Chapter 2 **Fluid Properties**

fluid is a substance whose molecules move freely past each other, *or it's a substance that* deforms continuously when subjected to a shear stress.

Shear stress exists in fluid when the particles of fluid move in different velocities relative to each other.

Shear stress is equal zero when

a) Fluid at rest b) Fluid is moving in common velocity to all of its particles.

Fluid "gasses or liquids"

Gases : Widely spaced molecules with small intermolecular forces Take volume and shape of container. Compressible: density is not constant with pressure, doesn't form a free surface.

Liquids: Closely spaced molecules with large intermolecular forces, Retain volume and take shape of container. Incompressible: density is constant with pressure. They form a free surface.

2.1 Properties Involving Mass and Weight

Density: the quantity of matter contained in a unit volume of the substance.

Mass density "$\boldsymbol{\rho}$"
the ratio of mass to volume at a point

$$\rho = \frac{mass}{Volume} \text{ "}kg/m^3\text{"}$$

Density of water at 4 ^oC= 1000 kg/m^3
Density of air at 4 ^oC= 1.27 kg/m^3

Specific weight "$\boldsymbol{\gamma}$"
The gravitational force per unit volume of fluid, or simply the weight per unit volume,

$$\gamma = \rho g \text{ "}N/m^3\text{"}$$

Specific weights of common liquids are given in Table A.4.

Specific Gravity "S"
The ratio of the specific weight of a given fluid to the specific weight of water at the standard reference temperature 4°C

$$S = \frac{\gamma_{fluid}}{\gamma_{water}} = \frac{\rho_{fluid}}{\rho_{water}}$$

Ideal Gas Law "ONLY FOR GASES"

$$P\forall = nR_uT$$

P: the absolute pressure, ∀: Volume, n: number of moles, R_u: universal gas constant "8.314 kJ/kmol-K", T: absolute temperature.

$$P = \frac{nM}{\forall}\frac{R_uT}{M}$$

M: molecular weight , $\frac{R_u}{M} = R$:gas constant ,$\frac{nM}{\forall}$: mass per unit volume, or density

$$P = \rho RT$$

EXAMPLE 2.1 DENSITY OF AIR
Air at standard sea-level pressure (*p* = 101 kN/m^2 "Pa") has a temperature of 4°C. What is the density of the air?
Problem Definition: Situation: Air with a known temperature and pressure.
Find: Density (kg/m3).
Properties: Air, 4°C, *p* at 101 kN/m^2 "Pa"; Table A.2, *R* = 287 J/kg K.
Plan
Apply the ideal gas law, Eq. 2.5, to solve for density, ρ
Solution

$$\rho = P/RT$$

$$\rho = \frac{101x10^3\ \mathrm{N}/m^2}{(287)\ \mathrm{J/kg\ K}(273+4)\ \mathrm{K}} = 1.27kg/m^3$$

Remember: Use absolute temperatures and pressures with the ideal gas law.
Remember: use *R* from Table A.2. Do not use *Ru*.

2.3 Properties Involving Thermal Energy

Specific Heat, *c*

The property that describes the capacity of a substance to store thermal energy
The specific heat of a gas depends on the process accompanying the change in temperature. If the *specific volume v* of the gas *(v* = 1/ρ) remains constant while the temperature changes, then the specific heat is identified as c_v. However, if the pressure is held constant during the change in state, then the specific heat is identified as c_p. The ratio c_p/c_v is given the symbol *k*. Values for *cp* and *k* for various gases are given in Table A.2.

Internal Energy

The energy that a substance possesses because of the state of the molecular activity in the substance. Internal energy is usually expressed as a specific quantity—that is, internal energy per unit mass. the *specific internal energy, u,* is given in joules per kilogram.

Enthalpy

The combination *u* + *p*/ρ is encountered frequently in equations for thermodynamics and compressible flow; it has been given the name *specific enthalpy.* For an ideal gas, *u* and *p*/ρ are functions of temperature alone. Consequently their sum, specific enthalpy, is also a function solely of temperature.

2.4 Viscosity, μ

(also called *dynamic viscosity*, or *absolute viscosity*) is a measure of a fluid's resistance to deformation under shear stress.

Shear stress, τ, is the ratio of force/area on a surface when the force is aligned parallel to the area. Shear strain is a change in an interior angle of a cubical element, Δφ, that was originally a right angle. The shear stress on a material element in solid mechanics is proportional to the strain, and the constant of proportionality is the shear modulus:

Shear stress = shear modulus x strain

In fluid flow, however, the shear stress on a fluid element is proportional to the rate (speed) of strain, and the constant of proportionality is the viscosity:

Shear stress = Viscosity x rate of strain

$$\tau = \mu \frac{dV}{dy} => \mu = \frac{\tau}{dV/dy} = N.s/m^2$$

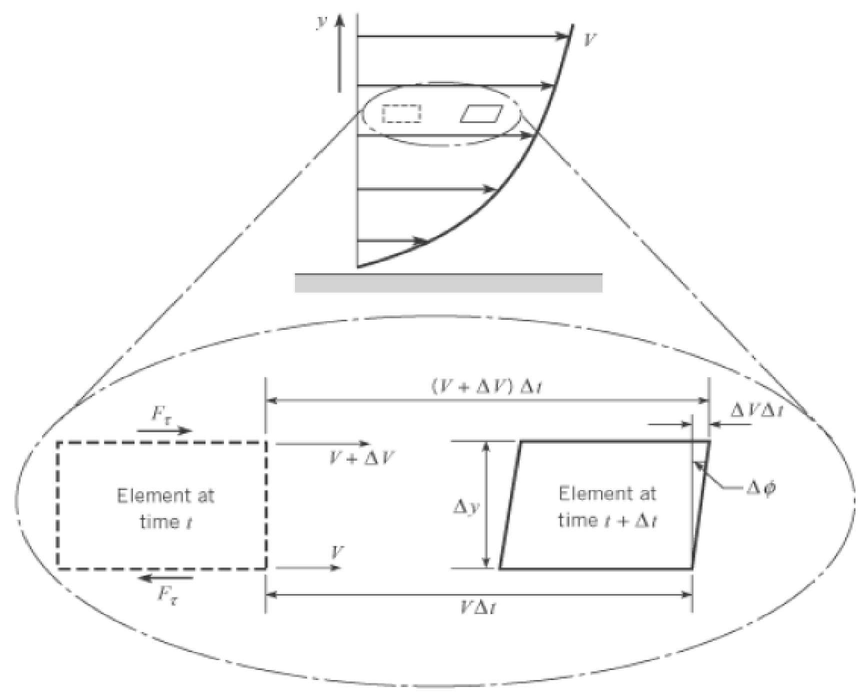
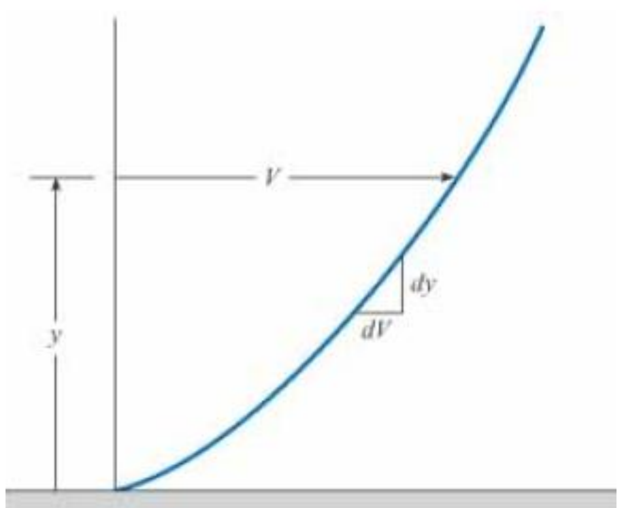


Figure 2.1 *Depiction of strain caused by a shear stress (force per area) in a fluid. The rate of strain is the rate of change of the interior angle of the original rectangle.*

Figure 2.1 depicts an initially rectangular element in a parallel flow field. As the element moves downstream, a shear force on the top of the element (and a corresponding shear stress in the opposite direction on the bottom of the element) causes the top surface to move faster (with velocity *V* + Δ*V*) than the bottom (at velocity *V*). The forward and rearward edges become inclined at an angle Δφ with respect to the vertical. The rate at which Δφ changes with time, given by φ, is the *rate of strain*, and can be related to the velocity difference between the two surfaces.

