76- Applications of Bernoulli's Principle:1

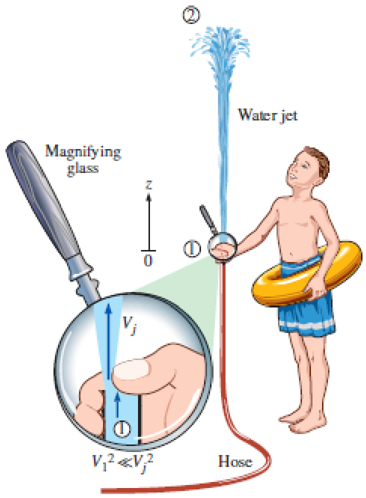


FIGURE 5–39
Schematic for Example 5–5. Inset shows a magnified view of the hose outlet region.

Applications of the Bernoulli Equation

So far, we have discussed the fundamental aspects of the Bernoulli equation. Now, we demonstrate its use in a wide range of applications through examples.

EXAMPLE 5–5 Spraying Water into the Air

Water is flowing from a garden hose (Fig. 5–39). A child places his thumb to cover most of the hose outlet, causing a thin jet of high-speed water to emerge. The pressure in the hose just upstream of his thumb is 400 kPa. If the hose is held upward, what is the maximum height that the jet could achieve?

SOLUTION Water from a hose attached to the water main is sprayed into the air. The maximum height the water jet can rise is to be determined.

Assumptions **1** The flow exiting into the air is steady, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The surface tension effects are negligible. **3** The friction between the water and air is negligible. **4** The irreversibilities that occur at the outlet of the hose due to abrupt contraction are not taken into account.

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

Analysis This problem involves the conversion of flow, kinetic, and potential energies to each other without involving any pumps, turbines, and wasteful components with large frictional losses, and thus it is suitable for the use of the Bernoulli equation. The water height will be maximum under the stated assumptions. The velocity inside the hose is negligibly small compared to that of the jet ($V_1^2 \ll V_j^2$, see magnified portion of Fig. 5–39) and we take the elevation just below the hose outlet as the reference level ($z_1 = 0$). At the top of the water trajectory $V_2 = 0$, and atmospheric pressure pertains. Then the Bernoulli equation along a streamline from 1 to 2 simplifies to

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

Solving for z_2 and substituting,

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

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

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

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

EXAMPLE 5–6 Water Discharge from a Large Tank

A large tank open to the atmosphere is filled with water to a height of 5 m from the outlet tap (Fig. 5–40). A tap near the bottom of the tank is now opened, and water flows out from the smooth and rounded outlet. Determine the maximum water velocity at the outlet.

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

Assumptions **1** The flow is incompressible and irrotational (except very close to the walls). **2** The water drains slowly enough that the flow can be approximated as steady (actually quasi-steady when the tank begins to drain). **3** Irreversible losses in the tap region are neglected.

Analysis This problem involves the conversion of flow, kinetic, and potential energies to each other without involving any pumps, turbines, and wasteful components with large frictional losses, and thus it is suitable for the use of the Bernoulli equation. We take point 1 to be at the free surface of water so that $P_1 = P_{atm}$ (open to the atmosphere), V_1 is negligibly small compared to V_2 (the tank diameter is very large relative to the outlet diameter), $z_1 = 5$ m, and $z_2 = 0$ (we take the reference level at the center of the outlet). Also, $P_2 = P_{atm}$ (water discharges into the atmosphere). For flow along a streamline from 1 to 2, the Bernoulli equation simplifies to

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

Solving for V_2 and substituting,

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

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

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

Discussion If the orifice were sharp-edged instead of rounded, then the flow would be disturbed, and the average exit velocity would be less than 9.9 m/s. Care must be exercised when attempting to apply the Bernoulli equation to situations where abrupt expansions or contractions occur since the friction and flow disturbance in such cases may not be negligible. From conservation of mass, $(V_1/V_2)^2 = (D_2/D_1)^4$. So, for example, if $D_2/D_1 = 0.1$, then $(V_1/V_2)^2 = 0.0001$, and our approximation that $V_1^2 \ll V_2^2$ is justified.

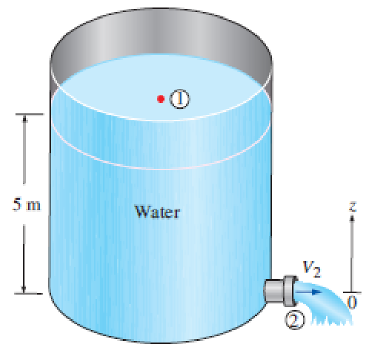
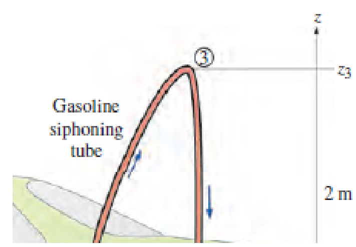


FIGURE 5–40
Schematic for Example 5–6.



(ref. 'Fluid Mechanics' by & Cimbala)

77- Applications of Bernoulli's Principle:2

EXAMPLE 5–8 **Velocity Measurement by a Pitot Tube**

A piezometer and a Pitot tube are tapped into a horizontal water pipe, as shown in Fig. 5–42, to measure static and stagnation (static + dynamic) pressures. For the indicated water column heights, determine the velocity at the center of the pipe.

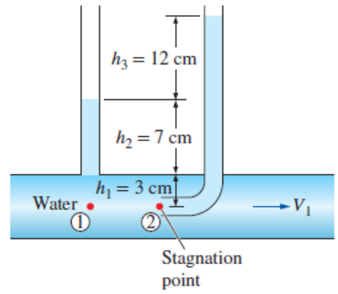


FIGURE 5–42
Schematic for Example 5–8.

SOLUTION The static and stagnation pressures in a horizontal pipe are measured. The velocity at the center of the pipe is to be determined.

Assumptions **1** The flow is steady and incompressible. **2** Points 1 and 2 are close enough together that the irreversible energy loss between these two points is negligible, and thus we can use the Bernoulli equation.

Analysis We take points 1 and 2 along the streamline at the centerline of the pipe, with point 1 directly under the piezometer and point 2 at the tip of the Pitot tube. This is a steady flow with straight and parallel streamlines, and the gage pressures at points 1 and 2 can be expressed as

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

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

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

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

Substituting the P_1 and P_2 expressions gives

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

Solving for V_1 and substituting,

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

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

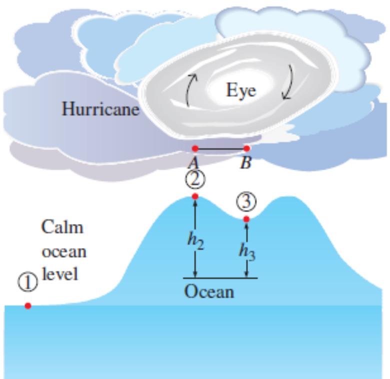


FIGURE 5–43
Schematic for Example 5–9. The vertical scale is greatly exaggerated.

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

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

Figure 5–43 depicts a hurricane hovering over the ocean swell below. The atmospheric pressure 200 mi from the eye is 30.0 in Hg (at point 1, generally normal for the ocean) and the winds are calm. The atmospheric pressure at the eye of the storm is 22.0 in Hg. Estimate the ocean swell at (*a*) the eye of the hurricane at point 3 and (*b*) point 2, where the wind velocity is 155 mph. Take the density of seawater and mercury to be 64 lbm/ft^3 and 848 lbm/ft^3, respectively, and the density of air at normal sea-level temperature and pressure to be 0.076 lbm/ft^3.

SOLUTION A hurricane is moving over the ocean. The amount of ocean swell at the eye and at active regions of the hurricane are to be determined.

Assumptions **1** The airflow within the hurricane is steady, incompressible, and irrotational (so that the Bernoulli equation is applicable). (This is certainly a very questionable assumption for a highly turbulent flow, but it is justified in the discussion.) **2** The effect of water sucked into the air is negligible.

Properties The densities of air at normal conditions, seawater, and mercury are given to be 0.076 lbm/ft^3, 64.0 lbm/ft^3, and 848 lbm/ft^3, respectively.

Analysis (*a*) Reduced atmospheric pressure over the water causes the water to rise. Thus, decreased pressure at point 2 relative to point 1 causes the ocean water to rise at point 2. The same is true at point 3, where the storm air velocity is negligible. The pressure difference given in terms of the mercury column height is expressed in terms of the seawater column height by

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

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

$$h_3 = \frac{\rho_{\text{Hg}}}{\rho_{\text{sw}}} h_{\text{Hg}} = \left(\frac{848 \text{ lbm/ft}^3}{64.0 \text{ lbm/ft}^3}\right)[(30 - 22) \text{ in Hg}]\left(\frac{1 \text{ ft}}{12 \text{ in}}\right) = \mathbf{8.83 \text{ ft}}$$

which is equivalent to the storm surge at the *eye of the hurricane* (Fig. 5–44) since the wind velocity there is negligible and there are no dynamic effects.

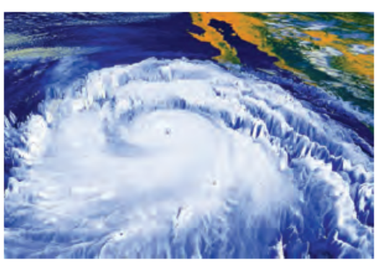


FIGURE 5–44
The eye of hurricane Linda (1997 in the Pacific Ocean near Baja California) is clearly visible in this satellite photo.
© Brand X Pictures/PunchStock RF

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

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

Substituting,

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

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

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

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

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

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

$$h_2 = h_3 + h_{\text{dynamic}} = 8.83 + 0.70 = \mathbf{9.53 \text{ ft}}$$

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

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

(ref. 'Fluid Mechanics' by & Cimbala)

78- Applications of Bernoulli’s Principle:3

EXAMPLE 5–10 Bernoulli Equation for Compressible Flow

Derive the Bernoulli equation when the compressibility effects are not negligible for an ideal gas undergoing (*a*) an isothermal process and (*b*) an isentropic process.

SOLUTION The Bernoulli equation for compressible flow is to be obtained for an ideal gas for isothermal and isentropic processes.

Assumptions **1** The flow is steady and frictional effects are negligible. **2** The fluid is an ideal gas, so the relation $P = \rho RT$ is applicable. **3** The specific heats are constant so that $P/\rho k =$ constant during an isentropic process.

Analysis (*a*) When the compressibility effects are significant and the flow cannot be assumed to be incompressible, the Bernoulli equation is given by Eq. 5–40 as

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

FIGURE 5–45
Compressible flow of a gas through turbine blades is often modeled as isentropic, and the compressible form of the Bernoulli equation is a reasonable approximation.
© Corbis RF

The compressibility effects can be properly accounted for by performing the integration $\int dP/\rho$ in Eq. 1. But this requires a relation between P and ρ for the process. For the *isothermal* expansion or compression of an ideal gas, the integral in Eq. 1 is performed easily by noting that T = constant and substituting $\rho = P/RT$,

$$\int \frac{dP}{\rho} = \int \frac{dP}{P/RT} = RT \ln P$$

Substituting into Eq. 1 gives the desired relation,

Isothermal process:
$$RT \ln P + \frac{V^2}{2} + gz = \text{constant} \tag{2}$$

(*b*) A more practical case of compressible flow is the *isentropic flow of ideal gases* through equipment that involves high-speed fluid flow such as nozzles, diffusers, and the passages between turbine blades (Fig. 5–45). Isentropic (i.e., reversible and adiabatic) flow is closely approximated by these devices, and it is characterized by the relation $P/\rho^k = C$ = constant, where k is the specific heat ratio of the gas. Solving for ρ from $P/\rho^k = C$ gives $\rho = C^{-1/k}P^{1/k}$. Performing the integration,

$$\int \frac{dP}{\rho} = \int C^{1/k}P^{-1/k}\, dP = C^{1/k}\frac{P^{-1/k+1}}{-1/k+1} = \frac{P^{1/k}}{\rho}\frac{P^{-1/k+1}}{-1/k+1} = \left(\frac{k}{k-1}\right)\frac{P}{\rho} \tag{3}$$

Substituting, the Bernoulli equation for steady, isentropic, compressible flow of an ideal gas becomes

Isentropic flow:
$$\left(\frac{k}{k-1}\right)\frac{P}{\rho} + \frac{V^2}{2} + gz = \text{constant} \tag{4a}$$

or

$$\left(\frac{k}{k-1}\right)\frac{P_1}{\rho_1} + \frac{V_1^2}{2} + gz_1 = \left(\frac{k}{k-1}\right)\frac{P_2}{\rho_2} + \frac{V_2^2}{2} + gz_2 \tag{4b}$$

A common practical situation involves the acceleration of a gas from rest (stagnation conditions at state 1) with negligible change in elevation. In that case we have $z_1 = z_2$ and $V_1 = 0$. Noting that $\rho = P/RT$ for ideal gases, P/ρ^k = constant for isentropic flow, and the Mach number is defined as Ma = V/c where $c = \sqrt{kRT}$ is the local speed of sound for ideal gases, Eq. 4b simplifies to

$$\frac{P_1}{P_2} = \left[1 + \left(\frac{k-1}{2}\right)\text{Ma}_2^2\right]^{k/(k-1)} \tag{4c}$$

where state 1 is the stagnation state and state 2 is any state along the flow.

Discussion It can be shown that the results obtained using the compressible and incompressible equations deviate no more than 2 percent when the Mach number is less than 0.3. Therefore, the flow of an ideal gas can be considered to be incompressible when Ma $\lesssim$ 0.3. For atmospheric air at normal conditions, this corresponds to a flow speed of about 100 m/s or 360 km/h.

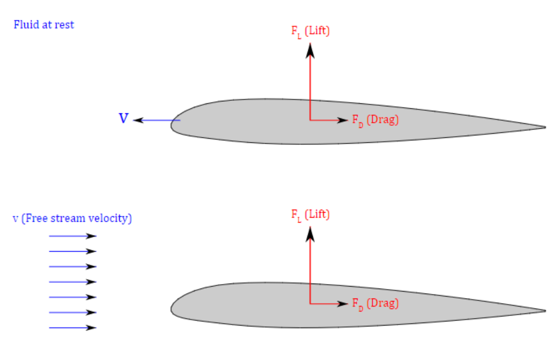
(ref. 'Fluid Mechanics' by & Cimbala)

79- Applications of Bernoulli’s Principle:4

Drag and Lift:

An object moving through a fluid will experience a complicated set of forces acting on it. In order to understand the behavior of the object as it moves through the fluid, we will focus on two resultant forces–drag and lift. The drag force acts parallel to the motion of the object while the lift force acts in a direction perpendicular to the motion.

Instead of considering an object moving through a fluid at rest, we can consider the same object at rest in a fluid stream of the same velocity. The object will experience the same set of forces as illustrated in the picture below. In this case the drag and lift forces will act parallel and perpendicular to the velocity of the fluid.



The upper figure shows an airfoil moving to the left with velocity v in a fluid at rest. The lower figure shows the same airfoil at rest in a fluid stream moving to the right with velocity v. Both airfoils will experience the same set of forces.

Drag Force equation

The drag force on an object is

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

In the above expression, C_D is a dimensionless number known as the drag coefficient. ρ is the density of the fluid. v is the free stream velocity of the fluid relative to the body. A is a *characteristic area* of the body and will be defined carefully for each object in the subsequent sections. It is usually taken as the largest cross-sectional area of the body perpendicular to the flow, also known as the projected area.

The combined term $\rho v^2/2$ is called the *dynamic pressure*. Note that the drag force is proportional to the dynamic pressure and therefore the velocity squared. If the velocity increases by a factor of two the drag force will increase by a factor of four.

The drag coefficient, C_D, depends on the shape and orientation of the body. It also depends on the Reynolds number and the roughness of the object’s surface. It can also be influenced by other bodies in the vicinity of the object. Ultimately the drag coefficient will be taken from experimental data.

(ref. https://kdusling.github.io/teaching/Applied-Fluids/Notes/DragAndLift)

80- Applications of Bernoulli's Principle:5

28.5: Worked Examples- Bernoulli's Equation

Example 28.5.1: Venturi Meter

Figure 28.8 shows a Venturi Meter, a device used to measure the speed of a fluid in a pipe. A fluid of density ρ_f is flowing through a pipe. A U-shaped tube partially filled with mercury of density ρ_{Hg} lies underneath the points 1 and 2.

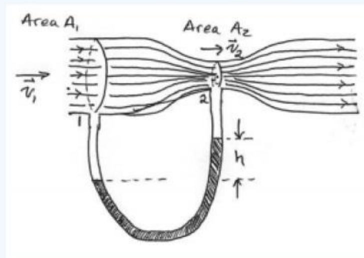


Figure 28.8: Venturi Meter

The cross-sectional areas of the pipe at points 1 and 2 are A_1 and A_2 respectively. Determine an expression for the flow speed at the point 1 in terms of the cross-sectional areas A_1 and A_2, and the difference in height h of the liquid levels of the two arms of the U-shaped tube.

Solution

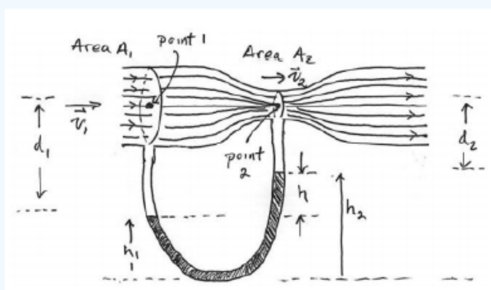


Figure 28.8: Coordinate system for Venturi tube

We shall assume that the pressure and speed are constant in the cross-sectional areas A_1 and A_2. We also assume the fluid is incompressible so the density ρ_f is constant throughout the tube. The two points 1 and 2 lie on the streamline passing through the midpoint of the tube so they are at the same height. Using $y_1 = y_2$ in Equation (28.4.8), the pressure and flow speeds at the two points 1 and 2 are related by

$$P_1 + \frac{1}{2}\rho_f v_1^2 = P_2 + \frac{1}{2}\rho_f v_2^2$$

We can rewrite Equation (28.4.10) as

$$P_1 - P_2 = \frac{1}{2}\rho_f \left(v_2^2 - v_1^2\right)$$

Let h1 and h2 denote the heights of the liquid level in the arms of the U-shaped tube directly beneath points 1 and 2 respectively. **Pascal's Law** relates the pressure difference between the two arms of the U-shaped tube according to in the left arm of the U-shaped tube according to

$$P_{\text{bottom}} = P_1 + \rho_f g d_1 + \rho_{Hg} g h_1$$

In a similar fashion, the pressure at point 2 is given by

$$P_{\text{bottom}} = P_2 + \rho_f g d_2 + \rho_{Hg} g h_2$$

Therefore, setting Equation (28.4.12) equal to Equation (28.4.13), we determine that the pressure difference on the two sides of the U-shaped tube is

$$P_1 - P_2 = \rho_f g (d_2 - d_1) + \rho_{Hg} g (h_2 - h_1)$$

From Figure 28.8, $d_2 + h_2 = d_1 + h_1$, therefore $d_2 - d_1 = h_1 - h_2 = -h$ We can rewrite Equation (28.4.14) as

$$P_1 - P_2 = (\rho_{Hg} - \rho_f)\, gh$$

Substituting Equation (28.4.11) into Equation (28.4.15) yields

$$\frac{1}{2}\rho_f \left(v_2^2 - v_1^2\right) = (\rho_{Hg} - \rho_f)\, gh$$

The mass continuity condition (Equation(28.3.5)) implies that $v_2 = (A_1/A_2)\, v_1$ and so we can rewrite Equation (28.4.16) as

$$\frac{1}{2}\rho_f \left((A_1/A_2)^2 - 1\right) v_1^2 = (\rho_{Hg} - \rho_f)\, gh$$

We can now solve Equation (28.4.17) for the speed of the flow at point 1;

$$v_1 = \sqrt{\frac{2\,(\rho_{Hg} - \rho_f)\, gh}{\rho_f \left((A_1/A_2)^2 - 1\right)}}$$

(ref. https://phys.libretexts.org/Bookshelves/Classical_Mechanics/Classical_Mechanics_(Dourmashkin)/28:_Fluid_Dynamics/28.05:_Worked_Examples-_Bernoullis_Equation)

81- Applications of Bernoulli's Principle:6

85 12.1 Flow Rate and Its Relation to Velocity

Summary

- Calculate flow rate.
- Define units of volume.
- Describe incompressible fluids.
- Explain the consequences of the equation of continuity.

Flow rate Q is defined to be the volume of fluid passing by some location through an area during a period of time, as seen in Figure 1. In symbols, this can be written as

$$Q = \frac{V}{t},$$

where V is the volume and t is the elapsed time.

The SI unit for flow rate is $\mathbf{m^3/s}$, but a number of other units for Q are in common use. For example, the heart of a resting adult pumps blood at a rate of 5.00 liters per minute (L/min). Note that a **liter** (L) is 1/1000 of a cubic meter or 1000 cubic centimeters ($\mathbf{10^{-3}\ m^3}$ or $\mathbf{10^3\ cm^3}$). In this text we shall use whatever metric units are most convenient for a given situation.

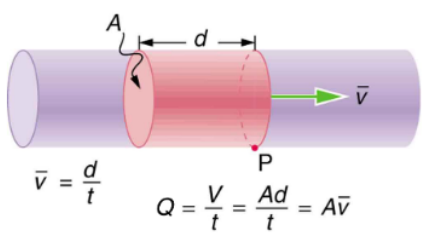


Figure 1. Flow rate is the volume of fluid per unit time flowing past a point through the area *A*. Here the shaded cylinder of fluid flows past point **P** in a uniform pipe in time *t*. The volume of the cylinder is *Ad* and the average velocity is $\bar{v}=d/t$ so that the flow rate is $Q=Ad/t=A\bar{v}$.

Example 1: Calculating Volume from Flow Rate: The Heart Pumps a Lot of Blood in a Lifetime

How many cubic meters of blood does the heart pump in a 75-year lifetime, assuming the average flow rate is 5.00 L/min?

Strategy

Time and flow rate Q are given, and so the volume V can be calculated from the definition of flow rate.

Solution

Solving $Q = V/t$ for volume gives

$$V = Qt.$$

Substituting known values yields

$$\begin{aligned} V &= \left(\frac{5.00\ \text{L}}{1\ \text{min}}\right)(75\ \text{y})\left(\frac{1\ \text{m}^3}{10^3\text{L}}\right)\left(5.26 \times 10^5\ \frac{\text{min}}{\text{y}}\right) \\ &= 2.0 \times 10^5\ \text{m}^3. \end{aligned}$$

Discussion

This amount is about 200,000 tons of blood. For comparison, this value is equivalent to about 200 times the volume of water contained in a 6-lane 50-m lap pool.

(ref. http://pressbooks-dev.oer.hawaii.edu/collegephysics/chapter/12-1-flow-rate-and-its-relation-to-velocity/#:~:text=We%20can%20useQ%3DA,the%20other%20variables%20are%20known.&text=The%20flow%20rate%20is%20given,2%20for%20a%20cylindrical%20vessel.)

82- Applications of Bernoulli's Principle:7

Poiseuille's Law

Poiseuille's law states that the flow of a fluid depends on different variables such as the radius (R) and length of the tube (L), pressure gradient (ΔP), and the viscosity of the fluid (ν) as per their relationship. Poiseuille's Formula is represented as

$$Q = \frac{\Delta P \pi R^4}{8 \nu L}.$$

Solved Example

Calculate the average speed of the blood when the blood flow through a large artery of radius 2.5 mm is found to be 20 cm long. The pressure across the ends of the artery is known as 380 Pa.

Solution: Blood viscosity n = 0.0027 N .s/m^2

l = 20 cm

Radius = 2.5 mm

The difference of pressure = 380 Pa (P1 – P2)

The average speed is given by

Q = $\Delta P \pi r^4 / 8 \eta l$

$$Q = \frac{(380 \times 3.906 \times 10 - 11 \times 3.14)}{(8 \times 0.0027 \times 0.20)}$$

The average speed becomes **1.0789** m / s

(ref. https://collegedunia.com/exams/poiseuille-law-formula-derivation-solved-examples-physics-articleid-4802)

83- Newton's Law of Motion: 1

CHAPTER

6

MOMENTUM ANALYSIS OF FLOW SYSTEMS

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

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

OBJECTIVES

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

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

(ref. 'Fluid Mechanics' by & Cimbala)

84- Newton's Law of Motion: 2

6–1 ▪ NEWTON'S LAWS

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

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

Newton's second law:
$$\vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = \frac{d(m\vec{V})}{dt} \qquad \textbf{(6–1)}$$

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

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

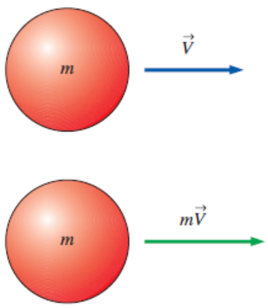


FIGURE 6–1
Linear momentum is the product of mass and velocity, and its direction is the direction of velocity.

