CHAPTER 3

FLUID FLOW

FLOWING fluids in HVAC&R systems can transfer heat, mass, and momentum. This chapter introduces the basics of fluid mechanics related to HVAC processes, reviews pertinent flow processes, and presents a general discussion of single-phase fluid flow analysis.

FLUID PROPERTIES

Solids and fluids react differently to shear stress: solids deform only a finite amount, whereas fluids deform continuously until the stress is removed. Both liquids and gases are fluids, although the natures of their molecular interactions differ strongly in both degree of compressibility and formation of a free surface (interface) in liquid. In general, liquids are considered incompressible fluids; gases may range from **compressible** to nearly **incompressible**. Liquids have unbalanced molecular cohesive forces at or near the surface (interface), so the liquid surface tends to contract and has properties similar to a stretched elastic membrane. A liquid surface, therefore, is under tension (**surface tension**).

Fluid motion can be described by several simplified models. The simplest is the **ideal-fluid** model, which assumes that the fluid has no resistance to shearing. Ideal fluid flow analysis is well developed (e.g., Schlichting 1979), and may be valid for a wide range of applications.

Viscosity is a measure of a fluid's resistance to shear. Viscous effects are taken into account by categorizing a fluid as either Newtonian or non-Newtonian. In **Newtonian fluids**, the rate of deformation is directly proportional to the shearing stress; most fluids in the HVAC industry (e.g., water, air, most refrigerants) can be treated as Newtonian. In **non-Newtonian fluids**, the relationship between the rate of deformation and shear stress is more complicated.

Density

The density ρ of a fluid is its mass per unit volume. The densities of air and water (Fox et al. 2004) at standard indoor conditions of 68°F and 14.696 psi (sea-level atmospheric pressure) are

$$\rho_{water} = 62.4 \text{ lb}_m/\text{ft}^3$$

$$\rho_{air} = 0.0753 \text{ lb}_m/\text{ft}^3$$

Viscosity

Viscosity is the resistance of adjacent fluid layers to shear. A classic example of shear is shown in Figure 1, where a fluid is between two parallel plates, each of area A separated by distance Y. The bottom plate is fixed and the top plate is moving, which induces a shearing force in the fluid. For a Newtonian fluid, the tangential force F per unit area required to slide one plate with velocity V parallel to the other is proportional to V/Y:

$$F/A = \mu (V/Y) \quad (1)$$

The preparation of this chapter is assigned to TC 1.3, Heat Transfer and Fluid Flow.

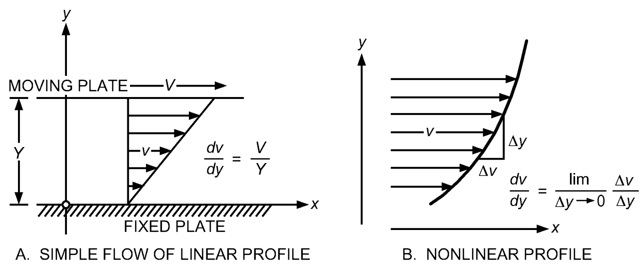

Fig. 1 Velocity Profiles and Gradients in Shear Flows

where the proportionality factor μ is the **absolute** or **dynamic viscosity** of the fluid. The ratio of F to A is the **shearing stress** τ, and V/Y is the **lateral velocity gradient** (Figure 1A). In complex flows, velocity and shear stress may vary across the flow field; this is expressed by

$$\tau = \mu \frac{dv}{dy} \quad (2)$$

The velocity gradient associated with viscous shear for a simple case involving flow velocity in the x direction but of varying magnitude in the y direction is illustrated in Figure 1B.

Absolute viscosity μ depends primarily on temperature. For gases (except near the critical point), viscosity increases with the square root of the absolute temperature, as predicted by the kinetic theory of gases. In contrast, a liquid's viscosity decreases as temperature increases. Absolute viscosities of various fluids are given in Chapter 33.

Absolute viscosity has dimensions of force × time/length^2. At standard indoor conditions, the absolute viscosities of water and dry air (Fox et al. 2004) are

$$\mu_{water} = 6.76 \times 10^{-4} \text{ lb}_m/\text{ft}\cdot\text{s} = 2.10 \times 10^{-5} \text{ lb}_f\cdot\text{s/ft}^2$$

$$\mu_{air} = 1.22 \times 10^{-5} \text{ lb}_m/\text{ft}\cdot\text{s} = 3.79 \times 10^{-7} \text{ lb}_f\cdot\text{s/ft}^2$$

Another common unit of viscosity is the **centipoise** (1 centipoise = 1 g/(s·m) = 1 mPa·s). At standard conditions, water has a viscosity close to 1.0 centipoise.

