \tag{5.87}$$

On an energy per unit volume basis we have

$$p_{out} + \frac{\rho\alpha_{out}\overline{V}_{out}^2}{2} + \gamma z_{out} = p_{in} + \frac{\rho\alpha_{in}\overline{V}_{in}^2}{2} + \gamma z_{in} + \rho w_{\substack{shaft\\net\ in}} - \rho(\text{loss}) \tag{5.88}$$

and on an energy per unit weight or head basis we have

$$\frac{p_{out}}{\gamma} + \frac{\alpha_{out}\overline{V}_{out}^2}{2g} + z_{out} = \frac{p_{in}}{\gamma} + \frac{\alpha_{in}\overline{V}_{in}^2}{2g} + z_{in} + \frac{w_{\substack{shaft\\net\ in}}}{g} - h_L \tag{5.89}$$

The following examples illustrate the use of the kinetic energy coefficient.

## EXAMPLE 5.26 Energy—Effect of Nonuniform Velocity Profile

**GIVEN** The small fan shown in Fig. E5.26 moves air at a mass flowrate of 0.1 kg/min. Upstream of the fan, the pipe diameter is 60 mm, the flow is laminar, the velocity distribution is parabolic, and the kinetic energy coefficient, $\alpha_1$, is equal to 2.0. Downstream of the fan, the pipe diameter is 30 mm, the flow is turbulent, the velocity profile is quite uniform, and the kinetic energy coefficient, $\alpha_2$, is equal to 1.08. The rise in static pressure across the fan is 0.1 kPa and the fan motor draws 0.14 W.

**FIND** Compare the value of loss calculated: **(a)** assuming uniform velocity distributions, **(b)** considering actual velocity distributions.

## SOLUTION

Application of Eq. 5.87 to the contents of the control volume shown in Fig. E5.26 leads to

$$\frac{p_2}{\rho} + \frac{\alpha_2 \overline{V}_2^2}{2} + \cancel{gz_2} = \frac{p_1}{\rho} + \frac{\alpha_1 \overline{V}_1^2}{2} + \cancel{gz_1} - \text{loss} + w_{\substack{\text{shaft}\\ \text{net in}}} \qquad (1)$$

(0 (change in $gz$ is negligible))

or solving Eq. 1 for loss we get

$$\text{loss} = w_{\substack{\text{shaft}\\ \text{net in}}} - \left(\frac{p_2 - p_1}{\rho}\right) + \frac{\alpha_1 \overline{V}_1^2}{2} - \frac{\alpha_2 \overline{V}_2^2}{2} \qquad (2)$$

To proceed further, we need values of $w_{\text{shaft net in}}$, $\overline{V}_1$, and $\overline{V}_2$. These quantities can be obtained as follows. For shaft work

$$w_{\substack{\text{shaft}\\ \text{net in}}} = \frac{\text{power to fan motor}}{\dot{m}}$$

or

$$w_{\substack{\text{shaft}\\ \text{net in}}} = \frac{(0.14 \text{ W})[(1 \text{ N} \cdot \text{m/s})/\text{W}]}{0.1 \text{ kg/min}} (60 \text{ s/min}) = 84.0 \text{ N} \cdot \text{m/kg} \qquad (3)$$

For the average velocity at section (1), $\overline{V}_1$, from Eq. 5.11 we obtain

$$\overline{V}_1 = \frac{\dot{m}}{\rho A_1} = \frac{\dot{m}}{\rho(\pi D_1^2/4)} = \frac{(0.1 \text{ kg/min})\,(1 \text{ min}/60 \text{ s})\,(1000 \text{ mm/m})^2}{(1.23 \text{ kg/m}^3)[\pi(60 \text{ mm})^2/4]} = 0.479 \text{ m/s} \qquad (4)$$

For the average velocity at section (2), $\overline{V}_2$,

$$\overline{V}_2 = \frac{(0.1 \text{ kg/min})\,(1 \text{ min}/60 \text{ s})\,(1000 \text{ mm/m})^2}{(1.23 \text{ kg/m}^3)[\pi(30 \text{ mm})^2/4]} = 1.92 \text{ m/s} \qquad (5)$$

**(a)** For the assumed uniform velocity profiles ($\alpha_1 = \alpha_2 = 1.0$), Eq. 2 yields

$$\text{loss} = w_{\substack{\text{shaft}\\ \text{net in}}} - \left(\frac{p_2 - p_1}{\rho}\right) + \frac{\overline{V}_1^2}{2} - \frac{\overline{V}_2^2}{2} \qquad (6)$$

Using Eqs. 3, 4, and 5 and the pressure rise given in the problem statement, Eq. 6 gives

$$\text{loss} = 84.0 \frac{\text{N} \cdot \text{m}}{\text{kg}} - \frac{(0.1 \text{ kPa})(1000 \text{ Pa/kPa})(1 \text{ N/m}^2/\text{Pa})}{1.23 \text{ kg/m}^3} + \frac{(0.479 \text{ m/s})^2}{2[1 \text{ (kg} \cdot \text{m)}/(\text{N} \cdot \text{s}^2)]} - \frac{(1.92 \text{ m/s})^2}{2[1 \text{ (kg} \cdot \text{m)}/(\text{N} \cdot \text{s}^2)]}$$

or

$$\text{loss} = 84.0 \text{ N} \cdot \text{m/kg} - 81.3 \text{ N} \cdot \text{m/kg} + 0.115 \text{ N} \cdot \text{m/kg} - 1.84 \text{ N} \cdot \text{m/kg} = 0.975 \text{ N} \cdot \text{m/kg} \quad \textbf{(Ans)}$$

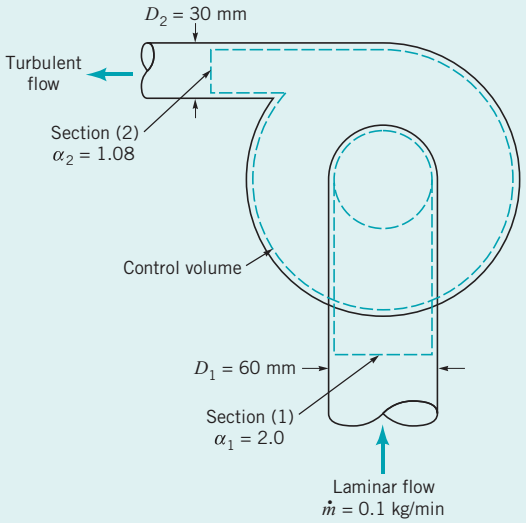



**FIGURE E5.26**

**(b)** For the actual velocity profiles ($\alpha_1 = 2$, $\alpha_2 = 1.08$), Eq. 1 gives

$$\text{loss} = w_{\substack{\text{shaft}\\ \text{net in}}} - \left(\frac{p_2 - p_1}{\rho}\right) + \alpha_1 \frac{\overline{V}_1^2}{2} - \alpha_2 \frac{\overline{V}_2^2}{2} \qquad (7)$$

If we use Eqs. 3, 4, and 5 and the given pressure rise, Eq. 7 yields

$$\text{loss} = 84 \text{ N} \cdot \text{m/kg} - \frac{(0.1 \text{ kPa})(1000 \text{ Pa/kPa})(1 \text{ N/m}^2/\text{Pa})}{1.23 \text{ kg/m}^3} + \frac{2(0.479 \text{ m/s})^2}{2[1 \text{ (kg} \cdot \text{m)}/(\text{N} \cdot \text{s}^2)]} - \frac{1.08(1.92 \text{ m/s})^2}{2[1 \text{ (kg} \cdot \text{m)}/(\text{N} \cdot \text{s}^2)]}$$

or

$$\text{loss} = 84.0 \text{ N} \cdot \text{m/kg} - 81.3 \text{ N} \cdot \text{m/kg} + 0.230 \text{ N} \cdot \text{m/kg} - 1.99 \text{ N} \cdot \text{m/kg} = 0.940 \text{ N} \cdot \text{m/kg} \quad \textbf{(Ans)}$$

**COMMENT** The difference in loss calculated assuming uniform velocity profiles and actual velocity profiles is not large compared to $w_{\text{shaft net in}}$ for this fluid flow situation.

## EXAMPLE 5.27 Energy—Effect of Nonuniform Velocity Profile

**GIVEN** Consider the flow situation of Example 5.14.

**FIND** Apply Eq. 5.87 to develop an expression for the fluid pressure drop that occurs between sections (1) and (2). By comparing the equation for pressure drop obtained presently with the result of Example 5.14, obtain an expression for loss between sections (1) and (2).

### SOLUTION

Application of Eq. 5.87 to the flow of Example 5.14 (see Fig. E5.14) leads to

$$\frac{p_2}{\rho}+\frac{\alpha_2\overline{w}_2^2}{2}+gz_2=\frac{p_1}{\rho}+\frac{\alpha_1\overline{w}_1^2}{2}+gz_1-\text{loss}+\underbrace{w_{\substack{\text{shaft}\\ \text{net in}}}}_{0\ (\text{no shaft work})} \tag{1}$$

Solving Eq. 1 for the pressure drop, $p_1-p_2$, we obtain

$$p_1-p_2=\rho\left[\frac{\alpha_2\overline{w}_2^2}{2}-\frac{\alpha_1\overline{w}_1^2}{2}+g(z_2-z_1)+\text{loss}\right] \tag{2}$$

Since the fluid velocity at section (1), $w_1$, is uniformly distributed over cross-sectional area $A_1$, the corresponding kinetic energy coefficient, $\alpha_1$, is equal to 1.0. The kinetic energy coefficient at section (2), $\alpha_2$, needs to be determined from the velocity profile distribution given in Example 5.14. Using Eq. 5.86 we get

$$\alpha_2=\frac{\displaystyle\int_{A_2}\rho w_2^3\,dA_2}{\dot{m}\overline{w}_2^2} \tag{3}$$

Substituting the parabolic velocity profile equation into Eq. 3 we obtain

$$\alpha_2=\frac{\rho\displaystyle\int_0^R(2w_1)^3[1-(r/R)^2]^3\,2\pi r\,dr}{(\rho A_2\overline{w}_2)\overline{w}_2^2}$$

From conservation of mass, since $A_1=A_2$

$$w_1=\overline{w}_2 \tag{4}$$

Then, substituting Eq. 4 into Eq. 3, we obtain

$$\alpha_2=\frac{\rho 8\overline{w}_2^3 2\pi\displaystyle\int_0^R[1-(r/R)^2]^3 r\,dr}{\rho\pi R^2\overline{w}_2^3}$$

or

$$\alpha_2=\frac{16}{R^2}\int_0^R[1-3(r/R)^2+3(r/R)^4-(r/R)^6]r\,dr=2 \tag{5}$$

Now we combine Eqs. 2 and 5 to get

$$p_1-p_2=\rho\left[\frac{2.0\overline{w}_2^2}{2}-\frac{1.0\overline{w}_1^2}{2}+g(z_2-z_1)+\text{loss}\right] \tag{6}$$

However, from conservation of mass $\overline{w}_2=\overline{w}_1=\overline{w}$ so that Eq. 6 becomes

$$p_1-p_2=\frac{\rho\overline{w}^2}{2}+\rho g(z_2-z_1)+\rho(\text{loss}) \tag{7}$$

The term associated with change in elevation, $\rho g(z_2-z_1)$, is equal to the weight per unit cross-sectional area, $\mathcal{W}/A$, of the water contained between sections (1) and (2) at any instant,

$$\rho g(z_2-z_1)=\frac{\mathcal{W}}{A} \tag{8}$$

Thus, combining Eqs. 7 and 8 we get

$$p_1-p_2=\frac{\rho\overline{w}^2}{2}+\frac{\mathcal{W}}{A}+\rho(\text{loss}) \tag{9}$$

The pressure drop between sections (1) and (2) is due to:

1. The change in kinetic energy between sections (1) and (2) associated with going from a uniform velocity profile to a parabolic velocity profile.
2. The weight of the water column, that is, hydrostatic pressure effect.
3. Viscous loss.

Comparing Eq. 9 for pressure drop with the one obtained in Example 5.14 (i.e., the answer of Example 5.14) we obtain

$$\frac{\rho\overline{w}^2}{2}+\frac{\mathcal{W}}{A}+\rho(\text{loss})=\frac{\rho\overline{w}^2}{3}+\frac{R_z}{A}+\frac{\mathcal{W}}{A} \tag{10}$$

or

$$\text{loss}=\frac{R_z}{\rho A}-\frac{\overline{w}^2}{6} \qquad \textbf{(Ans)}$$

**COMMENT** We conclude that while some of the pipe wall friction force, $R_z$, resulted in loss of available energy, a portion of this friction, $\rho A\overline{w}^2/6$, led to the velocity profile change.

### 5.3.5 Combination of the Energy Equation and the Moment-of-Momentum Equation[4]

If Eq. 5.82 is used for one-dimensional incompressible flow through a turbomachine, we can use Eq. 5.54, developed in Section 5.2.4 from the moment-of-momentum equation (Eq. 5.42), to evaluate

[4]This section may be omitted without loss of continuity in the text material. This section should not be considered without prior study of Sections 5.2.3 and 5.2.4. All of these sections are recommended for those interested in Chapter 12.

CMS Cepcor EX 1046 Page 148
CMS Cepcor v. Sandvik IPR2024-01380

shaft work. This application of both Eqs. 5.54 and 5.82 allows us to ascertain the amount of loss that occurs in incompressible turbomachine flows as is demonstrated in Example 5.28.

## EXAMPLE 5.28 Energy—Fan Performance

**GIVEN** Consider the fan of Example 5.19.

**FIND** Show that only some of the shaft power into the air is converted into useful effects. Develop a meaningful efficiency equation and a practical means for estimating lost shaft energy.

### SOLUTION

We use the same control volume used in Example 5.19. Application of Eq. 5.82 to the contents of this control volume yields

$$\frac{p_2}{\rho}+\frac{V_2^2}{2}+gz_2=\frac{p_1}{\rho}+\frac{V_1^2}{2}+gz_1+w_{\substack{\text{shaft}\\ \text{net in}}}-\text{loss} \qquad (1)$$

As in Example 5.26, we can see with Eq. 1 that a "useful effect" in this fan can be defined as

$$\text{useful effect}=w_{\substack{\text{shaft}\\ \text{net in}}}-\text{loss}$$

$$=\left(\frac{p_2}{\rho}+\frac{V_2^2}{2}+gz_2\right)-\left(\frac{p_1}{\rho}+\frac{V_1^2}{2}+gz_1\right) \qquad (2)\ \textbf{(Ans)}$$

In other words, only a portion of the shaft work delivered to the air by the fan blades is used to increase the available energy of the air; the rest is lost because of fluid friction.

A meaningful efficiency equation involves the ratio of shaft work converted into a useful effect (Eq. 2) to shaft work into the air, $w_{\text{shaft net in}}$. Thus, we can express efficiency, $\eta$, as

$$\eta=\frac{w_{\substack{\text{shaft}\\ \text{net in}}}-\text{loss}}{w_{\substack{\text{shaft}\\ \text{net in}}}} \qquad (3)$$

However, when Eq. 5.54, which was developed from the moment-of-momentum equation (Eq. 5.42), is applied to the contents of the control volume of Fig. E5.19, we obtain

$$w_{\substack{\text{shaft}\\ \text{net in}}}=+U_2V_{\theta 2} \qquad (4)$$

Combining Eqs. 2, 3, and 4, we obtain

$$\eta=\{[(p_2/\rho)+(V_2^2/2)+gz_2]-[(p_1/\rho)+(V_1^2/2)+gz_1]\}/U_2V_{\theta 2} \qquad (5)\ \textbf{(Ans)}$$

Equation 5 provides us with a practical means to evaluate the efficiency of the fan of Example 5.19.

Combining Eqs. 2 and 4, we obtain

$$\text{loss}=U_2V_{\theta 2}-\left[\left(\frac{p_2}{\rho}+\frac{V_2^2}{2}+gz_2\right)-\left(\frac{p_1}{\rho}+\frac{V_1^2}{2}+gz_1\right)\right] \qquad (6)\ \textbf{(Ans)}$$

**COMMENT** Equation 6 provides us with a useful method of evaluating the loss due to fluid friction in the fan of Example 5.19 in terms of fluid mechanical variables that can be measured.

## 5.4 Second Law of Thermodynamics—Irreversible Flow[5]

The second law of thermodynamics affords us with a means to formalize the inequality

$$\breve{u}_2-\breve{u}_1-q_{\substack{\text{net}\\ \text{in}}}\geq 0 \qquad (5.90)$$

*The second law of thermodynamics formalizes the notion of loss.*

for steady, incompressible, one-dimensional flow with friction (see Eq. 5.73). In this section we continue to develop the notion of loss of useful or available energy for flow with friction. Minimization of loss of available energy in any flow situation is of obvious engineering importance.

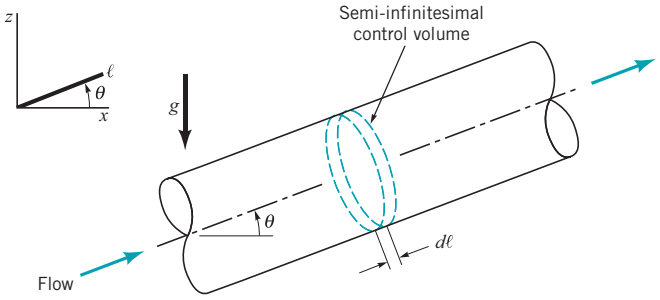
### 5.4.1 Semi-infinitesimal Control Volume Statement of the Energy Equation

If we apply the one-dimensional, steady flow energy equation, Eq. 5.70, to the contents of a control volume that is infinitesimally thin as illustrated in Fig 5.8, the result is

$$\dot{m}\left[d\breve{u}+d\left(\frac{p}{\rho}\right)+d\left(\frac{V^2}{2}\right)+g\,(dz)\right]=\delta\dot{Q}_{\substack{\text{net}\\ \text{in}}} \qquad (5.91)$$

[5]This entire section may be omitted without loss of continuity in the text material.

CMS Cepcor EX 1046 Page 149
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 5.9** **Semi-infinitesimal control volume.**

For all pure substances including common engineering working fluids, such as air, water, oil, and gasoline, the following relationship is valid (see, for example, Ref. 3).

$$T\,ds = d\check{u} + pd\left(\frac{1}{\rho}\right) \tag{5.92}$$

where $T$ is the absolute temperature and $s$ is the *entropy* per unit mass.

Combining Eqs. 5.91 and 5.92 we get

$$\dot{m}\left[T\,ds - pd\left(\frac{1}{\rho}\right) + d\left(\frac{p}{\rho}\right) + d\left(\frac{V^2}{2}\right) + g\,dz\right] = \delta\dot{Q}_{\substack{\text{net}\\\text{in}}}$$

or, dividing through by $\dot{m}$ and letting $\delta q_{\substack{\text{net}\\\text{in}}} = \delta\dot{Q}_{\substack{\text{net}\\\text{in}}}/\dot{m}$, we obtain

$$\frac{dp}{\rho} + d\left(\frac{V^2}{2}\right) + g\,dz = -(T\,ds - \delta q_{\substack{\text{net}\\\text{in}}}) \tag{5.93}$$

### 5.4.2 Semi-infinitesimal Control Volume Statement of the Second Law of Thermodynamics

A general statement of the second law of thermodynamics is

$$\frac{D}{Dt}\int_{\text{sys}} s\rho\,d\forall \geq \sum\left(\frac{\delta\dot{Q}^{\text{net}}_{\text{in}}}{T}\right)_{\text{sys}} \tag{5.94}$$

or in words,

*The second law of thermodynamics involves entropy, heat transfer, and temperature.*

| | | |
|---|---|---|
| the time rate of increase of the entropy of a system | ≥ | sum of the ratio of net heat transfer rate into system to absolute temperature for each particle of mass in the system receiving heat from surroundings |

The right-hand side of Eq. 5.94 is identical for the system and control volume at the instant when system and control volume are coincident; thus,

$$\sum\left(\frac{\delta\dot{Q}^{\text{net}}_{\text{in}}}{T}\right)_{\text{sys}} = \sum\left(\frac{\delta\dot{Q}^{\text{net}}_{\text{in}}}{T}\right)_{\text{cv}} \tag{5.95}$$

With the help of the Reynolds transport theorem (Eq. 4.19) the system time derivative can be expressed for the contents of the coincident control volume that is fixed and nondeforming. Using Eq. 4.19, we obtain

$$\frac{D}{Dt}\int_{\text{sys}} s\rho\,d\forall = \frac{\partial}{\partial t}\int_{\text{cv}} s\rho\,d\forall + \int_{\text{cs}} s\rho\mathbf{V}\cdot\hat{\mathbf{n}}\,dA \tag{5.96}$$

For a fixed, nondeforming control volume, Eqs. 5.94, 5.95, and 5.96 combine to give

$$\frac{\partial}{\partial t}\int_{cv} s\rho\, d\forall + \int_{cs} s\rho \mathbf{V}\cdot\hat{\mathbf{n}}\, dA \geq \sum\left(\frac{\delta\dot{Q}_{\substack{net\\in}}}{T}\right)_{cv} \tag{5.97}$$

At any instant for steady flow

$$\frac{\partial}{\partial t}\int_{cv} s\rho\, d\forall = 0 \tag{5.98}$$

If the flow consists of only one stream through the control volume and if the properties are uniformly distributed (one-dimensional flow), Eqs. 5.97 and 5.98 lead to

$$\dot{m}(s_{out} - s_{in}) \geq \sum \frac{\delta\dot{Q}_{\substack{net\\in}}}{T} \tag{5.99}$$

For the infinitesimally thin control volume of Fig. 5.8, Eq. 5.99 yields

$$\dot{m}\, ds \geq \sum \frac{\delta\dot{Q}_{\substack{net\\in}}}{T} \tag{5.100}$$

If all of the fluid in the infinitesimally thin control volume is considered as being at a uniform temperature, $T$, then from Eq. 5.100 we get

*The relationship between entropy and heat transfer rate depends on the process involved.*

$$T\, ds \geq \delta q_{\substack{net\\in}}$$

or

$$T\, ds - \delta q_{\substack{net\\in}} \geq 0 \tag{5.101}$$

The equality is for any reversible (frictionless) process; the inequality is for all irreversible (friction) processes.

### 5.4.3 Combination of the Equations of the First and Second Laws of Thermodynamics

Combining Eqs. 5.93 and 5.101, we conclude that

$$-\left[\frac{dp}{\rho} + d\left(\frac{V^2}{2}\right) + g\, dz\right] \geq 0 \tag{5.102}$$

The equality is for any steady, reversible (frictionless) flow, an important example being flow for which the Bernoulli equation (Eq. 3.7) is applicable. The inequality is for all steady, irreversible (friction) flows. The actual amount of the inequality has physical significance. It represents the extent of loss of useful or available energy which occurs because of irreversible flow phenomena including viscous effects. Thus, Eq. 5.102 can be expressed as

$$-\left[\frac{dp}{\rho} + d\left(\frac{V^2}{2}\right) + g\, dz\right] = \delta(\text{loss}) = (T\, ds - \delta q_{\substack{net\\in}}) \tag{5.103}$$

The irreversible flow loss is zero for a frictionless flow and greater than zero for a flow with frictional effects. Note that when the flow is frictionless, Eq. 5.103 multiplied by density, $\rho$, is identical to Eq. 3.5. Thus, for steady frictionless flow, Newton's second law of motion (see Section 3.1) and the first and second laws of thermodynamics lead to the same differential equation,

$$\frac{dp}{\rho} + d\left(\frac{V^2}{2}\right) + g\, dz = 0 \tag{5.104}$$

CMS Cepcor EX 1046 Page 151
CMS Cepcor v. Sandvik IPR2024-01380

If some shaft work is involved, then the flow must be at least locally unsteady in a cyclical way and the appropriate form of the energy equation for the contents of an infinitesimally thin control volume can be developed starting with Eq. 5.67. The resulting equation is

$$-\left[\frac{dp}{\rho}+d\left(\frac{V^2}{2}\right)+g\,dz\right]=\delta(\text{loss})-\delta w_{\substack{\text{shaft}\\ \text{net in}}} \tag{5.105}$$

Equations 5.103 and 5.105 are valid for incompressible and compressible flows. If we combine Eqs. 5.92 and 5.103, we obtain

$$d\breve{u}+pd\left(\frac{1}{\rho}\right)-\delta q_{\substack{\text{net}\\ \text{in}}}=\delta(\text{loss}) \tag{5.106}$$

For incompressible flow, $d(1/\rho)=0$ and, thus, from Eq. 5.106,

$$d\breve{u}-\delta q_{\substack{\text{net}\\ \text{in}}}=\delta(\text{loss}) \tag{5.107}$$

Applying Eq. 5.107 to a finite control volume, we obtain

$$\breve{u}_{\text{out}}-\breve{u}_{\text{in}}-q_{\substack{\text{net}\\ \text{in}}}=\text{loss}$$

which is the same conclusion we reached earlier (see Eq. 5.78) for incompressible flows.

For compressible flow, $d(1/\rho)\neq 0$, and thus when we apply Eq. 5.106 to a finite control volume we obtain

$$\breve{u}_{\text{out}}-\breve{u}_{\text{in}}+\int_{\text{in}}^{\text{out}}pd\left(\frac{1}{\rho}\right)-q_{\substack{\text{net}\\ \text{in}}}=\text{loss} \tag{5.108}$$

indicating that $u_{\text{out}}-u_{\text{in}}-q_{\text{net in}}$ is not equal to loss.

### 5.4.4 Application of the Loss Form of the Energy Equation

*Zero loss is associated with the Bernoulli equation.*

Steady flow along a pathline in an incompressible and frictionless flow field provides a simple application of the loss form of the energy equation (Eq. 5.105). We start with Eq. 5.105 and integrate it term by term from one location on the pathline, section (1), to another one downstream, section (2). Note that because the flow is frictionless, loss = 0. Also, because the flow is steady throughout, $w_{\text{shaft net in}}=0$. Since the flow is incompressible, the density is constant. The control volume in this case is an infinitesimally small diameter streamtube (Fig. 5.7). The resultant equation is

$$\frac{p_2}{\rho}+\frac{V_2^2}{2}+gz_2=\frac{p_1}{\rho}+\frac{V_1^2}{2}+gz_1 \tag{5.109}$$

which is identical to the Bernoulli equation (Eq. 3.7) already discussed in Chapter 3.

If the frictionless and steady pathline flow of the fluid particle considered above was compressible, application of Eq. 5.105 would yield

$$\int_1^2\frac{dp}{\rho}+\frac{V_2^2}{2}+gz_2=\frac{V_1^2}{2}+gz_1 \tag{5.110}$$

To carry out the integration required, $\int_1^2(dp/\rho)$, a relationship between fluid density, $\rho$, and pressure, $p$, must be known. If the frictionless compressible flow we are considering is adiabatic and involves the flow of an ideal gas, it is shown in Section 11.1 that

$$\frac{p}{\rho^k}=\text{constant} \tag{5.111}$$

where $k=c_p/c_v$ is the ratio of gas specific heats, $c_p$ and $c_v$, which are properties of the fluid. Using Eq. 5.111 we get

$$\int_1^2\frac{dp}{\rho}=\frac{k}{k-1}\left(\frac{p_2}{\rho_2}-\frac{p_1}{\rho_1}\right) \tag{5.112}$$

CMS Cepcor EX 1046 Page 152
CMS Cepcor v. Sandvik IPR2024-01380

Thus, Eqs. 5.110 and 5.112 lead to

$$\frac{k}{k-1}\frac{p_2}{\rho_2}+\frac{V_2^2}{2}+gz_2=\frac{k}{k-1}\frac{p_1}{\rho_1}+\frac{V_1^2}{2}+gz_1 \tag{5.113}$$

Note that this equation is identical to Eq. 3.24. An example application of Eqs. 5.109 and 5.113 follows.

## EXAMPLE 5.29 Energy—Comparison of Compressible and Incompressible Flow

**GIVEN** Air steadily expands adiabatically and without friction from stagnation conditions of 100 psia and 520 °R to 14.7 psia.

**FIND** Determine the velocity of the expanded air assuming (a) incompressible flow, (b) compressible flow.

### SOLUTION

**(a)** If the flow is considered incompressible, the Bernoulli equation, Eq. 5.109, can be applied to flow through an infinitesimal cross-sectional streamtube, like the one in Fig. 5.7, from the stagnation state (1) to the expanded state (2). From Eq. 5.109 we get

0 (1 is the stagnation state)

$$\frac{p_2}{\rho}+\frac{V_2^2}{2}+\cancel{gz_2}=\frac{p_1}{\rho}+\cancel{\frac{V_1^2}{2}}+gz_1 \tag{1}$$

0 (changes in $gz$ are negligible for air flow)

or

$$V_2=\sqrt{2\left(\frac{p_1-p_2}{\rho}\right)}$$

We can calculate the density at state (1) by assuming that air behaves like an ideal gas,

$$\rho=\frac{p_1}{RT_1}=\frac{(100\text{ psia})(144\text{ in.}^2/\text{ft}^2)}{(1716\text{ ft}\cdot\text{lb/slug}\cdot{}^\circ\text{R})(520\text{ }^\circ\text{R})}=0.0161\text{ slug/ft}^3 \tag{2}$$

Thus,

$$V_2=\sqrt{\frac{2(100\text{ psia}-14.7\text{ psia})(144\text{ in.}^2/\text{ft}^2)}{(0.016\text{ slug/ft}^3)[1\text{ (lb}\cdot\text{s}^2)/(\text{slug}\cdot\text{ft})]}}=1240\text{ ft/s}\quad\textbf{(Ans)}$$

The assumption of incompressible flow is not valid in this case since for air a change from 100 psia to 14.7 psia would undoubtedly result in a significant density change.

**(b)** If the flow is considered compressible, Eq. 5.113 can be applied to the flow through an infinitesimal cross-sectional control volume, like the one in Fig. 5.7, from the stagnation state (1) to the expanded state (2). We obtain

0 (1 is the stagnation state)

$$\frac{k}{k-1}\frac{p_2}{\rho_2}+\frac{V_2^2}{2}+\cancel{gz_2}=\frac{k}{k-1}\frac{p_1}{\rho_1}+\cancel{\frac{V_1^2}{2}}+gz_1 \tag{3}$$

0 (changes in $gz$ are negligible for air flow)

or

$$V_2=\sqrt{\frac{2k}{k-1}\left(\frac{p_1}{\rho_1}-\frac{p_2}{\rho_2}\right)} \tag{4}$$

Given in the problem statement are values of $p_1$ and $p_2$. A value of $\rho_1$ was calculated earlier (Eq. 2). To determine $\rho_2$ we need to make use of a property relationship for reversible (frictionless) and adiabatic flow of an ideal gas that is derived in Chapter 11; namely,

$$\frac{p}{\rho^k}=\text{constant} \tag{5}$$

where $k=1.4$ for air. Solving Eq. 5 for $\rho_2$ we get

$$\rho_2=\rho_1\left(\frac{p_2}{p_1}\right)^{1/k}$$

or

$$\rho_2=(0.0161\text{ slug/ft}^3)\left[\frac{14.7\text{ psia}}{100\text{ psia}}\right]^{1/1.4}=0.00409\text{ slug/ft}^3$$

Then, from Eq. 4, with $p_1=100\text{ lb/in.}^2(144\text{ in.}^2/\text{ft}^2)=14{,}400\text{ lb/ft}^2$ and $p_2=14.7\text{ lb/in.}^2(144\text{ in.}^2/\text{ft}^2)=2117\text{ lb/ft}^2$,

$$V_2=\sqrt{\frac{(2)(1.4)}{1.4-1}\left(\frac{14{,}400\text{ lb/ft}^2}{0.0161\text{ slug/ft}^3}-\frac{2117\text{ lb/ft}^2}{0.00409\text{ slug/ft}^3}\right)}=1620\text{ (lb}\cdot\text{ft/slug)}^{1/2}[(1\text{ slug}\cdot\text{ft/s}^2)/\text{lb}]^{1/2}$$

or

$$V_2=1620\text{ ft/s}\quad\textbf{(Ans)}$$

**COMMENT** A considerable difference exists between the air velocities calculated assuming incompressible and compressible flow. In Section 3.8.1, a discussion of when a fluid flow may be appropriately considered incompressible is provided. Basically, when flow speed is less than a third of the speed of sound in the fluid involved, incompressible flow may be assumed with only a small error.

## 5.5 Chapter Summary and Study Guide

In this chapter the flow of a fluid is analyzed by using important principles including conservation of mass, Newton's second law of motion, and the first and second laws of thermodynamics as applied to control volumes. The Reynolds transport theorem is used to convert basic system-orientated laws into corresponding control volume formulations.

The continuity equation, a statement of the fact that mass is conserved, is obtained in a form that can be applied to any flow—steady or unsteady, incompressible or compressible. Simplified forms of the continuity equation enable tracking of fluid everywhere in a control volume, where it enters, where it leaves, and within. Mass or volume flowrates of fluid entering or leaving a control volume and rate of accumulation or depletion of fluid within a control volume can be estimated.

The linear momentum equation, a form of Newton's second law of motion applicable to flow of fluid through a control volume, is obtained and used to solve flow problems. Net force results from or causes changes in linear momentum (velocity magnitude and/or direction) of fluid flowing through a control volume. Work and power associated with force can be involved.

The moment-of-momentum equation, which involves the relationship between torque and changes in angular momentum, is obtained and used to solve flow problems dealing with turbines (energy extracted from a fluid) and pumps (energy supplied to a fluid).

The steady-state energy equation, obtained from the first law of thermodynamics (conservation of energy), is written in several forms. The first (Eq. 5.69) involves power terms. The second form (Eq. 5.82 or 5.84) is termed the mechanical energy equation or the extended Bernoulli equation. It consists of the Bernoulli equation with extra terms that account for energy losses due to friction in the flow, as well as terms accounting for the work of pumps or turbines in the flow.

*conservation of mass*
*continuity equation*
*mass flowrate*
*linear momentum equation*
*moment-of-momentum equation*
*shaft power*
*shaft torque*
*first law of thermodynamics*
*heat transfer rate*
*energy equation*
*loss*
*shaft work head*
*head loss*
*kinetic energy coefficient*

The following checklist provides a study guide for this chapter. When your study of the entire chapter and end-of-chapter exercises has been completed you should be able to

- write out meanings of the terms listed here in the margin and understand each of the related concepts. These terms are particularly important and are set in ***italic, bold, and color*** type in the text.
- select an appropriate control volume for a given problem and draw an accurately labeled control volume diagram.
- use the continuity equation and a control volume to solve problems involving mass or volume flowrate.
- use the linear momentum equation and a control volume, in conjunction with the continuity equation as necessary, to solve problems involving forces related to linear momentum change.
- use the moment-of-momentum equation to solve problems involving torque and related work and power due to angular momentum change.
- use the energy equation, in one of its appropriate forms, to solve problems involving losses due to friction (head loss) and energy input by pumps or extraction by turbines.
- use the kinetic energy coefficient in the energy equation to account for nonuniform flows.

Some of the important equations in this chapter are given below.

| | | |
|---|---|---|
| Conservation of mass | $\dfrac{\partial}{\partial t}\displaystyle\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA = 0$ | **(5.5)** |
| Mass flowrate | $\dot{m} = \rho Q = \rho A V$ | **(5.6)** |
| Average velocity | $\overline{V} = \dfrac{\displaystyle\int_A \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA}{\rho A}$ | **(5.7)** |
| Steady flow mass conservation | $\sum \dot{m}_{out} - \sum \dot{m}_{in} = 0$ | **(5.9)** |
| Moving control volume mass conservation | $\dfrac{\partial}{\partial t}\displaystyle\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{W} \cdot \hat{\mathbf{n}} \, dA = 0$ | **(5.16)** |

| | | |
|---|---|---|
| Deforming control volume mass conservation | $\dfrac{DM_{sys}}{Dt} = \dfrac{\partial}{\partial t}\displaystyle\int_{cv} \rho \, d\forall + \int_{cs} \rho \mathbf{W} \cdot \hat{\mathbf{n}} \, dA = 0$ | (5.17) |
| Force related to change in linear momentum | $\dfrac{\partial}{\partial t}\displaystyle\int_{cv} \mathbf{V}\rho \, d\forall + \int_{cs} \mathbf{V}\rho\mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \sum \mathbf{F}_{\substack{\text{contents of the} \\ \text{control volume}}}$ | (5.22) |
| Moving control volume force related to change in linear momentum | $\displaystyle\int_{cs} \mathbf{W}\rho\mathbf{W} \cdot \hat{\mathbf{n}} \, dA = \sum \mathbf{F}_{\substack{\text{contents of the} \\ \text{control volume}}}$ | (5.29) |
| Vector addition of absolute and relative velocities | $\mathbf{V} = \mathbf{W} + \mathbf{U}$ | (5.43) |
| Shaft torque from force | $\sum \left[ (\mathbf{r} \times \mathbf{F})_{\substack{\text{contents of the} \\ \text{control volume}}} \right]_{\text{axial}} = \mathbf{T}_{\text{shaft}}$ | (5.45) |
| Shaft torque related to change in moment-of-momentum (angular momentum) | $T_{\text{shaft}} = (-\dot{m}_{\text{in}})(\pm r_{\text{in}} V_{\theta \text{in}}) + \dot{m}_{\text{out}}(\pm r_{\text{out}} V_{\theta \text{out}})$ | (5.50) |
| Shaft power related to change in moment-of-momentum (angular momentum) | $\dot{W}_{\text{shaft}} = (-\dot{m}_{\text{in}})(\pm U_{\text{in}} V_{\theta \text{in}}) + \dot{m}_{\text{out}}(\pm U_{\text{out}} V_{\theta \text{out}})$ | (5.53) |
| First law of thermodynamics (Conservation of energy) | $\dfrac{\partial}{\partial t}\displaystyle\int_{cv} e\rho \, d\forall + \int_{cs} \left( \check{u} + \frac{p}{\rho} + \frac{V^2}{2} + gz \right) \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA = \dot{Q}_{\substack{\text{net} \\ \text{in}}} + \dot{W}_{\substack{\text{shaft} \\ \text{net in}}}$ | (5.64) |
| Conservation of power | $\dot{m}\left[ \check{h}_{\text{out}} - \check{h}_{\text{in}} + \dfrac{V_{\text{out}}^2 - V_{\text{in}}^2}{2} + g(z_{\text{out}} - z_{\text{in}}) \right] = \dot{Q}_{\substack{\text{net} \\ \text{in}}} + \dot{W}_{\substack{\text{shaft} \\ \text{net in}}}$ | (5.69) |
| Conservation of mechanical energy | $\dfrac{p_{\text{out}}}{\rho} + \dfrac{V_{\text{out}}^2}{2} + gz_{\text{out}} = \dfrac{p_{\text{in}}}{\rho} + \dfrac{V_{\text{in}}^2}{2} + gz_{\text{in}} + w_{\substack{\text{shaft} \\ \text{net in}}} - \text{loss}$ | (5.82) |

## References


1. Eck, B., *Technische Stromungslehre*, Springer-Verlag, Berlin, Germany, 1957.
2. Dean, R. C., "On the Necessity of Unsteady Flow in Fluid Machines," *ASME Journal of Basic Engineering* 81D; 24–28, March 1959.
3. Moran, M. J., and Shapiro, H. N., *Fundamentals of Engineering Thermodynamics*, 6th Ed., Wiley, New York, 2008.


## Review Problems

Go to Appendix G for a set of review problems with answers. Detailed solutions can be found in *Student Solution Manual and Study Guide for Fundamentals of Fluid Mechanics*, by Munson et al. (© 2009 John Wiley and Sons, Inc.).

## Problems

**Note: Unless otherwise indicated, use the values of fluid properties found in the tables on the inside of the front cover. Problems designated with an (*) are intended to be solved with the aid of a programmable calculator or a computer. Problems designated with a (†) are "open-ended" problems and require critical thinking in that to work them one must make various assumptions and provide the necessary data. There is not a unique answer to these problems.**

**Answers to the even-numbered problems are listed at the end of the book. Access to the videos that accompany problems can be obtained through the book's web site, www.wiley.com/college/munson. The lab-type problems can also be accessed on this web site.**

### Section 5.1.1 Derivation of the Continuity Equation

**5.1** Explain why the mass of the contents of a system is constant with time.

**5.2** Explain how the mass of the contents of a control volume can vary with time or not.

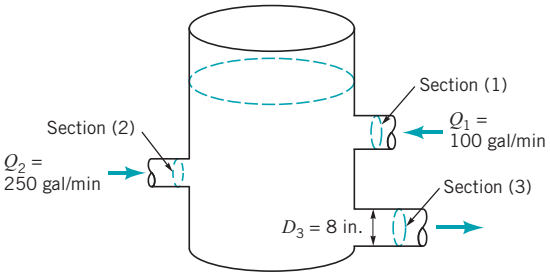
**5.3** Explain the concept of a coincident control volume and system and why it is useful.

**5.4** Obtain a photograph/image of a situation for which the conservation of mass law is important. Briefly describe the situation and its relevance.

CMS Cepcor EX 1046 Page 155
CMS Cepcor v. Sandvik IPR2024-01380

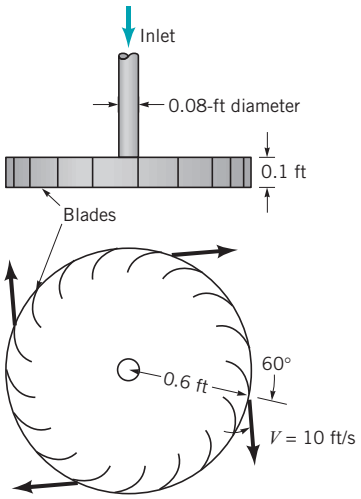
### Section 5.1.2 Fixed, Nondeforming Control Volume—Uniform Velocity Profile or Average Velocity.

**5.5** Water enters a cylindrical tank through two pipes at rates of 250 and 100 gal/min (see Fig. P5.5). If the level of the water in the tank remains constant, calculate the average velocity of the flow leaving the tank through an 8-in. inside-diameter pipe.



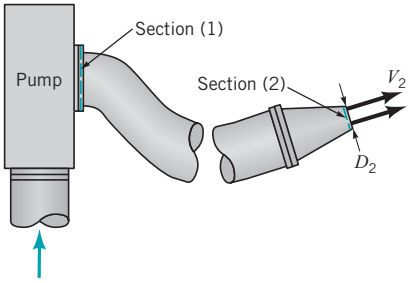

■ FIGURE P5.5

**5.6** Water flows out through a set of thin, closely spaced blades as shown in Fig. 5.6 with a speed of $V = 10$ ft/s around the entire circumference of the outlet. Determine the mass flowrate through the inlet pipe.



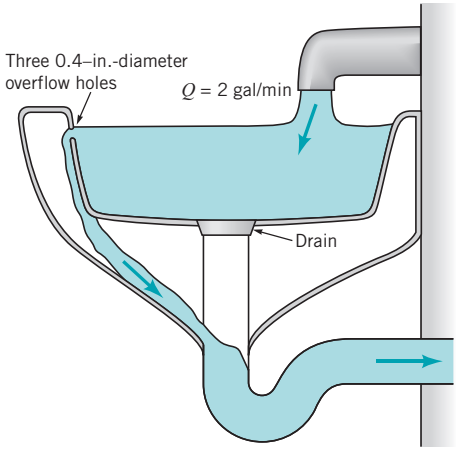

■ FIGURE P5.6

**5.7** The pump shown in Fig. P5.7 produces a steady flow of 10 gal/s through the nozzle. Determine the nozzle exit diameter, $D_2$, if the exit velocity is to be $V_2 = 100$ ft/s.



