Section 2.8

2.53 Cleaning processes involve the use of soaps and detergents to alter surface tension so that dirty surfaces are brought into contact with water, the universal solvent. Do you think soaps increase or decrease the surface tension? Why?

2.54 An air bubble in glycerin has a diameter of 2000 μm. What is the pressure difference across the surface of this bubble due to surface tension?

2.55 Calculate the height of capillary rise for water in a glass tube with $D = 0.5$ mm.

2.56. When a glass tube is inserted into liquid mercury, the depression is found to be 4 cm. Estimate the diameter of the tube.

2.57 What is the rise of water in a vertical crack formed by two glass plates 20 μm apart?

2.58 Calculate the capillary rise of water in a pair of the glass plates separated by 2 mm if the plates are inclined at 75° from the vertical.

Section 2.9

2.59 Water flows from a pipe at 6 m/s. What is the kinetic energy per unit mass of the water? What is the kinetic energy per unit volume of water?

2.60 A windmill extracts energy from air moving at 40 km/h. What is the total kinetic energy per unit volume of moving air?

2.61 A hydroelectric plant will employ a total elevation change of 350 ft. What is the gravitational potential energy change per unit volume of water?

2.62 What is the pressure potential energy stored in 75 L of water at 20 MPa?

3 CASE STUDIES IN FLUID MECHANICS

3.1 Introduction
3.2 Common Dimensionless Groups in Fluid Mechanics
3.3 Case Studies
3.3.1 Flow in a Round Pipe
3.3.2 Flow Through Area Change
3.3.3 Pump and Fan Laws
3.3.4 Flat Plate Boundary Layer
3.3.5 Drag on Cylinders and Spheres
3.3.6 Lift and Drag on Airfoils
3.4 Summary
Problems

3.1 INTRODUCTION

In Chapter 2 you learned how to combine your understanding of fluid and flow properties and a force balance based on Newton's second law to solve simple fluid mechanics problems. Here we focus our attention on some of the results that have been obtained by engineers for more complex fluid mechanics problems. We have selected some of these results to form the basis of a number of interesting case studies. In each case study, you will find a brief description of the flow field of interest, and one or more design formulas that can be used to calculate important quantities of engineering and design interest. These formulas rely primarily on results obtained by using experimental methods, and in particular on the dimensional analysis and modeling tools to be presented in Chapter 9. In some cases, the formulas can be developed or otherwise explained by means of the more sophisticated analysis tools you will also learn about in later chapters. In any case, the amount of information given in a case study is not unlike what you might find in an engineering handbook, and applying the material should not be difficult.

A twofold goal of this chapter is to expose you to interesting flow fields early in the text and to allow you to calculate some engineering characteristics of these flows at an early stage in the learning process. As we revisit these case studies in later chapters, our hope is that you will progress from a cautious first application of the case study results to a fuller understanding of the underlying flow fields. Furthermore, these results may help you better comprehend your laboratory course work.

CD/Video Library/Laminar and Turbulent Flow on a Flat Plate

At this point you might be wondering: Why do we need to rely on experimental results in fluid mechanics? Why not just use a better analytical model or a bigger computer to solve a flow problem? An answer to these questions lies in recognizing the difference between laminar and turbulent fluid flow. As the name implies, laminar flow involves the movement of fluid in "layers." As shown by the dye in the top of Figure 3.1, the motion of a fluid in laminar flow is orderly, often slow and steady, and generally amenable to observation, measurement, and prediction. Analytical and computational solutions to laminar flow problems are both feasible and common, and the need for experiments is often minimal. However, laminar flows are relatively rare both in nature and in engineering practice. This is because a laminar flow undergoes a transition (middle of Figure 3.1) and eventually becomes turbulent as flow speeds increase. Turbulent flow, as illustrated at the bottom of Figure 3.1, is encountered in almost all flows in nature and engineering practice. This type of flow consists of a chaotic, disordered, and unsteady motion of fluid that is generally difficult to visualize, measure, and predict. There are no analytical solutions for turbulent flow, and computational models of turbulence are limited in their applicability. Thus experimental results are necessary for engineering designs involving turbulent flows.

Although the future of fluid mechanics will undoubtedly be marked by an increasing dependence on computational solutions for both laminar and turbulent flows, models of turbulence and other physical processes of interest in fluid mechanics will continue to require calibration and verification by well chosen experiments.

In the case studies that follow, you will find frequent references to dimensionless groups. Examples of these groups include the Reynolds and Mach numbers. Simply put, a dimensionless group is an algebraic combination of the parameters describing a particular flow that proves to be both dimensionless as a whole and significant in terms of understanding the flow field. In fluid mechanics, the most important dimensionless group is called the Reynolds number. The Reynolds number of a flow, written as $Re = \rho V L/\mu$, is the product of density ρ, a fluid velocity scale V, and a length scale L, all divided by

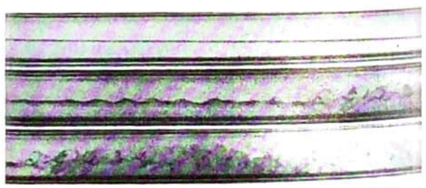


Figure 3.1 Dye injected into a pipe flow indicates laminar flow (top), transitional flow (middle), and turbulent flow (bottom).

viscosity, μ. In a given unit system Re is dimensionless, which we can demonstrate by writing the dimensions of each quantity in the Reynolds number to obtain

$$\{Re\} = \frac{\{\rho\}\{V\}\{L\}}{\{\mu\}} = \frac{[(M/L^3)(L/t)(L)]}{M/Lt} = \frac{ML^{-1}t^{-1}}{ML^{-1}t^{-1}} = 1$$

Flows with large Reynolds numbers are usually turbulent, an important consideration in understanding how a flow will behave.

3.2 COMMON DIMENSIONLESS GROUPS IN FLUID MECHANICS

FE

As you learn more about fluid mechanics you will discover that some dimensionless groups occur repeatedly in analyses of fluid mechanics problems. Most dimensionless groups have been given names in honor of their discovers or other prominent individuals in the study of fluid mechanics. It is important to become familiar with the common dimensionless groups to ensure that you present the results of your analysis in the form other engineers expect. Also, the numerical values of these traditional dimensionless groups are used in the classification of a particular fluid mechanics problem, in the selection of efficient solution techniques, and to compare results with those obtained by investigations of similar flows. Let us take a look at some of the more important dimensionless groups in fluid mechanics and learn about their relationship to various physical phenomena.

Reynolds Number: As discussed earlier, the Reynolds number, the most important dimensionless group in fluid mechanics, is defined to be

$$Re = \frac{\rho V L}{\mu} \tag{3.1}$$

where ρ is the fluid density, V is a fluid velocity scale, L is a length scale, and μ is the fluid viscosity. This dimensionless group is named in honor of Osborne Reynolds (1842–1912), a noted pioneer in the study of pipe flow and turbulence. The velocity and length scales involved in its definition are illustrated for internal and external flows in Figure 3.2.

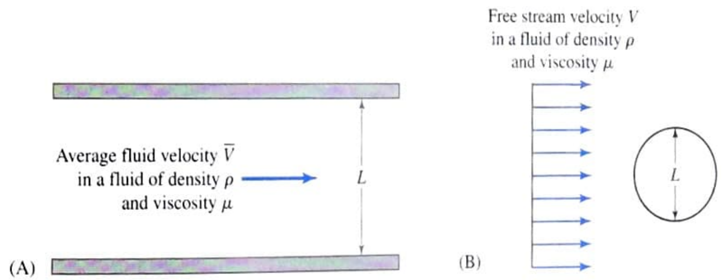


Figure 3.2 Velocity and length scales used in defining Re for examples of (A) internal flow and (B) external flow.

CD/History/Osborne Reynolds

It is important for you to have an understanding of the physical significance of the Reynolds number. One way to interpret Re is to think of it as a ratio of inertial to viscous forces in a fluid flow. An inertial force can be written using Newton's second law as $\mathbf{F} = M\mathbf{a}$. If we recognize that mass is equal to the product of density and volume and write the equation in terms of dimensions we find:

$$\{\mathbf{F}_I\} = \{M\}\{\mathbf{a}\} = \{\rho L^3\}\{Vt^{-1}\} = \{\rho L^3 Vt^{-1}\} = \{\rho V^2 L^2\} \tag{3.2}$$

where we have made use of the fact that the dimensions for velocity are $\{Lt^{-1}\}$. To generate a similar expression for the viscous force, we begin with Newton's law of viscosity, $\tau = \mu(du/dy)$, in dimensional form:

$$\{\tau\} = \{\mu\}\{VL^{-1}\} \tag{3.3}$$

But we require an expression for the viscous force, which is equal to the shear stress multiplied by the area over which that stress acts. Thus,

$$\{\mathbf{F}_V\} = \{\tau A\} = \{\mu\}\{VL^{-1}\}\{L^2\} = \{\mu VL\} \tag{3.4}$$

If we divide Eq. 3.2 by Eq. 3.4 we obtain:

$$\frac{\{\mathbf{F}_I\}}{\{\mathbf{F}_V\}} = \frac{\{\rho V^2 L^2\}}{\{\mu VL\}} = \left\{\frac{\rho VL}{\mu}\right\} \tag{3.5}$$

Since the right-hand side of this equation is equivalent to the Reynolds number, we are justified in interpreting Re as a ratio of inertial to viscous forces.

Except within a thin boundary layer near solid surfaces, high Re flows are dominated by inertial forces and are usually turbulent. Low Re flows, or creeping flows, are highly viscous in character and laminar. Flows at intermediate Re are often laminar, with inertial and viscous forces both playing significant roles in determining flow structure throughout the flow field.

CD/Video Library/Flow Past a Cylinder

The effect of Re on flow structure for flow over a cylinder is illustrated in Figure 3.3. At very low values, $Re = 0.038$ (Figure 3.3A), the inertia is so small that fluid particles easily flow around the cylinder while remaining in their laminar layers. At $Re = 19$ (Figure 3.3B) the inertia has increased to the point that some fluid particles cannot "make the turn," like Formula 1 racecar drivers who spin out going too fast through a curve. This phenomenon is called flow separation. As Re increases to 55 (Figure 3.3C), the separation bubble is pushed downstream. Thus Re indicates the presence of structural changes in the flow field. In Chapters 12 and 14 we will discuss in greater detail the flow over a cylinder and the interesting results that occur at higher Reynolds numbers.

Before we continue with an example, let us sound a note of caution concerning the interpretation of Re. It would be a gross simplification to consider Re to be only the ratio

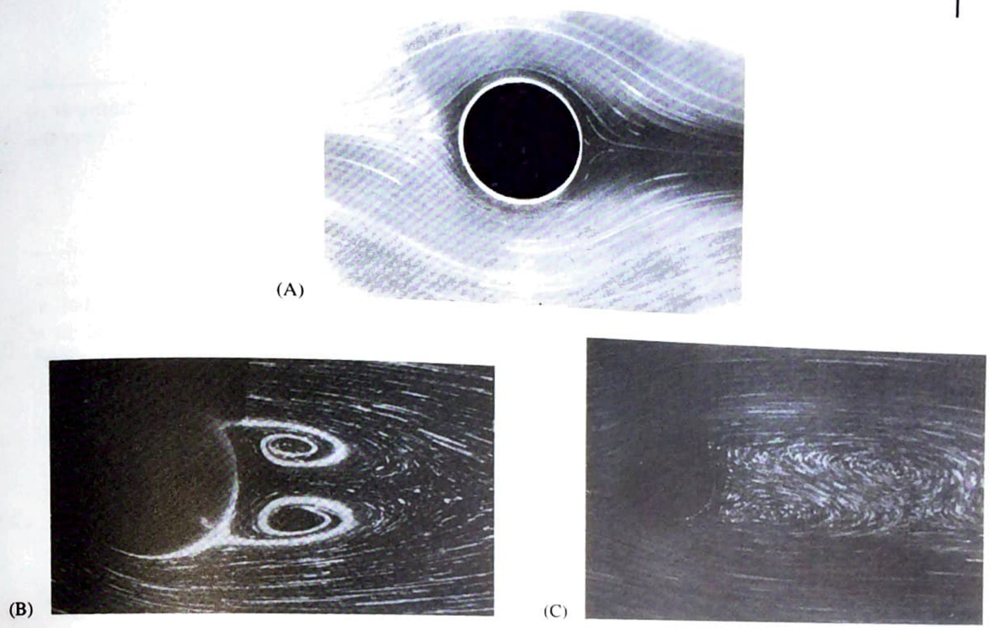


Figure 3.3 Flow field over a cylinder at (A) $Re = 0.038$, (B) $Re = 19$, and (C) $Re = 55$.