In fluid dynamics, **kinematic viscosity** ν is sometimes used in lieu of absolute or dynamic viscosity. Kinematic viscosity is the ratio of absolute viscosity to density:

$$\nu = \mu/\rho$$

At standard indoor conditions, the kinematic viscosities of water and dry air (Fox et al. 2004) are

$$\nu_{water} = 1.08 \times 10^{-5} \text{ ft}^2/\text{s}$$

$$\nu_{air} = 1.62 \times 10^{-4} \text{ ft}^2/\text{s}$$

The **stoke** (1 cm^2/s) and **centistoke** (1 mm^2/s) are common units for kinematic viscosity.

Note that the inch-pound system of units often requires the conversion factor g_c = 32.1740 $lb_m \cdot ft/s^2 \cdot lb_f$ to make some equations containing lb_f and lb_m dimensionally consistent. The conversion factor g_c is not shown in the equations, but is included as needed.

BASIC RELATIONS OF FLUID DYNAMICS

This section discusses fundamental principles of fluid flow for constant-property, homogeneous, incompressible fluids and introduces fluid dynamic considerations used in most analyses.

Continuity in a Pipe or Duct

Conservation of mass applied to fluid flow in a conduit requires that mass not be created or destroyed. Specifically, the mass flow rate into a section of pipe must equal the mass flow rate out of that section of pipe if no mass is accumulated or lost (e.g., from leakage). This requires that

$$\dot{m} = \int \rho v \, dA = \text{constant} \tag{3}$$

where $\dot{m}$ is mass flow rate across the area normal to flow, v is fluid velocity normal to differential area dA, and ρ is fluid density. Both ρ and v may vary over the cross section A of the conduit. When flow is effectively incompressible (ρ = constant) in a pipe or duct flow analysis, the **average velocity** is then $V = (1/A)\int v \, dA$, and the mass flow rate can be written as

$$\dot{m} = \rho V A \tag{4}$$

or

$$Q = \dot{m}/\rho = AV \tag{5}$$

where Q is **volumetric flow rate**.

Bernoulli Equation and Pressure Variation in Flow Direction

The **Bernoulli equation** is a fundamental principle of fluid flow analysis. It involves the conservation of momentum and energy along a streamline; it is not generally applicable across streamlines. Development is fairly straightforward. The first law of thermodynamics can apply to both mechanical flow energies (**kinetic** and **potential energy**) and thermal energies.

The change in energy content ΔE per unit mass of flowing fluid is a result of the work per unit mass w done on the system plus the heat per unit mass q absorbed or rejected:

$$\Delta E = w + q \tag{6}$$

Fluid energy is composed of kinetic, potential (because of elevation z), and internal (u) energies. Per unit mass of fluid, the energy change relation between two sections of the system is

$$\Delta\left(\frac{v^2}{2} + gz + u\right) = E_M - \Delta\left(\frac{p}{\rho}\right) + q \tag{7}$$

where the work terms are (1) external work E_M from a fluid machine (E_M is positive for a pump or blower) and (2) flow work p/ρ (where p = pressure), and g is the gravitational constant. Rearranging, the energy equation can be written as the **generalized Bernoulli equation**:

$$\Delta\left(\frac{v^2}{2} + gz + u + \frac{p}{\rho}\right) = E_M + q \tag{8}$$

The expression in parentheses in Equation (8) is the sum of the kinetic energy, potential energy, internal energy, and flow work per unit mass flow rate. In cases with no work interaction, no heat transfer, and no viscous frictional forces that convert mechanical energy into internal energy, this expression is constant and is known as the **Bernoulli constant *B***:

$$\frac{v^2}{2} + gz + \left(\frac{p}{\rho}\right) = B \tag{9}$$

Alternative forms of this relation are obtained through multiplication by ρ or division by g:

$$p + \frac{\rho v^2}{2} + \rho g z = \rho B \tag{10}$$

$$\frac{p}{\gamma} + \rho\frac{v^2}{2g} + z = \frac{B}{g} \tag{11}$$

where $\gamma = \rho g$ is the **specific weight** or **weight density**. Note that Equations (9) to (11) assume no frictional losses.

The units in the first form of the Bernoulli equation [Equation (9)] are energy per unit mass; in Equation (10), energy per unit volume; in Equation (11), energy per unit weight, usually called **head**. Note that the units for head reduce to just length (i.e., $ft \cdot lb_f/lb_f$ to ft). In gas flow analysis, Equation (10) is often used, and ρgz is negligible. Equation (10) should be used when density variations occur. For liquid flows, Equation (11) is commonly used. Identical results are obtained with the three forms if the units are consistent and fluids are homogeneous.