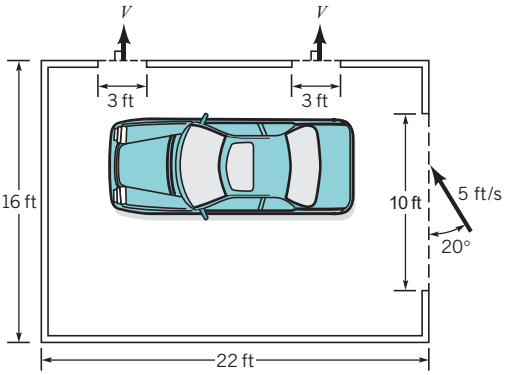

■ FIGURE P5.7

**5.8** Water flows into a sink as shown in Video V5.1 and Fig. P5.8 at a rate of 2 gallons per minute. Determine the average velocity through each of the three 0.4-in.-diameter overflow holes if the drain is closed and the water level in the sink remains constant.




■ FIGURE P5.8

**5.9** The wind blows through a 7 ft × 10 ft garage door opening with a speed of 5 ft/s as shown in Fig. P5.9. Determine the average speed, $V$, of the air through the two 3 ft × 4 ft openings in the windows.




■ FIGURE P5.9

**5.10** The human circulatory system consists of a complex branching pipe network ranging in diameter from the aorta (largest) to the capillaries (smallest). The average radii and the number of these vessels is shown in the table below. Does the average blood velocity increase, decrease, or remain constant as it travels from the aorta to the capillaries?

| Vessel | Average Radius, mm | Number |
|---|---|---|
| Aorta | 12.5 | 1 |
| Arteries | 2.0 | 159 |
| Arterioles | 0.03 | $1.4 \times 10^7$ |
| Capillaries | 0.006 | $3.9 \times 10^9$ |

**5.11** Air flows steadily between two cross sections in a long, straight section of 0.1-m inside diameter pipe. The static temperature and pressure at each section are indicated in Fig. P5.11. If the average air velocity at section (1) is 205 m/s, determine the average air velocity at section (2).

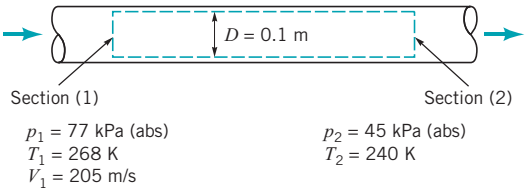



**FIGURE P5.11**

**5.12** A hydraulic jump (see Video V10.10) is in place downstream from a spillway as indicated in Fig. P5.12. Upstream of the jump, the depth of the stream is 0.6 ft and the average stream velocity is 18 ft/s. Just downstream of the jump, the average stream velocity is 3.4 ft/s. Calculate the depth of the stream, $h$, just downstream of the jump.

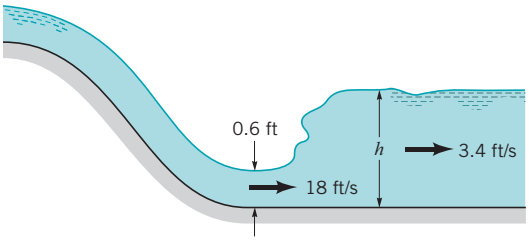



**FIGURE P5.12**

**5.13** An evaporative cooling tower (see Fig. P5.13) is used to cool water from 110 to 80°F. Water enters the tower at a rate of 250,000 lbm/hr. Dry air (no water vapor) flows into the tower at a rate of 151,000 lbm/hr. If the rate of wet air flow out of the tower is 156,900 lbm/hr, determine the rate of water evaporation in lbm/hr and the rate of cooled water flow in lbm/hr.

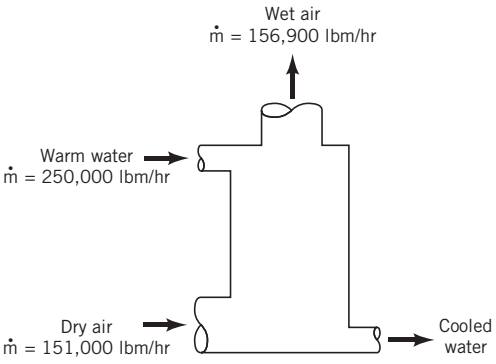



**FIGURE P5.13**

**5.14** At cruise conditions, air flows into a jet engine at a steady rate of 65 lbm/s. Fuel enters the engine at a steady rate of 0.60 lbm/s. The average velocity of the exhaust gases is 1500 ft/s relative to the engine. If the engine exhaust effective cross-sectional area is 3.5 ft$^2$, estimate the density of the exhaust gases in lbm/ft$^3$.

**5.15** Water at 0.1 m$^3$/s and alcohol ($SG$=0.8) at 0.3 m$^3$/s are mixed in a $y$-duct as shown in Fig. 5.15. What is the average density of the mixture of alcohol and water?

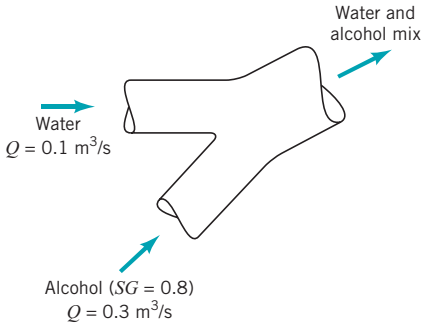



**FIGURE P5.15**

**5.16** Freshwater flows steadily into an open 55-gal drum initially filled with seawater. The freshwater mixes thoroughly with the seawater and the mixture overflows out of the drum. If the freshwater flowrate is 10 gal/min, estimate the time in seconds required to decrease the difference between the density of the mixture and the density of freshwater by 50%.

## Section 5.1.2 Fixed, Nondeforming Control Volume—Nonuniform Velocity Profile

**5.17** A water jet pump (see Fig. P5.17) involves a jet cross-sectional area of 0.01 m$^2$, and a jet velocity of 30 m/s. The jet is surrounded by entrained water. The total cross-sectional area associated with the jet and entrained streams is 0.075 m$^2$. These two fluid streams leave the pump thoroughly mixed with an average velocity of 6 m/s through a cross-sectional area of 0.075 m$^2$. Determine the pumping rate (i.e., the entrained fluid flowrate) involved in liters/s.

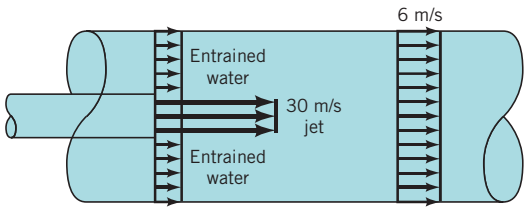



**FIGURE P5.17**

**5.18** Two rivers merge to form a larger river as shown in Fig. P5.18. At a location downstream from the junction (before the two streams completely merge), the nonuniform velocity profile is as shown and the depth is 6 ft. Determine the value of $V$.

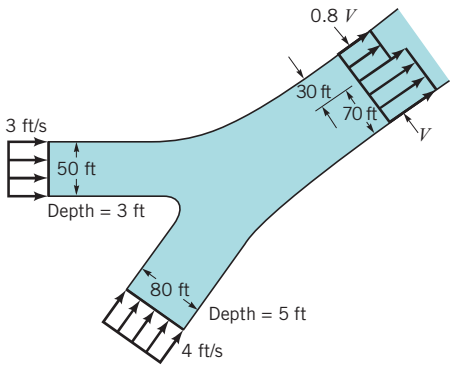
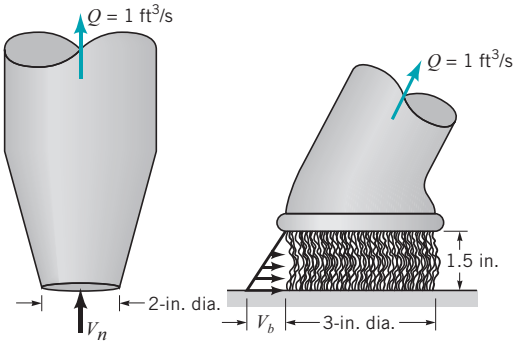



**FIGURE P5.18**

**5.19** Various types of attachments can be used with the shop vac shown in Video V5.2. Two such attachments are shown in Fig. P5.19—a nozzle and a brush. The flowrate is 1 ft$^3$/s. **(a)** Determine the average velocity through the nozzle entrance, $V_n$. **(b)** Assume the air enters the brush attachment in a radial direction all around the brush with a velocity profile that varies linearly from 0 to $V_b$ along the length of the bristles as shown in the figure. Determine the value of $V_b$.



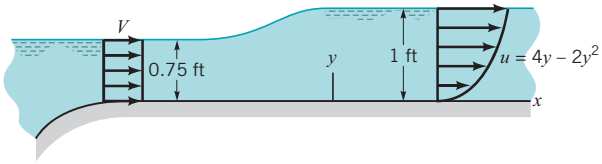

■ FIGURE P5.19

**5.20** An appropriate turbulent pipe flow velocity profile is

$$\mathbf{V} = u_c\left(\frac{R-r}{R}\right)^{1/n}\hat{\mathbf{i}}$$

where $u_c$ = centerline velocity, $r$ = local radius, $R$ = pipe radius, and $\hat{\mathbf{i}}$ = unit vector along pipe centerline. Determine the ratio of average velocity, $\bar{u}$, to centerline velocity, $u_c$, for **(a)** $n = 4$, **(b)** $n = 6$, **(c)** $n = 8$, **(d)** $n = 10$. Compare the different velocity profiles.

**5.21** As shown in Fig. P5.21, at the entrance to a 3-ft-wide channel the velocity distribution is uniform with a velocity $V$. Further downstream the velocity profile is given by $u = 4y - 2y^2$, where $u$ is in ft/s and $y$ is in ft. Determine the value of $V$.

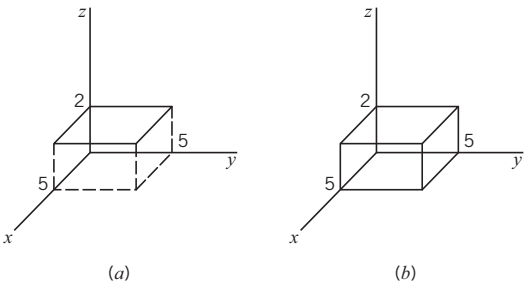



■ FIGURE P5.21

**5.22** A water flow situation is described by the velocity field equation

$$\mathbf{V} = (3x + 2)\,\hat{\mathbf{i}} + (2y - 4)\,\hat{\mathbf{j}} - 5z\hat{\mathbf{k}}\ \text{ft/s}$$

where $x$, $y$, and $z$ are in feet. **(a)** Determine the mass flowrate through the rectangular area in the plane corresponding to $z = 2$ feet having corners at $(x, y, z) = (0, 0, 2)$, $(5, 0, 2)$, $(5, 5, 2)$, and $(0, 5, 2)$ as shown in Fig P5.22*a*. **(b)** Show that mass is conserved in the control volume having corners at $(x, y, z) = (0, 0, 2)$, $(5, 0, 2)$, $(5, 5, 2)$, $(0, 5, 2)$, $(0, 0, 0)$, $(5, 0, 0)$, $(5, 5, 0)$, and $(0, 5, 0)$, as shown in Fig. P5.22*b*.

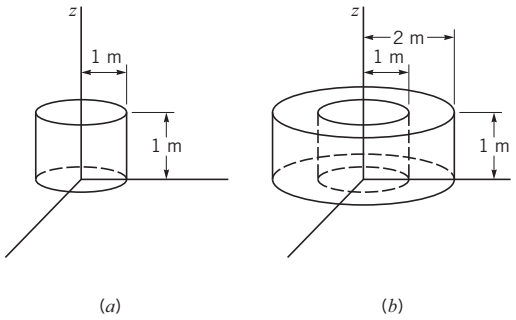



■ FIGURE P5.22

**5.23** An incompressible flow velocity field (water) is given as

$$\mathbf{V} = -\frac{1}{r}\hat{\mathbf{e}}_r + \frac{1}{r}\hat{\mathbf{e}}_\theta\ \text{m/s}$$

where $r$ is in meters. **(a)** Calculate the mass flowrate through the cylindrical surface at $r = 1$ m from $z = 0$ to $z = 1$ m as shown in Fig.P5.23*a*. **(b)** Show that mass is conserved in the annular control volume from $r = 1$ m to $r = 2$ m and $z = 0$ to $z = 1$ m as shown in Fig. P5.23*b*.

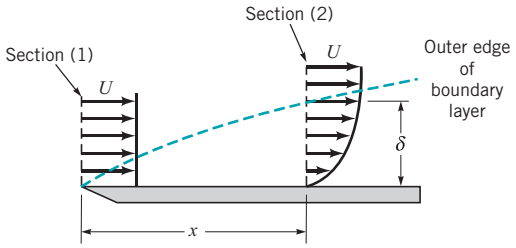



■ FIGURE P5.23

**5.24** Flow of a viscous fluid over a flat plate surface results in the development of a region of reduced velocity adjacent to the wetted surface as depicted in Fig. P5.24. This region of reduced flow is called a boundary layer. At the leading edge of the plate, the velocity profile may be considered uniformly distributed with a value $U$. All along the outer edge of the boundary layer, the fluid velocity component parallel to the plate surface is also $U$. If the $x$ direction velocity profile at section (2) is

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{1/7}$$

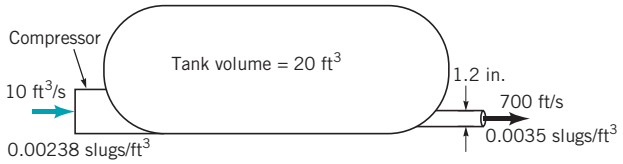
develop an expression for the volume flowrate through the edge of the boundary layer from the leading edge to a location downstream at $x$ where the boundary layer thickness is $\delta$.




■ FIGURE P5.24

## Section 5.1.2 Fixed, Nondeforming Control Volume—Unsteady Flow

**5.25** Air at standard conditions enters the compressor shown in Fig. P5.25 at a rate of 10 ft$^3$/s. It leaves the tank through a 1.2-in.-diameter pipe with a density of 0.0035 slugs/ft$^3$ and a uniform speed of 700 ft/s. **(a)** Determine the rate (slugs/s) at which the mass of air in the tank is increasing or decreasing. **(b)** Determine the average time rate of change of air density within the tank.




■ FIGURE P5.25

**5.26** Estimate the time required to fill with water a cone-shaped container (see Fig. P5.26) 5 ft high and 5 ft across at the top if the filling rate is 20 gal/min.

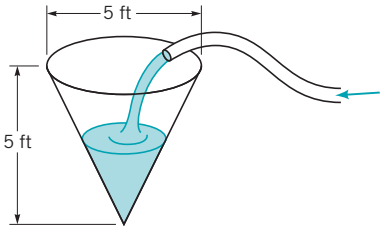



**FIGURE P5.26**

**†5.27** Estimate the maximum flowrate of rainwater (during a heavy rain) that you would expect from the downspout connected to the gutters of your house. List all assumptions and show all calculations.

### Section 5.1.3 Moving, Nondeforming Control Volume

**5.28** For an automobile moving along a highway, describe the control volume you would use to estimate the flowrate of air across the radiator. Explain how you would estimate the velocity of that air.

### Section 5.1.4 Deforming Control Volume

**5.29** A hypodermic syringe (see Fig. P5.29) is used to apply a vaccine. If the plunger is moved forward at the steady rate of 20 mm/s and if vaccine leaks past the plunger at 0.1 of the volume flowrate out the needle opening, calculate the average velocity of the needle exit flow. The inside diameters of the syringe and the needle are 20 mm and 0.7 mm.

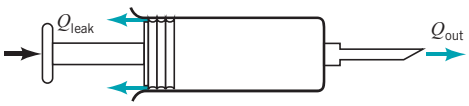



**FIGURE P5.29**

**5.30** The Hoover Dam (see **Video V2.4**) backs up the Colorado River and creates Lake Mead, which is approximately 115 miles long and has a surface area of approximately 225 square miles. If during flood conditions the Colorado River flows into the lake at a rate of 45,000 cfs and the outflow from the dam is 8000 cfs, how many feet per 24-hour day will the lake level rise?

**5.31** Storm sewer backup causes your basement to flood at the steady rate of 1 in. of depth per hour. The basement floor area is 1500 $ft^2$. What capacity (gal/min) pump would you rent to **(a)** keep the water accumulated in your basement at a constant level until the storm sewer is blocked off, and **(b)** reduce the water accumulation in your basement at a rate of 3 in./hr even while the backup problem exists?

**5.32** (See Fluids in the News article **"New 1.6 gpf standards,"** Section 5.1.2.) When a toilet is flushed, the water depth, $h$, in the tank as a function of time, $t$, is as given in the table. The size of the rectangular tank is 19 in. by 7.5 in. **(a)** Determine the volume of water used per flush, gpf. **(b)** Plot the flowrate for $0 \le t \le 6$ s.

| $t$ (s) | $h$ (in.) |
|---|---|
| 0 | 5.70 |
| 0.5 | 5.33 |
| 1.0 | 4.80 |
| 2.0 | 3.45 |
| 3.0 | 2.40 |
| 4.0 | 1.50 |
| 5.0 | 0.75 |
| 6.0 | 0 |

### Section 5.2.1 Derivation of the Linear Momentum Equation

**5.33** What is fluid linear momentum and the "flow" of linear momentum?

**5.34** Explain the physical meaning of each of the terms of the linear momentum equation (Eq. 5.22).

**5.35** What is an inertial control volume?

**5.36** Distinguish between body and surface forces.

**5.37** Obtain a photograph/image of a situation in which the linear momentum of a fluid changes during flow from one location to another. Explain briefly how force is involved.

### Section 5.2.2 Application of the Linear Momentum Equation (Also see Lab Problems 5.140, 5.141, 5.142, and 5.143.)

**5.38** A 10-mm diameter jet of water is deflected by a homogeneous rectangular block (15 mm by 200 mm by 100 mm) that weighs 6 N as shown in **Video V5.6** and Fig. P5.38. Determine the minimum volume flowrate needed to tip the block.

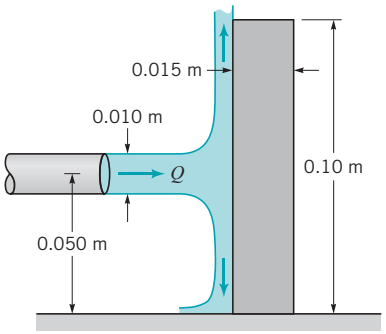



**FIGURE P5.38**

**5.39** Determine the anchoring force required to hold in place the conical nozzle attached to the end of the laboratory sink faucet shown in Fig. P5.39 when the water flowrate is 10 gal/min. The nozzle weight is 0.2 lb. The nozzle inlet and exit inside diameters are 0.6 and 0.2 in., respectively. The nozzle axis is vertical and the axial distance between sections (1) and (2) is 1.2 in. The pressure at section (1) is 68 psi.

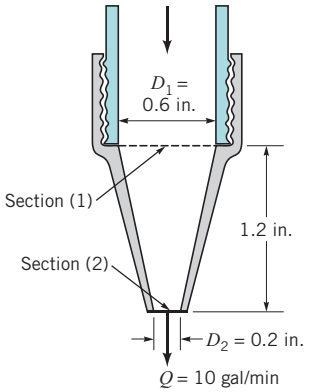
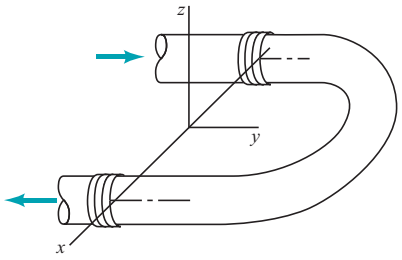



**FIGURE P5.39**

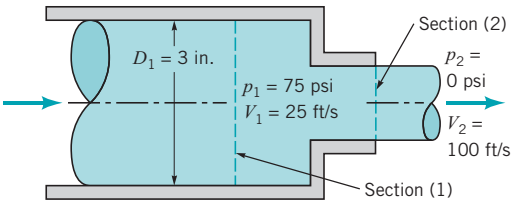
**5.40** Water flows through a horizontal, 180° pipe bend as is illustrated in Fig. P5.40. The flow cross section area is constant at a value of 9000 $mm^2$. The flow velocity everywhere in the bend is 15 m/s.




■ FIGURE P5.40

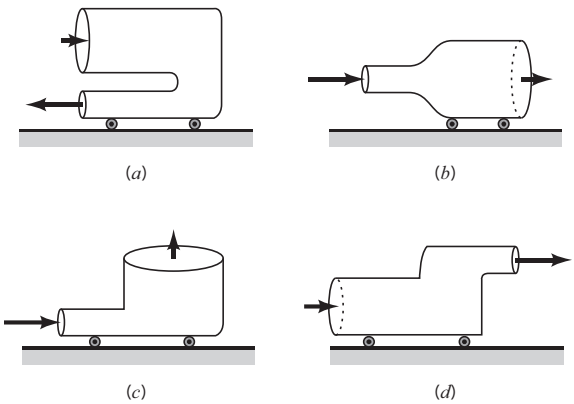
The pressures at the entrance and exit of the bend are 210 and 165 kPa, respectively. Calculate the horizontal ($x$ and $y$) components of the anchoring force needed to hold the bend in place.

**5.41** Water enters the horizontal, circular cross-sectional, sudden contraction nozzle sketched in Fig. P5.41 at section (1) with a uniformly distributed velocity of 25 ft/s and a pressure of 75 psi. The water exits from the nozzle into the atmosphere at section (2) where the uniformly distributed velocity is 100 ft/s. Determine the axial component of the anchoring force required to hold the contraction in place.



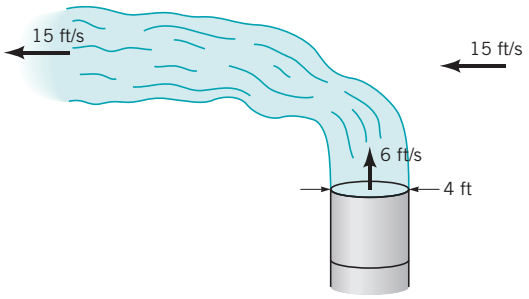

■ FIGURE P5.41

**5.42** The four devices shown in Fig. P5.42 rest on frictionless wheels, are restricted to move in the $x$ direction only, and are initially held stationary. The pressure at the inlets and outlets of each is atmospheric, and the flow is incompressible. The contents of each device is not known. When released, which devices will move to the right and which to the left? Explain.



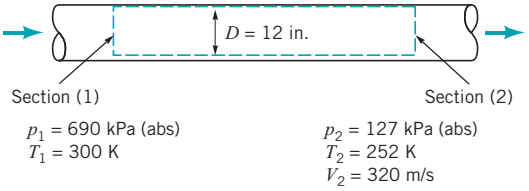

■ FIGURE P5.42

**5.43** Exhaust (assumed to have the properties of standard air) leaves the 4-ft-diameter chimney shown in **Video V5.4** and Fig. P5.43 with a speed of 6 ft/s. Because of the wind, after a few diameters downstream the exhaust flows in a horizontal direction with the speed of the wind, 15 ft/s. Determine the horizontal component of the force that the blowing wind puts on the exhaust gases.



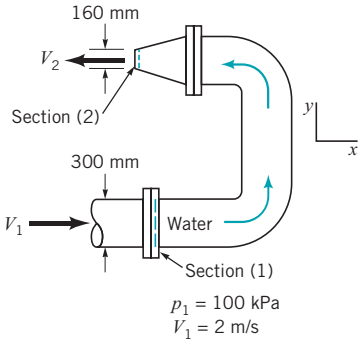

■ FIGURE P5.43

**5.44** Air flows steadily between two cross sections in a long, straight section of 12-in.-inside diameter pipe. The static temperature and pressure at each section are indicated in Fig P5.44. If the average air velocity at section (2) is 320 m/s, determine the average air velocity at section (1). Determine the frictional force exerted by the pipe wall on the air flowing between sections (1) and (2). Assume uniform velocity distributions at each section.



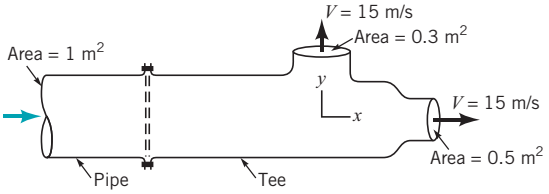

■ FIGURE P5.44

**5.45** Determine the magnitude and direction of the anchoring force needed to hold the horizontal elbow and nozzle combination shown in Fig. P5.45 in place. Atmospheric pressure is 100 kPa(abs). The gage pressure at section (1) is 100 kPa. At section (2), the water exits to the atmosphere.




■ FIGURE P5.45

**5.46** Water flows as two free jets from the tee attached to the pipe shown in Fig. P5.46. The exit speed is 15 m/s. If viscous effects and gravity are negligible, determine the $x$ and $y$ components of the force that the pipe exerts on the tee.




■ FIGURE P5.46

**5.47** A converging elbow (see Fig. P5.47) turns water through an angle of 135° in a vertical plane. The flow cross section diameter is 400 mm at the elbow inlet, section (1), and 200 mm at the elbow outlet, section (2). The elbow flow passage volume is 0.2 $m^3$ between sections (1) and (2). The water volume flowrate is 0.4 $m^3/s$ and the elbow inlet and outlet pressures are 150 kPa and 90 kPa. The elbow mass is 12 kg. Calculate the horizontal ($x$ direction) and vertical ($z$ direction) anchoring forces required to hold the elbow in place.

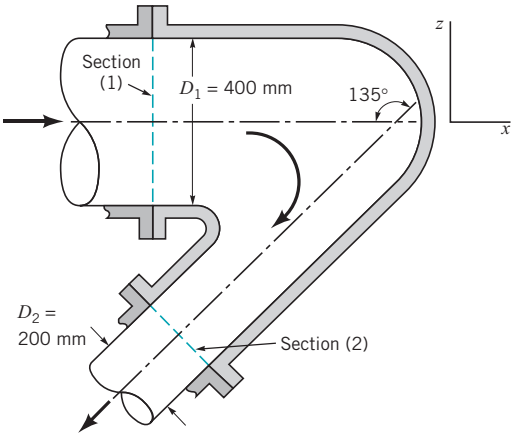



**FIGURE P5.47**

**5.48** The hydraulic dredge shown in Fig. P5.48 is used to dredge sand from a river bottom. Estimate the thrust needed from the propeller to hold the boat stationary. Assume the specific gravity of the sand/water mixture is $SG = 1.2$.

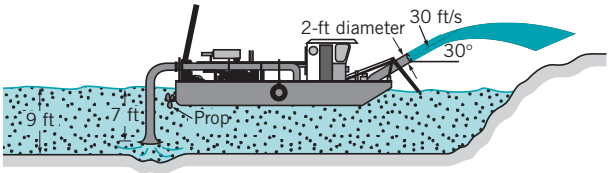



**FIGURE P5.48**

**5.49** A static thrust stand is to be designed for testing a specific jet engine. Knowing the following conditions for a typical test,

intake air velocity = 700 ft/s
exhaust gas velocity = 1640 ft/s
intake cross section area = 10 $ft^2$
intake static pressure = 11.4 psia
intake static temperature = 480 °R
exhaust gas pressure = 0 psi

estimate a nominal thrust to design for.

**5.50** A horizontal, circular cross-sectional jet of air having a diameter of 6 in. strikes a conical deflector as shown in Fig. P5.50. A horizontal anchoring force of 5 lb is required to hold the cone in place. Estimate the nozzle flowrate in $ft^3/s$. The magnitude of the velocity of the air remains constant.

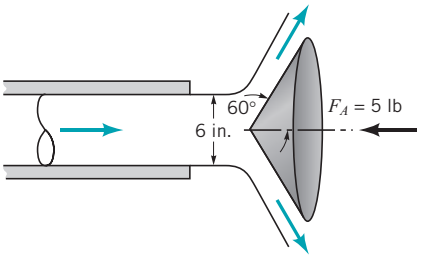



**FIGURE P5.50**

**5.51** A vertical, circular cross-sectional jet of air strikes a conical deflector as indicated in Fig. P5.51. A vertical anchoring force of 0.1 N is required to hold the deflector in place. Determine the mass (kg) of the deflector. The magnitude of velocity of the air remains constant.

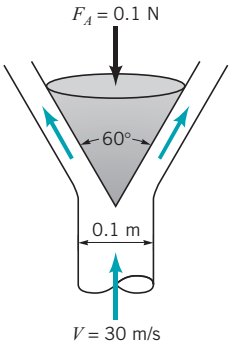



**FIGURE P5.51**

**5.52** Water flows from a large tank into a dish as shown in Fig. P5.52. **(a)** If at the instant shown the tank and the water in it weigh $W_1$ lb, what is the tension, $T_1$, in the cable supporting the tank? **(b)** If at the instant shown the dish and the water in it weigh $W_2$ lb, what is the force, $F_2$, needed to support the dish?

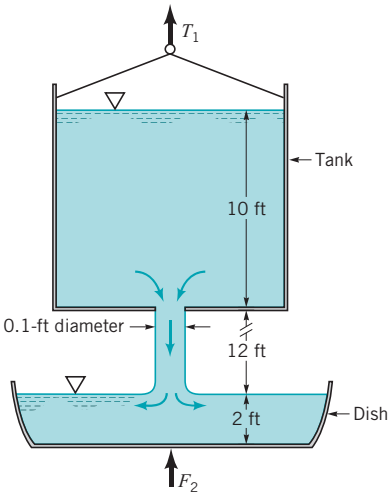



**FIGURE P5.52**

**5.53** Two water jets of equal size and speed strike each other as shown in Fig. P5.53. Determine the speed, $V$, and direction, $\theta$, of the resulting combined jet. Gravity is negligible.

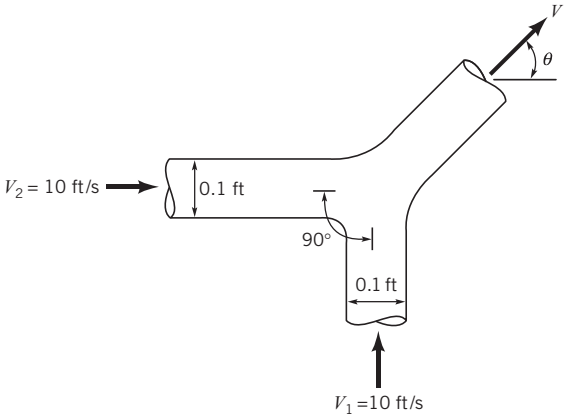
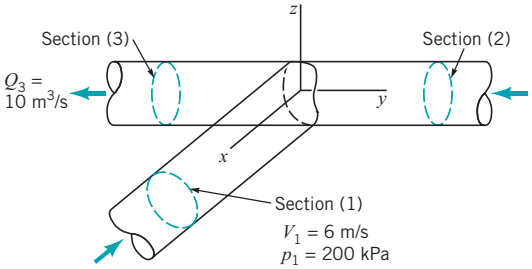



**FIGURE P5.53**

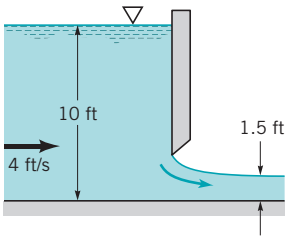
CMS Cepcor EX 1046 Page 161
CMS Cepcor v. Sandvik IPR2024-01380

**5.54** Assuming frictionless, incompressible, one-dimensional flow of water through the horizontal tee connection sketched in Fig. P5.54, estimate values of the $x$ and $y$ components of the force exerted by the tee on the water. Each pipe has an inside diameter of 1 m.



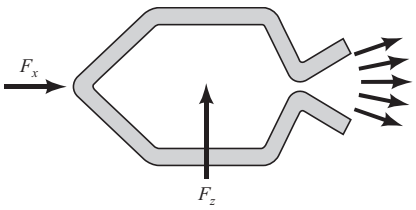

■ FIGURE P5.54

**5.55** Determine the magnitude of the horizontal component of the anchoring force required to hold in place the sluice gate shown in Fig. 5.55. Compare this result with the size of the horizontal component of the anchoring force required to hold in place the sluice gate when it is closed and the depth of water upstream is 10 ft.



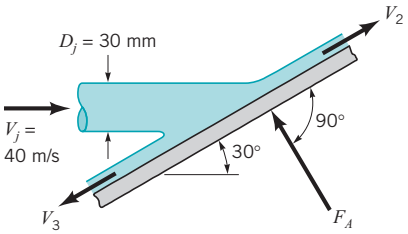

■ FIGURE P5.55

**5.56** The rocket shown in Fig. P5.56. is held stationary by the horizontal force, $F_x$, and the vertical force, $F_z$. The velocity and pressure of the exhaust gas are 5000 ft/s and 20 psia at the nozzle exit, which has a cross section area of 60 in.$^2$. The exhaust mass flowrate is constant at 21 lbm/s. Determine the value of the restraining force $F_x$. Assume the exhaust flow is essentially horizontal.



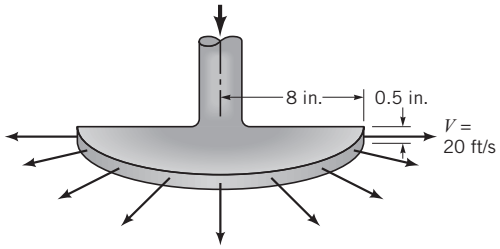

■ FIGURE P5.56

**5.57** A horizontal circular jet of air strikes a stationary flat plate as indicated in Fig. 5.57. The jet velocity is 40 m/s and the jet diameter is 30 mm. If the air velocity magnitude remains constant as the air flows over the plate surface in the directions shown, determine: **(a)** the magnitude of $F_A$, the anchoring force required to hold the plate stationary; **(b)** the fraction of mass flow along the plate surface in each of the two directions shown; **(c)** the magnitude of $F_A$, the anchoring force required to allow the plate to move to the right at a constant speed of 10 m/s.



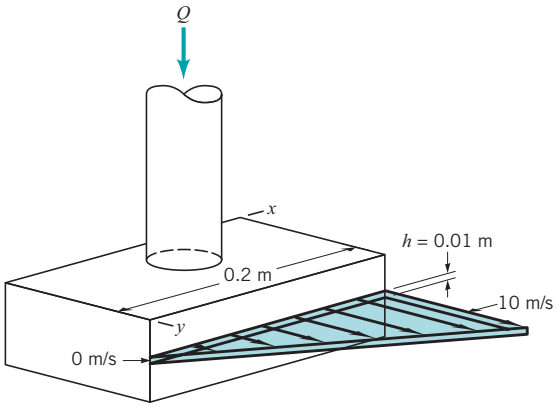

■ FIGURE P5.57

**5.58** Water is sprayed radially outward over 180° as indicated in Fig. P5.58. The jet sheet is in the horizontal plane. If the jet velocity at the nozzle exit is 20 ft/s, determine the direction and magnitude of the resultant horizontal anchoring force required to hold the nozzle in place.



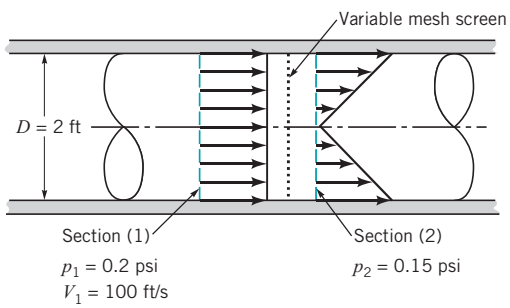

■ FIGURE P5.58

**5.59** A sheet of water of uniform thickness ($h$ = 0.01 m) flows from the device shown in Fig. P5.59. The water enters vertically through the inlet pipe and exits horizontally with a speed that varies linearly from 0 to 10 m/s along the 0.2-m length of the slit. Determine the $y$ component of anchoring force necessary to hold this device stationary.




■ FIGURE P5.59

**5.60** A variable mesh screen produces a linear and axisymmetric velocity profile as indicated in Fig. P5.60 in the air flow through a




■ FIGURE P5.60

2-ft-diameter circular cross section duct. The static pressures upstream and downstream of the screen are 0.2 and 0.15 psi and are uniformly distributed over the flow cross section area. Neglecting the force exerted by the duct wall on the flowing air, calculate the screen drag force.

**5.61** Water flows vertically upward in a circular cross-sectional pipe as shown in Fig. P5.61. At section (1), the velocity profile over the cross-sectional area is uniform. At section (2), the velocity profile is

$$\mathbf{V} = w_c\left(\frac{R-r}{R}\right)^{1/7}\hat{\mathbf{k}}$$

where $\mathbf{V}$ = local velocity vector, $w_c$ = centerline velocity in the axial direction, $R$ = pipe radius, and $r$ = radius from pipe axis. Develop an expression for the fluid pressure drop that occurs between sections (1) and (2).

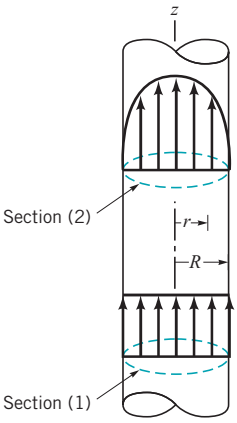



**FIGURE P5.61**

**5.62** In a laminar pipe flow that is fully developed, the axial velocity profile is parabolic. That is,

$$u = u_c\left[1-\left(\frac{r}{R}\right)^2\right]$$

as is illustrated in Fig. P5.62. Compare the axial direction momentum flowrate calculated with the average velocity, $\bar{u}$, with the axial direction momentum flowrate calculated with the nonuniform velocity distribution taken into account.

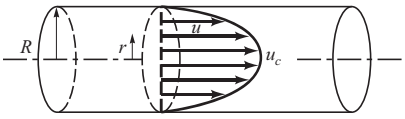



**FIGURE P5.62**

**†5.63** Water from a garden hose is sprayed against your car to rinse dirt from it. Estimate the force that the water exerts on the car. List all assumptions and show calculations.

**5.64** A Pelton wheel vane directs a horizontal, circular cross-sectional jet of water symmetrically as indicated in Fig. P5.64 and **Video V5.6**. The jet leaves the nozzle with a velocity of 100 ft/s. Determine the $x$ direction component of anchoring force required to **(a)** hold the vane stationary, **(b)** confine the speed of the vane to a value of 10 ft/s to the right. The fluid speed magnitude remains constant along the vane surface.

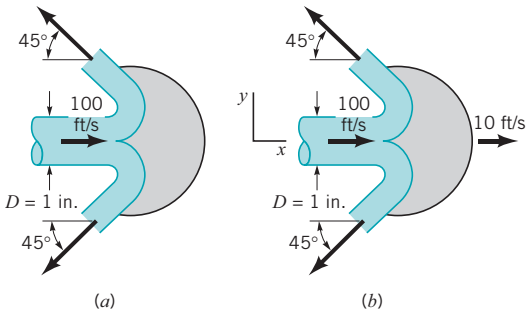



**FIGURE P5.64**

**5.65** How much power is transferred to the moving vane of Problem 5.64?

**5.66** The thrust developed to propel the jet ski shown in **Video V9.11** and Fig. P5.66 is a result of water pumped through the vehicle and exiting as a high-speed water jet. For the conditions shown in the figure, what flowrate is needed to produce a 300-lb thrust? Assume the inlet and outlet jets of water are free jets.

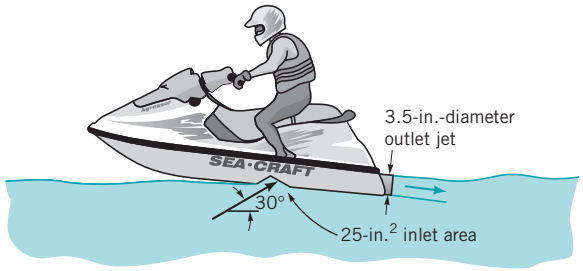



**FIGURE P5.66**

**5.67** (See Fluids in the News article titled **"Where the plume goes,"** Section 5.2.2.) Air flows into the jet engine shown in Fig. P5.67 at a rate of 9 slugs/s and a speed of 300 ft/s. Upon landing, the engine exhaust exits through the reverse thrust mechanism with a speed of 900 ft/s in the direction indicated. Determine the reverse thrust applied by the engine to the airplane. Assume the inlet and exit pressures are atmospheric and that the mass flowrate of fuel is negligible compared to the air flowrate through the engine.

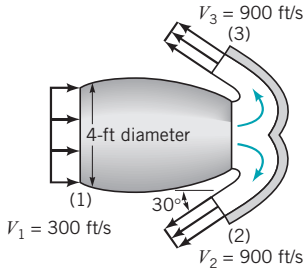
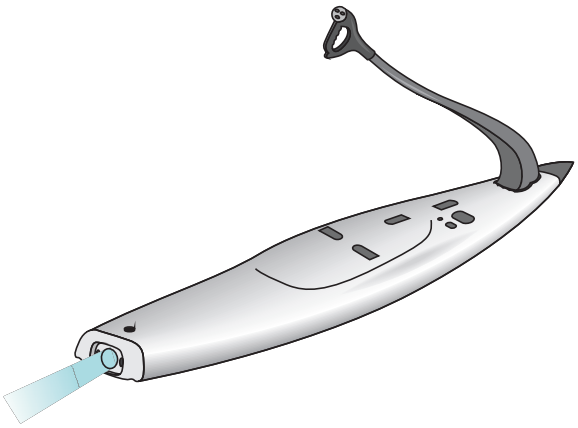



**FIGURE P5.67**

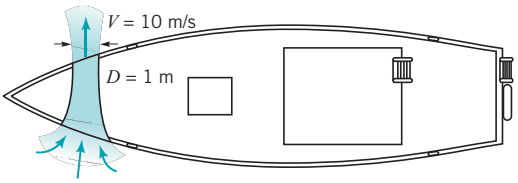
**5.68** (See Fluids in the News article titled **"Motorized surfboard,"** Section 5.2.2.) The thrust to propel the powered surfboard shown in Fig. P5.68 is a result of water pumped through the board that exits as a high-speed 2.75-in.-diameter jet. Determine the flowrate and the velocity of the exiting jet if the thrust is to be 300 lb. Neglect the momentum of the water entering the pump.

CMS Cepcor EX 1046 Page 163
CMS Cepcor v. Sandvik IPR2024-01380



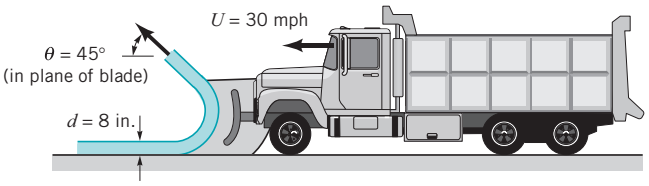
■ FIGURE P5.68

**5.69** (See Fluids in the News article titled **"Bow thrusters,"** Section 5.2.2). The bow thruster on the boat shown in Fig. P5.69 is used to turn the boat. The thruster produces a 1-m-diameter jet of water with a velocity of 10 m/s. Determine the force produced by the thruster. Assume that the inlet and outlet pressures are zero and that the momentum of the water entering the thruster is negligible.