For strain in flow near a wall, as shown in Fig. 2.2, the term *dV/dy* represents the velocity gradient (or change of velocity with distance from the wall), where *V* is the fluid velocity and *y* is the distance measured from the wall. several observations:
First, the velocity gradient at the boundary is finite. The curve of velocity variation cannot be tangent to the boundary because this would imply an infinite velocity gradient and, in turn, an infinite shear stress, which is impossible. Second, a velocity gradient that becomes less steep (*dV/dy* becomes smaller) with distance from the boundary has a maximum shear stress at the boundary, and the shear stress decreases with distance from the boundary. Also note that the velocity of the fluid is zero at the stationary boundary. That is, at the boundary surface the fluid has the velocity of the boundary—no slip occurs between the fluid and the boundary. This is referred to as the *no-slip condition*.



Kinematic Viscosity, ν

$$\boldsymbol{\nu} = \mu/\rho = m^2/s$$

Temperature Dependency

The effect of temperature on viscosity is different for liquids and gases. The viscosity of liquids decreases as the temperature increases, whereas the viscosity of gases increases with increasing temperature; this trend is also true for kinematic viscosity.
An equation for the variation of liquid viscosity with temperature is

$$\mu = Ce^{b/T}$$

where *C* and *b* are empirical constants that require viscosity data at two temperatures for evaluation. This equation should be used primarily for data interpolation.

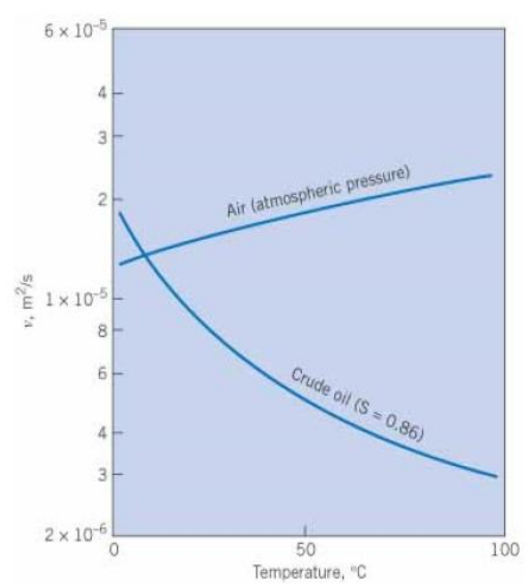


Figure 2.3 *Kinematic viscosity for air and crude oil.*

An estimate for the variation of gas viscosity with temperature is *Sutherland's equation*,

$$\frac{\mu}{\mu_0} = \left(\frac{T}{T_0}\right)^{3/2} \frac{T_0 + S}{T + S}$$

where μ0 is the viscosity at temperature *T*0, and *S* is Sutherland's constant. All temperatures are absolute. Sutherland's constant for air is 111 K; values for other gases are given in Table A.2. Using Sutherland's equation for air yields viscosities with an accuracy of ± 2% for temperatures between 170 K and 1900 K. In general, the effect of pressure on the viscosity of common gases is minimal for pressures less than 10 atmospheres.

Q2.37) Suppose that glycerin is flowing *(T* = 20°C) and that the pressure gradient *dp*/*dx* is -1.6kN/m^3. What are the velocity and shear stress at a distance of 12 mm from the wall if the space *B* between the walls is 5.0 cm? What are the shear stress and velocity at the wall? The velocity distribution for viscous flow between stationary plates is

$u = -\frac{1}{2\mu}\frac{dp}{dx}(By - y^2)$

Properties: Glycerin at 20 ◦C from Table A.4: μ = 1.41N.s/m^2.

a.) Velocity and shear stress (at y = 12mm)

velocity

$u = -\frac{1}{2(1.41)\text{N}.\frac{\text{s}}{\text{m}^2}}(-1600)\text{N/m}^3(0.05m \times\ 0.012m - (0.012)^2\text{m}^2)$ = 0.2587 m/s

Rate of strain

$$\frac{du}{dy} = \frac{d}{dy}\left[-\frac{1}{2\mu}\frac{dp}{dx}(By - y^2)\right] = -\frac{1}{2\mu}\frac{dp}{dx}(b - 2y)$$

$$\frac{du}{dy}(y = 0.012) = -\frac{1}{2(1.41)\text{N}.\frac{\text{s}}{\text{m}^2}}(-1600)\text{N/m}^3[0.05m - 2 \times 0.012\text{m}] = 14.75s^{-1}$$

Shear stress

$\tau = \mu\frac{du}{dy} = (1.41)\text{N}.\frac{\text{s}}{\text{m}^2} \times 14.75s^{-1} = 20.798\ Pa$

b.) Velocity and shear stress (at y = 0mm)

$$u = -\frac{1}{2(1.41)\text{N}.\frac{\text{s}}{\text{m}^2}}(-1600)\text{N/m}^3(0.05m \times\ 0\ m - (0)^2\text{m}^2) = 0.0m/s$$

Rate of strain @ y=0

$$\frac{du}{dy}(y = 0) = -\frac{1}{2(1.41)\text{N}.\frac{\text{s}}{\text{m}^2}}(-1600)\text{N/m}^3[0.05m - 2 \times 0.0\text{ m}] = 28.37s^{-1}$$

Shear stress

$\tau = \mu\frac{du}{dy} = (1.41)\text{N}.\frac{\text{s}}{\text{m}^2} \times 28.37s^{-1} = 40.0\ Pa$

Newtonian Versus Non-Newtonian Fluids

Fluids for which the shear stress is directly proportional to the rate of strain are called *Newtonian fluids.* Because shear stress is directly proportional to the shear strain, *dV/dy*, a plot relating these variables results in a straight line passing through the origin. The slope of this line is the value of the dynamic (absolute) viscosity.

For some fluids the shear stress may not be directly proportional to the rate of strain; these are called *non-Newtonian fluids*.

1. Shear-thinning fluids, has the interesting property that the ratio of shear stress to shear strain decreases as the shear strain increases. Some common shear-thinning fluids are toothpaste, catsup, paints, and printer's ink.
2. Shear-thickening fluids, the viscosity increases with shear rate. Some examples of these fluids are mixtures of glass particles in water and gypsum-water mixtures.
3. Bingham plastic, acts like a solid for small values of shear stress and then behaves as a fluid at higher shear stress.