Many systems of pipes, ducts, pumps, and blowers can be considered as one-dimensional flow along a streamline (i.e., variation in velocity across the pipe or duct is ignored, and local velocity v = average velocity V). When v varies significantly across the cross section, the kinetic energy term in the Bernoulli constant B is expressed as $\alpha V^2/2$, where the **kinetic energy factor** ($\alpha > 1$) expresses the ratio of the true kinetic energy of the velocity profile to that of the average velocity. For laminar flow in a wide rectangular channel, α = 1.54, and in a pipe, α = 2.0. For turbulent flow in a duct, $\alpha \approx 1$.

Heat transfer q may often be ignored. Conversion of mechanical energy to internal energy Δu may be expressed as a loss E_L. The change in the Bernoulli constant ($\Delta B = B_2 - B_1$) between stations 1 and 2 along the conduit can be expressed as

$$\left(\frac{p}{\rho} + \alpha\frac{V^2}{2} + gz\right)_1 + E_M - E_L = \left(\frac{p}{\rho} + \alpha\frac{V^2}{2} + gz\right)_2 \tag{12}$$

or, by dividing by g, in the form

$$\left(\frac{p}{\gamma} + \alpha\frac{V^2}{2g} + z\right)_1 + H_M - H_L = \left(\frac{p}{\gamma} + \alpha\frac{V^2}{2g} + z\right)_2 \tag{13}$$

Note that Equation (12) has units of energy per mass, whereas each term in Equation (13) has units of energy per weight, or head. The terms E_M and E_L are defined as positive, where $gH_M = E_M$ represents energy added to the conduit flow by pumps or blowers. A turbine or fluid motor thus has a negative H_M or E_M. The terms E_M and H_M (= E_M/g) are defined as positive, and represent energy added to the fluid by pumps or blowers. *The simplicity of Equation (13) should be noted*; the total head at station 1 (pressure head plus velocity head plus elevation head) plus the head added by a pump (H_M) minus the head lost through friction (H_L) is the total head at station 2.

Laminar Flow

When real-fluid effects of viscosity or turbulence are included, the continuity relation in Equation (5) is not changed, but V must be evaluated from the integral of the velocity profile, using local velocities. In fluid flow past fixed boundaries, velocity at the boundary is zero, velocity gradients exist, and shear stresses are produced. The equations of motion then become complex, and exact solutions are difficult to find except in simple cases for laminar flow between flat plates, between rotating cylinders, or within a pipe or tube.

For steady, fully developed laminar flow between two parallel plates (Figure 2), shear stress τ varies linearly with distance y from the centerline (transverse to the flow; $y = 0$ in the center of the channel). For a wide rectangular channel $2b$ tall, τ can be written as

$$\tau = \left(\frac{y}{b}\right)\tau_w = \mu \frac{dv}{dy} \tag{14}$$

where τ_w is wall shear stress $[b(dp/ds)]$, and s is flow direction. Because velocity is zero at the wall ($y = b$), Equation (14) can be integrated to yield

$$v = \left(\frac{b^2 - y^2}{2\mu}\right)\frac{dp}{ds} \tag{15}$$

The resulting parabolic velocity profile in a wide rectangular channel is commonly called **Poiseuille flow**. Maximum velocity occurs at the centerline ($y = 0$), and the average velocity V is 2/3 of the maximum velocity. From this, the longitudinal pressure drop in terms of V can be written as

$$\frac{dp}{ds} = -\left(\frac{3\mu V}{b^2}\right) \tag{16}$$

A parabolic velocity profile can also be derived for a pipe of radius R. V is 1/2 of the maximum velocity, and the pressure drop can be written as

$$\frac{dp}{ds} = -\left(\frac{8\mu V}{R^2}\right) \tag{17}$$

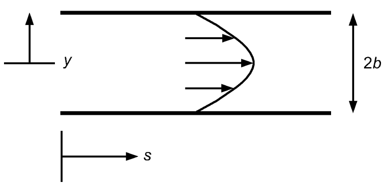

Fig. 2 Dimensions for Steady, Fully Developed Laminar Flow Equations

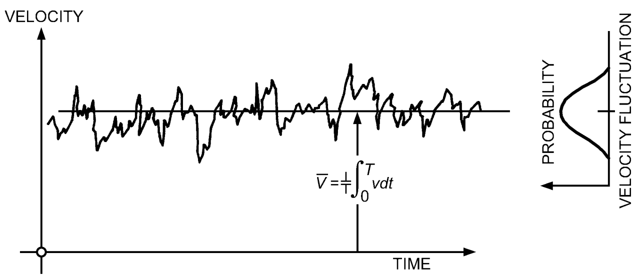

Fig. 3 Velocity Fluctuation at Point in Turbulent Flow

Turbulence

Fluid flows are generally turbulent, involving random perturbations or fluctuations of the flow (velocity and pressure), characterized by an extensive hierarchy of scales or frequencies (Robertson 1963). Flow disturbances that are not chaotic but have some degree of periodicity (e.g., the oscillating vortex trail behind bodies) have been erroneously identified as turbulence. Only flows involving random perturbations without any order or periodicity are turbulent; velocity in such a flow varies with time or locale of measurement (Figure 3).