CD/Dynamics/Reynolds Number: Inertia and Viscosity

of inertial to viscous forces. For example, $Re = 1$ should not be interpreted as inertial and viscous forces being equal. The choice of length and velocity scales used in Re have most often been chosen for convenience, not physical significance. Thus Re should be compared and interpreted for a single flow field only, not between flow fields. Consider the critical Re_{cr}, where the transition of a laminar flow to turbulent flow is an important application of the Reynolds number: Re_{cr} can differ by several orders of magnitude between an internal flow and an external flow. Thus the physical meaning cannot be precisely the same.

Mach Number: The Mach number, named in honor of Ernst Mach (1838–1916), a pioneer in the study of high speed flow, was introduced in Section 2.6.1 and is defined to be the ratio of fluid velocity V to c, the speed of sound in the fluid. Thus the Mach number is given by

$$M = \frac{V}{c} \tag{3.6}$$

EXAMPLE 3.1

A good serve from a professional tennis player may reach 190 km/h. If the diameter of a tennis ball is approximately 6.5 cm, what is the Reynolds number for the flow over the ball?

SOLUTION

The Reynolds number for a tennis ball is found using Eq. 3.1: $Re = \rho V L/\mu$. The characteristic velocity is $V = 190$ km/h, and we will use the diameter of the tennis ball as the characteristic length scale so that $L = 6.5$ cm. The density and viscosity of air at STP are found in Appendix A to be $\rho = 1.204$ kg/m^3 and $\mu = 1.82 \times 10^{-5}$ (N-s)/m^2. Substituting these values into the expression for Re and using the appropriate unit conversion factors found in Appendix C yields:

$$Re = \frac{\rho V L}{\mu}$$

$$= \frac{1.204\ \text{kg/m}^3 \left[(190\ \text{km/h})\left(\frac{1\ \text{h}}{3600\ \text{s}}\right)\left(\frac{1000\ \text{m}}{1\ \text{km}}\right)\right]\left[(6.5\ \text{cm})\left(\frac{1\ \text{m}}{100\ \text{cm}}\right)\right]}{1.82 \times 10^{-5}[(\text{N-s})/\text{m}^2]\left[\frac{1\ (\text{kg-m})/\text{s}^2}{1\ \text{N}}\right]}$$

$$Re = 2.27 \times 10^5$$

This is a high value of Re (we will define "high value" later in the context of specific types of flows); thus for the movement of the tennis ball through the air, inertial forces are significant and viscous forces will be important only in the boundary layer.

The Mach number provides a measure of the effects of compressibility on a flow. An incompressible fluid, i.e., a liquid, has $M \approx 0$ because the sound speed is very large in comparison to a typical liquid flow speed. Gases tend to flow much faster than liquids relative to their sound speeds, hence Mach number is of great interest in classifying the flow of a gas such as air. When air flows with a small Mach number, nominally $M < 0.3$, the air behaves like an incompressible fluid. Thus a flow with $M < 0.3$ is called an incompressible flow. A flow with a Mach number greater than this is termed a compressible flow, since variations in the density of the air must be accounted for. We further classify compressible flows according to Mach number as subsonic if $M < 1$ and supersonic if $M > 1$. Flows near the sonic velocity have unique characteristics such that $0.9 < M < 1.2$ flows are classified as transonic. Flows at very high velocity, $M > 5$, are termed hypersonic.

CD/Video Library/Shock Waves

Figure 3.4 A ship's wake, photograph from the space shuttle. The wake trails several miles behind the ship.

Froude Number: The Froude number is defined to be the ratio

$$Fr = \frac{V}{\sqrt{gL}} \tag{3.7}$$

where V is a fluid velocity scale, L is a length scale, and g is the acceleration of gravity. This dimensionless group is named in honor of William Froude (1810–1879), who used models to perform pioneering studies of the drag on ships due to wave making (Figure 3.4).

The Froude number can be interpreted as the ratio of inertial forces to gravitational forces. From Eq. 3.2 we know that the dimensions for the inertial force can be written as $\{\mathbf{F}_I\} = \{M\}\{a\} = \{\rho L^3\}\{Vt^{-1}\} = \{\rho L^3 V t^{-1}\} = \{\rho V^2 L^2\}$. Similarly, the dimensions for the gravitational force are:

$$\{\mathbf{F}_G\} = \{M\}\{g\} = \{\rho L^3\}\{g\} \tag{3.8}$$

Taking the ratio of the inertial force to the gravitational force yields:

$$\frac{\{\mathbf{F}_I\}}{\{\mathbf{F}_G\}} = \frac{\{\rho V^2 L^2\}}{\{\rho L^3 g\}} = \left\{\frac{V^2}{gL}\right\} \tag{3.9}$$

Since this ratio is clearly dimensionless (units of force in the numerator and denominator), the square root of the ratio is also dimensionless, and we see that the Froude number can in fact be interpreted as a ratio of inertial to gravitational forces.

The Froude number is important in ship hydrodynamics, in the study of water waves, and in the classification of free surface flows, which do not involve a moving body. In such cases the length scale is often taken to be the liquid depth. Free surface flows are of interest to civil engineers involved in large-scale projects such as canals, weirs, spillways, and waterways of all kinds.

CD/Video Library/River Flow

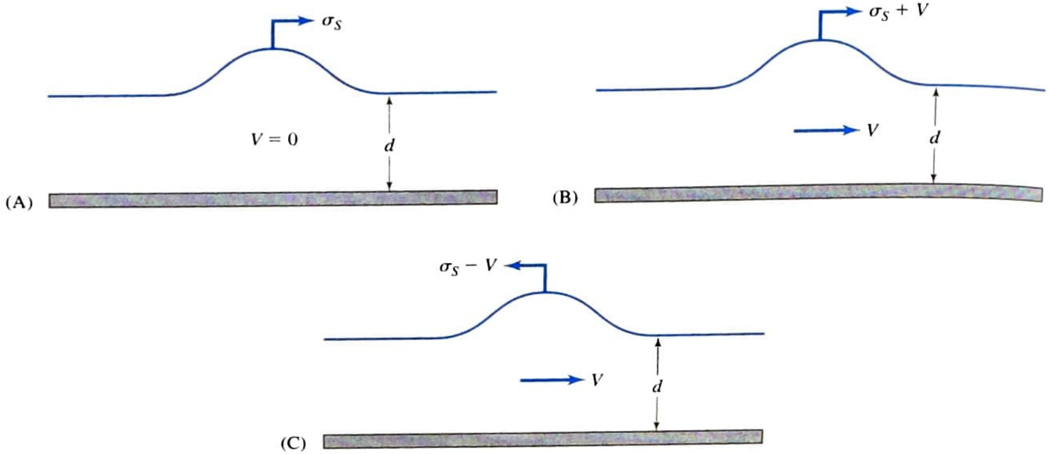


Figure 3.5 Infinitesimal wave moves (A) to the right on stationary fluid (B) to the right on fluid moving to the right and (C) to the left on fluid moving to the right. To an observer moving with the fluid, the wave speed is σ_S in both cases. For an observer on the shore, the wave speed is σ_S for (A), $\sigma_S + V$ for (B), and $\sigma_S - V$ for (C).

Free surface flows with $Fr < 1$ are said to be subcritical; those with $Fr > 1$ are supercritical, and a flow at $Fr = 1$ is said to be critical. An understanding of the physical phenomenon behind the use of the adjective critical in free surface flows can be gained by noting that the wave propagation speed of an infinitesimal wave in stationary water of depth d is

$$\sigma_S = \sqrt{gd} \tag{3.10a}$$

Here σ_S is the speed at which the wave moves relative to the water (see Figure 3.5A). The Froude number in a problem involving wave propagation in water moving at speed V is

$$Fr = \frac{V}{\sqrt{gd}} = \frac{V}{\sigma_S} \tag{3.10b}$$

If the water is moving at a velocity V to the right, then, as shown in Figure 3.5B, a wave moving to the right (in the flow direction) travels at a velocity $\sigma_S + V$, and to the left at a velocity $\sigma_S - V$ (Figure 3.5C). If the water is moving at a velocity $V = \sigma_S$, then a wave cannot propagate upstream. This is the critical water speed for a free surface flow of depth d, and Eq. 3.10b shows that this speed corresponds to $Fr = 1$. In a subcritical flow, $Fr < 1$ and $V < \sigma_S$, so waves may travel in both directions. In a supercritical flow, $Fr > 1$ and $V > \sigma_S$, so waves can travel downstream only.

Weber Number: The Weber number is an important dimensionless group in flow problems involving surface tension. It is named after Moritz Weber (1871–1951), who worked on problems involving capillary effects. In a problem involving a moving liquid, the Weber number is defined by

$$We = \frac{\rho V^2 L}{\sigma} \tag{3.11a}$$

where σ is the surface tension, and V and L are velocity and length scales, respectively. The Weber number in a moving liquid can be thought of as the ratio of inertial force to surface tension (or equivalently a ratio of kinetic energy to surface energy). In a problem involving liquid at rest in a gravitation field g, the importance of surface tension can be characterized by defining the Weber number as

$$We = \frac{\rho g L^2}{\sigma} \tag{3.11b}$$

In this case We may be viewed as the ratio of gravitational forces to surface tension (or equivalently, gravitational potential energy to surface energy). Surface tension effects are only important when $We \ll 1$. Otherwise the effects of surface tension can be safely

EXAMPLE 3.2

The flow in a wide tidal channel separating a back bay from the ocean may approach 0.75 m/s. If the tidal channel is 6 m deep, what are the Reynolds and Froude numbers for the flow?

SOLUTION

The Reynolds and Froude numbers for this flow are found using Eqs. 3.1 and 3.10b: $Re = \rho VL/\mu$ and $Fr = V/\sqrt{gd} = V/\sigma_S$. The characteristic velocity is $V = 0.75$ m/s and the depth of the tidal channel serves as the characteristic length scale such that $L = d = 6$ m. We assume conditions at 20°C for water and use Appendix A to find: $\rho = 998$ kg/m³ and $\mu = 1 \times 10^{-3}$ (N-s)/m². Also note that $g = 9.81$ m/s². Substituting these values into the expressions for Re and Fr and using the definition of a newton as a unit conversion factor, we have:

$$Re = \frac{\rho VL}{\mu} = \frac{(998\ \text{kg/m}^3)(0.75\ \text{m/s})(6\ \text{m})}{1 \times 10^{-3}[(\text{N-s})/\text{m}^2]\left[\frac{1\ (\text{kg-m})/\text{s}^2}{1\ \text{N}}\right]}$$

$$= 4.49 \times 10^6$$

$$Fr = \frac{V}{\sqrt{gd}} = \frac{0.75\ \text{m/s}}{\sqrt{(9.81\ \text{m/s}^2)(6\ \text{m})}} = 0.098$$

A Reynolds number of this magnitude would result in turbulent flow in the channel, and since the Froude number is less than one, we can conclude that the flow is subcritical.

EXAMPLE 3.3

Water flows from a 1 mm diameter orifice at 4 m/s. Is it likely that surface tension effects will be important in this application?

SOLUTION

The Weber number for this flow is found by using Eq. 3.11a: $We = \rho V^2 L/\sigma$. The characteristic velocity is $V = 4$ m/s, and the diameter of the orifice serves as the characteristic length scale so that $L = 1$ mm $= 0.001$ m. Assuming conditions at 20°C for water, we use Appendix A to find $\rho = 998$ kg/m^3 and $\sigma = 0.073$ N/m. Note that we have used the surface tension for a water–air interface because we are assuming that water exits the orifice into air. Substituting the appropriate values into the expressions for *We* and using the definition of a Newton as a unit conversion factor yields:

$$We = \frac{\rho V^2 L}{\sigma} = \frac{(998\ \text{kg/m}^3)(4\ \text{m/s})^2(0.001\ \text{m})}{(0.073\ \text{N/m})\left[\dfrac{1\ (\text{kg-m})/\text{s}^2}{1\ \text{N}}\right]} = 219$$

Since $We \gg 1$, we can safely neglect surface tension effects in this application.