■ FIGURE P5.69

**5.70** A snowplow mounted on a truck clears a path 12 *ft* through heavy wet snow, as shown in Figure P5.70. The snow is 8 in. deep and its density is 10 lbm/ft$^3$. The truck travels at 30 mph. The snow is discharged from the plow at an angle of 45° from the direction of travel and 45° above the horizontal, as shown in Figure P5.70. Estimate the force required to push the plow.




■ FIGURE P5.70

## Section 5.2.3 Derivation of the Moment-of-Momentum Equation

**5.71** What is fluid moment-of-momentum (angular momentum) and the "flow" of moment-of-momentum (angular momentum)?

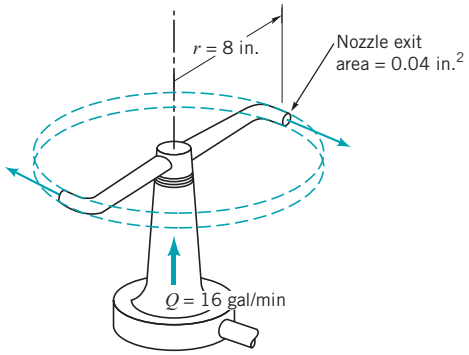
**5.72** Describe the orthogonal components of the moment-of-momentum equation (Eq. 5.42) and comment on the direction of each.

**5.73** Describe a few examples (include photographs/images) of turbines where the force/torque of a flowing fluid leads to rotation of a shaft.

**5.74** Describe a few examples (include photographs/images) of pumps where a fluid is forced to move by "blades" mounted on a rotating shaft.

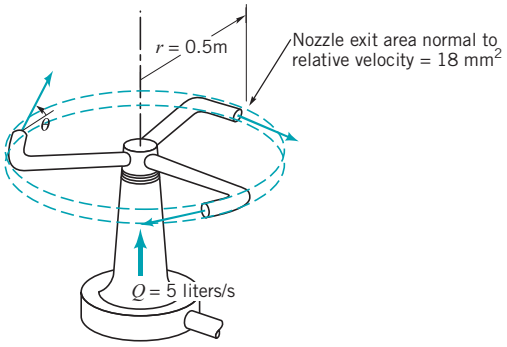
## Section 5.2.4 Application of the Moment-of-Momentum Equation

**5.75** Water enters a rotating lawn sprinkler through its base at the steady rate of 16 gal/min as shown in Fig. P5.75. The exit cross-sectional area of each of the two nozzles is 0.04 in.$^2$, and the flow leaving each nozzle is tangential. The radius from the axis of rotation to the centerline of each nozzle is 8 in. **(a)** Determine the resisting torque required to hold the sprinkler head stationary. **(b)** Determine the resisting torque associated with the sprinkler rotating with a constant speed of 500 rev/min. **(c)** Determine the angular velocity of the sprinkler if no resisting torque is applied.




■ FIGURE P5.75

**5.76** Five liters/s of water enter the rotor shown in **Video V5.10** and Fig. P5.76 along the axis of rotation. The cross-sectional area of each of the three nozzle exits normal to the relative velocity is 18 mm$^2$. How large is the resisting torque required to hold the rotor stationary? How fast will the rotor spin steadily if the resisting torque is reduced to zero and **(a)** $\theta = 0°$, **(b)** $\theta = 30°$, **(c)** $\theta = 60°$?




■ FIGURE P5.76

**5.77** Shown in Fig. P5.77 is a toy "helicopter" powered by air escaping from a balloon. The air from the balloon flows radially through each of the three propeller blades and out through small nozzles at the tips of the blades. Explain physically how this flow can cause the rotation necessary to rotate the blades to produce the needed lifting force.

**5.78** A simplified sketch of a hydraulic turbine runner is shown in Fig. P5.78. Relative to the rotating runner, water enters at section (1) (cylindrical cross section area $A_1$ at $r_1$=1.5 m) at an angle of 100° from the tangential direction and leaves at section (2) (cylindrical cross section area $A_2$ at $r_2$=0.85 m ) at an angle of 50° from the tangential direction. The blade height at sections (1) and (2) is 0.45 m and the volume flowrate through the turbine is 30 m$^3$/s. The runner speed is 130 rpm in the direction shown. Determine the shaft power developed.

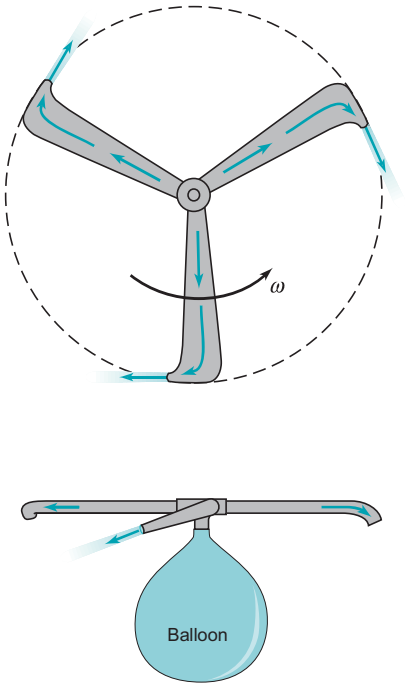



FIGURE P5.77

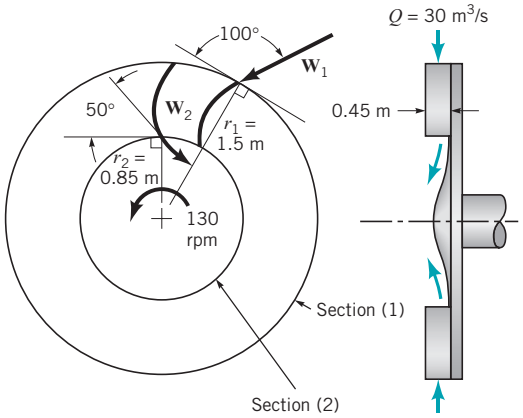



FIGURE P5.78

**5.79** A water turbine with radial flow has the dimensions shown in Fig.P5.79.The absolute entering velocity is 50 ft/s, and it makes an angle of 30° with the tangent to the rotor. The absolute exit velocity is directed radially inward. The angular speed of the rotor is 120 rpm. Find the power delivered to the shaft of the turbine.

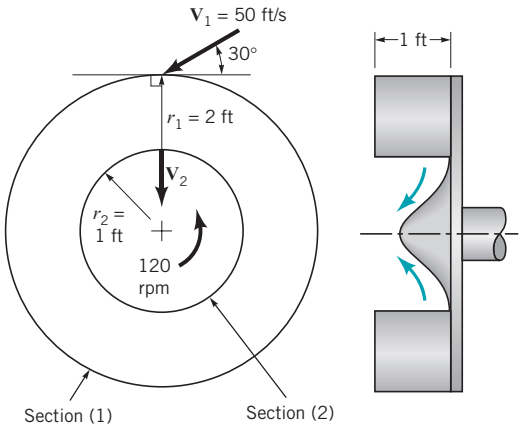



FIGURE P5.79

**5.80** Shown in Fig. P5.80 are front and side views of a centrifugal pump rotor or impeller. If the pump delivers 200 liters/s of water and the blade exit angle is 35° from the tangential direction, determine the power requirement associated with flow leaving at the blade angle. The flow entering the rotor blade row is essentially radial as viewed from a stationary frame.

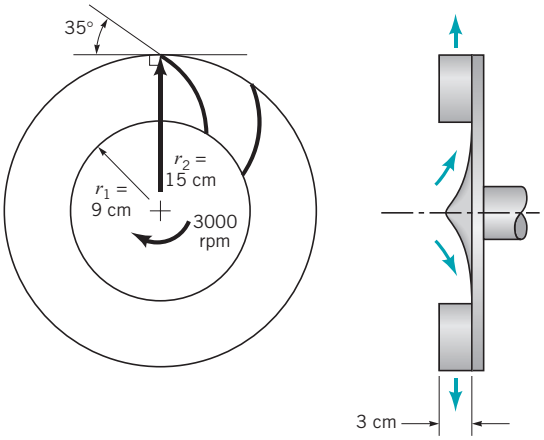



FIGURE P5.80

**5.81** The velocity triangles for water flow through a radial pump rotor are as indicated in Fig. P5.81. **(a)** Determine the energy added to each unit mass (kg) of water as it flows through the rotor. **(b)** Sketch an appropriate blade section.

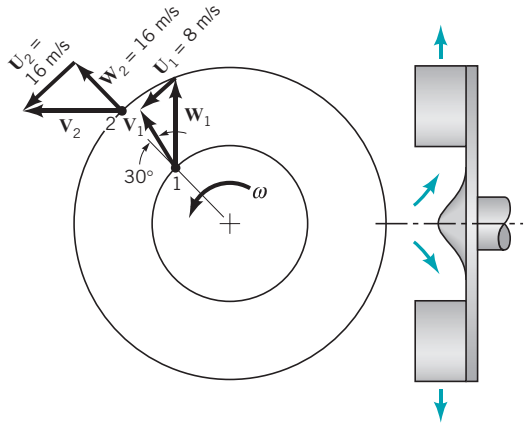



FIGURE P5.81

**5.82** An axial flow turbomachine rotor involves the upstream (1) and downstream (2) velocity triangles shown in Fig.P5.82. Is this turbomachine a turbine or a fan? Sketch an appropriate blade section and determine energy transferred per unit mass of fluid.

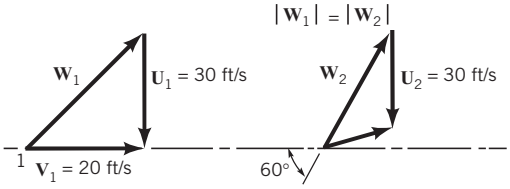
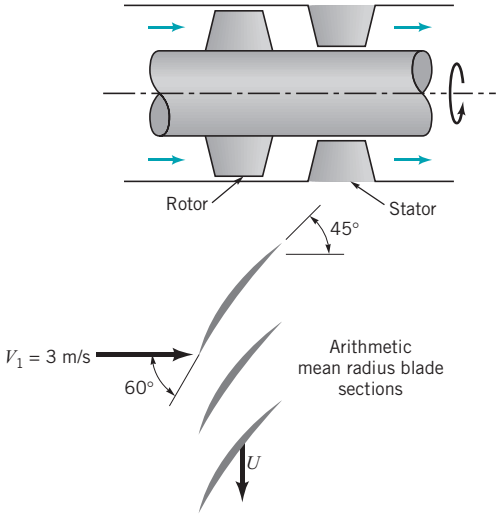


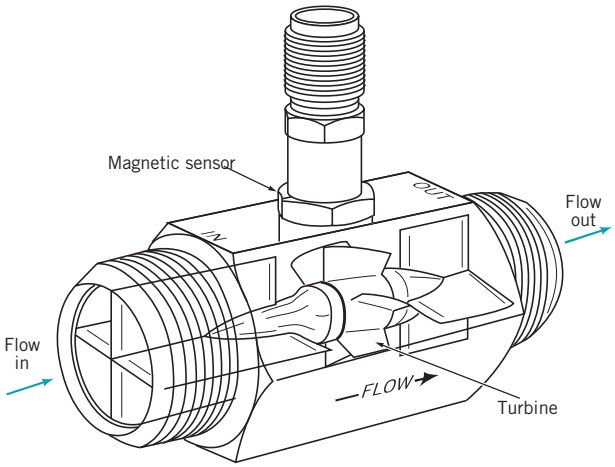

FIGURE P5.82

**5.83** An axial flow gasoline pump (see Fig. P5.83) consists of a rotating row of blades (rotor) followed downstream by a stationary row of blades (stator). The gasoline enters the rotor axially (without any angular momentum) with an absolute velocity of 3 m/s. The rotor blade inlet and exit angles are 60° and 45° from the axial direction. The pump annulus passage cross-sectional area is constant. Consider the flow as being tangent to the blades involved. Sketch velocity triangles for flow just upstream and downstream of the rotor and just downstream of the stator where the flow is axial. How much energy is added to each kilogram of gasoline? Is this an actual or ideal amount?




■ FIGURE P5.83

**5.84** Sketch the velocity triangles for the flows entering and leaving the rotor of the turbine-type flow meter shown in Fig. P5.84. Show how rotor angular velocity is proportional to average fluid velocity.




■ FIGURE P5.84 **(Courtesy of EG&G Flow Technology, Inc.)**

**5.85** By using velocity triangles for flow upstream (1) and downstream (2) of a turbomachine rotor, prove that the shaft work in per unit mass flowing through the rotor is

$$w_{\substack{\text{shaft}\\ \text{net in}}} = \frac{V_2^2 - V_1^2 + U_2^2 - U_1^2 + W_1^2 - W_2^2}{2}$$

where $V$ = absolute flow velocity magnitude, $W$ = relative flow velocity magnitude, and $U$ = blade speed.

## Section 5.3.1 Derivation of the Energy Equation

**5.86** Distiguish between shaft work and other kinds of work associated with a flowing fluid.

**5.87** Define briefly what heat transfer is. What is an adiabatic flow? Give several practical examples of nearly adiabatic flows.

## Section 5.3.2 Application of the Energy Equation – No Shaft Work and Section 5.3.3 Comparison of the Energy Equation with the Bernoulli Equation

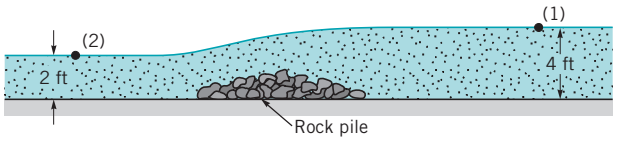
**5.88** What is enthalpy and why is it useful for energy considerations in fluid mechanics?

**5.89** Cite a few examples of evidence of loss of available energy in actual fluid flows. Why does loss occur?

**5.90** Is zero heat transfer a necessary condition for application of the Bernoulli equation (Eq. 5.75)?

**5.91** A 1000-m-high waterfall involves steady flow from one large body to another. Detemine the temperature rise associated with this flow.

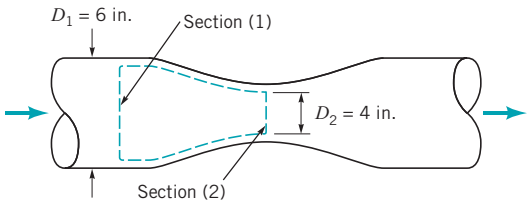
**5.92** A 100-ft-wide river with a flowrate of 2400 $ft^3/s$ flows over a rock pile as shown in Fig. P5.92. Determine the direction of flow and the head loss associated with the flow across the rock pile.




■ FIGURE P5.92

**5.93** Air steadily expands adiabatically and without friction from stagnation conditions of 690 kpa (abs) and 290 K to a static pressure of 101 kpa (abs). Determine the velocity of the expanded air assuming: **(a)** incompressible flow; **(b)** compressible flow.

**5.94** A horizontal Venturi flow meter consists of a converging–diverging conduit as indicated in Fig. P5.94. The diameters of cross sections (1) and (2) are 6 and 4 in. The velocity and static pressure are uniformly distributed at cross sections (1) and (2). Determine the volume flowrate ($ft^3/s$) through the meter if $p_1 - p_2 = 3$ psi, the flowing fluid is oil ($\rho = 56$ lbm/$ft^3$), and the loss per unit mass from (1) to (2) is negligibly small.




■ FIGURE P5.94

**5.95** Oil ($SG = 0.9$) flows downward through a vertical pipe contraction as shown in Fig. P5.95. If the mercury manometer reading, $h$, is 100 mm, determine the volume flowrate for frictionless flow. Is the actual flowrate more or less than the frictionless value? Explain.

**5.96** An incompressible liquid flows steadily along the pipe shown in Fig. P5.96. Determine the direction of flow and the head loss over the 6-m length of pipe.

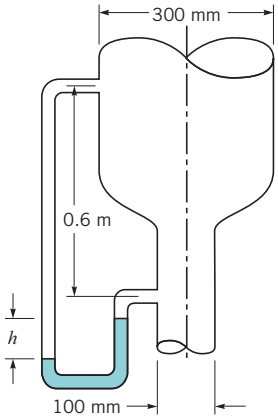



FIGURE P5.95

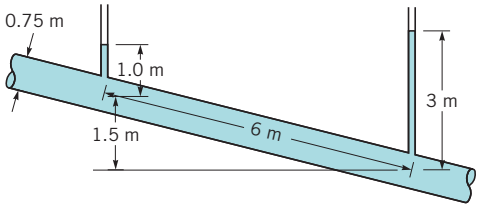



FIGURE P5.96

**5.97** Water flows through a vertical pipe, as is indicated in Fig. P5.97. Is the flow up or down in the pipe? Explain.

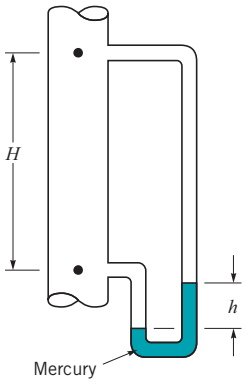



FIGURE P5.97

**5.98** A circular disk can be lifted up by blowing on it with the device shown in Fig. P5.98. Explain why this happens.

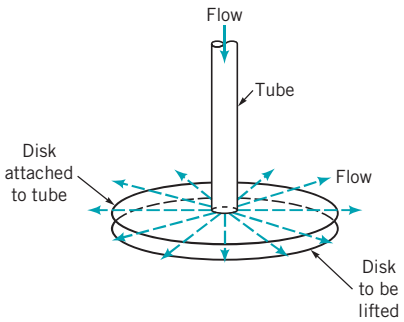



FIGURE P5.98

**5.99** A siphon is used to draw water at 20°C from a large container as indicated in Fig. P5.99. Does changing the elevation, $h$, of the siphon centerline above the water level in the tank vary the flowrate through the siphon? Explain. What is the maximum allowable value of $h$?

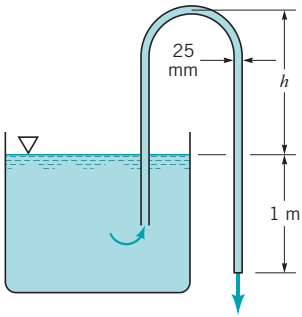



FIGURE P5.99

**5.100** A water siphon having a constant inside diameter of 3 in. is arranged as shown in Fig. P5.100. If the friction loss between $A$ and $B$ is $0.8V^2/2$, where $V$ is the velocity of flow in the siphon, determine the flowrate involved.

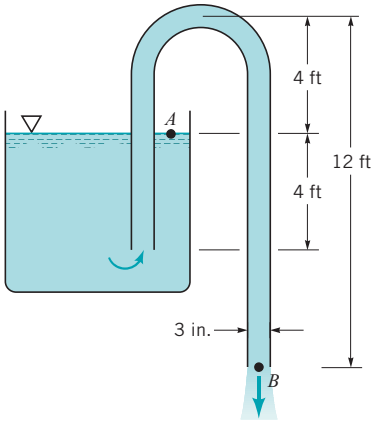



FIGURE P5.100

**5.101** Water flows through a valve (see Fig.P5.101) at the rate of 1000 lbm/s. The pressure just upstream of the valve is 90 psi and the pressure drop across the valve is 50 psi. The inside diameters of the valve inlet and exit pipes are 12 and 24 in. If the flow through the valve occurs in a horizontal plane determine the loss in available energy across the valve.

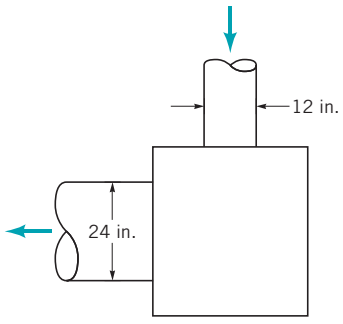
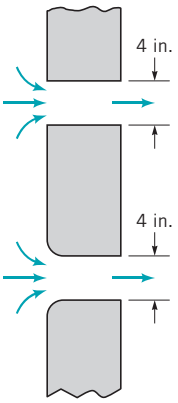



FIGURE P5.101

**5.102** Compare the volume flowrates associated with two different vent configurations, a cylindrical hole in the wall having a diameter of 4 in. and the same diameter cylindrical hole in the wall

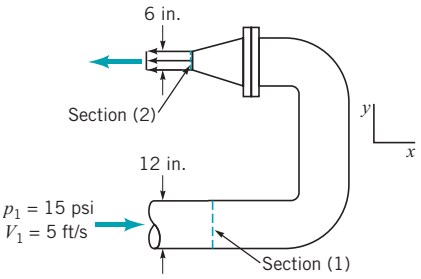
but with a well-rounded entrance (see Fig. P5.102). The room is held at a constant pressure of 1.5 psi above atmospheric. Both vents exhaust into the atmosphere. The loss in available energy associated with flow through the cylindrical vent from the room to the vent exit is $0.5V_2^2/2$, where $V_2$ is the uniformly distributed exit velocity of air. The loss in available energy associated with flow through the rounded entrance vent from the room to the vent exit is $0.05V_2^2/2$, where $V_2$ is the uniformly distributed exit velocity of air.




■ FIGURE P5.102

**5.103** A gas expands through a nozzle from a pressure of 300 psia to a pressure of 5 psia. The enthalpy change involved, $\check{h}_1 - \check{h}_2$, is 150 Btu/lbm. If the expansion is adiabatic but with frictional effects and the inlet gas speed is negligibly small, determine the exit gas velocity.

**5.104** For the 180° elbow and nozzle flow shown in Fig. P5.104, determine the loss in available energy from section (1) to section (2). How much additional available energy is lost from section (2) to where the water comes to rest?

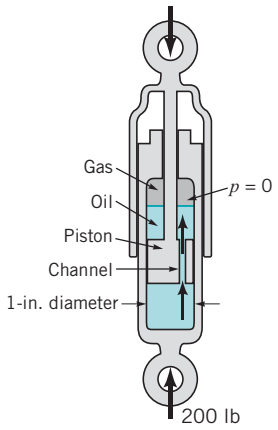



■ FIGURE P5.104

**5.105** An automobile engine will work best when the back pressure at the interface of the exhaust manifold and the engine block is minimized. Show how reduction of losses in the exhaust manifold, piping, and muffler will also reduce the back pressure. How could losses in the exhaust system be reduced? What primarily limits the minimization of exhaust system losses?

**†5.106** Explain how, in terms of the loss of available energy involved, a home sink water faucet valve works to vary the flow from the shutoff condition to maximum flow. Explain how you would estimate the size of the overflow drain holes needed in the sink of **Video V5.1** (**Video V3.9** may be helpful).

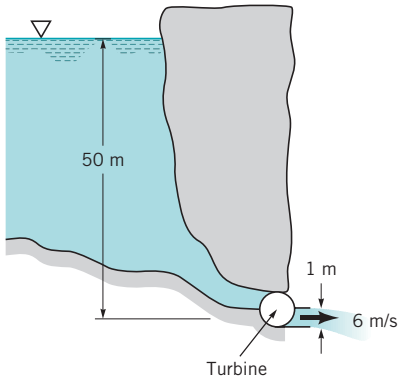
**5.107** (See Fluids in the News article titled **"Smart shocks,"** Section 5.3.3.) A 200-lb force applied to the end of the piston of the shock absorber shown in Fig. P5.107 causes the two ends of the shock absorber to move toward each other with a speed of 5 ft/s. Determine the head loss associated with the flow of the oil through the channel. Neglect gravity and any friction force between the piston and cylinder walls.




■ FIGURE P5.107

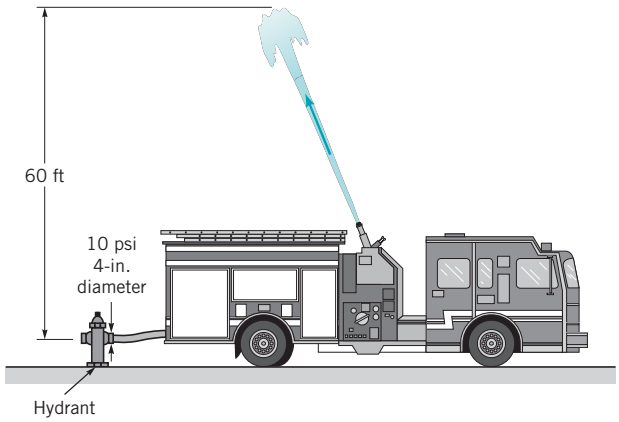
## Section 5.3.2 Application of the Energy Equation–With Shaft Work

**5.108** What is the maximum possible power output of the hydroelectric turbine shown in Fig.P5.108?




■ FIGURE P5.108

**5.109** The pumper truck shown in Fig. P5.109 is to deliver 1.5 $ft^3/s$ to a maximum elevation of 60 ft above the hydrant. The pressure at the 4-in.-diameter outlet of the hydrant is 10 psi. If head losses are negligibly small, determine the power that the pump must add to the water.




■ FIGURE P5.109

**5.110** The hydroelectric turbine shown in Fig. P5.110 passes 8 million gal/min across a head of 600 ft. What is the maximum amount of power output possible? Why will the actual amount be less?

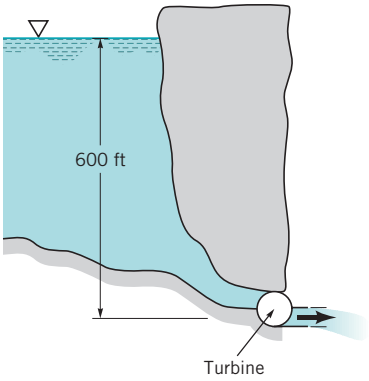



**FIGURE P5.110**

**5.111** A pump is to move water from a lake into a large, pressurized tank as shown in Fig. P5.111 at a rate of 1000 gal in 10 min or less. Will a pump that adds 3 hp to the water work for this purpose? Support your answer with appropriate calculations. Repeat the problem if the tank were pressurized to 3, rather than 2, atmospheres.

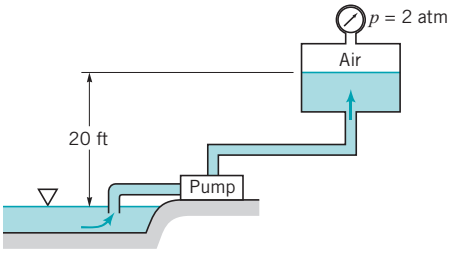



**FIGURE P5.111**

**5.112** A hydraulic turbine is provided with 4.25 $m^3/s$ of water at 415 kPa. A vacuum gage in the turbine discharge 3 m below the turbine inlet centerline reads 250 mm Hg vacuum. If the turbine shaft output power is 1100 kW, calculate the power loss through the turbine. The supply and discharge pipe inside diameters are identically 80 mm.

**5.113** Water is supplied at 150 $ft^3/s$ and 60 psi to a hydraulic turbine through a 3-ft inside diameter inlet pipe as indicated in Fig. P5.113. The turbine discharge pipe has a 4-ft inside diameter. The static pressure at section (2), 10 ft below the turbine inlet, is 10-in. Hg vacuum. If the turbine develops 2500 hp, determine the power lost between sections (1) and (2).

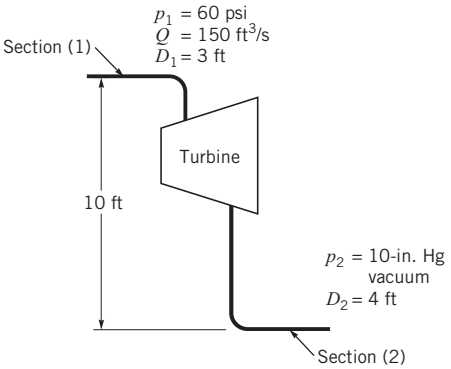



**FIGURE P5.113**

**5.114** A centrifugal air compressor stage operates between an inlet stagnation pressure of 14.7 psia and an exit stagnation pressure of 60 psia. The inlet stagnation temperature is 80 °F. If the loss of total pressure through the compressor stage associated with irreversible flow phenomena is 10 psi, estimate the actual and ideal stagnation temperature rise through the compressor. Estimate the ratio of ideal to actual temperature rise to obtain an approximate value of the efficiency.

**5.115** Water is pumped through a 4-in.-diameter pipe as shown in Fig. P5.115*a*. The pump characteristics (pump head versus flowrate) are given in Fig. P5.115*b*. Determine the flowrate if the head loss in the pipe is $h_L = 8V^2/2g$.

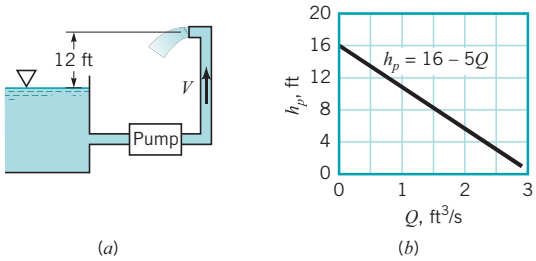



**FIGURE P5.115**

**5.116** Water is pumped from the large tank shown in Fig. P5.116. The head loss is known to be equal to $4V^2/2g$ and the pump head is $h_p = 20 - 4Q^2$, where $h_p$ is in ft when $Q$ is in $ft^3/s$. Determine the flowrate.

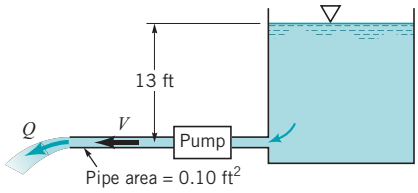



**FIGURE P5.116**

**5.117** When a fan or pump is tested at the factory, head curves (head across the fan or pump versus volume flowrate) are often produced. A generic fan or pump head curve is shown in Fig.P5.117*a*. For any piping system, the drop in pressure or head involved because of loss can be estimated as a function of volume flowrate. A generic piping system loss curve is shown in Fig.P5.117*b*. When the pump or fan and piping system associated with the two curves of Fig.P5.117 are combined, what will the flowrate be? Why? How can the flowrate through this combined system be varied?

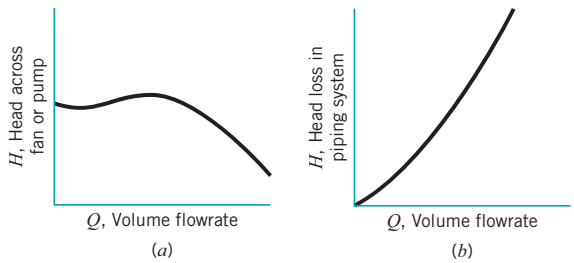
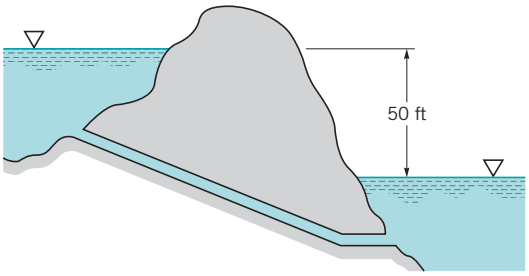


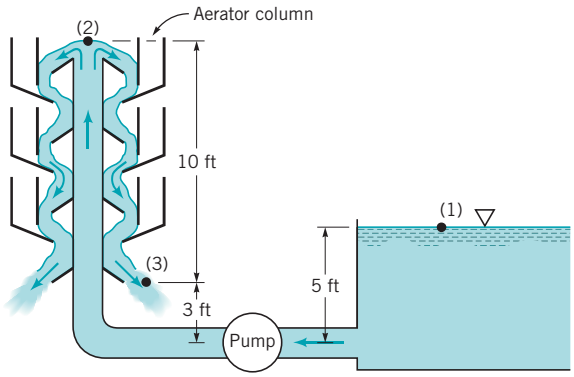

**FIGURE P5.117**

**5.118** Water flows by gravity from one lake to another as sketched in Fig. P5.118 at the steady rate of 80 gpm. What is the loss in available energy associated with this flow? If this same amount of loss is associated with pumping the fluid from the lower lake to the higher one at the same flowrate, estimate the amount of pumping power required.



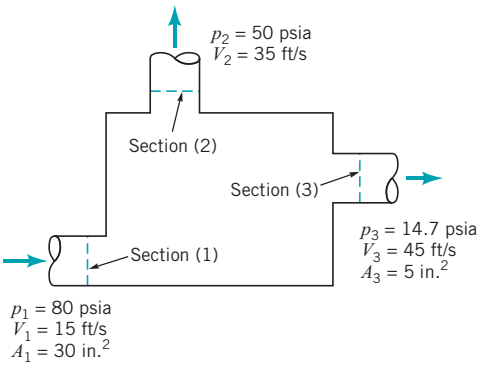

■ FIGURE P5.118

**5.119** Water is pumped from a tank, point (1), to the top of a water plant aerator, point (2), as shown in Video V5.14 and Fig. P5.119 at a rate of 3.0 ft$^3$/s. (a) Determine the power that the pump adds to the water if the head loss from (1) to (2) where $V_2 = 0$ is 4 ft. (b) Determine the head loss from (2) to the bottom of the aerator column, point (3), if the average velocity at (3) is $V_3 = 2$ ft/s.



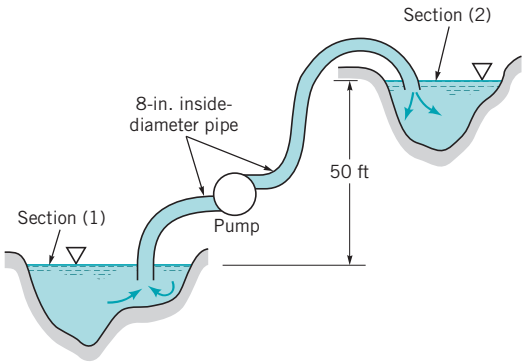

■ FIGURE P5.119

**5.120** A liquid enters a fluid machine at section (1) and leaves at sections (2) and (3) as shown in Fig. P5.120. The density of the fluid is constant at 2 slugs/ft$^3$. All of the flow occurs in a horizontal plane and is frictionless and adiabatic. For the above-mentioned and additional conditions indicated in Fig. P5.120, determine the amount of shaft power involved.



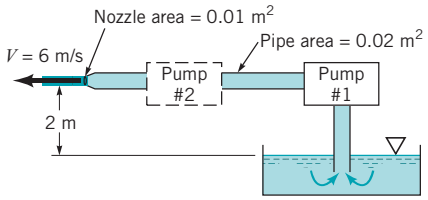

■ FIGURE P5.120

**5.121** Water is to be moved from one large reservoir to another at a higher elevation as indicated in Fig. P5.121. The loss of available energy associated with 2.5 ft$^3$/s being pumped from sections (1) to (2) is loss $= 61\overline{V}^2/2$ ft$^2$/s$^2$, where $\overline{V}$ is the average velocity of water in the 8-in. inside diameter piping involved. Determine the amount of shaft power required.



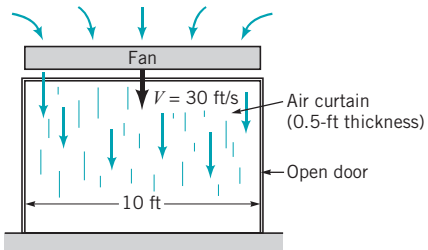

■ FIGURE P5.121

**5.122** Water is to be pumped from the large tank shown in Fig. P5.122 with an exit velocity of 6 m/s. It was determined that the original pump (pump 1) that supplies 1 kW of power to the water did not produce the desired velocity. Hence, it is proposed that an additional pump (pump 2) be installed as indicated to increase the flowrate to the desired value. How much power must pump 2 add to the water? The head loss for this flow is $h_L = 250Q^2$, where $h_L$ is in m when $Q$ is in m$^3$/s.




■ FIGURE P5.122

**5.123** (See Fluids in the News article titled **"Curtain of air,"** Section 5.3.3.) The fan shown in Fig. P5.123 produces an air curtain to separate a loading dock from a cold storage room. The air curtain is a jet of air 10 ft wide, 0.5 ft thick moving with speed $V = 30$ ft/s. The loss associated with this flow is loss $= K_L V^2/2$, where $K_L = 5$. How much power must the fan supply to the air to produce this flow?




■ FIGURE P5.123

## Section 5.3.2 Application of the Energy Equation—Combined with Linear momentum

**5.124** If a $\frac{3}{4}$-hp motor is required by a ventilating fan to produce a 24-in. stream of air having a velocity of 40 ft/s as shown in Fig. P5.124, estimate **(a)** the efficiency of the fan and **(b)** the thrust of the supporting member on the conduit enclosing the fan.

**5.125** Air flows past an object in a pipe of 2-m diameter and exits as a free jet as shown in Fig. P5.125. The velocity and pressure upstream are uniform at 10 m/s and 50 N/m$^2$, respectively. At the

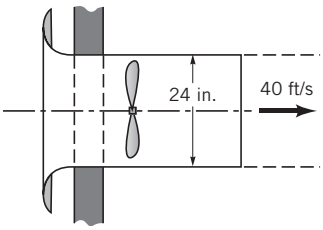



FIGURE P5.124

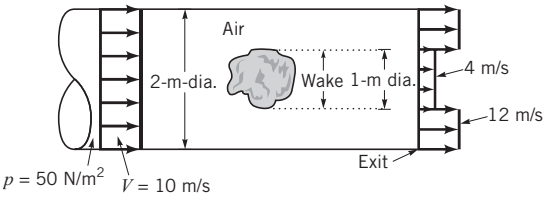



FIGURE P5.125

pipe exit the velocity is nonuniform as indicated. The shear stress along the pipe wall is negligible. **(a)** Determine the head loss associated with a particle as it flows from the uniform velocity upstream of the object to a location in the wake at the exit plane of the pipe. **(b)** Determine the force that the air puts on the object.

**5.126** Water flows through a 2-ft-diameter pipe arranged horizontally in a circular arc as shown in Fig. P5.126. If the pipe discharges to the atmosphere ($p = 14.7$ psia) determine the $x$ and $y$ components of the resultant force exerted by the water on the piping between sections (1) and (2). The steady flowrate is 3000 ft$^3$/min. The loss in pressure due to fluid friction between sections (1) and (2) is 60 psi.

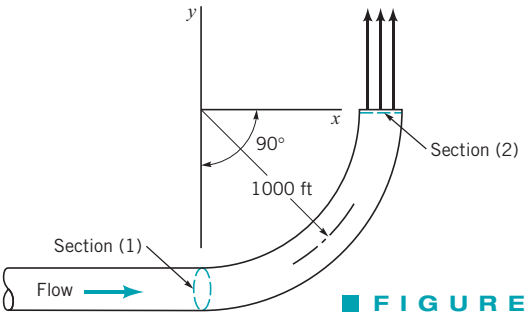



FIGURE P5.126

**5.127** Water flows steadily down the inclined pipe as indicated in Fig. P5.127. Determine the following: **(a)** the difference in pressure $p_1 - p_2$, **(b)** the loss between sections (1) and (2), **(c)** the net axial force exerted by the pipe wall on the flowing water between sections (1) and (2).

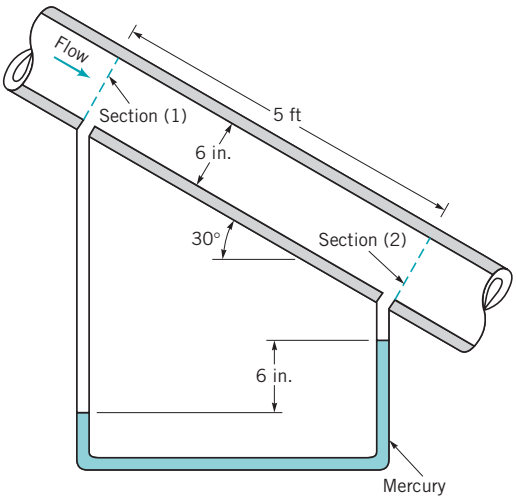



FIGURE P5.127

**5.128** Water flows steadily in a pipe and exits as a free jet through an end cap that contains a filter as shown in Fig. P5.128. The flow is in a horizontal plane. The axial component, $R_y$, of the anchoring force needed to keep the end cap stationary is 60 lb. Determine the head loss for the flow through the end cap.

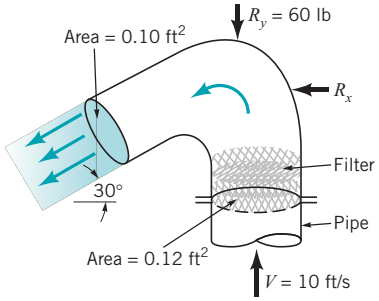



FIGURE P5.128

**5.129** When fluid flows through an abrupt expansion as indicated in Fig. P5.129, the loss in available energy across the expansion, $\text{loss}_{ex}$, is often expressed as

$$\text{loss}_{ex} = \left(1 - \frac{A_1}{A_2}\right)^2 \frac{V_1^2}{2}$$

where $A_1$ = cross-sectional area upstream of expansion, $A_2$ = cross-sectional area downstream of expansion, and $V_1$ = velocity of flow upstream of expansion. Derive this relationship.

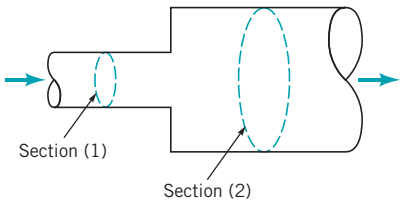



FIGURE P5.129

**5.130** Two water jets collide and form one homogeneous jet as shown in Fig. P5.130. **(a)** Determine the speed, $V$, and direction, $\theta$, of the combined jet. **(b)** Determine the loss for a fluid particle flowing from (1) to (3), from (2) to (3). Gravity is negligible.

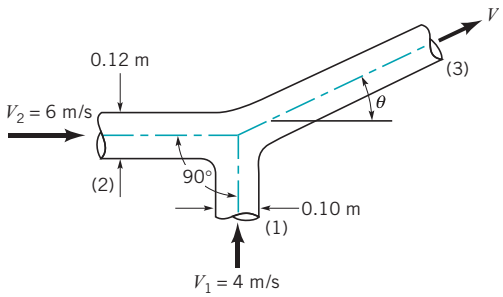



FIGURE P5.130

CMS Cepcor EX 1046 Page 171
CMS Cepcor v. Sandvik IPR2024-01380

### Section 5.3.4 Application of the Energy Equation to Nonuniform Flows

**5.131** Water flows vertically upward in a circular cross-sectional pipe. At section (1), the velocity profile over the cross-sectional area is uniform. At section (2), the velocity profile is

$$\mathbf{V} = w_c\left(\frac{R-r}{R}\right)^{1/7}\hat{\mathbf{k}}$$

where $\mathbf{V}$ = local velocity vector, $w_c$ = centerline velocity in the axial direction, $R$ = pipe inside radius, and, $r$ = radius from pipe axis. Develop an expression for the loss in available energy between sections (1) and (2).

**5.132** The velocity profile in a turbulent pipe flow may be approximated with the expression

$$\frac{u}{u_c} = \left(\frac{R-r}{R}\right)^{1/n}$$

where $u$ = local velocity in the axial direction, $u_c$ = centerline velocity in the axial direction, $R$ = pipe inner radius from pipe axis, $r$ = local radius from pipe axis, and $n$ = constant. Determine the kinetic energy coefficient, $\alpha$, for **(a)** $n = 5$, **(b)** $n = 6$, **(c)** $n = 7$, **(d)** $n = 8$, **(e)** $n = 9$, **(f)** $n = 10$.