Turbulence can be quantified statistically. The velocity most often used is the time-averaged velocity. The strength of turbulence is characterized by the root mean square (RMS) of the instantaneous variation in velocity about this mean. Turbulence causes the fluid to transfer momentum, heat, and mass very rapidly across the flow.

Laminar and turbulent flows can be differentiated using the **Reynolds number Re**, which is a dimensionless relative ratio of inertial forces to viscous forces:

$$Re_L = VL/\nu \tag{18}$$

where L is the characteristic length scale and ν is the kinematic viscosity of the fluid. In flow through pipes, tubes, and ducts, the characteristic length scale is the **hydraulic diameter D_h**, given by

$$D_h = 4A/P_w \tag{19}$$

where A is the cross-sectional area of the pipe, duct, or tube, and P_w is the wetted perimeter.

For a round pipe, D_h equals the pipe diameter. In general, **laminar flow** in pipes or ducts exists when the Reynolds number (based on D_h) is less than 2300. Fully **turbulent flow** exists when $Re_{D_h} >$ 10,000. For $2300 < Re_{D_h} <$ 10,000, transitional flow exists, and predictions are unreliable.

BASIC FLOW PROCESSES

Wall Friction

At the boundary of real-fluid flow, the relative tangential velocity at the fluid surface is zero. Sometimes in turbulent flow studies, velocity at the wall may appear finite and nonzero, implying a **fluid slip** at the wall. However, this is not the case; the conflict results from difficulty in velocity measurements near the wall (Goldstein 1938). Zero wall velocity leads to high shear stress near the wall boundary, which slows adjacent fluid layers. Thus, a velocity profile develops near a wall, with velocity increasing from zero at the wall to an exterior value within a finite lateral distance.

Laminar and turbulent flow differ significantly in their velocity profiles. Turbulent flow profiles are flat and laminar profiles are more pointed (Figure 4). As discussed, fluid velocities of the turbulent profile near the wall must drop to zero more rapidly than those of the laminar profile, so shear stress and friction are much greater in turbulent flow. Fully developed conduit flow may be characterized by the **pipe factor**, which is the ratio of average to maximum (centerline) velocity. Viscous velocity profiles result in pipe factors

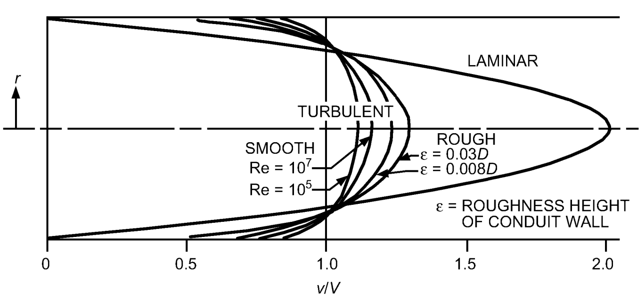

Fig. 4 Velocity Profiles of Flow in Pipes

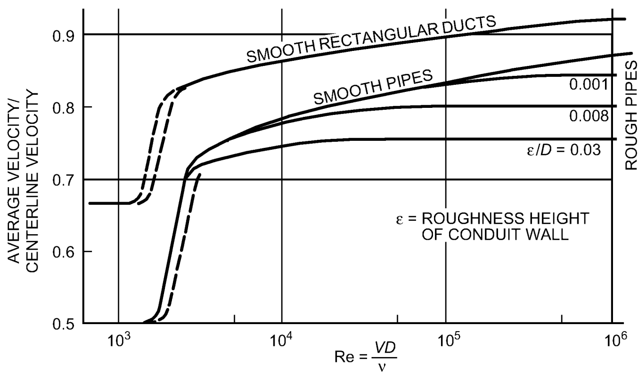

Fig. 5 Pipe Factor for Flow in Conduits

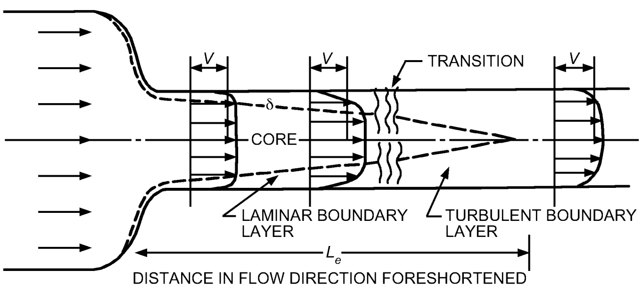

Fig. 6 Flow in Conduit Entrance Region

of 0.667 and 0.50 for wide rectangular and axisymmetric conduits. Figure 5 indicates much higher values for rectangular and circular conduits for turbulent flow. Because of the flat velocity profiles, the kinetic energy factor α in Equations (12) and (13) ranges from 1.01 to 1.10 for fully developed turbulent pipe flow.