(ref. 'Fluid Mechanics' by & Cimbala)

85- Newton's Law of Motion: 3

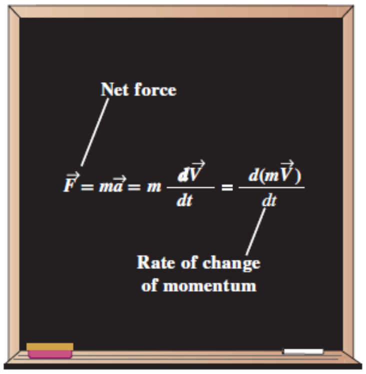


FIGURE 6–2
Newton's second law is also expressed as *the rate of change of the momentum of a body is equal to the net force acting on it.*

The momentum of a system remains constant only when the net force acting on it is zero, and thus the momentum of such a system is conserved. This is known as the *conservation of momentum principle.* This principle has proven to be a very useful tool when analyzing collisions such as those between balls; between balls and rackets, bats, or clubs; and between atoms or subatomic particles; and explosions such as those that occur in rockets, missiles, and guns. In fluid mechanics, however, the net force acting on a system is typically *not* zero, and we prefer to work with the linear momentum equation rather than the conservation of momentum principle.

Note that force, acceleration, velocity, and momentum are vector quantities, and as such they have direction as well as magnitude. Also, momentum is a constant multiple of velocity, and thus the direction of momentum is the direction of velocity as shown in Fig. 6–1. Any vector equation can be written in scalar form for a specified direction using magnitudes, e.g., $F_x = ma_x = d(mV_x)/dt$ in the x-direction.

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

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

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

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

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

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

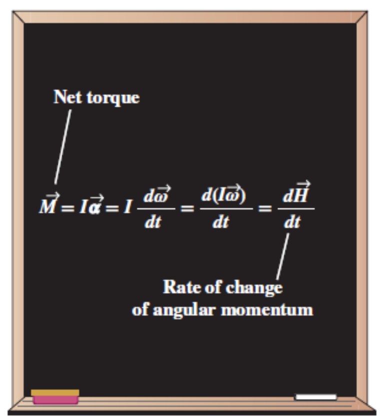


FIGURE 6–3
The rate of change of the angular momentum of a body is equal to the net torque acting on it.

(ref. 'Fluid Mechanics' by & Cimbala)

86- Forces Acting On a Control Volume:1

6–3 ▪ FORCES ACTING ON A CONTROL VOLUME

The forces acting on a control volume consist of **body forces** that act throughout the entire body of the control volume (such as gravity, electric, and magnetic forces) and **surface forces** that act on the control surface (such as pressure and viscous forces and reaction forces at points of contact). Only external forces are considered in the analysis. Internal forces (such as the pressure force between a fluid and the inner surfaces of the flow section) are not considered in a control volume analysis unless they are exposed by passing the control surface through that area.

ıd

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

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

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

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

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

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

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

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

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

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

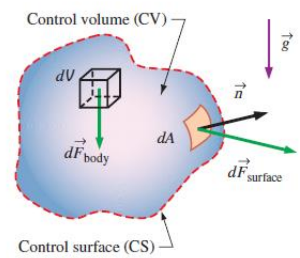


FIGURE 6–5
The total force acting on a control volume is composed of body forces and surface forces; body force is shown on a differential volume element, and surface force is shown on a differential surface element.

dv

(ref. ‘Fluid Mechanics’ by & Cimbala)

87- Forces Acting On a Control Volume:2

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

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

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

When the face is not parallel to one of the coordinate axes, mathematical laws for axes rotation and tensors can be used to calculate the normal and tangential components acting at the face. In addition, an alternate notation called **tensor notation** is convenient when working with tensors but is usually reserved for graduate studies. (For a more in-depth analysis of tensors and tensor notation see, for example, Kundu and Cohen, 2011.)

In Eq. 6–8, σ_{ij} is defined as the stress (force per unit area) in the *j*-direction acting on a face whose normal is in the *i*-direction. Note that *i* and *j* are merely *indices* of the tensor and are not the same as unit vectors $\vec{i}$ and $\vec{j}$. For example, σ_{xy} is defined as positive for the stress pointing in the *y*-direction on a face whose outward normal is in the *x*-direction. This component of the stress tensor, along with the other eight components, is shown in Fig. 6–8 for the case of a differential fluid element aligned with the axes in Cartesian coordinates. All the components in Fig. 6–8 are shown on positive faces (right, top, and front) and in their positive orientation by definition. Positive stress components on the *opposing* faces of the fluid element (not shown) point in exactly opposite directions.

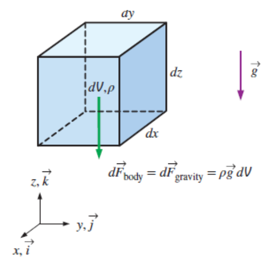


FIGURE 6–6
The gravitational force acting on a differential volume element of fluid is equal to its weight; the axes are oriented so that the gravity vector acts *downward* in the negative *z*-direction.

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

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

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

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

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

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

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

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

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

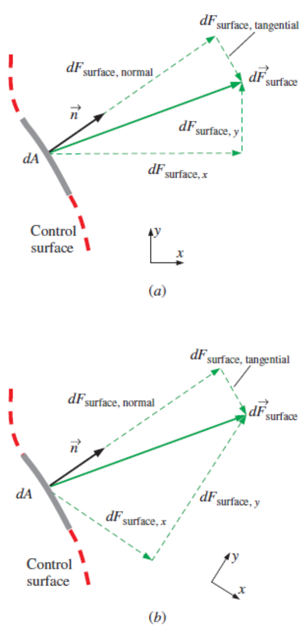


FIGURE 6–7
When coordinate axes are rotated (*a*) to (*b*), the components of the surface force change, even though the force itself remains the same; only two dimensions are shown here.

(ref. 'Fluid Mechanics' by & Cimbala)

88- Forces Acting On a Control Volume:3

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

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

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

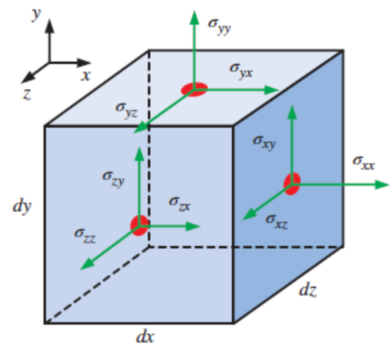


FIGURE 6–8
Components of the stress tensor in Cartesian coordinates on the right, top, and front faces.

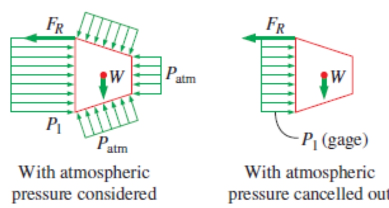
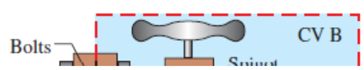


FIGURE 6–9
Atmospheric pressure acts in all directions, and thus it can be ignored when performing force balances since its effect cancels out in every direction.



(ref. 'Fluid Mechanics' by & Cimbala)

89- The Linear Momentum Equations

6–4 ▪ THE LINEAR MOMENTUM EQUATION

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

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

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

$$\sum \vec{F} = \frac{d}{dt}\int_{\text{sys}} \rho\vec{V}\,dV \qquad \textbf{(6–14)}$$

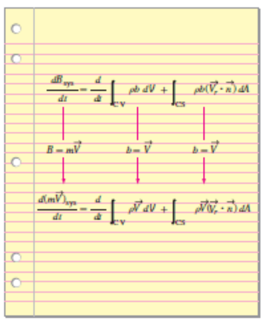


FIGURE 6–11
The linear momentum equation is obtained by replacing B in the Reynolds transport theorem by the momentum $m\vec{V}$, and b by the momentum per unit mass $\vec{V}$.

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

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

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

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

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

which is stated in words as

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

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

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

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

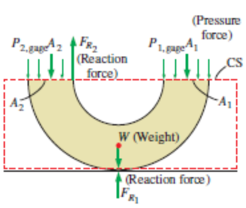


FIGURE 6–12
In most flow systems, the sum of forces $\Sigma\vec{F}$ consists of weights, pressure forces, and reaction forces. Gage pressures are used here since atmospheric pressure cancels out on all sides of the control surface.

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

(ref. 'Fluid Mechanics' by & Cimbala)

90- Review of Rotational Motion and Angular Momentum

6–5 ▪ REVIEW OF ROTATIONAL MOTION AND ANGULAR MOMENTUM

The motion of a rigid body can be considered to be the combination of translational motion of its center of mass and rotational motion about its center of mass. The translational motion is analyzed using the linear momentum equation, Eq. 6–1. Now we discuss the rotational motion—a motion during which all points in the body move in circles about the axis of rotation. Rotational motion is described with angular quantities such as angular distance θ, angular velocity $\vec{\omega}$, and angular acceleration $\vec{\alpha}$.

Newton's second law requires that there must be a force acting in the tangential direction to cause angular acceleration. The strength of the rotating effect, called the *moment* or *torque,* is proportional to the magnitude of the force and its distance from the axis of rotation. The perpendicular distance from the axis of rotation to the line of action of the force is called the *moment arm,* and the magnitude of torque M acting on a point mass m at normal distance r from the axis of rotation is expressed as

$$M = rF_t = rma_t = mr^2\alpha \tag{6–32}$$

The total torque acting on a rotating rigid body about an axis is determined by integrating the torque acting on differential mass δm over the entire body to give

Magnitude of torque:
$$M = \int_{\text{mass}} r^2\alpha\, \delta m = \left[\int_{\text{mass}} r^2\, \delta m\right]\alpha = I\alpha \tag{6–33}$$

where I is the *moment of inertia* of the body about the axis of rotation, which is a measure of the inertia of a body against rotation. The relation $M = I\alpha$ is the counterpart of Newton's second law, with torque replacing force, moment of inertia replacing mass, and angular acceleration replacing linear acceleration (Fig. 6–29). Note that unlike mass, the rotational inertia of a body also depends on the distribution of the mass of the body with respect to the axis of rotation. Therefore, a body whose mass is closely packed about its axis of rotation has a small resistance against angular acceleration, while a body whose mass is concentrated at its periphery has a large resistance against angular acceleration. A flywheel is a good example of the latter.

The linear momentum of a body of mass m having velocity $\vec{V}$ is $m\vec{V}$, and the direction of linear momentum is identical to the direction of velocity.

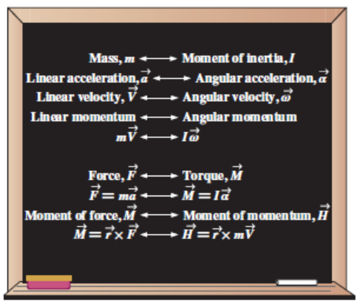


FIGURE 6–29
Analogy between corresponding linear and angular quantities.

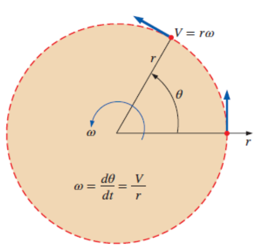


FIGURE 6–28
The relations between angular distance θ, angular velocity ω, and linear velocity V in a plane.

The amount of rotation of a point in a body is expressed in terms of the angle θ swept by a line of length r that connects the point to the axis of rotation and is perpendicular to the axis. The angle θ is expressed in radians (rad), which is the arc length corresponding to θ on a circle of unit radius. Noting that the circumference of a circle of radius r is $2\pi r$, the angular distance traveled by any point in a rigid body during a complete rotation is 2π rad. The physical distance traveled by a point along its circular path is $l = \theta r$, where r is the normal distance of the point from the axis of rotation and θ is the angular distance in rad. Note that 1 rad corresponds to $360/(2\pi) \cong 57.3°$.

The magnitude of angular velocity ω is the angular distance traveled per unit time, and the magnitude of angular acceleration α is the rate of change of angular velocity. They are expressed as (Fig. 6–28),

$$\omega = \frac{d\theta}{dt} = \frac{d(l/r)}{dt} = \frac{1}{r}\frac{dl}{dt} = \frac{V}{r} \quad \text{and} \quad \alpha = \frac{d\omega}{dt} = \frac{d^2\theta}{dt^2} = \frac{1}{r}\frac{dV}{dt} = \frac{a_t}{r} \tag{6–30}$$

or

$$V = r\omega \quad \text{and} \quad a_t = r\alpha \tag{6–31}$$

where V is the linear velocity and a_t is the linear acceleration in the tangential direction for a point located at a distance r from the axis of rotation. Note that ω and α are the same for all points of a rotating rigid body, but V and a_t are not (they are proportional to r).

(ref. 'Fluid Mechanics' by & Cimbala)

Noting that the moment of a force is equal to the product of the force and the normal distance, the magnitude of the moment of momentum, called the **angular momentum**, of a point mass m about an axis is expressed as $H = rmV = r^2m\omega$, where r is the normal distance from the axis of rotation to the line of action of the momentum vector (Fig. 6–30). Then the total angular momentum of a rotating rigid body is determined by integration to be

Magnitude of angular momentum: $$H = \int_{\text{mass}} r^2\omega\,\delta m = \left[\int_{\text{mass}} r^2\,\delta m\right]\omega = I\omega \quad \textbf{(6–34)}$$

where again I is the *moment of inertia* of the body about the axis of rotation. It can also be expressed more generally in vector form as

$$\vec{H} = I\vec{\omega} \quad \textbf{(6–35)}$$

Note that the angular velocity $\vec{\omega}$ is the same at every point of a rigid body.

Newton's second law $\vec{F} = m\vec{a}$ was expressed in terms of the rate of change of linear momentum in Eq. 6–1 as $\vec{F} = d(m\vec{V})/dt$. Likewise, the counterpart of Newton's second law for rotating bodies $\vec{M} = I\vec{\alpha}$ is expressed in Eq. 6–2 in terms of the rate of change of angular momentum as

Angular momentum equation: $$\vec{M} = I\vec{\alpha} = I\frac{d\vec{\omega}}{dt} = \frac{d(I\vec{\omega})}{dt} = \frac{d\vec{H}}{dt} \quad \textbf{(6–36)}$$

where $\vec{M}$ is the net torque applied on the body about the axis of rotation.

The angular velocity of rotating machinery is typically expressed in rpm (number of revolutions per minute) and denoted by $\dot{n}$. Noting that velocity is distance traveled per unit time and the angular distance traveled during each revolution is 2π, the angular velocity of rotating machinery is $\omega = 2\pi\dot{n}$ rad/min or

Angular velocity versus rpm: $$\omega = 2\pi\dot{n}\ \text{(rad/min)} = \frac{2\pi\dot{n}}{60}\ \text{(rad/s)} \quad \textbf{(6–37)}$$

Consider a constant force F acting in the tangential direction on the outer surface of a shaft of radius r rotating at an rpm of $\dot{n}$. Noting that work W is force times distance, and power $\dot{W}$ is work done per unit time and thus force times velocity, we have $\dot{W}_{\text{shaft}} = FV = Fr\omega = M\omega$. Therefore, the power transmitted by a shaft rotating at an rpm of $\dot{n}$ under the influence of an applied torque M is (Fig. 6–31)

Shaft power: $$\dot{W}_{\text{shaft}} = \omega M = 2\pi\dot{n}M \quad \textbf{(6–38)}$$

The kinetic energy of a body of mass m during translational motion is $\text{KE} = \frac{1}{2}mV^2$. Noting that $V = r\omega$, the rotational kinetic energy of a body of mass m at a distance r from the axis of rotation is $\text{KE} = \frac{1}{2}mr^2\omega^2$. The total rotational kinetic energy of a rotating rigid body about an axis is determined by integrating the rotational kinetic energies of differential masses dm over the entire body to give

Rotational kinetic energy: $$\text{KE}_r = \frac{1}{2}I\omega^2 \quad \textbf{(6–39)}$$

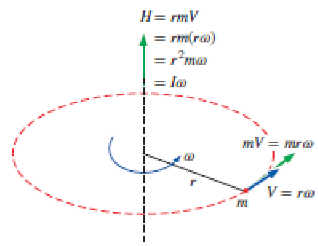


FIGURE 6–30
Angular momentum of point mass m rotating at angular velocity ω at distance r from the axis of rotation.

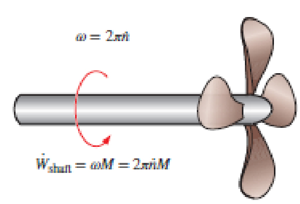


FIGURE 6–31
The relations between angular velocity, rpm, and the power transmitted through a rotating shaft.

(ref. 'Fluid Mechanics' by & Cimbala)

91- The Angular Momentum Equation:1

$\vec{M} = \vec{r} \times \vec{F}$

$M = Fr \sin\theta$

FIGURE 6–32
The moment of a force $\vec{F}$ about a point O is the vector product of the position vector $\vec{r}$ and $\vec{F}$.

6–6 ▪ THE ANGULAR MOMENTUM EQUATION

The linear momentum equation discussed in Section 6–4 is useful for determining the relationship between the linear momentum of flow streams and the resultant forces. Many engineering problems involve the moment of the linear momentum of flow streams, and the rotational effects caused by them. Such problems are best analyzed by the *angular momentum equation,* also called the *moment of momentum equation.* An important class of fluid devices, called *turbomachines,* which include centrifugal pumps, turbines, and fans, is analyzed by the angular momentum equation.

The *moment of a force* $\vec{F}$ about a point O is the vector (or cross) product (Fig. 6–32)

Moment of a force: $\vec{M} = \vec{r} \times \vec{F}$ **(6–40)**

where $\vec{r}$ is the position vector from point O to any point on the line of action of $\vec{F}$. The vector product of two vectors is a vector whose line of action is normal to the plane that contains the crossed vectors ($\vec{r}$ and $\vec{F}$ in this case) and whose magnitude is

Magnitude of the moment of a force: $M = Fr \sin\theta$ **(6–41)**

where θ is the angle between the lines of action of the vectors $\vec{r}$ and $\vec{F}$. Therefore, the magnitude of the moment about point O is equal to the magnitude of the force multiplied by the normal distance of the line of action of the force from the point O. The sense of the moment vector $\vec{M}$ is determined by the right-hand rule: when the fingers of the right hand are curled in the direction that the force tends to cause rotation, the thumb points the direction of the moment vector (Fig. 6–33). Note that a force whose line of action passes through point O produces zero moment about point O.

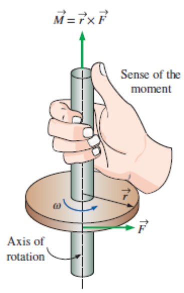


FIGURE 6–33
The determination of the direction of the moment by the right-hand rule.

The vector product of $\vec{r}$ and the momentum vector $m\vec{V}$ gives the *moment of momentum,* also called the *angular momentum,* about a point O as

Moment of momentum: $\vec{H} = \vec{r} \times m\vec{V}$ **(6–42)**

Therefore, $\vec{r} \times \vec{V}$ represents the angular momentum per unit mass, and the angular momentum of a differential mass $\delta m = \rho\, dV$ is $d\vec{H} = (\vec{r} \times \vec{V})\rho\, dV$. Then the angular momentum of a system is determined by integration to be

Moment of momentum (system): $\vec{H}_{sys} = \int_{sys} (\vec{r} \times \vec{V})\rho\, dV$ **(6–43)**

The rate of change of the moment of momentum is

Rate of change of moment of momentum: $\dfrac{d\vec{H}_{sys}}{dt} = \dfrac{d}{dt}\int_{sys} (\vec{r} \times \vec{V})\rho\, dV$ **(6–44)**

The angular momentum equation for a system was expressed in Eq. 6–2 as

$$\sum \vec{M} = \frac{d\vec{H}_{sys}}{dt} \qquad \textbf{(6–45)}$$

where $\Sigma\vec{M} = \Sigma(\vec{r} \times \vec{F})$ is the net torque or moment applied on the system, which is the vector sum of the moments of all forces acting on the system, and $d\vec{H}_{sys}/dt$ is the rate of change of the angular momentum of the system. Equation 6–45 is stated as *the rate of change of angular momentum of a system is equal to the net torque acting on the system.* This equation is valid for a fixed quantity of mass and an inertial reference frame, i.e., a reference frame that is fixed or moves with a constant velocity in a straight path.

The general control volume formulation of the angular momentum equation is obtained by setting $b = \vec{r} \times \vec{V}$ and thus $B = \vec{H}$ in the general Reynolds transport theorem. It gives (Fig. 6–34)

$$\frac{d\vec{H}_{sys}}{dt} = \frac{d}{dt}\int_{CV} (\vec{r} \times \vec{V})\rho\, dV + \int_{CS} (\vec{r} \times \vec{V})\rho(\vec{V}_r \cdot \vec{n})\, dA \qquad \textbf{(6–46)}$$

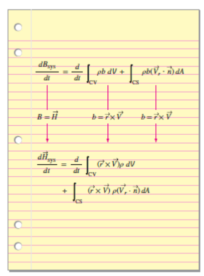


FIGURE 6–34
The angular momentum equation is obtained by replacing B in the Reynolds transport theorem by the angular momentum $\vec{H}$, and b by the angular momentum per unit mass $\vec{r} \times \vec{V}$.

The left-hand side of this equation is, from Eq. 6–45, equal to $\Sigma\vec{M}$. Substituting, the angular momentum equation for a general control volume (stationary or moving, fixed shape or distorting) is

General: $\sum \vec{M} = \dfrac{d}{dt}\int_{CV} (\vec{r} \times \vec{V})\rho\, dV + \int_{CS} (\vec{r} \times \vec{V})\rho(\vec{V}_r \cdot \vec{n})\, dA$ **(6–47)**

which is stated in words as

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

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

(ref. 'Fluid Mechanics' by & Cimbala)

92- The Angular Momentum Equation:2

FIGURE 6–35
A rotating lawn sprinkler is a good example of application of the angular momentum equation.
© John A. Rizzo/Getty Images RF

Special Cases

During *steady flow,* the amount of angular momentum within the control volume remains constant, and thus the time rate of change of angular momentum of the contents of the control volume is zero. Then,

Steady flow:
$$\sum \vec{M} = \int_{CS} (\vec{r} \times \vec{V})\rho(\vec{V}_r \cdot \vec{n})\, dA \qquad \textbf{(6–49)}$$

In many practical applications, the fluid crosses the boundaries of the control volume at a certain number of inlets and outlets, and it is convenient to replace the area integral by an algebraic expression written in terms of the average properties over the cross-sectional areas where the fluid enters or leaves the control volume. In such cases, the angular momentum flow rate can be expressed as the difference in the angular momentum of outgoing and incoming streams. Furthermore, in many cases the moment arm $\vec{r}$ is either constant along the inlet or outlet (as in radial flow turbomachines) or is large compared to the diameter of the inlet or outlet pipe (as in rotating lawn sprinklers, Fig. 6–35). In such cases, the *average* value of $\vec{r}$ is used throughout the cross-sectional area of the inlet or outlet. Then, an approximate form of the angular momentum equation in terms of average properties at inlets and outlets becomes

$$\sum \vec{M} \cong \frac{d}{dt}\int_{CV} (\vec{r} \times \vec{V})\rho\, dV + \sum_{out} (\vec{r} \times \dot{m}\vec{V}) - \sum_{in} (\vec{r} \times \dot{m}\vec{V}) \qquad \textbf{(6–50)}$$

You may be wondering why we don't introduce a correction factor into Eq. 6–50, like we did for conservation of energy (Chap. 5) and for conservation of linear momentum (Section 6–4). The reason is that the cross product of $\vec{r}$ and $\dot{m}\vec{V}$ is dependent on problem geometry, and thus, such a correction factor would vary from problem to problem. Therefore, whereas we can readily calculate a kinetic energy flux correction factor and a momentum flux correction factor for fully developed pipe flow that can be applied to various problems, we cannot do so for angular momentum. Fortunately, in many problems of practical engineering interest, the error associated with using average values of radius and velocity is small, and the approximation of Eq. 6–50 is reasonable.