The Bond number $B = [g(\rho_1 - \rho_2)D^2]/\sigma$ is used to characterize problems involving fluid droplets or bubbles of density ρ_1 immersed in another fluid of density ρ_2. It can be thought of as a measure of the ratio of buoyancy force to surface tension force.

ignored. Can you confirm that the Weber number for the capillary rise of water ($\sigma = 0.073$ N/m) in a round glass tube 1 mm in diameter is $We = 0.134$? How large would the diameter of the tube need to be for you to predict that capillary rise would be negligible?

Euler Number: The Euler number is defined to be

$$Eu = \frac{p - p_0}{\frac{1}{2}\rho V^2} \tag{3.12a}$$

where $p - p_0$ is the difference between a local value of pressure and that at some reference location. Leonhard Euler (1707–1783) was a great mathematician who first derived many fundamentals of fluid mechanics. The Euler number can be interpreted as a measure of the ratio of pressure force to inertial force. A number of variations on the Euler number appear in fluid mechanics. In aerodynamics, the pressure difference in the Euler number refers to the upstream static pressure p_∞, and the Euler number then becomes the pressure coefficient

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho V^2} \tag{3.12b}$$

The Euler number does not have the great physical significance of the Mach or Froude numbers; however, as with all dimensionless groups, it allows for the compact communication of data.

Figure 3.6 The Karman vortex street in the wake of a cylinder.

Slender structures such as suspended power transmissions lines, struts on small airplanes, and smokestacks are known to have natural vibration frequencies that can be calculated by using techniques from structural mechanics. When the natural vibrational frequency of a structure coincides with the frequency of the flow-induced Karman vortices, a condition known as resonance develops. During resonance, the amplitude of the structural vibrations can increase significantly. The Karman vortices are implicated in the wind-induced failure of the Tacoma Narrows suspension bridge in 1940 (Figure 3.7); however, there is still disagreement about the precise cause of the disaster.

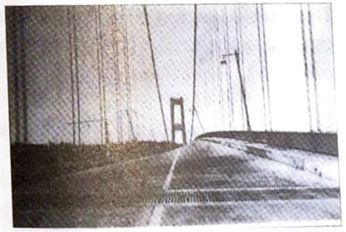


Figure 3.7 The Tacoma Narrows Bridge shortly before its collapse in 1940. The sidewalk to the right is over 28 ft above the one to the left.

Strouhal Number: The Strouhal number, which is defined as

$$St = \frac{\omega L}{V} \tag{3.13}$$

is important in problems involving flow oscillations in which the frequency of the oscillations is ω. The Strouhal number can be interpreted as the ratio of vibrational velocity to translational velocity. Many flows over bluff bodies develop oscillations. The most well known is the generation of Karman vortices that are shed periodically from the wake of a cylinder (see Figure 3.6). In this case it is known that over a range of Reynolds numbers $10^2 \le Re \le 10^7$, the Strouhal number is approximately 0.21 if the frequency of vortex shedding is measured in radians per second. Thus *St* can be used to predict the expected frequency of vortex shedding. Vincenz Strouhal (1850–1922) did pioneering work on the vibration or "singing" of wires due to this effect.

CD/Video Library/Tacoma Narrows Bridge Disaster

Prandtl Number: Fluid mechanics is integrally related to the field of convective heat transfer, which is the study of heat transport processes in fluid flows. In fact, an important dimensionless number used in convective heat transfer is the Prandtl number, named after Ludwig Prandtl (1875–1953), one of the giants of twentieth-century fluid mechanics. The Prandtl number

$$Pr = \frac{\nu}{\alpha} \tag{3.14}$$

EXAMPLE 3.4

A smokestack at a power plant is 9 ft in diameter. The natural vibrational frequency for this structure is known to be 7 rad/s. Calculate the wind velocity that would induce Karman vortex shedding at a frequency of 7 rad/s and comment on the likelihood of wind-induced resonance leading to structural failure.

SOLUTION

The Strouhal number for this flow is found by using Eq. 3.13: $St = \omega L/V$. Resonance occurs when the wind-induced vortex shedding frequency calculated in this way corresponds to the natural frequency of the structure. Thus, we must determine the wind velocity at which the vortex shedding frequency is $\omega = 7$ rad/s. The diameter of the smokestack serves as the characteristic length scale, so that is $L = 9$ ft. Assuming that the critical Strouhal number is 0.21 and substituting the appropriate values into Eq. 3.7 after solving for V yields:

$$V = \frac{\omega L}{St} = \frac{(7\ \text{rad/s})(9\ \text{ft})}{0.21} = 300\ \text{ft/s}$$

$$V = (300\ \text{ft/s})\left(\frac{1\ \text{mph}}{1.467\ \text{ft/s}}\right) = 204\ \text{mph}$$

Since it is unlikely that the smokestack will experience wind speeds in excess of 200 mph, one need not be concerned about vortex shedding leading to structural failure.

is the ratio of kinematic viscosity ν to thermal diffusivity α. Heat transfer between a solid surface and a fluid that is in motion due to external means (e.g., a fan or pump) is called forced convection. In cases of forced convection the heat transfer rate depends on the Prandtl and Reynolds numbers.

Other Dimensionless Groups: There are many more named dimensionless groups in fluid mechanics as well as some that are simply physically descriptive and not named after a particular historical figure. For example, the dimensionless group known as the relative roughness e/D occurs in pipe flow. This group is defined as the ratio of the average height of the pipe wall roughness e to inside diameter of the pipe D.

3.3 CASE STUDIES

The following case studies represent a varied selection of the type of information available to engineers. Our emphasis in selecting these particular studies is their broad applicability in engineering design. Engineers use results like these to successfully practice design after a single course in fluid mechanics. Each of these flow problems has been investigated theoretically, but the majority of useful results have been obtained empirically. If you are careful to apply the formulas developed in a case study in the

original context, your analysis will provide answers to design questions within the range of normal engineering accuracy.

The empirical results presented here give the impression of simplicity because they involve only global characteristics of the flow field. Notice that nothing is said in any of the case studies about local details of the flow field. Actually, all these flow fields are quite complex. Our purpose in designating these problems as case studies is to give you an early introduction to the design aspects of engineering fluid mechanics and to emphasize the relevance of the subsequent theoretical chapters to developing a better understanding of the fluid mechanics of engineering problems. We do this by revisiting these same flow problems in later chapters and using the new tools we have developed to better understand the sources of the case study formulas.

3.3.1 Flow in a Round Pipe

FE

Pumping a fluid through a pipe or duct is a common, and arguably the most important, application of fluid mechanics. Society could not function without the water, steam, air, natural gas, oil, and other hydrocarbons transported via piping systems. Our homes and workplaces depend on central heating, ventilation, and air conditioning. Indeed, social historians in the United States have commented that the migration of people to the southern states after World War II would not have occurred without the universal availability of air conditioning. Virtually all engines require delivery of fuel, lubricant, and coolant through a pipe or hose. Can you think of other important technical applications of these systems? Does pipe flow also occur in biological systems?

In this first case study we consider steady, fully developed incompressible flow in a straight, horizontal, round pipe as shown in Figure 3.8. The adjective "steady" implies that the flow is unchanging in time, and "fully developed" implies that the flow is the same at every location along the pipe. "Incompressible" here implies that the fluid density is constant. This type of flow commonly occurs in the movement of liquid through relatively long pipes subjected to a continuous pumping action. In later chapters we show how to handle a rectangular, square, or other shape for the pipe or duct, as well as flows that are not steady or fully developed. Low speed gas flow occurs at constant density, so the techniques developed in this case study may also be used to analyze flow of air in heating, ventilating, and air-conditioning systems.

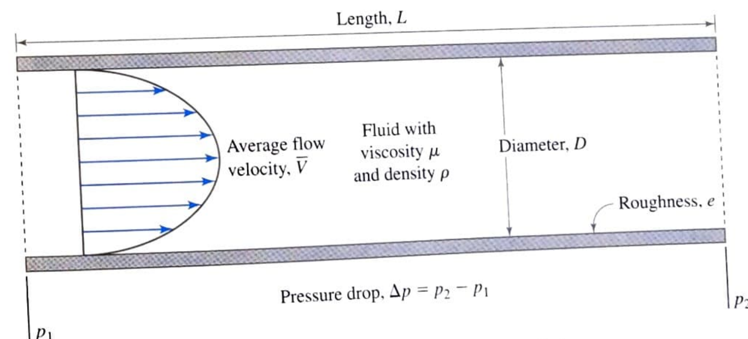


Figure 3.8 Variables for fully developed flow in a horizontal pipe.

Note that this is a transcendental equation and will require iteration to determine the friction factor for known values of relative roughness and Reynolds number. We may also determine the friction factor by means of the Chen equation, another empirically based relationship:

$$f = \left\{-2.0\log\left[\frac{e/D}{3.7065} - \frac{5.0452}{Re}\log\left(\frac{(e/D)^{1.1098}}{2.8257} + \frac{5.8506}{Re^{0.8981}}\right)\right]\right\}^{-2} \quad (3.19b)$$

EXAMPLE 3.5

Normal saline solution flows with an average velocity of $\bar{V} = 0.5$ mm/s in a 2 m length of polymer tubing before entering a patient's arm intravenously. If the inside diameter of tubing is $D = 2$ mm, determine the friction factor, volume flowrate, and pressure drop in the tubing. Assume that the saline solution has the same properties as water, and that the IV line is horizontal.

SOLUTION

We are asked to determine f, Q, and Δp for a flow of saline through a horizontal tube with $L = 2$ m and $D = 2$ mm. This problem can be solved without the aid of a sketch. We are given $\bar{V} = 0.5$ mm/s and assume conditions at 20°C for water. From Appendix A we find $\rho = 998$ kg/m^3 and $\mu = 1 \times 10^{-3}$ (N-s)/m^2. The problem is solved by using Eq. 3.15 $[\Delta p = \rho f(L/D)(\bar{V}^2/2)]$ to find Δp and Eq. 3.16 to find Q. We begin, however, by returning to Eq. 3.1 ($Re = \rho \bar{V} L/\mu$) to determine Re and then using either Eq. 3.18 or 3.19 to determine the appropriate friction factor for the calculated value of Re.

Substituting the foregoing values into the expression for Re yields:

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{(998\ \text{kg/m}^3)(0.5 \times 10^{-3}\ \text{m/s})(2 \times 10^{-3}\ \text{m})}{[1 \times 10^{-3}\ (\text{N-s})/\text{m}^2]\left[\frac{1\ (\text{kg-m})/\text{s}^2}{1\ \text{N}}\right]} = 1$$

Since $Re < 2300$, the flow is laminar, and we can use Eq. 3.18 to find the friction factor: $f = 64/Re = \frac{64}{1} = 64$. Next, use Eq. 3.15 to solve for the pressure drop:

$$\Delta p = \rho f \frac{L}{D}\frac{\bar{V}^2}{2} = (998\ \text{kg/m}^3)(64)\left(\frac{2\ \text{m}}{0.002\ \text{m}}\right)\left[\frac{(0.5 \times 10^{-3}\ \text{m/s})^2}{2}\right]$$
$$= 7.98\ \text{N/m}^2 = 7.98\ \text{Pa}$$

Finally we use Eq. 3.16 to find the volume flowrate:

$$Q = \bar{V}A = \bar{V}\frac{\pi D^2}{4} = (0.5 \times 10^{-3}\ \text{m/s})\left[\frac{\pi(2 \times 10^{-3}\ \text{m})^2}{4}\right] = 1.57 \times 10^{-9}\ \text{m}^3/\text{s}$$

A very small volume flowrate like this can also be expressed in milliliters per minute:

$$Q = (1.57 \times 10^{-9}\ \text{m}^3/\text{s})\left(\frac{10^2\ \text{cm}}{\text{m}}\right)^3\left(\frac{1\ \text{mL}}{1\ \text{cm}^3}\right)\left(\frac{60\ \text{s}}{1\ \text{min}}\right) = 0.09\ \text{mL/min}$$

EXAMPLE 3.6

Gasoline flows with an average velocity of $\bar{V} = 4$ ft/s in a horizontal steel pipe of length $L = 100$ ft with an inside diameter $D = 1$ in. The pipe connects the bulk storage tank to the pump at a gas station as shown in Figure 3.10. Determine the friction factor, volume flowrate, pressure drop, and pump horsepower required for this flow if the relative roughness of the pipe is $e/D = 0.001$. The fluid properties are $\rho = 42.45$ lb$_m$/ft^3 and $\mu = 1.96 \times 10^{-4}$ lb$_m$/(ft-s).