**5.133** A small fan moves air at a mass flowrate of 0.004 lbm/s. Upstream of the fan, the pipe diameter is 2.5 in., the flow is laminar, the velocity distribution is parabolic, and the kinetic energy coefficient, $\alpha_1$, is equal to 2.0. Downstream of the fan, the pipe diameter is 1 in., the flow is turbulent, the velocity profile is quite flat, and the kinetic energy coefficient, $\alpha_2$, is equal to 1.08. If the rise in static pressure across the fan is 0.015 psi and the fan shaft draws 0.00024 hp, compare the value of loss calculated: **(a)** assuming uniform velocity distributions, **(b)** considering actual velocity distributions.

### Section 5.3.5 Combination of the Energy Equation and the Moment-of-Momentum Equation

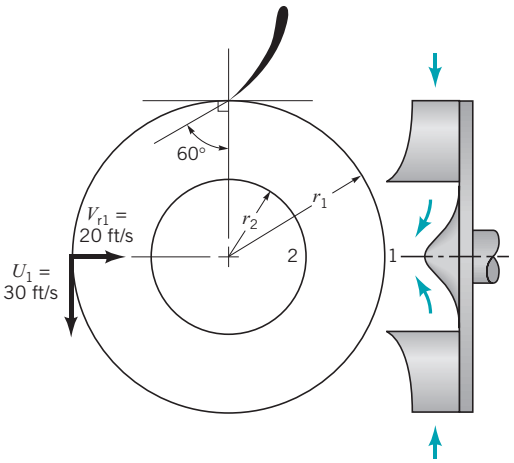
**5.134** Air enters a radial blower with zero angular momentum. It leaves with an absolute tangential velocity, $V_\theta$, of 200 ft/s. The rotor blade speed at rotor exit is 170 ft/s. If the stagnation pressure rise across the rotor is 0.4 psi, calculate the loss of available energy across the rotor and the rotor efficiency.

**5.135** Water enters a pump impeller radially. It leaves the impeller with a tangential component of absolute velocity of 10 m/s. The impeller exit diameter is 60 mm, and the impeller speed is 1800 rpm. If the stagnation pressure rise across the impeller is 45 kPa, determine the loss of available energy across the impeller and the hydraulic efficiency of the pump.

**5.136** Water enters an axial-flow turbine rotor with an absolute velocity tangential component, $V_\theta$, of 15 ft/s. The corresponding blade velocity, $U$, is 50 ft/s. The water leaves the rotor blade row with no angular momentum. If the stagnation pressure drop across the turbine is 12 psi, determine the hydraulic efficiency of the turbine.

**5.137** An inward flow radial turbine (see Fig. P5.137) involves a nozzle angle, $\alpha_1$, of 60° and an inlet rotor tip speed, $U_1$, of 30 ft/s. The ratio of rotor inlet to outlet diameters is 2.0. The radial component of velocity remains constant at 20 ft/s through the rotor, and the flow leaving the rotor at section (2) is without angular momentum. If the flowing fluid is water and the stagnation pressure drop across the rotor is 16 psi, determine the loss of available energy across the rotor and the hydraulic efficiency involved.

**5.138** An inward flow radial turbine (see Fig. P5.137) involves a nozzle angle, $\alpha_1$, of 60° and an inlet rotor tip speed of 30 ft/s. The ratio of rotor inlet to outlet diameters is 2.0. The radial component of velocity remains constant at 20 ft/s through the rotor, and the flow leaving the rotor at section (2) is without angular momentum. If the flowing fluid is air and the static pressure drop across the rotor is 0.01 psi, determine the loss of available energy across the rotor and the rotor aerodynamic efficiency.




■ FIGURE P5.137

### Section 5.4 Second Law of Thermodynamics—Irreversible Flow

**5.139** Why do all actual fluid flows involve loss of available energy?

#### ■ Lab Problems

**5.140** This problem involves the force that a jet of air exerts on a flat plate as the air is deflected by the plate. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

**5.141** This problem involves the pressure distribution produced on a flat plate that deflects a jet of air. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

**5.142** This problem involves the force that a jet of water exerts on a vane when the vane turns the jet through a given angle. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

**5.143** This problem involves the force needed to hold a pipe elbow stationary. To proceed with this problem, go to Appendix H which is located on the book's web site, www.wiley.com/college/munson.

#### ■ Life Long Learning Problems

**5.144** What are typical efficiencies associated with swimming and how can they be improved?

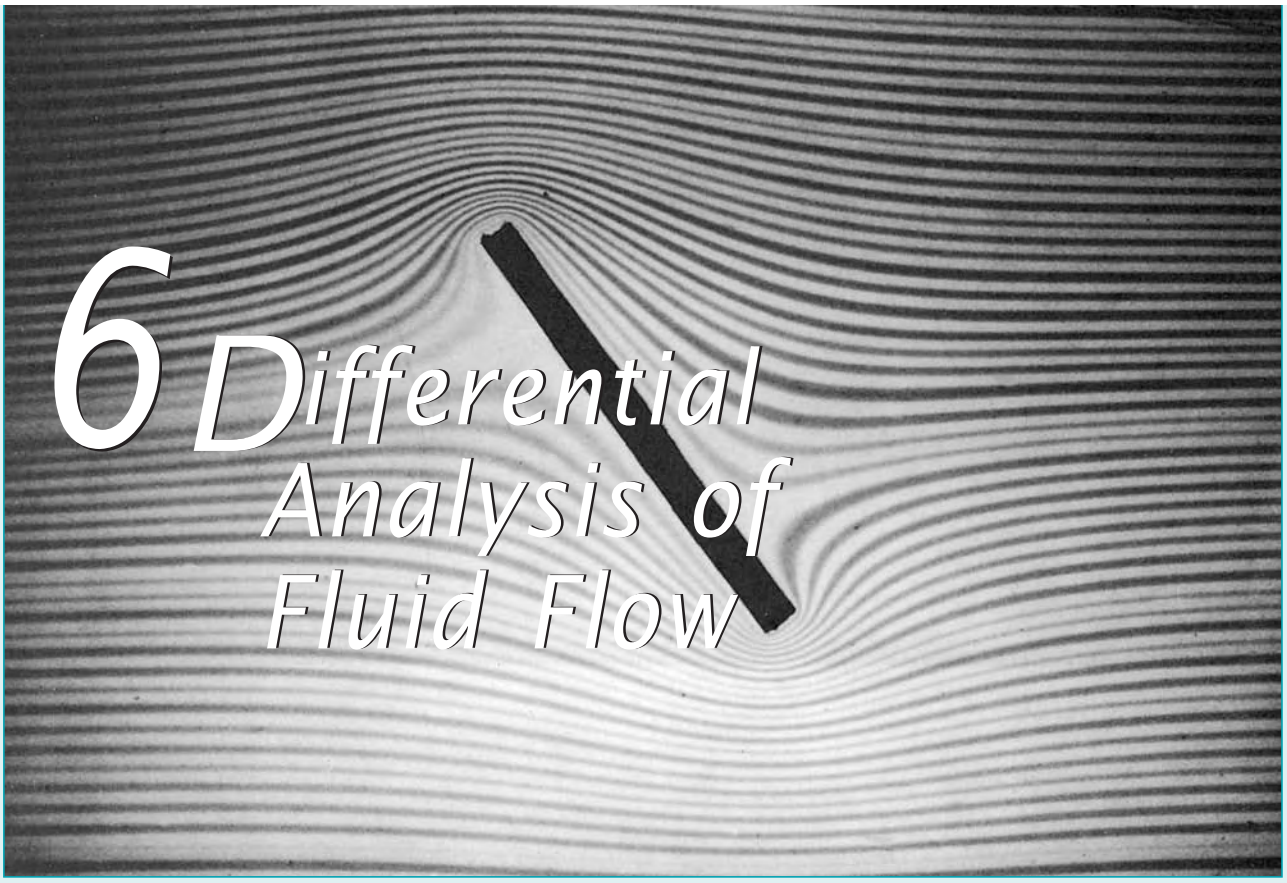
**5.145** Explain how local ionization of flowing air can accelerate it. How can this be useful?

**5.146** Discuss the main causes of loss of available energy in a turbo-pump and how they can be minimized. What are typical turbo-pump efficiencies?

**5.147** Discuss the main causes of loss of available energy in a turbine and how they can be minimized. What are typical turbine efficiencies?

#### ■ FE Exam Problems

Sample FE (Fundamentals of Engineering) exam questions for fluid mechanics are provided on the book's web site, www.wiley.com/college/munson.



# 6 Differential Analysis of Fluid Flow

CHAPTER OPENING PHOTO: Flow past an inclined plate: The streamlines of a viscous fluid flowing slowly past a two-dimensional object placed between two closely spaced plates (a Hele-Shaw cell) approximate inviscid, irrotational (potential) flow. (Dye in water between glass plates spaced 1 mm apart.) (Photography courtesy of D. H. Peregrine.)

## Learning Objectives

After completing this chapter, you should be able to:

- determine various kinematic elements of the flow given the velocity field.
- explain the conditions necessary for a velocity field to satisfy the continuity equation.
- apply the concepts of stream function and velocity potential.
- characterize simple potential flow fields.
- analyze certain types of flows using the Navier–Stokes equations.

In the previous chapter attention is focused on the use of finite control volumes for the solution of a variety of fluid mechanics problems. This approach is very practical and useful, since it does not generally require a detailed knowledge of the pressure and velocity variations within the control volume. Typically, we found that only conditions on the surface of the control volume were needed, and thus problems could be solved without a detailed knowledge of the flow field. Unfortunately, there are many situations that arise in which the details of the flow are important and the finite control volume approach will not yield the desired information. For example, we may need to know how the velocity varies over the cross section of a pipe, or how the pressure and shear stress vary along the surface of an airplane wing. In these circumstances we need to develop relationships that apply at a point, or at least in a very small infinitesimal region within a given flow field. This approach, which involves an *infinitesimal control volume*, as distinguished from a finite control volume, is commonly referred to as *differential analysis*, since (as we will soon discover) the governing equations are differential equations.

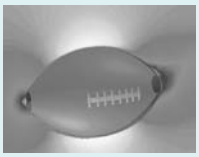


CMS Cepcor EX 1046 Page 173
CMS Cepcor v. Sandvik IPR2024-01380

In this chapter we will provide an introduction to the differential equations that describe (in detail) the motion of fluids. Unfortunately, we will also find that these equations are rather complicated, non-linear partial differential equations that cannot be solved exactly except in a few cases, where simplifying assumptions are made. Thus, although differential analysis has the potential for supplying very detailed information about flow fields, this information is not easily extracted. Nevertheless, this approach provides a fundamental basis for the study of fluid mechanics. We do not want to be too discouraging at this point, since there are some exact solutions for laminar flow that can be obtained, and these have proved to be very useful. A few of these are included in this chapter. In addition, by making some simplifying assumptions many other analytical solutions can be obtained. For example, in some circumstances it may be reasonable to assume that the effect of viscosity is small and can be neglected. This rather drastic assumption greatly simplifies the analysis and provides the opportunity to obtain detailed solutions to a variety of complex flow problems. Some examples of these so-called *inviscid flow* solutions are also described in this chapter.



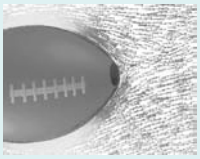
*V6.1 Spinning football-velocity contours*



It is known that for certain types of flows the flow field can be conceptually divided into two regions—a very thin region near the boundaries of the system in which viscous effects are important, and a region away from the boundaries in which the flow is essentially inviscid. By making certain assumptions about the behavior of the fluid in the thin layer near the boundaries, and using the assumption of inviscid flow outside this layer, a large class of problems can be solved using differential analysis. These boundary layer problems are discussed in Chapter 9. Finally, it is to be noted that with the availability of powerful computers it is feasible to attempt to solve the differential equations using the techniques of numerical analysis. Although it is beyond the scope of this book to delve extensively into this approach, which is generally referred to as *computational fluid dynamics* (CFD), the reader should be aware of this approach to complex flow problems. CFD has become a common engineering tool and a brief introduction can be found in Appendix A. To introduce the power of CFD, two animations based on the numerical computations are provided as shown in the margin.



*V6.2 Spinning football-velocity vectors*

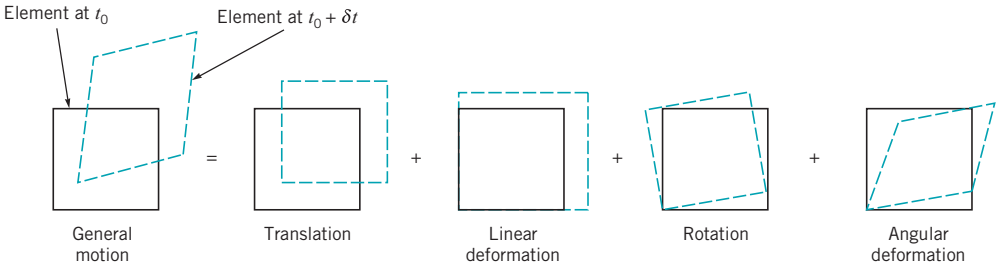


We begin our introduction to differential analysis by reviewing and extending some of the ideas associated with fluid kinematics that were introduced in Chapter 4. With this background the remainder of the chapter will be devoted to the derivation of the basic differential equations (which will be based on the principle of conservation of mass and Newton's second law of motion) and to some applications.

## 6.1 Fluid Element Kinematics

*Fluid element motion consists of translation, linear deformation, rotation, and angular deformation.*

In this section we will be concerned with the mathematical description of the motion of fluid elements moving in a flow field. A small fluid element in the shape of a cube which is initially in one position will move to another position during a short time interval $\delta t$ as illustrated in Fig. 6.1. Because of the generally complex velocity variation within the field, we expect the element not only to translate from one position but also to have its volume changed (linear deformation), to rotate, and to undergo a change in shape (angular deformation). Although these movements and deformations occur simultaneously, we can consider each one separately as illustrated in Fig. 6.1. Since element motion and deformation are intimately related to the velocity and variation of velocity throughout the flow field, we will briefly review the manner in which velocity and acceleration fields can be described.

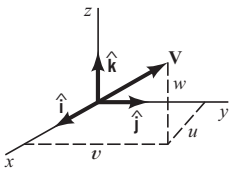



■ **FIGURE 6.1** **Types of motion and deformation for a fluid element.**

### 6.1.1 Velocity and Acceleration Fields Revisited

As discussed in detail in Section 4.1, the velocity field can be described by specifying the velocity **V** at all points, and at all times, within the flow field of interest. Thus, in terms of rectangular coordinates, the notation $\mathbf{V}(x, y, z, t)$ means that the velocity of a fluid particle depends on where it is located within the flow field (as determined by its coordinates, $x$, $y$, and $z$) and when it occupies the particular point (as determined by the time, $t$). As is pointed out in Section 4.1.1, this method of describing the fluid motion is called the Eulerian method. It is also convenient to express the velocity in terms of three rectangular components so that

$$\mathbf{V} = u\hat{\mathbf{i}} + v\hat{\mathbf{j}} + w\hat{\mathbf{k}} \tag{6.1}$$




where $u$, $v$, and $w$ are the velocity components in the $x$, $y$, and $z$ directions, respectively, and $\hat{\mathbf{i}}$, $\hat{\mathbf{j}}$, and $\hat{\mathbf{k}}$ are the corresponding unit vectors, as shown by the figure in the margin. Of course, each of these components will, in general, be a function of $x$, $y$, $z$, and $t$. One of the goals of differential analysis is to determine how these velocity components specifically depend on $x$, $y$, $z$, and $t$ for a particular problem.

With this description of the velocity field it was also shown in Section 4.2.1 that the acceleration of a fluid particle can be expressed as

$$\mathbf{a} = \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z} \tag{6.2}$$

and in component form:

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \tag{6.3a}$$

$$a_y = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \tag{6.3b}$$

$$a_z = \frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \tag{6.3c}$$

*The acceleration of a fluid particle is described using the concept of the material derivative.*

The acceleration is also concisely expressed as

$$\mathbf{a} = \frac{D\mathbf{V}}{Dt} \tag{6.4}$$

where the operator

$$\frac{D(\ )}{Dt} = \frac{\partial(\ )}{\partial t} + u\frac{\partial(\ )}{\partial x} + v\frac{\partial(\ )}{\partial y} + w\frac{\partial(\ )}{\partial z} \tag{6.5}$$

is termed the *material derivative*, or *substantial derivative*. In vector notation

$$\frac{D(\ )}{Dt} = \frac{\partial(\ )}{\partial t} + (\mathbf{V}\cdot\nabla)(\ ) \tag{6.6}$$

where the gradient operator, $\nabla(\ )$, is

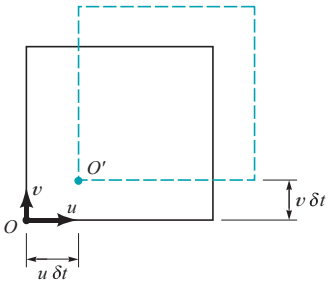
$$\nabla(\ ) = \frac{\partial(\ )}{\partial x}\hat{\mathbf{i}} + \frac{\partial(\ )}{\partial y}\hat{\mathbf{j}} + \frac{\partial(\ )}{\partial z}\hat{\mathbf{k}} \tag{6.7}$$

which was introduced in Chapter 2. As we will see in the following sections, the motion and deformation of a fluid element depend on the velocity field. The relationship between the motion and the forces causing the motion depends on the acceleration field.

### 6.1.2 Linear Motion and Deformation

The simplest type of motion that a fluid element can undergo is translation, as illustrated in Fig. 6.2. In a small time interval $\delta t$ a particle located at point $O$ will move to point $O'$ as is illustrated in the figure. If all points in the element have the same velocity (which is only true if there are no velocity gradients), then the element will simply translate from one position to another. However,

CMS Cepcor EX 1046 Page 175
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 6.2** **Translation of a fluid element.**

because of the presence of velocity gradients, the element will generally be deformed and rotated as it moves. For example, consider the effect of a single velocity gradient, $\partial u/\partial x$, on a small cube having sides $\delta x$, $\delta y$, and $\delta z$. As is shown in Fig. 6.3*a*, if the $x$ component of velocity of $O$ and $B$ is $u$, then at nearby points $A$ and $C$ the $x$ component of the velocity can be expressed as $u + (\partial u/\partial x)\,\delta x$. This difference in velocity causes a "stretching" of the volume element by an amount $(\partial u/\partial x)(\delta x)(\delta t)$ during the short time interval $\delta t$ in which line $OA$ stretches to $OA'$ and $BC$ to $BC'$ (Fig. 6.3*b*). The corresponding change in the original volume, $\delta \forall = \delta x\,\delta y\,\delta z$, would be

$$\text{Change in } \delta \forall = \left(\frac{\partial u}{\partial x}\,\delta x\right)(\delta y\,\delta z)(\delta t)$$

and the *rate* at which the volume $\delta \forall$ is changing *per unit volume* due to the gradient $\partial u/\partial x$ is

$$\frac{1}{\delta \forall}\frac{d(\delta \forall)}{dt} = \lim_{\delta t \to 0}\left[\frac{(\partial u/\partial x)\,\delta t}{\delta t}\right] = \frac{\partial u}{\partial x} \tag{6.8}$$

*The rate of volume change per unit volume is related to the velocity gradients.*

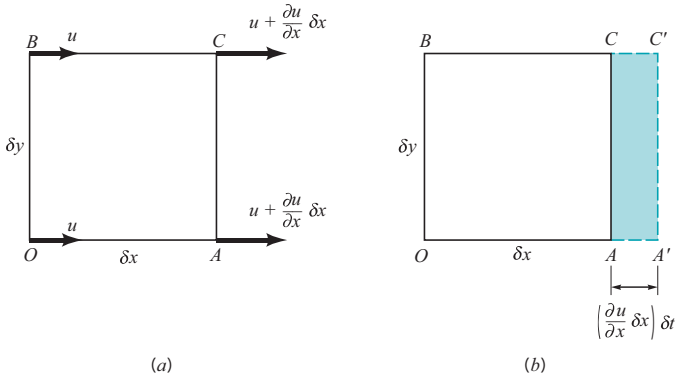
If velocity gradients $\partial v/\partial y$ and $\partial w/\partial z$ are also present, then using a similar analysis it follows that, in the general case,

$$\frac{1}{\delta \forall}\frac{d(\delta \forall)}{dt} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = \nabla \cdot \mathbf{V} \tag{6.9}$$

This rate of change of the volume per unit volume is called the ***volumetric dilatation rate.*** Thus, we see that the volume of a fluid may change as the element moves from one location to another in the flow field. However, for an *incompressible fluid* the volumetric dilatation rate is zero, since the element volume cannot change without a change in fluid density (the element mass must be conserved). Variations in the velocity in the direction of the velocity, as represented by the derivatives $\partial u/\partial x$, $\partial v/\partial y$, and $\partial w/\partial z$, simply cause a *linear deformation* of the element in the sense that the shape of the element does not change. Cross derivatives, such as $\partial u/\partial y$ and $\partial v/\partial x$, will cause the element to rotate and generally to undergo an *angular deformation*, which changes the shape of the element.

### 6.1.3 Angular Motion and Deformation

For simplicity we will consider motion in the $x$–$y$ plane, but the results can be readily extended to the more general three dimensional case. The velocity variation that causes rotation and angular deformation is illustrated in Fig. 6.4*a*. In a short time interval $\delta t$ the line segments $OA$ and $OB$ will




■ **FIGURE 6.3**
**Linear deformation of a fluid element.**

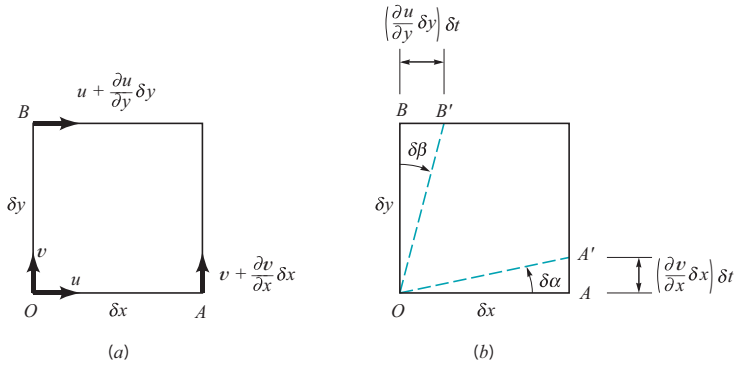



**FIGURE 6.4**
**Angular motion and deformation of a fluid element.**



*V6.3 Shear deformation*



rotate through the angles $\delta\alpha$ and $\delta\beta$ to the new positions $OA'$ and $OB'$, as is shown in Fig. 6.4*b*. The angular velocity of line $OA$, $\omega_{OA}$, is

$$\omega_{OA} = \lim_{\delta t \to 0} \frac{\delta\alpha}{\delta t}$$

For small angles

$$\tan \delta\alpha \approx \delta\alpha = \frac{(\partial v/\partial x)\,\delta x\,\delta t}{\delta x} = \frac{\partial v}{\partial x}\,\delta t \tag{6.10}$$

so that

$$\omega_{OA} = \lim_{\delta t \to 0}\left[\frac{(\partial v/\partial x)\,\delta t}{\delta t}\right] = \frac{\partial v}{\partial x}$$

Note that if $\partial v/\partial x$ is positive, $\omega_{OA}$ will be counterclockwise. Similarly, the angular velocity of the line $OB$ is

$$\omega_{OB} = \lim_{\delta t \to 0} \frac{\delta\beta}{\delta t}$$

and

$$\tan \delta\beta \approx \delta\beta = \frac{(\partial u/\partial y)\,\delta y\,\delta t}{\delta y} = \frac{\partial u}{\partial y}\,\delta t \tag{6.11}$$

so that

$$\omega_{OB} = \lim_{\delta t \to 0}\left[\frac{(\partial u/\partial y)\,\delta t}{\delta t}\right] = \frac{\partial u}{\partial y}$$

*Rotation of fluid particles is related to certain velocity gradients in the flow field.*

In this instance if $\partial u/\partial y$ is positive, $\omega_{OB}$ will be clockwise. The *rotation*, $\omega_z$, of the element about the $z$ axis is defined as the average of the angular velocities $\omega_{OA}$ and $\omega_{OB}$ of the two mutually perpendicular lines $OA$ and $OB$.[1] Thus, if counterclockwise rotation is considered to be positive, it follows that

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \tag{6.12}$$

Rotation of the field element about the other two coordinate axes can be obtained in a similar manner with the result that for rotation about the $x$ axis

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) \tag{6.13}$$

and for rotation about the $y$ axis

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) \tag{6.14}$$

[1]With this definition $\omega_z$ can also be interpreted to be the angular velocity of the bisector of the angle between the lines $OA$ and $OB$.

CMS Cepcor EX 1046 Page 177
CMS Cepcor v. Sandvik IPR2024-01380

The three components, $\omega_x$, $\omega_y$, and $\omega_z$ can be combined to give the rotation vector, $\boldsymbol{\omega}$, in the form

$$\boldsymbol{\omega} = \omega_x\hat{\mathbf{i}} + \omega_y\hat{\mathbf{j}} + \omega_z\hat{\mathbf{k}} \tag{6.15}$$

An examination of this result reveals that $\boldsymbol{\omega}$ is equal to one-half the curl of the velocity vector. That is,

$$\boldsymbol{\omega} = \tfrac{1}{2}\,\text{curl}\,\mathbf{V} = \tfrac{1}{2}\,\nabla \times \mathbf{V} \tag{6.16}$$

since by definition of the vector operator $\nabla \times \mathbf{V}$

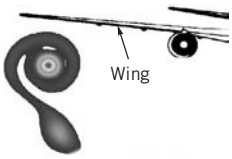
$$\frac{1}{2}\nabla \times \mathbf{V} = \frac{1}{2}\begin{vmatrix} \hat{\mathbf{i}} & \hat{\mathbf{j}} & \hat{\mathbf{k}} \\ \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial z} \\ u & v & w \end{vmatrix}$$

$$= \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\hat{\mathbf{i}} + \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\hat{\mathbf{j}} + \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\hat{\mathbf{k}}$$

*Vorticity in a flow field is related to fluid particle rotation.*

The ***vorticity***, $\boldsymbol{\zeta}$, is defined as a vector that is twice the rotation vector; that is,

$$\boldsymbol{\zeta} = 2\,\boldsymbol{\omega} = \nabla \times \mathbf{V} \tag{6.17}$$




The use of the vorticity to describe the rotational characteristics of the fluid simply eliminates the $(\frac{1}{2})$ factor associated with the rotation vector. The figure in the margin shows vorticity contours of the wing tip vortex flow shortly after an aircraft has passed. The lighter colors indicate stronger vorticity. (See also Fig. 4.3.)

We observe from Eq. 6.12 that the fluid element will rotate about the $z$ axis as an *undeformed* block (i.e., $\omega_{OA} = -\omega_{OB}$) only when $\partial u/\partial y = -\partial v/\partial x$. Otherwise the rotation will be associated with an angular deformation. We also note from Eq. 6.12 that when $\partial u/\partial y = \partial v/\partial x$ the rotation around the $z$ axis is zero. More generally if $\nabla \times \mathbf{V} = 0$, then the rotation (and the vorticity) are zero, and flow fields for which this condition applies are termed ***irrotational.*** We will find in Section 6.4 that the condition of irrotationality often greatly simplifies the analysis of complex flow fields. However, it is probably not immediately obvious why some flow fields would be irrotational, and we will need to examine this concept more fully in Section 6.4.

## EXAMPLE 6.1 Vorticity

**GIVEN** For a certain two-dimensional flow field the velocity is given by the equation

$$\mathbf{V} = (x^2 - y^2)\hat{\mathbf{i}} - 2xy\hat{\mathbf{j}}$$

**FIND** Is this flow irrotational?

### SOLUTION

For an irrotational flow the rotation vector, $\boldsymbol{\omega}$, having the components given by Eqs. 6.12, 6.13, and 6.14 must be zero. For the prescribed velocity field

$$u = x^2 - y^2 \qquad v = -2xy \qquad w = 0$$

and therefore

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) = 0$$

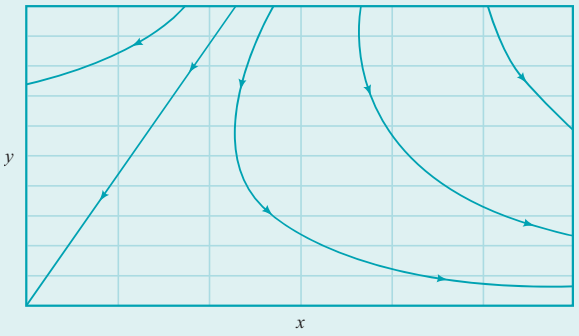
$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) = 0$$

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = \frac{1}{2}\left[(-2y) - (-2y)\right] = 0$$

Thus, the flow is irrotational. **(Ans)**

**COMMENTS** It is to be noted that for a two-dimensional flow field (where the flow is in the $x$–$y$ plane) $\omega_x$ and $\omega_y$ will always be zero, since by definition of two-dimensional flow $u$ and $v$ are not functions of $z$, and $w$ is zero. In this instance the condition for irrotationality simply becomes $\omega_z = 0$ or $\partial v/\partial x = \partial u/\partial y$.

The streamlines for the steady, two-dimensional flow of this example are shown in Fig. E6.1. (Information about how to calculate




■ FIGURE E6.1

streamlines for a given velocity field is given in Sections 4.1.4 and 6.2.3.) It is noted that all of the streamlines (except for the one through the origin) are curved. However, because the flow is irrotational, there is no rotation of the fluid elements. That is, lines *OA* and *OB* of Fig. 6.4 rotate with the same speed but in opposite directions.

As shown by Eq. 6.17, the condition of irrotationality is equivalent to the fact that the vorticity, $\boldsymbol{\zeta}$, is zero or the curl of the velocity is zero.

In addition to the rotation associated with the derivatives $\partial u/\partial y$ and $\partial v/\partial x$, it is observed from Fig. 6.4*b* that these derivatives can cause the fluid element to undergo an *angular deformation*, which results in a change in shape of the element. The change in the original right angle formed by the lines *OA* and *OB* is termed the shearing strain, $\delta\gamma$, and from Fig. 6.4*b*

$$\delta\gamma = \delta\alpha + \delta\beta$$

where $\delta\gamma$ is considered to be positive if the original right angle is decreasing. The rate of change of $\delta\gamma$ is called the *rate of shearing strain* or the *rate of angular deformation* and is commonly denoted with the symbol $\dot{\gamma}$. The angles $\delta\alpha$ and $\delta\beta$ are related to the velocity gradients through Eqs. 6.10 and 6.11 so that

$$\dot{\gamma} = \lim_{\delta t \to 0} \frac{\delta\gamma}{\delta t} = \lim_{\delta t \to 0} \left[ \frac{(\partial v/\partial x)\,\delta t + (\partial u/\partial y)\,\delta t}{\delta t} \right]$$

and, therefore,

$$\dot{\gamma} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \tag{6.18}$$

As we will learn in Section 6.8, the rate of angular deformation is related to a corresponding shearing stress which causes the fluid element to change in shape. From Eq. 6.18 we note that if $\partial u/\partial y = -\partial v/\partial x$, the rate of angular deformation is zero, and this condition corresponds to the case in which the element is simply rotating as an undeformed block (Eq. 6.12). In the remainder of this chapter we will see how the various kinematical relationships developed in this section play an important role in the development and subsequent analysis of the differential equations that govern fluid motion.

## 6.2 Conservation of Mass

*Conservation of mass requires that the mass of a system remain constant.*

As is discussed in Section 5.1, conservation of mass requires that the mass, $M$, of a system remain constant as the system moves through the flow field. In equation form this principle is expressed as

$$\frac{DM_{\text{sys}}}{Dt} = 0$$

We found it convenient to use the control volume approach for fluid flow problems, with the control volume representation of the conservation of mass written as

$$\frac{\partial}{\partial t} \int_{\text{cv}} \rho \, d\forall + \int_{\text{cs}} \rho \mathbf{V} \cdot \hat{\mathbf{n}} \, dA = 0 \tag{6.19}$$

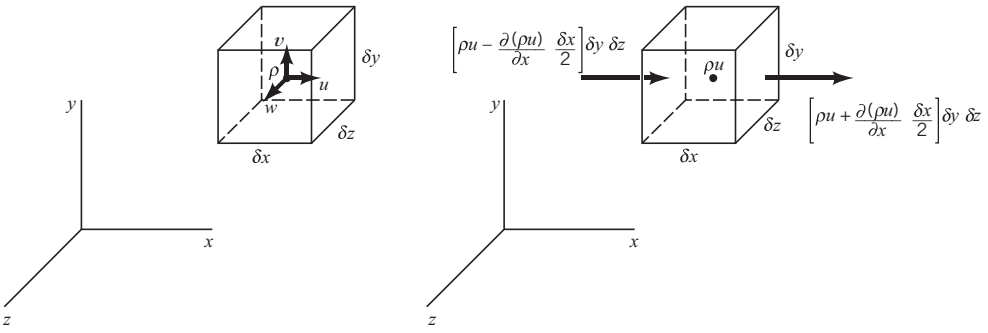
where the equation (commonly called the ***continuity equation***) can be applied to a finite control volume (cv), which is bounded by a control surface (cs). The first integral on the left side of Eq. 6.19 represents the rate at which the mass within the control volume is changing, and the second integral represents the net rate at which mass is flowing out through the control surface (rate of mass outflow − rate of mass inflow). To obtain the differential form of the continuity equation, Eq. 6.19 is applied to an infinitesimal control volume.

### 6.2.1 Differential Form of Continuity Equation

We will take as our control volume the small, stationary cubical element shown in Fig. 6.5*a*. At the center of the element the fluid density is $\rho$ and the velocity has components $u$, $v$, and $w$. Since the element is small, the volume integral in Eq. 6.19 can be expressed as

$$\frac{\partial}{\partial t} \int_{\text{cv}} \rho \, d\forall \approx \frac{\partial \rho}{\partial t} \delta x \, \delta y \, \delta z \tag{6.20}$$

CMS Cepcor EX 1046 Page 179
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 6.5** **A differential element for the development of conservation of mass equation.**

The rate of mass flow through the surfaces of the element can be obtained by considering the flow in each of the coordinate directions separately. For example, in Fig. 6.5*b* flow in the $x$ direction is depicted. If we let $\rho u$ represent the $x$ component of the mass rate of flow per unit area at the center of the element, then on the right face

$$\rho u\big|_{x+(\delta x/2)} = \rho u + \frac{\partial(\rho u)}{\partial x}\frac{\delta x}{2} \tag{6.21}$$

and on the left face

$$\rho u\big|_{x-(\delta x/2)} = \rho u - \frac{\partial(\rho u)}{\partial x}\frac{\delta x}{2} \tag{6.22}$$

Note that we are really using a Taylor series expansion of $\rho u$ and neglecting higher order terms such as $(\delta x)^2$, $(\delta x)^3$, and so on. When the right-hand sides of Eqs. 6.21 and 6.22 are multiplied by the area $\delta y\,\delta z$, the rate at which mass is crossing the right and left sides of the element are obtained as is illustrated in Fig. 6.5*b*. When these two expressions are combined, the net rate of mass flowing from the element through the two surfaces can be expressed as

$$\text{Net rate of mass outflow in } x \text{ direction} = \left[\rho u + \frac{\partial(\rho u)}{\partial x}\frac{\delta x}{2}\right]\delta y\,\delta z - \left[\rho u - \frac{\partial(\rho u)}{\partial x}\frac{\delta x}{2}\right]\delta y\,\delta z = \frac{\partial(\rho u)}{\partial x}\delta x\,\delta y\,\delta z \tag{6.23}$$

For simplicity, only flow in the $x$ direction has been considered in Fig. 6.5*b*, but, in general, there will also be flow in the $y$ and $z$ directions. An analysis similar to the one used for flow in the $x$ direction shows that

$$\text{Net rate of mass outflow in } y \text{ direction} = \frac{\partial(\rho v)}{\partial y}\delta x\,\delta y\,\delta z \tag{6.24}$$

and

$$\text{Net rate of mass outflow in } z \text{ direction} = \frac{\partial(\rho w)}{\partial z}\delta x\,\delta y\,\delta z \tag{6.25}$$

Thus,

$$\text{Net rate of mass outflow} = \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z}\right]\delta x\,\delta y\,\delta z \tag{6.26}$$

From Eqs. 6.19, 6.20, and 6.26 it now follows that the differential equation for conservation of mass is

$$\boxed{\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0} \tag{6.27}$$

*The continuity equation is one of the fundamental equations of fluid mechanics.*

As previously mentioned, this equation is also commonly referred to as the continuity equation.

The continuity equation is one of the fundamental equations of fluid mechanics and, as expressed in Eq. 6.27, is valid for steady or unsteady flow, and compressible or incompressible fluids. In vector notation, Eq. 6.27 can be written as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \mathbf{V} = 0 \tag{6.28}$$

Two special cases are of particular interest. For *steady* flow of *compressible* fluids

$$\nabla \cdot \rho \mathbf{V} = 0$$

or

$$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \tag{6.29}$$

*For incompressible fluids the continuity equation reduces to a simple relationship involving certain velocity gradients.*

This follows since by definition $\rho$ is not a function of time for steady flow, but could be a function of position. For *incompressible* fluids the fluid density, $\rho$, is a constant throughout the flow field so that Eq. 6.28 becomes

$$\nabla \cdot \mathbf{V} = 0 \tag{6.30}$$

or

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{6.31}$$

Equation 6.31 applies to both steady and unsteady flow of incompressible fluids. Note that Eq. 6.31 is the same as that obtained by setting the volumetric dilatation rate (Eq. 6.9) equal to zero. This result should not be surprising since both relationships are based on conservation of mass for incompressible fluids. However, the expression for the volumetric dilatation rate was developed from a system approach, whereas Eq. 6.31 was developed from a control volume approach. In the former case the deformation of a particular differential mass of fluid was studied, and in the latter case mass flow through a fixed differential volume was studied.

## EXAMPLE 6.2 Continuity Equation

**GIVEN** The velocity components for a certain incompressible, steady flow field are

$$u = x^2 + y^2 + z^2$$
$$v = xy + yz + z$$
$$w = ?$$

**FIND** Determine the form of the $z$ component, $w$, required to satisfy the continuity equation.

### SOLUTION

Any physically possible velocity distribution must for an incompressible fluid satisfy conservation of mass as expressed by the continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

For the given velocity distribution

$$\frac{\partial u}{\partial x} = 2x \qquad \text{and} \qquad \frac{\partial v}{\partial y} = x + z$$

so that the required expression for $\partial w/\partial z$ is

$$\frac{\partial w}{\partial z} = -2x - (x + z) = -3x - z$$

Integration with respect to $z$ yields

$$w = -3xz - \frac{z^2}{2} + f(x, y) \qquad \textbf{(Ans)}$$

**COMMENT** The third velocity component cannot be explicitly determined since the function $f(x, y)$ can have any form and conservation of mass will still be satisfied. The specific form of this function will be governed by the flow field described by these velocity components—that is, some additional information is needed to completely determine $w$.

CMS Cepcor EX 1046 Page 181
CMS Cepcor v. Sandvik IPR2024-01380

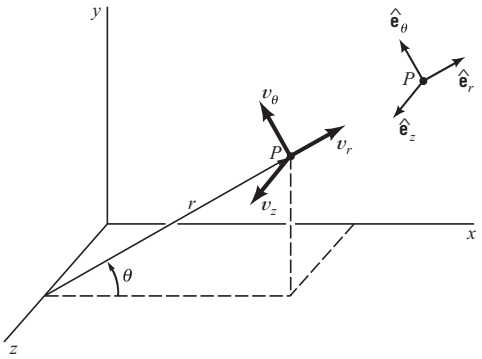



**FIGURE 6.6** The representation of velocity components in cylindrical polar coordinates.

### 6.2.2 Cylindrical Polar Coordinates

*For some problems, velocity components expressed in cylindrical polar coordinates will be convenient.*

For some problems it is more convenient to express the various differential relationships in cylindrical polar coordinates rather than Cartesian coordinates. As is shown in Fig. 6.6, with cylindrical coordinates a point is located by specifying the coordinates $r$, $\theta$, and $z$. The coordinate $r$ is the radial distance from the $z$ axis, $\theta$ is the angle measured from a line parallel to the $x$ axis (with counterclockwise taken as positive), and $z$ is the coordinate along the $z$ axis. The velocity components, as sketched in Fig. 6.6, are the radial velocity, $v_r$, the tangential velocity, $v_\theta$, and the axial velocity, $v_z$. Thus, the velocity at some arbitrary point $P$ can be expressed as

$$\mathbf{V} = v_r\hat{\mathbf{e}}_r + v_\theta\hat{\mathbf{e}}_\theta + v_z\hat{\mathbf{e}}_z \tag{6.32}$$

where $\hat{\mathbf{e}}_r$, $\hat{\mathbf{e}}_\theta$, and $\hat{\mathbf{e}}_z$ are the unit vectors in the $r$, $\theta$, and $z$ directions, respectively, as are illustrated in Fig. 6.6. The use of cylindrical coordinates is particularly convenient when the boundaries of the flow system are cylindrical. Several examples illustrating the use of cylindrical coordinates will be given in succeeding sections in this chapter.

The differential form of the continuity equation in cylindrical coordinates is

$$\boxed{\frac{\partial\rho}{\partial t} + \frac{1}{r}\frac{\partial(r\rho v_r)}{\partial r} + \frac{1}{r}\frac{\partial(\rho v_\theta)}{\partial\theta} + \frac{\partial(\rho v_z)}{\partial z} = 0} \tag{6.33}$$

This equation can be derived by following the same procedure used in the preceding section (see Problem 6.20). For steady, compressible flow

$$\frac{1}{r}\frac{\partial(r\rho v_r)}{\partial r} + \frac{1}{r}\frac{\partial(\rho v_\theta)}{\partial\theta} + \frac{\partial(\rho v_z)}{\partial z} = 0 \tag{6.34}$$

For incompressible fluids (for steady or unsteady flow)

$$\frac{1}{r}\frac{\partial(rv_r)}{\partial r} + \frac{1}{r}\frac{\partial v_\theta}{\partial\theta} + \frac{\partial v_z}{\partial z} = 0 \tag{6.35}$$

### 6.2.3 The Stream Function

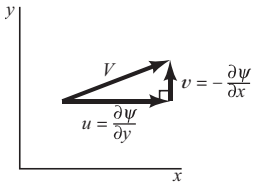
Steady, incompressible, plane, two-dimensional flow represents one of the simplest types of flow of practical importance. By plane, two-dimensional flow we mean that there are only two velocity components, such as $u$ and $v$, when the flow is considered to be in the $x$–$y$ plane. For this flow the continuity equation, Eq. 6.31, reduces to

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{6.36}$$

*Velocity components in a two-dimensional flow field can be expressed in terms of a stream function.*

We still have two variables, $u$ and $v$, to deal with, but they must be related in a special way as indicated by Eq. 6.36. This equation suggests that if we define a function $\psi(x, y)$, called the ***stream function***, which relates the velocities shown by the figure in the margin as

$$u = \frac{\partial \psi}{\partial y} \qquad v = -\frac{\partial \psi}{\partial x} \tag{6.37}$$




then the continuity equation is identically satisfied. This conclusion can be verified by simply substituting the expressions for $u$ and $v$ into Eq. 6.36 so that

$$\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial y}\right) + \frac{\partial}{\partial y}\left(-\frac{\partial \psi}{\partial x}\right) = \frac{\partial^2 \psi}{\partial x\, \partial y} - \frac{\partial^2 \psi}{\partial y\, \partial x} = 0$$

Thus, whenever the velocity components are defined in terms of the stream function we know that conservation of mass will be satisfied. Of course, we still do not know what $\psi(x, y)$ is for a particular problem, but at least we have simplified the analysis by having to determine only one unknown function, $\psi(x, y)$, rather than the two functions, $u(x, y)$ and $v(x, y)$.