Boundary Layer

The boundary layer is the region close to the wall where wall friction affects flow. Boundary layer thickness (usually denoted by δ) is thin compared to downstream flow distance. For external flow over a body, fluid velocity varies from zero at the wall to a maximum at distance δ from the wall. Boundary layers are generally laminar near the start of their formation but may become turbulent downstream.

A significant boundary-layer occurrence exists in a pipeline or conduit following a well-rounded entrance (Figure 6). Layers grow from the walls until they meet at the center of the pipe. Near the start of the straight conduit, the layer is very thin and most likely laminar, so the uniform velocity core outside has a velocity only slightly greater than the average velocity. As the layer grows in thickness, the slower velocity near the wall requires a velocity increase in the uniform core to satisfy continuity. As flow proceeds, the wall layers grow (and centerline velocity increases) until they join, after an **entrance length L_e**. Applying the Bernoulli relation of Equation (10) to core flow indicates a decrease in pressure along the layer. Ross (1956) shows that, although the entrance length L_e is many diameters, the length in which pressure drop significantly exceeds that for fully developed flow is on the order of 10 hydraulic diameters for turbulent flow in smooth pipes.

In more general boundary-layer flows, as with wall layer development in a diffuser or for the layer developing along the surface of a strut or turning vane, pressure gradient effects can be severe and may even lead to boundary layer separation. When the outer flow velocity (v_1 in Figure 7) decreases in the flow direction, an adverse pressure gradient can cause separation, as shown in the figure. Downstream from the separation point, fluid backflows near the

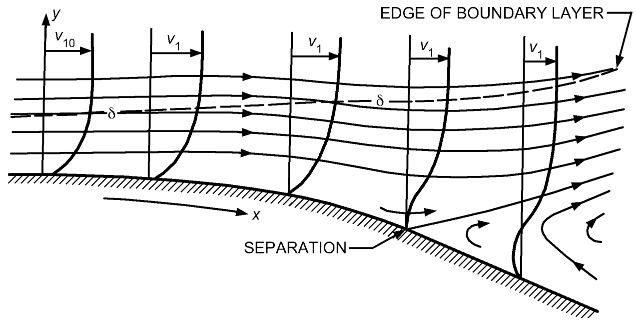

Fig. 7 Boundary Layer Flow to Separation

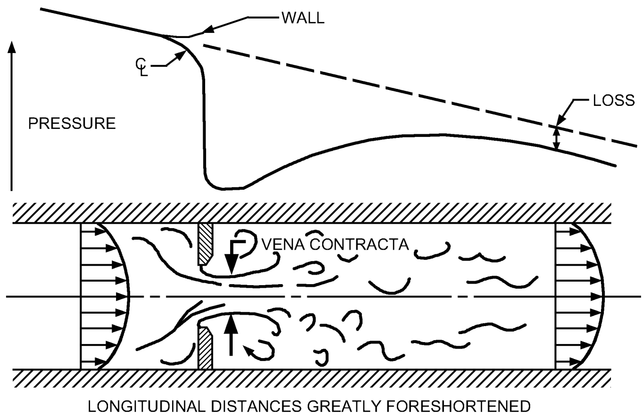

Fig. 8 Geometric Separation, Flow Development, and Loss in Flow Through Orifice

wall. Separation is caused by frictional velocity (thus local kinetic energy) reduction near the wall. Flow near the wall no longer has energy to move into the higher pressure imposed by the decrease in v_1 at the edge of the layer. The locale of this separation is difficult to predict, especially for the turbulent boundary layer. Analyses verify the experimental observation that a turbulent boundary layer is less subject to separation than a laminar one because of its greater kinetic energy.

Flow Patterns with Separation

In technical applications, flow with separation is common and often accepted if it is too expensive to avoid. Flow separation may be geometric or dynamic. Dynamic separation is shown in Figure 7. Geometric separation (Figures 8 and 9) results when a fluid stream passes over a very sharp corner, as with an orifice; the fluid generally leaves the corner irrespective of how much its velocity has been reduced by friction.

For geometric separation in orifice flow (Figure 8), the outer streamlines separate from the sharp corners and, because of fluid inertia, contract to a section smaller than the orifice opening. The smallest section is known as the **vena contracta** and generally has a limiting area of about six-tenths of the orifice opening. After the vena contracta, the fluid stream expands rather slowly through turbulent or laminar interaction with the fluid along its sides. Outside the jet, fluid velocity is comparatively small. Turbulence helps spread out the jet, increases losses, and brings the velocity distribution back to a more uniform profile. Finally, downstream, the velocity profile returns to the fully developed flow of Figure 4. The entrance and exit profiles can profoundly affect the vena contracta and pressure drop (Coleman 2004).