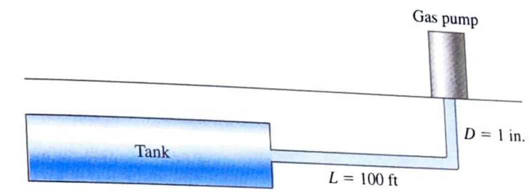


Figure 3.10 Schematic of gas station for Example 3.6.

SOLUTION

We are asked to determine f, Q, Δp, and P required for a flow of gasoline through a horizontal pipe with $L = 100$ ft and $D = 1$ in. Figure 3.10 is an adequate sketch of the flow situation. We are given $\bar{V} = 4$ ft/s, $\rho = 42.45$ lb$_m$/ft^3, and $\mu = 1.96 \times 10^{-4}$ lb$_m$/(ft-s). The problem is solved by using Eq. 3.15 $[\Delta p = \rho f(L/D)(\bar{V}^2/2)]$ to find Δp, Eq. 3.16 ($Q = \bar{V}A$) to find Q, and Eq. 3.17 ($P = Q\Delta p$) to find P. We begin of course by using Eq. 3.1 ($Re = \rho \bar{V} L/\mu$) to determine Re and then use either Eq. 3.18 or 3.19 to determine the friction factor.

Substituting appropriate values into the expression for Re yields:

$$Re = \frac{\rho \bar{V} D}{\mu} = \frac{(42.45\ \text{lb}_m/\text{ft}^3)(4\ \text{ft/s})(1\ \text{in.})(1\ \text{ft}/12\ \text{in.})}{1.96 \times 10^{-4}\ \text{lb}_m/(\text{ft-s})} = 7.22 \times 10^4$$

Since $Re > 2300$, the flow is turbulent and we must use Eq. 3.19a or 3.19b to find the friction factor. Choosing Eq. 3.19a and substituting $e/D = 0.001$ and $Re = 72{,}200$ gives:

$$\frac{1}{\sqrt{f}} = -2.0\log\left(2.7 \times 10^{-4} + \frac{3.48 \times 10^{-5}}{\sqrt{f}}\right)$$

Using repeated hand calculations (painful), a spreadsheet (still time-consuming), or a symbolic manipulator program (good idea) we find $f = 0.023$.

Next, use Eq. 3.15 to solve for the pressure drop.

$$\Delta p = \rho f \frac{L}{D}\frac{\bar{V}^2}{2} = (42.45\ \text{lb}_m/\text{ft}^3)(0.023)\left[\frac{100\ \text{ft}}{(1\ \text{in.})(1\ \text{ft}/12\ \text{in.})}\right]\left[\frac{(4\ \text{ft/s})^2}{2}\right]$$

$$= 9373\ \text{lb}_m/(\text{ft-s}^2)$$

To obtain Δp in common units, we use the definition of g_c as a unit conversion factor:

$$\Delta p = 9373\ \text{lb}_m/(\text{ft-s}^2)\left(\frac{1\ \text{lb}_f\text{-s}^2}{32.2\ \text{lb}_m\text{-ft}}\right) = 291\ \text{lb}_f/\text{ft}^2$$

$$\Delta p = 291\ \text{lb}_f/\text{ft}^2\left(\frac{1\ \text{ft}}{12\ \text{in.}}\right)^2 = 2.0\ \text{psi}$$

Next we use Eq. 3.16 to find the volume flowrate:

$$Q = \bar{V}A = \bar{V}\frac{\pi D^2}{4} = (4\ \text{ft/s})\left[\frac{\pi(1\ \text{in.})^2(1\ \text{ft}/12\ \text{in.})^2}{4}\right] = 2.18 \times 10^{-2}\ \text{ft}^3/\text{s}$$

$$Q = (2.18 \times 10^{-2}\ \text{ft}^3/\text{s})\left(\frac{1\ \text{gal}}{0.13368\ \text{ft}^3}\right)\left(\frac{60\ \text{s}}{1\ \text{min}}\right) = 9.8\ \text{gal/min}$$

Finally, use Eq. 3.17 to find the pump horsepower:

$$P = Q\Delta p = (2.18 \times 10^{-2}\ \text{ft}^3/\text{s})(291\ \text{lb}_f/\text{ft}^2)\left[\frac{1\ \text{hp}}{550\ (\text{ft-lb}_f)/\text{s}}\right] = 0.01\ \text{hp}$$

The volume flowrate of ~10 gal/min seems reasonable for a gasoline pump at a service station. The power required to operate the pump is small because we are considering only the pressure drop needed to overcome friction. In most cases the pump must also produce enough pressure to overcome the hydrostatic pressure variation due to elevation change as well as losses due to valves and other fittings in the pipe network. We will discuss these additional aspects of piping system design in Chapter 13. This problem can also be solved by using the Chen equation, Eq. 3.19b, to estimate the friction factor.

3.3.2 Flow Through Area Change

If you examine a pipe or duct system in a building, it is evident that changes in the cross-sectional area of a flow passage are quite common (Figure 3.11). The area change is often abrupt owing to space limitations, and turbulent flow is the norm in these systems. In this section we provide a method for using a loss coefficient to estimate the frictional pressure drop in steady incompressible turbulent flow through a sudden area change. Frictional pressure drops also occur when flow passes through nozzles, diffusers, bends, valves, entrances, exits, and other features of a pipe or duct system. Methods to compute the pressure drop through these elements will also be described later (see Chapter 13).

In examining the flow through an area change it is critical to realize that even in the absence of frictional effects, there is always a pressure change due to the change in the speed of the flow as it passes through the area change. (This is the change in pressure predicted by the Bernoulli equation as discussed in Chapter 2.) The total change in pressure as a flow passes through an area change may therefore be thought of as the sum of a pressure change associated with the change in average flow velocity (which may be either positive or negative depending on whether the flow slows down or speeds up) and a frictional pressure drop (a negative pressure change). We model this effect in turbulent flow as

$$p_2 - p_1 = \left[\tfrac{1}{2}\rho(\bar{V}_1^2 - \bar{V}_2^2)\right] - \Delta p_F \tag{3.20}$$

where p_2 is the downstream pressure, p_1 is the upstream pressure, and Δp_F is the frictional pressure loss. The velocities $\bar{V}_1$ and $\bar{V}_2$ in this formula are the average velocities in the upstream and downstream sections. We can calculate Δp_F by using empirical results. Note from Eq. 3.16 that since the same volume flowrate passes through each section the average velocities are related by

$$\bar{V}_1 A_1 = \bar{V}_2 A_2 \tag{3.21}$$

Did you recognize that if the frictional pressure drop is set to zero in Eq. 3.20, this equation becomes identical to Bernoulli's equation (Eq. 2.11) for a flow along a horizontal path? Notice also how the empirical model here (Eq. 3.20) builds on an earlier ideal result by adding a term to account for friction.

Now consider what happens in the idealized case of a frictionless flow through an area decrease. Since the frictional pressure loss Δp_F is assumed to be zero, Eqs. 3.20 and 3.21 show that the value of the pressure downstream is less than that upstream because the area decrease causes the flow to speed up. Conversely, for an increase in area the value of the pressure downstream is greater than that upstream because the flow slows down in the larger area downstream. Equation 3.20 shows that the effect of friction is to cause a lower pressure downstream than the ideal result irrespective of the area change.

The four basic types of cross-sectional area change are shown in Figure 3.12. As noted earlier, flows in systems of engineering interest usually have high Reynolds numbers and are turbulent. Because the section of a pipe or duct in which area change occurs is often relatively short, the portion of the frictional pressure loss due to viscous effects at the walls is negligible in comparison to the loss caused by turbulence. Thus, fluid viscosity is not an important parameter in these flows. Observation suggests that for

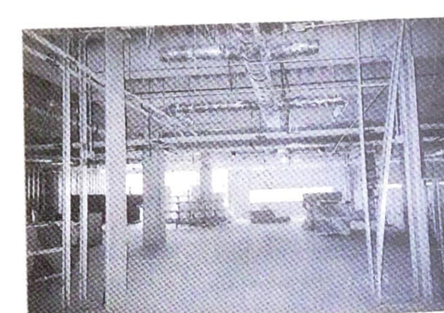


Figure 3.11 Ductwork system with several area changes.

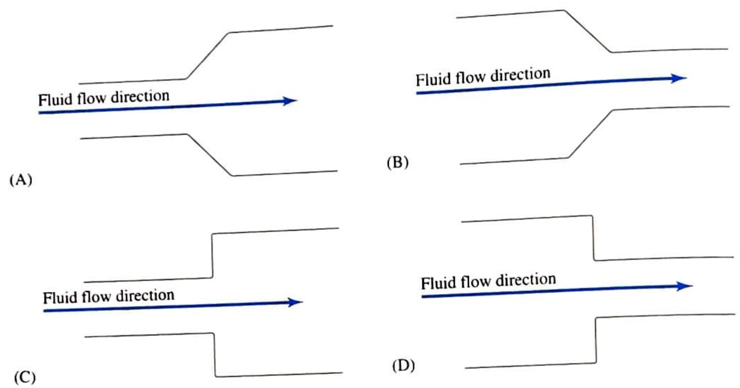


Figure 3.12 Schematics of area changes: (A) enlargement, (B) gradual contraction, (C) sudden expansion, and (D) sudden contraction.

gradual enlargements or contractions, the pressure loss in turbulent flow is a function of the inlet and outlet areas, fluid density, average velocity through the section, and an angle defining the geometry of the area change. For a sudden area change, however, there is no angle to consider, hence the pressure change depends only upon the remaining variables.

CD/Video Library/Flow Past a Back Step

Sudden Expansion

Suppose we analyze the case of an abrupt enlargement of a round pipe as shown in Figure 3.12C. We assume that in a turbulent flow through a sudden area change the frictional pressure loss Δp_F is described by a functional relationship of the form $\Delta p_F = f(A_1, A_2, \rho, \bar{V}_1)$, where A_1 is the inlet area, A_2 is the outlet area, ρ is the fluid density, and $\bar{V}_1$ is the average inlet velocity. Note that we do not have to include $\bar{V}_2$ in our analysis since Eq. 3.16 makes its inclusion redundant. By using dimensional analysis, we can write the frictional pressure drop as

$$\Delta p_F = K_E \tfrac{1}{2}\rho \bar{V}_1^2 \tag{3.22}$$

where conventional engineering practice introduces a dimensionless loss coefficient K_E for the enlargement. Note that we can think of $\frac{1}{2}\rho\bar{V}_1^2$ as representing the kinetic energy per unit volume in the upstream flow. Thus, the result suggests that the frictional pressure drop may be represented as some fraction of the upstream kinetic energy content of the fluid. From the available experimental data we can also deduce that K_E is a function of the area ratio of the enlargement. The problem reduces to finding the enlargement loss coefficient, since when K_E is known, the frictional pressure drop can be calculated from Eq. 3.22.

The enlargement loss coefficient for high Reynolds number turbulent flow is shown in Figure 3.13. Note that the enlargement loss coefficient is always positive. If the inlet

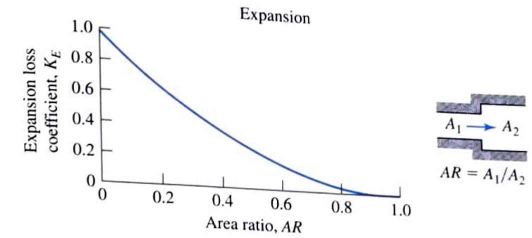


Figure 3.13 Loss coefficients for flow through a sudden expansion.

and outlet areas are equal, there is no frictional pressure loss, and the loss coefficient must be zero. If the ratio of the outlet area to inlet area is very large, the loss coefficient should approach unity because all the kinetic energy in the incoming flow is dissipated.