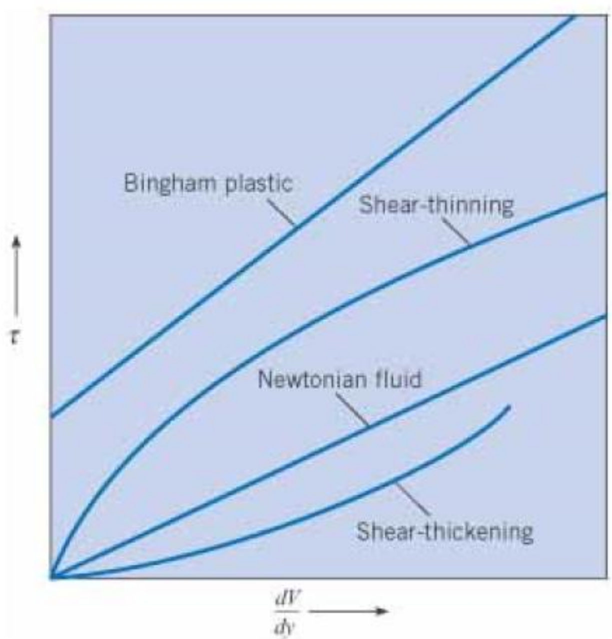


Figure 2.6 *Shear stress relations for different types of fluids.*

Bulk Modulus of Elasticity

The *bulk modulus of elasticity*, E_v, is a property that relates changes in pressure to changes in volume (e.g., expansion or contraction)

$$E_v = -\frac{dp}{d\forall/\forall} = -\frac{change\ in\ pressure}{fractional\ change\ in\ Volume}$$

where: *dp* is the differential pressure change, $d\forall$ is the differential volume change, and $\forall$ is the volume of fluid.

Negative sign is used in the definition to yield a positive E_v since $d\forall/\forall$ is negative for positive dp.

$$m = \rho \forall$$

$dm = \rho d\forall + \forall d\rho = 0 \Rightarrow \rho d\forall = -\forall d\rho$ ➔ $-d\forall/\forall = d\rho/\rho$

$$E_v = \frac{dp}{d\rho/\rho} = \frac{change\ in\ pressure}{fractional\ change\ in\ density}$$

The bulk modulus of elasticity of water is approximately 2.2 GN/m2, which corresponds to a 0.05% change in volume for a change of 1 MN/m2 in pressure. Obviously, the term *incompressible* is justifiably applied to water because it has such a small change in volume for a very large change in pressure.

The elasticity of an ideal gas is proportional to the pressure, according to the ideal gas law. For an isothermal (constant-temperature) process

$\frac{dp}{d\rho} = RT$ hence $E_v = \rho \frac{dp}{d\rho} = \rho RT = P$

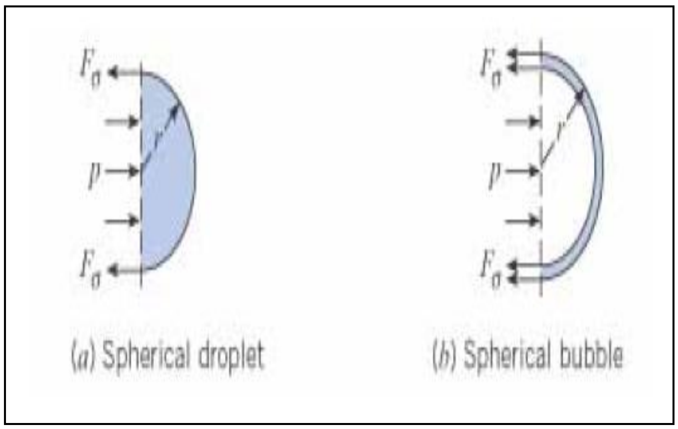
For an adiabatic process "$\frac{P}{\rho^k} = constant$", $E_v = kp$, where k is the ratio of specific heats, c_P/c_v.

Surface Tension σ

Surface tension, σ (sigma), is a material property whereby a liquid at a material interface, usually liquid-gas, exerts a force per unit length along the surface.

Surface tension is the result of molecular attraction near a free surface, causing the surface to act like a stretched membrane.

$F_\sigma = \sigma L$, where L is the length over which the surface tension acts.



Vapor Pressure

The pressure at which a liquid will vaporize, or boil, at a given temperature, is called its *vapor pressure*. This means that boiling occurs whenever the local pressure equals the vapor pressure. Vapor pressure increases with temperature. Note that there are two ways to boil a liquid. One way is to raise the temperature, assuming that the pressure is fixed. @ 1 atm (p= 101.325 KPa), [p=14.7psi]water boils at 100 °C [212°F] thus reaching the temperature where the vapor pressure is equal to the same value.

Other way is to reduce pressure, this causes the water to boil at temperature much less than 100 °C [212°F]. For example, the vapor pressure of water at 50°F (10°C) is 0.178 psia (approximately 1% of standard atmospheric pressure). Therefore, if the pressure in water at 50°F is reduced to 0.178 psia, the water boils. Such boiling often occurs in localized low-pressure zones of flowing liquids, such as on the suction side of a pump. When localized low-pressure boiling does occur in flowing liquids, vapor bubbles start growing in local regions of very low pressure and then collapse in regions of higher pressure downstream. This phenomenon, which is called *cavitation*, can cause extensive damage to fluids systems.

Table A.5 gives values of vapor pressure for water.

Chapter 3

Fluid Statics

3.1 Pressure

- *Pressure* : The ratio of normal force to area at a point.

$$p = \lim_{\Delta A \to 0} \frac{\left|\overrightarrow{\Delta F}_{normal}\right|}{\Delta A} = \frac{d\vec{F}_{normal}}{dA}$$