Other geometric separations (Figure 9) occur in conduits at sharp entrances, inclined plates or dampers, or sudden expansions. For these geometries, a vena contracta can be identified; for sudden

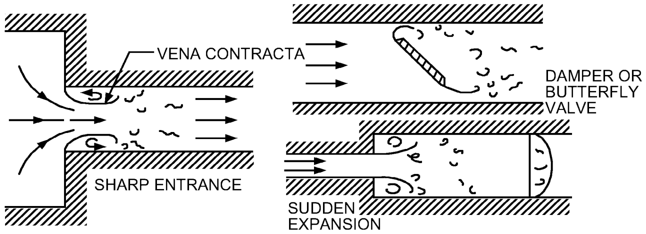

Fig. 9 Examples of Geometric Separation Encountered in Flows in Conduits

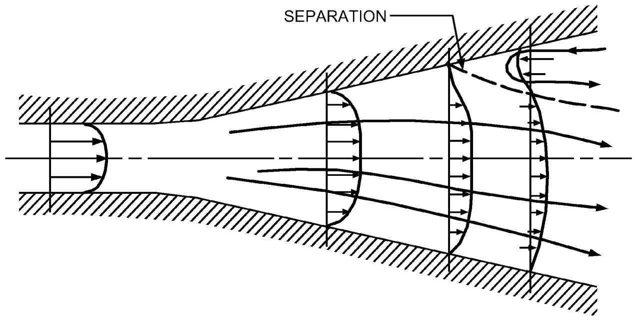

Fig. 10 Separation in Flow in Diffuser

expansion, its area is that of the upstream contraction. Ideal-fluid theory, using free streamlines, provides insight and predicts contraction coefficients for valves, orifices, and vanes (Robertson 1965). These geometric flow separations produce large losses. To expand a flow efficiently or to have an entrance with minimum losses, design the device with gradual contours, a diffuser, or a rounded entrance.

Flow devices with gradual contours are subject to separation that is more difficult to predict, because it involves the dynamics of boundary-layer growth under an adverse pressure gradient rather than flow over a sharp corner. A diffuser is used to reduce the loss in expansion; it is possible to expand the fluid some distance at a gentle angle without difficulty, particularly if the boundary layer is turbulent. Eventually, separation may occur (Figure 10), which is frequently asymmetrical because of irregularities. Downstream flow involves flow reversal (backflow) and excess losses. Such separation is commonly called **stall** (Kline 1959). Larger expansions may use splitters that divide the diffuser into smaller sections that are less likely to have separations (Moore and Kline 1958). Another technique for controlling separation is to bleed some low-velocity fluid near the wall (Furuya et al. 1976). Alternatively, Heskested (1970) shows that suction at the corner of a sudden expansion has a strong positive effect on geometric separation.

Drag Forces on Bodies or Struts

Bodies in moving fluid streams are subjected to appreciable fluid forces or **drag**. Conventionally, the drag force F_D on a body can be expressed in terms of a **drag coefficient** $\boldsymbol{C_D}$:

$$F_D = C_D \rho A \left(\frac{V^2}{2}\right) \tag{20}$$

where A is the projected (normal to flow) area of the body. The drag coefficient C_D is a strong function of the body's shape and angularity, and the Reynolds number of the relative flow in terms of the body's characteristic dimension.

For Reynolds numbers of 10^3 to 10^5, the C_D of most bodies is constant because of flow separation, but above 10^5, the C_D of rounded bodies drops suddenly as the surface boundary layer undergoes transition to turbulence. Typical C_D values are given in Table 1; Hoerner (1965) gives expanded values.

Table 1 Drag Coefficients

Body Shape	$10^3 < \text{Re} < 2 \times 10^5$	$\text{Re} > 3 \times 10^5$
Sphere	0.36 to 0.47	~0.1
Disk	1.12	1.12
Streamlined strut	0.1 to 0.3	< 0.1
Circular cylinder	1.0 to 1.1	0.35
Elongated rectangular strut	1.0 to 1.2	1.0 to 1.2
Square strut	~2.0	~2.0