EXAMPLE 3.7

What is the frictional pressure drop in air flowing in a round duct due to a sudden change in diameter from 0.4 m to 0.6 m? The flowrate in the duct is 0.5 m^3/s. What is the total pressure change across this enlargement?

SOLUTION

We are asked to find the frictional pressure drop and total pressure change across a sudden enlargement in a pipe. Figure 3.13 will serve as a sketch of the geometry of the enlargement. The first part of this problem is solved by using Eq. 3.22 ($\Delta p_F = K_E \frac{1}{2}\rho \bar{V}_1^2$). The area ratio is found to be

$$\frac{A_1}{A_2} = \frac{\pi D_1^2/4}{\pi D_2^2/4} = \left(\frac{D_1}{D_2}\right)^2 = \left(\frac{0.4\ \text{m}}{0.6\ \text{m}}\right)^2 = 0.444$$

By using Figure 3.13, we find a loss coefficient of $K_E \approx 0.3$. Next we determine the upstream average velocity, $\bar{V}_1$, using the definition of volume flowrate given in Eq. 3.16 ($Q = \bar{V}A$). Solving this expression for $\bar{V}_1$ and substituting known values gives

$$\bar{V}_1 = \frac{Q}{A_1} = \frac{Q}{\pi D_1^2/4} = \frac{0.5\ \text{m}^3/\text{s}}{\pi (0.4\ \text{m})^2/4} = 3.98\ \text{m/s}$$

Next use Eq. 3.22, along with the density of air at 20°C (Appendix A) $\rho = 1.204$ kg/m^3, to find the frictional pressure loss:

$$\Delta p_F = K_E \tfrac{1}{2}\rho \bar{V}_1^2 = 0.3\left(\tfrac{1}{2}\right)(1.204\ \text{kg/m}^3)(3.98\ \text{m/s})^2 = 2.86\ \text{Pa}$$

Next find $\bar{V}_2$ using the volume flowrate:

$$\bar{V}_2 = \frac{Q}{A_2} = \frac{Q}{\pi D_2^2/4} = \frac{0.5\ \text{m}^3/\text{s}}{\pi(0.6\ \text{m})^2/4} = 1.77\ \text{m/s}$$

Finally, use Eq. 3.20 to find the total pressure change across the enlargement.

$$p_2 - p_1 = \left[\tfrac{1}{2}\rho\left(\bar{V}_1^2 - \bar{V}_2^2\right)\right] - \Delta p_F$$
$$= \left\{\tfrac{1}{2}(1.204\ \text{kg/m}^3)[(3.98\ \text{m/s})^2 - (1.77\ \text{m/s})^2]\right\} - 2.86\ \text{Pa}$$
$$p_2 - p_1 = 7.65\ \text{Pa} - 2.86\ \text{Pa} = 4.79\ \text{Pa}.$$

CD/Video Library/Forward Facing Step

Sudden Contraction

A similar analysis of the turbulent flow in a sudden contraction as shown in Figure 3.14 leads to the introduction of the contraction loss coefficient K_C and the following formula for calculating the pressure drop

$$\Delta p_F = K_C \tfrac{1}{2}\rho \bar{V}_2^2 \quad (3.23)$$

Note carefully that the contraction loss coefficient is defined in terms of the kinetic energy in the higher speed outlet flow. The value of the contraction loss coefficient can be found in Figure 3.14. If the inlet and outlet areas are equal, there is no pressure loss, and the contraction loss coefficient must be zero. For very small ratios of outlet area to inlet area, the loss coefficient has been found to approach 0.5.

3.3.3 Pump and Fan Laws

FE

The preceding case studies have dealt with calculating the frictional pressure drop in a section of a pipe or in a sudden area change. We now consider the problem of choosing a pump or fan with the performance needed to move fluid through a system once the total pressure drop at the desired flowrate has been determined. It is beyond the scope of this section to address the question of what type of pump or fan should be selected. For example, in air-handling applications one can choose a centrifugal or vane–axial fan (Figure 3.15A). Similar choices exist for pumps (Figure 3.15B). When a certain type of device has been chosen, and the manufacturer selected, it is necessary to pick the appropriate size machine from the manufacturer's family of geometrically similar equipment. In fact, the overall process of choosing a pump or fan is called sizing. The pump and fan laws developed next will allow you to use information provided by the manufacturer to predict the characteristics of geometrically similar, differently sized devices. They will also give you the ability to predict the performance of a specific device under different operating conditions.

In our earlier analysis of flow in a pipe or duct system the focus was on the frictional pressure drop. There are other contributions to the total pressure drop in a system, for example, a change in elevation. It is customary to use a parameter called total head, H, in the design of pipe and duct systems. This total head, with dimensions of energy per unit mass (or equivalently $\{L^2t^{-2}\}$), is a measure of the total load seen by a pump or fan moving fluid through the system. The power, P, required by the pump or fan is also an important parameter in the design of these systems. Thus, in analyzing the performance of a pump or fan, both the head and power are considered to be important dependent variables.

We begin our analysis with the observation that for geometrically similar machines of a given type, only one length scale is required to specify the machine geometry. This length scale is conveniently taken to be the diameter D of the impeller or other rotating element. We assume that the head and power of a fan or pump depends on ω, the angular speed of the impeller, the volume flowrate, and the density and viscosity of the fluid. Thus, we postulate that the head and power are functions of these variables:

$$H = f_1(D, Q, \omega, \rho, \mu) \quad \text{and} \quad P = f_2(D, Q, \omega, \rho, \mu)$$

A dimensional analysis (to be performed in Chapter 9) would show that the dimensionless head can be expressed as follows

$$\frac{H}{\omega^2 D^2} = g_1\left(\frac{Q}{\omega D^3}, \frac{\rho D^2 \omega}{\mu}\right) \quad (3.24a)$$

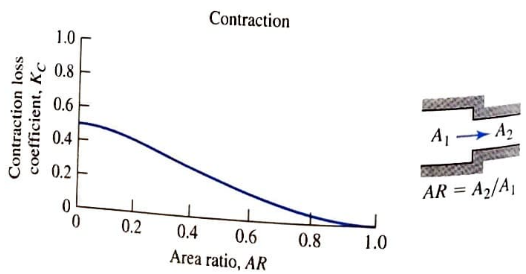


Figure 3.14 Loss coefficients for flow through a sudden contraction.

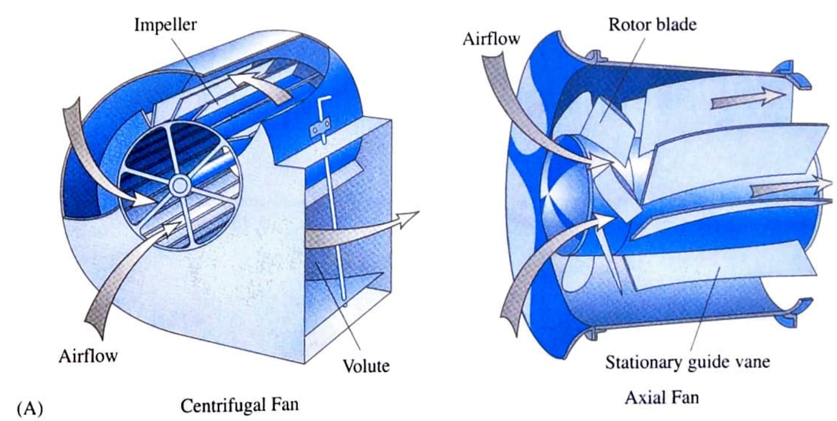
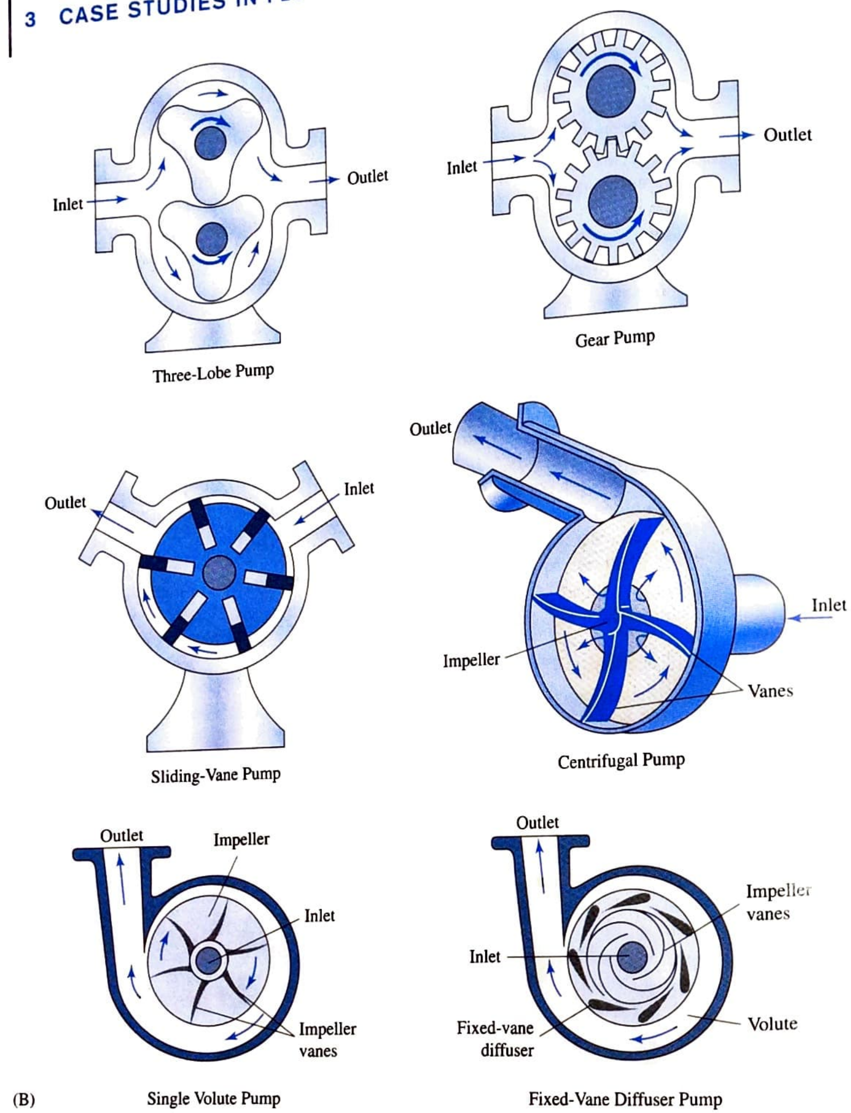


Figure 3.15 Schematics of common designs of (A) fans and (B) pumps. The three-lobe, gear, and sliding-vane devices are all rotary pumps.



(B)

Figure 3.15 *Continued.*

while the dimensionless power may be written as

$$\frac{P}{\rho\omega^3 D^5} = g_2\left(\frac{Q}{\omega D^3}, \frac{\rho D^2\omega}{\mu}\right) \tag{3.24b}$$

In pump and fan engineering, the dependent dimensionless groups $H/\omega^2 D^2$ and $P/\rho\omega^3 D^5$ are known as the head and power coefficients, respectively. The independent dimensionless group $Q/\omega D^3$ is known as the flow coefficient, while the group $\rho D^2\omega/\mu$ can be considered to be a form of the Reynolds number because the product $D\omega$ has the dimensions of velocity.

In considering the scaling of two geometrically similar systems, the principle of similitude, (also discussed in Chapter 9) tells us that all independent dimensionless groups must be the same for each system. However, in dealing with pumps and fans of reasonable size, it is found that the performance is independent of *Re* as defined earlier. Thus the appropriate scaling law for comparing two pumps or fans in the same family is

$$\frac{Q_1}{\omega_1 D_1^3} = \frac{Q_2}{\omega_2 D_2^3} \tag{3.25a}$$

If the flow coefficient of two machines are equal, then the head and power coefficients are also equal:

$$\frac{H_1}{\omega_1^2 D_1^2} = \frac{H_2}{\omega_2^2 D_2^2} \quad \text{and} \quad \frac{P_1}{\rho\omega_1^3 D_1^5} = \frac{P_2}{\rho\omega_2^3 D_2^5} \tag{3.25b}$$

These equations are known as the pump laws or fan laws. Not only do they relate the performance of two differently sized machines in the same family, but they also allow us to determine how a given machine will operate under a new set of operating conditions.