If the flow is *steady,* Eq. 6–50 further reduces to (Fig. 6–36)

Steady flow:
$$\sum \vec{M} = \sum_{out} (\vec{r} \times \dot{m}\vec{V}) - \sum_{in} (\vec{r} \times \dot{m}\vec{V}) \qquad \textbf{(6–51)}$$

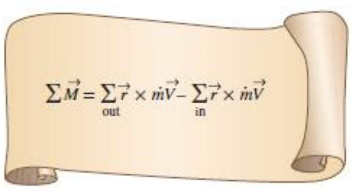


FIGURE 6–36
The net torque acting on a control volume during steady flow is equal to the difference between the outgoing and incoming angular momentum flow rates.

(ref. 'Fluid Mechanics' by & Cimbala)

93- The Angular Momentum Equation:3

Flow with No External Moments

When there are no external moments applied, the angular momentum equation Eq. 6–50 reduces to

No external moments:
$$0 = \frac{d\vec{H}_{CV}}{dt} + \sum_{out} (\vec{r} \times \dot{m}\vec{V}) - \sum_{in} (\vec{r} \times \dot{m}\vec{V}) \qquad \textbf{(6–53)}$$

This is an expression of the conservation of angular momentum principle, which can be stated as *in the absence of external moments, the rate of change of the angular momentum of a control volume is equal to the difference between the incoming and outgoing angular momentum fluxes.*

When the moment of inertia I of the control volume remains constant, the first term on the right side of Eq. 6–53 becomes simply moment of inertia times angular acceleration, $I\vec{\alpha}$. Therefore, the control volume in this case can be treated as a solid body, with a net torque of

$$\vec{M}_{body} = I_{body}\,\vec{\alpha} = \sum_{in} (\vec{r} \times \dot{m}\vec{V}) - \sum_{out} (\vec{r} \times \dot{m}\vec{V}) \qquad \textbf{(6–54)}$$

(due to a change of angular momentum) acting on it. This approach can be used to determine the angular acceleration of space vehicles and aircraft when a rocket is fired in a direction different than the direction of motion.

Radial-Flow Devices

Many rotary-flow devices such as centrifugal pumps and fans involve flow in the radial direction normal to the axis of rotation and are called *radial-flow devices* (Chap. 14). In a centrifugal pump, for example, the fluid enters the device in the axial direction through the eye of the impeller, turns outward as it flows through the passages between the blades of the impeller, collects in the scroll, and is discharged in the tangential direction, as shown in Fig. 6–37. Axial-flow devices are easily analyzed using the linear momentum equation. But radial-flow devices involve large changes in angular momentum of the fluid and are best analyzed with the help of the angular momentum equation.

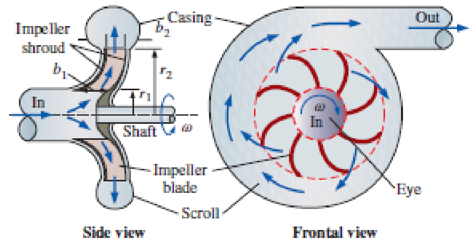


FIGURE 6–37
Side and frontal views of a typical centrifugal pump.

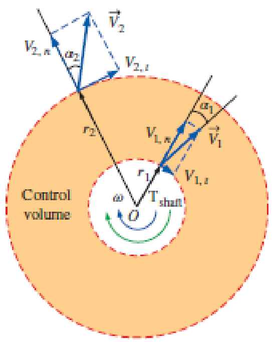


FIGURE 6–38
An annular control volume that encloses the impeller section of a centrifugal pump.

To analyze a centrifugal pump, we choose the annular region that encloses the impeller section as the control volume, as shown in Fig. 6–38. Note that the average flow velocity, in general, has normal and tangential components at both the inlet and the outlet of the impeller section. Also, when the shaft rotates at angular velocity ω, the impeller blades have tangential velocity ωr_1 at the inlet and ωr_2 at the outlet. For steady, incompressible flow, the conservation of mass equation is written as

$$\dot{V}_1 = \dot{V}_2 = \dot{V} \quad \rightarrow \quad (2\pi r_1 b_1)V_{1,n} = (2\pi r_2 b_2)V_{2,n} \qquad \textbf{(6–55)}$$

where b_1 and b_2 are the flow widths at the inlet where $r = r_1$ and at the outlet where $r = r_2$, respectively. (Note that the actual circumferential cross-sectional area is somewhat less than $2\pi rb$ since the blade thickness is not zero.) Then the average normal components $V_{1,n}$ and $V_{2,n}$ of absolute velocity can be expressed in terms of the volumetric flow rate $\dot{V}$ as

$$V_{1,n} = \frac{\dot{V}}{2\pi r_1 b_1} \quad \text{and} \quad V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} \qquad \textbf{(6–56)}$$

The normal velocity components $V_{1,n}$ and $V_{2,n}$ as well as pressure acting on the inner and outer circumferential areas pass through the shaft center, and thus they do not contribute to torque about the origin. Then only the tangential velocity components contribute to torque, and the application of the angular momentum equation $\sum M = \sum_{out} r\dot{m}V - \sum_{in} r\dot{m}V$ to the control volume gives

Euler's turbine equation:

$$T_{shaft} = \dot{m}(r_2 V_{2,t} - r_1 V_{1,t}) \qquad \textbf{(6–57)}$$

which is known as **Euler's turbine equation**. When the angles α_1 and α_2 between the direction of absolute flow velocities and the radial direction are known, Eq. 6–57 becomes

$$T_{shaft} = \dot{m}(r_2 V_2 \sin \alpha_2 - r_1 V_1 \sin \alpha_1) \qquad \textbf{(6–58)}$$

In the idealized case of the tangential fluid velocity being equal to the blade angular velocity both at the inlet and the exit, we have $V_{1,t} = \omega r_1$ and $V_{2,t} = \omega r_2$, and the torque becomes

$$T_{shaft,\,ideal} = \dot{m}\omega(r_2^2 - r_1^2) \qquad \textbf{(6–59)}$$

where $\omega = 2\pi\dot{n}$ is the angular velocity of the blades. When the torque is known, the shaft power is determined from $\dot{W}_{shaft} = \omega T_{shaft} = 2\pi\dot{n}T_{shaft}$.

(ref. 'Fluid Mechanics' by & Cimbala)

94- Summary

SUMMARY

This chapter deals mainly with the conservation of momentum for finite control volumes. The forces acting on the control volume consist of *body forces* that act throughout the entire body of the control volume (such as gravity, electric, and magnetic forces) and *surface forces* that act on the control surface (such as the pressure forces and reaction forces at points of contact). The sum of all forces acting on the control volume at a particular instant in time is represented by $\sum \vec{F}$ and is expressed as

$$\underbrace{\sum \vec{F}}_{\text{total force}} = \underbrace{\sum \vec{F}_{\text{gravity}}}_{\text{body force}} + \underbrace{\sum \vec{F}_{\text{pressure}} + \sum \vec{F}_{\text{viscous}} + \sum \vec{F}_{\text{other}}}_{\text{surface forces}}$$

Newton's second law can be stated as *the sum of all external forces acting on a system is equal to the time rate of change of linear momentum of the system.* Setting $b = \vec{V}$ and thus $B = m\vec{V}$ in the Reynolds transport theorem and utilizing Newton's second law gives the *linear momentum equation* for a control volume as

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

which reduces to the following special cases:

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

Unsteady flow (algebraic form):

$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho \vec{V}\, dV + \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$$

Steady flow (algebraic form): $$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$$

No external forces: $$0 = \frac{d(m\vec{V})_{CV}}{dt} + \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$$

where β is the momentum-flux correction factor. A control volume whose mass m remains constant can be treated as a solid body (a fixed-mass system) with a *net thrusting force* (also called simply the *thrust*) of

$$\vec{F}_{\text{thrust}} = m_{CV}\vec{a} = \sum_{\text{in}} \beta \dot{m}\vec{V} - \sum_{\text{out}} \beta \dot{m}\vec{V}$$

acting on the body.

Newton's second law can also be stated as *the rate of change of angular momentum of a system is equal to the net torque acting on the system.* Setting $b = \vec{r} \times \vec{V}$ and thus $B = \vec{H}$ in the general Reynolds transport theorem gives the *angular momentum equation* as

$$\sum \vec{M} = \frac{d}{dt}\int_{CV} (\vec{r} \times \vec{V})\rho\, dV + \int_{CS} (\vec{r} \times \vec{V})\rho(\vec{V}_r \cdot \vec{n})\, dA$$

which reduces to the following special cases:

Steady flow: $$\sum \vec{M} = \int_{CS} (\vec{r} \times \vec{V})\rho(\vec{V}_r \cdot \vec{n})\, dA$$

Unsteady flow (algebraic form):

$$\sum \vec{M} = \frac{d}{dt}\int_{CV} (\vec{r} \times \vec{V})\rho\, dV + \sum_{\text{out}} \vec{r} \times \dot{m}\vec{V} - \sum_{\text{in}} \vec{r} \times \dot{m}\vec{V}$$

Steady and uniform flow:

$$\sum \vec{M} = \sum_{\text{out}} \vec{r} \times \dot{m}\vec{V} - \sum_{\text{in}} \vec{r} \times \dot{m}\vec{V}$$

Scalar form for one direction:

$$\sum M = \sum_{\text{out}} r\dot{m}V - \sum_{\text{in}} r\dot{m}V$$

No external moments:

$$0 = \frac{d\vec{H}_{CV}}{dt} + \sum_{\text{out}} \vec{r} \times \dot{m}\vec{V} - \sum_{\text{in}} \vec{r} \times \dot{m}\vec{V}$$

A control volume whose moment of inertia I remains constant can be treated as a solid body (a fixed-mass system), with a net torque of

$$\vec{M}_{CV} = I_{CV}\vec{\alpha} = \sum_{\text{in}} \vec{r} \times \dot{m}\vec{V} - \sum_{\text{out}} \vec{r} \times \dot{m}\vec{V}$$

acting on the body. This relation is used to determine the angular acceleration of a spacecraft when a rocket is fired.

The linear and angular momentum equations are of fundamental importance in the analysis of turbomachinery and are used extensively in Chap. 14.

(ref. 'Fluid Mechanics' by & Cimbala)

95- Dimensional Analysis Background

DIMENSIONAL ANALYSIS AND MODELING

In this chapter, we first review the concepts of *dimensions* and *units*. We then review the fundamental principle of *dimensional homogeneity,* and show how it is applied to equations in order to *nondimensionalize* them and to identify *dimensionless groups*. We discuss the concept of *similarity* between a *model* and a *prototype*. We also describe a powerful tool for engineers and scientists called *dimensional analysis,* in which the combination of dimensional variables, nondimensional variables, and dimensional constants into *nondimensional parameters* reduces the number of necessary independent parameters in a problem. We present a step-by-step method for obtaining these nondimensional parameters, called the *method of repeating variables,* which is based solely on the dimensions of the variables and constants. Finally, we apply this technique to several practical problems to illustrate both its utility and its limitations.

(ref. 'Fluid Mechanics' by & Cimbala)

96- Dimensional Analysis

OBJECTIVES

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

- Develop a better understanding of dimensions, units, and dimensional homogeneity of equations
- Understand the numerous benefits of dimensional analysis
- Know how to use the method of repeating variables to identify nondimensional parameters
- Understand the concept of dynamic similarity and how to apply it to experimental modeling

(ref. ‘Fluid Mechanics’ by & Cimbala)

97- Dimensions and Units

7–1 ▪ DIMENSIONS AND UNITS

A **dimension** is a measure of a physical quantity (without numerical values), while a **unit** is a way to assign a *number* to that dimension. For example, length is a dimension that is measured in units such as microns (μm), feet (ft), centimeters (cm), meters (m), kilometers (km), etc. (Fig. 7–1). There are seven **primary dimensions** (also called **fundamental** or **basic dimensions**)—mass, length, time, temperature, electric current, amount of light, and amount of matter.

> All nonprimary dimensions can be formed by some combination of the seven primary dimensions.

, For example, force has the same dimensions as mass times acceleration (by Newton's second law). Thus, in terms of primary dimensions,

Dimensions of force:

$$\{\text{Force}\} = \left\{\text{Mass}\frac{\text{Length}}{\text{Time}^2}\right\} = \{\text{mL/t}^2\} \qquad \textbf{(7–1)}$$

where the brackets indicate "the dimensions of" and the abbreviations are taken from Table 7–1. You should be aware that some authors prefer force instead of mass as a primary dimension—we do not follow that practice.

TABLE 7–1

Primary dimensions and their associated primary SI and English units

Dimension	Symbol*	SI Unit	English Unit
Mass	m	kg (kilogram)	lbm (pound-mass)
Length	L	m (meter)	ft (foot)
Time†	t	s (second)	s (second)
Temperature	T	K (kelvin)	R (rankine)
Electric current	I	A (ampere)	A (ampere)
Amount of light	C	cd (candela)	cd (candela)
Amount of matter	N	mol (mole)	mol (mole)

(ref. 'Fluid Mechanics' by & Cimbala)

98- Dimensional Homogeneity

7–2 ▪ DIMENSIONAL HOMOGENEITY

We've all heard the old saying, You can't add apples and oranges (Fig. 7–3). This is actually a simplified expression of a far more global and fundamental mathematical law for equations, the **law of dimensional homogeneity**, stated as

Every additive term in an equation must have the same dimensions.

Consider, for example, the change in total energy of a simple compressible closed system from one state and/or time (1) to another (2), as illustrated in Fig. 7–4. The change in total energy of the system (ΔE) is given by

Change of total energy of a system:

$$\Delta E = \Delta U + \Delta \mathrm{KE} + \Delta \mathrm{PE} \qquad \textbf{(7–2)}$$

where E has three components: internal energy (U), kinetic energy (KE), and potential energy (PE). These components can be written in terms of the system mass (m); measurable quantities and thermodynamic properties at each of the two states, such as speed (V), elevation (z), and specific internal energy (u); and the gravitational acceleration constant (g),

$$\Delta U = m(u_2 - u_1) \qquad \Delta \mathrm{KE} = \frac{1}{2} m(V_2^2 - V_1^2) \qquad \Delta \mathrm{PE} = mg(z_2 - z_1) \qquad \textbf{(7–3)}$$

It is straightforward to verify that the left side of Eq. 7–2 and all three additive terms on the right side of Eq. 7–2 have the same dimensions—energy. Using the definitions of Eq. 7–3, we write the primary dimensions of each term,

$$\{\Delta E\} = \{\text{Energy}\} = \{\text{Force} \times \text{Length}\} \;\rightarrow\; \{\Delta E\} = \{\mathrm{mL^2/t^2}\}$$

$$\{\Delta U\} = \left\{\text{Mass}\,\frac{\text{Energy}}{\text{Mass}}\right\} = \{\text{Energy}\} \;\rightarrow\; \{\Delta U\} = \{\mathrm{mL^2/t^2}\}$$

$$\{\Delta \mathrm{KE}\} = \left\{\text{Mass}\,\frac{\text{Length}^2}{\text{Time}^2}\right\} \;\rightarrow\; \{\Delta \mathrm{KE}\} = \{\mathrm{mL^2/t^2}\}$$

$$\{\Delta \mathrm{PE}\} = \left\{\text{Mass}\,\frac{\text{Length}}{\text{Time}^2}\,\text{Length}\right\} \;\rightarrow\; \{\Delta \mathrm{PE}\} = \{\mathrm{mL^2/t^2}\}$$

(ref. 'Fluid Mechanics' by & Cimbala)

99- Dimensional Homogeneity (Repetition)

EXAMPLE 7–2 Dimensional Homogeneity of the Bernoulli Equation

Probably the most well-known (and most misused) equation in fluid mechanics is the Bernoulli equation (Fig. 7–6), discussed in Chap. 5. One standard form of the Bernoulli equation for incompressible irrotational fluid flow is

Bernoulli equation:

$$P + \frac{1}{2}\rho V^2 + \rho g z = C \qquad \textbf{(1)}$$

(*a*) Verify that each additive term in the Bernoulli equation has the same dimensions. (*b*) What are the dimensions of the constant C?

SOLUTION We are to verify that the primary dimensions of each additive term in Eq. 1 are the same, and we are to determine the dimensions of constant C.

Analysis (*a*) Each term is written in terms of primary dimensions,

$$\{P\} = \{\text{Pressure}\} = \left\{\frac{\text{Force}}{\text{Area}}\right\} = \left\{\text{Mass}\,\frac{\text{Length}}{\text{Time}^2}\,\frac{1}{\text{Length}^2}\right\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

$$\left\{\frac{1}{2}\rho V^2\right\} = \left\{\frac{\text{Mass}}{\text{Volume}}\left(\frac{\text{Length}}{\text{Time}}\right)^2\right\} = \left\{\frac{\text{Mass}\times\text{Length}^2}{\text{Length}^3\times\text{Time}^2}\right\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

$$\{\rho g z\} = \left\{\frac{\text{Mass}}{\text{Volume}}\,\frac{\text{Length}}{\text{Time}^2}\,\text{Length}\right\} = \left\{\frac{\text{Mass}\times\text{Length}^2}{\text{Length}^3\times\text{Time}^2}\right\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

Indeed, **all three additive terms have the same dimensions**.

(*b*) From the law of dimensional homogeneity, the constant must have the same dimensions as the other additive terms in the equation. Thus,

Primary dimensions of the Bernoulli constant:

$$\{C\} = \left\{\frac{\textbf{m}}{\textbf{t}^2\textbf{L}}\right\}$$

Discussion If the dimensions of any of the terms were different from the others, it would indicate that an error was made somewhere in the analysis.

(ref. ‘Fluid Mechanics’ by & Cimbala)

100- Nondimensionalisation of Equations:1

Nondimensionalization of Equations

The law of dimensional homogeneity guarantees that every additive term in an equation has the same dimensions. It follows that if we divide each term in the equation by a collection of variables and constants whose product has those same dimensions, the equation is rendered **nondimensional** (Fig. 7–7). If, in addition, the nondimensional terms in the equation are of order unity, the equation is called **normalized**. Normalization is thus more restrictive than nondimensionalization, even though the two terms are sometimes (incorrectly) used interchangeably.

Each term in a nondimensional equation is dimensionless.

In the process of nondimensionalizing an equation of motion, **nondimensional parameters** often appear—most of which are named after a notable scientist or engineer (e.g., the Reynolds number and the Froude number). This process is referred to by some authors as **inspectional analysis**.

(ref. 'Fluid Mechanics' by & Cimbala)

101- Nondimensionalisation of Equations:2

As a simple example, consider the equation of motion describing the elevation z of an object falling by gravity through a vacuum (no air drag), as in Fig. 7–8. The initial location of the object is z_0 and its initial velocity is w_0 in the z-direction. From high school physics,

Equation of motion:
$$\frac{d^2z}{dt^2} = -g \tag{7–4}$$

Dimensional variables are defined as dimensional quantities that change or vary in the problem. For the simple differential equation given in Eq. 7–4, there are two dimensional variables: z (dimension of length) and t (dimension of time). **Nondimensional** (or **dimensionless**) **variables** are defined as quantities that change or vary in the problem, but have no dimensions; an example is angle of rotation, measured in degrees or radians which are dimensionless units. Gravitational constant g, while dimensional, remains constant and is called a **dimensional constant**. Two additional dimensional constants are relevant to this particular problem, initial location z_0 and initial vertical speed w_0. While dimensional constants may change from problem to problem, they are fixed for a particular problem and are thus distinguished from dimensional variables. We use the term **parameters** for the combined set of dimensional variables, nondimensional variables, and dimensional constants in the problem.

Equation 7–4 is easily solved by integrating twice and applying the initial conditions. The result is an expression for elevation z at any time t:

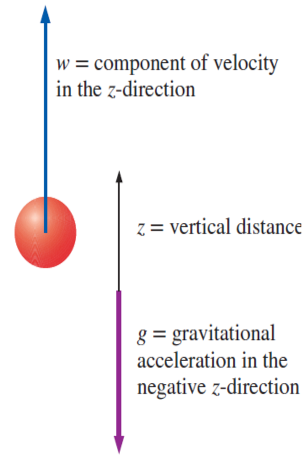


FIGURE 7–8
Object falling in a vacuum. Vertical velocity is drawn positively, so $w < 0$ for a falling object.

(ref. 'Fluid Mechanics' by & Cimbala)

102- Dimensional Variables

Dimensional result:
$$z = z_0 + w_0 t - \frac{1}{2} g t^2 \quad \textbf{(7–5)}$$

The constant $\frac{1}{2}$ and the exponent 2 in Eq. 7–5 are dimensionless results of the integration. Such constants are called **pure constants**. Other common examples of pure constants are π and e.

(ref. ‘Fluid Mechanics’ by & Cimbala)

103-

Nondimensional or Dimensional Variables

To nondimensionalize Eq. 7–4, we need to select **scaling parameters**, based on the primary dimensions contained in the original equation. In fluid flow problems there are typically at least *three* scaling parameters, e.g., L, V, and $P_0 - P_\infty$ (Fig. 7–9), since there are at least three primary dimensions in the general problem (e.g., mass, length, and time). In the case of the falling object being discussed here, there are only two primary dimensions, length and time, and thus we are limited to selecting only *two* scaling parameters. We have some options in the selection of the scaling parameters since we have three available dimensional constants g, z_0, and w_0. We choose z_0 and w_0. You are invited to repeat the analysis with g and z_0 and/or with g and w_0. With these two chosen scaling parameters we nondimensionalize the dimensional variables z and t. The first step is to list the primary dimensions of *all* dimensional variables and dimensional constants in the problem,

Primary dimensions of all parameters:

$$\{z\} = \{\text{L}\} \qquad \{t\} = \{\text{t}\} \qquad \{z_0\} = \{\text{L}\} \qquad \{w_0\} = \{\text{L/t}\} \qquad \{g\} = \{\text{L/t}^2\}$$

The second step is to use our two scaling parameters to nondimensionalize z and t (by inspection) into nondimensional variables z^* and t^*,

Nondimensionalized variables:

$$z^* = \frac{z}{z_0} \qquad t^* = \frac{w_0 t}{z_0} \tag{7–6}$$

Substitution of Eq. 7–6 into Eq. 7–4 gives

$$\frac{d^2z}{dt^2} = \frac{d^2(z_0 z^*)}{d(z_0 t^*/w_0)^2} = \frac{w_0^2}{z_0}\frac{d^2z^*}{dt^{*2}} = -g \quad \rightarrow \quad \frac{w_0^2}{gz_0}\frac{d^2z^*}{dt^{*2}} = -1 \tag{7–7}$$

which is the desired nondimensional equation. The grouping of dimensional constants in Eq. 7–7 is the square of a well-known **nondimensional parameter** or **dimensionless group** called the **Froude number**,

Froude number:

$$\text{Fr} = \frac{w_0}{\sqrt{gz_0}} \tag{7–8}$$

The Froude (pronounced "Frude") number also appears as a nondimensional parameter in free-surface flows (Chap. 13), and can be thought of as the ratio of inertial force to gravitational force (Fig. 7–10). You should note that in some older textbooks, Fr is defined as the *square* of the parameter shown in Eq. 7–8. Substitution of Eq. 7–8 into Eq. 7–7 yields

Nondimensionalized equation of motion:

$$\frac{d^2z^*}{dt^{*2}} = -\frac{1}{\text{Fr}^2} \tag{7–9}$$

In dimensionless form, only one parameter remains, namely the Froude number. Equation 7–9 is easily solved by integrating twice and applying the initial conditions. The result is an expression for dimensionless elevation z^* as a function of dimensionless time t^*:

Nondimensional result:

$$z^* = 1 + t^* - \frac{1}{2\text{Fr}^2}t^{*2} \tag{7–10}$$

Comparison of Eqs. 7–5 and 7–10 reveals that they are equivalent. In fact, for practice, substitute Eqs. 7–6 and 7–8 into Eq. 7–5 to verify Eq. 7–10.

(ref. 'Fluid Mechanics' by & Cimbala)

104- Nondimensionalisation of Equations:3

EXAMPLE 7–3 Illustration of the Advantages of Nondimensionalization

Your little brother's high school physics class conducts experiments in a large vertical pipe whose inside is kept under vacuum conditions. The students are able to remotely release a steel ball at initial height z_0 between 0 and 15 m (measured from the bottom of the pipe), and with initial vertical speed w_0 between 0 and 10 m/s. A computer coupled to a network of photosensors along the pipe enables students to plot the trajectory of the steel ball (height z plotted as a function of time t) for each test. The students are unfamiliar with dimensional analysis or nondimensionalization techniques, and therefore conduct several "brute force" experiments to determine how the trajectory is affected by initial conditions z_0 and w_0. First they hold w_0 fixed at 4 m/s and conduct experiments at five different values of z_0: 3, 6, 9, 12, and 15 m. The experimental results are shown in Fig. 7–12*a*. Next, they hold z_0 fixed at 10 m and conduct experiments at five different values of w_0: 2, 4, 6, 8, and 10 m/s. These results are shown in Fig. 7–12*b*. Later that evening, your brother shows you the data and the trajectory plots and tells you that they plan to conduct more experiments at different values of z_0 and w_0. You explain to him that by first nondimensionalizing the data, the problem can be reduced to just *one* parameter, and no further experiments are required. Prepare a nondimensional plot to prove your point and discuss.