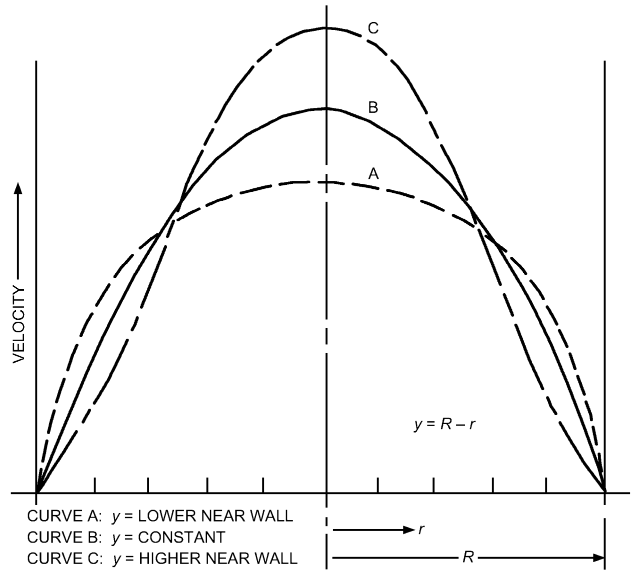

Fig. 11 Effect of Viscosity Variation on Velocity Profile of Laminar Flow in Pipe

Nonisothermal Effects

When appreciable temperature variations exist, the primary fluid properties (density and viscosity) may no longer assumed to be constant, but vary across or along the flow. The Bernoulli equation [Equations (9) to (11)] must be used, because volumetric flow is not constant. With gas flows, the thermodynamic process involved must be considered. In general, this is assessed using Equation (9), written as

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gz = B \tag{21}$$

Effects of viscosity variations also appear. In nonisothermal laminar flow, the parabolic velocity profile (see Figure 4) is no longer valid. In general, for gases, viscosity increases with the square root of absolute temperature; for liquids, viscosity decreases with increasing temperature. This results in opposite effects.

For fully developed pipe flow, the linear variation in shear stress from the wall value τ_w to zero at the centerline is independent of the temperature gradient. In the section on Laminar Flow, τ is defined as $\tau = (y/b)\,\tau_w$, where y is the distance from the centerline and $2b$ is the wall spacing. For pipe radius $R = D/2$ and distance from the wall $y = R - r$ (see Figure 11), then $\tau = \tau_w(R - y)/R$. Then, solving Equation (2) for the change in velocity yields

$$dv = \left[\frac{\tau_w(R-y)}{R\mu}\right] dy = -\left(\frac{\tau_w}{R\mu}\right) r\, dr \tag{22}$$

When fluid viscosity is lower near the wall than at the center (because of external heating of liquid or cooling of gas by heat transfer through the pipe wall), the velocity gradient is steeper near the wall and flatter near the center, so the profile is generally flattened. When liquid is cooled or gas is heated, the velocity profile is more pointed for laminar flow (Figure 11). Calculations for such flows of gases and liquid metals in pipes are in Deissler (1951). Occurrences in turbulent flow are less apparent than in laminar flow. If enough heating is applied to gaseous flows, the viscosity increase can cause reversion to laminar flow.

Buoyancy effects and the gradual approach of the fluid temperature to equilibrium with that outside the pipe can cause considerable variation in the velocity profile along the conduit. Colborne and Drobitch (1966) found the pipe factor for upward vertical flow of hot air at a Re < 2000 reduced to about 0.6 at 40 diameters from the entrance, then increased to about 0.8 at 210 diameters, and finally decreased to the isothermal value of 0.5 at the end of 320 diameters.

FLOW ANALYSIS

Fluid flow analysis is used to correlate pressure changes with flow rates and the nature of the conduit. For a given pipeline, either the pressure drop for a certain flow rate, or the flow rate for a certain pressure difference between the ends of the conduit, is needed. Flow analysis ultimately involves comparing a pump or blower to a conduit piping system for evaluating the expected flow rate.

Generalized Bernoulli Equation

Internal energy differences are generally small, and usually the only significant effect of heat transfer is to change the density ρ. For gas or vapor flows, use the generalized Bernoulli equation in the pressure-over-density form of Equation (12), allowing for the thermodynamic process in the pressure-density relation:

$$-\int_1^2 \frac{dp}{\rho} + \alpha_1\frac{V_1^2}{2} + E_M = \alpha_2\frac{V_2^2}{2} + E_L \tag{23}$$

Elevation changes involving z are often negligible and are dropped. The pressure form of Equation (10) is generally unacceptable when appreciable density variations occur, because the volumetric flow rate differs at the two stations. This is particularly serious in friction-loss evaluations where the density usually varies over considerable lengths of conduit (Benedict and Carlucci 1966). When the flow is essentially incompressible, Equation (20) is satisfactory.

Example 1. Specify a blower to produce isothermal airflow of 400 cfm through a ducting system (Figure 12). Accounting for intake and fitting losses, equivalent conduit lengths are 60 and 165 ft, and flow is isothermal. Head at the inlet (station 1) and following the discharge (station 4), where velocity is zero, is the same. Frictional losses H_L are evaluated as 24.5 ft of air between stations 1 and 2, and 237 ft between stations 3 and 4.