EXAMPLE 3.8

To upgrade a ventilation system it is required that the flowrate be increased from 5000 ft^3/min to 8000 ft^3/min. This is to be accomplished by increasing the angular velocity of the ventilation fan. If the current system operates at a fan rpm of 1000, what fan rpm is required for the upgrade? What will be the power increase for the upgrade?

SOLUTION

Use the fan law, Eq. 3.25a, to determine the new angular velocity (noting that the fan is the same so the characteristic dimension D is constant).

$$\left(\frac{Q}{D^3\omega}\right)_{\text{upgrade}} = \left(\frac{Q}{D^3\omega}\right)_{\text{existing}}$$

$$\omega_{\text{upgrade}} = Q_{\text{upgrade}}\left(\frac{\omega}{Q}\right)_{\text{existing}} = (8000\ \text{ft}^3/\text{min})\left(\frac{1000\ \text{rpm}}{5000\ \text{ft}^3/\text{min}}\right) = 1600\ \text{rpm}$$

Now use Eq. 3.25b to find the increase in power (with fluid density constant).

$$\left(\frac{P}{\omega^3 D^5 \rho}\right)_{\text{upgrade}} = \left(\frac{P}{\omega^3 D^5 \rho}\right)_{\text{existing}}$$

$$\frac{P_{\text{upgrade}}}{P_{\text{existing}}} = \frac{\omega^3_{\text{upgrade}}}{\omega^3_{\text{existing}}} = \frac{(1600\text{ rpm})^3}{(1000\text{ rpm})^3} = 4.096$$

Thus, the increase in rpm of 60% results in a power increase of over 300%.

CD/Boundary Layers

3.3.4 Flat Plate Boundary Layer

FE

The case studies thus far have involved internal flow. A flow is classified as internal if the fluid moves within an interior space defined by a number of bounding walls. Pipe flow is obviously of this type, as is the flow in a pump. Engineers also deal with many important external flows, i.e., flows in which a fluid moves around an object. An external flow also occurs whenever a body such as a vehicle moves through a fluid. The next three case studies deal with external flows.

Consider what happens when flow occurs over a flat plate. As shown in Figure 3.16, the fluid at the plate surface does not move relative to the plate. A short distance away from the plate, however, the fluid is moving at the free stream velocity. The effect of viscosity is to create a boundary layer near the plate in which the velocity changes smoothly and continuously from zero on the plate to the free stream value.[1] The boundary layer thickness increases downstream of the leading edge, and the flow in the boundary layer eventually changes from laminar to turbulent (see Figure 3.17). Because there is a transverse velocity gradient at the plate surface, the fluid exerts a shear stress on the plate that results in a drag force (recall that Newton's law of viscosity relates the shear stress to the velocity gradient via the fluid viscosity).

CD/History/Ludwig Prandtl

A quantity of great interest in the flat plate boundary layer is the wall shear stress. If we know how the wall shear stress varies along the plate, we can calculate the

[1] Given that the fluid velocity and viscous effects are likely to be important, which dimensionless group do you expect to see play a major role in the model for the flat plate boundary layer?

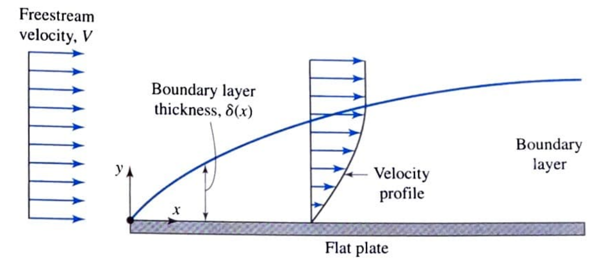


Figure 3.16 Development of the boundary layer on a flat plate.

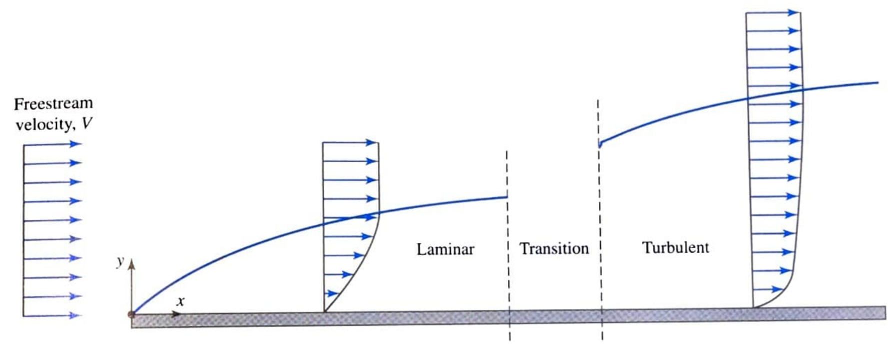


Figure 3.17 Laminar-to-turbulent transition of the boundary layer on a flat plate.

The concept of a boundary layer was conceived by Ludwig Prandtl, who reasoned that in a high Reynolds number flow over a body, viscous effects would be significant only within the boundary layer. His boundary layer theory was one of the most important contributions to fluid mechanics in the twentieth century.

frictional force applied by the fluid to the plate. The flat plate boundary layer may be used to model flow over relatively flat surfaces such as ship hulls and the walls of various structures, and as a crude approximation to the more complex boundary layers on airplane wings, fuselages, and similar surfaces.

Observations suggest that in an incompressible flow at high *Re* the shear stress τ_W on the wall in a flat plate boundary layer (Figure 3.18) depends on the distance from the leading edge x, the freestream velocity V, and the fluid density and viscosity. Thus we propose a relationship between these variables of the form:

$$\tau_W = f(x, V, \rho, \mu)$$

Dimensional analysis reveals that this relationship can be expressed as

$$\frac{\tau_W}{\frac{1}{2}\rho V^2} = g\left(\frac{\rho V x}{\mu}\right) \tag{3.26}$$

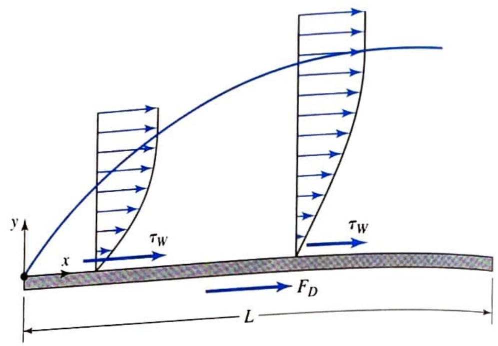


Figure 3.18 Shear stress due to flow over a flat plate.

It is customary in boundary layer analysis to define the skin friction coefficient C_f as

$$C_f = \frac{\tau_W}{\frac{1}{2}\rho V^2} \tag{3.27}$$

and to define a Reynolds number based on the distance x from the leading edge as

$$Re_x = \frac{\rho V x}{\mu} \tag{3.28}$$

From the dimensional analysis we can also conclude that there is a relationship between the skin friction coefficient and the Reynolds number of the form

$$C_f = C_f(Re_x) \tag{3.29}$$

The force exerted by the shear stress on one side of a plate of width w and length L shown in Figure 3.18 is found by integrating the (variable) shear stress along the length of the plate. This frictional force, or drag (since it acts in the flow direction), is given by

$$F_D = w \int_0^L \tau_W(x)\,dx$$

and can also be written in terms of the skin friction coefficient as

$$F_D = w \int_0^L \frac{1}{2}\rho V^2 C_f(x)\,dx \tag{3.30}$$

We can calculate the drag on a flat plate due to a laminar or turbulent boundary layer by using Eq. 3.30, provided we have an expression for the appropriate skin friction coefficient.

CD/Special Features/Blasius Boundary Layer Growth

Laminar Boundary Layer: H. Blasius, a student of Prandtl's, developed an approximate solution for the laminar flat plate boundary layer that gave the following expression for the skin friction coefficient:

$$C_f = \frac{0.664}{\sqrt{Re_x}} \tag{3.31}$$

From empirical observation we know that the transition to turbulence occurs at about $Re_x = 5 \times 10^5$, so Eq. 3.31 is limited to $Re_x < 5 \times 10^5$.

A laminar boundary layer usually transitions to turbulence very close to the leading edge of a plate, so close that in calculating the drag, the laminar portion of the boundary layer can often be ignored and the whole boundary layer treated as if it were turbulent from the leading edge.

Turbulent Boundary Layer: An approximate model of the velocity distribution in turbulent flow (see Figure 3.17) yields an expression for the skin friction coefficient of the form:

$$C_f = \frac{0.0594}{(Re_x)^{1/5}} \tag{3.32}$$

EXAMPLE 3.9

A cruise missile 5 m long and 1 m in diameter is cruising at 200 m/s at an altitude of 500 m. If the boundary layer on the missile skin is modeled as that over a flat plate, what is the drag force on the missile due to skin friction?

SOLUTION

From Appendix A we find for air at 500 m, $\rho = 1.17$ kg/m^3 and $\mu = 1.77 \times 10^{-5}$ (N-s)/m^2. First we use the critical Reynolds number of 5×10^5 to locate the transition to turbulence:

$$x_{cr} = Re_{cr}\frac{\mu}{\rho V} = 5 \times 10^5 \frac{[1.77 \times 10^{-5}\ \text{(N-s)/m}^2]}{(1.17\ \text{kg/m}^3)(200\ \text{m/s})} = 0.04\ \text{m}$$

The laminar region is small enough to be neglected. We will use Eq. 3.32 for the skin friction coefficient and calculate the drag force on the wetted surface by using Eq. 3.30.

$$F_D = w\frac{1}{2}\rho V^2 \int_0^L C_f\,dx = w\frac{1}{2}\rho V^2 \int_0^L \frac{0.0594}{(Re_x)^{1/5}}\,dx = w\frac{1}{2}\rho V^2 \int_0^L \frac{0.0594}{(\rho V x/\mu)^{1/5}}\,dx$$

$$F_D = w\frac{1}{2}\rho V^2 \left(\frac{0.0595}{(\rho V/\mu)^{1/5}}\right)\int_0^L x^{-1/5}\,dx = w\frac{1}{2}\rho V^2 \left[\frac{0.0595}{(\rho V/\mu)^{1/5}}\right]\left(\frac{x^{4/5}}{4/5}\bigg|_0^L\right)$$

In this case the "width" is the circumference of the missile, πD. Substituting appropriate numerical values yields:

$$F_D = \pi(1\ \text{m})\frac{1}{2}(1.17\ \text{kg/m}^3)(200\ \text{m/s})^2 \frac{0.0594}{\left\{\left[\frac{(1.17\ \text{kg/m}^3)(200\ \text{m/s})}{1.77 \times 10^{-5}\ \text{(N-s)/m}^2}\right]\left[\frac{1\ \text{N}}{1\ \text{(kg-m)/s}^2}\right]\right\}^{1/5}} \frac{(5\ \text{m})^{4/5}}{4/5}$$

$$= 744\ \text{N}$$

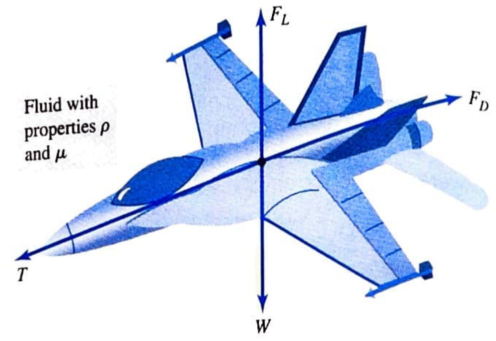


Figure 3.19 Free body diagram of a nonaccelerating body showing that the drag and thrust are equal.