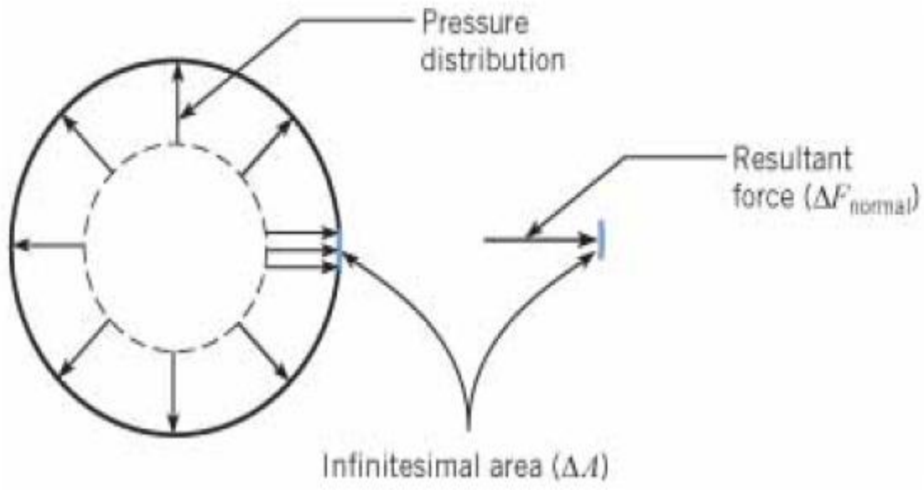


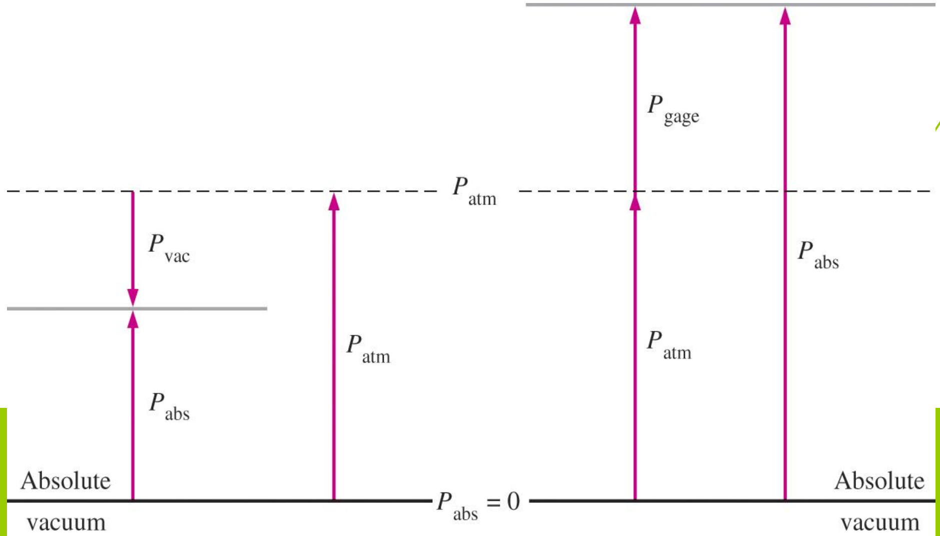
Figure 3.2 *Pressure acting on the walls of a sphere.*

- Pressure often varies from point to point.
- Pressure is a scalar quantity; it has magnitude only

- It produces a resultant force by its action on an area. The resultant force is normal to the area and acts in a direction toward the surface "Compression"
- Units: SI units: Newton/m^2=Pascal (Pa)
- Standard atmospheric pressure, which is the air pressure at sea level, can be written using multiple units: 1.0 atm =101.3 KPa = 14.7 Psi = 33.9 ft-H_2O = 760 mm-Hg = 29.92 in-Hg = 2116 Psf

Absolute Pressure, Gage Pressure, and Vacuum Pressure

- The pressure in a perfect vacuum :<u>absolute zero</u>
- Pressure measured relative to this zero pressure is termed *absolute pressure*
- *gage pressure* : measured relative to local atmospheric pressure
- *Vacuum pressure :* When pressure is less than atmospheric
- $P_{abs} = P_{atm} + P_{gage}$
- $P_{abs} = P_{atm} – P_{vacuum}$
- $P_{vacuum} = - P_{gage}$



Hydraulic Machines

- A *hydraulic machine* uses components such as pistons, pumps, and hoses to transmit forces and energy using fluids.
- braking systems, forklift trucks, power steering systems, and airplane control systems.
- Pascals Law: pressure applied to an enclosed and continuous body of fluid is transmitted undiminished “غير منقوص” to every portion of that fluid and to the walls of the containing vessel.

- F=100N , find F2

$$\sum M_c = 0$$

$$(0.33\ m)(100N) - (0.03m)F_1 = 0$$

$$F_1 = \frac{(0.33\ m)(100N)}{(0.03m)} = 1100N$$

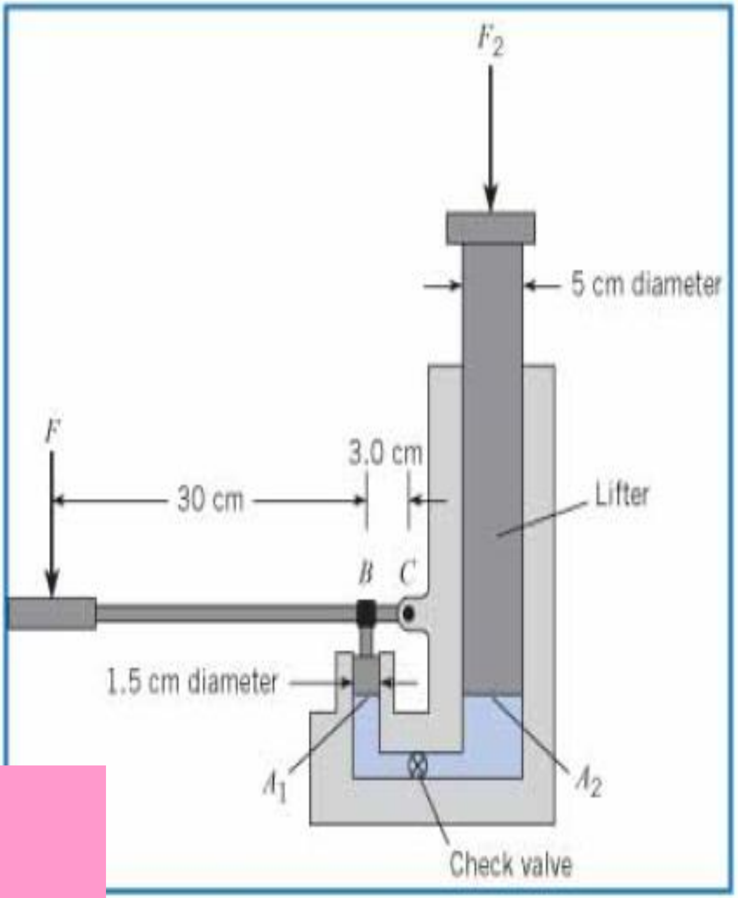
$$\sum F_{small\ piston} = P_1 A_1 - F_1 = 0$$

$$P_1 A_1 = F_1 = 1100\ N$$

$$\Rightarrow P_1 = \frac{F_1}{A_1} = \frac{F_1}{\pi d_1^2 / 4} = \frac{1100}{\pi (0.015)^2 / 4} = 6.22 \times 10^6 Pa$$

$$\sum F_{lifter} = F_2 - P_2 A_2 = 0,\ P_2 = P_1,\ A_2 = \frac{\pi d_2^2}{4},\quad d_2 = 0.05m$$

$$F_2 = P_1 A_2 = (6.22 \times 10^6 Pa) \times \frac{\pi (0.05)^2}{4} m^2 = 12.2KN$$

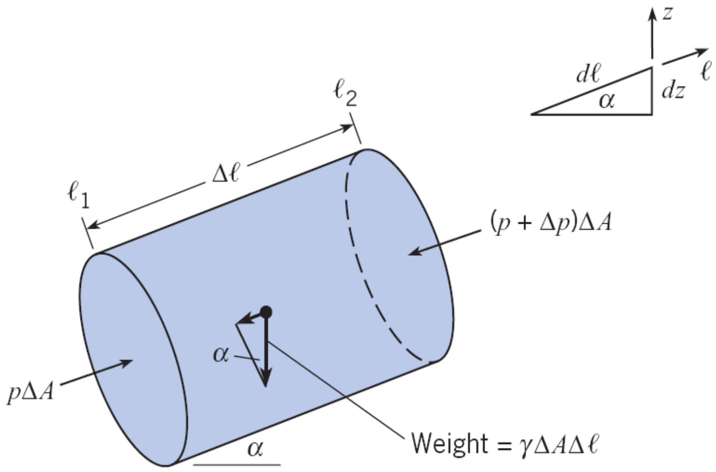


3.2 Pressure Variation with Elevation
Hydrostatic Differential Equation

$$\sum F_l = 0 \Rightarrow F_{pressure} - F_{weight} = 0$$

$$p\Delta A - (p + \Delta p)\Delta A - \gamma \Delta l\, \Delta A \sin\alpha = 0$$

$$-\Delta p \Delta A = \gamma \Delta l\, \Delta A \sin\alpha$$



Divide the volume '$\Delta l \Delta A$' to get

$$\frac{\Delta p}{\Delta l} = -\gamma \sin\alpha$$

$$\text{also } \sin\alpha = \frac{\Delta z}{\Delta l}$$

$$\lim_{\Delta z \to 0} \frac{\Delta p}{\Delta z} = -\gamma \Rightarrow \frac{dp}{dz} = -\gamma$$

Hydrostatic Equation

- The hydrostatic equation is used to predict pressure variation in a fluid with constant density

integrate the differential equation $p + \gamma z = p_z = \text{constant}$

- where the term *z* is elevation, which is the height (vertical distance) above a fixed reference point called a datum, and is *piezometric pressure*.
- Dividing by γ gives ➔ ➔➔ $\frac{p_z}{\gamma} = \left(\frac{p}{\gamma} + z\right) = h$

h is the *piezometric head*.