Another particular advantage of using the stream function is related to the fact that *lines along which $\psi$ is constant are streamlines*. Recall from Section 4.1.4 that streamlines are lines in the flow field that are everywhere tangent to the velocities, as is illustrated in Fig. 6.7. It follows from the definition of the streamline that the slope at any point along a streamline is given by

$$\frac{dy}{dx} = \frac{v}{u}$$

The change in the value of $\psi$ as we move from one point $(x, y)$ to a nearby point $(x + dx, y + dy)$ is given by the relationship:

$$d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy = -v\, dx + u\, dy$$

Along a line of constant $\psi$ we have $d\psi = 0$ so that

$$-v\, dx + u\, dy = 0$$

and, therefore, along a line of constant $\psi$

$$\frac{dy}{dx} = \frac{v}{u}$$

which is the defining equation for a streamline. Thus, if we know the function $\psi(x, y)$ we can plot lines of constant $\psi$ to provide the family of streamlines that are helpful in visualizing the pattern

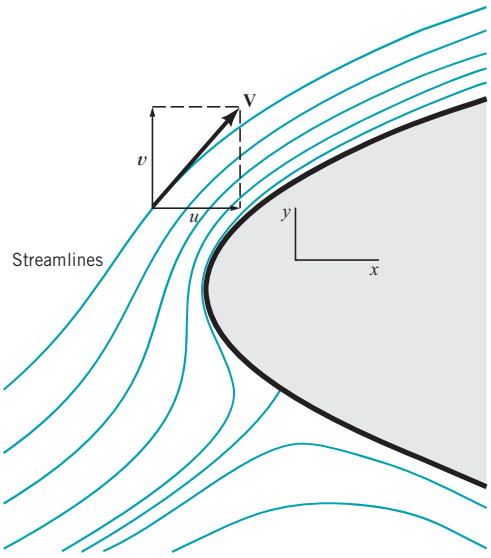
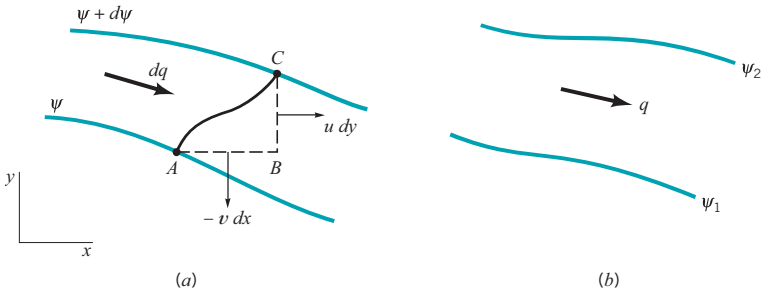



**FIGURE 6.7** Velocity and velocity components along a streamline.




■ **FIGURE 6.8** **The flow between two streamlines.**

of flow. There are an infinite number of streamlines that make up a particular flow field, since for each constant value assigned to $\psi$ a streamline can be drawn.

The actual numerical value associated with a particular streamline is not of particular significance, but the change in the value of $\psi$ is related to the volume rate of flow. Consider two closely spaced streamlines, shown in Fig. 6.8*a*. The lower streamline is designated $\psi$ and the upper one $\psi + d\psi$. Let $dq$ represent the volume rate of flow (per unit width perpendicular to the $x$–$y$ plane) passing between the two streamlines. Note that flow never crosses streamlines, since by definition the velocity is tangent to the streamline. From conservation of mass we know that the inflow, $dq$, crossing the arbitrary surface $AC$ of Fig. 6.8*a* must equal the net outflow through surfaces $AB$ and $BC$. Thus,

*The change in the value of the stream function is related to the volume rate of flow.*

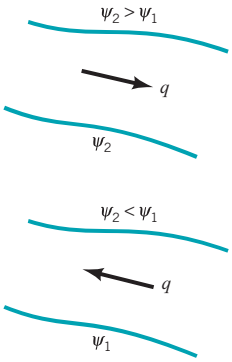
$$dq = u\,dy - v\,dx$$

or in terms of the stream function

$$dq = \frac{\partial \psi}{\partial y}\,dy + \frac{\partial \psi}{\partial x}\,dx \tag{6.38}$$

The right-hand side of Eq. 6.38 is equal to $d\psi$ so that

$$dq = d\psi \tag{6.39}$$




Thus, the volume rate of flow, $q$, between two streamlines such as $\psi_1$ and $\psi_2$ of Fig. 6.8*b* can be determined by integrating Eq. 6.39 to yield

$$q = \int_{\psi_1}^{\psi_2} d\psi = \psi_2 - \psi_1 \tag{6.40}$$

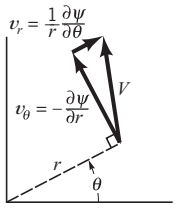
The relative value of $\psi_2$ with respect to $\psi_1$ determines the direction of flow, as shown by the figure in the margin.

In cylindrical coordinates the continuity equation (Eq. 6.35) for incompressible, plane, two-dimensional flow reduces to

$$\frac{1}{r}\frac{\partial (r v_r)}{\partial r} + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} = 0 \tag{6.41}$$

and the velocity components, $v_r$ and $v_\theta$, can be related to the stream function, $\psi(r, \theta)$, through the equations

$$v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \qquad v_\theta = -\frac{\partial \psi}{\partial r} \tag{6.42}$$




as shown by the figure in the margin.

Substitution of these expressions for the velocity components into Eq. 6.41 shows that the continuity equation is identically satisfied. The stream function concept can be extended to axisymmetric flows, such as flow in pipes or flow around bodies of revolution, and to two-dimensional compressible flows. However, the concept is not applicable to general three-dimensional flows.

## EXAMPLE 6.3 Stream Function

**GIVEN** The velocity components in a steady, incompressible, two-dimensional flow field are

$$u = 2y$$
$$v = 4x$$

**FIND**

**(a)** Determine the corresponding stream function and

**(b)** Show on a sketch several streamlines. Indicate the direction of flow along the streamlines.

### SOLUTION

**(a)** From the definition of the stream function (Eqs. 6.37)

$$u = \frac{\partial \psi}{\partial y} = 2y$$

and

$$v = -\frac{\partial \psi}{\partial x} = 4x$$

The first of these equations can be integrated to give

$$\psi = y^2 + f_1(x)$$

where $f_1(x)$ is an arbitrary function of $x$. Similarly from the second equation

$$\psi = -2x^2 + f_2(y)$$

where $f_2(y)$ is an arbitrary function of $y$. It now follows that in order to satisfy both expressions for the stream function

$$\psi = -2x^2 + y^2 + C \qquad \textbf{(Ans)}$$

where $C$ is an arbitrary constant.

**COMMENT** Since the velocities are related to the derivatives of the stream function, an arbitrary constant can always be added to the function, and the value of the constant is actually of no consequence. Usually, for simplicity, we set $C = 0$ so that for this particular example the simplest form for the stream function is

$$\psi = -2x^2 + y^2 \qquad \textbf{(1)} \quad \textbf{(Ans)}$$

Either answer indicated would be acceptable.

**(b)** Streamlines can now be determined by setting $\psi$ = constant and plotting the resulting curve. With the above expression for $\psi$ (with $C = 0$) the value of $\psi$ at the origin is zero so that the equation of the streamline passing through the origin (the $\psi = 0$ streamline) is

$$0 = -2x^2 + y^2$$

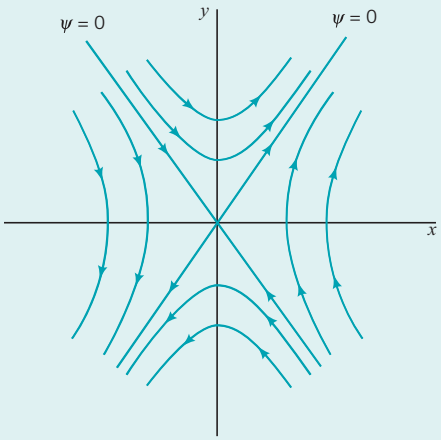



FIGURE E6.3

or

$$y = \pm\sqrt{2}x$$

Other streamlines can be obtained by setting $\psi$ equal to various constants. It follows from Eq. 1 that the equations of these streamlines (for $\psi \neq 0$) can be expressed in the form

$$\frac{y^2}{\psi} - \frac{x^2}{\psi/2} = 1$$

which we recognize as the equation of a hyperbola. Thus, the streamlines are a family of hyperbolas with the $\psi = 0$ streamlines as asymptotes. Several of the streamlines are plotted in Fig. E6.3. Since the velocities can be calculated at any point, the direction of flow along a given streamline can be easily deduced. For example, $v = -\partial\psi/\partial x = 4x$ so that $v > 0$ if $x > 0$ and $v < 0$ if $x < 0$. The direction of flow is indicated on the figure.

## 6.3 Conservation of Linear Momentum

To develop the differential momentum equations we can start with the linear momentum equation

$$\mathbf{F} = \left.\frac{D\mathbf{P}}{Dt}\right|_{\text{sys}} \qquad \textbf{(6.43)}$$

where $\mathbf{F}$ is the resultant force acting on a fluid mass, $\mathbf{P}$ is the linear momentum defined as

$$\mathbf{P} = \int_{\text{sys}} \mathbf{V}\, dm$$

CMS Cepcor EX 1046 Page 185
CMS Cepcor v. Sandvik IPR2024-01380

and the operator $D(\ )/Dt$ is the material derivative (see Section 4.2.12). In the last chapter it was demonstrated how Eq. 6.43 in the form

$$\sum \mathbf{F}_{\substack{\text{contents of the}\\ \text{control volume}}} = \frac{\partial}{\partial t}\int_{cv} \mathbf{V}\rho\, d\forall + \int_{cs} \mathbf{V}\rho\mathbf{V}\cdot\hat{\mathbf{n}}\, dA \tag{6.44}$$

could be applied to a finite control volume to solve a variety of flow problems. To obtain the differential form of the linear momentum equation, we can either apply Eq. 6.43 to a differential system, consisting of a mass, $\delta m$, or apply Eq. 6.44 to an infinitesimal control volume, $\delta\forall$, which initially bounds the mass $\delta m$. It is probably simpler to use the system approach since application of Eq. 6.43 to the differential mass, $\delta m$, yields

$$\delta\mathbf{F} = \frac{D(\mathbf{V}\,\delta m)}{Dt}$$

where $\delta\mathbf{F}$ is the resultant force acting on $\delta m$. Using this system approach $\delta m$ can be treated as a constant so that

$$\delta\mathbf{F} = \delta m\frac{D\mathbf{V}}{Dt}$$

But $D\mathbf{V}/Dt$ is the acceleration, $\mathbf{a}$, of the element. Thus,

$$\delta\mathbf{F} = \delta m\,\mathbf{a} \tag{6.45}$$

which is simply Newton's second law applied to the mass $\delta_m$. This is the same result that would be obtained by applying Eq. 6.44 to an infinitesimal control volume (see Ref. 1). Before we can proceed, it is necessary to examine how the force $\delta\mathbf{F}$ can be most conveniently expressed.

### 6.3.1 Description of Forces Acting on the Differential Element

*Both surface forces and body forces generally act on fluid particles.*

In general, two types of forces need to be considered: *surface forces*, which act on the surface of the differential element, and *body forces*, which are distributed throughout the element. For our purpose, the only body force, $\delta\mathbf{F}_b$, of interest is the weight of the element, which can be expressed as

$$\delta\mathbf{F}_b = \delta m\,\mathbf{g} \tag{6.46}$$

where $\mathbf{g}$ is the vector representation of the acceleration of gravity. In component form

$$\delta F_{bx} = \delta m\, g_x \tag{6.47a}$$

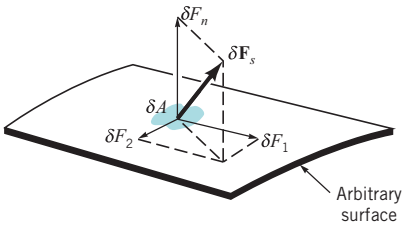
$$\delta F_{by} = \delta m\, g_y \tag{6.47b}$$

$$\delta F_{bz} = \delta m\, g_z \tag{6.47c}$$

where $g_x$, $g_y$, and $g_z$ are the components of the acceleration of gravity vector in the $x$, $y$, and $z$ directions, respectively.

Surface forces act on the element as a result of its interaction with its surroundings. At any arbitrary location within a fluid mass, the force acting on a small area, $\delta A$, which lies in an arbitrary surface, can be represented by $\delta\mathbf{F}_s$, as is shown in Fig. 6.9. In general, $\delta\mathbf{F}_s$ will be inclined with respect to the surface. The force $\delta\mathbf{F}_s$ can be resolved into three components, $\delta F_n$, $\delta F_1$, and $\delta F_2$, where $\delta F_n$ is normal to the area, $\delta A$, and $\delta F_1$ and $\delta F_2$ are parallel to the area and orthogonal to each other. The *normal stress*, $\sigma_n$, is defined as

$$\sigma_n = \lim_{\delta A\to 0}\frac{\delta F_n}{\delta A}$$




■ **FIGURE 6.9** **Components of force acting on an arbitrary differential area.**

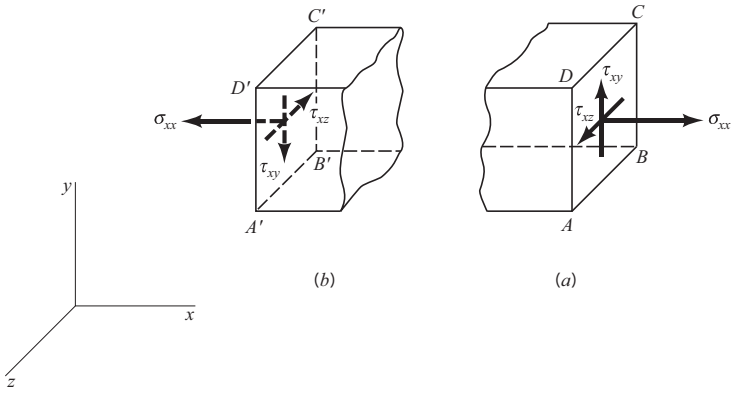



**FIGURE 6.10** **Double subscript notation for stresses.**

and the *shearing stresses* are defined as

$$\tau_1 = \lim_{\delta A \to 0} \frac{\delta F_1}{\delta A}$$

and

$$\tau_2 = \lim_{\delta A \to 0} \frac{\delta F_2}{\delta A}$$

We will use $\sigma$ for normal stresses and $\tau$ for shearing stresses. The intensity of the force per unit area at a point in a body can thus be characterized by a normal stress and two shearing stresses, if the orientation of the area is specified. For purposes of analysis it is usually convenient to reference the area to the coordinate system. For example, for the rectangular coordinate system shown in Fig. 6.10 we choose to consider the stresses acting on planes parallel to the coordinate planes. On the plane *ABCD* of Fig. 6.10*a*, which is parallel to the *y*–*z* plane, the normal stress is denoted $\sigma_{xx}$ and the shearing stresses are denoted as $\tau_{xy}$ and $\tau_{xz}$. To easily identify the particular stress component we use a double subscript notation. The first subscript indicates the direction of the *normal* to the plane on which the stress acts, and the second subscript indicates the direction of the stress. Thus, normal stresses have repeated subscripts, whereas the subscripts for the shearing stresses are always different.

It is also necessary to establish a sign convention for the stresses. We define the positive direction for the stress as the positive coordinate direction on the surfaces for which the outward normal is in the positive coordinate direction. This is the case illustrated in Fig. 6.10*a* where the outward normal to the area *ABCD* is in the positive *x* direction. The positive directions for $\sigma_{xx}$, $\tau_{xy}$, and $\tau_{xz}$ are as shown in Fig. 6.10*a*. If the outward normal points in the negative coordinate direction, as in Fig. 6.10*b* for the area $A'B'C'D'$, then the stresses are considered positive if directed in the negative coordinate directions. Thus, the stresses shown in Fig. 6.10*b* are considered to be positive when directed as shown. Note that positive normal stresses are tensile stresses; that is, they tend to "stretch" the material.

It should be emphasized that the state of stress at a point in a material is not completely defined by simply three components of a "stress vector." This follows, since any particular stress vector depends on the orientation of the plane passing through the point. However, it can be shown that the normal and shearing stresses acting on *any* plane passing through a point can be expressed in terms of the stresses acting on three orthogonal planes passing through the point (Ref. 2).

*Surface forces can be expressed in terms of the shear and normal stresses.*

We now can express the surface forces acting on a small cubical element of fluid in terms of the stresses acting on the faces of the element as shown in Fig. 6.11. It is expected that in general the stresses will vary from point to point within the flow field. Thus, through the use of Taylor series expansions we will express the stresses on the various faces in terms of the corresponding stresses at the center of the element of Fig. 6.11 and their gradients in the coordinate directions. For simplicity only the forces in the *x* direction are shown. Note that the stresses must be multiplied by the area on which they act to obtain the force. Summing all these forces in the *x* direction yields

$$\delta F_{sx} = \left( \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) \delta x\, \delta y\, \delta z \qquad \textbf{(6.48a)}$$

CMS Cepcor EX 1046 Page 187
CMS Cepcor v. Sandvik IPR2024-01380

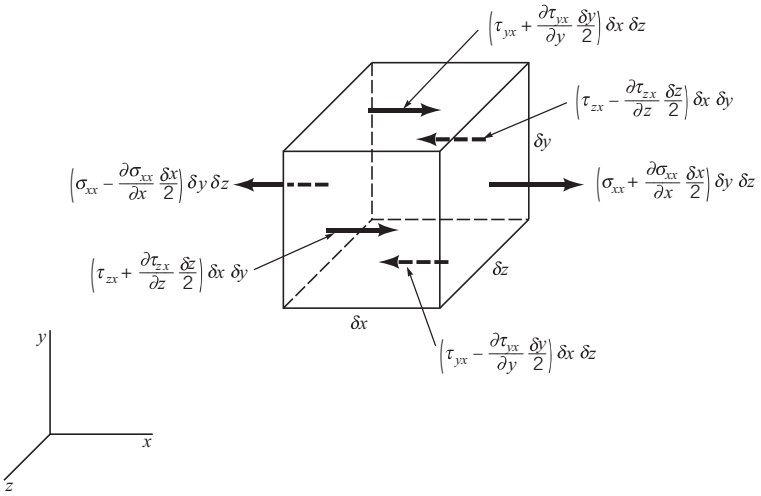



**FIGURE 6.11** **Surface forces in the $x$ direction acting on a fluid element.**

for the resultant surface force in the $x$ direction. In a similar manner the resultant surface forces in the $y$ and $z$ directions can be obtained and expressed as

$$\delta F_{sy} = \left(\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z}\right)\delta x\,\delta y\,\delta z \tag{6.48b}$$

$$\delta F_{sz} = \left(\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z}\right)\delta x\,\delta y\,\delta z \tag{6.48c}$$

The resultant surface force can now be expressed as

$$\delta \mathbf{F}_s = \delta F_{sx}\hat{\mathbf{i}} + \delta F_{sy}\hat{\mathbf{j}} + \delta F_{sz}\hat{\mathbf{k}} \tag{6.49}$$

and this force combined with the body force, $\delta \mathbf{F}_b$, yields the resultant force, $\delta \mathbf{F}$, acting on the differential mass, $\delta m$. That is, $\delta \mathbf{F} = \delta \mathbf{F}_s + \delta \mathbf{F}_b$.

### 6.3.2 Equations of Motion

The expressions for the body and surface forces can now be used in conjunction with Eq. 6.45 to develop the equations of motion. In component form Eq. 6.45 can be written as

$$\delta F_x = \delta m\, a_x$$

$$\delta F_y = \delta m\, a_y$$

$$\delta F_z = \delta m\, a_z$$

where $\delta m = \rho\,\delta x\,\delta y\,\delta z$, and the acceleration components are given by Eq. 6.3. It now follows (using Eqs. 6.47 and 6.48 for the forces on the element) that

*The motion of a fluid is governed by a set of nonlinear differential equations.*

$$\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} = \rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) \tag{6.50a}$$

$$\rho g_y + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} = \rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) \tag{6.50b}$$

$$\rho g_z + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_{zz}}{\partial z} = \rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) \tag{6.50c}$$

where the element volume $\delta x\,\delta y\,\delta z$ cancels out.

Equations 6.50 are the general differential equations of motion for a fluid. In fact, they are applicable to any continuum (solid or fluid) in motion or at rest. However, before we can use the equations to solve specific problems, some additional information about the stresses must be obtained.

Otherwise, we will have more unknowns (all of the stresses and velocities and the density) than equations. It should not be too surprising that the differential analysis of fluid motion is complicated. We are attempting to describe, in detail, complex fluid motion.

## 6.4 Inviscid Flow

As is discussed in Section 1.6, shearing stresses develop in a moving fluid because of the viscosity of the fluid. We know that for some common fluids, such as air and water, the viscosity is small, and therefore it seems reasonable to assume that under some circumstances we may be able to simply neglect the effect of viscosity (and thus shearing stresses). Flow fields in which the shearing stresses are assumed to be negligible are said to be *inviscid*, *nonviscous*, or *frictionless*. These terms are used interchangeably. As is discussed in Section 2.1, for fluids in which there are no shearing stresses the normal stress at a point is independent of direction—that is, $\sigma_{xx} = \sigma_{yy} = \sigma_{zz}$. In this instance we define the pressure, $p$, as the negative of the normal stress so that

$$-p = \sigma_{xx} = \sigma_{yy} = \sigma_{zz}$$

The negative sign is used so that a *compressive* normal stress (which is what we expect in a fluid) will give a *positive* value for $p$.

In Chapter 3 the inviscid flow concept was used in the development of the Bernoulli equation, and numerous applications of this important equation were considered. In this section we will again consider the Bernoulli equation and will show how it can be derived from the general equations of motion for inviscid flow.

### 6.4.1 Euler's Equations of Motion

For an inviscid flow in which all the shearing stresses are zero, and the normal stresses are replaced by $-p$, the general equations of motion (Eqs. 6.50) reduce to

$$\rho g_x - \frac{\partial p}{\partial x} = \rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) \tag{6.51a}$$

$$\rho g_y - \frac{\partial p}{\partial y} = \rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) \tag{6.51b}$$

$$\rho g_z - \frac{\partial p}{\partial z} = \rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) \tag{6.51c}$$

These equations are commonly referred to as ***Euler's equations of motion,*** named in honor of Leonhard Euler (1707–1783), a famous Swiss mathematician who pioneered work on the relationship between pressure and flow. In vector notation Euler's equations can be expressed as

*Euler's equations of motion apply to an inviscid flow field.*

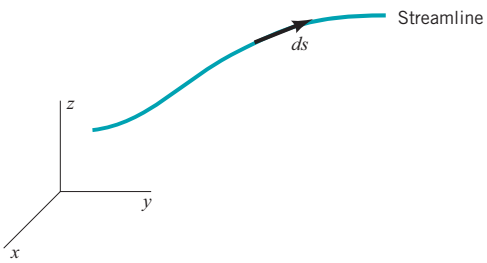
$$\rho\mathbf{g} - \nabla p = \rho\left[\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V}\cdot\nabla)\mathbf{V}\right] \tag{6.52}$$

Although Eqs. 6.51 are considerably simpler than the general equations of motion, Eqs. 6.50, they are still not amenable to a general analytical solution that would allow us to determine the pressure and velocity at all points within an inviscid flow field. The main difficulty arises from the nonlinear velocity terms ($u\,\partial u/\partial x$, $v\,\partial u/\partial y$, etc.), which appear in the convective acceleration. Because of these terms, Euler's equations are nonlinear partial differential equations for which we do not have a general method of solving. However, under some circumstances we can use them to obtain useful information about inviscid flow fields. For example, as shown in the following section we can integrate Eq. 6.52 to obtain a relationship (the Bernoulli equation) between elevation, pressure, and velocity along a streamline.

### 6.4.2 The Bernoulli Equation

In Section 3.2 the Bernoulli equation was derived by a direct application of Newton's second law to a fluid particle moving along a streamline. In this section we will again derive this important

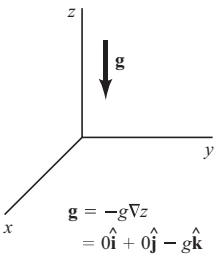
CMS Cepcor EX 1046 Page 189
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 6.12** **The notation for differential length along a streamline.**

equation, starting from Euler's equations. Of course, we should obtain the same result since Euler's equations simply represent a statement of Newton's second law expressed in a general form that is useful for flow problems and maintains the restriction of zero viscosity. We will restrict our attention to steady flow so Euler's equation in vector form becomes

$$\rho\mathbf{g} - \nabla p = \rho(\mathbf{V}\cdot\nabla)\mathbf{V} \tag{6.53}$$




We wish to integrate this differential equation along some arbitrary streamline (Fig. 6.12) and select the coordinate system with the $z$ axis vertical (with "up" being positive) so that, as shown by the figure in the margin, the acceleration of gravity vector can be expressed as

$$\mathbf{g} = -g\nabla z$$

where $g$ is the magnitude of the acceleration of gravity vector. Also, it will be convenient to use the vector identity

$$(\mathbf{V}\cdot\nabla)\mathbf{V} = \tfrac{1}{2}\nabla(\mathbf{V}\cdot\mathbf{V}) - \mathbf{V}\times(\nabla\times\mathbf{V})$$

Equation 6.53 can now be written in the form

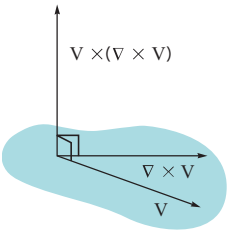
$$-\rho g\nabla z - \nabla p = \frac{\rho}{2}\nabla(\mathbf{V}\cdot\mathbf{V}) - \rho\mathbf{V}\times(\nabla\times\mathbf{V})$$

and this equation can be rearranged to yield

$$\frac{\nabla p}{\rho} + \frac{1}{2}\nabla(V^2) + g\nabla z = \mathbf{V}\times(\nabla\times\mathbf{V})$$

We next take the dot product of each term with a differential length $d\mathbf{s}$ along a streamline (Fig. 6.12). Thus,

$$\frac{\nabla p}{\rho}\cdot d\mathbf{s} + \frac{1}{2}\nabla(V^2)\cdot d\mathbf{s} + g\nabla z\cdot d\mathbf{s} = [\mathbf{V}\times(\nabla\times\mathbf{V})]\cdot d\mathbf{s} \tag{6.54}$$




Since $d\mathbf{s}$ has a direction along the streamline, the vectors $d\mathbf{s}$ and $\mathbf{V}$ are parallel. However, as shown by the figure in the margin, the vector $\mathbf{V}\times(\nabla\times\mathbf{V})$ is perpendicular to $\mathbf{V}$ (why?), so it follows that

$$[\mathbf{V}\times(\nabla\times\mathbf{V})]\cdot d\mathbf{s} = 0$$

Recall also that the dot product of the gradient of a scalar and a differential length gives the differential change in the scalar in the direction of the differential length. That is, with $d\mathbf{s} = dx\,\mathbf{i} + dy\,\mathbf{j} + dz\,\mathbf{k}$ we can write $\nabla p\cdot d\mathbf{s} = (\partial p/\partial x)\,dx + (\partial p/\partial y)dy + (\partial p/\partial z)dz = dp$. Thus, Eq. 6.54 becomes

$$\frac{dp}{\rho} + \frac{1}{2}d(V^2) + g\,dz = 0 \tag{6.55}$$

*Euler's equations can be arranged to give the relationship among pressure, velocity, and elevation for inviscid fluids*

where the change in $p$, $V$, and $z$ is along the streamline. Equation 6.55 can now be integrated to give

$$\int\frac{dp}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.56}$$

which indicates that the sum of the three terms on the left side of the equation must remain a constant along a given streamline. Equation 6.56 is valid for both compressible and incompressible

inviscid flows, but for compressible fluids the variation in $\rho$ with $p$ must be specified before the first term in Eq. 6.56 can be evaluated.

For inviscid, incompressible fluids (commonly called ***ideal fluids***) Eq. 6.56 can be written as

$$\frac{p}{\rho} + \frac{V^2}{2} + gz = \text{constant along a streamline} \tag{6.57}$$

and this equation is the ***Bernoulli equation*** used extensively in Chapter 3. It is often convenient to write Eq. 6.57 between two points (1) and (2) along a streamline and to express the equation in the "head" form by dividing each term by $g$ so that

$$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + z_2 \tag{6.58}$$

It should be again emphasized that the Bernoulli equation, as expressed by Eqs. 6.57 and 6.58, is restricted to the following:

- inviscid flow
- steady flow
- incompressible flow
- flow along a streamline

You may want to go back and review some of the examples in Chapter 3 that illustrate the use of the Bernoulli equation.

### 6.4.3 Irrotational Flow

*The vorticity is zero in an irrotational flow field.*

If we make one additional assumption—that the flow is *irrotational*—the analysis of inviscid flow problems is further simplified. Recall from Section 6.1.3 that the rotation of a fluid element is equal to $\frac{1}{2}(\nabla \times \mathbf{V})$, and an irrotational flow field is one for which $\nabla \times \mathbf{V} = 0$ (i.e., the curl of velocity is zero). Since the vorticity, $\zeta$, is defined as $\nabla \times \mathbf{V}$, it also follows that in an irrotational flow field the vorticity is zero. The concept of irrotationality may seem to be a rather strange condition for a flow field. Why would a flow field be irrotational? To answer this question we note that if $\frac{1}{2}(\nabla \times \mathbf{V}) = 0$, then each of the components of this vector, as are given by Eqs. 6.12, 6.13, and 6.14, must be equal to zero. Since these components include the various velocity gradients in the flow field, the condition of irrotationality imposes specific relationships among these velocity gradients. For example, for rotation about the $z$ axis to be zero, it follows from Eq. 6.12 that

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = 0$$

and, therefore,

$$\frac{\partial v}{\partial x} = \frac{\partial u}{\partial y} \tag{6.59}$$

Similarly from Eqs. 6.13 and 6.14

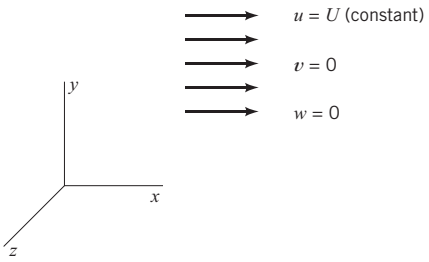
$$\frac{\partial w}{\partial y} = \frac{\partial v}{\partial z} \tag{6.60}$$

$$\frac{\partial u}{\partial z} = \frac{\partial w}{\partial x} \tag{6.61}$$

A general flow field would not satisfy these three equations. However, a uniform flow as is illustrated in Fig. 6.13 does. Since $u = U$ (a constant), $v = 0$, and $w = 0$, it follows that Eqs. 6.59, 6.60, and 6.61 are all satisfied. Therefore, a uniform flow field (in which there are no velocity gradients) is certainly an example of an irrotational flow.

Uniform flows by themselves are not very interesting. However, many interesting and important flow problems include uniform flow in some part of the flow field. Two examples are

CMS Cepcor EX 1046 Page 191
CMS Cepcor v. Sandvik IPR2024-01380

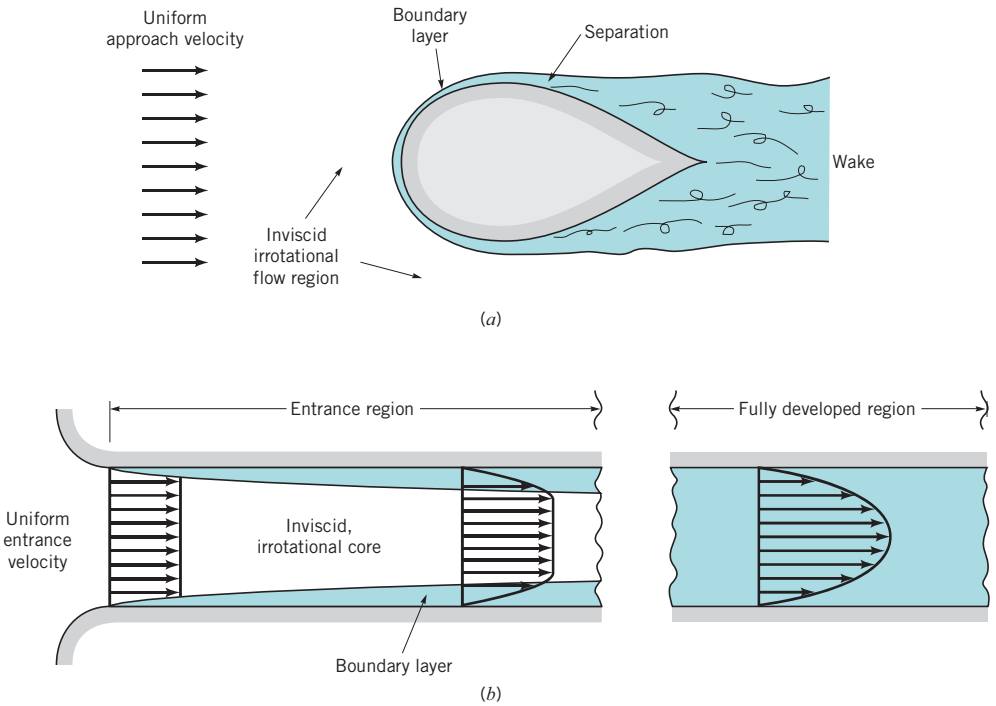



■ **FIGURE 6.13** **Uniform flow in the $x$ direction.**

shown in Fig. 6.14. In Fig. 6.14*a* a solid body is placed in a uniform stream of fluid. Far away from the body the flow remains uniform, and in this far region the flow is irrotational. In Fig. 6.14*b*, flow from a large reservoir enters a pipe through a streamlined entrance where the velocity distribution is essentially uniform. Thus, at the entrance the flow is irrotational.

*Flow fields involving real fluids often include both regions of negligible shearing stresses and regions of significant shearing stresses.*

For an inviscid fluid there are no shearing stresses—the only forces acting on a fluid element are its weight and pressure forces. Since the weight acts through the element center of gravity, and the pressure acts in a direction normal to the element surface, neither of these forces can cause the element to rotate. Therefore, for an inviscid fluid, if some part of the flow field is irrotational, the fluid elements emanating from this region will not take on any rotation as they progress through the flow field. This phenomenon is illustrated in Fig. 6.14*a* in which fluid elements flowing far away from the body have irrotational motion, and as they flow around the body the motion remains irrotational except very near the boundary. Near the boundary the velocity changes rapidly from zero at the boundary (no-slip condition) to some relatively large value in a short distance from the boundary. This rapid change in velocity gives rise to a large velocity gradient normal to the boundary and produces significant shearing stresses, even though the viscosity is small. Of course if we had a truly inviscid fluid, the fluid would simply "slide" past the boundary and the flow would be irrotational everywhere. But this is not the case for real fluids, so we will typically have a layer (usually very thin) near any fixed surface in a moving stream in which shearing stresses are not negligible. This layer is called the *boundary layer*. Outside the boundary layer the flow can be treated as an irrotational flow. Another possible consequence of the boundary layer is that the main stream may "separate" from the surface and form a *wake* downstream from the body. (See the




■ **FIGURE 6.14** **Various regions of flow: (*a*) around bodies; (*b*) through channels.**

photographs at the beginning of Chapters 7, 9, and 11.) The wake would include a region of slow, perhaps randomly moving fluid. To completely analyze this type of problem it is necessary to consider both the inviscid, irrotational flow outside the boundary layer, and the viscous, rotational flow within the boundary layer and to somehow "match" these two regions. This type of analysis is considered in Chapter 9.

As is illustrated in Fig. 6.14*b*, the flow in the entrance to a pipe may be uniform (if the entrance is streamlined), and thus will be irrotational. In the central core of the pipe the flow remains irrotational for some distance. However, a boundary layer will develop along the wall and grow in thickness until it fills the pipe. Thus, for this type of internal flow there will be an *entrance region* in which there is a central irrotational core, followed by a so-called *fully developed region* in which viscous forces are dominant. The concept of irrotationality is completely invalid in the fully developed region. This type of internal flow problem is considered in detail in Chapter 8.

The two preceding examples are intended to illustrate the possible applicability of irrotational flow to some "real fluid" flow problems and to indicate some limitations of the irrotationality concept. We proceed to develop some useful equations based on the assumptions of inviscid, incompressible, irrotational flow, with the admonition to use caution when applying the equations.

### 6.4.4 The Bernoulli Equation for Irrotational Flow

In the development of the Bernoulli equation in Section 6.4.2, Eq. 6.54 was integrated along a streamline. This restriction was imposed so the right side of the equation could be set equal to zero; that is,

$$[\mathbf{V} \times (\nabla \times \mathbf{V})] \cdot d\mathbf{s} = 0$$

(since $d\mathbf{s}$ is parallel to $\mathbf{V}$). However, for irrotational flow, $\nabla \times \mathbf{V} = 0$, so the right side of Eq. 6.54 is zero regardless of the direction of $d\mathbf{s}$. We can now follow the same procedure used to obtain Eq. 6.55, where the differential changes $dp$, $d(V^2)$, and $dz$ can be taken in any direction. Integration of Eq. 6.55 again yields

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{6.62}$$

where for irrotational flow the constant is the same throughout the flow field. Thus, for incompressible, irrotational flow the Bernoulli equation can be written as

$$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + z_2 \tag{6.63}$$

*The Bernoulli equation can be applied between any two points in an irrotational flow field.*

between *any two points in the flow field*. Equation 6.63 is exactly the same form as Eq. 6.58 but is not limited to application along a streamline. However, Eq. 6.63 is restricted to

- inviscid flow
- steady flow
- incompressible flow
- irrotational flow

It may be worthwhile to review the use and misuse of the Bernoulli equation for rotational flow as is illustrated in Example 3.18.

### 6.4.5 The Velocity Potential

For an irrotational flow the velocity gradients are related through Eqs. 6.59, 6.60, and 6.61. It follows that in this case the velocity components can be expressed in terms of a scalar function $\phi(x, y, z, t)$ as

$$u = \frac{\partial \phi}{\partial x} \qquad v = \frac{\partial \phi}{\partial y} \qquad w = \frac{\partial \phi}{\partial z} \tag{6.64}$$

CMS Cepcor EX 1046 Page 193
CMS Cepcor v. Sandvik IPR2024-01380

where $\phi$ is called the ***velocity potential.*** Direct substitution of these expressions for the velocity components into Eqs. 6.59, 6.60, and 6.61 will verify that a velocity field defined by Eqs. 6.64 is indeed irrotational. In vector form, Eqs. 6.64 can be written as

$$\boxed{\mathbf{V} = \nabla\phi} \tag{6.65}$$

so that for an irrotational flow the velocity is expressible as the gradient of a scalar function $\phi$.

The velocity potential is a consequence of the irrotationality of the flow field, whereas the stream function is a consequence of conservation of mass (see Section 6.2.3). It is to be noted, however, that the velocity potential can be defined for a general three-dimensional flow, whereas the stream function is restricted to two-dimensional flows.

For an incompressible fluid we know from conservation of mass that

$$\nabla \cdot \mathbf{V} = 0$$

*Inviscid, incompressible, irrotational flow fields are governed by Laplace's equation and are called potential flows.*

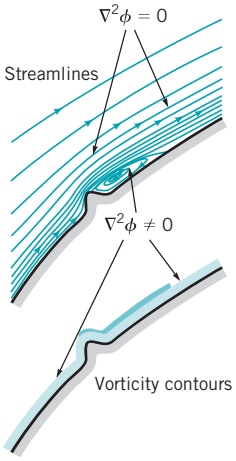
and therefore for incompressible, irrotational flow (with $\mathbf{V} = \nabla\phi$) it follows that

$$\nabla^2\phi = 0 \tag{6.66}$$

where $\nabla^2(\ ) = \nabla \cdot \nabla(\ )$ is the *Laplacian operator*. In Cartesian coordinates

$$\frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} + \frac{\partial^2\phi}{\partial z^2} = 0$$

This differential equation arises in many different areas of engineering and physics and is called *Laplace's equation*. Thus, inviscid, incompressible, irrotational flow fields are governed by Laplace's equation. This type of flow is commonly called a ***potential flow.*** To complete the mathematical formulation of a given problem, boundary conditions have to be specified. These are usually velocities specified on the boundaries of the flow field of interest. It follows that if the potential function can be determined, then the velocity at all points in the flow field can be determined from Eq. 6.64, and the pressure at all points can be determined from the Bernoulli equation (Eq. 6.63). Although the concept of the velocity potential is applicable to both steady and unsteady flow, we will confine our attention to steady flow.




Potential flows, governed by Eqs. 6.64 and 6.66, are irrotational flows. That is, the vorticity is zero throughout. If vorticity is present (e.g., boundary layer, wake), then the flow cannot be described by Laplace's equation. The figure in the margin illustrates a flow in which the vorticity is not zero in two regions—the separated region behind the bump and the boundary layer next to the solid surface. This is discussed in detail in Chapter 9.