SOLUTION A nondimensional plot is to be generated from all the available trajectory data. Specifically, we are to plot z^* as a function of t^*.

Assumptions The inside of the pipe is subjected to strong enough vacuum pressure that aerodynamic drag on the ball is negligible.

Properties The gravitational constant is 9.81 m/s^2.

Analysis Equation 7–4 is valid for this problem, as is the nondimensionalization that resulted in Eq. 7–9. As previously discussed, this problem combines three of the original dimensional parameters (g, z_0, and w_0) into *one* nondimensional parameter, the Froude number. After converting to the dimensionless variables of Eq. 7–6, the 10 trajectories of Fig. 7–12*a* and *b* are replotted in dimensionless format in Fig. 7–13. It is clear that all the trajectories are of the same family, with the Froude number as the only remaining parameter. Fr^2 varies from about 0.041 to about 1.0 in these experiments. If any more experiments are to be conducted, they should include combinations of z_0 and w_0 that produce Froude numbers outside of this range. A large number of additional experiments would be unnecessary, since all the trajectories would be of the same family as those plotted in Fig. 7–13.

Discussion At low Froude numbers, gravitational forces are much larger than inertial forces, and the ball falls to the floor in a relatively short time. At large values of Fr on the other hand, inertial forces dominate initially, and the ball rises a significant distance before falling; it takes much longer for the ball to hit the ground. The students are obviously not able to adjust the gravitational constant, but if they could, the brute force method would require many more experiments to document the effect of g. If they nondimensionalize first, however, the dimensionless trajectory plots already obtained and shown in Fig. 7–13 would be valid for *any* value of g; no further experiments would be required unless Fr were outside the range of tested values.

(ref. ‘Fluid Mechanics’ by & Cimbala)

105-

Examples of Nondimensionalisation Equations

EXAMPLE 7–4 **Extrapolation of Nondimensionalized Data**

The gravitational constant at the surface of the moon is only about one-sixth of that on earth. An astronaut on the moon throws a baseball at an initial speed of 21.0 m/s at a 5° angle above the horizon and at 2.0 m above the moon's surface (Fig. 7–14). (*a*) Using the dimensionless data of Example 7–3 shown in Fig. 7–13, predict how long it takes for the baseball to fall to the ground. (*b*) Do an *exact* calculation and compare the result to that of part (*a*).

SOLUTION Experimental data obtained on earth are to be used to predict the time required for a baseball to fall to the ground on the moon.

Assumptions **1** The horizontal velocity of the baseball is irrelevant. **2** The surface of the moon is perfectly flat near the astronaut. **3** There is no aerodynamic drag on the ball since there is no atmosphere on the moon. **4** Moon gravity is one-sixth that of earth.

Properties The gravitational constant on the moon is $g_{\text{moon}} \cong 9.81/6 = 1.63\ \text{m/s}^2$.

Analysis (*a*) The Froude number is calculated based on the value of g_{moon} and the vertical component of initial speed,

$$w_0 = (21.0\ \text{m/s})\sin(5°) = 1.830\ \text{m/s}$$

from which

$$\text{Fr}^2 = \frac{w_0^2}{g_{\text{moon}} z_0} = \frac{(1.830\ \text{m/s})^2}{(1.63\ \text{m/s}^2)(2.0\ \text{m})} = 1.03$$

This value of Fr^2 is nearly the same as the largest value plotted in Fig. 7–13. Thus, in terms of dimensionless variables, the baseball strikes the ground at $t^* \cong 2.75$, as determined from Fig. 7–13. Converting back to dimensional variables using Eq. 7–6,

Estimated time to strike the ground: $$t = \frac{t^* z_0}{w_0} = \frac{2.75(2.0\ \text{m})}{1.830\ \text{m/s}} = \mathbf{3.01\ s}$$

(*b*) An exact calculation is obtained by setting z equal to zero in Eq. 7–5 and solving for time t (using the quadratic formula), *Exact time to strike the ground*:

$$t = \frac{w_0 + \sqrt{w_0^2 + 2z_0 g}}{g}$$

$$= \frac{1.830 \text{ m/s} + \sqrt{(1.830 \text{ m/s})^2 + 2(2.0 \text{ m})(1.63 \text{ m/s}^2)}}{1.63 \text{ m/s}^2} = \mathbf{3.05\ s}$$

Discussion If the Froude number had landed between two of the trajectories of Fig. 7–13, interpolation would have been required. Since some of the numbers are precise to only two significant digits, the small difference between the results of part (*a*) and part (*b*) is of no concern. The final result is $t = 3.0$ s to two significant digits.

(ref. 'Fluid Mechanics' by & Cimbala)

106- Nondimensionalisation of Equations:4

Some Dimensionless Parameters

Euler number	$\text{Eu} = \frac{\Delta P}{\rho V^2} \left(\text{sometimes } \frac{\Delta P}{\frac{1}{2}\rho V^2}\right)$	$\frac{\text{Pressure difference}}{\text{Dynamic pressure}}$
Froude number	$\text{Fr} = \frac{V}{\sqrt{gL}} \left(\text{sometimes } \frac{V^2}{gL}\right)$	$\frac{\text{Inertial force}}{\text{Gravitational force}}$
Reynolds number	$\text{Re} = \frac{\rho V L}{\mu} = \frac{VL}{\nu}$	$\frac{\text{Inertial force}}{\text{Viscous force}}$
Strouhal number	$\text{St (sometimes S or Sr)} = \frac{fL}{V}$	$\frac{\text{Characteristic flow time}}{\text{Period of oscillation}}$

(ref. ‘Fluid Mechanics’ by & Cimbala)

107- Dimensional Analysis and Similarity:1

7–3 ▪ DIMENSIONAL ANALYSIS AND SIMILARITY

Nondimensionalization of an equation by inspection is useful only when we know the equation to begin with. However, in many cases in real-life engineering, the equations are either not known or too difficult to solve; oftentimes *experimentation* is the only method of obtaining reliable information. In most experiments, to save time and money, tests are performed on a geometrically scaled **model**, rather than on the full-scale **prototype**. In such cases, care must be taken to properly scale the results. We introduce here a powerful technique called **dimensional analysis**. While typically taught in fluid mechanics, dimensional analysis is useful in *all* disciplines, especially when it is necessary to design and conduct experiments. You are encouraged to use this powerful tool in other subjects as well, not just in fluid mechanics. The three primary purposes of dimensional analysis are

- To generate nondimensional parameters that help in the design of experiments (physical and/or numerical) and in the reporting of experimental results
- To obtain scaling laws so that prototype performance can be predicted from model performance
- To (sometimes) predict trends in the relationship between parameters

(ref. ‘Fluid Mechanics’ by & Cimbala)

108- Dimensional Analysis and Similarity:2

Before discussing the *technique* of dimensional analysis, we first explain the underlying *concept* of dimensional analysis—the principle of **similarity**. There are three necessary conditions for complete similarity between a model and a prototype. The first condition is **geometric similarity**—the model must be the same shape as the prototype, but may be scaled by some constant scale factor. The second condition is **kinematic similarity**, which means that the velocity at any point in the model flow must be proportional

(by a constant scale factor) to the velocity at the corresponding point in the prototype flow (Fig. 7–16). Specifically, for kinematic similarity the velocity at corresponding points must scale in magnitude and must point in the same relative direction. You may think of geometric similarity as *length-scale* equivalence and kinematic similarity as *time-scale* equivalence. *Geometric similarity is a prerequisite for kinematic similarity*. Just as the geometric scale factor can be less than, equal to, or greater than one, so can the velocity scale factor. In Fig. 7–16, for example, the geometric scale factor is less than one (model smaller than prototype), but the velocity scale is greater than one (velocities around the model are greater than those around the prototype). You may recall from Chap. 4 that streamlines are kinematic phenomena; hence, the streamline pattern in the model flow is a geometrically scaled copy of that in the prototype flow when kinematic similarity is achieved.

The third and most restrictive similarity condition is that of **dynamic similarity**. Dynamic similarity is achieved when all *forces* in the model flow scale by a constant factor to corresponding forces in the prototype flow (*force-scale* equivalence). As with geometric and kinematic similarity, the scale factor for forces can be less than, equal to, or greater than one. In Fig. 7–16 for example, the force-scale factor is less than one since the force on the model building is less than that on the prototype. *Kinematic similarity is a necessary but insufficient condition for dynamic similarity*. It is thus possible for a model flow and a prototype flow to achieve both geometric and kinematic similarity, yet not dynamic similarity. All three similarity conditions must exist for complete similarity to be ensured.

In a general flow field, complete similarity between a model and prototype is achieved only when there is geometric, kinematic, and dynamic similarity.

We let uppercase Greek letter Pi (Π) denote a nondimensional parameter. In Section 7–2, we have already discussed one Π, namely the Froude number, Fr. In a general dimensional analysis problem, there is one Π that we call the **dependent** Π, giving it the notation Π_1. The parameter Π_1 is in general a function of several other Π's, which we call **independent** Π's. The functional relationship is

Functional relationship between Π's: $$\Pi_1 = f(\Pi_2, \Pi_3, \ldots, \Pi_k) \qquad \textbf{(7–11)}$$

where k is the total number of Π's.

Consider an experiment in which a scale model is tested to simulate a prototype flow. To ensure complete similarity between the model and the prototype, each independent Π of the model (subscript m) must be identical to the corresponding independent Π of the prototype (subscript p), i.e., $\Pi_{2,m} = \Pi_{2,p}$, $\Pi_{3,m} = \Pi_{3,p}$, . . . , $\Pi_{k,m} = \Pi_{k,p}$.

To ensure complete similarity, the model and prototype must be geometrically similar, and all independent Π groups must match between model and prototype.

Under these conditions the *dependent* Π of the model ($\Pi_{1,m}$) is guaranteed to also equal the dependent Π of the prototype ($\Pi_{1,p}$). Mathematically, we write a conditional statement for achieving similarity,

$$\text{If} \quad \Pi_{2,m} = \Pi_{2,p} \quad \text{and} \quad \Pi_{3,m} = \Pi_{3,p} \ldots \quad \text{and} \quad \Pi_{k,m} = \Pi_{k,p},$$
$$\text{then} \quad \Pi_{1,m} = \Pi_{1,p} \qquad \textbf{(7–12)}$$

Consider, for example, the design of a new sports car, the aerodynamics of which is to be tested in a wind tunnel. To save money, it is desirable to test a small, geometrically scaled model of the car rather than a full-scale prototype of the car (Fig. 7–17). In the case of aerodynamic drag on an automobile, it turns out that if the flow is approximated as incompressible, there are only two Π's in the problem,

$$\Pi_1 = f(\Pi_2) \quad \text{where} \quad \Pi_1 = \frac{F_D}{\rho V^2 L^2} \quad \text{and} \quad \Pi_2 = \frac{\rho V L}{\mu} \qquad \textbf{(7–13)}$$

The procedure used to generate these Π's is discussed in Section 7–4. In Eq. 7–13, F_D is the magnitude of the aerodynamic drag on the car, ρ is the air density, V is the car's speed (or the speed of the air in the wind tunnel), L is the length of the car, and μ is the viscosity of the air. Π_1 is a nonstandard form of the drag coefficient, and Π_2 is the **Reynolds number**, Re. You will find that many problems in fluid mechanics involve a Reynolds number (Fig. 7–18).

The Reynolds number is the most well known and useful dimensionless parameter in all of fluid mechanics.

In the problem at hand there is only one independent Π, and Eq. 7–12 ensures that if the independent Π's match (the Reynolds numbers match: $\Pi_{2,m} = \Pi_{2,p}$), then the dependent Π's also match ($\Pi_{1,m} = \Pi_{1,p}$). This enables engineers to measure the aerodynamic drag on the model car and then use this value to predict the aerodynamic drag on the prototype car.

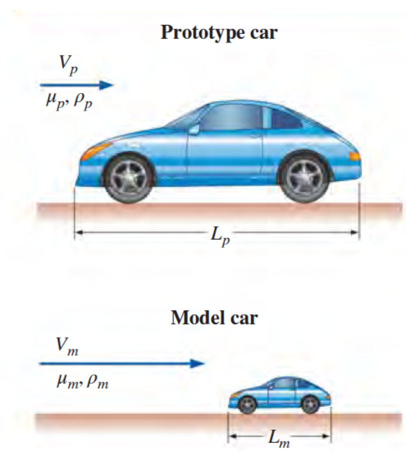


FIGURE 7–17
Geometric similarity between a prototype car of length L_p and a model car of length L_m.

(ref. 'Fluid Mechanics' by & Cimbala)

109- Similarity between Model and Pro-Type Car

EXAMPLE 7–5 Similarity between Model and Prototype Cars

The aerodynamic drag of a new sports car is to be predicted at a speed of 50.0 mi/h at an air temperature of 25°C. Automotive engineers build a one-fifth scale model of the car to test in a wind tunnel. It is winter and the wind tunnel is located in an unheated building; the temperature of the wind tunnel air is only about 5°C. Determine how fast the engineers should run the wind tunnel in order to achieve similarity between the model and the prototype.

SOLUTION We are to utilize the concept of similarity to determine the speed of the wind tunnel.

Assumptions **1** Compressibility of the air is negligible (the validity of this approximation is discussed later). **2** The wind tunnel walls are far enough away so as to not interfere with the aerodynamic drag on the model car. **3** The model is geometrically similar to the prototype. **4** The wind tunnel has a moving belt to simulate the ground under the car, as in Fig. 7–19. (The moving belt is necessary in order to achieve kinematic similarity everywhere in the flow, in particular underneath the car.)

Properties For air at atmospheric pressure and at $T = 25°C$, $\rho = 1.184\ \text{kg/m}^3$ and $\mu = 1.849 \times 10^{-5}$ kg/m·s. Similarly, at $T = 5°C$, $\rho = 1.269\ \text{kg/m}^3$ and $\mu = 1.754 \times 10^{-5}$ kg/m·s.

Analysis Since there is only one independent Π in this problem, the similarity equation (Eq. 7–12) holds if $\Pi_{2,m} = \Pi_{2,p}$, where Π_2 is given by Eq. 7–13, and we call it the Reynolds number. Thus, we write

$$\Pi_{2,m} = \text{Re}_m = \frac{\rho_m V_m L_m}{\mu_m} = \Pi_{2,p} = \text{Re}_p = \frac{\rho_p V_p L_p}{\mu_p}$$

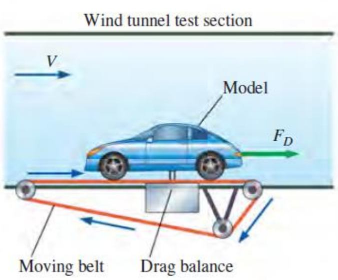


FIGURE 7–19
A *drag balance* is a device used in a wind tunnel to measure the aerodynamic drag of a body. When testing automobile models, a *moving belt* is often added to the floor of the wind tunnel to simulate the moving ground (from the car's frame of reference).

which we solve for the unknown wind tunnel speed for the model tests, V_m,

$$V_m = V_p\left(\frac{\mu_m}{\mu_p}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{L_p}{L_m}\right)$$

$$= (50.0\ \text{mi/h})\left(\frac{1.754 \times 10^{-5}\ \text{kg/m·s}}{1.849 \times 10^{-5}\ \text{kg/m·s}}\right)\left(\frac{1.184\ \text{kg/m}^3}{1.269\ \text{kg/m}^3}\right)(5) = \mathbf{221\ mi/h}$$

Thus, to ensure similarity, the wind tunnel should be run at 221 mi/h (to three significant digits). Note that we were never given the actual length of either car, but the ratio of L_p to L_m is known because the prototype is five times larger than the scale model. When the dimensional parameters are rearranged as nondimensional ratios (as done here), the unit system is irrelevant. Since the units in each numerator cancel those in each denominator, no unit conversions are necessary.

Discussion This speed is quite high (about 100 m/s), and the wind tunnel may not be able to run at that speed. Furthermore, the incompressible approximation may come into question at this high speed (we discuss this in more detail in Example 7–8).

(ref. 'Fluid Mechanics' by & Cimbala)

110- Nondimensionalization of Equations

EXAMPLE 7–6 **Prediction of Aerodynamic Drag Force on a Prototype Car**

This example is a follow-up to Example 7–5. Suppose the engineers run the wind tunnel at 221 mi/h to achieve similarity between the model and the prototype. The aerodynamic drag force on the model car is measured with a **drag balance** (Fig. 7–19). Several drag readings are recorded, and the average drag force on the model is 21.2 lbf. Predict the aerodynamic drag force on the prototype (at 50 mi/h and 25°C).

SOLUTION Because of similarity, the model results are to be scaled up to predict the aerodynamic drag force on the prototype.

Analysis The similarity equation (Eq. 7–12) shows that since $\Pi_{2,m} = \Pi_{2,p}$, $\Pi_{1,m} = \Pi_{1,p}$, where Π_1 is given for this problem by Eq. 7–13. Thus, we write

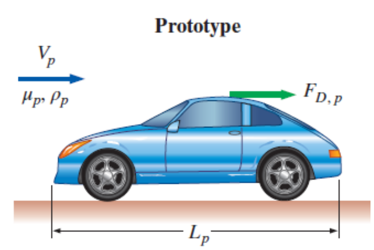
$$\Pi_{1,m} = \frac{F_{D,m}}{\rho_m V_m^2 L_m^2} = \Pi_{1,p} = \frac{F_{D,p}}{\rho_p V_p^2 L_p^2}$$

which we solve for the unknown aerodynamic drag force on the prototype car, $F_{D,p}$,

$$F_{D,p} = F_{D,m}\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{V_m}{V_p}\right)^2\left(\frac{L_m}{L_p}\right)^2$$

$$= (21.2\text{ lbf})\left(\frac{1.184\text{ kg/m}^3}{1.269\text{ kg/m}^3}\right)\left(\frac{50.0\text{ mi/h}}{221\text{ mi/h}}\right)^2(5)^2 = \mathbf{25.3\ lbf}$$

Discussion By arranging the dimensional parameters as nondimensional ratios, the units cancel nicely even though they are a mixture of SI and English units. Because both velocity and length are squared in the equation for Π_1, the higher speed in the wind tunnel nearly compensates for the model's smaller size, and the drag force on the model is nearly the same as that on the prototype. In fact, if the density and viscosity of the air in the wind tunnel were *identical* to those of the air flowing over the prototype, the two drag forces would be identical as well (Fig. 7–20).



(ref. 'Fluid Mechanics' by & Cimbala)

111- Buckingham Pi Theorem

7–4 ▪ THE METHOD OF REPEATING VARIABLES AND THE BUCKINGHAM PI THEOREM

We have seen several examples of the usefulness and power of dimensional analysis. Now we are ready to learn how to *generate* the nondimensional parameters, i.e., the Π's. There are several methods that have been developed for this purpose, but the most popular (and simplest) method is the **method of repeating variables**, popularized by Edgar Buckingham (1867–1940). The method was first published by the Russian scientist Dimitri Riabouchinsky (1882–1962) in 1911. We can think of this method as a step-by-step procedure or "recipe" for obtaining nondimensional parameters. There are six steps, listed concisely in Fig. 7–22, and in more detail in Table 7–2. These steps are explained in further detail as we work through a number of example problems.

As with most new procedures, the best way to learn is by example and practice. As a simple first example, consider a ball falling in a vacuum as discussed in Section 7–2. Let us pretend that we do not know that Eq. 7–4 is appropriate for this problem, nor do we know much physics concerning falling objects. In fact, suppose that all we know is that the instantaneous

The Method of Repeating Variables

Step 1: List the parameters in the problem and count their total number n.

Step 2: List the primary dimensions of each of the n parameters.

Step 3: Set the *reduction* j as the number of primary dimensions. Calculate k, the expected number of Π's,
$k = n - j$

Step 4: Choose j *repeating parameters*.

Step 5: Construct the k Π's, and manipulate as necessary.

Step 6: Write the final functional relationship and check your algebra.

FIGURE 7–22
A concise summary of the six steps that comprise the *method of repeating variables*.

TABLE 7–2
Detailed description of the six steps that comprise the *method of repeating variables**

Step 1	List the parameters (dimensional variables, nondimensional variables, and dimensional constants) and count them. Let n be the total number of parameters in the problem, including the dependent variable. Make sure that any listed independent parameter is indeed independent of the others, i.e., it cannot be expressed in terms of them. (For example, don't include radius r and area $A = \pi r^2$, since r and A are *not* independent.)
Step 2	List the primary dimensions for each of the n parameters.
Step 3	Guess the **reduction** j. As a first guess, set j equal to the number of primary dimensions represented in the problem. The expected number of Π's (k) is equal to n minus j, according to the **Buckingham Pi theorem**, *The Buckingham Pi theorem:* $k = n - j$ **(7–14)** If at this step or during any subsequent step, the analysis does not work out, verify that you have included enough parameters in step 1. Otherwise, go back and *reduce j by one* and try again.
Step 4	Choose j **repeating parameters** that will be used to construct each Π. Since the repeating parameters have the potential to appear in each Π, be sure to choose them *wisely* (Table 7–3).
Step 5	Generate the Π's one at a time by grouping the j repeating parameters with one of the remaining parameters, forcing the product to be dimensionless. In this way, construct all k Π's. By convention the first Π, designated as Π_1, is the *dependent* Π (the one on the left side of the list). Manipulate the Π's as necessary to achieve established dimensionless groups (Table 7–5).
Step 6	Check that all the Π's are indeed dimensionless. Write the final functional relationship in the form of Eq. 7–11.

* This is a step-by-step method for finding the dimensionless Π groups when performing a dimensional analysis.

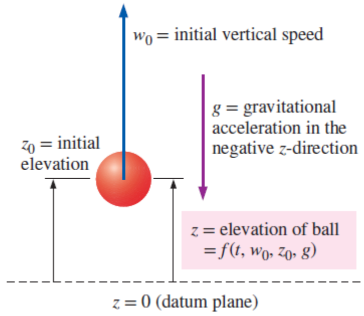


FIGURE 7–23
Setup for dimensional analysis of a ball falling in a vacuum. Elevation z is a function of time t, initial vertical speed w_0, initial elevation z_0, and gravitational constant g.

elevation z of the ball must be a function of time t, initial vertical speed w_0, initial elevation z_0, and gravitational constant g (Fig. 7–23). The beauty of dimensional analysis is that the only other thing we need to know is the primary dimensions of each of these quantities. As we go through each step of the method of repeating variables, we explain some of the subtleties of the technique in more detail using the falling ball as an example.

Step 1

There are five parameters (dimensional variables, nondimensional variables, and dimensional constants) in this problem; $n = 5$. They are listed in functional form, with the dependent variable listed as a function of the independent variables and constants:

List of relevant parameters: $z = f(t, w_0, z_0, g) \quad n = 5$

Step 2

The primary dimensions of each parameter are listed here. We recommend writing each dimension with exponents since this helps with later algebra.

z	t	w_0	z_0	g
$\{L^1\}$	$\{t^1\}$	$\{L^1t^{-1}\}$	$\{L^1\}$	$\{L^1t^{-2}\}$

Step 3

As a first guess, j is set equal to 2, the number of primary dimensions represented in the problem (L and t).

Reduction: $j = 2$

If this value of j is correct, the number of Π's predicted by the Buckingham Pi theorem is

Number of expected Π's: $k = n - j = 5 - 2 = 3$

Step 4

We need to choose two repeating parameters since $j = 2$. Since this is often the hardest (or at least the most mysterious) part of the method of repeating variables, several guidelines about choosing repeating parameters are listed in Table 7–3.

Following the guidelines of Table 7–3 on the next page, the wisest choice of two repeating parameters is w_0 and z_0.

Repeating parameters: w_0 and z_0

Step 5

Now we combine these repeating parameters into products with each of the remaining parameters, one at a time, to create the Π's. The first Π is always the *dependent* Π and is formed with the dependent variable z.

Dependent Π: $$\Pi_1 = z w_0^{a_1} z_0^{b_1} \tag{7–15}$$

where a_1 and b_1 are constant exponents that need to be determined. We apply the primary dimensions of step 2 into Eq. 7–15 and *force* the Π to be dimensionless by setting the exponent of each primary dimension to zero:

Dimensions of Π_1: $$\{\Pi_1\} = \{L^0 t^0\} = \{z w_0^{a_1} z_0^{b_1}\} = \{L^1 (L^1 t^{-1})^{a_1} L^{b_1}\}$$

Since primary dimensions are by definition independent of each other, we equate the exponents of each primary dimension independently to solve for exponents a_1 and b_1 (Fig. 7–24).