Since *h* is constant

$$\left(\frac{p_1}{\gamma} + z_1\right) = \left(\frac{p_2}{\gamma} + z_2\right)$$

or $\quad p_1 + \gamma z_1 = p_2 + \gamma z_2$

Problem 3.11) For the closed tank with Bourdon-tube gages tapped into it, what is the specific gravity of the oil and the pressure reading on gage *C*?

ANALYSIS

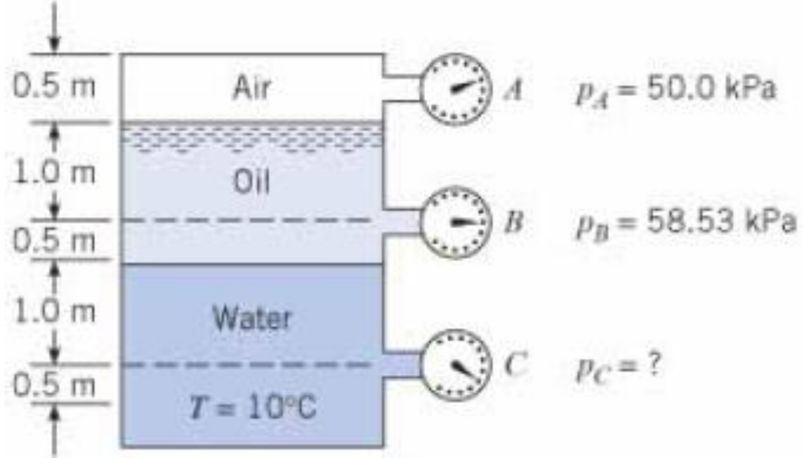
Hydrostatic equation (from oil surface to elevation B)

$$\begin{aligned} p_A + \gamma z_A &= p_B + \gamma z_B \\ 50,000 \text{ N/m}^3 + \gamma_{\text{oil}}(1 \text{ m}) &= 58,530 \text{ N/m}^3 + \gamma_{\text{oil}}(0 \text{ m}) \\ \gamma_{\text{oil}} &= 8530 \text{ N/m}^3 \end{aligned}$$

Specific gravity

$$S = \frac{\gamma_{\text{oil}}}{\gamma_{\text{water}}} = \frac{8530 \text{ N/m}^3}{9810 \text{ N/m}^3}$$

$$\boxed{S_{\text{oil}} = 0.87}$$



PROBLEM 3.11

Hydrostatic equation (in oil)

$$p_{\text{btm of oil}} = (58,530 \text{ Pa} + \gamma_{\text{oil}} \times 0.5 \text{ m})$$

Hydrostatic equation (in water)

$$p_c = (p_{\text{btm of oil}}) + \gamma_{\text{water}}(1 \text{ m})$$

Hydrostatic equation (in oil)

$$p_{\text{btm of oil}} = (58,530 \text{ Pa} + \gamma_{\text{oil}} \times 0.5 \text{ m})$$

Combine equations

$$\begin{aligned} p_c &= (58,530 \text{ Pa} + \gamma_{\text{oil}} \times 0.5 \text{ m}) + \gamma_{\text{water}}(1 \text{ m}) \\ &= (58,530 + 8530 \times 0.5) + 9810(1) \\ &= 72,605 \text{ N/m}^2 \end{aligned}$$

$$\boxed{p_c = 72.6 \text{ kPa}}$$

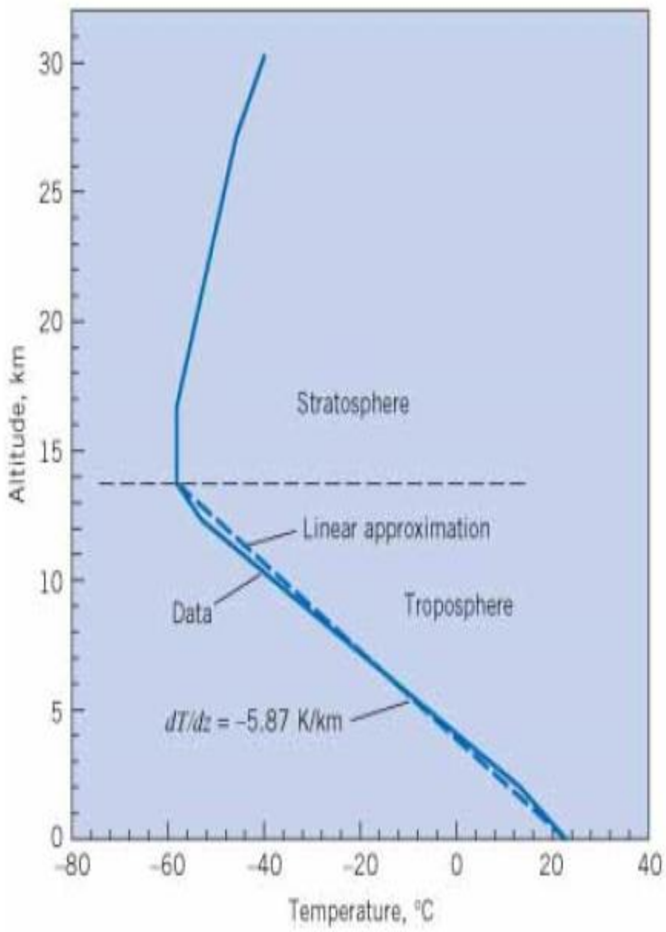
Pressure Variation in the Atmosphere

The ideal gas law $\rho = (p / RT)$

$$\gamma = \rho g = \frac{pg}{RT}$$

The last equation requires Temp-vs-elevation data for the atmosphere

Troposphere height =13.7km



Temperature variation with altitude for the U.S. standard atmosphere in July 1.