For some problems it will be convenient to use cylindrical coordinates, $r$, $\theta$, and $z$. In this coordinate system the gradient operator is

$$\nabla(\ ) = \frac{\partial(\ )}{\partial r}\hat{\mathbf{e}}_r + \frac{1}{r}\frac{\partial(\ )}{\partial\theta}\hat{\mathbf{e}}_\theta + \frac{\partial(\ )}{\partial z}\hat{\mathbf{e}}_z \tag{6.67}$$

so that

$$\nabla\phi = \frac{\partial\phi}{\partial r}\hat{\mathbf{e}}_r + \frac{1}{r}\frac{\partial\phi}{\partial\theta}\hat{\mathbf{e}}_\theta + \frac{\partial\phi}{\partial z}\hat{\mathbf{e}}_z \tag{6.68}$$

where $\phi = \phi(r, \theta, z)$. Since

$$\mathbf{V} = v_r\hat{\mathbf{e}}_r + v_\theta\hat{\mathbf{e}}_\theta + v_z\hat{\mathbf{e}}_z \tag{6.69}$$

it follows for an irrotational flow (with $\mathbf{V} = \nabla\phi$)

$$v_r = \frac{\partial\phi}{\partial r} \qquad v_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} \qquad v_z = \frac{\partial\phi}{\partial z} \tag{6.70}$$

Also, Laplace's equation in cylindrical coordinates is

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial\phi}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2\phi}{\partial\theta^2} + \frac{\partial^2\phi}{\partial z^2} = 0 \tag{6.71}$$

CMS Cepcor EX 1046 Page 194
CMS Cepcor v. Sandvik IPR2024-01380

## EXAMPLE 6.4 Velocity Potential and Inviscid Flow Pressure

**GIVEN** The two-dimensional flow of a nonviscous, incompressible fluid in the vicinity of the 90° corner of Fig. E6.4*a* is described by the stream function

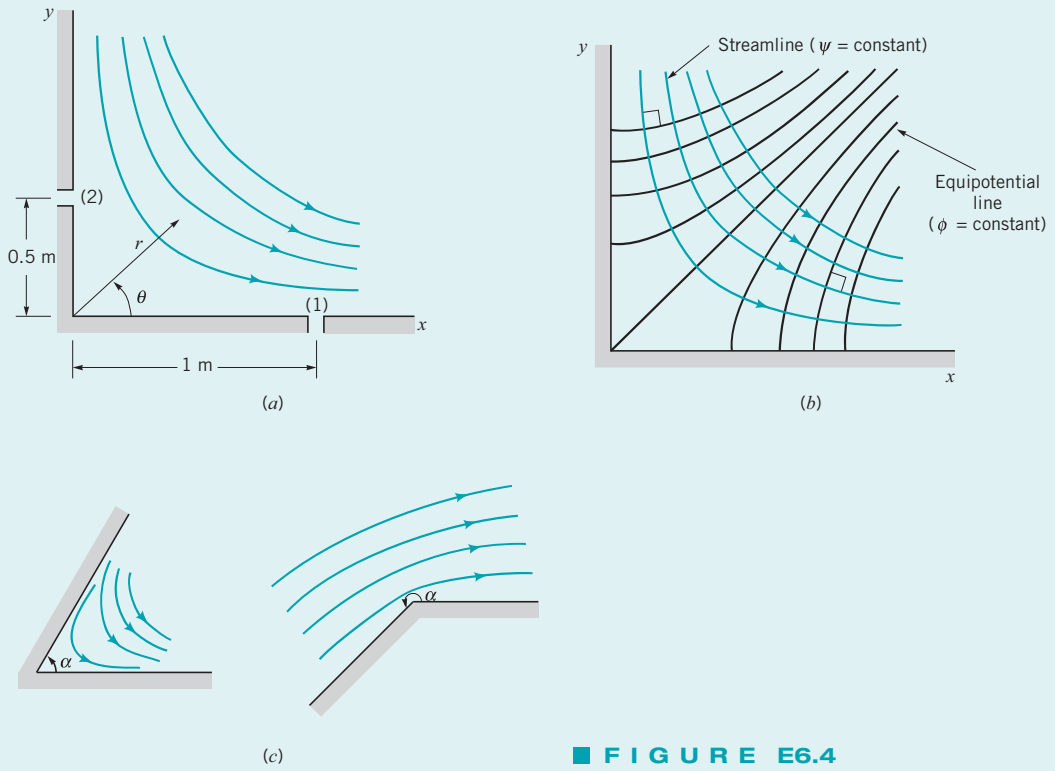
$$\psi = 2r^2 \sin 2\theta$$

where $\psi$ has units of $m^2/s$ when $r$ is in meters. Assume the fluid density is $10^3$ kg/m$^3$ and the $x$–$y$ plane is horizontal—that is, there is no difference in elevation between points (1) and (2).

**FIND**

**(a)** Determine, if possible, the corresponding velocity potential.

**(b)** If the pressure at point (1) on the wall is 30 kPa, what is the pressure at point (2)?




■ FIGURE E6.4

## SOLUTION

**(a)** The radial and tangential velocity components can be obtained from the stream function as (see Eq. 6.42)

$$v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = 4r \cos 2\theta$$

and

$$v_\theta = -\frac{\partial \psi}{\partial r} = -4r \sin 2\theta$$

Since

$$v_r = \frac{\partial \phi}{\partial r}$$

it follows that

$$\frac{\partial \phi}{\partial r} = 4r \cos 2\theta$$

and therefore by integration

$$\phi = 2r^2 \cos 2\theta + f_1(\theta) \qquad (1)$$

where $f_1(\theta)$ is an arbitrary function of $\theta$. Similarly

$$v_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = -4r \sin 2\theta$$

and integration yields

$$\phi = 2r^2 \cos 2\theta + f_2(r) \qquad (2)$$

where $f_2(r)$ is an arbitrary function of $r$. To satisfy both Eqs. 1 and 2, the velocity potential must have the form

$$\phi = 2r^2 \cos 2\theta + C \qquad \textbf{(Ans)}$$

where $C$ is an arbitrary constant. As is the case for stream functions, the specific value of $C$ is not important, and it is customary to let $C = 0$ so that the velocity potential for this corner flow is

$$\phi = 2r^2 \cos 2\theta \qquad \textbf{(Ans)}$$

CMS Cepcor EX 1046 Page 195
CMS Cepcor v. Sandvik IPR2024-01380

**COMMENT** In the statement of this problem it was implied by the wording "if possible" that we might not be able to find a corresponding velocity potential. The reason for this concern is that we can always define a stream function for two-dimensional flow, but the flow must be *irrotational* if there is a corresponding velocity potential. Thus, the fact that we were able to determine a velocity potential means that the flow is irrotational. Several streamlines and lines of constant $\phi$ are plotted in Fig. E6.4*b*. These two sets of lines are *orthogonal*. The reason why streamlines and lines of constant $\phi$ are always orthogonal is explained in Section 6.5.

**(b)** Since we have an irrotational flow of a nonviscous, incompressible fluid, the Bernoulli equation can be applied between any two points. Thus, between points (1) and (2) with no elevation change

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

or

$$p_2 = p_1 + \frac{\rho}{2}(V_1^2 - V_2^2) \tag{3}$$

Since

$$V^2 = v_r^2 + v_\theta^2$$

it follows that for any point within the flow field

$$\begin{aligned} V^2 &= (4r\cos 2\theta)^2 + (-4r\sin 2\theta)^2 \\ &= 16r^2(\cos^2 2\theta + \sin^2 2\theta) \\ &= 16r^2 \end{aligned}$$

This result indicates that the square of the velocity at any point depends only on the radial distance, $r$, to the point. Note that the constant, 16, has units of $s^{-2}$. Thus,

$$V_1^2 = (16\ s^{-2})(1\ m)^2 = 16\ m^2/s^2$$

and

$$V_2^2 = (16\ s^{-2})(0.5\ m)^2 = 4\ m^2/s^2$$

Substitution of these velocities into Eq. 3 gives

$$\begin{aligned} p_2 &= 30 \times 10^3\ N/m^2 + \frac{10^3\ kg/m^3}{2}(16\ m^2/s^2 - 4\ m^2/s^2) \\ &= 36\ kPa \end{aligned} \quad \textbf{(Ans)}$$

**COMMENT** The stream function used in this example could also be expressed in Cartesian coordinates as

$$\psi = 2r^2 \sin 2\theta = 4r^2 \sin\theta\cos\theta$$

or

$$\psi = 4xy$$

since $x = r\cos\theta$ and $y = r\sin\theta$. However, in the cylindrical polar form the results can be generalized to describe flow in the vicinity of a corner of angle $\alpha$ (see Fig. E6.4*c*) with the equations

$$\psi = Ar^{\pi/\alpha}\sin\frac{\pi\theta}{\alpha}$$

and

$$\phi = Ar^{\pi/\alpha}\cos\frac{\pi\theta}{\alpha}$$

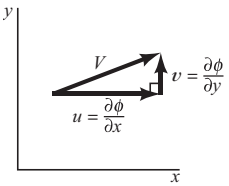
where $A$ is a constant.

## 6.5 Some Basic, Plane Potential Flows

*For potential flow, basic solutions can be simply added to obtain more complicated solutions.*

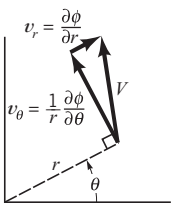
A major advantage of Laplace's equation is that it is a linear partial differential equation. Since it is linear, various solutions can be added to obtain other solutions—that is, if $\phi_1(x, y, z)$ and $\phi_2(x, y, z)$ are two solutions to Laplace's equation, then $\phi_3 = \phi_1 + \phi_2$ is also a solution. The practical implication of this result is that if we have certain basic solutions we can combine them to obtain more complicated and interesting solutions. In this section several basic velocity potentials, which describe some relatively simple flows, will be determined. In the next section these basic potentials will be combined to represent complicated flows.

For simplicity, only plane (two-dimensional) flows will be considered. In this case, by using Cartesian coordinates




$$u = \frac{\partial\phi}{\partial x} \qquad v = \frac{\partial\phi}{\partial y} \tag{6.72}$$

or by using cylindrical coordinates




$$v_r = \frac{\partial\phi}{\partial r} \qquad v_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} \tag{6.73}$$

as shown by the figure in the margin. Since we can define a stream function for plane flow, we can also let

$$u = \frac{\partial\psi}{\partial y} \qquad v = -\frac{\partial\psi}{\partial x} \tag{6.74}$$

or

$$v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \qquad v_\theta = -\frac{\partial \psi}{\partial r} \tag{6.75}$$

where the stream function was previously defined in Eqs. 6.37 and 6.42. We know that by defining the velocities in terms of the stream function, conservation of mass is identically satisfied. If we now impose the condition of irrotationality, it follows from Eq. 6.59 that

$$\frac{\partial u}{\partial y} = \frac{\partial v}{\partial x}$$

and in terms of the stream function

$$\frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial y}\right) = \frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial x}\right)$$

or

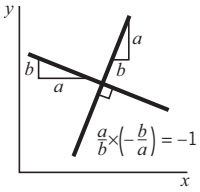
$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

Thus, for a plane irrotational flow we can use either the velocity potential or the stream function—both must satisfy Laplace's equation in two dimensions. It is apparent from these results that the velocity potential and the stream function are somehow related. We have previously shown that lines of constant $\psi$ are streamlines; that is,

$$\left.\frac{dy}{dx}\right|_{\text{along } \psi = \text{constant}} = \frac{v}{u} \tag{6.76}$$

The change in $\phi$ as we move from one point $(x, y)$ to a nearby point $(x + dx, y + dy)$ is given by the relationship

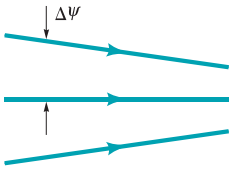
$$d\phi = \frac{\partial \phi}{\partial x}dx + \frac{\partial \phi}{\partial y}dy = u\,dx + v\,dy$$



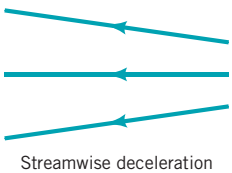

Along a line of constant $\phi$ we have $d\phi = 0$ so that

$$\left.\frac{dy}{dx}\right|_{\text{along } \phi = \text{constant}} = -\frac{u}{v} \tag{6.77}$$

A comparison of Eqs. 6.76 and 6.77 shows that lines of constant $\phi$ (called ***equipotential lines***) are orthogonal to lines of constant $\psi$ (streamlines) at all points where they intersect. (Recall that two lines are orthogonal if the product of their slopes is $-1$, as illustrated by the figure in the margin.) For any potential flow field a "***flow net***" can be drawn that consists of a family of streamlines and equipotential lines. The flow net is useful in visualizing flow patterns and can be used to obtain graphical solutions by sketching in streamlines and equipotential lines and adjusting the lines until the lines are approximately orthogonal at all points where they intersect. An example of a flow net is shown in Fig. 6.15. Velocities can be estimated from the flow net, since the velocity is inversely proportional to the streamline spacing, as shown by the figure in the margin. Thus, for example, from Fig. 6.15 we can see that the velocity near the inside corner will be higher than the velocity along the outer part of the bend. (See the photographs at the beginning of Chapters 3 and 6.)




Streamwise acceleration

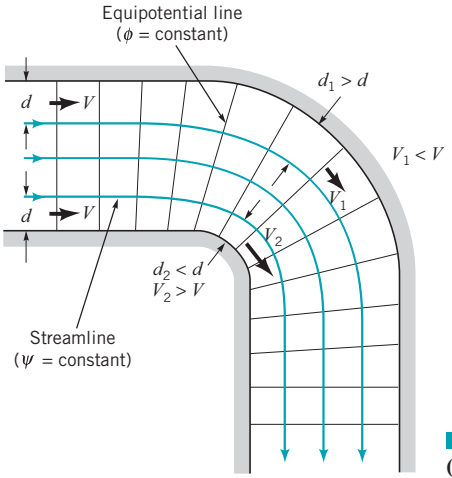


Streamwise deceleration

### 6.5.1 Uniform Flow

The simplest plane flow is one for which the streamlines are all straight and parallel, and the magnitude of the velocity is constant. This type of flow is called a ***uniform flow.*** For example, consider a uniform flow in the positive $x$ direction as is illustrated in Fig. 6.16*a*. In this instance, $u = U$ and $v = 0$, and in terms of the velocity potential

$$\frac{\partial \phi}{\partial x} = U \qquad \frac{\partial \phi}{\partial y} = 0$$




■ **FIGURE 6.15** **Flow net for a 90° bend. (From Ref. 3, used by permission.)**

These two equations can be integrated to yield

$$\phi = Ux + C$$

where $C$ is an arbitrary constant, which can be set equal to zero. Thus, for a uniform flow in the positive $x$ direction

$$\phi = Ux \tag{6.78}$$

The corresponding stream function can be obtained in a similar manner, since

$$\frac{\partial \psi}{\partial y} = U \qquad \frac{\partial \psi}{\partial x} = 0$$

and, therefore,

$$\psi = Uy \tag{6.79}$$

These results can be generalized to provide the velocity potential and stream function for a uniform flow at an angle $\alpha$ with the $x$ axis, as in Fig. 6.16*b*. For this case

$$\phi = U(x \cos \alpha + y \sin \alpha) \tag{6.80}$$

and

$$\psi = U(y \cos \alpha - x \sin \alpha) \tag{6.81}$$

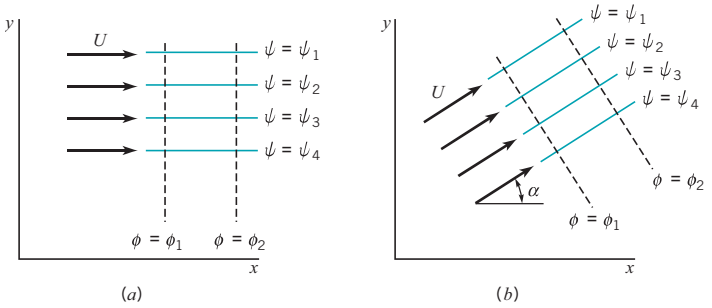
### 6.5.2 Source and Sink

Consider a fluid flowing radially outward from a line through the origin perpendicular to the $x$–$y$ plane as is shown in Fig. 6.17. Let $m$ be the volume rate of flow emanating from the line (per unit length), and therefore to satisfy conservation of mass

$$(2\pi r)v_r = m$$

or

$$v_r = \frac{m}{2\pi r}$$




■ **FIGURE 6.16**
**Uniform flow: (*a*) in the $x$ direction; (*b*) in an arbitrary direction, $\alpha$.**

CMS Cepcor EX 1046 Page 198
CMS Cepcor v. Sandvik IPR2024-01380

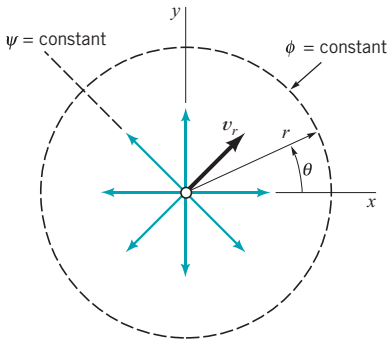



**FIGURE 6.17** The streamline pattern for a source.

*A source or sink represents a purely radial flow.*

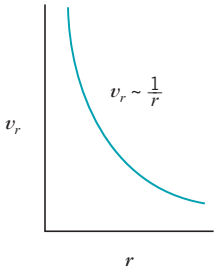
Also, since the flow is a purely radial flow, $v_\theta = 0$, the corresponding velocity potential can be obtained by integrating the equations

$$\frac{\partial \phi}{\partial r} = \frac{m}{2\pi r} \qquad \frac{1}{r}\frac{\partial \phi}{\partial \theta} = 0$$

It follows that

$$\phi = \frac{m}{2\pi} \ln r \tag{6.82}$$

If $m$ is positive, the flow is radially outward, and the flow is considered to be a ***source*** flow. If $m$ is negative, the flow is toward the origin, and the flow is considered to be a ***sink*** flow. The flowrate, $m$, is the *strength* of the source or sink.




As shown by the figure in the margin, at the origin where $r = 0$ the velocity becomes infinite, which is of course physically impossible. Thus, sources and sinks do not really exist in real flow fields, and the line representing the source or sink is a mathematical *singularity* in the flow field. However, some real flows can be approximated at points away from the origin by using sources or sinks. Also, the velocity potential representing this hypothetical flow can be combined with other basic velocity potentials to approximately describe some real flow fields. This idea is further discussed in Section 6.6.

The stream function for the source can be obtained by integrating the relationships

$$v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = \frac{m}{2\pi r} \qquad v_\theta = -\frac{\partial \psi}{\partial r} = 0$$

to yield

$$\psi = \frac{m}{2\pi}\theta \tag{6.83}$$

It is apparent from Eq. 6.83 that the streamlines (lines of $\psi$ = constant) are radial lines, and from Eq. 6.82 the equipotential lines (lines of $\phi$ = constant) are concentric circles centered at the origin.

## EXAMPLE 6.5 Potential Flow—Sink

**GIVEN** A nonviscous, incompressible fluid flows between wedge-shaped walls into a small opening as shown in Fig. E6.5. The velocity potential (in $ft^2/s$), which approximately describes this flow is

$$\phi = -2 \ln r$$

**FIND** Determine the volume rate of flow (per unit length) into the opening.

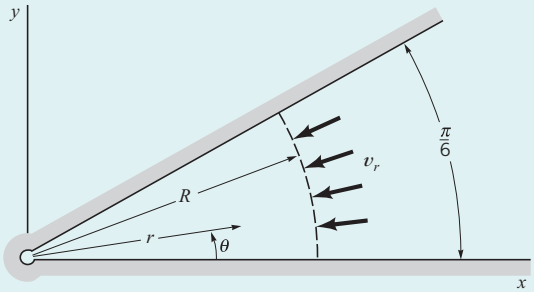



**FIGURE E6.5**

## SOLUTION

The components of velocity are

$$v_r = \frac{\partial \phi}{\partial r} = -\frac{2}{r} \qquad v_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = 0$$

which indicates we have a purely radial flow. The flowrate per unit width, $q$, crossing the arc of length $R\pi/6$ can thus be obtained by integrating the expression

$$q = \int_0^{\pi/6} v_r R\, d\theta = -\int_0^{\pi/6} \left(\frac{2}{R}\right) R\, d\theta$$

$$= -\frac{\pi}{3} = -1.05 \text{ ft}^2/\text{s} \qquad \textbf{(Ans)}$$

**COMMENT** Note that the radius $R$ is arbitrary since the flowrate crossing any curve between the two walls must be the same. The negative sign indicates that the flow is toward the opening, that is, in the negative radial direction.

### 6.5.3 Vortex

We next consider a flow field in which the streamlines are concentric circles—that is, we interchange the velocity potential and stream function for the source. Thus, let

$$\phi = K\theta \tag{6.84}$$

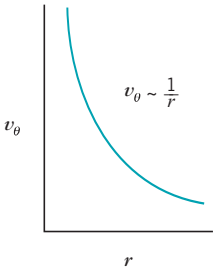
*A vortex represents a flow in which the streamlines are concentric circles.*

and

$$\psi = -K \ln r \tag{6.85}$$

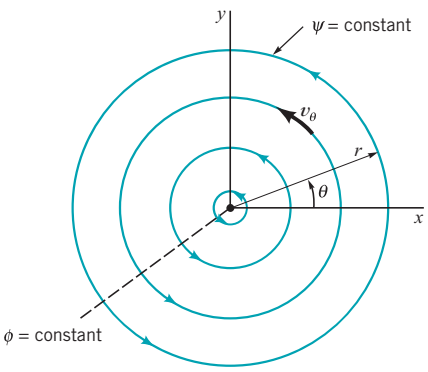
where $K$ is a constant. In this case the streamlines are concentric circles as are illustrated in Fig. 6.18, with $v_r = 0$ and

$$v_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = -\frac{\partial \psi}{\partial r} = \frac{K}{r} \tag{6.86}$$




This result indicates that the tangential velocity varies inversely with the distance from the origin, as shown by the figure in the margin, with a singularity occurring at $r = 0$ (where the velocity becomes infinite).

It may seem strange that this ***vortex*** motion is irrotational (and it is since the flow field is described by a velocity potential). However, it must be recalled that rotation refers to the orientation of a fluid element and not the path followed by the element. Thus, for an irrotational vortex, if a pair of small sticks were placed in the flow field at location $A$, as indicated in Fig. 6.19$a$, the sticks would rotate as they move to location $B$. One of the sticks, the one that is aligned along the streamline, would follow a circular path and rotate in a counterclockwise




■ **FIGURE 6.18** **The streamline pattern for a vortex.**

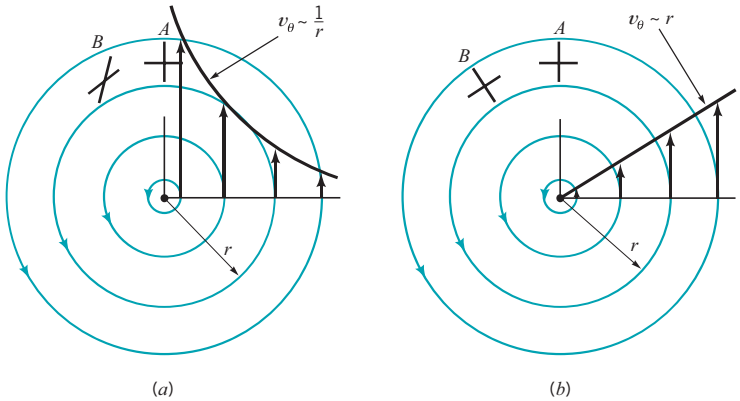



**FIGURE 6.19** **Motion of fluid element from *A* to *B*: (*a*) for irrotational (free) vortex; (*b*) for rotational (forced) vortex.**

direction. The other stick would rotate in a clockwise direction due to the nature of the flow field—that is, the part of the stick nearest the origin moves faster than the opposite end. Although both sticks are rotating, the average angular velocity of the two sticks is zero since the flow is irrotational.

If the fluid were rotating as a rigid body, such that $v_\theta = K_1 r$ where $K_1$ is a constant, then sticks similarly placed in the flow field would rotate as is illustrated in Fig. 6.19*b*. This type of vortex motion is *rotational* and cannot be described with a velocity potential. The rotational vortex is commonly called a *forced vortex*, whereas the irrotational vortex is usually called a *free vortex*. The swirling motion of the water as it drains from a bathtub is similar to that of a free vortex, whereas the motion of a liquid contained in a tank that is rotated about its axis with angular velocity $\omega$ corresponds to a forced vortex.

*Vortex motion can be either rotational or irrotational.*

A *combined vortex* is one with a forced vortex as a central core and a velocity distribution corresponding to that of a free vortex outside the core. Thus, for a combined vortex

$$v_\theta = \omega r \qquad r \leq r_0 \tag{6.87}$$

and

$$v_\theta = \frac{K}{r} \qquad r > r_0 \tag{6.88}$$

where $K$ and $\omega$ are constants and $r_0$ corresponds to the radius of the central core. The pressure distribution in both the free and forced vortex was previously considered in Example 3.3. (See Fig. E6.6*a* for an approximation of this type of flow.)

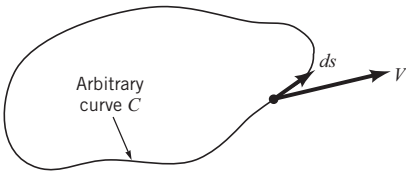
## Fluids in the News

**Some hurricane facts** One of the most interesting, yet potentially devastating, naturally occurring fluid flow phenomenan is a hurricane. Broadly speaking a hurricane is a rotating mass of air circulating around a low pressure central core. In some respects the motion is similar to that of a *free vortex*. The Caribbean and Gulf of Mexico experience the most hurricanes, with the official hurricane season being from June 1 to November 30. Hurricanes are usually 300 to 400 miles wide and are structured around a central eye in which the air is relatively calm. The eye is surrounded by an eye wall which is the region of strongest winds and precipitation. As one goes from the eye wall to the eye the wind speeds decrease sharply and within the eye the air is relatively calm and clear of clouds. However, in the eye the pressure is at a minimum and may be 10% less than standard atmospheric pressure. This low pressure creates strong downdrafts of dry air from above. Hurricanes are classified into five categories based on their wind speeds:

Category one—74–95 mph
Category two—96–110 mph
Category three—111–130 mph
Category four—131–155 mph
Category five—greater than 155 mph.

(See Problem 6.58.)

CMS Cepcor EX 1046 Page 201
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 6.20** **The notation for determining circulation around closed curve *C*.**

A mathematical concept commonly associated with vortex motion is that of ***circulation.*** The circulation, $\Gamma$, is defined as the line integral of the tangential component of the velocity taken around a closed curve in the flow field. In equation form, $\Gamma$ can be expressed as

$$\Gamma = \oint_C \mathbf{V} \cdot d\mathbf{s} \tag{6.89}$$

where the integral sign means that the integration is taken around a closed curve, $C$, in the counterclockwise direction, and $d\mathbf{s}$ is a differential length along the curve as is illustrated in Fig. 6.20. For an irrotational flow, $\mathbf{V} = \nabla\phi$ so that $\mathbf{V} \cdot d\mathbf{s} = \nabla\phi \cdot d\mathbf{s} = d\phi$ and, therefore,

$$\Gamma = \oint_C d\phi = 0$$

This result indicates that for an irrotational flow the circulation will generally be zero. (Chapter 9 has further discussion of circulation in real flows.) However, if there are singularities enclosed within the curve the circulation may not be zero. For example, for the free vortex with $v_\theta = K/r$ the circulation around the circular path of radius $r$ shown in Fig. 6.21 is

$$\Gamma = \int_0^{2\pi} \frac{K}{r}(r\, d\theta) = 2\pi K$$

*The numerical value of the circulation may depend on the particular closed path considered.*

which shows that the circulation is nonzero and the constant $K = \Gamma/2\pi$. However, for irrotational flows the circulation around any path that does not include a singular point will be zero. This can be easily confirmed for the closed path $ABCD$ of Fig. 6.21 by evaluating the circulation around that path.

The velocity potential and stream function for the free vortex are commonly expressed in terms of the circulation as

$$\phi = \frac{\Gamma}{2\pi}\theta \tag{6.90}$$



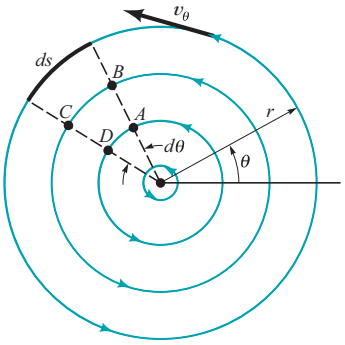
*V6.4 Vortex in a beaker*



and

$$\psi = -\frac{\Gamma}{2\pi}\ln r \tag{6.91}$$

The concept of circulation is often useful when evaluating the forces developed on bodies immersed in moving fluids. This application will be considered in Section 6.6.3.




■ **FIGURE 6.21** **Circulation around various paths in a free vortex.**

## EXAMPLE 6.6 Potential Flow—Free Vortex

**GIVEN** A liquid drains from a large tank through a small opening as illustrated in Fig. E6.6*a*. A vortex forms whose velocity distribution away from the tank opening can be approximated as that of a free vortex having a velocity potential

$$\phi = \frac{\Gamma}{2\pi}\theta$$

**FIND** Determine an expression relating the surface shape to the strength of the vortex as specified by the circulation $\Gamma$.

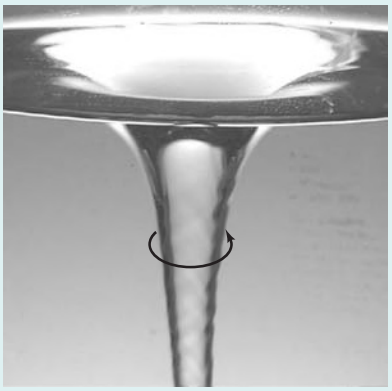


FIGURE E6.6*a*

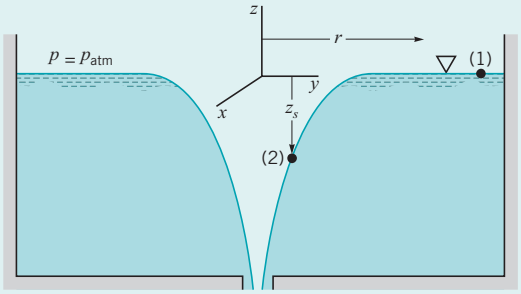



FIGURE E6.6*b*

### SOLUTION

Since the free vortex represents an irrotational flow field, the Bernoulli equation

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

can be written between any two points. If the points are selected at the free surface, $p_1 = p_2 = 0$, so that

$$\frac{V_1^2}{2g} = z_s + \frac{V_2^2}{2g} \qquad (1)$$

where the free surface elevation, $z_s$, is measured relative to a datum passing through point (1) as shown in Fig. E6.6*b*.

The velocity is given by the equation

$$v_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = \frac{\Gamma}{2\pi r}$$

We note that far from the origin at point (1), $V_1 = v_\theta \approx 0$ so that Eq. 1 becomes

$$z_s = -\frac{\Gamma^2}{8\pi^2 r^2 g} \qquad \text{(Ans)}$$

which is the desired equation for the surface profile.

**COMMENT** The negative sign indicates that the surface falls as the origin is approached as shown in Fig. E6.6. This solution is not valid very near the origin since the predicted velocity becomes excessively large as the origin is approached.

### 6.5.4 Doublet

*A doublet is formed by an appropriate source–sink pair.*

The final, basic potential flow to be considered is one that is formed by combining a source and sink in a special way. Consider the equal strength, source–sink pair of Fig. 6.22. The combined stream function for the pair is

$$\psi = -\frac{m}{2\pi}(\theta_1 - \theta_2)$$

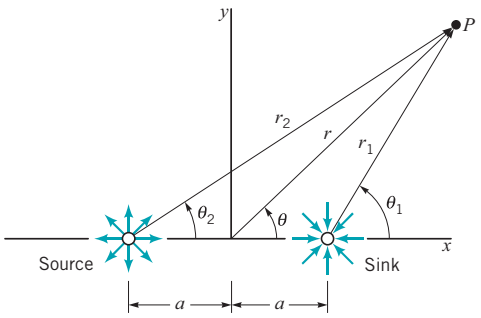



FIGURE 6.22 **The combination of a source and sink of equal strength located along the *x* axis.**

which can be rewritten as

$$\tan\left(-\frac{2\pi\psi}{m}\right) = \tan(\theta_1 - \theta_2) = \frac{\tan\theta_1 - \tan\theta_2}{1 + \tan\theta_1 \tan\theta_2} \tag{6.92}$$

From Fig. 6.22 it follows that

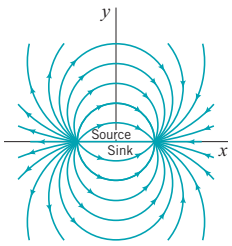
$$\tan\theta_1 = \frac{r\sin\theta}{r\cos\theta - a}$$

and

$$\tan\theta_2 = \frac{r\sin\theta}{r\cos\theta + a}$$

These results substituted into Eq. 6.92 give

$$\tan\left(-\frac{2\pi\psi}{m}\right) = \frac{2ar\sin\theta}{r^2 - a^2}$$




so that

$$\psi = -\frac{m}{2\pi}\tan^{-1}\left(\frac{2ar\sin\theta}{r^2 - a^2}\right) \tag{6.93}$$

The figure in the margin shows typical streamlines for this flow. For small values of the distance $a$

$$\psi = -\frac{m}{2\pi}\frac{2ar\sin\theta}{r^2 - a^2} = -\frac{mar\sin\theta}{\pi(r^2 - a^2)} \tag{6.94}$$

since the tangent of an angle approaches the value of the angle for small angles.

*A doublet is formed by letting a source and sink approach one another.*

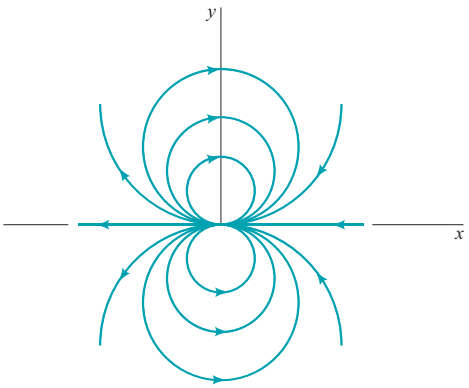
The so-called ***doublet*** is formed by letting the source and sink approach one another ($a \to 0$) while increasing the strength $m$ ($m \to \infty$) so that the product $ma/\pi$ remains constant. In this case, since $r/(r^2 - a^2) \to 1/r$, Eq. 6.94 reduces to

$$\psi = -\frac{K\sin\theta}{r} \tag{6.95}$$

where $K$, a constant equal to $ma/\pi$, is called the *strength* of the doublet. The corresponding velocity potential for the doublet is

$$\phi = \frac{K\cos\theta}{r} \tag{6.96}$$

Plots of lines of constant $\psi$ reveal that the streamlines for a doublet are circles through the origin tangent to the $x$ axis as shown in Fig. 6.23. Just as sources and sinks are not physically realistic entities, neither are doublets. However, the doublet when combined with other basic potential flows




■ **FIGURE 6.23** **Streamlines for a doublet.**

**TABLE 6.1**
**Summary of Basic, Plane Potential Flows**

| Description of Flow Field | Velocity Potential | Stream Function | Velocity Components[a] |
|---|---|---|---|
| Uniform flow at angle $\alpha$ with the $x$ axis (see Fig. 6.16*b*) | $\phi = U(x\cos\alpha + y\sin\alpha)$ | $\psi = U(y\cos\alpha - x\sin\alpha)$ | $u = U\cos\alpha$<br>$v = U\sin\alpha$ |
| Source or sink (see Fig. 6.17)<br>$m > 0$ source<br>$m < 0$ sink | $\phi = \frac{m}{2\pi}\ln r$ | $\psi = \frac{m}{2\pi}\theta$ | $v_r = \frac{m}{2\pi r}$<br>$v_\theta = 0$ |
| Free vortex (see Fig. 6.18)<br>$\Gamma > 0$ counterclockwise motion<br>$\Gamma < 0$ clockwise motion | $\phi = \frac{\Gamma}{2\pi}\theta$ | $\psi = -\frac{\Gamma}{2\pi}\ln r$ | $v_r = 0$<br>$v_\theta = \frac{\Gamma}{2\pi r}$ |
| Doublet (see Fig. 6.23) | $\phi = \frac{K\cos\theta}{r}$ | $\psi = -\frac{K\sin\theta}{r}$ | $v_r = -\frac{K\cos\theta}{r^2}$<br>$v_\theta = -\frac{K\sin\theta}{r^2}$ |

[a]Velocity components are related to the velocity potential and stream function through the relationships:

$$u = \frac{\partial\phi}{\partial x} = \frac{\partial\psi}{\partial y} \qquad v = \frac{\partial\phi}{\partial y} = -\frac{\partial\psi}{\partial x} \qquad v_r = \frac{\partial\phi}{\partial r} = \frac{1}{r}\frac{\partial\psi}{\partial\theta} \qquad v_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} = -\frac{\partial\psi}{\partial r}.$$

provides a useful representation of some flow fields of practical interest. For example, we will determine in Section 6.6.3 that the combination of a uniform flow and a doublet can be used to represent the flow around a circular cylinder. Table 6.1 provides a summary of the pertinent equations for the basic, plane potential flows considered in the preceding sections.

## 6.6 Superposition of Basic, Plane Potential Flows

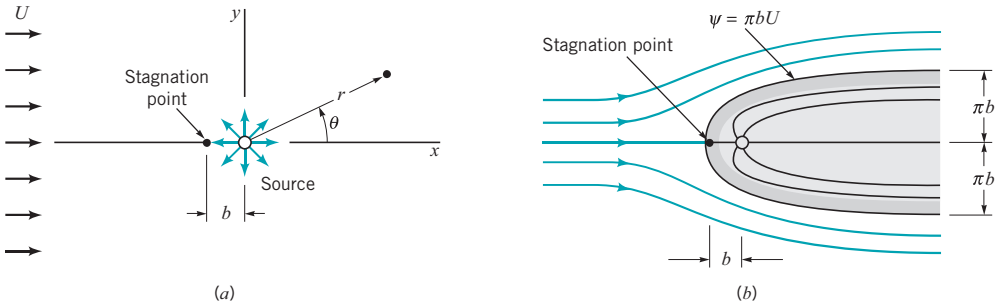
As was discussed in the previous section, potential flows are governed by Laplace's equation, which is a linear partial differential equation. It therefore follows that the various basic velocity potentials and stream functions can be combined to form new potentials and stream functions. (Why is this true?) Whether such combinations yield useful results remains to be seen. It is to be noted that *any streamline in an inviscid flow field can be considered as a solid boundary*, since the conditions along a solid boundary and a streamline are the same—that is, there is no flow through the boundary or the streamline. Thus, if we can combine some of the basic velocity potentials or stream functions to yield a streamline that corresponds to a particular body shape of interest, that combination can be used to describe in detail the flow around that body. This method of solving some interesting flow problems, commonly called the ***method of superposition***, is illustrated in the following three sections.

### 6.6.1 Source in a Uniform Stream—Half-Body

*Flow around a half-body is obtained by the addition of a source to a uniform flow.*

Consider the superposition of a source and a uniform flow as shown in Fig. 6.24*a*. The resulting stream function is

$$\begin{aligned}\psi &= \psi_{\text{uniform flow}} + \psi_{\text{source}} \\ &= Ur\sin\theta + \frac{m}{2\pi}\theta\end{aligned} \tag{6.97}$$




**■ FIGURE 6.24** **The flow around a half-body: (*a*) superposition of a source and a uniform flow; (*b*) replacement of streamline $\psi = \pi bU$ with solid boundary to form half-body.**

and the corresponding velocity potential is

$$\phi = Ur\cos\theta + \frac{m}{2\pi}\ln r \tag{6.98}$$



*V6.5 Half-body*



It is clear that at some point along the negative $x$ axis the velocity due to the source will just cancel that due to the uniform flow and a stagnation point will be created. For the source alone

$$v_r = \frac{m}{2\pi r}$$

so that the stagnation point will occur at $x = -b$ where

$$U = \frac{m}{2\pi b}$$

or

$$b = \frac{m}{2\pi U} \tag{6.99}$$

The value of the stream function at the stagnation point can be obtained by evaluating $\psi$ at $r = b$ and $\theta = \pi$, which yields from Eq. 6.97

$$\psi_{\text{stagnation}} = \frac{m}{2}$$

Since $m/2 = \pi bU$ (from Eq. 6.99) it follows that the equation of the streamline passing through the stagnation point is

$$\pi bU = Ur\sin\theta + bU\theta$$

or

$$r = \frac{b(\pi - \theta)}{\sin\theta} \tag{6.100}$$

where $\theta$ can vary between 0 and $2\pi$. A plot of this streamline is shown in Fig. 6.24*b*. If we replace this streamline with a solid boundary, as indicated in the figure, then it is clear that this combination of a uniform flow and a source can be used to describe the flow around a streamlined body placed in a uniform stream. The body is open at the downstream end, and thus is called a ***half-body***. Other streamlines in the flow field can be obtained by setting $\psi$ = constant in Eq. 6.97 and plotting the resulting equation. A number of these streamlines are shown in Fig. 6.24*b*. Although the streamlines inside the body are shown, they are actually of no interest in this case, since we are concerned with the flow field outside the body. It should be noted that the singularity in the flow field (the source) occurs inside the body, and there are no singularities in the flow field of interest (outside the body).

*For inviscid flow, a streamline can be replaced by a solid boundary.*

The width of the half-body asymptotically approaches $2\pi b$. This follows from Eq. 6.100, which can be written as

$$y = b(\pi - \theta)$$

so that as $\theta \to 0$ or $\theta \to 2\pi$ the half-width approaches $\pm b\pi$. With the stream function (or velocity potential) known, the velocity components at any point can be obtained. For the half-body, using the stream function given by Eq. 6.97,

$$v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = U\cos\theta + \frac{m}{2\pi r}$$

and

$$v_\theta = -\frac{\partial \psi}{\partial r} = -U\sin\theta$$

Thus, the square of the magnitude of the velocity, $V$, at any point is

$$V^2 = v_r^2 + v_\theta^2 = U^2 + \frac{Um\cos\theta}{\pi r} + \left(\frac{m}{2\pi r}\right)^2$$

and since $b = m/2\pi U$

$$V^2 = U^2\left(1 + 2\frac{b}{r}\cos\theta + \frac{b^2}{r^2}\right) \tag{6.101}$$

With the velocity known, the pressure at any point can be determined from the Bernoulli equation, which can be written between any two points in the flow field since the flow is irrotational. Thus, applying the Bernoulli equation between a point far from the body, where the pressure is $p_0$ and the velocity is $U$, and some arbitrary point with pressure $p$ and velocity $V$, it follows that

$$p_0 + \tfrac{1}{2}\rho U^2 = p + \tfrac{1}{2}\rho V^2 \tag{6.102}$$

where elevation changes have been neglected. Equation 6.101 can now be substituted into Eq. 6.102 to obtain the pressure at any point in terms of the reference pressure, $p_0$, and the upstream velocity, $U$.

*For a potential flow the fluid is allowed to slip past a fixed solid boundary.*

This relatively simple potential flow provides some useful information about the flow around the front part of a streamlined body, such as a bridge pier or strut placed in a uniform stream. An important point to be noted is that the velocity tangent to the surface of the body is not zero; that is, the fluid "slips" by the boundary. This result is a consequence of neglecting viscosity, the fluid property that causes real fluids to stick to the boundary, thus creating a "no-slip" condition. All potential flows differ from the flow of real fluids in this respect and do not accurately represent the velocity very near the boundary. However, outside this very thin boundary layer the velocity distribution will generally correspond to that predicted by potential flow theory if flow separation does not occur. (See Section 9.2.6.) Also, the pressure distribution along the surface will closely approximate that predicted from the potential flow theory, since the boundary layer is thin and there is little opportunity for the pressure to vary through the thin layer. In fact, as discussed in more detail in Chapter 9, the pressure distribution obtained from potential flow theory is used in conjunction with viscous flow theory to determine the nature of flow within the boundary layer.

## EXAMPLE 6.7 Potential Flow—Half-body

**GIVEN** A 40 mi/hr wind blows toward a hill arising from a plain that can be approximated with the top section of a half-body as illustrated in Fig. E6.7*a*. The height of the hill approaches 200 ft as shown. Assume an air density of 0.00238 slugs/ft$^3$.