3.3.5 Drag on Cylinders and Spheres

One of the most important problems in fluid mechanics is to determine the drag on a body immersed in a moving fluid. Drag is the component of the total retarding force acting on the body in the direction of the oncoming stream. A bit of thought shows that drag can be due to unbalanced pressures on the fore and aft surfaces of a body as well as to skin friction in the form of shear stress on the wetted surface. Applying Newton's second law to the body shown in Figure 3.19 shows that the thrust and drag forces acting on a nonaccelerating body are equal and opposite. Thus, estimating the force (thrust) needed to move a body through a stationary fluid at constant velocity requires estimating the drag. The power required to move the body through the fluid is the product of the magnitude of the thrust (or drag) and the speed of the body.

The ability to calculate drag is a critical element in the design of virtually all modern modes of transportation. Historically, problems of this type have been investigated

EXAMPLE 3.10

What is the power required to fly the cruise missile in Example 3.9? Assume that the drag is primarily due to skin friction.

SOLUTION

The power required is the product of the thrust and the flight speed. Since the missile is at constant velocity the thrust is equal to the drag 720 N and the flight speed is 200 m/s, the power required is

$$P = (720\ \text{N})(200\ \text{m/s})\left(\frac{1\ \text{J}}{1\ \text{N-m}}\right)\left(\frac{1\ \text{W}}{1\ \text{J/s}}\right) = 144{,}000\ \text{W} = 144\ \text{kW}$$

Another approach to this problem would be to find a drag coefficient that includes both the effects of skin friction and the pressure distribution as discussed shortly.

experimentally by using a wind tunnel to provide a flow over a scale model, with the results presented in terms of the drag coefficients. Analytical results are available to estimate the drag force in a very few cases, but generally engineers rely on a large body of empirical results. In this section we discuss the drag in steady, incompressible flow for two very simple geometries: an infinitely long circular cylinder and a sphere. For simplicity, the flow approaching the cylinder is required to be perpendicular to the axis of the cylinder, and neither the cylinder nor the sphere is rotating.

CD/Video Library/Flow Past a Cylinder

Cylinder

The circular cylinder is a common structural shape. Examples include bridge cables, chimney pipes, wing struts, and flagpoles. Although the geometry of a circular cylinder is simple, the wake of a cylinder can be quite complex (see Figure 3.3).

Now consider the steady flow over a cylinder. We are interested in the drag force F_D on a cylinder of diameter D and length L. The drag will depend on these two geometric parameters as well as on the velocity, density and viscosity of the fluid. We summarize the proposed relationship mathematically as $F_D = f(D, L, V, \rho, \mu)$. Dimensional analysis (details to be provided in Chapter 9) then shows that the relationship between these groups is

$$\frac{F_D}{\frac{1}{2}\rho V^2 DL} = g\left(Re, \frac{L}{D}\right) \tag{3.33}$$

where the Re is based on the cylinder diameter. The standard way to present this result is to write:

$$F_D = C_D \tfrac{1}{2}\rho V^2 DL \tag{3.34}$$

where the drag coefficient for a cylinder is defined as

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 DL} \tag{3.35}$$

Note that from Eq. 3.33 the drag coefficient is $C_D = g(Re, L/D)$, or simply

$$C_D = C_D\left(Re, \frac{L}{D}\right) \tag{3.36}$$

From Eq. 3.36 we conclude that the drag on a cylindrical body depends on the Reynolds number and on the aspect ratio of the cylinder. As the length of the cylindrical body approaches infinity, the flow over the cylinder anywhere along its length must become independent of position. In this limiting case of long cylinders, the drag coefficient for a cylinder is only a function of Re:

$$C_D = C_D(Re) \tag{3.37}$$

CD/Video Library/Flow Past a Sphere

Sphere

The drag on a sphere is needed to predict the behavior of spherical objects of all sizes including pollen and the particles in mists and smoke, as well as the balls used in golf, soccer, and baseball. The drag force F_D on a smooth sphere of diameter D will depend on this single geometric parameter as well as on the fluid velocity, density, and viscosity.

In this case we postulate the relationship as

$$F_D = f(D, V, \rho, \mu)$$

and find that the drag is given by

$$F_D = C_D \frac{1}{2}\rho V^2 \frac{\pi D^2}{4} \tag{3.38}$$

The drag coefficient is defined for a sphere by

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2(\pi D^2/4)} \tag{3.39}$$

Since there is only one length scale in the flow, namely the sphere diameter, the drag coefficient for a sphere depends only on the Reynolds number:

$$C_D = C_D(Re) \tag{3.40}$$

where Re is based on the sphere diameter.

Drag Coefficient

At this point the problem of calculating the drag on a cylinder or sphere is reduced to finding information on the variation of the drag coefficient with Re. Reynolds numbers of interest may range from near zero to 10^8 or even larger, depending on the application: contrast the Re for wind flow over a strand of a spiderweb with that for a guide wire on an early biplane in flight at 90 mph.

Flows for which $Re \ll 1$ are called creeping flows. A creeping flow is dominated by viscous forces. There are analytical results for creeping flows over cylinders and spheres, and we can take advantage of these to deduce the drag coefficients for $Re \ll 1$. An approximate solution due to Oseen for creeping flow over a very long cylinder gives the following formula for the drag coefficient:

$$C_D = \frac{4\pi}{Re\left[\ln\left(2\frac{L}{D}\right) - 0.72\right]} \quad \text{or} \quad C_D = \frac{8\pi}{Re\left[\log_{10}\left(\frac{7.4}{Re}\right)\right]} \tag{3.41}$$

The exact solution for creeping flow over a sphere was derived by Stokes. This solution gives the drag coefficient for a sphere in the creeping flow regime as

$$C_D = \frac{24}{Re} \tag{3.42}$$

For higher Reynolds numbers we can take advantage of empirical data and read the drag coefficients for flow over a sphere or cylinder from Figure 3.20. The interesting variations in drag coefficient with increasing Reynolds number reflect changes in the flow structure. These changes will be discussed in more detail in Chapter 14 on external flow.

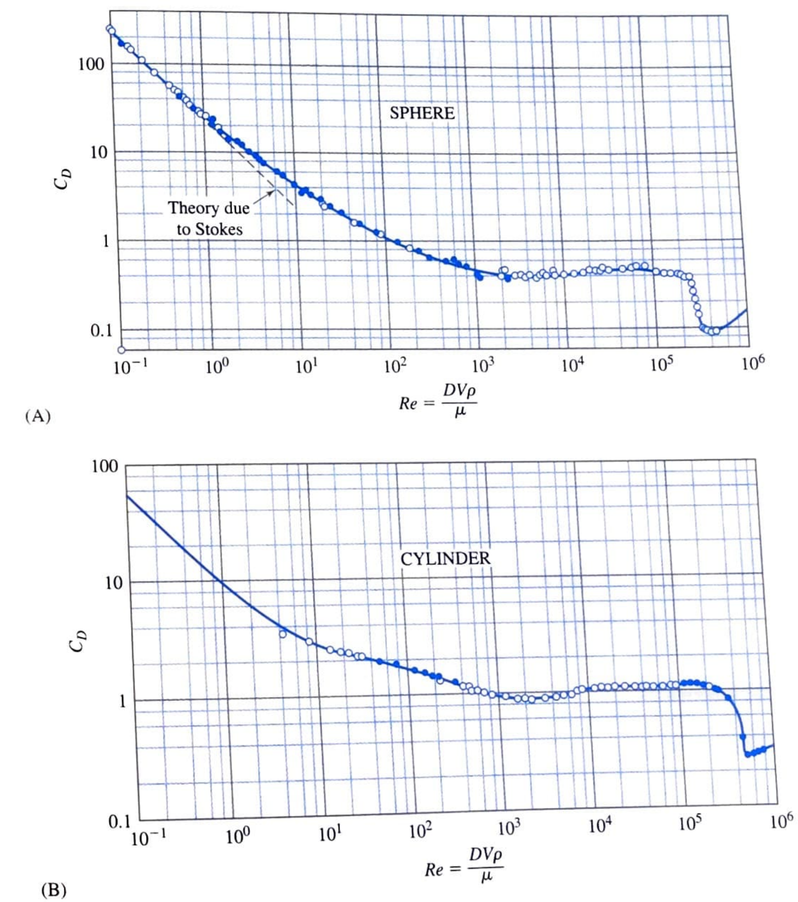


Figure 3.20 Drag coefficient for (A) a smooth sphere and (B) an infinite cylinder as a function of Reynolds number.

EXAMPLE 3.11

What is the drag force on a spherical particle 1 μm in diameter settling in air at $V = 0.1$ m/s?

SOLUTION

From Appendix A we find for air at 20°C: $\rho = 1.204$ kg/m^3 and $\mu = 1.82 \times 10^{-5}$ (N-s)/m^2. The first step is to calculate the Reynolds number:

$$Re = \frac{\rho V D}{\mu} = \frac{(1.204\ \text{kg/m}^3)(0.1\ \text{m/s})(1 \times 10^{-6}\ \text{m})}{1.82 \times 10^{-5}\ \text{(N-s)/m}^2} = 6.6 \times 10^{-3}$$

Since $Re \ll 1$, we can use Eq. 3.42, $C_D = 24/Re$, for the drag coefficient. Finally, we use Eq. 3.38 to calculate the drag force:

$$F_D = C_D \frac{1}{2} \rho V^2 \frac{\pi D^2}{4} = \left(\frac{24}{Re}\right) \frac{1}{2} \rho V^2 \frac{\pi D^2}{4}$$

$$F_D = \left(\frac{24}{6.6 \times 10^{-3}}\right) \frac{1}{2} (1.204\ \text{kg/m}^3)(0.1\ \text{m/s})^2 \frac{\pi(1 \times 10^{-6}\ \text{m})^2}{4} = 1.7 \times 10^{-11}\ \text{N}$$

EXAMPLE 3.12

A radio transmission tower is 1000 ft tall and employs 0.5 in. diameter wire cables to stabilize and strengthen the structure as shown in Figure 3.21. What is the normal force on a cable in the highest expected wind of 100 mph (146.7 ft/s)?

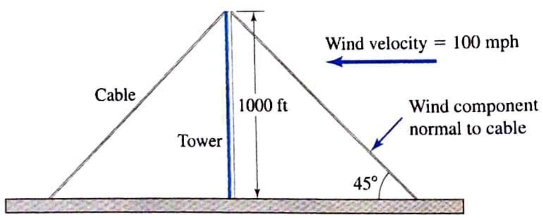


Figure 3.21 Schematic of radio transmission tower for Example 3.12.

SOLUTION

From the geometry, the length of the longest cable is 1414 ft, and the wind velocity is 146.7 ft/s. The component of wind velocity normal to the cable is

(146.7 ft/s cos 45°) = 103.7 ft/s. From Appendix A we find for air at 70°F: $\rho = 0.002329$ slug/ft^3 and $\mu = 3.82 \times 10^{-7}$ (lb$_f$-s)/ft^2. Next calculate Reynolds number as

$$Re = \frac{\rho V D}{\mu} = \frac{(0.002329\ \text{slug/ft}^3)(103.7\ \text{ft/s})(0.5/12\ \text{ft})}{3.82 \times 10^{-7}\ (\text{lb}_f\text{-s})/\text{ft}^2} = 2.63 \times 10^4$$

From Figure 3.20b we read a drag coefficient for a cylinder of ~1.2. Next we compute the force acting normal to the cable with Eq. 3.34:

$$F_D = C_D \tfrac{1}{2} \rho V^2 D L = (1.2)(0.5)(0.002329\ \text{slug/ft}^3)(103.7\ \text{ft/s})^2(0.5/12\ \text{ft})(1000\ \text{ft}) = 626\ \text{lb}_f$$

The aspect ratio of the cable is over 3×10^4, so the assumption, implicit in using Figure 3.20b, that it is an infinite cylinder is appropriate.

CD/Video Library/Flow Past an Airfoil

3.3.6 Lift and Drag on Airfoils

A wing is a specially shaped body designed to produce lift when exposed to a stream of fluid. Lift is defined to be the component of fluid force acting on a body at a right angle to the oncoming stream. Thus, lift is a vertical force for a vehicle or object in level flight and may be thought of as being created by unbalanced pressures acting on the top and bottom of the object. The pressure on a wing, for example, is much higher on the bottom surface than on the top surface. The total lift developed by a wing supports the weight of an aircraft.