Pressure Variation in the Troposphere

Linear Temperature profile approximation

$$T = T_0 - \alpha(z - z_0)$$

T_0 **: T at a reference level where the pressure is known**

α **: the lapse rate.**

$$\frac{\mathrm{dp}}{\mathrm{dz}} = -\gamma = -\frac{pg}{RT}$$

Substitute for T

$$\frac{\mathrm{dp}}{\mathrm{dz}} = -\frac{pg}{R[T_0 - \alpha(z - z_0)]}$$

$$\frac{\mathrm{p}}{p_0} = \left[\frac{T_0 - \alpha(z - z_0)}{T_0}\right]^{g/\alpha R}$$

or

$$\mathrm{p} = p_0\left[\frac{T_0 - \alpha(z - z_0)}{T_0}\right]^{g/\alpha R}$$

Pressure Variation in the Lower Stratosphere

In the lower part of the stratosphere (13.7 to 16.8 km above the earth's surface) temperature is approximately constant

$$\frac{\mathrm{dp}}{\mathrm{dz}} = -\gamma = -\frac{pg}{RT}$$

$$\ln \mathrm{p} = -\frac{zg}{RT} + C$$

$$\textbf{At } \boldsymbol{z} = \boldsymbol{z}_0 \ , \ \boldsymbol{p} = \boldsymbol{p}_0$$

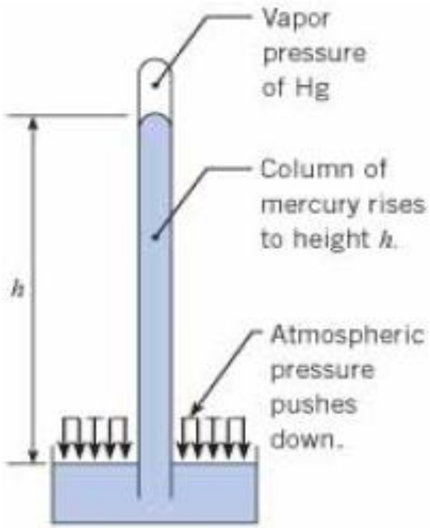
$$\frac{\mathrm{p}}{p_0} = e^{(z-z_0)g/RT}$$

$$or \qquad \boldsymbol{p} = p_0 e^{(z-z_0)g/RT}$$

3.3 Pressure Measurements

- **Barometer:**

Mercury Barometer



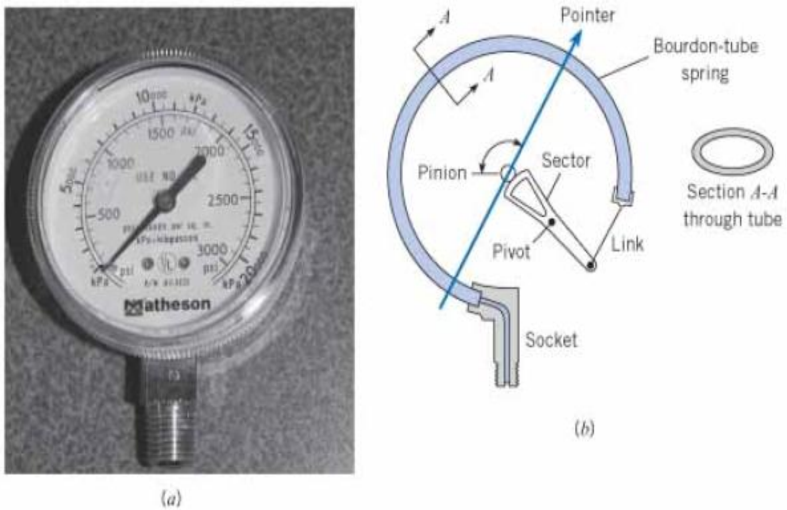
A mercury barometer.

$$p_{atm} = \gamma_{Hg} h + p_v \approx \gamma_{Hg} h$$

$p_v = 2.4 \times 10^{-6}$ atm at 20°C

Bourdon-Tube Gage

pressure by sensing the deflection of a coiled tube

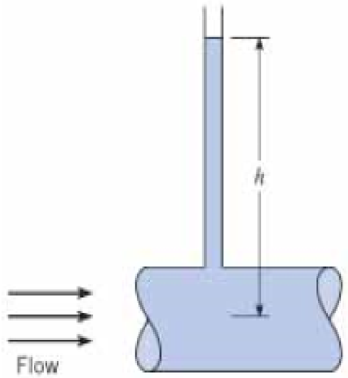


Bourdon-tube gage.
(a) View of typical gage. (b) Internal mechanism (schematic).

3.3 Pressure Measurements

Piezometer

$$p = \gamma h$$



Piexometer attached to a pipe.

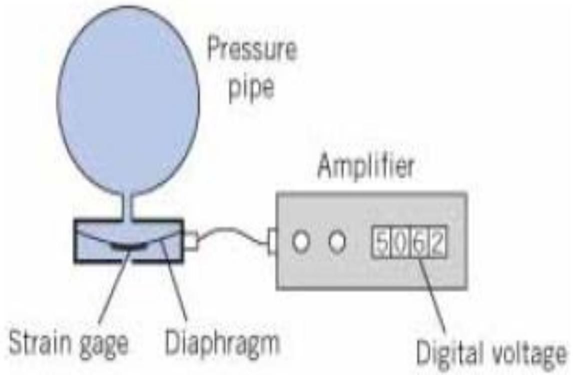
vertical tube-transparent- in which a liquid rises in response to a positive gage pressure.

Simplicity, direct measurement (no need for calibration) & accuracy.

Limited to low pressure ,not easy measure   Gas Pressure

Pressure transducers

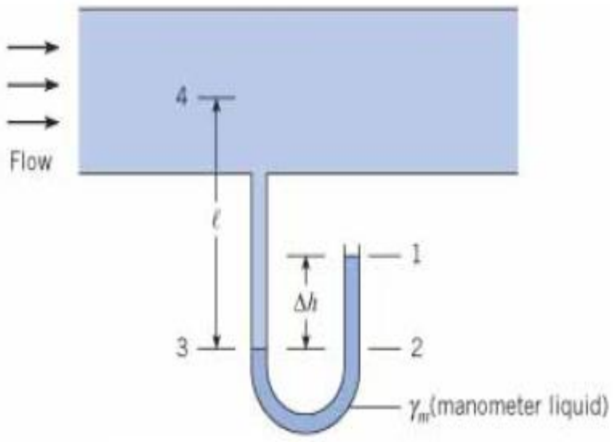
They convert pressure to an electrical signal



Schematic diagram of strain-gage pressure transducer.

3.3 Pressure Measurements

Manometer



U-tube manometer.

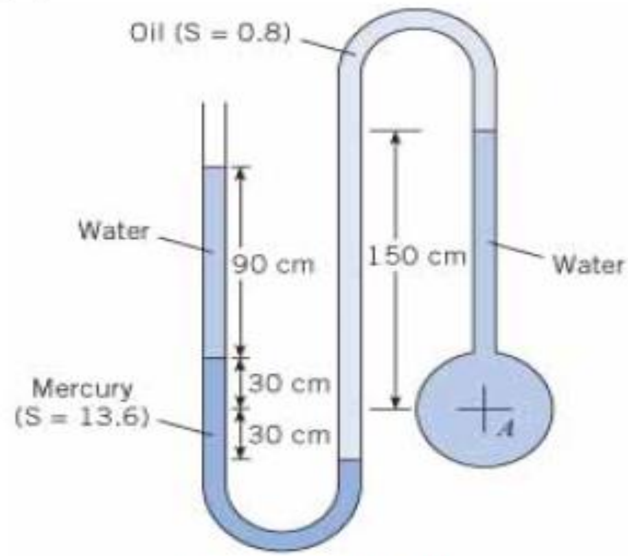
$$p_2 = p_1 + \sum_{down} \gamma_i h_i - \sum_{up} \gamma_i h_i$$

γ_i **the specific weight**

h_i **deflection "Height"**

P3.39

Find the pressure at the center of pipe *A*.