Time: $$\{t^0\} = \{t^{-a_1}\} \qquad 0 = -a_1 \qquad a_1 = 0$$

Length: $$\{L^0\} = \{L^1 L^{a_1} L^{b_1}\} \qquad 0 = 1 + a_1 + b_1 \qquad b_1 = -1 - a_1 \qquad b_1 = -1$$

Equation 7–15 thus becomes

$$\Pi_1 = \frac{z}{z_0} \tag{7–16}$$

In similar fashion we create the first independent Π (Π_2) by combining the repeating parameters with independent variable t.

First independent Π: $$\Pi_2 = t w_0^{a_2} z_0^{b_2}$$

Dimensions of Π_2: $$\{\Pi_2\} = \{L^0 t^0\} = \{t w_0^{a_2} z_0^{b_2}\} = \{t (L^1 t^{-1})^{a_2} L^{b_2}\}$$

Equating exponents,

Time: $\{t^0\} = \{t^1 t^{-a_2}\} \qquad 0 = 1 - a_2 \qquad a_2 = 1$

Length: $\{L^0\} = \{L^{a_2} L^{b_2}\} \qquad 0 = a_2 + b_2 \qquad b_2 = -a_2 \qquad b_2 = -1$

Π_2 is thus

$$\Pi_2 = \frac{w_0 t}{z_0} \tag{7–17}$$

Finally we create the second independent Π (Π_3) by combining the repeating parameters with g and *forcing* the Π to be dimensionless (Fig. 7–26).

Second independent Π: $\Pi_3 = g w_0^{a_3} z_0^{b_3}$

Dimensions of Π_3: $\{\Pi_3\} = \{L^0 t^0\} = \{g w_0^{a_3} z_0^{b_3}\} = \{L^1 t^{-2} (L^1 t^{-1})^{a_3} L^{b_3}\}$

Equating exponents,

Time: $\{t^0\} = \{t^{-2} t^{-a_3}\} \qquad 0 = -2 - a_3 \qquad a_3 = -2$

Length: $\{L^0\} = \{L^1 L^{a_3} L^{b_3}\} \qquad 0 = 1 + a_3 + b_3 \qquad b_3 = -1 - a_3 \qquad b_3 = 1$

Π_3 is thus

$$\Pi_3 = \frac{g z_0}{w_0^2} \tag{7–18}$$

All three Π's have been found, but at this point it is prudent to examine them to see if any manipulation is required. We see immediately that Π_1 and Π_2 are the same as the nondimensionalized variables z^* and t^* defined by Eq. 7–6—no manipulation is necessary for these. However, we recognize that the third Π must be raised to the power of $-\frac{1}{2}$ to be of the same form as an established dimensionless parameter, namely the Froude number of Eq. 7–8:

Modified Π_3:

$$\Pi_{3,\text{ modified}} = \left(\frac{g z_0}{w_0^2}\right)^{-1/2} = \frac{w_0}{\sqrt{g z_0}} = \text{Fr} \tag{7–19}$$

Such manipulation is often necessary to put the Π's into proper established form. The Π of Eq. 7–18 is not *wrong,* and there is certainly no mathematical advantage of Eq. 7–19 over Eq. 7–18. Instead, we like to say that Eq. 7–19 is more "socially acceptable" than Eq. 7–18, since it is a named, established nondimensional parameter that is commonly used in the literature. Table 7–4 lists some guidelines for manipulation of nondimensional Π groups into established nondimensional parameters.

Table 7–5 lists some established nondimensional parameters, most of which are named after a notable scientist or engineer (see Fig. 7–27 and the Historical Spotlight on p. 317). This list is by no means exhaustive. Whenever possible, you should manipulate your Π's as necessary in order to convert them into established nondimensional parameters.

Step 6

We should double-check that the Π's are indeed dimensionless (Fig. 7–28). You can verify this on your own for the present example. We are finally ready to write the functional relationship between the nondimensional parameters. Combining Eqs. 7–16, 7–17, and 7–19 into the form of Eq. 7–11,

Relationship between Π's: $$\Pi_1 = f(\Pi_2, \Pi_3) \quad \rightarrow \quad \frac{z}{z_0} = f\left(\frac{w_0 t}{z_0}, \frac{w_0}{\sqrt{g z_0}}\right)$$

Or, in terms of the nondimensional variables z^* and t^* defined previously by Eq. 7–6 and the definition of the Froude number,

Final result of dimensional analysis: $$z^* = f(t^*, \mathrm{Fr}) \qquad \textbf{(7–20)}$$

It is useful to compare the result of dimensional analysis, Eq. 7–20, to the exact analytical result, Eq. 7–10. The method of repeating variables properly predicts the functional relationship between dimensionless groups. However,

The method of repeating variables cannot predict the exact mathematical form of the equation.

(ref. 'Fluid Mechanics' by & Cimbala)

112- Differential Analysis of Fluid Flow: Introduction

DIFFERENTIAL ANALYSIS OF FLUID FLOW

In this chapter we derive the differential equations of fluid motion, namely, conservation of mass (the *continuity equation*) and Newton's second law (the *Navier–Stokes equation*). These equations apply to every point in the flow field and thus enable us to solve for all details of the flow everywhere in the *flow domain*. Unfortunately, most differential equations encountered in fluid mechanics are very difficult to solve and often require the aid of a computer. Also, these equations must be combined when necessary with additional equations, such as an equation of state and an equation for energy and/or species transport. We provide a step-by-step procedure for solving this set of differential equations of fluid motion and obtain analytical solutions for several simple examples. We also introduce the concept of the *stream function*; curves of constant stream function turn out to be *streamlines* in two-dimensional flow fields.

(ref. 'Fluid Mechanics' by & Cimbala)

113- Differential Analysis of Fluid Flow: Preliminaries 1

9–1 ▪ INTRODUCTION

In Chap. 5, we derived control volume versions of the laws of conservation of mass and energy, and in Chap. 6 we did the same for momentum. The control volume technique is useful when we are interested in the overall features of a flow, such as mass flow rate into and out of the control volume or net forces applied to bodies. An example is sketched in Fig. 9–1*a* for the case of wind flowing around a satellite dish. A rectangular control volume is taken around the vicinity of the satellite dish, as sketched. If we know the air velocity along the entire control surface, we can calculate the net reaction force on the stand without ever knowing any details about the geometry of the satellite dish. The interior of the control volume is in fact treated like a "black box" in control volume analysis—we *cannot* obtain detailed knowledge about flow properties such as velocity or pressure at points *inside* the control volume.

Differential analysis, on the other hand, involves application of differential equations of fluid motion to *any* and *every* point in the flow field over a region called the **flow domain**. You can think of the differential technique as the analysis of millions of tiny control volumes stacked end to end and on top of each other all throughout the flow field. In the limit as the number of tiny control volumes goes to infinity, and the size of each control volume shrinks to a point, the conservation equations simplify to a set of partial differential equations that are valid at any point in the flow. When solved, these differential equations yield details about the velocity, density, pressure, etc., at *every* point throughout the *entire* flow domain. In Fig. 9–1*b*, for example, differential analysis of airflow around the satellite dish yields streamline shapes, a detailed pressure distribution around the dish, etc. From these details, we can integrate to find gross features of the flow such as the net force on the satellite dish.

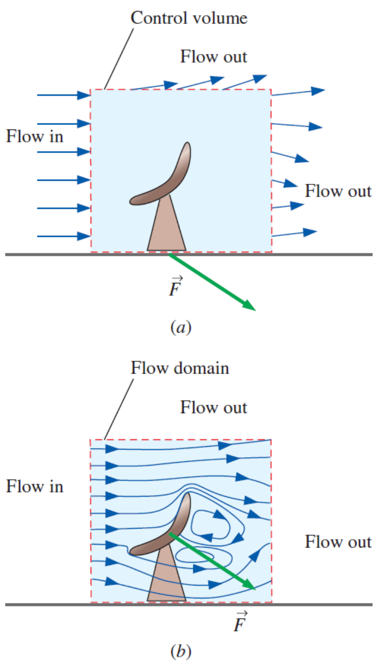


FIGURE 9–1
(*a*) In control volume analysis, the interior of the control volume is treated like a black box, but (*b*) in differential analysis, *all* the details of the flow are solved at *every* point within the flow domain.

In a fluid flow problem such as the one illustrated in Fig. 9–1 in which air density and temperature changes are insignificant, it is sufficient to solve two differential equations of motion—conservation of mass and Newton's second law (the linear momentum equation). For three-dimensional incompressible flow, there are *four unknowns* (velocity components u, v, w, and pressure P) and *four equations* (one from conservation of mass, which is a scalar equation, and three from Newton's second law, which is a vector equation). As we shall see, the equations are **coupled**, meaning that some of the variables appear in all four equations; the set of differential equations must therefore be solved simultaneously for all four unknowns. In addition, **boundary conditions** for the variables must be specified at *all boundaries of the flow domain,* including inlets, outlets, and walls. Finally, if the flow is unsteady, we must march our solution along in time as the flow field changes. You can see how differential analysis of fluid flow can become quite complicated and difficult. Computers are a tremendous help here, as discussed in Chap. 15. Nevertheless, there is much we can do analytically, and we start by deriving the differential equation for conservation of mass.

(ref. 'Fluid Mechanics' by & Cimbala)

114- Differential Analysis of Fluid Flow: Preliminaries 2

9–2 ▪ CONSERVATION OF MASS—THE CONTINUITY EQUATION

Through application of the Reynolds transport theorem (Chap. 4), we have the following general expression for conservation of mass as applied to a control volume:

Conservation of mass for a CV:

$$0 = \int_{CV} \frac{\partial \rho}{\partial t}\, dV + \int_{CS} \rho \vec{V} \cdot \vec{n}\, dA \qquad \textbf{(9–1)}$$

Recall that Eq. 9–1 is valid for both fixed and moving control volumes, provided that the velocity vector is the ***absolute*** velocity (as seen by a fixed observer). When there are well-defined inlets and outlets, Eq. 9–1 is rewritten as

$$\int_{CV} \frac{\partial \rho}{\partial t}\, dV = \sum_{in} \dot{m} - \sum_{out} \dot{m} \qquad \textbf{(9–2)}$$

In words, the net rate of change of mass within the control volume is equal to the rate at which mass flows into the control volume minus the rate at which mass flows out of the control volume. Equation 9–2 applies to *any* control volume, regardless of its size. To generate a differential equation for conservation of mass, we imagine the control volume shrinking to infinitesimal size, with dimensions dx, dy, and dz (Fig. 9–2). In the limit, the entire control volume shrinks to a *point* in the flow.

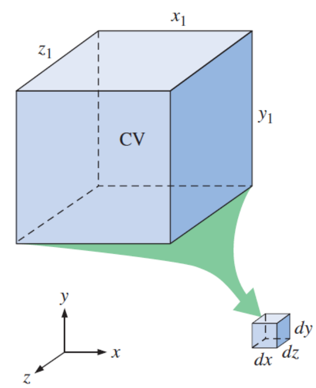


FIGURE 9–2
To derive a differential conservation equation, we imagine shrinking a control volume to infinitesimal size.

(ref. ‘Fluid Mechanics’ by & Cimbala)

115- Differential Analysis of Fluid Flow: Preliminaries 3

Derivation Using the Divergence Theorem

The quickest and most straightforward way to derive the differential form of conservation of mass is to apply the **divergence theorem** to Eq. 9–1. The divergence theorem is also called **Gauss's theorem**, named after the German mathematician Johann Carl Friedrich Gauss (1777–1855). The divergence theorem allows us to transform a volume integral of the divergence of a vector into an area integral over the surface that defines the volume. For any vector $\vec{G}$, the **divergence** of $\vec{G}$ is defined as $\vec{\nabla}\cdot\vec{G}$, and the divergence theorem is written as

Divergence theorem:

$$\int_V \vec{\nabla}\cdot\vec{G}\,dV = \oint_A \vec{G}\cdot\vec{n}\,dA \quad \textbf{(9–3)}$$

The circle on the area integral is used to emphasize that the integral must be evaluated around the *entire closed area A* that surrounds volume V. Note that the control surface of Eq. 9–1 *is* a closed area, even though we do not always add the circle to the integral symbol. Equation 9–3 applies to *any* volume, so we choose the control volume of Eq. 9–1. We also let $\vec{G} = \rho\vec{V}$ since $\vec{G}$ can be any vector. Substitution of Eq. 9–3 into Eq. 9–1 converts the area integral into a volume integral,

$$0 = \int_{CV} \frac{\partial \rho}{\partial t}\,dV + \int_{CV} \vec{\nabla}\cdot(\rho\vec{V})\,dV$$

We now combine the two volume integrals into one,

$$\int_{CV} \left[\frac{\partial \rho}{\partial t} + \vec{\nabla}\cdot(\rho\vec{V})\right] dV = 0 \quad \textbf{(9–4)}$$

Finally, we argue that Eq. 9–4 must hold for *any* control volume regardless of its size or shape. This is possible only if the integrand (the terms within square brackets) is identically zero. Hence, we have a general differential equation for conservation of mass, better known as the **continuity equation**:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{V}) = 0 \qquad \textbf{(9–5)}$$

Equation 9–5 is the compressible form of the continuity equation since we have not assumed incompressible flow. It is valid at any point in the flow domain.

(ref. ‘Fluid Mechanics’ by & Cimbala)

116- Differential Analysis of Fluid Flow: Preliminaries 4

Alternative Form of the Continuity Equation

We expand Eq. 9–5 by using the product rule on the divergence term,

$$\frac{\partial \rho}{\partial t} + \vec{\nabla}\cdot(\rho\vec{V}) = \underbrace{\frac{\partial \rho}{\partial t} + \vec{V}\cdot\vec{\nabla}\rho}_{\text{Material derivative of } \rho} + \rho\vec{\nabla}\cdot\vec{V} = 0 \qquad \textbf{(9–9)}$$

Recognizing the *material derivative* in Eq. 9–9 (see Chap. 4), and dividing by ρ, we write the compressible continuity equation in an alternative form,

Alternative form of the continuity equation:

$$\frac{1}{\rho}\frac{D\rho}{Dt} + \vec{\nabla}\cdot\vec{V} = 0 \qquad \textbf{(9–10)}$$

Equation 9–10 shows that as we follow a fluid element through the flow field (we call this a **material element**), its density changes as $\vec{\nabla}\cdot\vec{V}$ changes (Fig. 9–9).

(ref. 'Fluid Mechanics' by & Cimbala)

117- Differential Analysis of Fluid Flow: Continuity Equation

On the other hand, if changes in the density of the material element are negligibly small compared to the magnitude of the density itself as the element moves around, then both terms in Eq. 9–10 are negligibly small; $\vec{\nabla} \cdot \vec{V} \cong 0$ and $\rho^{-1}\, D\rho/Dt \cong 0$, and the flow is approximated as **incompressible**.

(ref. ‘Fluid Mechanics’ by & Cimbala)

118- Differential Analysis of Fluid Flow: Objectives

Derivation Using an Infinitesimal Control Volume

We derive the continuity equation in a different way, by starting with a control volume on which we apply conservation of mass. Consider an infinitesimal box-shaped control volume aligned with the axes in Cartesian coordinates (Fig. 9–3). The dimensions of the box are dx, dy, and dz, and the center of the box is shown at some arbitrary point P from the origin (the box can be located anywhere in the flow field). At the center of the box we define the density as ρ and the velocity components as u, v, and w, as shown. At locations away from the center of the box, we use a **Taylor series expansion** about the center of the box (point P). [The series expansion is named in honor of its creator, the English mathematician Brook Taylor (1685–1731).] For example, the center of the right-most face of the box is located a distance $dx/2$ from the middle of the box in the x-direction; the value of ρu at that point is

$$(\rho u)_{\text{center of right face}} = \rho u + \frac{\partial(\rho u)}{\partial x}\frac{dx}{2} + \frac{1}{2!}\frac{\partial^2(\rho u)}{\partial x^2}\left(\frac{dx}{2}\right)^2 + \cdots \qquad \textbf{(9–6)}$$

(ref. ‘Fluid Mechanics’ by & Cimbala)

119- Derivation of Continuity Equation: Using the Divergence Theorem

As the box representing the control volume shrinks to a point, however, second-order and higher terms become negligible. For example, suppose $dx/L = 10^{-3}$, where L is some characteristic length scale of the flow domain. Then $(dx/L)^2 = 10^{-6}$, a factor of a thousand less than dx/L. In fact, the smaller dx, the better the assumption that second-order terms are negligible. Applying this truncated Taylor series expansion to the density times the normal velocity component at the center point of each of the six faces of the box, we have

Center of right face: $$(\rho u)_{\text{center of right face}} \cong \rho u + \frac{\partial(\rho u)}{\partial x}\frac{dx}{2}$$

Center of left face: $$(\rho u)_{\text{center of left face}} \cong \rho u - \frac{\partial(\rho u)}{\partial x}\frac{dx}{2}$$

Center of front face: $$(\rho w)_{\text{center of front face}} \cong \rho w + \frac{\partial(\rho w)}{\partial z}\frac{dz}{2}$$

Center of rear face: $$(\rho w)_{\text{center of rear face}} \cong \rho w - \frac{\partial(\rho w)}{\partial z}\frac{dz}{2}$$

Center of top face: $$(\rho v)_{\text{center of top face}} \cong \rho v + \frac{\partial(\rho v)}{\partial y}\frac{dy}{2}$$

Center of bottom face: $$(\rho v)_{\text{center of bottom face}} \cong \rho v - \frac{\partial(\rho v)}{\partial y}\frac{dy}{2}$$

(ref. ‘Fluid Mechanics’ by & Cimbala)

120- Derivation of The Continuity Equation Using an Infinitesimal Control Volume

The mass flow rate into or out of one of the faces is equal to the density times the normal velocity component at the center point of the face times the surface area of the face. In other words, $\dot{m} = \rho V_n A$ at each face, where V_n is the magnitude of the normal velocity through the face and A is the surface area of the face (Fig. 9–4). The mass flow rate through each face of our infinitesimal control volume is illustrated in Fig. 9–5. We could construct truncated Taylor series expansions at the center of each face for the remaining (nonnormal) velocity components as well, but this is unnecessary since these components are *tangential* to the face under consideration. For example, the value of ρv at the center of the right face can be estimated by a similar expansion, but since v is tangential to the right face of the box, it contributes nothing to the mass flow rate into or out of that face.

As the control volume shrinks to a point, the value of the volume integral on the left-hand side of Eq. 9–2 becomes

Rate of change of mass within CV:

$$\int_{CV} \frac{\partial \rho}{\partial t} dV \cong \frac{\partial \rho}{\partial t} dx\, dy\, dz \tag{9–7}$$

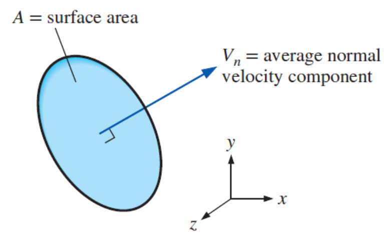


FIGURE 9–4
The mass flow rate through a surface is equal to $\rho V_n A$.

(ref. 'Fluid Mechanics' by & Cimbala)

121- Continuity Equation: Derivation Using an Infinitesimal Control Volume

since the volume of the box is $dx\,dy\,dz$. We now apply the approximations of Fig. 9–5 to the right-hand side of Eq. 9–2. We add up all the mass flow rates into and out of the control volume through the faces. The left, bottom, and back faces contribute to mass *inflow,* and the first term on the right-hand side of Eq. 9–2 becomes

Net mass flow rate into CV:

$$\sum_{\text{in}} \dot{m} \cong \underbrace{\left(\rho u - \frac{\partial(\rho u)}{\partial x}\frac{dx}{2}\right) dy\,dz}_{\text{left face}} + \underbrace{\left(\rho v - \frac{\partial(\rho v)}{\partial y}\frac{dy}{2}\right) dx\,dz}_{\text{bottom face}} + \underbrace{\left(\rho w - \frac{\partial(\rho w)}{\partial z}\frac{dz}{2}\right) dx\,dy}_{\text{rear face}}$$

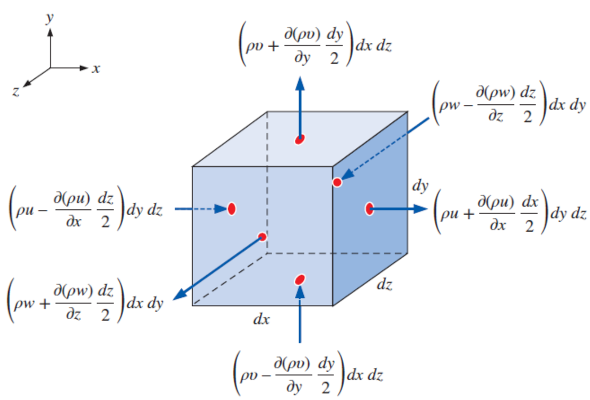


FIGURE 9–5
The inflow or outflow of mass through each face of the differential control volume; the red dots indicate the center of each face.

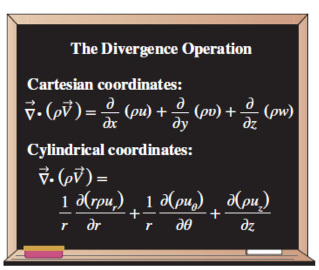


FIGURE 9–6
The divergence operation in Cartesian and cylindrical coordinates.

Similarly, the right, top, and front faces contribute to mass *outflow,* and the second term on the right-hand side of Eq. 9–2 becomes

Net mass flow rate out of CV:

$$\sum_{\text{out}} \dot{m} \cong \underbrace{\left(\rho u + \frac{\partial(\rho u)}{\partial x}\frac{dx}{2}\right) dy\,dz}_{\text{right face}} + \underbrace{\left(\rho v + \frac{\partial(\rho v)}{\partial y}\frac{dy}{2}\right) dx\,dz}_{\text{top face}} + \underbrace{\left(\rho w + \frac{\partial(\rho w)}{\partial z}\frac{dz}{2}\right) dx\,dy}_{\text{front face}}$$

We substitute Eq. 9–7 and these two equations for mass flow rate into Eq. 9–2. Many of the terms cancel each other out; after combining and simplifying the remaining terms, we are left with

$$\frac{\partial \rho}{\partial t} dx\,dy\,dz = -\frac{\partial(\rho u)}{\partial x} dx\,dy\,dz - \frac{\partial(\rho v)}{\partial y} dx\,dy\,dz - \frac{\partial(\rho w)}{\partial z} dx\,dy\,dz$$

The volume of the box, $dx\,dy\,dz$, appears in each term and can be eliminated. After rearrangement we end up with the following differential equation for conservation of mass in Cartesian coordinates:

Continuity equation in Cartesian coordinates:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad \textbf{(9–8)}$$

Equation 9–8 is the compressible form of the continuity equation in Cartesian coordinates. It is written in more compact form by recognizing the divergence operation (Fig. 9–6), yielding the same equation as Eq. 9–5.

(ref. ‘Fluid Mechanics’ by & Cimbala)

122- The Continuity Equation: Alternative Form of the Continuity Equation

Alternative Form of the Continuity Equation

We expand Eq. 9–5 by using the product rule on the divergence term,

$$\frac{\partial \rho}{\partial t} + \vec{\nabla}\cdot(\rho\vec{V}) = \underbrace{\frac{\partial \rho}{\partial t} + \vec{V}\cdot\vec{\nabla}\rho}_{\text{Material derivative of } \rho} + \rho\vec{\nabla}\cdot\vec{V} = 0 \quad \text{(9–9)}$$

Recognizing the *material derivative* in Eq. 9–9 (see Chap. 4), and dividing by ρ, we write the compressible continuity equation in an alternative form,

Alternative form of the continuity equation:

$$\frac{1}{\rho}\frac{D\rho}{Dt} + \vec{\nabla}\cdot\vec{V} = 0 \quad \text{(9–10)}$$

Equation 9–10 shows that as we follow a fluid element through the flow field (we call this a **material element**), its density changes as $\vec{\nabla}\cdot\vec{V}$ changes (Fig. 9–9).

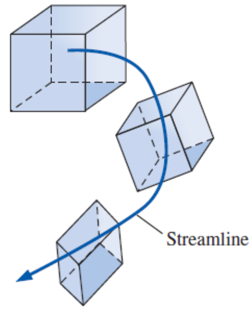


FIGURE 9–9
As a material element moves through a flow field, its density changes according to Eq. 9–10.