Solution: The following form of the generalized Bernoulli relation is used in place of Equation (12), which also could be used:

$$(p_1/\rho_1 g) + \alpha_1(V_1^2/2g) + z_1 + H_M = (p_2/\rho_2 g) + \alpha_2(V_2^2/2g) + z_2 + H_L \tag{24}$$

The term $V_1^2/2g$ can be calculated as follows:

$$A_1 = \pi\left(\frac{D}{2}\right)^2 = \pi\left(\frac{9/12}{2}\right)^2 = 0.44 \text{ ft}^2$$

$$V_1 = Q/A_1 = \left(\frac{400 \text{ ft}^3}{\text{min}}\right)\left(\frac{1 \text{ min}}{60\text{s}}\right)/0.44 \text{ ft}^2 = 15.1 \text{ ft/s} \tag{25}$$

$$V_1^2/2g = (15.1)^2/2(32) = 3.56 \text{ ft}$$

The term $V_2^2/2g$ can be calculated in a similar manner.

In Equation (24), H_M is evaluated by applying the relation between any two points on opposite sides of the blower. Because conditions at stations 1 and 4 are known, they are used, and the location-specifying subscripts on the right side of Equation (24) are changed to 4. Note that $p_1 = p_4 = p$, $\rho_1 = \rho_4 = \rho$, and $V_1 = V_4 = 0$. Thus,

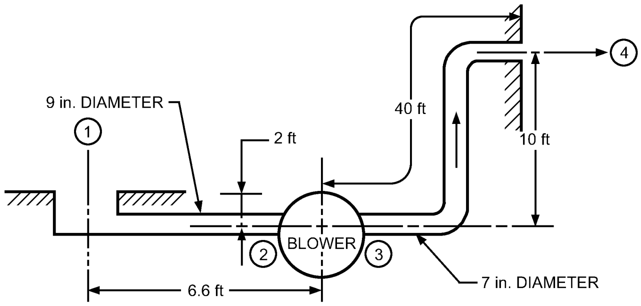

Fig. 12 Blower and Duct System for Example 1

$$(p/\rho g) + 0 + 2 + H_M = (p/\rho g) + 0 + 10 + (24.5 + 237) \tag{26}$$

so H_M = 269.5 ft of air. For standard air, this corresponds to 3.89 in. of water.

The head difference measured across the blower (between stations 2 and 3) is often taken as H_M. It can be obtained by calculating the static pressure at stations 2 and 3. Applying Equation (24) successively between stations 1 and 2 and between 3 and 4 gives

$$(p_1/\rho g) + 0 + 2 + 0 = (p_2/\rho g) + (1.06 \times 3.56) + 0 + 24.5$$
$$(p_3/\rho g) + (1.03 \times 9.70) + 0 + 0 = (p_4/\rho g) + 0 + 10 + 237 \tag{27}$$

where α just ahead of the blower is taken as 1.06, and just after the blower as 1.03; the latter value is uncertain because of possible uneven discharge from the blower. Static pressures p_1 and p_4 may be taken as zero gage. Thus,

$$p_2/\rho g = -26.2 \text{ ft of air}$$
$$p_3/\rho g = 237 \text{ ft of air} \tag{28}$$

The difference between these two numbers is 263.2 ft, which is not the H_M calculated after Equation (24) as 269.5 ft. The apparent discrepancy results from ignoring the velocity at stations 2 and 3. Actually, H_M is

$$\begin{aligned} H_M &= (p_3/\rho g) + \alpha_3(V_3^2/2g) - [(p_2/\rho g) + \alpha_2(V_2^2/2g)] \\ &= 237 + (1.03 \times 9.70) - [-26.2 + (1.06 \times 3.54)] \\ &= 247 - (-22.5) = 269.5 \text{ ft of air} \end{aligned} \tag{29}$$

The required blower head is the same, no matter how it is evaluated. It is the specific energy added to the system by the machine. Only when the conduit size and velocity profiles on both sides of the machine are the same is E_M or H_M simply found from $\Delta p = p_3 - p_2$.

Conduit Friction

The loss term E_L or H_L of Equation (12) or (13) accounts for friction caused by conduit-wall shearing stresses and losses from conduit-section changes. H_L is the head loss (i.e., loss of energy per unit weight).

In real-fluid flow, a frictional shear occurs at bounding walls, gradually influencing flow further away from the boundary. A lateral velocity profile is produced and flow energy is converted into heat (fluid internal energy), which is generally unrecoverable (a loss). This loss in fully developed conduit flow is evaluated using the **Darcy-Weisbach equation**:

$$H_{Lf} = f\left(\frac{L}{D}\right)\left(\frac{V^2}{2g}\right) \tag{30}$$

where L is the length of conduit of diameter D and f is the **Darcy-Weisbach friction factor**. Sometimes a numerically different relation is used with the **Fanning friction factor** (1/4 of the Darcy friction factor f). The value of f is nearly constant for turbulent flow, varying only from about 0.01 to 0.05.