PROBLEM 3.39

$$p_A = p_B + \sum_{down} \gamma_i h_i - \sum_{up} \gamma_i h_i$$

$$p_A = 0 + (0.9 + 0.6 * 13.6 - 0.8 * 1.8 + 1.5)9810$$

$$p_A = 89.47 \ kPa \ gage$$

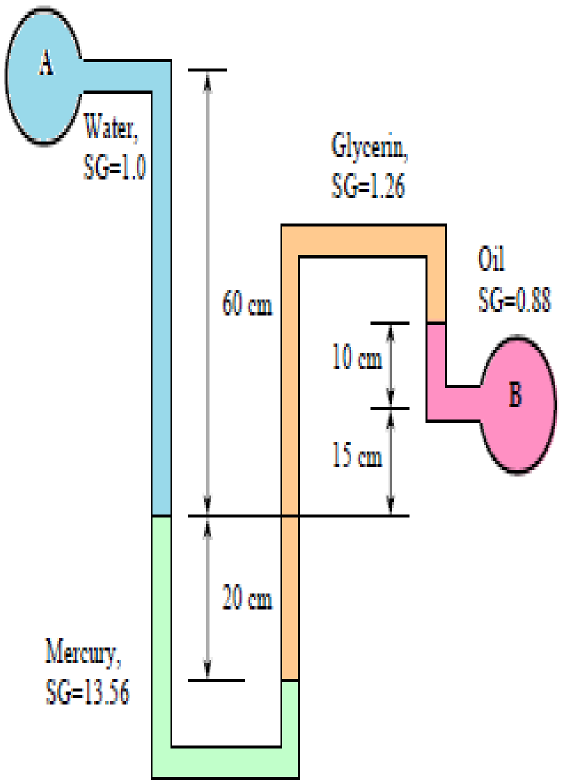
Calculate the pressure difference between the pipes

Note : SG: specific Gravity "S"

$$P_A + (\rho gh)_w + (\rho gh)_{Hg} - (\rho gh)_{gly} + (\rho gh)_{oil} = P_B$$

$$P_B - P_A = (\rho gh)_w + (\rho g)_w (S\,h)_{Hg} - (\rho g)_w (S\,h)_{gly} + (\rho g)_w (S\,h)_{oil}$$

$$P_B - P_A = (\rho g)_w [(S\,h)_{w} + (S\,h)_{Hg} - (S\,h)_{gly} + (S\,h)_{oil}]$$



$$P_B - P_A = (9.81 \text{m/s})(1000 \text{ kg/m})[1(0.6 \text{ m}) + 13.5(0.2\text{m}) - 1.26(0.45\text{m}) + 0.88(0.1\text{m})]$$

$$P_B - P_A = 27700 \text{ N/m}^2$$

$$P_B - P_A = 27.7 \text{ kN/m}^2 = 27.7 \text{ kPa}$$

3.4 Forces on Plane Surfaces (Panels)

Uniform Pressure Distribution

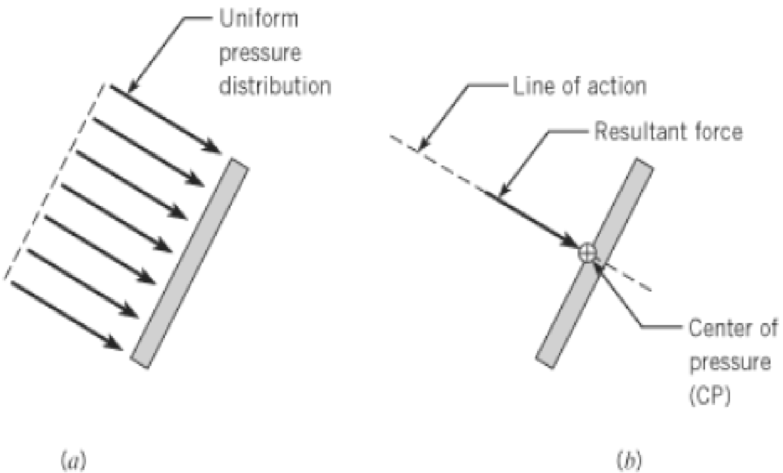


Figure 3.14 *(a) Uniform pressure distribution, and (b) equivalent force.*

$$F = \int_A p\,dA = \bar{p}A$$

Hydrostatic Pressure Distribution

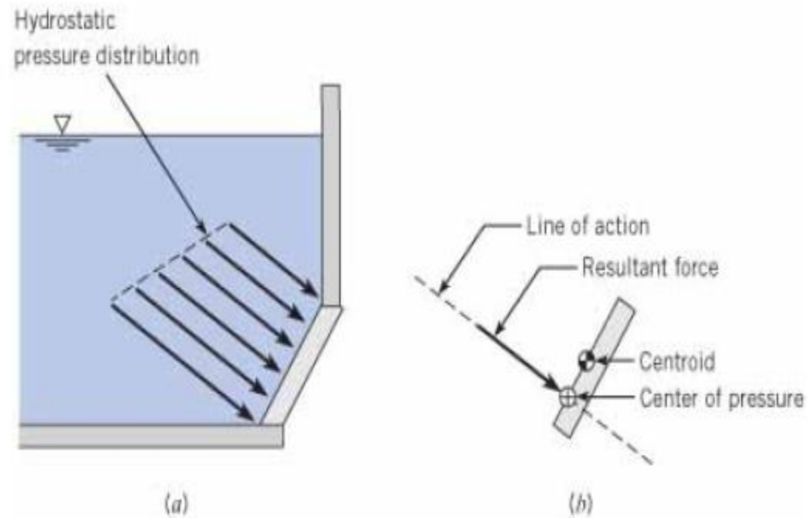
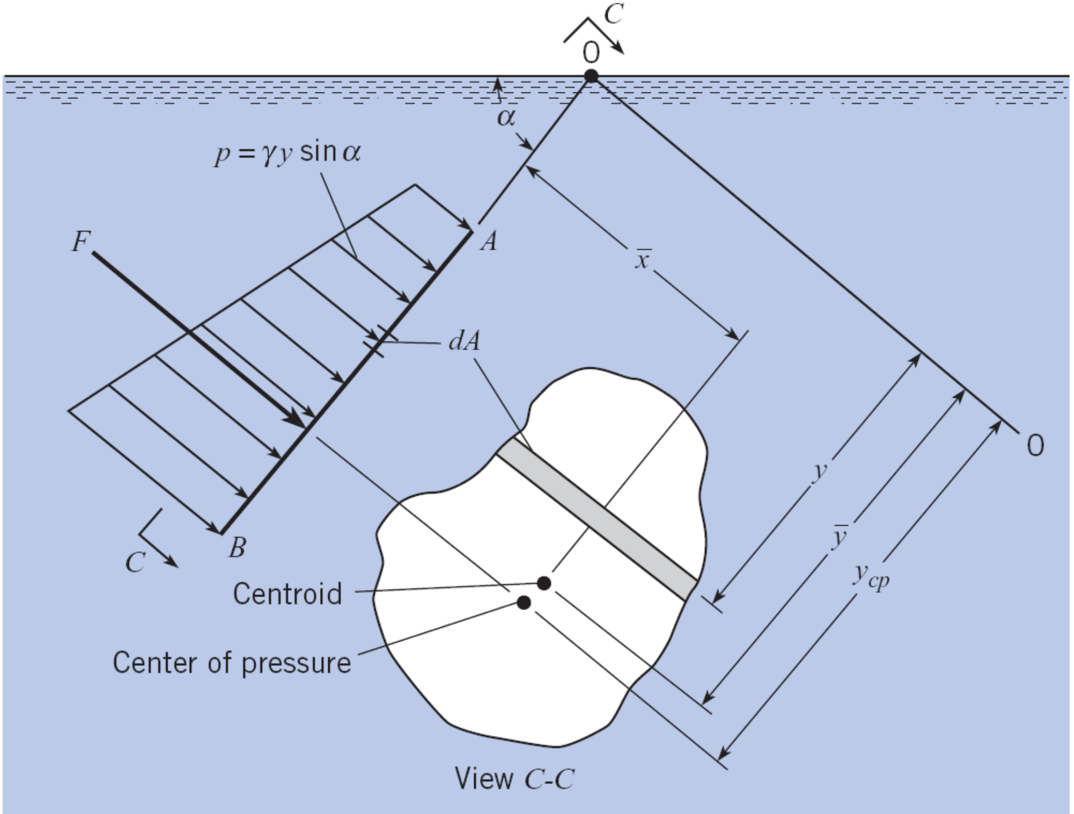