On the other hand, if changes in the density of the material element are negligibly small compared to the magnitude of the density itself as the element moves around, then both terms in Eq. 9–10 are negligibly small; $\vec{\nabla}\cdot\vec{V} \cong 0$ and $\rho^{-1}\, D\rho/Dt \cong 0$, and the flow is approximated as **incompressible**.

(ref. ‘Fluid Mechanics’ by & Cimbala)

123- Continuity Equation in Cylindrical Coordinates

Continuity Equation in Cylindrical Coordinates

Many problems in fluid mechanics are more conveniently solved in **cylindrical coordinates** (r, θ, z) (often called **cylindrical polar coordinates**), rather than in Cartesian coordinates. For simplicity, we introduce cylindrical coordinates in two dimensions first (Fig. 9–10*a*). By convention, r is the radial distance from the origin to some point (P), and θ is the angle measured from the x-axis (θ is always defined as mathematically positive in the counterclockwise direction). Velocity components, u_r and u_θ, and unit vectors, $\vec{e}_r$ and $\vec{e}_\theta$, are also shown in Fig. 9–10*a*. In three dimensions, imagine sliding everything in Fig. 9–10*a* out of the page along the z-axis (normal to the xy-plane) by some distance z. We have attempted to draw this in Fig. 9–10*b*. In three dimensions, we have a third velocity component, u_z, and a third unit vector, $\vec{e}_z$, also sketched in Fig. 9–10*b*.

The following coordinate transformations are obtained from Fig. 9–10:

Coordinate transformations:

$$r = \sqrt{x^2 + y^2} \qquad x = r\cos\theta \qquad y = r\sin\theta \qquad \theta = \tan^{-1}\frac{y}{x} \tag{9–11}$$

Coordinate z is the same in cylindrical and Cartesian coordinates.

To obtain an expression for the continuity equation in cylindrical coordinates, we have two choices. First, we can use Eq. 9–5 directly, since it was derived without regard to our choice of coordinate system. We simply look up the expression for the divergence operator in cylindrical coordinates in a vector calculus book (e.g., Spiegel, 1968; see also Fig. 9–6). Second, we can draw a three-dimensional infinitesimal fluid element in cylindrical coordinates and analyze mass flow rates into and out of the element, similar to what we did before in Cartesian coordinates. Either way, we end up with

Continuity equation in cylindrical coordinates:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial(r\rho u_r)}{\partial r} + \frac{1}{r}\frac{\partial(\rho u_\theta)}{\partial \theta} + \frac{\partial(\rho u_z)}{\partial z} = 0 \tag{9–12}$$

(ref. ‘Fluid Mechanics’ by & Cimbala)

124- Special Cases of the Continuity Equation

Special Cases of the Continuity Equation

We now look at two special cases, or simplifications, of the continuity equation. In particular, we first consider steady compressible flow, and then incompressible flow.

Special Case 1: Steady Compressible Flow

If the flow is compressible but steady, $\partial/\partial t$ of any variable is equal to zero. Thus, Eq. 9–5 reduces to

Steady continuity equation:
$$\vec{\nabla}\cdot(\rho\vec{V}) = 0 \tag{9–13}$$

In Cartesian coordinates, Eq. 9–13 reduces to

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{9–14}$$

In cylindrical coordinates, Eq. 9–13 reduces to

$$\frac{1}{r}\frac{\partial(r\rho u_r)}{\partial r} + \frac{1}{r}\frac{\partial(\rho u_\theta)}{\partial \theta} + \frac{\partial(\rho u_z)}{\partial z} = 0 \tag{9–15}$$

Special Case 2: Incompressible Flow

If the flow is approximated as incompressible, density is not a function of time or space. Thus the unsteady term in Eq. 9–5 disappears and ρ can be taken outside of the divergence operator. Equation 9–5 therefore reduces to

Incompressible continuity equation:
$$\vec{\nabla}\cdot\vec{V} = 0 \tag{9–16}$$

The same result is obtained if we start with Eq. 9–10 and recognize that for an incompressible flow, density does not change appreciably following a fluid particle, as pointed out previously. Thus the material derivative of ρ is approximately zero, and Eq. 9–10 reduces immediately to Eq. 9–16.

You may have noticed that *no time derivatives remain in Eq. 9–16*. We conclude from this that *even if the flow is unsteady, Eq. 9–16 applies at any instant in time*. Physically, this means that as the velocity field changes in one part of an incompressible flow field, the entire rest of the flow field immediately adjusts to the change such that Eq. 9–16 is satisfied at all times. For compressible flow this is not the case. In fact, a disturbance in

(ref. 'Fluid Mechanics' by & Cimbala)

125- Examples of Application of the continuity equation

In Cartesian coordinates, Eq. 9–16 is

Incompressible continuity equation in Cartesian coordinates:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad \textbf{(9–17)}$$

Equation 9–17 is the form of the continuity equation you will probably encounter most often. It applies to steady or unsteady, incompressible, three-dimensional flow, and you would do well to memorize it.

In cylindrical coordinates, Eq. 9–16 is

Incompressible continuity equation in cylindrical coordinates:

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{1}{r}\frac{\partial(u_\theta)}{\partial \theta} + \frac{\partial(u_z)}{\partial z} = 0 \qquad \textbf{(9–18)}$$

(ref. ‘Fluid Mechanics’ by & Cimbala)

126- Examples of Application of the continuity equation: Design of a converging duct

EXAMPLE 9–2 Design of a Compressible Converging Duct

A two-dimensional converging duct is being designed for a high-speed wind tunnel. The bottom wall of the duct is to be flat and horizontal, and the top wall is to be curved in such a way that the axial wind speed u increases approximately linearly from $u_1 = 100$ m/s at section (1) to $u_2 = 300$ m/s at section

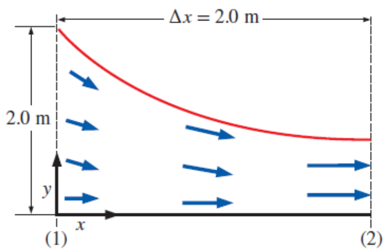


FIGURE 9–12
Converging duct, designed for a high-speed wind tunnel (not to scale).

(2) (Fig. 9–12). Meanwhile, the air density ρ is to decrease approximately linearly from $\rho_1 = 1.2$ kg/m³ at section (1) to $\rho_2 = 0.85$ kg/m³ at section (2). The converging duct is 2.0 m long and is 2.0 m high at section (1). (*a*) Predict the *y-component* of velocity, $v(x, y)$, in the duct. (*b*) Plot the approximate shape of the duct, ignoring friction on the walls. (*c*) How high should the duct be at section (2), the exit of the duct?

SOLUTION For given velocity component u and density ρ, we are to predict velocity component v, plot an approximate shape of the duct, and predict its height at the duct exit.

Assumptions **1** The flow is steady and two-dimensional in the xy-plane. **2** Friction on the walls is ignored. **3** Axial velocity u increases linearly with x, and density ρ decreases linearly with x.

Properties The fluid is air at room temperature (25°C). The speed of sound is about 346 m/s, so the flow is subsonic, but compressible.

Analysis (*a*) We write expressions for u and ρ, forcing them to be linear in x,

$$u = u_1 + C_u x \quad \text{where} \quad C_u = \frac{u_2 - u_1}{\Delta x} = \frac{(300 - 100)\text{ m/s}}{2.0\text{ m}} = 100\text{ s}^{-1} \tag{1}$$

and

$$\rho = \rho_1 + C_\rho x \quad \text{where} \quad C_\rho = \frac{\rho_2 - \rho_1}{\Delta x} = \frac{(0.85 - 1.2)\text{ kg/m}^3}{2.0\text{ m}} \tag{2}$$

$$= -0.175\text{ kg/m}^4$$

The steady continuity equation (Eq. 9–14) for this two-dimensional compressible flow simplifies to

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \underbrace{\frac{\partial(\rho w)}{\partial z}}_{0} = 0 \quad \rightarrow \quad \frac{\partial(\rho v)}{\partial y} = -\frac{\partial(\rho u)}{\partial x} \tag{3}$$

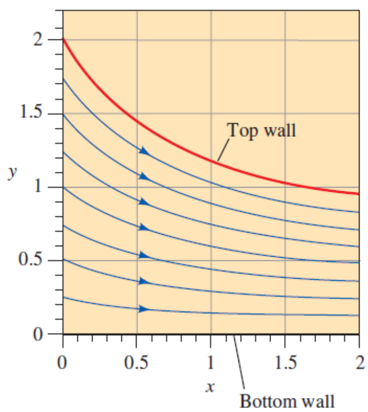


FIGURE 9–13
Streamlines for the converging duct of Example 9–2.

Substituting Eqs. 1 and 2 into Eq. 3 and noting that C_u and C_ρ are constants,

$$\frac{\partial(\rho v)}{\partial y} = -\frac{\partial[(\rho_1 + C_\rho x)(u_1 + C_u x)]}{\partial x} = -(\rho_1 C_u + u_1 C_\rho) - 2C_u C_\rho x$$

Integration with respect to y gives

$$\rho v = -(\rho_1 C_u + u_1 C_\rho)y - 2C_u C_\rho xy + f(x) \quad \textbf{(4)}$$

Note that since the integration is a *partial* integration, we have added an arbitrary function of x instead of simply a constant of integration. Next, we apply boundary conditions. We argue that since the bottom wall is flat and horizontal, v must equal zero at $y = 0$ for any x. This is possible only if $f(x) = 0$. Solving Eq. 4 for v gives

$$v = \frac{-(\rho_1 C_u + u_1 C_\rho)y - 2C_u C_\rho xy}{\rho} \quad \rightarrow \quad v = \frac{-(\rho_1 C_u + u_1 C_\rho)y - 2C_u C_\rho xy}{\rho_1 + C_\rho x} \quad \textbf{(5)}$$

(*b*) Using Eqs. 1 and 5 and the technique described in Chap. 4, we plot several streamlines between $x = 0$ and $x = 2.0$ m in Fig. 9–13. The streamline starting at $x = 0$, $y = 2.0$ m approximates the top wall of the duct.

(*c*) At section (2), the top streamline crosses $y = 0.941$ m at $x = 2.0$ m. Thus, the predicted height of the duct at section (2) is **0.941 m**.

Discussion You can verify that the combination of Eqs. 1, 2, and 5 satisfies the continuity equation. However, this alone does not guarantee that the density and velocity components will actually *follow* these equations if the duct were to be built as designed here. The actual flow depends on the *pressure drop* between sections (1) and (2); only one unique pressure drop can yield the desired flow acceleration. Temperature may also change considerably in this kind of compressible flow in which the air accelerates toward sonic speeds.

(ref. ‘Fluid Mechanics’ by & Cimbala)

127- Examples of Application of the continuity equation: Two-Dimensional Flow

EXAMPLE 9–3 Incompressibility of an Unsteady Two-Dimensional Flow

Consider the velocity field of Example 4–5—an unsteady, two-dimensional velocity field given by $\vec{V} = (u, v) = (0.5 + 0.8x)\vec{i} + [1.5 + 2.5 \sin(\omega t) - 0.8y]\vec{j}$, where angular frequency ω is equal to 2π rad/s (a physical frequency of 1 Hz). Verify that this flow field can be approximated as incompressible.

SOLUTION We are to verify that a given velocity field is incompressible.

Assumptions **1** The flow is two-dimensional, implying no z-component of velocity and no variation of u or v with z.

Analysis The components of velocity in the x- and y-directions, respectively, are

$$u = 0.5 + 0.8x \qquad \text{and} \qquad v = 1.5 + 2.5 \sin(\omega t) - 0.8y$$

If the flow is incompressible, Eq. 9–16 must apply. More specifically, in Cartesian coordinates Eq. 9–17 must apply. Let's check:

$$\underbrace{\frac{\partial u}{\partial x}}_{0.8} + \underbrace{\frac{\partial v}{\partial y}}_{-0.8} + \underbrace{\frac{\partial w}{\partial z}}_{0 \text{ since 2-D}} = 0 \qquad \rightarrow \qquad 0.8 - 0.8 = 0$$

So we see that the incompressible continuity equation is indeed satisfied at any instant in time, and **this flow field may be approximated as incompressible**.

Discussion Although there is an unsteady term in v, it has no y-derivative and drops out of the continuity equation.

(ref. 'Fluid Mechanics' by & Cimbala)

128- Examples of Application of the continuity equation: Three-Dimensional Flow

EXAMPLE 9–4 Finding a Missing Velocity Component

The u velocity component of a steady, two-dimensional, incompressible flow field is $u = ax + by$, where a and b are constants. Velocity component v is missing (Fig. 9–14). Generate an expression for v as a function of x and y.

SOLUTION We are to find the *y component* of velocity v, using a given expression for u.

Assumptions **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the xy-plane, implying that $w = 0$ and neither u nor v depends on z.

Analysis We plug the velocity components into the steady incompressible continuity equation,

Condition for incompressibility:

$$\frac{\partial v}{\partial y} = -\underbrace{\frac{\partial u}{\partial x}}_{a} - \underbrace{\frac{\partial w}{\partial z}}_{0} \quad \rightarrow \quad \frac{\partial v}{\partial y} = -a$$

Next we integrate with respect to y. Note that since the integration is a *partial* integration, we must add some arbitrary function of x instead of simply a constant of integration.

Solution: $$v = -ay + f(x)$$

If the flow were three-dimensional, we would add a function of x and z instead.

Discussion To satisfy the incompressible continuity equation, any function of x will work since there are no derivatives of v with respect to x in the continuity equation. Not all functions of x are necessarily physically possible, however, since the flow may not be able to satisfy the steady conservation of momentum equation.

(ref. 'Fluid Mechanics' by & Cimbala)

129- Examples of Application of the continuity equation: Three-Dimensional Flow

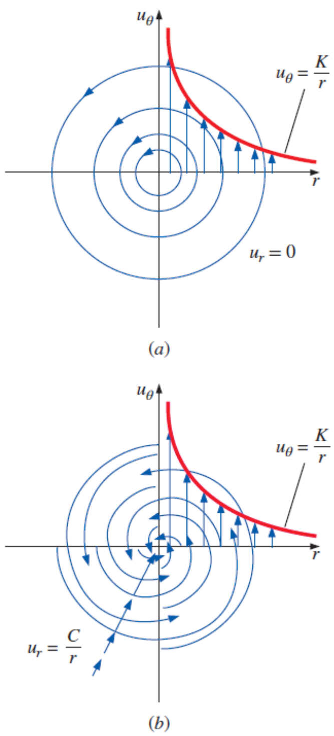


FIGURE 9–15
Streamlines and velocity profiles for (*a*) a line vortex flow and (*b*) a spiraling line vortex/sink flow.

EXAMPLE 9–5 Two-Dimensional, Incompressible, Vortical Flow

Consider a two-dimensional, incompressible flow in cylindrical coordinates; the tangential velocity component is $u_\theta = K/r$, where K is a constant. This represents a class of vortical flows. Generate an expression for the other velocity component, u_r.

SOLUTION For a given tangential velocity component, we are to generate an expression for the radial velocity component.

Assumptions **1** The flow is two-dimensional in the xy- ($r\theta$-) plane (velocity is not a function of z, and $u_z = 0$ everywhere). **2** The flow is incompressible.

Analysis The incompressible continuity equation (Eq. 9–18) for this two-dimensional case simplifies to

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \underbrace{\frac{\partial u_z}{\partial z}}_{0\ (2\text{-D})} = 0 \quad \rightarrow \quad \frac{\partial(ru_r)}{\partial r} = -\frac{\partial u_\theta}{\partial \theta} \tag{1}$$

The given expression for u_θ is not a function of θ, and therefore Eq. 1 reduces to

$$\frac{\partial(ru_r)}{\partial r} = 0 \quad \rightarrow \quad ru_r = f(\theta, t) \tag{2}$$

where we have introduced an arbitrary function of θ and t instead of a constant of integration, since we performed a *partial* integration with respect to r. Solving for u_r,

$$u_r = \frac{f(\theta, t)}{r} \tag{3}$$

Thus, *any radial velocity component of the form given by Eq. 3 yields a two-dimensional, incompressible velocity field that satisfies the continuity equation.*

We discuss some specific cases. The simplest case is when $f(\theta, t) = 0$ ($u_r = 0$, $u_\theta = K/r$). This yields the **line vortex** discussed in Chap. 4, as sketched in Fig. 9–15*a*. Another simple case is when $f(\theta, t) = C$, where C is a constant. This yields a radial velocity whose magnitude decays as $1/r$. For negative C, imagine a spiraling line vortex/sink flow, in which fluid elements not only revolve around the origin, but get sucked into a sink at the origin (actually a line sink along the z-axis). This is illustrated in Fig. 9–15*b*.

(ref. 'Fluid Mechanics' by & Cimbala)

130- Examples of Application of the continuity equation: Incompressible Flow

EXAMPLE 9–7 **Conditions for Incompressible Flow**

Consider a steady velocity field given by $\vec{V} = (u, v, w) = a(x^2y + y^2)\vec{i} + bxy^2\vec{j} + cx\vec{k}$, where a, b, and c are constants. Under what conditions is this flow field incompressible?

SOLUTION We are to determine a relationship between constants a, b, and c that ensures incompressibility.

Assumptions **1** The flow is steady. **2** The flow is incompressible (under certain constraints to be determined).

Analysis We apply Eq. 9–17 to the given velocity field,

$$\underbrace{\frac{\partial u}{\partial x}}_{2axy} + \underbrace{\frac{\partial v}{\partial y}}_{2bxy} + \underbrace{\cancel{\frac{\partial w}{\partial z}}}_{0} = 0 \quad \rightarrow \quad 2axy + 2bxy = 0$$

Thus to guarantee incompressibility, constants a and b must be equal in magnitude but opposite in sign.

Condition for incompressibility: $\boldsymbol{a = -b}$

Discussion If a were not equal to $-b$, this might still be a valid flow field, but density would have to vary with location in the flow field. In other words, the flow would be *compressible,* and Eq. 9–14 would need to be satisfied in place of Eq. 9–17.

(ref. 'Fluid Mechanics' by & Cimbala)

131- The Stream Function in Cartesian Coordinates

9–3 ▪ THE STREAM FUNCTION

The Stream Function in Cartesian Coordinates

Consider the simple case of incompressible, two-dimensional flow in the *xy*-plane. The continuity equation (Eq. 9–17) in Cartesian coordinates reduces to

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{9–19}$$

A clever variable transformation enables us to rewrite Eq. 9–19 in terms of *one* dependent variable (ψ) instead of *two* dependent variables (u and v). We define the **stream function** ψ as (Fig. 9–17)

Incompressible, two-dimensional stream function in Cartesian coordinates:

$$u = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = -\frac{\partial \psi}{\partial x} \tag{9–20}$$

The stream function and the corresponding velocity potential function (Chap. 10) were first introduced by the Italian mathematician Joseph Louis Lagrange (1736–1813). Substitution of Eq. 9–20 into Eq. 9–19 yields

$$\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial y}\right) + \frac{\partial}{\partial y}\left(-\frac{\partial \psi}{\partial x}\right) = \frac{\partial^2 \psi}{\partial x\, \partial y} - \frac{\partial^2 \psi}{\partial y\, \partial x} = 0$$

which is identically satisfied for any smooth function $\psi(x, y)$, because the order of differentiation (y then x versus x then y) is irrelevant.

You may ask why we chose to put the negative sign on v rather than on u. (We could have defined the stream function with the signs reversed, and continuity would still have been identically satisfied.) The answer is that although the sign is arbitrary, the definition of Eq. 9–20 leads to flow from left to right as ψ increases in the y-direction, which is usually preferred. Most fluid mechanics books define ψ in this way, although sometimes ψ is defined with the opposite signs (e.g., in some British textbooks and in the indoor air quality field, Heinsohn and Cimbala, 2003).

What have we gained by this transformation? First, as already mentioned, a single variable (ψ) replaces *two* variables (u and v)—once ψ is known, we can generate both u and v via Eq. 9–20, and we are guaranteed that the solution satisfies continuity, Eq. 9–19. Second, it turns out that the stream function has useful physical significance (Fig. 9–18). Namely,

Curves of constant ψ are **streamlines** of the flow.

This is easily proven by considering a streamline in the xy-plane, as sketched in Fig. 9–19. Recall from Chap. 4 that along such a streamline,

$$\textit{Along a streamline:}\qquad \frac{dy}{dx}=\frac{v}{u}\quad\rightarrow\quad \underbrace{-v}_{\partial\psi/\partial x}\,dx+\underbrace{u}_{\partial\psi/\partial y}\,dy=0$$

where we have applied Eq. 9–20, the definition of ψ. Thus,

$$\textit{Along a streamline:}\qquad \frac{\partial\psi}{\partial x}dx+\frac{\partial\psi}{\partial y}dy=0 \tag{9–21}$$

But for any smooth function ψ of two variables x and y, we know by the chain rule of mathematics that the total change of ψ from point (x, y) to another point $(x + dx, y + dy)$ some infinitesimal distance away is

$$\textit{Total change of }\psi\textit{:}\qquad d\psi=\frac{\partial\psi}{\partial x}dx+\frac{\partial\psi}{\partial y}dy \tag{9–22}$$

By comparing Eq. 9–21 to Eq. 9–22 we see that $d\psi = 0$ along a streamline; thus we have proven the statement that ψ is constant along streamlines.

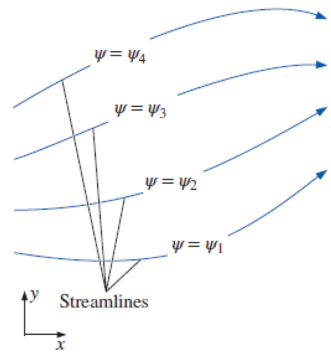
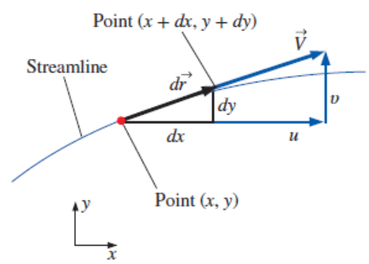


FIGURE 9–18
Curves of constant stream function represent streamlines of the flow.



EXAMPLE 9–8 Calculation of the Velocity Field

(ref. 'Fluid Mechanics' by & Cimbala)

132- The Stream Function

EXAMPLE 9–8 Calculation of the Velocity Field from the Stream Function

A steady, two-dimensional, incompressible flow field in the *xy*-plane has a stream function given by $\psi = ax^3 + by + cx$, where a, b, and c are constants: $a = 0.50\ (\text{m}\cdot\text{s})^{-1}$, $b = -2.0$ m/s, and $c = -1.5$ m/s. (*a*) Obtain expressions for velocity components u and υ. (*b*) Verify that the flow field satisfies the incompressible continuity equation. (*c*) Plot several streamlines of the flow in the upper-right quadrant.

SOLUTION For a given stream function, we are to calculate the velocity components, verify incompressibility, and plot flow streamlines.

Assumptions **1** The flow is steady. **2** The flow is incompressible (this assumption is to be verified). 3 The flow is two-dimensional in the *xy*-plane, implying that $w = 0$ and neither u nor υ depend on z.

Analysis (*a*) We use Eq. 9–20 to obtain expressions for u and υ by differentiating the stream function,

$$u = \frac{\partial \psi}{\partial y} = \mathbf{b} \qquad \text{and} \qquad \upsilon = -\frac{\partial \psi}{\partial x} = \mathbf{-3ax^2 - c}$$

(*b*) Since u is not a function of x, and υ is not a function of y, we see immediately that the two-dimensional, incompressible continuity equation (Eq. 9–19) is satisfied. In fact, since ψ is smooth in x and y, the two-dimensional, incompressible

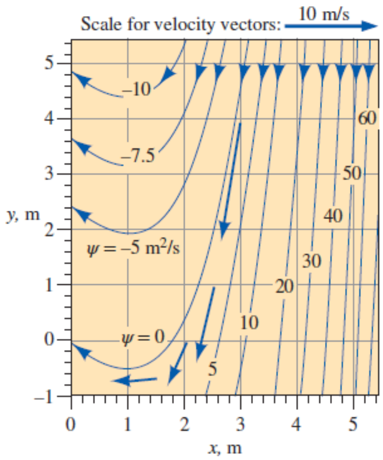


FIGURE 9–20
Streamlines for the velocity field of Example 9–8; the value of constant ψ is indicated for each streamline, and velocity vectors are shown at four locations.

continuity equation in the xy-plane is automatically satisfied by the very definition of ψ. We conclude that **the flow is indeed incompressible**.