Many factors influence the design of a wing. The cross section at any given point along a wing has the form known as an airfoil. This airfoil shape is carefully designed to maximize lift and minimize drag. There are many different airfoil shapes for different applications such as airplane wings, propellers, and impeller blades in turbomachines. Example airfoil shapes are shown in Figure 3.22. In this section we discuss the problem of calculating the total lift and drag produced by a wing with a constant airfoil shape all along its length under the assumption that the wing is effectively infinitely long. Real wings of finite length are subject to end effects, which lower their performance. Airfoils are discussed in more detail in Chapter 14 on external flow.

The standard nomenclature for airfoil geometry is illustrated in Figure 3.23. In steady subsonic flow the lift and drag forces, F_L and F_D, respectively, are each found to depend on the thickness t, span b, chord length c, and angle of attack α. They also depend on the freestream velocity V, and on the fluid density and viscosity. If we postulate the dependence of lift and drag on the physical parameters as

$$F_L = f(t, b, c, V, \rho, \mu) \quad \text{and} \quad F_D = f(t, b, c, V, \rho, \mu)$$

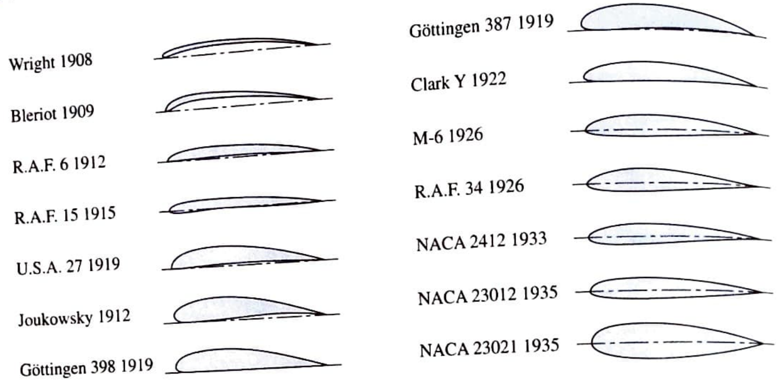


Figure 3.22 Important airfoil shapes in the history of aerodynamics.

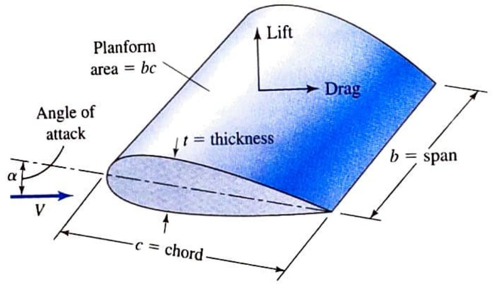


Figure 3.23 Airfoil nomenclature.

dimensional analysis leads to the following standard relationships among dimensionless groups:

$$\frac{F_L}{\frac{1}{2}\rho V^2 bc} = g_1\left(Re_c, \frac{t}{c}, \frac{b}{c}, \alpha\right) \quad \text{and} \quad \frac{F_D}{\frac{1}{2}\rho V^2 bc} = g_2\left(Re_c, \frac{t}{c}, \frac{b}{c}, \alpha\right)$$

where Re_c is the Reynolds number based on chord length, i.e., $Re_c = \rho V c/\mu$. The lift and drag coefficients for an airfoil section are defined as

$$C_L = C_L\left[Re_c, \frac{t}{c}, \frac{b}{c}, \alpha\right] \quad \text{and} \quad C_D = C_D\left[Re_c, \frac{t}{c}, \frac{b}{c}, \alpha\right]$$

thus the lift and drag are given by

$$F_L = C_L \tfrac{1}{2}\rho V^2 bc \quad \text{and} \quad F_D = C_D \tfrac{1}{2}\rho V^2 bc \qquad (3.43a, b)$$

where the product bc is called the planform area. For an infinitely long wing, the ratio of span to chord, b/c, disappears from the expressions for C_L and C_D, and we conclude

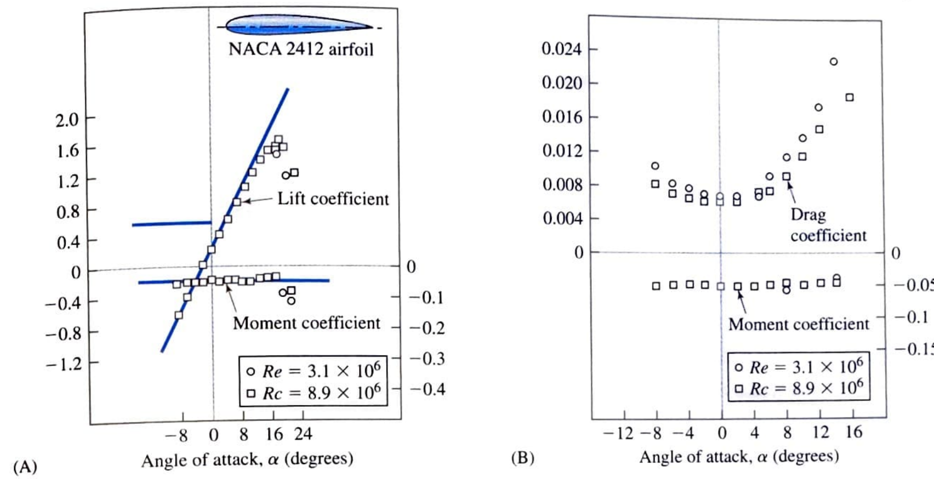


Figure 3.24 Experimental (A) lift and (B) drag coefficients as a function of angle of attack for a NACA 2412 airfoil.

that the lift and drag coefficients of a long wing are a function only of Reynolds number, the geometry of the airfoil as expressed by the ratio of thickness to chord, and the angle of attack.

Lift and drag data were made available for a large number of airfoils by the predecessor to NASA, the National Advisory Committee for Aeronautics (NACA). Figure 3.24 shows lift and drag coefficients for a typical airfoil shape, NACA 2412.

EXAMPLE 3.13

Calculate the lift force on a Cessna 150 wing cruising at an airspeed of 120 mph at an altitude of 5000 ft. The wing is constructed of a NACA 2412 airfoil at an angle of attack of 2°. Its span is 32 ft., 8 in., and the wing planform area is 157 ft².

SOLUTION

From Appendix B we find for air at 5000 ft ($T = 41°$F): $\rho = 2.048 \times 10^{-3}$ slug/ft³ and $\mu = 3.637 \times 10^{-7}$ (lb$_f$-s)/ft². The lift coefficient for the NACA 2412 at 2° angle of attack is ~0.3 from Figure 3.24. Next, use Eq. 3.43a to calculate the lift as

$$F_L = C_L \tfrac{1}{2}\rho V^2 bc$$

$$= (0.3)\frac{1}{2}(2.048 \times 10^{-3}\ \text{slug/ft}^3)\left[120\ \text{mph}\left(1.4667\ \frac{\text{ft/s}}{\text{mph}}\right)\right]^2 157\ \text{ft}^2 = 1.5 \times 10^3\ \text{lb}_f$$

Note that the Reynolds number based on the chord length (calculated as area divided by length) is

$$Re_c = \frac{\rho V c}{\mu}$$

$$= \frac{(2.048 \times 10^{-3}\ \text{slug/ft}^3)(120\ \text{mph})\left(1.4667\ \dfrac{\text{ft/s}}{\text{mph}}\right)(157\ \text{ft}^2/32.667\ \text{ft})}{3.637 \times 10^{-7}(\text{lb}_\text{f}\text{-s})/\text{ft}^2}$$

$$= 4.8 \times 10^6$$

which is within the range of the experimental data given in Figure 3.24.

3.4 SUMMARY

In this chapter several case studies were introduced. Each case study had two parts, a brief description of the flow field of interest and the introduction of design formulas used to calculate important quantities of engineering interest. These formulas rely primarily on results obtained using experimental methods, and in particular on the dimensional analysis and modeling tools, which you will learn about eventually, in Chapter 9. The amount of information given in a case study is not unlike what you might find in an engineering handbook.

The case studies included frequent references to dimensionless groups. A dimensionless group is an algebraic combination of the parameters describing a particular flow problem that proves to be both dimensionless as a whole and significant in terms of understanding the flow field. The use of dimensionless groups allows an engineer to classify a fluid mechanics problem, relate it to work by others, and select an effective solution method. Although a large number of dimensionless groups occur in fluid mechanics, only a limited number of them are used on a regular basis. We list five examples.

1. Reynolds number, $Re = \rho VL/\mu$, is the most common dimensionless group in fluid mechanics. It can be interpreted as the ratio of inertial forces to viscous forces. If the *Re* is small, viscous forces dominate the flow and inertial forces can be neglected. Conversely, if *Re* is large, inertial forces dominate outside of boundary layers.
2. Euler number, $Eu = \Delta p/\rho V^2$, is the ratio of pressure forces to inertial forces.
3. Froude number, $Fr = V^2/gL$, is the ratio of inertial forces to gravity forces. It is important in the classification of free surface flows.
4. Mach number, $M = V/c$, is the ratio of the velocity scale to the speed of sound in the fluid. The Mach number is important in compressible fluid mechanics and is used to determine when compressible effects must be considered.
5. Weber number, $We = \rho V^2 L/\sigma$, is the ratio of inertial forces to surface tension forces. The Weber number is important in a limited number of instances such as capillary flows.

This chapter concludes with six important case studies: fully developed flow in pipes and ducts, flow through sudden area change, pump and fan laws, flat plate boundary layer, drag on cylinders and spheres, and lift and drag on airfoils. These case studies represent a substantial amount of the material with which an engineer could practice design after a single course in fluid mechanics. Each of these problems may be studied theoretically, but the majority of useful results have been obtained empirically. In this chapter we have introduced each flow, indicated the important dimensionless groups that can be used to understand the flow, and provided formulaic solutions to each flow. These problems were designated as case studies to emphasize the relevance of the subsequent theoretical chapters to everyday engineering problems. We do this by revisiting these problems throughout the book.

PROBLEMS

Section 3.2

3.1 For each of the common dimensionless groups listed, demonstrate that the group is, in fact, dimensionless. In addition, offer a physical interpretation of each dimensionless group.
(*a*) Reynolds number, *Re*
(*b*) Froude number, *Fr*
(*c*) Euler number, *Eu*
(*d*) Prandtl number, *Pr*

3.2 For each of the common dimensionless groups listed, demonstrate that the group is, in fact, dimensionless. In addition, offer a physical interpretation of each dimensionless group.
(*a*) Reynolds number, *Re*
(*b*) Mach number, *M*
(*c*) Weber number, *We*
(*d*) Strouhal number, *St*

3.3 Air initially at STP is flowing over an airplane wing with a chord of 2 m. If the air velocity is 300 km/h, determine the Reynolds number and Mach number for this flow. At what speed must the plane fly in a standard atmosphere at an altitude of 3000 m for the flow over the wing to have the same value of *Re*?

3.4 A sphere of diameter 2 mm is moving through glycerin at a velocity of 5 mm/s.
(*a*) Calculate the Reynolds number for this flow.
(*b*) Would you characterize this flow as turbulent, laminar, or creeping flow?
(*c*) Do you think viscous effects are important in this flow?
(*d*) Do you think inertial effects are important in the previous flow?

3.5 As mentioned in Chapter 2, an engineer must consider compressibility effects when the Mach number exceeds 0.3. What is the flight speed for an aircraft flying in a standard atmosphere at 10,000 ft necessary to achieve this Mach number?

3.6 As mentioned in Chapter 2, an engineer must consider compressibility effects when the Mach number exceeds 0.3. For water at STP, what velocity must the fluid reach to achieve this Mach number? If the water is moving through a 1 in. diameter pipe at $M = 0.3$, what is the corresponding *Re* number?

3.7 A thin film of SAE 30W oil is experiencing a velocity of 0.75 m/s at a depth of 1.5 mm below its free surface. Calculate the Froude number and the Weber number for this flow. What is the significance of the relative values of *Fr* and *We* in this flow?

3.8 In open channel flows the characteristic dimension in the Froude number is the depth of the fluid. What is the minimum fluid velocity necessary to achieve supercritical flow in a channel that is 100 ft deep? What does it mean