**FIND**

**(a)** What is the magnitude of the air velocity at a point on the hill directly above the origin [point (2)]?

**(b)** What is the elevation of point (2) above the plain and what is the difference in pressure between point (1) on the plain far from the hill and point (2)?

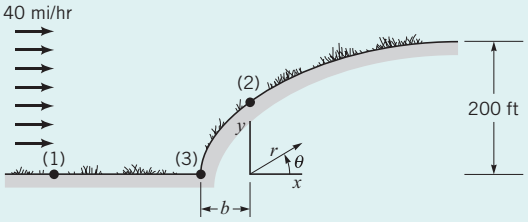



FIGURE E6.7*a*

## SOLUTION

**(a)** The velocity is given by Eq. 6.101 as

$$V^2 = U^2\left(1 + 2\frac{b}{r}\cos\theta + \frac{b^2}{r^2}\right)$$

At point (2), $\theta = \pi/2$, and since this point is on the surface (Eq. 6.100)

$$r = \frac{b(\pi - \theta)}{\sin\theta} = \frac{\pi b}{2} \qquad \textbf{(1)}$$

Thus,

$$V_2^2 = U^2\left[1 + \frac{b^2}{(\pi b/2)^2}\right]$$
$$= U^2\left(1 + \frac{4}{\pi^2}\right)$$

and the magnitude of the velocity at (2) for a 40 mi/hr approaching wind is

$$V_2 = \left(1 + \frac{4}{\pi^2}\right)^{1/2}(40\text{ mi/hr}) = 47.4\text{ mi/hr} \qquad \textbf{(Ans)}$$

**(b)** The elevation at (2) above the plain is given by Eq. 1 as

$$y_2 = \frac{\pi b}{2}$$

Since the height of the hill approaches 200 ft and this height is equal to $\pi b$, it follows that

$$y_2 = \frac{200\text{ ft}}{2} = 100\text{ ft} \qquad \textbf{(Ans)}$$

From the Bernoulli equation (with the $y$ axis the vertical axis)

$$\frac{p_1}{\gamma} + \frac{V_1^2}{2g} + y_1 = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + y_2$$

so that

$$p_1 - p_2 = \frac{\rho}{2}(V_2^2 - V_1^2) + \gamma(y_2 - y_1)$$

and with

$$V_1 = (40\text{ mi/hr})\left(\frac{5280\text{ ft/mi}}{3600\text{ s/hr}}\right) = 58.7\text{ ft/s}$$

and

$$V_2 = (47.4\text{ mi/hr})\left(\frac{5280\text{ ft/mi}}{3600\text{ s/hr}}\right) = 69.5\text{ ft/s}$$

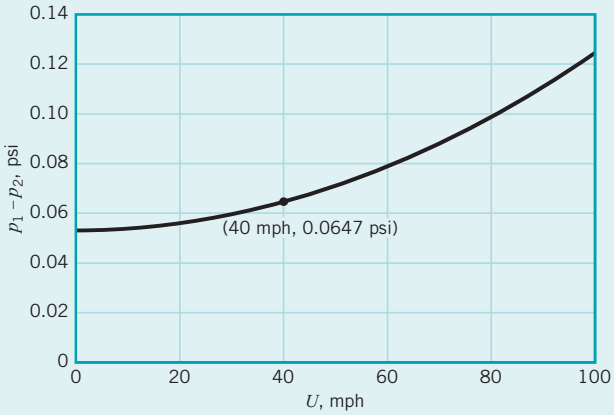
it follows that

$$p_1 - p_2 = \frac{(0.00238\text{ slugs/ft}^3)}{2}[(69.5\text{ ft/s})^2 - (58.7\text{ ft/s})^2]$$
$$+ (0.00238\text{ slugs/ft}^3)(32.2\text{ ft/s}^2)(100\text{ ft} - 0\text{ ft})$$
$$= 9.31\text{ lb/ft}^2 = 0.0647\text{ psi} \qquad \textbf{(Ans)}$$

**COMMENTS** This result indicates that the pressure on the hill at point (2) is slightly lower than the pressure on the plain at some distance from the base of the hill with a 0.0533 psi difference due to the elevation increase and a 0.0114 psi difference due to the velocity increase.

By repeating the calculations for various values of the upstream wind speed, $U$, the results shown in Fig. E6.7*b* are obtained. Note that as the wind speed increases, the pressure difference increases from the calm conditions of $p_1 - p_2 = 0.0533$ psi.

The maximum velocity along the hill surface does not occur at point (2) but farther up the hill at $\theta = 63°$. At this point $V_{surface} = 1.26U$. The minimum velocity ($V = 0$) and maximum pressure occur at point (3), the stagnation point.




■ FIGURE E6.7*b*

### 6.6.2 Rankine Ovals

The half-body described in the previous section is a body that is "open" at one end. To study the flow around a closed body, a source and a sink of equal strength can be combined with a uniform flow as shown in Fig. 6.25*a*. The stream function for this combination is

$$\psi = Ur\sin\theta - \frac{m}{2\pi}(\theta_1 - \theta_2) \qquad \textbf{(6.103)}$$

and the velocity potential is

$$\phi = Ur\cos\theta - \frac{m}{2\pi}(\ln r_1 - \ln r_2) \qquad \textbf{(6.104)}$$

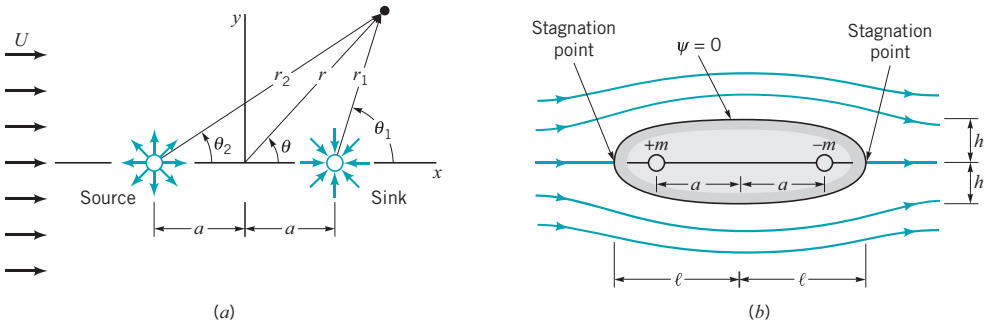



**FIGURE 6.25** **The flow around a Rankine oval: (*a*) superposition of source–sink pair and a uniform flow; (*b*) replacement of streamline $\psi = 0$ with solid boundary to form Rankine oval.**

As discussed in Section 6.5.4, the stream function for the source–sink pair can be expressed as in Eq. 6.93 and, therefore, Eq. 6.103 can also be written as

$$\psi = Ur\sin\theta - \frac{m}{2\pi}\tan^{-1}\left(\frac{2ar\sin\theta}{r^2 - a^2}\right)$$

or

$$\psi = Uy - \frac{m}{2\pi}\tan^{-1}\left(\frac{2ay}{x^2 + y^2 - a^2}\right) \tag{6.105}$$

*Rankine ovals are formed by combining a source and sink with a uniform flow.*

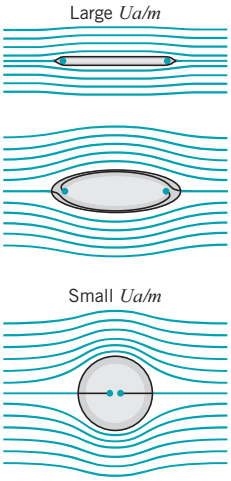
The corresponding streamlines for this flow field are obtained by setting $\psi$ = constant. If several of these streamlines are plotted, it will be discovered that the streamline $\psi = 0$ forms a closed body as is illustrated in Fig. 6.25*b*. We can think of this streamline as forming the surface of a body of length $2\ell$ and width $2h$ placed in a uniform stream. The streamlines inside the body are of no practical interest and are not shown. Note that since the body is closed, all of the flow emanating from the source flows into the sink. These bodies have an oval shape and are termed ***Rankine ovals.***

Stagnation points occur at the upstream and downstream ends of the body as are indicated in Fig. 6.25*b*. These points can be located by determining where along the $x$ axis the velocity is zero. The stagnation points correspond to the points where the uniform velocity, the source velocity, and the sink velocity all combine to give a zero velocity. The locations of the stagnation points depend on the value of $a$, $m$, and $U$. The body half-length, $\ell$ (the value of $|x|$ that gives $\mathbf{V} = 0$ when $y = 0$), can be expressed as

$$\ell = \left(\frac{ma}{\pi U} + a^2\right)^{1/2} \tag{6.106}$$

or

$$\frac{\ell}{a} = \left(\frac{m}{\pi U a} + 1\right)^{1/2} \tag{6.107}$$




The body half-width, $h$, can be obtained by determining the value of $y$ where the $y$ axis intersects the $\psi = 0$ streamline. Thus, from Eq. 6.105 with $\psi = 0$, $x = 0$, and $y = h$, it follows that

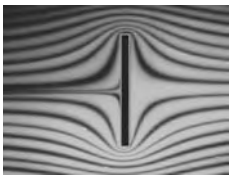
$$h = \frac{h^2 - a^2}{2a}\tan\frac{2\pi Uh}{m} \tag{6.108}$$

or

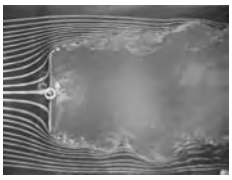
$$\frac{h}{a} = \frac{1}{2}\left[\left(\frac{h}{a}\right)^2 - 1\right]\tan\left[2\left(\frac{\pi Ua}{m}\right)\frac{h}{a}\right] \tag{6.109}$$

Equations 6.107 and 6.109 show that both $\ell/a$ and $h/a$ are functions of the dimensionless parameter, $\pi Ua/m$. Although for a given value of $Ua/m$ the corresponding value of $\ell/a$ can be determined directly from Eq. 6.107, $h/a$ must be determined by a trial and error solution of Eq. 6.109.

A large variety of body shapes with different length to width ratios can be obtained by using different values of $Ua/m$, as shown by the figure in the margin. As this parameter becomes large, flow



Potential Flow



Viscous Flow

around a long slender body is described, whereas for small values of the parameter, flow around a more blunt shape is obtained. Downstream from the point of maximum body width the surface pressure increases with distance along the surface. This condition (called an adverse pressure gradient) typically leads to separation of the flow from the surface, resulting in a large low pressure wake on the downstream side of the body. Separation is not predicted by potential theory (which simply indicates a symmetrical flow). This is illustrated by the figure in the margin for an extreme blunt shape. Therefore, the potential solution for the Rankine ovals will give a reasonable approximation of the velocity outside the thin, viscous boundary layer and the pressure distribution on the front part of the body only.

### 6.6.3 Flow around a Circular Cylinder

As was noted in the previous section, when the distance between the source–sink pair approaches zero, the shape of the Rankine oval becomes more blunt and in fact approaches a circular shape. Since the doublet described in Section 6.5.4 was developed by letting a source–sink pair approach one another, it might be expected that a uniform flow in the positive $x$ direction combined with a doublet could be used to represent flow around a circular cylinder. This combination gives for the stream function

*A doublet combined with a uniform flow can be used to represent flow around a circular cylinder.*

$$\psi = Ur \sin\theta - \frac{K \sin\theta}{r} \tag{6.110}$$

and for the velocity potential

$$\phi = Ur \cos\theta + \frac{K \cos\theta}{r} \tag{6.111}$$

In order for the stream function to represent flow around a circular cylinder it is necessary that $\psi$ = constant for $r = a$, where $a$ is the radius of the cylinder. Since Eq. 6.110 can be written as

$$\psi = \left(U - \frac{K}{r^2}\right) r \sin\theta$$

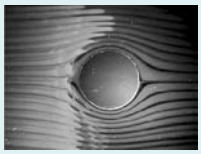
it follows that $\psi = 0$ for $r = a$ if

$$U - \frac{K}{a^2} = 0$$

which indicates that the doublet strength, $K$, must be equal to $Ua^2$. Thus, the stream function for flow around a circular cylinder can be expressed as



*V6.6 Circular cylinder*

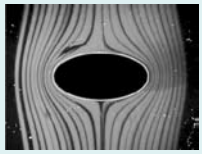


$$\psi = Ur\left(1 - \frac{a^2}{r^2}\right) \sin\theta \tag{6.112}$$

and the corresponding velocity potential is



*V6.7 Ellipse*



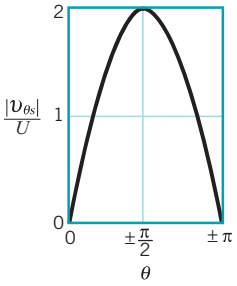
$$\phi = Ur\left(1 + \frac{a^2}{r^2}\right) \cos\theta \tag{6.113}$$

A sketch of the streamlines for this flow field is shown in Fig. 6.26.

The velocity components can be obtained from either Eq. 6.112 or 6.113 as

$$v_r = \frac{\partial\phi}{\partial r} = \frac{1}{r}\frac{\partial\psi}{\partial\theta} = U\left(1 - \frac{a^2}{r^2}\right)\cos\theta \tag{6.114}$$

and




$$v_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} = -\frac{\partial\psi}{\partial r} = -U\left(1 + \frac{a^2}{r^2}\right)\sin\theta \tag{6.115}$$

On the surface of the cylinder ($r = a$) it follows from Eq. 6.114 and 6.115 that $v_r = 0$ and

$$v_{\theta s} = -2U \sin\theta$$

As shown by the figure in the margin, the maximum velocity occurs at the top and bottom of the cylinder ($\theta = \pm\pi/2$) and has a magnitude of twice the upstream velocity, $U$. As we move away from the cylinder along the ray $\theta = \pi/2$ the velocity varies, as is illustrated in Fig. 6.26.

CMS Cepcor EX 1046 Page 210
CMS Cepcor v. Sandvik IPR2024-01380

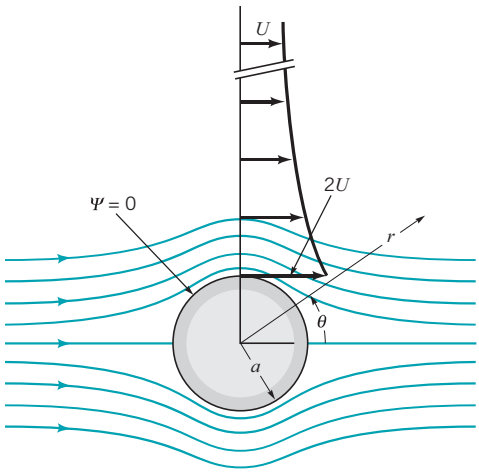



**FIGURE 6.26** The flow around a circular cylinder.

*The pressure distribution on the cylinder surface is obtained from the Bernoulli equation.*

The pressure distribution on the cylinder surface is obtained from the Bernoulli equation written from a point far from the cylinder where the pressure is $p_0$ and the velocity is $U$ so that

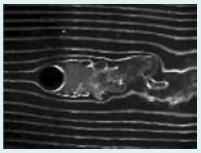
$$p_0 + \tfrac{1}{2}\rho U^2 = p_s + \tfrac{1}{2}\rho v_{\theta s}^2$$

where $p_s$ is the surface pressure. Elevation changes are neglected. Since $v_{\theta s} = -2U \sin\theta$, the surface pressure can be expressed as

$$p_s = p_0 + \tfrac{1}{2}\rho U^2(1 - 4\sin^2\theta) \tag{6.116}$$



*V6.8 Circular cylinder with separation*



A comparison of this theoretical, symmetrical pressure distribution expressed in dimensionless form with a typical measured distribution is shown in Fig. 6.27. This figure clearly reveals that only on the upstream part of the cylinder is there approximate agreement between the potential flow and the experimental results. Because of the viscous boundary layer that develops on the cylinder, the main flow separates from the surface of the cylinder, leading to the large difference between the theoretical, frictionless fluid solution and the experimental results on the downstream side of the cylinder (see Chapter 9).

The resultant force (per unit length) developed on the cylinder can be determined by integrating the pressure over the surface. From Fig. 6.28 it can be seen that

$$F_x = -\int_0^{2\pi} p_s \cos\theta \, a \, d\theta \tag{6.117}$$

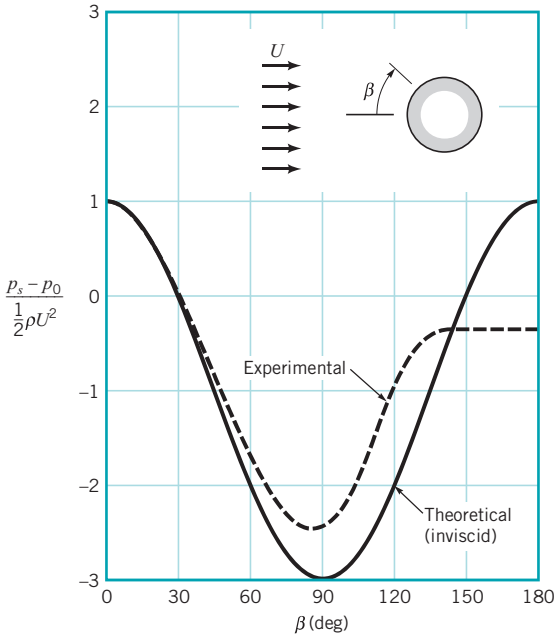
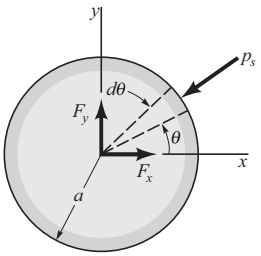



**FIGURE 6.27** A comparison of theoretical (inviscid) pressure distribution on the surface of a circular cylinder with typical experimental distribution.

CMS Cepcor EX 1046 Page 211
CMS Cepcor v. Sandvik IPR2024-01380




■ **FIGURE 6.28** **The notation for determining lift and drag on a circular cylinder.**



*V6.9 Potential and viscous flow*




*Potential theory incorrectly predicts that the drag on a cylinder is zero.*

and

$$F_y = -\int_0^{2\pi} p_s \sin\theta \, a \, d\theta \tag{6.118}$$

where $F_x$ is the *drag* (force parallel to direction of the uniform flow) and $F_y$ is the *lift* (force perpendicular to the direction of the uniform flow). Substitution for $p_s$ from Eq. 6.116 into these two equations, and subsequent integration, reveals that $F_x = 0$ and $F_y = 0$ (Problem 6.73). These results indicate that both the drag and lift as predicted by potential theory for a fixed cylinder in a uniform stream are zero. Since the pressure distribution is symmetrical around the cylinder, this is not really a surprising result. However, we know from experience that there is a significant drag developed on a cylinder when it is placed in a moving fluid. This discrepancy is known as d'Alembert's paradox. The paradox is named after Jean le Rond d'Alembert (1717–1783), a French mathematician and philosopher, who first showed that the drag on bodies immersed in inviscid fluids is zero. It was not until the latter part of the nineteenth century and the early part of the twentieth century that the role viscosity plays in the steady fluid motion was understood and d'Alembert's paradox explained (see Section 9.1).

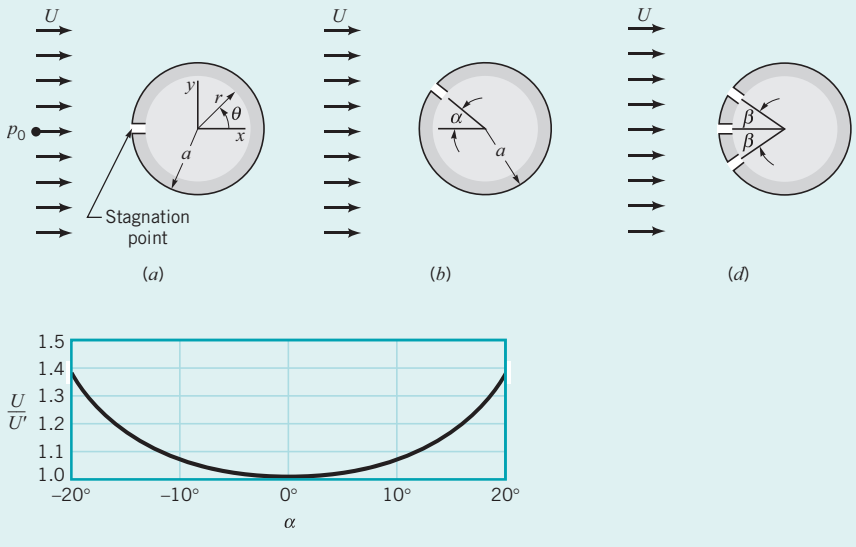
## EXAMPLE 6.8 Potential Flow—Cylinder

**GIVEN** When a circular cylinder is placed in a uniform stream, a stagnation point is created on the cylinder as is shown in Fig. E6.8*a*. If a small hole is located at this point, the stagnation pressure, $p_{stag}$, can be measured and used to determine the approach velocity, $U$.

**FIND**

**(a)** Show how $p_{stag}$ and $U$ are related.

**(b)** If the cylinder is misaligned by an angle $\alpha$ (Figure E6.8*b*), but the measured pressure is still interpreted as the stagnation pressure, determine an expression for the ratio of the true velocity, $U$, to the predicted velocity, $U'$. Plot this ratio as a function of $\alpha$ for the range $-20° \le \alpha \le 20°$.




■ **FIGURE E6.8**

## SOLUTION

**(a)** The velocity at the stagnation point is zero so the Bernoulli equation written between a point on the stagnation streamline upstream from the cylinder and the stagnation point gives

$$\frac{p_0}{\gamma} + \frac{U^2}{2g} = \frac{p_{\text{stag}}}{\gamma}$$

Thus,

$$U = \left[\frac{2}{\rho}(p_{\text{stag}} - p_0)\right]^{1/2} \qquad \textbf{(Ans)}$$

**COMMENT** A measurement of the difference between the pressure at the stagnation point and the upstream pressure can be used to measure the approach velocity. This is, of course, the same result that was obtained in Section 3.5 for Pitot-static tubes.

**(b)** If the direction of the fluid approaching the cylinder is not known precisely, it is possible that the cylinder is misaligned by some angle, $\alpha$. In this instance the pressure actually measured, $p_\alpha$, will be different from the stagnation pressure, but if the misalignment is not recognized the predicted approach velocity, $U'$, would still be calculated as

$$U' = \left[\frac{2}{\rho}(p_\alpha - p_0)\right]^{1/2}$$

Thus,

$$\frac{U(\text{true})}{U'(\text{predicted})} = \left(\frac{p_{\text{stag}} - p_0}{p_\alpha - p_0}\right)^{1/2} \qquad \textbf{(1)}$$

The velocity on the surface of the cylinder, $v_\theta$, where $r = a$, is obtained from Eq. 6.115 as

$$v_\theta = -2U \sin\theta$$

If we now write the Bernoulli equation between a point upstream of the cylinder and the point on the cylinder where $r = a$, $\theta = \alpha$, it follows that

$$p_0 + \frac{1}{2}\rho U^2 = p_\alpha + \frac{1}{2}\rho(-2U\sin\alpha)^2$$

and, therefore,

$$p_\alpha - p_0 = \tfrac{1}{2}\rho U^2(1 - 4\sin^2\alpha) \qquad \textbf{(2)}$$

Since $p_{\text{stag}} - p_0 = \frac{1}{2}\rho U^2$ it follows from Eqs. 1 and 2 that

$$\frac{U(\text{true})}{U'(\text{predicted})} = (1 - 4\sin^2\alpha)^{-1/2} \qquad \textbf{(Ans)}$$

This velocity ratio is plotted as a function of the misalignment angle $\alpha$ in Fig. E6.8*c*.

**COMMENT** It is clear from these results that significant errors can arise if the stagnation pressure tap is not aligned with the stagnation streamline. As is discussed in Section 3.5, if two additional, symmetrically located holes are drilled on the cylinder, as are illustrated in Fig. E6.8*d*, the correct orientation of the cylinder can be determined. The cylinder is rotated until the pressures in the two symmetrically placed holes are equal, thus indicating that the center hole coincides with the stagnation streamline. For $\beta = 30°$ the pressure at the two holes theoretically corresponds to the upstream pressure, $p_0$. With this orientation a measurement of the difference in pressure between the center hole and the side holes can be used to determine $U$.

An additional, interesting potential flow can be developed by adding a free vortex to the stream function or velocity potential for the flow around a cylinder. In this case

$$\psi = Ur\left(1 - \frac{a^2}{r^2}\right)\sin\theta - \frac{\Gamma}{2\pi}\ln r \qquad \textbf{(6.119)}$$

and

$$\phi = Ur\left(1 + \frac{a^2}{r^2}\right)\cos\theta + \frac{\Gamma}{2\pi}\theta \qquad \textbf{(6.120)}$$

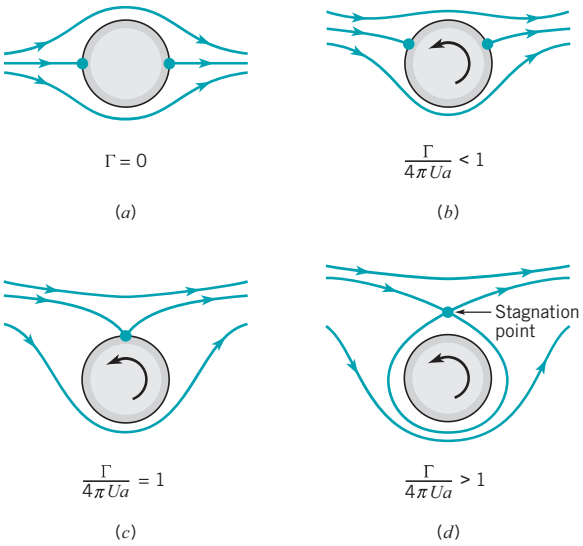
where $\Gamma$ is the circulation. We note that the circle $r = a$ will still be a streamline (and thus can be replaced with a solid cylinder), since the streamlines for the added free vortex are all circular. However, the tangential velocity, $v_\theta$, on the surface of the cylinder ($r = a$) now becomes

$$v_{\theta s} = -\left.\frac{\partial\psi}{\partial r}\right|_{r=a} = -2U\sin\theta + \frac{\Gamma}{2\pi a} \qquad \textbf{(6.121)}$$

*Flow around a rotating cylinder is approximated by the addition of a free vortex.*

This type of flow field could be approximately created by placing a rotating cylinder in a uniform stream. Because of the presence of viscosity in any real fluid, the fluid in contact with the rotating cylinder would rotate with the same velocity as the cylinder, and the resulting flow field would resemble that developed by the combination of a uniform flow past a cylinder and a free vortex.

CMS Cepcor EX 1046 Page 213
CMS Cepcor v. Sandvik IPR2024-01380




**■ FIGURE 6.29**
**The location of stagnation points on a circular cylinder: (*a*) without circulation; (*b*, *c*, *d*) with circulation.**

A variety of streamline patterns can be developed, depending on the vortex strength, $\Gamma$. For example, from Eq. 6.121 we can determine the location of stagnation points on the surface of the cylinder. These points will occur at $\theta = \theta_{stag}$ where $v_\theta = 0$ and therefore from Eq. 6.121

$$\sin\theta_{stag} = \frac{\Gamma}{4\pi Ua} \tag{6.122}$$

If $\Gamma = 0$, then $\theta_{stag} = 0$ or $\pi$—that is, the stagnation points occur at the front and rear of the cylinder as are shown in Fig. 6.29*a*. However, for $-1 \le \Gamma/4\pi Ua \le 1$, the stagnation points will occur at some other location on the surface as illustrated in Figs. 6.29*b*,*c*. If the absolute value of the parameter $\Gamma/4\pi Ua$ exceeds 1, Eq. 6.122 cannot be satisfied, and the stagnation point is located away from the cylinder as shown in Fig. 6.29*d*.

The force per unit length developed on the cylinder can again be obtained by integrating the differential pressure forces around the circumference as in Eqs. 6.117 and 6.118. For the cylinder with circulation, the surface pressure, $p_s$, is obtained from the Bernoulli equation (with the surface velocity given by Eq. 6.121)

$$p_0 + \frac{1}{2}\rho U^2 = p_s + \frac{1}{2}\rho\left(-2U\sin\theta + \frac{\Gamma}{2\pi a}\right)^2$$

or

$$p_s = p_0 + \frac{1}{2}\rho U^2\left(1 - 4\sin^2\theta + \frac{2\Gamma\sin\theta}{\pi aU} - \frac{\Gamma^2}{4\pi^2 a^2 U^2}\right) \tag{6.123}$$

Equation 6.123 substituted into Eq. 6.117 for the drag, and integrated, again yields (Problem 6.74)

$$F_x = 0$$

*Potential flow past a cylinder with circulation gives zero drag but non-zero lift.*

That is, even for the rotating cylinder no force in the direction of the uniform flow is developed. However, use of Eq. 6.123 with the equation for the lift, $F_y$ (Eq. 6.118), yields (Problem 6.74)

$$F_y = -\rho U\Gamma \tag{6.124}$$

Thus, for the cylinder with circulation, lift is developed equal to the product of the fluid density, the upstream velocity, and the circulation. The negative sign means that if $U$ is positive (in the positive $x$ direction) and $\Gamma$ is positive (a free vortex with counterclockwise rotation), the direction of the $F_y$ is downward.

Of course, if the cylinder is rotated in the clockwise direction ($\Gamma < 0$) the direction of $F_y$ would be upward. This can be seen by studying the surface pressure distribution (Eq. 6.123), which is plotted in Fig. 6.30 for two situations. One has $\Gamma/4\pi Ua = 0$, which corresponds to no rotation of the cylinder. The other has $\Gamma/4\pi Ua = -0.25$, which corresponds to clockwise rotation of the cylinder. With no

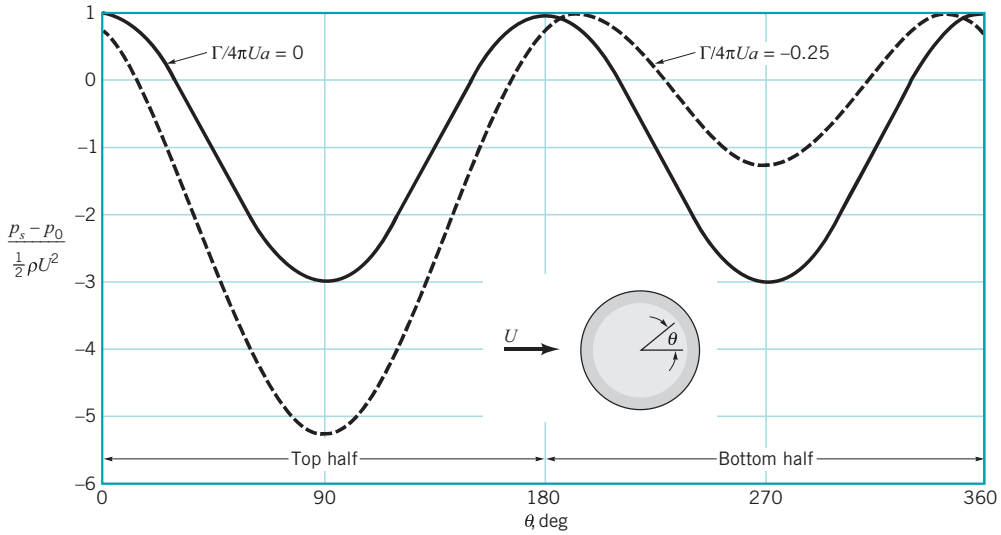



**FIGURE 6.30** **Pressure distribution on a circular cylinder with and without rotation.**

rotation the flow is symmetrical both top to bottom and front to back on the cylinder. With rotation the flow is symmetrical front to back, but not top to bottom. In this case the two stagnation points [i.e., $(p_s - p_0)/(\rho U^2/2) = 1$] are located on the bottom of the cylinder and the average pressure on the top half of the cylinder is less than that on the bottom half. The result is an upward lift force. It is this force acting in a direction perpendicular to the direction of the approach velocity that causes baseballs and golf balls to curve when they spin as they are propelled through the air. The development of this lift on rotating bodies is called the *Magnus effect.* (See Section 9.4 for further comments.)

Although Eq. 6.124 was developed for a cylinder with circulation, it gives the lift per unit length for any two-dimensional object of any cross-sectional shape placed in a uniform, inviscid stream. The circulation is determined around any closed curve containing the body. The generalized equation relating lift to fluid density, velocity, and circulation is called the *Kutta–Joukowski law,* and is commonly used to determine the lift on airfoils (see Section 9.4.2 and Refs. 2–6).

## Fluids in the News

**A sailing ship without sails** A sphere or cylinder spinning about its axis when placed in an airstream develops a force at right angles to the direction of the airstream. This phenomenon is commonly referred to as the *Magnus effect* and is responsible for the curved paths of baseballs and golf balls. Another lesser-known application of the Magnus effect was proposed by a German physicist and engineer, Anton Flettner, in the 1920s. Flettner's idea was to use the Magnus effect to make a ship move. To demonstrate the practicality of the "rotor-ship" he purchased a sailing schooner and replaced the ship's masts and rigging with two vertical cylinders that were 50 feet high and 9 feet in diameter. The cylinders looked like smokestacks on the ship. Their spinning motion was developed by 45-hp motors. The combination of a wind and the rotating cylinders created a force (Magnus effect) to push the ship forward. The ship, named the *Baden Baden*, made a successful voyage across the Atlantic, arriving in New York Harbor on May 9, 1926. Although the feasibility of the rotor-ship was clearly demonstrated, it proved to be less efficient and practical than more conventional vessels and the idea was not pursued. (See Problem 6.72.)

## 6.7 Other Aspects of Potential Flow Analysis

In the preceding section the method of superposition of basic potentials has been used to obtain detailed descriptions of irrotational flow around certain body shapes immersed in a uniform stream. For the cases considered, two or more of the basic potentials were combined and the question is asked: What kind of flow does this combination represent? This approach is relatively simple and does not require the use of advanced mathematical techniques. It is, however, restrictive in its general applicability. It does not allow us to specify a priori the body shape and then determine the velocity potential or stream function that describes the flow around the particular body.

CMS Cepcor EX 1046 Page 215
CMS Cepcor v. Sandvik IPR2024-01380

Determining the velocity potential or stream function for a given body shape is a much more complicated problem.

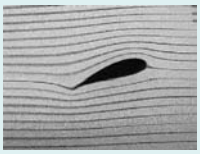
It is possible to extend the idea of superposition by considering a *distribution* of sources and sinks, or doublets, which when combined with a uniform flow can describe the flow around bodies of arbitrary shape. Techniques are available to determine the required distribution to give a prescribed body shape. Also, for plane potential flow problems it can be shown that complex variable theory (the use of real and imaginary numbers) can be effectively used to obtain solutions to a great variety of important flow problems. There are, of course, numerical techniques that can be used to solve not only plane two-dimensional problems, but the more general three-dimensional problems. Since potential flow is governed by Laplace's equation, any procedure that is available for solving this equation can be applied to the analysis of irrotational flow of frictionless fluids. Potential flow theory is an old and well-established discipline within the general field of fluid mechanics. The interested reader can find many detailed references on this subject, including Refs. 2, 3, 4, 5, and 6 given at the end of this chapter.

*Potential flow solutions are always approximate because the fluid is assumed to be frictionless.*

An important point to remember is that regardless of the particular technique used to obtain a solution to a potential flow problem, the solution remains approximate because of the fundamental assumption of a frictionless fluid. Thus, "exact" solutions based on potential flow theory represent, at best, only approximate solutions to real fluid problems. The applicability of potential flow theory to real fluid problems has been alluded to in a number of examples considered in the previous section. As a rule of thumb, potential flow theory will usually provide a reasonable approximation in those circumstances when we are dealing with a low viscosity fluid moving at a relatively high velocity, in regions of the flow field in which the flow is accelerating. Under these circumstances we generally find that the effect of viscosity is confined to the thin boundary layer that develops at a solid boundary. Outside the boundary layer the velocity distribution and the pressure distribution are closely approximated by the potential flow solution. However, in those regions of the flow field in which the flow is decelerating (for example, in the rearward portion of a bluff body or in the expanding region of a conduit), the pressure near a solid boundary will increase in the direction of flow. This so-called adverse pressure gradient can lead to flow separation, a phenomenon that causes dramatic changes in the flow field which are generally not accounted for by potential theory. However, as discussed in Chapter 9, in which boundary layer theory is developed, it is found that potential flow theory is used to obtain the appropriate pressure distribution that can then be combined with the viscous flow equations to obtain solutions near the boundary (and also to predict separation). The general differential equations that describe viscous fluid behavior and some simple solutions to these equations are considered in the remaining sections of this chapter.



*V6.10 Potential flow*



## 6.8 Viscous Flow

To incorporate viscous effects into the differential analysis of fluid motion we must return to the previously derived general equations of motion, Eqs. 6.50. Since these equations include both stresses and velocities, there are more unknowns than equations, and therefore before proceeding it is necessary to establish a relationship between the stresses and velocities.

### 6.8.1 Stress–Deformation Relationships

For incompressible Newtonian fluids it is known that the stresses are linearly related to the rates of deformation and can be expressed in Cartesian coordinates as (for normal stresses)

$$\sigma_{xx} = -p + 2\mu \frac{\partial u}{\partial x} \tag{6.125a}$$

$$\sigma_{yy} = -p + 2\mu \frac{\partial v}{\partial y} \tag{6.125b}$$

$$\sigma_{zz} = -p + 2\mu \frac{\partial w}{\partial z} \tag{6.125c}$$

(for shearing stresses)

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \tag{6.125d}$$

$$\tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) \tag{6.125e}$$

$$\tau_{zx} = \tau_{xz} = \mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right) \tag{6.125f}$$

where $p$ is the pressure, the negative of the average of the three normal stresses; that is, $-p = (\frac{1}{3})(\sigma_{xx} + \sigma_{yy} + \sigma_{zz})$. For viscous fluids in motion the normal stresses are not necessarily the same in different directions, thus, the need to define the pressure as the average of the three normal stresses. For fluids at rest, or frictionless fluids, the normal stresses are equal in all directions. (We have made use of this fact in the chapter on fluid statics and in developing the equations for inviscid flow.) Detailed discussions of the development of these stress–velocity gradient relationships can be found in Refs. 3, 7, and 8. An important point to note is that whereas for elastic solids the stresses are linearly related to the deformation (or strain), for Newtonian fluids the stresses are linearly related to the rate of deformation (or rate of strain).

*For Newtonian fluids, stresses are linearly related to the rate of strain.*

In cylindrical polar coordinates the stresses for incompressible Newtonian fluids are expressed as (for normal stresses)

$$\sigma_{rr} = -p + 2\mu\frac{\partial v_r}{\partial r} \tag{6.126a}$$

$$\sigma_{\theta\theta} = -p + 2\mu\left(\frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{v_r}{r}\right) \tag{6.126b}$$

$$\sigma_{zz} = -p + 2\mu\frac{\partial v_z}{\partial z} \tag{6.126c}$$

(for shearing stresses)

$$\tau_{r\theta} = \tau_{\theta r} = \mu\left[r\frac{\partial}{\partial r}\left(\frac{v_\theta}{r}\right) + \frac{1}{r}\frac{\partial v_r}{\partial \theta}\right] \tag{6.126d}$$

$$\tau_{\theta z} = \tau_{z\theta} = \mu\left(\frac{\partial v_\theta}{\partial z} + \frac{1}{r}\frac{\partial v_z}{\partial \theta}\right) \tag{6.126e}$$

$$\tau_{zr} = \tau_{rz} = \mu\left(\frac{\partial v_r}{\partial z} + \frac{\partial v_z}{\partial r}\right) \tag{6.126f}$$

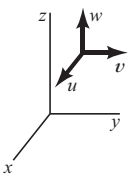
The double subscript has a meaning similar to that of stresses expressed in Cartesian coordinates—that is, the first subscript indicates the plane on which the stress acts, and the second subscript the direction. Thus, for example, $\sigma_{rr}$ refers to a stress acting on a plane perpendicular to the radial direction and in the radial direction (thus a normal stress). Similarly, $\tau_{r\theta}$ refers to a stress acting on a plane perpendicular to the radial direction but in the tangential ($\theta$ direction) and is therefore a shearing stress.

### 6.8.2 The Navier–Stokes Equations

The stresses as defined in the preceding section can be substituted into the differential equations of motion (Eqs. 6.50) and simplified by using the continuity equation (Eq. 6.31) to obtain:

($x$ direction)

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \rho g_x + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \tag{6.127a}$$




($y$ direction)

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \rho g_y + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \tag{6.127b}$$

CMS Cepcor EX 1046 Page 217
CMS Cepcor v. Sandvik IPR2024-01380

($z$ direction)

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{6.127c}$$

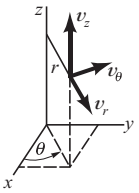
where $u$, $v$, and $w$ are the $x$, $y$, and $z$ components of velocity as shown in the figure in the margin of the previous page. We have rearranged the equations so the acceleration terms are on the left side and the force terms are on the right. These equations are commonly called the ***Navier–Stokes equations,*** named in honor of the French mathematician L. M. H. Navier (1785–1836) and the English mechanician Sir G. G. Stokes (1819–1903), who were responsible for their formulation. These three equations of motion, when combined with the conservation of mass equation (Eq. 6.31), provide a complete mathematical description of the flow of incompressible Newtonian fluids. We have four equations and four unknowns ($u$, $v$, $w$, and $p$), and therefore the problem is "well-posed" in mathematical terms. Unfortunately, because of the general complexity of the Navier–Stokes equations (they are nonlinear, second-order, partial differential equations), they are not amenable to exact mathematical solutions except in a few instances. However, in those few instances in which solutions have been obtained and compared with experimental results, the results have been in close agreement. Thus, the Navier–Stokes equations are considered to be the governing differential equations of motion for incompressible Newtonian fluids.

*The Navier–Stokes equations are the basic differential equations describing the flow of Newtonian fluids.*

In terms of cylindrical polar coordinates (see the figure in the margin), the Navier–Stokes equations can be written as

($r$ direction)

$$\rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_r}{\partial \theta}-\frac{v_\theta^2}{r}+v_z\frac{\partial v_r}{\partial z}\right)$$
$$=-\frac{\partial p}{\partial r}+\rho g_r+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_r}{\partial r}\right)-\frac{v_r}{r^2}+\frac{1}{r^2}\frac{\partial^2 v_r}{\partial \theta^2}-\frac{2}{r^2}\frac{\partial v_\theta}{\partial \theta}+\frac{\partial^2 v_r}{\partial z^2}\right] \tag{6.128a}$$




($\theta$ direction)

$$\rho\left(\frac{\partial v_\theta}{\partial t}+v_r\frac{\partial v_\theta}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r v_\theta}{r}+v_z\frac{\partial v_\theta}{\partial z}\right)$$
$$=-\frac{1}{r}\frac{\partial p}{\partial \theta}+\rho g_\theta+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_\theta}{\partial r}\right)-\frac{v_\theta}{r^2}+\frac{1}{r^2}\frac{\partial^2 v_\theta}{\partial \theta^2}+\frac{2}{r^2}\frac{\partial v_r}{\partial \theta}+\frac{\partial^2 v_\theta}{\partial z^2}\right] \tag{6.128b}$$

($z$ direction)

$$\rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+\frac{v_\theta}{r}\frac{\partial v_z}{\partial \theta}+v_z\frac{\partial v_z}{\partial z}\right)$$
$$=-\frac{\partial p}{\partial z}+\rho g_z+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{1}{r^2}\frac{\partial^2 v_z}{\partial \theta^2}+\frac{\partial^2 v_z}{\partial z^2}\right] \tag{6.128c}$$

To provide a brief introduction to the use of the Navier–Stokes equations, a few of the simplest exact solutions are developed in the next section. Although these solutions will prove to be relatively simple, this is not the case in general. In fact, only a few other exact solutions have been obtained.