(c) To plot streamlines, we solve the given equation for either y as a function of x and ψ, or x as a function of y and ψ. In this case, the former is easier, and we have

Equation for a streamline: $$y = \frac{\psi - ax^3 - cx}{b}$$

This equation is plotted in Fig. 9–20 for several values of ψ, and for the provided values of a, b, and c. The flow is nearly straight down at large values of x, but veers upward for $x < 1$ m.

Discussion You can verify that $\upsilon = 0$ at $x = 1$ m. In fact, υ is negative for $x > 1$ m and positive for $x < 1$ m. The direction of the flow can also be determined by picking an arbitrary point in the flow, say ($x = 3$ m, $y = 4$ m), and calculating the velocity there. We get $u = -2.0$ m/s and $\upsilon = -12.0$ m/s at this point, either of which shows that fluid flows to the lower left in this region of the flow field. For clarity, the velocity vector at this point is also plotted in Fig. 9–20; it is clearly parallel to the streamline near that point. Velocity vectors at three other locations are also plotted.

EXAMPLE 9–9 **Calculation of Stream Function for a Known Velocity Field**

(ref. ‘Fluid Mechanics’ by & Cimbala)

133- Stream Function

EXAMPLE 9–9 Calculation of Stream Function for a Known Velocity Field

Consider a steady, two-dimensional, incompressible velocity field with $u = ax + b$ and $\upsilon = -ay + cx$, where a, b, and c are constants: $a = 0.50\ s^{-1}$, $b = 1.5$ m/s, and $c = 0.35\ \text{s}^{-1}$. Generate an expression for the stream function and plot some streamlines of the flow in the upper-right quadrant.

SOLUTION For a given velocity field we are to generate an expression for ψ and plot several streamlines for given values of constants a, b, and c.

Assumptions **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the xy-plane, implying that $w = 0$ and neither u nor υ depend on z.

Analysis We start by picking one of the two parts of Eq. 9–20 that define the stream function (it doesn't matter which part we choose—the solution will be identical).

$$\frac{\partial \psi}{\partial y} = u = ax + b$$

Next we integrate with respect to y, noting that this is a *partial* integration, so we add an arbitrary function of the other variable, x, rather than a constant of integration,

$$\psi = axy + by + g(x) \qquad \textbf{(1)}$$

Now we choose the other part of Eq. 9–20, differentiate Eq. 1, and rearrange as follows:

$$\upsilon = -\frac{\partial \psi}{\partial x} = -ay - g'(x) \qquad \textbf{(2)}$$

where $g'(x)$ denotes dg/dx since g is a function of only one variable, x. We now have two expressions for velocity component υ, the equation given in the problem statement and Eq. 2. We equate these and integrate with respect to x to find $g(x)$,

$$v = -ay + cx = -ay - g'(x) \quad \rightarrow \quad g'(x) = -cx \quad \rightarrow \quad g(x) = -c\frac{x^2}{2} + C \qquad \textbf{(3)}$$

Note that here we have added an arbitrary constant of integration C since g is a function of x only. Finally, substituting Eq. 3 into Eq. 1 yields the final expression for ψ,

Solution:
$$\psi = axy + by - c\frac{x^2}{2} + C \qquad \textbf{(4)}$$

To plot the streamlines, we note that Eq. 4 represents a *family* of curves, one unique curve for each value of the constant $(\psi - C)$. Since C is arbitrary, it is common to set it equal to zero, although it can be set to any desired value. For simplicity we set $C = 0$ and solve Eq. 4 for y as a function of x, yielding

Equation for streamlines:
$$y = \frac{\psi + cx^2/2}{ax + b} \qquad \textbf{(5)}$$

For the given values of constants a, b, and c, we plot Eq. 5 for several values of ψ in Fig. 9–21; these curves of constant ψ are streamlines of the flow. From Fig. 9–21 we see that this is a smoothly converging flow in the upper-right quadrant.

Discussion It is always good to check your algebra. In this example, you should substitute Eq. 4 into Eq. 9–20 to verify that the correct velocity components are obtained.

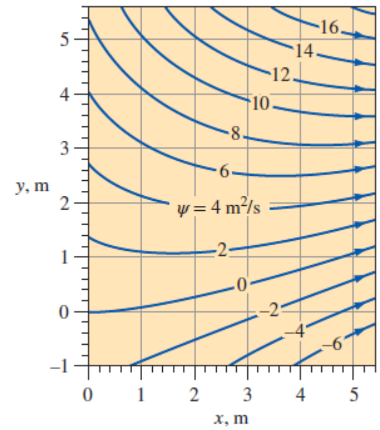


FIGURE 9–21
Streamlines for the velocity field of Example 9–9; the value of constant ψ is indicated for each streamline.

(ref. 'Fluid Mechanics' by & Cimbala)

134- The Stream Function in Cylindrical Coordinates

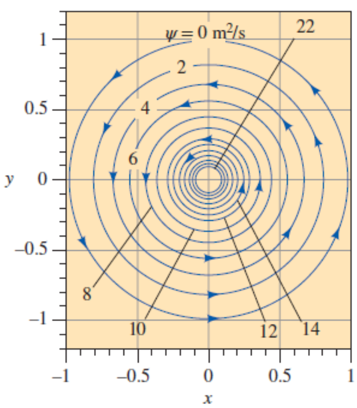


FIGURE 9–28
Streamlines for the velocity field of Example 9–12, with $K = 10$ m²/s and $C = 0$; the value of constant ψ is indicated for several streamlines.

EXAMPLE 9–12 Stream Function in Cylindrical Coordinates

Consider a line vortex, defined as steady, planar, incompressible flow in which the velocity components are $u_r = 0$ and $u_\theta = K/r$, where K is a constant. This flow is represented in Fig. 9–15*a*. Derive an expression for the stream function $\psi(r, \theta)$, and prove that the streamlines are circles.

SOLUTION For a given velocity field in cylindrical coordinates, we are to derive an expression for the stream function and show that the streamlines are circular.

Assumptions **1** The flow is steady. **2** The flow is incompressible. **3** The flow is planar in the $r\theta$-plane.

Analysis We use the definition of stream function given by Eq. 9–27. We can choose either component to start with; we choose the tangential component,

$$\frac{\partial \psi}{\partial r} = -u_\theta = -\frac{K}{r} \quad \rightarrow \quad \psi = -K \ln r + f(\theta) \tag{1}$$

Now we use the other component of Eq. 9–27,

$$u_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = \frac{1}{r} f'(\theta) \tag{2}$$

where the prime denotes a derivative with respect to θ. By equating u_r from the given information to Eq. 2, we see that

$$f'(\theta) = 0 \quad \rightarrow \quad f(\theta) = C$$

where C is an arbitrary constant of integration. Equation 1 is thus

Solution: $$\psi = -K \ln r + C \tag{3}$$

Finally, we see from Eq. 3 that curves of constant ψ are produced by setting r to a constant value. Since curves of constant r are circles by definition, **streamlines (curves of constant ψ) must therefore be circles about the origin, as in Fig. 9–15*a***.

For given values of C and ψ, we solve Eq. 3 for r to plot the streamlines,

Equation for streamlines: $$r = e^{-(\psi - C)/K} \tag{4}$$

For $K = 10$ m²/s and $C = 0$, streamlines from $\psi = 0$ to 22 are plotted in Fig. 9–28.

Discussion Notice that for a uniform increment in the value of ψ, the streamlines get closer and closer together near the origin as the tangential velocity increases. This is a direct result of the statement that the difference in the value of ψ from one streamline to another is equal to the volume flow rate per unit width between the two streamlines.

(ref. 'Fluid Mechanics' by & Cimbala)

135- The Differential Linear Momentum Equation- Cauchy's Equation

9–4 ▪ THE DIFFERENTIAL LINEAR MOMENTUM EQUATION—CAUCHY'S EQUATION

Through application of the Reynolds transport theorem (Chap. 4), we have the general expression for the linear momentum equation as applied to a control volume,

$$\sum \vec{F} = \int_{CV} \rho \vec{g}\, dV + \int_{CS} \sigma_{ij} \cdot \vec{n}\, dA = \int_{CV} \frac{\partial}{\partial t}(\rho \vec{V})\, dV + \int_{CS} (\rho \vec{V})\vec{V} \cdot \vec{n}\, dA \quad \textbf{(9–32)}$$

where σ_{ij} is the **stress tensor** introduced in Chap. 6. Components of σ_{ij} on the positive faces of an infinitesimal rectangular control volume are shown in Fig. 9–29. Equation 9–32 applies to both fixed and moving control volumes, provided that $\vec{V}$ is the absolute velocity (as seen from a fixed observer). For the special case of flow with well defined inlets and outlets, Eq. 9–32 is simplified as follows:

$$\sum \vec{F} = \sum \vec{F}_{\text{body}} + \sum \vec{F}_{\text{surface}} = \int_{CV} \frac{\partial}{\partial t}(\rho \vec{V})\, dV + \sum_{\text{out}} \beta \dot{m} \vec{V} - \sum_{\text{in}} \beta \dot{m} \vec{V} \quad \textbf{(9–33)}$$

where $\vec{V}$ in the last two terms is taken as the average velocity at an inlet or outlet, and β is the momentum flux correction factor (Chap. 6). In words, the total force acting on the control volume is equal to the rate at which momentum changes within the control volume plus the rate at which momentum flows out of the control volume minus the rate at which momentum flows into the control volume. Equation 9–33 applies to *any* control volume, regardless of its size. To generate a differential linear momentum equation, we imagine the control volume shrinking to infinitesimal size. In the limit, the entire control volume shrinks to a *point* in the flow (Fig. 9–2). We take the same approach here as we did for conservation of mass; namely, we show more than one way to derive the differential form of the linear momentum equation.

(ref. 'Fluid Mechanics' by & Cimbala)

136- Cauchy's Equation: Derivation Using the Divergence Theorem

Derivation Using the Divergence Theorem

The most straightforward (and most elegant) way to derive the differential form of the momentum equation is to apply the divergence theorem of Eq. 9–3. A more general form of the divergence theorem applies not only to vectors, but to other quantities as well, such as tensors, as illustrated in Fig. 9–30. Specifically, if we replace G_{ij} in the extended divergence theorem of Fig. 9–30 with the quantity $(\rho\vec{V})\vec{V}$, a second-order tensor, the last term in Eq. 9–32 becomes

$$\int_{CS} (\rho\vec{V})\vec{V}\cdot\vec{n}\,dA = \int_{CV} \vec{\nabla}\cdot(\rho\vec{V}\vec{V})\,dV \qquad \textbf{(9–34)}$$

where $\vec{V}\,\vec{V}$ is a vector product called the *outer product* of the velocity vector with itself. (The outer product of two vectors is *not* the same as the inner or dot product, nor is it the same as the cross product of the two vectors.) Similarly, if we replace G_{ij} in Fig. 9–30 by the stress tensor σ_{ij}, the second term on the left-hand side of Eq. 9–32 becomes

$$\int_{CS} \sigma_{ij}\cdot\vec{n}\,dA = \int_{CV} \vec{\nabla}\cdot\sigma_{ij}\,dV \qquad \textbf{(9–35)}$$

Thus, the two surface integrals of Eq. 9–32 become volume integrals by applying Eqs. 9–34 and 9–35. We combine and rearrange the terms, and rewrite Eq. 9–32 as

$$\int_{CV} \left[\frac{\partial}{\partial t}(\rho\vec{V}) + \vec{\nabla}\cdot(\rho\vec{V}\vec{V}) - \rho\vec{g} - \vec{\nabla}\cdot\sigma_{ij}\right] dV = 0 \qquad \textbf{(9–36)}$$

Finally, we argue that Eq. 9–36 must hold for *any* control volume regardless of its size or shape. This is possible only if the integrand (enclosed by square brackets) is identically zero. Hence, we have a general differential equation for linear momentum, known as **Cauchy's equation**,

Cauchy's equation:
$$\frac{\partial}{\partial t}(\rho\vec{V}) + \vec{\nabla}\cdot(\rho\vec{V}\vec{V}) = \rho\vec{g} + \vec{\nabla}\cdot\sigma_{ij} \qquad \textbf{(9–37)}$$

Equation 9–37 is named in honor of the French engineer and mathematician Augustin Louis de Cauchy (1789–1857). It is valid for compressible as well as incompressible flow since we have not made any assumptions about incompressibility. It is valid at any point in the flow domain (Fig. 9–31). Note that Eq. 9–37 is a *vector* equation, and thus represents three scalar equations, one for each coordinate axis in three-dimensional problems.

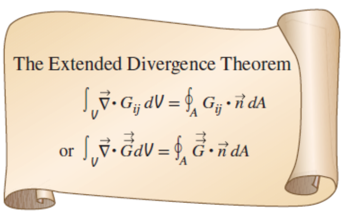
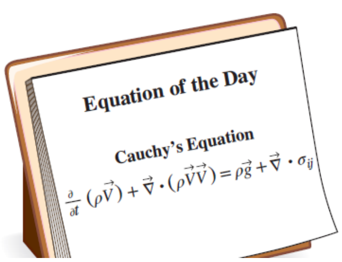


FIGURE 9–30
An extended form of the divergence theorem is useful not only for vectors, but also for tensors. In the equation, G_{ij} (or $\vec{\vec{G}}$) is a second-order tensor, V is a volume, and A is the surface area that encloses and defines the volume.



(ref. 'Fluid Mechanics' by & Cimbala)

137- The Navier–Stokes Equation

9–5 ▪ THE NAVIER–STOKES EQUATION

Introduction

Cauchy's equation (Eq. 9–37 or its alternative form Eq. 9–48) is not very useful to us as is, because the stress tensor σ_{ij} contains nine components, six of which are independent (because of symmetry). Thus, in addition to density and the three velocity components, there are six additional unknowns, for a total of 10 unknowns. (In Cartesian coordinates the unknowns are ρ, u, υ, w, σ_{xx}, σ_{xy}, σ_{xz}, σ_{yy}, σ_{yz}, and σ_{zz}.) Meanwhile, we have discussed only four equations so far—continuity (one equation) and Cauchy's equation (three equations). Of course, to be mathematically solvable, the number of equations must equal the number of unknowns, and thus we need six more equations. These equations are called **constitutive equations**, and they enable us to write the components of the stress tensor in terms of the velocity field and pressure field.

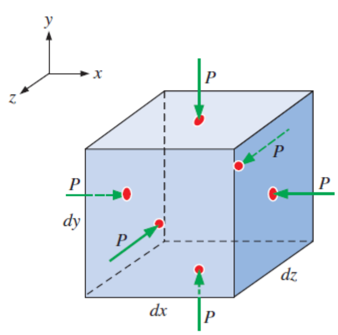


FIGURE 9–37
For fluids at rest, the only stress on a fluid element is the hydrostatic pressure, which always acts inward and normal to any surface. Note that we are ignoring gravity in this case; otherwise pressure would increase in the direction of the gravitational acceleration.

The first thing we do is separate the pressure stresses and the viscous stresses. When a fluid is at rest, the only stress acting at *any* surface of *any* fluid element is pressure P, which always acts *inward* and *normal* to the surface (Fig. 9–37). Thus, regardless of the orientation of the coordinate axes, for a fluid at rest the stress tensor reduces to

Fluid at rest:

$$\sigma_{ij} = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix} = \begin{pmatrix} -P & 0 & 0 \\ 0 & -P & 0 \\ 0 & 0 & -P \end{pmatrix} \quad \text{(9–52)}$$

Pressure P in Eq. 9–52 is the same as the **thermodynamic pressure** with which we are familiar from our study of thermodynamics. P is related to temperature and density through some type of **equation of state** (e.g., the ideal gas law). As a side note, this further complicates a compressible fluid flow analysis because we introduce yet another unknown, namely, temperature T. This new unknown requires another equation—the differential form of the energy equation—which is not discussed in this text.

When a fluid is *moving,* pressure still acts inwardly normal, but viscous stresses may also exist. We generalize Eq. 9–52 for moving fluids as

Moving fluids:

$$\sigma_{ij} = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix} = \begin{pmatrix} -P & 0 & 0 \\ 0 & -P & 0 \\ 0 & 0 & -P \end{pmatrix} + \begin{pmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{pmatrix} \quad \textbf{(9–53)}$$

where we have introduced a new tensor, τ_{ij}, called the **viscous stress tensor** or the **deviatoric stress tensor**. Mathematically, we have not helped the situation because we have replaced the six unknown components of σ_{ij} with six unknown components of τ_{ij}, and have added *another* unknown, pressure P. Fortunately, however, there are constitutive equations that express τ_{ij} in terms of the velocity field and measurable fluid properties such as viscosity. The actual form of the constitutive relations depends on the type of fluid, as discussed shortly.

As a side note, there are some subtleties associated with the pressure in Eq. 9–53. If the fluid is *incompressible,* we have no equation of state (it is replaced by the equation ρ = constant), and we can no longer define P as the thermodynamic pressure. Instead, we define P in Eq. 9–53 as the **mechanical pressure**,

Mechanical pressure:

$$P_m = -\frac{1}{3}(\sigma_{xx} + \sigma_{yy} + \sigma_{zz}) \quad \textbf{(9–54)}$$

We see from Eq. 9–54 that *mechanical pressure is the mean normal stress acting inwardly on a fluid element*. It is therefore also called **mean pressure** by some authors. Thus, when dealing with incompressible fluid flows, pressure variable P is always interpreted as the mechanical pressure P_m. For *compressible* flow fields however, pressure P in Eq. 9–53 *is* the thermodynamic pressure, but the mean normal stress felt on the surfaces of a fluid element is not necessarily the same as P (pressure variable P and mechanical pressure P_m are not necessarily equivalent). You are referred to Panton (1996) or Kundu et al. (2011) for a more detailed discussion of mechanical pressure.

(ref. 'Fluid Mechanics' by & Cimbala)

138- Navier–Stokes Equation: Newtonian versus Non-Newtonian Fluids

Newtonian versus Non-Newtonian Fluids

The study of the deformation of flowing fluids is called **rheology**; the rheological behavior of various fluids is sketched in Fig. 9–38. In this text, we concentrate on **Newtonian fluids**, defined as *fluids for which the stress tensor is linearly proportional to the strain rate tensor.* Newtonian fluids (stress proportional to strain rate) are analogous to elastic solids (Hooke's law: stress proportional to strain). Many common fluids, such as air and other gases, water, kerosene, gasoline, and other oil-based liquids, are Newtonian fluids. Fluids for which the stress tensor is *not* linearly related to the strain rate tensor are called **non-Newtonian fluids**. Examples include slurries and colloidal suspensions, polymer solutions, blood, paste, and cake batter. Some non-Newtonian fluids exhibit a "memory"—the shear stress depends not only on the local strain rate, but also on its *history*. A fluid that returns (partially) to its original shape after the applied stress is released is called **viscoelastic**.

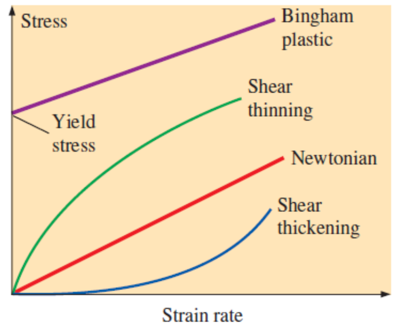


FIGURE 9–38
Rheological behavior of fluids—stress as a function of strain rate.

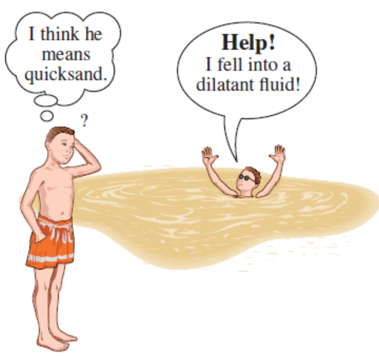


FIGURE 9–39
When an engineer falls into quicksand (a *dilatant fluid*), the faster he tries to move, the more viscous the fluid becomes.

Some non-Newtonian fluids are called **shear thinning fluids** or **pseudoplastic fluids**, because the more the fluid is sheared, the less viscous it becomes. A good example is paint. Paint is very viscous when poured from the can or when picked up by a paintbrush, since the shear rate is small. However, as we apply the paint to the wall, the thin layer of paint between the paintbrush and the wall is subjected to a large shear rate, and it becomes much less viscous. **Plastic fluids** are those in which the shear thinning effect is extreme. In some fluids a finite stress called the **yield stress** is required before the fluid begins to flow at all; such fluids are called **Bingham plastic fluids**. Certain pastes such as acne cream and toothpaste are examples of Bingham plastic fluids. If you hold the tube upside down, the paste does not flow, even though there is a nonzero stress due to gravity. However, if you squeeze the tube (greatly increasing the stress), the paste flows like a very viscous fluid. Other fluids show the opposite effect and are called **shear thickening fluids** or **dilatant fluids**; the more the fluid is sheared, the *more* viscous it becomes. The best example is quicksand, a thick mixture of sand and water. As we all know from Hollywood movies, it is easy to move *slowly* through quicksand, since the viscosity is low; but if you panic and try to move quickly, the viscous resistance increases considerably and you get "stuck" (Fig. 9–39). You can create your own quicksand by mixing two parts cornstarch with one part water—try it! Shear thickening fluids are used in some exercise equipment—the faster you pull, the more resistance you encounter.

(ref. 'Fluid Mechanics' by & Cimbala)

139- Derivation of the Navier–Stokes Equation for Incompressible, Isothermal Flow

Derivation of the Navier–Stokes Equation for Incompressible, Isothermal Flow

From this point on, we limit our discussion to Newtonian fluids, where by definition the stress tensor is linearly proportional to the strain rate tensor. The general result (for compressible flow) is rather involved and is not included here. Instead, we assume incompressible flow (ρ = constant). We also assume nearly isothermal flow—namely, that local changes in temperature are small or nonexistent; this eliminates the need for a differential energy equation. A further consequence of the latter assumption is that fluid properties, such as dynamic viscosity μ and kinematic viscosity ν, are constant as well (Fig. 9–40). With these assumptions, it can be shown (Kundu et al., 2011) that the viscous stress tensor reduces to

Viscous stress tensor for an incompressible Newtonian fluid with constant properties:

$$\tau_{ij} = 2\mu\varepsilon_{ij} \qquad \textbf{(9–55)}$$

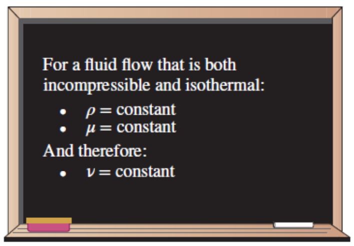


FIGURE 9–40
The incompressible flow approximation implies constant density, and the isothermal approximation implies constant viscosity.

where ε_{ij} is the strain rate tensor defined in Chap. 4. Equation 9–55 shows that stress is linearly proportional to strain. In Cartesian coordinates, the nine components of the viscous stress tensor are listed, only six of which are independent due to symmetry:

$$\tau_{ij} = \begin{pmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{pmatrix} = \begin{pmatrix} 2\mu\dfrac{\partial u}{\partial x} & \mu\left(\dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x}\right) & \mu\left(\dfrac{\partial u}{\partial z} + \dfrac{\partial w}{\partial x}\right) \\ \mu\left(\dfrac{\partial v}{\partial x} + \dfrac{\partial u}{\partial y}\right) & 2\mu\dfrac{\partial v}{\partial y} & \mu\left(\dfrac{\partial v}{\partial z} + \dfrac{\partial w}{\partial y}\right) \\ \mu\left(\dfrac{\partial w}{\partial x} + \dfrac{\partial u}{\partial z}\right) & \mu\left(\dfrac{\partial w}{\partial y} + \dfrac{\partial v}{\partial z}\right) & 2\mu\dfrac{\partial w}{\partial z} \end{pmatrix} \qquad \textbf{(9–56)}$$

In Cartesian coordinates the stress tensor of Eq. 9–53 thus becomes

$$\sigma_{ij} = \begin{pmatrix} -P & 0 & 0 \\ 0 & -P & 0 \\ 0 & 0 & -P \end{pmatrix} + \begin{pmatrix} 2\mu\dfrac{\partial u}{\partial x} & \mu\left(\dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x}\right) & \mu\left(\dfrac{\partial u}{\partial z} + \dfrac{\partial w}{\partial x}\right) \\ \mu\left(\dfrac{\partial v}{\partial x} + \dfrac{\partial u}{\partial y}\right) & 2\mu\dfrac{\partial v}{\partial y} & \mu\left(\dfrac{\partial v}{\partial z} + \dfrac{\partial w}{\partial y}\right) \\ \mu\left(\dfrac{\partial w}{\partial x} + \dfrac{\partial u}{\partial z}\right) & \mu\left(\dfrac{\partial w}{\partial y} + \dfrac{\partial v}{\partial z}\right) & 2\mu\dfrac{\partial w}{\partial z} \end{pmatrix} \qquad \textbf{(9–57)}$$

Now we substitute Eq. 9–57 into the three Cartesian components of Cauchy's equation. Let's consider the *x-component* first. Equation 9–51a becomes

$$\rho\frac{Du}{Dt} = -\frac{\partial P}{\partial x} + \rho g_x + 2\mu\frac{\partial^2 u}{\partial x^2} + \mu\frac{\partial}{\partial y}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) + \mu\frac{\partial}{\partial z}\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right) \qquad \textbf{(9–58)}$$

Notice that since pressure consists of a normal stress only, it contributes only one term to Eq. 9–58. However, since the viscous stress tensor consists of both normal and shear stresses, it contributes *three* terms. (This is a direct result of taking the divergence of a second-order tensor, by the way.)