## 6.9 Some Simple Solutions for Laminar, Viscous, Incompressible Fluids

A principal difficulty in solving the Navier–Stokes equations is because of their nonlinearity arising from the convective acceleration terms (i.e., $u\,\partial u/\partial x$, $w\,\partial v/\partial z$, etc.). There are no general analytical schemes for solving nonlinear partial differential equations (e.g., superposition of solutions cannot be used), and each problem must be considered individually. For most practical flow problems, fluid particles do have accelerated motion as they move from one location to another in the flow field. Thus, the convective acceleration terms are usually important. However, there are a few special cases for which the convective acceleration vanishes because of the nature of the geometry of the flow

system. In these cases exact solutions are often possible. The Navier–Stokes equations apply to both laminar and turbulent flow, but for turbulent flow each velocity component fluctuates randomly with respect to time and this added complication makes an analytical solution intractable. Thus, the exact solutions referred to are for laminar flows in which the velocity is either independent of time (steady flow) or dependent on time (unsteady flow) in a well-defined manner.

### 6.9.1 Steady, Laminar Flow between Fixed Parallel Plates

*An exact solution can be obtained for steady laminar flow between fixed parallel plates.*

We first consider flow between the two horizontal, infinite parallel plates of Fig. 6.31*a*. For this geometry the fluid particles move in the $x$ direction parallel to the plates, and there is no velocity in the $y$ or $z$ direction—that is, $v = 0$ and $w = 0$. In this case it follows from the continuity equation (Eq. 6.31) that $\partial u/\partial x = 0$. Furthermore, there would be no variation of $u$ in the $z$ direction for infinite plates, and for steady flow $\partial u/\partial t = 0$ so that $u = u(y)$. If these conditions are used in the Navier–Stokes equations (Eqs. 6.127), they reduce to

$$0 = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial y^2}\right) \tag{6.129}$$

$$0 = -\frac{\partial p}{\partial y} - \rho g \tag{6.130}$$

$$0 = -\frac{\partial p}{\partial z} \tag{6.131}$$

where we have set $g_x = 0$, $g_y = -g$, and $g_z = 0$. That is, the $y$ axis points up. We see that for this particular problem the Navier–Stokes equations reduce to some rather simple equations.

Equations 6.130 and 6.131 can be integrated to yield

$$p = -\rho g y + f_1(x) \tag{6.132}$$

which shows that the pressure varies hydrostatically in the $y$ direction. Equation 6.129, rewritten as

$$\frac{d^2 u}{dy^2} = \frac{1}{\mu}\frac{\partial p}{\partial x}$$

can be integrated to give

$$\frac{du}{dy} = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right) y + c_1$$



*V6.11 No-slip boundary condition*



and integrated again to yield

$$u = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) y^2 + c_1 y + c_2 \tag{6.133}$$

Note that for this simple flow the pressure gradient, $\partial p/\partial x$, is treated as constant as far as the integration is concerned, since (as shown in Eq. 6.132) it is not a function of $y$. The two constants $c_1$ and $c_2$ must be determined from the boundary conditions. For example, if the two plates are

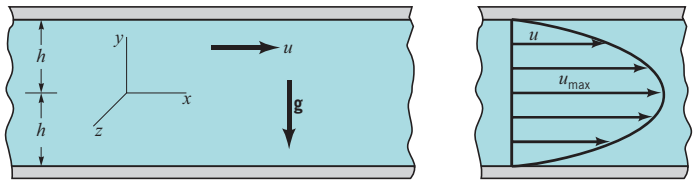



**FIGURE 6.31** **The viscous flow between parallel plates: (*a*) coordinate system and notation used in analysis; (*b*) parabolic velocity distribution for flow between parallel fixed plates.**

fixed, then $u = 0$ for $y = \pm h$ (because of the no-slip condition for viscous fluids). To satisfy this condition $c_1 = 0$ and

$$c_2 = -\frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)h^2$$

Thus, the velocity distribution becomes

$$u = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - h^2) \tag{6.134}$$

Equation 6.134 shows that the velocity profile between the two fixed plates is parabolic as illustrated in Fig. 6.31*b*.



*V6.12 Liquid–liquid no-slip*



The volume rate of flow, $q$, passing between the plates (for a unit width in the $z$ direction) is obtained from the relationship

$$q = \int_{-h}^{h} u\,dy = \int_{-h}^{h} \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - h^2)\,dy$$

or

$$q = -\frac{2h^3}{3\mu}\left(\frac{\partial p}{\partial x}\right) \tag{6.135}$$

The pressure gradient $\partial p/\partial x$ is negative, since the pressure decreases in the direction of flow. If we let $\Delta p$ represent the pressure *drop* between two points a distance $\ell$ apart, then

$$\frac{\Delta p}{\ell} = -\frac{\partial p}{\partial x}$$

and Eq. 6.135 can be expressed as

$$q = \frac{2h^3\Delta p}{3\mu\ell} \tag{6.136}$$

The flow is proportional to the pressure gradient, inversely proportional to the viscosity, and strongly dependent $(\sim h^3)$ on the gap width. In terms of the mean velocity, $V$, where $V = q/2h$, Eq. 6.136 becomes

$$V = \frac{h^2\Delta p}{3\mu\ell} \tag{6.137}$$

Equations 6.136 and 6.137 provide convenient relationships for relating the pressure drop along a parallel-plate channel and the rate of flow or mean velocity. The maximum velocity, $u_{max}$, occurs midway $(y = 0)$ between the two plates, as shown in Fig. 6.31*b*, so that from Eq. 6.134

$$u_{max} = -\frac{h^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)$$

or

$$u_{max} = \tfrac{3}{2}V \tag{6.138}$$

*The Navier–Stokes equations provide detailed flow characteristics for laminar flow between fixed parallel plates.*

The details of the steady laminar flow between infinite parallel plates are completely predicted by this solution to the Navier–Stokes equations. For example, if the pressure gradient, viscosity, and plate spacing are specified, then from Eq. 6.134 the velocity profile can be determined, and from Eqs. 6.136 and 6.137 the corresponding flowrate and mean velocity determined. In addition, from Eq. 6.132 it follows that

$$f_1(x) = \left(\frac{\partial p}{\partial x}\right)x + p_0$$

where $p_0$ is a reference pressure at $x = y = 0$, and the pressure variation throughout the fluid can be obtained from

$$p = -\rho g y + \left(\frac{\partial p}{\partial x}\right)x + p_0 \tag{6.139}$$

For a given fluid and reference pressure, $p_0$, the pressure at any point can be predicted. This relatively simple example of an exact solution illustrates the detailed information about the flow field which can be obtained. The flow will be laminar if the Reynolds number, $\mathrm{Re} = \rho V(2h)/\mu$, remains below about 1400. For flow with larger Reynolds numbers the flow becomes turbulent and the preceding analysis is not valid since the flow field is complex, three-dimensional, and unsteady.

## Fluids in the News

**10 tons on 8 psi** Place a golf ball on the end of a garden hose and then slowly turn the water on a small amount until the ball just barely lifts off the end of the hose, leaving a small gap between the ball and the hose. The ball is free to rotate. This is the idea behind the new "floating ball water fountains" developed in Finland. Massive, 10-ton, 6-ft-diameter stone spheres are supported by the pressure force of the water on the curved surface within a pedestal and rotate so easily that even a small child can change their direction of rotation. The key to the fountain design is the ability to grind and polish stone to an accuracy of a few thousandths of an inch. This allows the gap between the ball and its pedestal to be very small (on the order of 5/1000 in.) and the water flowrate correspondingly small (on the order of 5 gallons per minute). Due to the small gap, the flow in the gap is essentially that of *flow between parallel plates.* Although the sphere is very heavy, the pressure under the sphere within the pedestal needs to be only about 8 psi. (See Problem 6.88.)

### 6.9.2 Couette Flow

*For a given flow geometry, the character and details of the flow are strongly dependent on the boundary conditions.*

Another simple parallel-plate flow can be developed by fixing one plate and letting the other plate move with a constant velocity, $U$, as is illustrated in Fig. 6.32*a*. The Navier–Stokes equations reduce to the same form as those in the preceding section, and the solution for the pressure and velocity distribution are still given by Eqs. 6.132 and 6.133, respectively. However, for the moving plate problem the boundary conditions for the velocity are different. For this case we locate the origin of the coordinate system at the bottom plate and designate the distance between the two plates as $b$ (see Fig. 6.32*a*). The two constants $c_1$ and $c_2$ in Eq. 6.133 can be determined from the boundary conditions, $u = 0$ at $y = 0$ and $u = U$ at $y = b$. It follows that

$$u = U\frac{y}{b} + \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - by) \tag{6.140}$$

or, in dimensionless form,

$$\frac{u}{U} = \frac{y}{b} - \frac{b^2}{2\mu U}\left(\frac{\partial p}{\partial x}\right)\left(\frac{y}{b}\right)\left(1 - \frac{y}{b}\right) \tag{6.141}$$

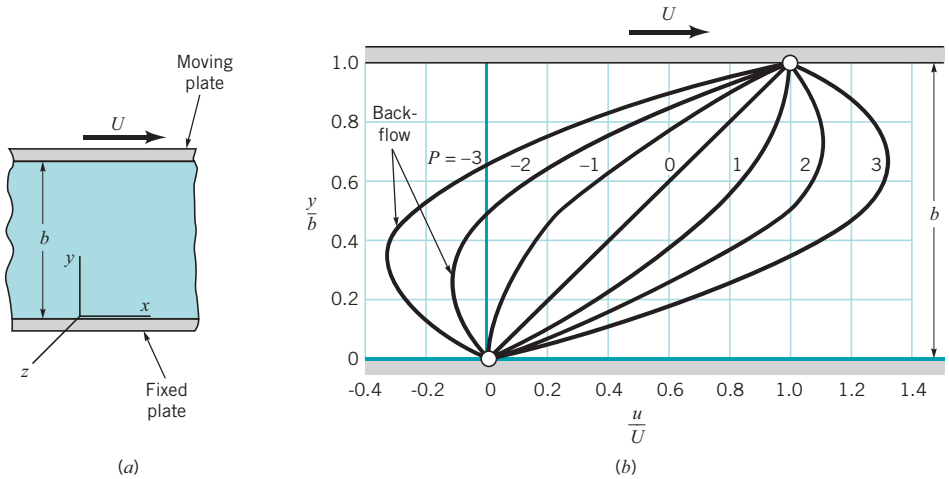
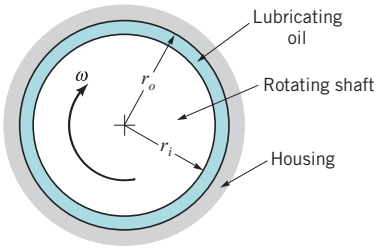



**FIGURE 6.32** **The viscous flow between parallel plates with bottom plate fixed and upper plate moving (Couette flow): (*a*) coordinate system and notation used in analysis; (*b*) velocity distribution as a function of parameter, $P$, where $P = -(b^2/2\mu U)\,\partial p/\partial x$. (From Ref. 8, used by permission.)**




■ **FIGURE 6.33** **Flow in the narrow gap of a journal bearing.**

The actual velocity profile will depend on the dimensionless parameter

$$P = -\frac{b^2}{2\mu U}\left(\frac{\partial p}{\partial x}\right)$$

*Flow between parallel plates with one plate fixed and the other moving is called Couette flow.*

Several profiles are shown in Fig. 6.32*b*. This type of flow is called ***Couette flow.***

The simplest type of Couette flow is one for which the pressure gradient is zero; that is, the fluid motion is caused by the fluid being dragged along by the moving boundary. In this case, with $\partial p/\partial x = 0$, Eq. 6.140 simply reduces to

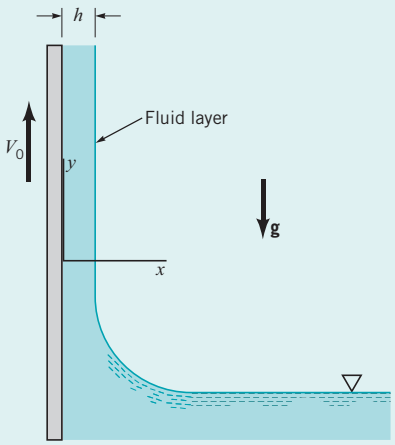
$$u = U\frac{y}{b} \tag{6.142}$$

which indicates that the velocity varies linearly between the two plates as shown in Fig. 6.31*b* for $P = 0$. This situation would be approximated by the flow between closely spaced concentric cylinders in which one cylinder is fixed and the other cylinder rotates with a constant angular velocity, $\omega$. As illustrated in Fig. 6.33, the flow in an unloaded journal bearing might be approximated by this simple Couette flow if the gap width is very small (i.e., $r_o - r_i \ll r_i$). In this case $U = r_i\omega$, $b = r_o - r_i$, and the shearing stress resisting the rotation of the shaft can be simply calculated as $\tau = \mu r_i\omega/(r_o - r_i)$. When the bearing is loaded (i.e., a force applied normal to the axis of rotation), the shaft will no longer remain concentric with the housing and the flow cannot be treated as flow between parallel boundaries. Such problems are dealt with in lubrication theory (see, for example, Ref. 9).

## EXAMPLE 6.9 Plane Couette Flow

**GIVEN** A wide moving belt passes through a container of a viscous liquid. The belt moves vertically upward with a constant velocity, $V_0$, as illustrated in Fig. E6.9*a*. Because of viscous forces the belt picks up a film of fluid of thickness $h$. Gravity tends to make the fluid drain down the belt. Assume that the flow is laminar, steady, and fully developed.

**FIND** Use the Navier–Stokes equations to determine an expression for the average velocity of the fluid film as it is dragged up the belt.




■ **FIGURE E6.9*a***

### SOLUTION

Since the flow is assumed to be fully developed, the only velocity component is in the $y$ direction (the $v$ component) so that $u = w = 0$. It follows from the continuity equation that $\partial v/\partial y = 0$, and for steady flow $\partial v/\partial t = 0$, so that $v = v(x)$. Under these conditions the Navier–Stokes equations for the $x$ direction (Eq. 6.127*a*) and the $z$ direction (perpendicular to the paper) (Eq. 6.127*c*) simply reduce to

$$\frac{\partial p}{\partial x} = 0 \qquad \frac{\partial p}{\partial z} = 0$$

This result indicates that the pressure does not vary over a horizontal plane, and since the pressure on the surface of the film ($x = h$) is atmospheric, the pressure throughout the film must be

atmospheric (or zero gage pressure). The equation of motion in the $y$ direction (Eq. 6.127$b$) thus reduces to

$$0 = -\rho g + \mu \frac{d^2v}{dx^2}$$

or

$$\frac{d^2v}{dx^2} = \frac{\gamma}{\mu} \tag{1}$$

Integration of Eq. 1 yields

$$\frac{dv}{dx} = \frac{\gamma}{\mu}x + c_1 \tag{2}$$

On the film surface ($x = h$) we assume the shearing stress is zero—that is, the drag of the air on the film is negligible. The shearing stress at the free surface (or any interior parallel surface) is designated as $\tau_{xy}$, where from Eq. 6.125$d$

$$\tau_{xy} = \mu\left(\frac{dv}{dx}\right)$$

Thus, if $\tau_{xy} = 0$ at $x = h$, it follows from Eq. 2 that

$$c_1 = -\frac{\gamma h}{\mu}$$

A second integration of Eq. 2 gives the velocity distribution in the film as

$$v = \frac{\gamma}{2\mu}x^2 - \frac{\gamma h}{\mu}x + c_2$$

At the belt ($x = 0$) the fluid velocity must match the belt velocity, $V_0$, so that

$$c_2 = V_0$$

and the velocity distribution is therefore

$$v = \frac{\gamma}{2\mu}x^2 - \frac{\gamma h}{\mu}x + V_0 \tag{3}$$

With the velocity distribution known we can determine the flowrate per unit width, $q$, from the relationship

$$q = \int_0^h v\,dx = \int_0^h \left(\frac{\gamma}{2\mu}x^2 - \frac{\gamma h}{\mu}x + V_0\right)dx$$

and thus

$$q = V_0 h - \frac{\gamma h^3}{3\mu}$$

The average film velocity, $V$ (where $q = Vh$), is therefore

$$V = V_0 - \frac{\gamma h^2}{3\mu} \tag{Ans}$$

**COMMENT** Equation (3) can be written in dimensionless form as

$$\frac{v}{V_0} = c\left(\frac{x}{h}\right)^2 - 2c\left(\frac{x}{h}\right) + 1$$

where $c = \gamma h^2/2\mu V_0$. This velocity profile is shown in Fig. E6.9$b$. Note that even though the belt is moving upward, for $c > 1$ (e.g., for fluids with small enough viscosity or with a small enough belt speed) there are portions of the fluid that flow downward (as indicated by $v/V_0 < 0$).

It is interesting to note from this result that there will be a net upward flow of liquid (positive $V$) only if $V_0 > \gamma h^2/3\mu$. It takes a relatively large belt speed to lift a small viscosity fluid.

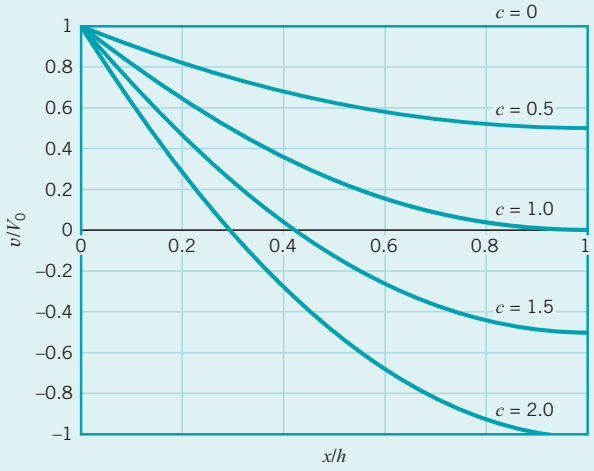


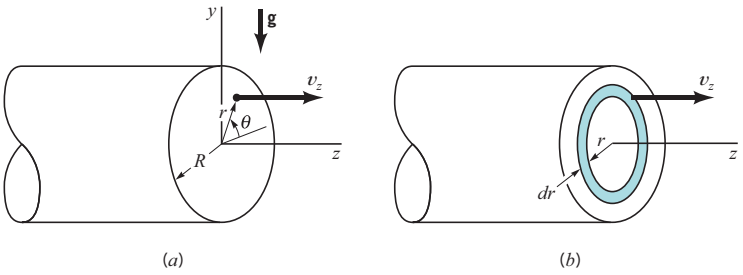

**FIGURE E6.9b**

### 6.9.3 Steady, Laminar Flow in Circular Tubes

*An exact solution can be obtained for steady, incompressible, laminar flow in circular tubes.*

Probably the best known exact solution to the Navier–Stokes equations is for steady, incompressible, laminar flow through a straight circular tube of constant cross section. This type of flow is commonly called *Hagen–Poiseuille flow*, or simply *Poiseuille flow.* It is named in honor of J. L. Poiseuille (1799–1869), a French physician, and G. H. L. Hagen (1797–1884), a German hydraulic engineer. Poiseuille was interested in blood flow through capillaries and deduced experimentally the resistance laws for laminar flow through circular tubes. Hagen's investigation of flow in tubes was also experimental. It was actually after the work of Hagen and Poiseuille that the theoretical results presented in this section were determined, but their names are commonly associated with the solution of this problem.

Consider the flow through a horizontal circular tube of radius $R$ as is shown in Fig. 6.34$a$. Because of the cylindrical geometry it is convenient to use cylindrical coordinates. We assume that the flow is parallel to the walls so that $v_r = 0$ and $v_\theta = 0$, and from the continuity equation (6.34) $\partial v_z/\partial z = 0$. Also, for steady, axisymmetric flow, $v_z$ is not a function of $t$ or $\theta$ so the velocity, $v_z$,




**■ FIGURE 6.34**
**The viscous flow in a horizontal, circular tube: (*a*) coordinate system and notation used in analysis; (*b*) flow through differential annular ring.**

is only a function of the radial position within the tube—that is, $v_z = v_z(r)$. Under these conditions the Navier–Stokes equations (Eqs. 6.128) reduce to

$$0 = -\rho g \sin\theta - \frac{\partial p}{\partial r} \tag{6.143}$$

$$0 = -\rho g \cos\theta - \frac{1}{r}\frac{\partial p}{\partial \theta} \tag{6.144}$$

$$0 = -\frac{\partial p}{\partial z} + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)\right] \tag{6.145}$$

where we have used the relationships $g_r = -g \sin\theta$ and $g_\theta = -g \cos\theta$ (with $\theta$ measured from the horizontal plane).

Equations 6.143 and 6.144 can be integrated to give

$$p = -\rho g(r \sin\theta) + f_1(z)$$

or

$$p = -\rho g y + f_1(z) \tag{6.146}$$

Equation 6.146 indicates that the pressure is hydrostatically distributed at any particular cross section, and the $z$ component of the pressure gradient, $\partial p/\partial z$, is not a function of $r$ or $\theta$.



*V6.13 Laminar flow*



The equation of motion in the $z$ direction (Eq. 6.145) can be written in the form

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right) = \frac{1}{\mu}\frac{\partial p}{\partial z}$$

and integrated (using the fact that $\partial p/\partial z$ = constant) to give

$$r\frac{\partial v_z}{\partial r} = \frac{1}{2\mu}\left(\frac{\partial p}{\partial z}\right)r^2 + c_1$$

Integrating again we obtain

$$v_z = \frac{1}{4\mu}\left(\frac{\partial p}{\partial z}\right)r^2 + c_1 \ln r + c_2 \tag{6.147}$$

Since we wish $v_z$ to be finite at the center of the tube ($r = 0$), it follows that $c_1 = 0$ [since $\ln(0) = -\infty$]. At the wall ($r = R$) the velocity must be zero so that

$$c_2 = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial z}\right)R^2$$

and the velocity distribution becomes

*The velocity distribution is parabolic for steady, laminar flow in circular tubes.*

$$v_z = \frac{1}{4\mu}\left(\frac{\partial p}{\partial z}\right)(r^2 - R^2) \tag{6.148}$$

Thus, at any cross section the velocity distribution is parabolic.

To obtain a relationship between the volume rate of flow, $Q$, passing through the tube and the pressure gradient, we consider the flow through the differential, washer-shaped ring of Fig. 6.34*b*. Since $v_z$ is constant on this ring, the volume rate of flow through the differential area $dA = (2\pi r)\,dr$ is

$$dQ = v_z(2\pi r)\,dr$$

and therefore

$$Q = 2\pi \int_0^R v_z r \, dr \tag{6.149}$$



*V6.14 Complex pipe flow*



Equation 6.148 for $v_z$ can be substituted into Eq. 6.149, and the resulting equation integrated to yield

$$Q = -\frac{\pi R^4}{8\mu}\left(\frac{\partial p}{\partial z}\right) \tag{6.150}$$

This relationship can be expressed in terms of the pressure *drop*, $\Delta p$, which occurs over a length, $\ell$, along the tube, since

$$\frac{\Delta p}{\ell} = -\frac{\partial p}{\partial z}$$

and therefore

$$Q = \frac{\pi R^4 \Delta p}{8\mu\ell} \tag{6.151}$$

*Poiseuille's law relates pressure drop and flowrate for steady, laminar flow in circular tubes.*

For a given pressure drop per unit length, the volume rate of flow is inversely proportional to the viscosity and proportional to the tube radius to the fourth power. A doubling of the tube radius produces a 16-fold increase in flow! Equation 6.151 is commonly called ***Poiseuille's law.***

In terms of the mean velocity, $V$, where $V = Q/\pi R^2$, Eq. 6.151 becomes

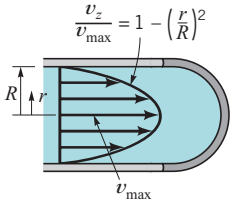
$$V = \frac{R^2 \Delta p}{8\mu\ell} \tag{6.152}$$

The maximum velocity $v_{max}$ occurs at the center of the tube, where from Eq. 6.148

$$v_{max} = -\frac{R^2}{4\mu}\left(\frac{\partial p}{\partial z}\right) = \frac{R^2 \Delta p}{4\mu\ell} \tag{6.153}$$

so that

$$v_{max} = 2V$$




The velocity distribution, as shown by the figure in the margin, can be written in terms of $v_{max}$ as

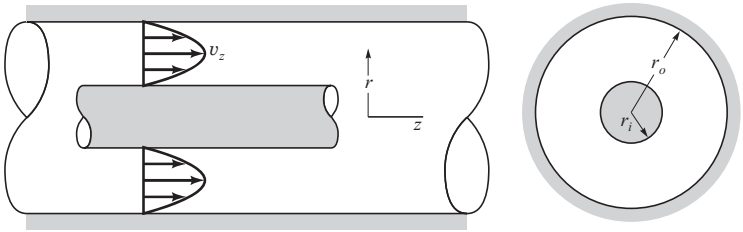
$$\frac{v_z}{v_{max}} = 1 - \left(\frac{r}{R}\right)^2 \tag{6.154}$$

As was true for the similar case of flow between parallel plates (sometimes referred to as *plane Poiseuille flow*), a very detailed description of the pressure and velocity distribution in tube flow results from this solution to the Navier–Stokes equations. Numerous experiments performed to substantiate the theoretical results show that the theory and experiment are in agreement for the laminar flow of Newtonian fluids in circular tubes or pipes. In general, the flow remains laminar for Reynolds numbers, $\mathrm{Re} = \rho V(2R)/\mu$, below 2100. Turbulent flow in tubes is considered in Chapter 8.

## Fluids in the News

**Poiseuille's law revisited** Poiseuille's law governing *laminar flow* of fluids in tubes has an unusual history. It was developed in 1842 by a French physician, J. L. M. Poiseuille, who was interested in the flow of blood in capillaries. Poiseuille, through a series of carefully conducted experiments using water flowing through very small tubes, arrived at the formula, $Q = K\Delta p\, D^4/\ell$. In this formula $Q$ is the flowrate, $K$ an empirical constant, $\Delta p$ the pressure drop over the length $\ell$, and $D$ the tube diameter. Another formula was given for the value of $K$ as a function of the water temperature. It was not until the concept of viscosity was introduced at a later date that Poiseuille's law was derived mathematically and the constant $K$ found to be equal to $\pi/8\mu$, where $\mu$ is the fluid viscosity. The experiments by Poiseuille have long been admired for their accuracy and completeness considering the laboratory instrumentation available in the mid nineteenth century.

CMS Cepcor EX 1046 Page 225
CMS Cepcor v. Sandvik IPR2024-01380




■ FIGURE 6.35 The viscous flow through an annulus.

### 6.9.4 Steady, Axial, Laminar Flow in an Annulus

*An exact solution can be obtained for axial flow in the annular space between two fixed, concentric cylinders.*

The differential equations (Eqs. 6.143, 6.144, 6.145) used in the preceding section for flow in a tube also apply to the axial flow in the annular space between two fixed, concentric cylinders (Fig. 6.35). Equation 6.147 for the velocity distribution still applies, but for the stationary annulus the boundary conditions become $v_z = 0$ at $r = r_o$ and $v_z = 0$ for $r = r_i$. With these two conditions the constants $c_1$ and $c_2$ in Eq. 6.147 can be determined and the velocity distribution becomes

$$v_z = \frac{1}{4\mu}\left(\frac{\partial p}{\partial z}\right)\left[r^2 - r_o^2 + \frac{r_i^2 - r_o^2}{\ln(r_o/r_i)}\ln\frac{r}{r_o}\right] \tag{6.155}$$

The corresponding volume rate of flow is

$$Q = \int_{r_i}^{r_o} v_z(2\pi r)\,dr = -\frac{\pi}{8\mu}\left(\frac{\partial p}{\partial z}\right)\left[r_o^4 - r_i^4 - \frac{(r_o^2 - r_i^2)^2}{\ln(r_o/r_i)}\right]$$

or in terms of the pressure drop, $\Delta p$, in length $\ell$ of the annulus

$$Q = \frac{\pi \Delta p}{8\mu\ell}\left[r_o^4 - r_i^4 - \frac{(r_o^2 - r_i^2)^2}{\ln(r_o/r_i)}\right] \tag{6.156}$$

The velocity at any radial location within the annular space can be obtained from Eq. 6.155. The maximum velocity occurs at the radius $r = r_m$ where $\partial v_z/\partial r = 0$. Thus,

$$r_m = \left[\frac{r_o^2 - r_i^2}{2\ln(r_o/r_i)}\right]^{1/2} \tag{6.157}$$

An inspection of this result shows that the maximum velocity does not occur at the midpoint of the annular space, but rather it occurs nearer the inner cylinder. The specific location depends on $r_o$ and $r_i$.

These results for flow through an annulus are valid only if the flow is laminar. A criterion based on the conventional Reynolds number (which is defined in terms of the tube diameter) cannot be directly applied to the annulus, since there are really "two" diameters involved. For tube cross sections other than simple circular tubes it is common practice to use an "effective" diameter, termed the *hydraulic diameter*, $D_h$, which is defined as

$$D_h = \frac{4 \times \text{cross-sectional area}}{\text{wetted perimeter}}$$

The wetted perimeter is the perimeter in contact with the fluid. For an annulus

$$D_h = \frac{4\pi(r_o^2 - r_i^2)}{2\pi(r_o + r_i)} = 2(r_o - r_i)$$

In terms of the hydraulic diameter, the Reynolds number is $\text{Re} = \rho D_h V/\mu$ (where $V = Q/$ cross-sectional area), and it is commonly assumed that if this Reynolds number remains below 2100 the flow will be laminar. A further discussion of the concept of the hydraulic diameter as it applies to other noncircular cross sections is given in Section 8.4.3.

## EXAMPLE 9.10 Laminar Flow in an Annulus

**GIVEN** A viscous liquid ($\rho = 1.18 \times 10^3$ kg/m$^3$; $\mu = 0.0045$ N $\cdot$ s/m$^2$) flows at a rate of 12 ml/s through a horizontal, 4-mm-diameter tube.

**FIND** (a) Determine the pressure drop along a l-m length of the tube which is far from the tube entrance so that the only component of velocity is parallel to the tube axis. (b) If a 2-mm-diameter rod is placed in the 4-mm-diameter tube to form a symmetric annulus, what is the pressure drop along a l-m length if the flowrate remains the same as in part (a)?

### SOLUTION

**(a)** We first calculate the Reynolds number, Re, to determine whether or not the flow is laminar. With the diameter $D = 4$ mm $= 0.004$ m, the mean velocity is

$$V = \frac{Q}{(\pi/4)D^2} = \frac{(12 \text{ ml/s})(10^{-6} \text{ m}^3/\text{ml})}{(\pi/4)(0.004 \text{ m})^2} = 0.955 \text{ m/s}$$

and, therefore,

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(1.18 \times 10^3 \text{ kg/m}^3)(0.955 \text{ m/s})(0.004 \text{ m})}{0.0045 \text{ N} \cdot \text{s/m}^2} = 1000$$

Since the Reynolds number is well below the critical value of 2100 we can safely assume that the flow is laminar. Thus, we can apply Eq. 6.151, which gives for the pressure drop

$$\Delta p = \frac{8\mu \ell Q}{\pi R^4} = \frac{8(0.0045 \text{ N} \cdot \text{s/m}^2)(1 \text{ m})(12 \times 10^{-6} \text{ m}^3/\text{s})}{\pi(0.002 \text{ m})^4} = 8.59 \text{ kPa} \quad \textbf{(Ans)}$$

**(b)** For flow in the annulus with an outer radius $r_o = 0.002$ m and an inner radius $r_i = 0.001$ m, the mean velocity is

$$V = \frac{Q}{\pi(r_o^2 - r_i^2)} = \frac{12 \times 10^{-6} \text{ m}^3/\text{s}}{(\pi)[(0.002 \text{ m})^2 - (0.001 \text{ m})^2]} = 1.27 \text{ m/s}$$

and the Reynolds number [based on the hydraulic diameter, $D_h = 2(r_o - r_i) = 2(0.002 \text{ m} - 0.001 \text{ m}) = 0.002$ m] is

$$\text{Re} = \frac{\rho D_h V}{\mu} = \frac{(1.18 \times 10^3 \text{ kg/m}^3)\,(0.002 \text{ m})\,(1.27 \text{ m/s})}{0.0045 \text{ N} \cdot \text{s/m}^2} = 666$$

This value is also well below 2100 so the flow in the annulus should also be laminar. From Eq. 6.156,

$$\Delta p = \frac{8\mu \ell Q}{\pi}\left[r_o^4 - r_i^4 - \frac{(r_o^2 - r_i^2)^2}{\ln(r_o/r_i)}\right]^{-1}$$

so that

$$\Delta p = \frac{8(0.0045 \text{ N} \cdot \text{s/m}^2)(1 \text{ m})(12 \times 10^{-6} \text{ m}^3/\text{s})}{\pi} \times \left\{(0.002 \text{ m})^4 - (0.001 \text{ m})^4 - \frac{[(0.002 \text{ m})^2 - (0.001 \text{ m})^2]^2}{\ln(0.002 \text{ m}/0.001 \text{ m})}\right\}^{-1}$$

$$= 68.2 \text{ kPa} \quad \textbf{(Ans)}$$

**COMMENTS** The pressure drop in the annulus is much larger than that of the tube. This is not a surprising result, since to maintain the same flow in the annulus as that in the open tube, the average velocity must be larger (the cross-sectional area is smaller) and the pressure difference along the annulus must overcome the shearing stresses that develop along both an inner and an outer wall.

By repeating the calculations for various radius ratios, $r_i/r_o$, the results shown in Fig. E6.10 are obtained. It is seen that the pressure drop ratio, $\Delta p_{\text{annulus}}/\Delta p_{\text{tube}}$ (i.e., the pressure drop in the annulus compared to that in a tube with a radius equal to the outer radius of the annulus, $r_o$), is a strong function of the radius ratio. Even an annulus with a very small inner radius will have a pressure drop significantly larger than that of a tube. For example, if the inner radius is only 1/100 of the outer radius, $\Delta p_{\text{annulus}}/\Delta p_{\text{tube}} = 1.28$. As shown in the figure, for larger inner radii, the pressure drop ratio is much larger [i.e., $\Delta p_{\text{annulus}}/\Delta p_{\text{tube}} = 7.94$ for $r_i/r_o = 0.50$ as in part (b) of this example].

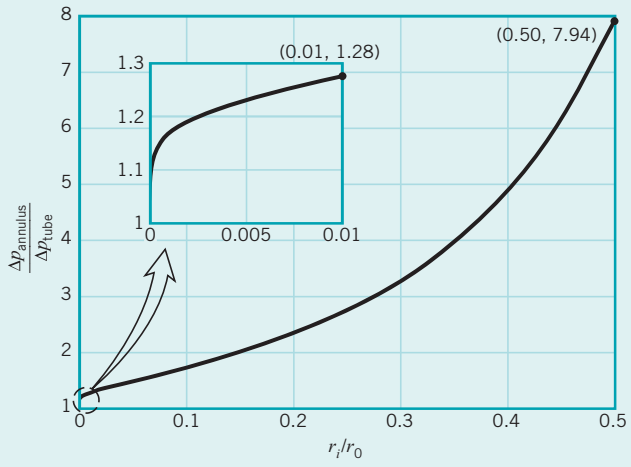



**FIGURE E6.10**

## 6.10 Other Aspects of Differential Analysis

In this chapter the basic differential equations that govern the flow of fluids have been developed. The Navier–Stokes equations, which can be compactly expressed in vector notation as

$$\rho\left(\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V}\cdot\nabla\mathbf{V}\right) = -\nabla p + \rho\mathbf{g} + \mu\nabla^2\mathbf{V} \tag{6.158}$$

along with the continuity equation

$$\nabla\cdot\mathbf{V} = 0 \tag{6.159}$$

are the general equations of motion for incompressible Newtonian fluids. Although we have restricted our attention to incompressible fluids, these equations can be readily extended to include compressible fluids. It is well beyond the scope of this introductory text to consider in depth the variety of analytical and numerical techniques that can be used to obtain both exact and approximate solutions to the Navier–Stokes equations. Students, however, should be aware of the existence of these very general equations, which are frequently used as the basis for many advanced analyses of fluid motion. A few relatively simple solutions have been obtained and discussed in this chapter to indicate the type of detailed flow information that can be obtained by using differential analysis. However, it is hoped that the relative ease with which these solutions were obtained does not give the false impression that solutions to the Navier–Stokes equations are readily available. This is certainly not true, and as previously mentioned there are actually very few practical fluid flow problems that can be solved by using an exact analytical approach. In fact, there are no known analytical solutions to Eq. 6.158 for flow past any object such as a sphere, cube, or airplane.

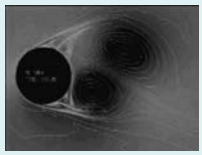
*Very few practical fluid flow problems can be solved using an exact analytical approach.*

Because of the difficulty in solving the Navier–Stokes equations, much attention has been given to various types of approximate solutions. For example, if the viscosity is set equal to zero, the Navier–Stokes equations reduce to Euler's equations. Thus, the frictionless fluid solutions discussed previously are actually approximate solutions to the Navier–Stokes equations. At the other extreme, for problems involving slowly moving fluids, viscous effects may be dominant and the nonlinear (convective) acceleration terms can be neglected. This assumption greatly simplifies the analysis, since the equations now become linear. There are numerous analytical solutions to these "*slow flow*" or "*creeping flow*" problems. Another broad class of approximate solutions is concerned with flow in the very thin boundary layer. L. Prandtl showed in 1904 how the Navier–Stokes equations could be simplified to study flow in boundary layers. Such "boundary layer solutions" play a very important role in the study of fluid mechanics. A further discussion of boundary layers is given in Chapter 9.

### 6.10.1 Numerical Methods



*V6.15 CFD example*



Numerical methods using digital computers are, of course, commonly utilized to solve a wide variety of flow problems. As discussed previously, although the differential equations that govern the flow of Newtonian fluids [the Navier–Stokes equations (6.158)] were derived many years ago, there are few known analytical solutions to them. With the advent of high-speed digital computers it has become possible to obtain numerical solutions to these (and other fluid mechanics) equations for many different types of problems. A brief introduction to computational fluid dynamics (CFD) is given in Appendix A.

Access to a program called FlowLab is available with this textbook. FlowLab is an educational version of a commercial CFD program. The backbone of FlowLab is the Fluent CFD package, which was used to create the numerical animations of flow past a spinning football referenced at the beginning of the chapter (V6.1 and V6.2). FlowLab provides a virtual laboratory for fluids experiments that makes use of the power of CFD, but with a student-friendly interface. Chapters 7–9 contain fluids problems that require the use of FlowLab to obtain the solutions.

CMS Cepcor EX 1046 Page 228
CMS Cepcor v. Sandvik IPR2024-01380

## Fluids in the News

**Fluids in the Academy Awards** A computer science professor at Stanford University and his colleagues were awarded a Scientific and Technical Academy Award for applying the Navier–Stokes equations for use in Hollywood movies. These researchers make use of computational algorithms to numerically solve the Navier–Stokes equations (also termed computational fluid dynamics, or CFD) and simulate complex liquid flows. The realism of the simulations has found application in the entertainment industry. Movie producers have used the power of these numerical tools to simulate flows from ocean waves in "Pirates of the Caribbean" to lava flows in the final duel in "Star Wars: Revenge of the Sith." Therefore, even Hollywood has recognized the usefulness of CFD.

## 6.11 Chapter Summary and Study Guide

*volumetric dilatation rate*
*vorticity*
*irrotational flow*
*continuity equation*
*stream function*
*Euler's equations of motion*
*ideal fluid*
*Bernoulli equation*
*velocity potential*
*potential flow*
*equipotential lines*
*flow net*
*uniform flow*
*source and sink*
*vortex*
*circulation*
*doublet*
*method of superposition*
*half-body*
*Rankine oval*
*Navier–Stokes equations*
*Couette flow*
*Poiseuille's law*

Differential analysis of fluid flow is concerned with the development of concepts and techniques that can be used to provide a detailed, point by point, description of a flow field. Concepts related to the motion and deformation of a fluid element are introduced, including the Eulerian method for describing the velocity and acceleration of fluid particles. Linear deformation and angular deformation of a fluid element are described through the use of flow characteristics such as the volumetric dilatation rate, rate of angular deformation, and vorticity. The differential form of the conservation of mass equation (continuity equation) is derived in both rectangular and cylindrical polar coordinates.

Use of the stream function for the study of steady, incompressible, plane, two-dimensional flow is introduced. The general equations of motion are developed, and for inviscid flow these equations are reduced to the simpler Euler equations of motion. The Euler equations are integrated to give the Bernoulli equation, and the concept of irrotational flow is introduced. Use of the velocity potential for describing irrotational flow is considered in detail, and several basic velocity potentials are described, including those for a uniform flow, source or sink, vortex, and doublet. The technique of using various combinations of these basic velocity potentials, by superposition, to form new potentials is described. Flows around a half-body, a Rankine oval, and around a circular cylinder are obtained using this superposition technique.

Basic differential equations describing incompressible, viscous flow (the Navier–Stokes equations) are introduced. Several relatively simple solutions for steady, viscous, laminar flow between parallel plates and through circular tubes are included.

The following checklist provides a study guide for this chapter. When your study of the entire chapter and end-of-chapter exercises has been completed you should be able to

- write out meanings of the terms listed here in the margin and understand each of the related concepts. These terms are particularly important and are set in ***italic bold, and color*** type in the text.
- determine the acceleration of a fluid particle, given the equation for the velocity field.
- determine the volumetric dilatation rate, vorticity, and rate of angular deformation for a fluid element, given the equation for the velocity field.
- show that a given velocity field satisfies the continuity equation.
- use the concept of the stream function to describe a flow field.
- use the concept of the velocity potential to describe a flow field.
- use superposition of basic velocity potentials to describe simple potential flow fields.
- use the Navier–Stokes equations to determine the detailed flow characteristics of incompressible, steady, laminar, viscous flow between parallel plates and through circular tubes.

Some of the important equations in this chapter are:

| | | |
|---|---|---|
| Acceleration of fluid particle | $\mathbf{a} = \frac{\partial \mathbf{V}}{\partial t} + u\frac{\partial \mathbf{V}}{\partial x} + v\frac{\partial \mathbf{V}}{\partial y} + w\frac{\partial \mathbf{V}}{\partial z}$ | **(6.2)** |
| Vorticity | $\zeta = 2\,\boldsymbol{\omega} = \nabla \times \mathbf{V}$ | **(6.17)** |
| Conservation of mass | $\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0$ | **(6.27)** |