We note that as long as the velocity components are smooth functions of x, y, and z, the order of differentiation is irrelevant. For example, the first part of the last term in Eq. 9–58 can be rewritten as

$$\mu \frac{\partial}{\partial z}\left(\frac{\partial w}{\partial x}\right) = \mu \frac{\partial}{\partial x}\left(\frac{\partial w}{\partial z}\right)$$

After some clever rearrangement of the viscous terms in Eq. 9–58,

$$\rho\frac{Du}{Dt} = -\frac{\partial P}{\partial x} + \rho g_x + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x}\frac{\partial u}{\partial x} + \frac{\partial}{\partial x}\frac{\partial v}{\partial y} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial x}\frac{\partial w}{\partial z} + \frac{\partial^2 u}{\partial z^2}\right]$$

$$= -\frac{\partial P}{\partial x} + \rho g_x + \mu\left[\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) + \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

The term in parentheses is zero because of the continuity equation for incompressible flow (Eq. 9–17). We also recognize the last three terms as the **Laplacian** of velocity component u in Cartesian coordinates (Fig. 9–41). Thus, we write the x-component of the momentum equation as

$$\rho\frac{Du}{Dt} = -\frac{\partial P}{\partial x} + \rho g_x + \mu\nabla^2 u \qquad \textbf{(9–59a)}$$

Similarly, the y- and z-components of the momentum equation reduce to

$$\rho\frac{Dv}{Dt} = -\frac{\partial P}{\partial y} + \rho g_y + \mu\nabla^2 v \qquad \textbf{(9–59b)}$$

and

$$\rho\frac{Dw}{Dt} = -\frac{\partial P}{\partial z} + \rho g_z + \mu\nabla^2 w \qquad \textbf{(9–59c)}$$

respectively. Finally, we combine the three components into one vector equation; the result is the **Navier–Stokes equation** for incompressible flow with constant viscosity.

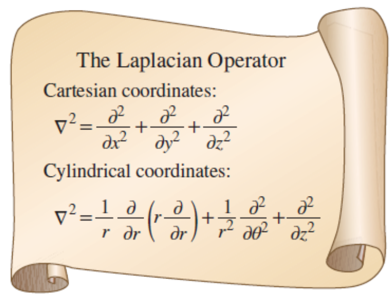


FIGURE 9–41
The Laplacian operator, shown here in both Cartesian and cylindrical coordinates, appears in the viscous term of the incompressible Navier–Stokes equation.

(ref. 'Fluid Mechanics' by & Cimbala)

140- Continuity and Navier–Stokes Equation in Cylindrical Coordinates

Continuity and Navier–Stokes Equations in Cartesian Coordinates

The continuity equation (Eq. 9–16) and the Navier–Stokes equation (Eq. 9–60) are expanded in Cartesian coordinates (x, y, z) and (u, υ, w):

Incompressible continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial \upsilon}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad \textbf{(9–61a)}$$

x-component of the incompressible Navier–Stokes equation:

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + \upsilon\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial P}{\partial x} + \rho g_x + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \textbf{(9–61b)}$$

y-component of the incompressible Navier–Stokes equation:

$$\rho\left(\frac{\partial \upsilon}{\partial t} + u\frac{\partial \upsilon}{\partial x} + \upsilon\frac{\partial \upsilon}{\partial y} + w\frac{\partial \upsilon}{\partial z}\right) = -\frac{\partial P}{\partial y} + \rho g_y + \mu\left(\frac{\partial^2 \upsilon}{\partial x^2} + \frac{\partial^2 \upsilon}{\partial y^2} + \frac{\partial^2 \upsilon}{\partial z^2}\right) \qquad \textbf{(9–61c)}$$

z-component of the incompressible Navier–Stokes equation:

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + \upsilon\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial P}{\partial z} + \rho g_z + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \qquad \textbf{(9–61d)}$$

Continuity and Navier–Stokes Equations in Cylindrical Coordinates

The continuity equation (Eq. 9–16) and the Navier–Stokes equation (Eq. 9–60) are expanded in cylindrical coordinates (r, θ, z) and (u_r, u_θ, u_z):

Incompressible continuity equation:

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{1}{r}\frac{\partial(u_\theta)}{\partial \theta} + \frac{\partial(u_z)}{\partial z} = 0 \qquad \textbf{(9–62a)}$$

r-component of the incompressible Navier–Stokes equation:

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta} - \frac{u_\theta^2}{r} + u_z\frac{\partial u_r}{\partial z}\right)$$

$$= -\frac{\partial P}{\partial r} + \rho g_r + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right) - \frac{u_r}{r^2} + \frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2}\right] \qquad \textbf{(9–62b)}$$

θ-component of the incompressible Navier–Stokes equation:

$$\rho\left(\frac{\partial u_\theta}{\partial t} + u_r\frac{\partial u_\theta}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{u_r u_\theta}{r} + u_z\frac{\partial u_\theta}{\partial z}\right)$$

$$= -\frac{1}{r}\frac{\partial P}{\partial \theta} + \rho g_\theta + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\theta}{\partial r}\right) - \frac{u_\theta}{r^2} + \frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial \theta^2} + \frac{2}{r^2}\frac{\partial u_r}{\partial \theta} + \frac{\partial^2 u_\theta}{\partial z^2}\right] \qquad \textbf{(9–62c)}$$

z-component of the incompressible Navier–Stokes equation:

$$\rho\left(\frac{\partial u_z}{\partial t} + u_r\frac{\partial u_z}{\partial r} + \frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta} + u_z\frac{\partial u_z}{\partial z}\right)$$

$$= -\frac{\partial P}{\partial z} + \rho g_z + \mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_z}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2}\right] \qquad \textbf{(9–62d)}$$

Alternative Form of the Viscous Terms

It can be shown that

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right) - \frac{u_r}{r^2} = \frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(ru_r)\right)$$

and

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\theta}{\partial r}\right) - \frac{u_\theta}{r^2} = \frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(ru_\theta)\right)$$

FIGURE 9–43
An alternative form for the first two viscous terms in the r- and θ-components of the Navier–Stokes equation.

(ref. 'Fluid Mechanics' by & Cimbala)

141- Analysis of Fluid Flow Problems

Three-Dimensional Incompressible Flow

Four variables or unknowns:

- Pressure P
- Three components of velocity $\vec{V}$

Four equations of motion:

- Continuity,

$$\vec{\nabla} \cdot \vec{V} = 0$$

- Three components of Navier–Stokes,

$$\rho \frac{D\vec{V}}{Dt} = -\vec{\nabla} P + \rho \vec{g} + \mu \nabla^2 \vec{V}$$

FIGURE 9–45
A general three-dimensional but incompressible flow field with constant properties requires four equations to solve for four unknowns.

9–6 ▪ DIFFERENTIAL ANALYSIS OF FLUID FLOW PROBLEMS

In this section we show how to apply the differential equations of motion in both Cartesian and cylindrical coordinates. There are two types of problems for which the differential equations (continuity and Navier–Stokes) are useful:

- Calculating the pressure field for a known velocity field
- Calculating both the velocity and pressure fields for a flow of known geometry and known boundary conditions

For simplicity, we consider only incompressible flow, eliminating calculation of ρ as a variable. In addition, the form of the Navier–Stokes equation derived in Section 9–5 is valid only for Newtonian fluids with constant properties (viscosity, thermal conductivity, etc.). Finally, we assume negligible temperature variations, so that T is not a variable. We are left with four variables or unknowns (pressure plus three components of velocity), and we have four differential equations (Fig. 9–45).

(ref. ‘Fluid Mechanics’ by & Cimbala)

142- Boundary Layer

10–6 ▪ THE BOUNDARY LAYER APPROXIMATION

As discussed in Sections 10–4 and 10–5, there are at least two flow situations in which the viscous term in the Navier–Stokes equation can be neglected. The first occurs in high Reynolds number regions of flow where net viscous forces are known to be negligible compared to inertial and/or pressure forces; we call these *inviscid regions of flow*. The second situation occurs when the vorticity is negligibly small; we call these *irrotational* or *potential regions of flow*. In either case, removal of the viscous terms from the Navier–Stokes equation yields the Euler equation (Eq. 10–13 and also Eq. 10–25). While the math is greatly simplified by dropping the viscous terms, there are some serious deficiencies associated with application of the Euler equation to practical engineering flow problems. High on the list of deficiencies is the inability to specify the no-slip condition at solid walls. This leads to unphysical results such as zero viscous shear forces on solid walls and zero aerodynamic drag on bodies immersed in a free stream. We can therefore think of the Euler equation and the Navier–Stokes equation as two mountains separated by a huge chasm (Fig. 10–75*a*). We make the following statement about the boundary layer approximation:

> The boundary layer approximation bridges the gap between the Euler equation and the Navier–Stokes equation, and between the slip condition and the no-slip condition at solid walls (Fig. 10–75*b*).

From a historical perspective, by the mid-1800s, the Navier–Stokes equation was known, but couldn't be solved except for flows of very simple geometries. Meanwhile, mathematicians were able to obtain beautiful analytical solutions of the Euler equation and of the potential flow equations for flows of complex geometry, but their results were often physically meaningless. Hence, the only reliable way to study fluid flows was empirically, i.e., with experiments. A major breakthrough in fluid mechanics occurred in 1904 when Ludwig Prandtl (1875–1953) introduced the **boundary layer approximation**. Prandtl's idea was to divide the flow into two regions: an **outer flow region** that is inviscid and/or irrotational, and an inner flow region called a **boundary layer**—a very thin region of flow near a solid wall where viscous forces and rotationality cannot be ignored (Fig. 10–76). In the outer flow region, we use the continuity and Euler equations to obtain the outer flow velocity field, and the Bernoulli equation to obtain the pressure field. Alternatively, if the outer flow region is irrotational, we may use the potential flow techniques discussed in Section 10–5 (e.g., superposition) to obtain the outer flow velocity field. In either case, we solve for the outer flow region *first*, and then fit in a thin boundary layer in regions where rotationality and viscous forces cannot be neglected. Within the boundary layer we solve the **boundary layer equations**, to be discussed shortly. (Note that the boundary layer equations are themselves approximations of the full Navier–Stokes equation, as we will see.)

The key to successful application of the boundary layer approximation is the assumption that the boundary layer is very *thin*. The classic example is a uniform stream flowing parallel to a long flat plate aligned with the x-axis. **Boundary layer thickness** δ at some location x along the plate is sketched in Fig. 10–77. By convention, δ is usually defined as the distance away from the wall at which the velocity component parallel to the wall is 99 percent of the fluid speed outside the boundary layer. It turns out that for a given fluid and plate, the higher the free-stream speed V, the thinner the boundary layer (Fig. 10–77). In nondimensional terms, we define the Reynolds number based on distance x along the wall,

Reynolds number along a flat plate:
$$\text{Re}_x = \frac{\rho V x}{\mu} = \frac{Vx}{\nu} \qquad \textbf{(10–60)}$$

Hence,

> At a given x-location, the higher the Reynolds number, the thinner the boundary layer.

In other words, the higher the Reynolds number, all else being equal, the more reliable the boundary layer approximation. We are confident that the boundary layer is thin when $\delta \ll x$ (or, expressed nondimensionally, $\delta/x \ll 1$).

The shape of the boundary layer profile can be obtained experimentally by flow visualization. An example is shown in Fig. 10–78 for a laminar boundary layer on a flat plate. Taken over 60 years ago by F. X. Wortmann, this is now considered a classic photograph of a laminar flat plate boundary layer profile.

(ref. ‘Fluid Mechanics’ by & Cimbala)

143- What is a Boundary Layer?

The Boundary Layer Equations

Now that we have a physical feel for boundary layers, we need the equations of motion to be used in boundary layer calculations—the **boundary layer equations**. For simplicity we consider only steady, two-dimensional flow in the xy-plane in Cartesian coordinates. The methodology used here can be extended, however, to axisymmetric boundary layers or to three-dimensional boundary layers in any coordinate system. We neglect gravity since we are not dealing with free surfaces or with buoyancy-driven flows (free convection flows), where gravitational effects dominate. We consider only *laminar* boundary layers; turbulent boundary layer equations are beyond the scope of this text. For the case of a boundary layer along a solid wall, we adopt a coordinate system in which x is everywhere parallel to the wall and y is everywhere normal to the wall (Fig. 10–85). This coordinate system is called a **boundary layer coordinate system**. When we solve the boundary layer equations, we do so at one x-location at a time, using this coordinate system *locally*, and it is *locally orthogonal*. It is not critical where we define $x = 0$, but for flow over a body, as in Fig. 10–85, we typically set $x = 0$ at the front stagnation point.

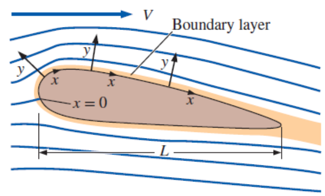


FIGURE 10–85
The boundary layer coordinate system for flow over a body; x follows the surface and is typically set to zero at the front stagnation point of the body, and y is everywhere normal to the surface locally.

How do we decide which terms to keep and which to neglect? To answer this question, we redo the nondimensionalization of the equations of motion based on appropriate length and velocity scales within the boundary layer. A magnified view of a portion of the boundary layer of Fig. 10–85 is sketched in Fig. 10–86. Since the order of magnitude of x is L, we use L as an appropriate length scale for distances in the streamwise direction and for derivatives of velocity and pressure with respect to x. However, this length scale is much too large for derivatives with respect to y. It makes more sense to use δ as the length scale for distances in the direction normal to the streamwise direction and for derivatives with respect to y. Similarly, while the characteristic velocity scale is V for the whole flow field, it is more appropriate to use U as the characteristic velocity scale for boundary layers, where *U is the magnitude of the velocity component parallel to the wall at a location just above the boundary layer* (Fig. 10–86). U is in general a function of x. Thus, within the boundary layer at some value of x, the orders of magnitude are

$$u \sim U \qquad P - P_\infty \sim \rho U^2 \qquad \frac{\partial}{\partial x} \sim \frac{1}{L} \qquad \frac{\partial}{\partial y} \sim \frac{1}{\delta} \tag{10–62}$$

The order of magnitude of velocity component υ is not specified in Eq. 10–62, but is instead obtained from the continuity equation. Applying the orders of magnitude in Eq. 10–62 to the incompressible continuity equation in two dimensions,

$$\underbrace{\frac{\partial u}{\partial x}}_{\sim U/L} + \underbrace{\frac{\partial \upsilon}{\partial y}}_{\sim \upsilon/\delta} = 0 \qquad \rightarrow \qquad \frac{U}{L} \sim \frac{\upsilon}{\delta}$$

Since the two terms have to balance each other, they must be of the same order of magnitude. Thus we obtain the order of magnitude of velocity component υ,

$$\upsilon \sim \frac{U\delta}{L} \tag{10–63}$$

Since $\delta/L \ll 1$ in a boundary layer (the boundary layer is very thin), we conclude that $\upsilon \ll u$ in a boundary layer (Fig. 10–87). From Eqs. 10–62 and 10–63, we define the following nondimensional variables within the boundary layer:

$$x^* = \frac{x}{L} \qquad y^* = \frac{y}{\delta} \qquad u^* = \frac{u}{U} \qquad \upsilon^* = \frac{\upsilon L}{U\delta} \qquad P^* = \frac{P - P_\infty}{\rho U^2}$$

Since we used appropriate scales, all these nondimensional variables are of order unity—i.e., they are *normalized* variables (Chap. 7).

We now consider the x- and y-components of the Navier–Stokes equation. We substitute these nondimensional variables into the y-momentum equation, giving

$$\underbrace{u}_{u^* U}\underbrace{\frac{\partial \upsilon}{\partial x}}_{\frac{\partial}{\partial x^*}\frac{\upsilon^* U\delta}{L^2}} + \underbrace{\upsilon}_{\upsilon^*\frac{U\delta}{L}}\underbrace{\frac{\partial \upsilon}{\partial y}}_{\frac{\partial}{\partial y^*}\frac{\upsilon^* U\delta}{L\delta}} = \underbrace{-\frac{1}{\rho}\frac{\partial P}{\partial y}}_{\frac{1}{\rho}\frac{\partial}{\partial y^*}\frac{P^*\rho U^2}{\delta}} + \underbrace{\nu\frac{\partial^2 \upsilon}{\partial x^2}}_{\nu\frac{\partial^2}{\partial x^{*2}}\frac{\upsilon^* U\delta}{L^3}} + \underbrace{\nu\frac{\partial^2 \upsilon}{\partial y^2}}_{\nu\frac{\partial^2}{\partial y^{*2}}\frac{\upsilon^* U\delta}{L\delta^2}}$$

After some algebra and after multiplying each term by $L^2/(U^2\delta)$, we get

$$u^*\frac{\partial \upsilon^*}{\partial x^*} + \upsilon^*\frac{\partial \upsilon^*}{\partial y^*} = -\left(\frac{L}{\delta}\right)^2\frac{\partial P^*}{\partial y^*} + \left(\frac{\nu}{UL}\right)\frac{\partial^2 \upsilon^*}{\partial x^{*2}} + \left(\frac{\nu}{UL}\right)\left(\frac{L}{\delta}\right)^2\frac{\partial^2 \upsilon^*}{\partial y^{*2}} \qquad \textbf{(10–64)}$$

Comparing terms in Eq. 10–64, the middle term on the right side is clearly orders of magnitude smaller than any other term since $\text{Re}_L = UL/\nu \gg 1$. For the same reason, the last term on the right is much smaller than the first term on the right. Neglecting these two terms leaves the two terms on the left and the first term on the right. However, since $L \gg \delta$, the pressure gradient term is orders of magnitude greater than the advective terms on the left side of the equation. Thus, the only term left in Eq. 10–64 is the pressure term. Since no other term in the equation can balance that term, we have no choice but to set it equal to zero. Thus, the nondimensional y-momentum equation reduces to

$$\frac{\partial P^*}{\partial y^*} \cong 0$$

or, in terms of the physical variables,

Normal pressure gradient through a boundary layer: $$\frac{\partial P}{\partial y} \cong 0 \qquad \textbf{(10–65)}$$

In words, although pressure may vary *along* the wall (in the x-direction), there is negligible change in pressure in the direction *normal* to the wall. This is illustrated in Fig. 10–88. At $x = x_1$, $P = P_1$ at all values of y across the boundary layer from the wall to the outer flow. At some other x-location, $x = x_2$, the pressure may have changed, but $P = P_2$ at all values of y across that portion of the boundary layer.

The pressure across a boundary layer (y-direction) is nearly constant.

Physically, because the boundary layer is so thin, streamlines within the boundary layer have negligible *curvature* when observed at the scale of the boundary layer thickness. Curved streamlines require a *centripetal acceleration,* which comes from a pressure gradient along the radius of curvature. Since the streamlines are not significantly curved in a thin boundary layer, there is no significant pressure gradient across the boundary layer.

Returning to the development of the boundary layer equations, we use Eq. 10–65 to greatly simplify the x-component of the momentum equation. Specifically, since P is not a function of y, we replace $\partial P/\partial x$ by dP/dx, where P is the value of pressure calculated from our outer flow approximation (using either continuity plus Euler, or the potential flow equations plus Bernoulli). The x-component of the Navier–Stokes equation becomes

$$\underbrace{u}_{u^*U}\underbrace{\frac{\partial u}{\partial x}}_{\frac{\partial}{\partial x^*}\frac{u^*U}{L}} + \underbrace{\upsilon}_{\upsilon^*\frac{U\delta}{L}}\underbrace{\frac{\partial u}{\partial y}}_{\frac{\partial}{\partial y^*}\frac{u^*U}{\delta}} = \underbrace{-\frac{1}{\rho}\frac{dP}{dx}}_{\frac{1}{\rho}\frac{\partial}{\partial x^*}\frac{P^*\rho U^2}{L}} + \underbrace{\nu\frac{\partial^2 u}{\partial x^2}}_{\upsilon\frac{\partial^2}{\partial x^{*2}}\frac{u^*U}{L^2}} + \underbrace{\nu\frac{\partial^2 u}{\partial y^2}}_{\upsilon\frac{\partial^2}{\partial y^{*2}}\frac{u^*U}{\delta^2}}$$

After some algebra, and after multiplying each term by L/U^2, we get

$$u^*\frac{\partial u^*}{\partial x^*} + \upsilon^*\frac{\partial u^*}{\partial y^*} = -\frac{dP^*}{dx^*} + \left(\frac{\nu}{UL}\right)\frac{\partial^2 u^*}{\partial x^{*2}} + \left(\frac{\nu}{UL}\right)\left(\frac{L}{\delta}\right)^2\frac{\partial^2 u^*}{\partial y^{*2}} \tag{10–66}$$

Comparing terms in Eq. 10–66, the middle term on the right side is clearly orders of magnitude smaller than the terms on the left side, since $\mathrm{Re}_L = UL/\nu \gg 1$. What about the last term on the right? If we neglect this term, we throw out all the viscous terms and are back to the Euler equation. Clearly this term must remain. Furthermore, since all the remaining terms in Eq. 10–66 are of order unity, the combination of parameters in parentheses in the last term on the right side of Eq. 10–66 must also be of order unity,

$$\left(\frac{\nu}{UL}\right)\left(\frac{L}{\delta}\right)^2 \sim 1$$

Again recognizing that $\mathrm{Re}_L = UL/\nu$, we see immediately that

$$\frac{\delta}{L} \sim \frac{1}{\sqrt{\mathrm{Re}_L}} \tag{10–67}$$

This confirms our previous statement that at a given streamwise location along the wall, the larger the Reynolds number, the thinner the boundary layer. If we substitute x for L in Eq. 10–67, we also conclude that for a laminar boundary layer on a flat plate, where $U(x) = V = \text{constant}$, δ grows like the square root of x (Fig. 10–90).

In terms of the original (physical) variables, Eq. 10–66 is written as

x-momentum boundary layer equation: $$u\frac{\partial u}{\partial x}+\upsilon\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{dP}{dx}+\nu\frac{\partial^2 u}{\partial y^2} \quad \textbf{(10–68)}$$

Note that the last term in Eq. 10–68 is not negligible in the boundary layer, since the y-derivative of velocity gradient $\partial u/\partial y$ is sufficiently large to offset the (typically small) value of kinematic viscosity ν. Finally, since we know from our y-momentum equation analysis that the pressure across the boundary layer is the same as that outside the boundary layer (Eq. 10–65), we apply the Bernoulli equation to the outer flow region. Differentiating with respect to x we get

$$\frac{P}{\rho}+\frac{1}{2}U^2=\text{constant} \quad \rightarrow \quad \frac{1}{\rho}\frac{dP}{dx}=-U\frac{dU}{dx} \quad \textbf{(10–69)}$$

where we note that both P and U are functions of x only, as illustrated in Fig. 10–91. Substitution of Eq. 10–69 into Eq. 10–68 yields

$$u\frac{\partial u}{\partial x}+\upsilon\frac{\partial u}{\partial y}=U\frac{dU}{dx}+\nu\frac{\partial^2 u}{\partial y^2} \quad \textbf{(10–70)}$$

and we have eliminated pressure from the boundary layer equations.

We summarize the set of equations of motion for a steady, incompressible, laminar boundary layer in the xy-plane without significant gravitational effects,

Boundary layer equations:

$$\frac{\partial u}{\partial x}+\frac{\partial \upsilon}{\partial y}=0$$

$$u\frac{\partial u}{\partial x}+\upsilon\frac{\partial u}{\partial y}=U\frac{dU}{dx}+\nu\frac{\partial^2 u}{\partial y^2} \quad \textbf{(10–71)}$$

(ref. ‘Fluid Mechanics’ by & Cimbala)