Figure 3.15 *(a) Hydrostatic pressure distribution, and (b) resultant force F acting at the center of pressure.*

Distribution of hydrostatic pressure on a plane surface



Pressure on the differential area can be computed if the y distance to the point is known

dF = p dA = (γ y sinα) dA

Integrating the differential force over the entire area A

$$F = \gamma\, Sin\alpha \int_A y\, dA = \gamma\, Sin\alpha\, \bar{y}\, A$$

Integral is the first moment of the area

Hydrostatic Force

$$F = \gamma\, \bar{y}\, Sin\,\alpha A = \bar{p}\, A$$

Pressure at the centroid

Line of Action of the Resultant Force

The torque due to the resultant force *F* will balance the torque due to the pressure distribution

$$y_{cp}F = \int y\, dF,$$ $dF = p\, dA$ ➔ $$y_{cp}F = \int_A yp\, dA$$ also $$p = \gamma\, y \sin\alpha$$

$$y_{cp}F = \int_A (y)(\gamma\, y \sin\alpha)dA = \int_A (\gamma\, y^2 \sin\alpha)dA = \gamma \sin\alpha \int_A y^2 dA$$

$$I = \int_A y^2 dA \text{ "the second moment of the area"}$$

$$I_0 = \bar{I} + \bar{y}^2 A$$

$$y_{cp}F = \gamma \sin\alpha(\bar{I} + \bar{y}^2 A) \text{ also } F = \left(\gamma \bar{y} \sin\alpha\right) A$$

$$y_{cp} = \bar{y} + \frac{\bar{I}}{\bar{y} A}$$

$$y_{cp} - \bar{y} = \frac{\bar{I}}{\bar{y} A}$$

Hydrostatic Force

Hydrostatic Force Terms

h: Vertical distance from centroid to the water surface

(This distance determines the pressure at the centroid)

$$\bar{p} = \gamma h = \gamma \bar{y} \sin\alpha$$

$\bar{y}$: Inclined distance from water surface to the centroid

y_{cp}: Inclined distance from water surface to *centre of pressure*

$\bar{p}$: The pressure at the centroid

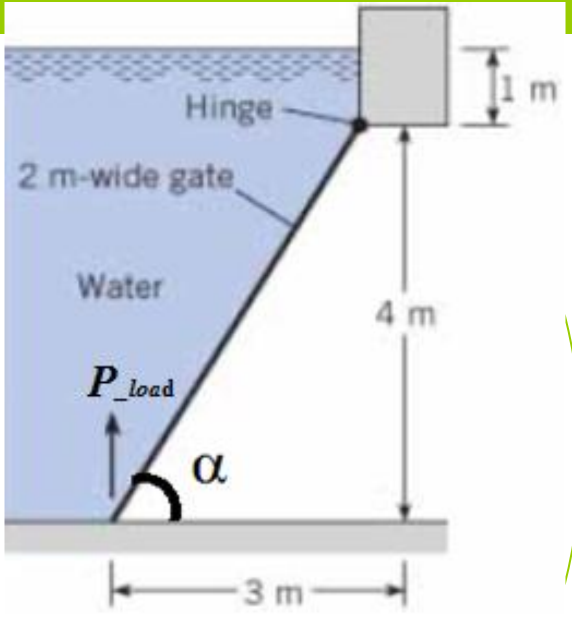
Problem 3.63 Determine *P_load* necessary to just start opening the 2 m–wide gate.

The length of the gate = $\sqrt{(3)^2+(4)^2} = 5\ m$

Hydrostatic force $F = \bar{p}\,A = \gamma \bar{h}\,A$

h:the water VERTICAL depth at the centroid ***h=2+1=3***

$$F = (3m)\left(9810\frac{N}{m^3}\right)(5m\times 2m) = 294.3kN$$



PROBLEM 3.63

$y_{cp} - \bar{y} = \dfrac{\bar{I}}{\bar{y}A}$ **where** $\bar{y} = \mathbf{h/sin}\alpha$ **: slanted distance from the surface**

$$y_{cp} - \bar{y} = \frac{(2\times 5^3)/12}{(3.75)(2\times 5\ )} = 0.5556\ \ m$$

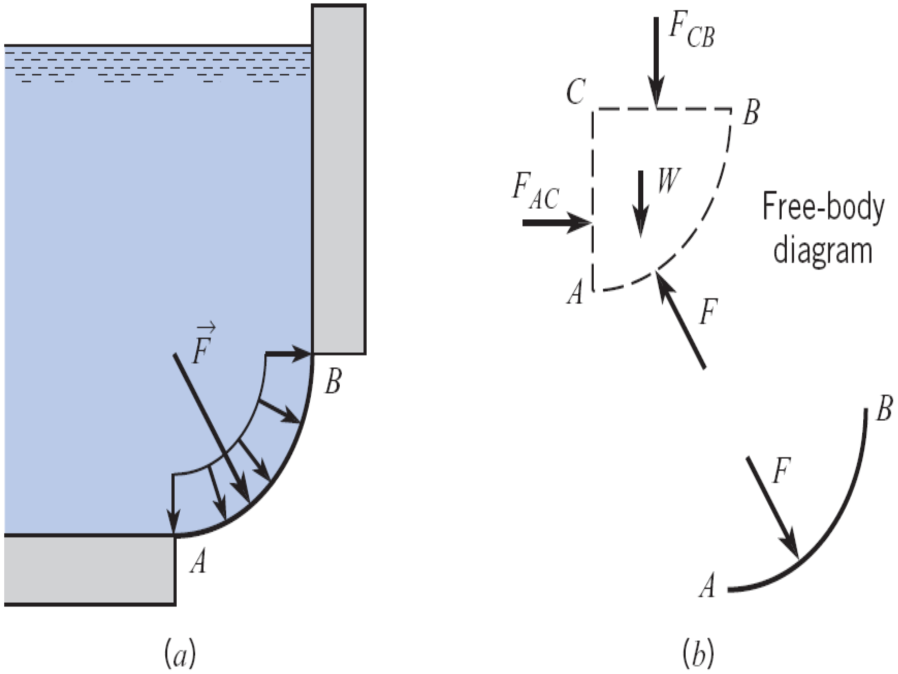
$$\sum M_{\text{hinge}} = 0$$

$$294.3\times(2.5+0.5556) - 3P\_laod = 0 \Rightarrow P\_laod = 300\boldsymbol{kN}$$

Force F is ⊥ to surface
Resultant force F acts at the center of pressure y_{cp}
y_{cp} and ȳ slanted distance from surface

Hydrostatic Forces on Curved Surfaces

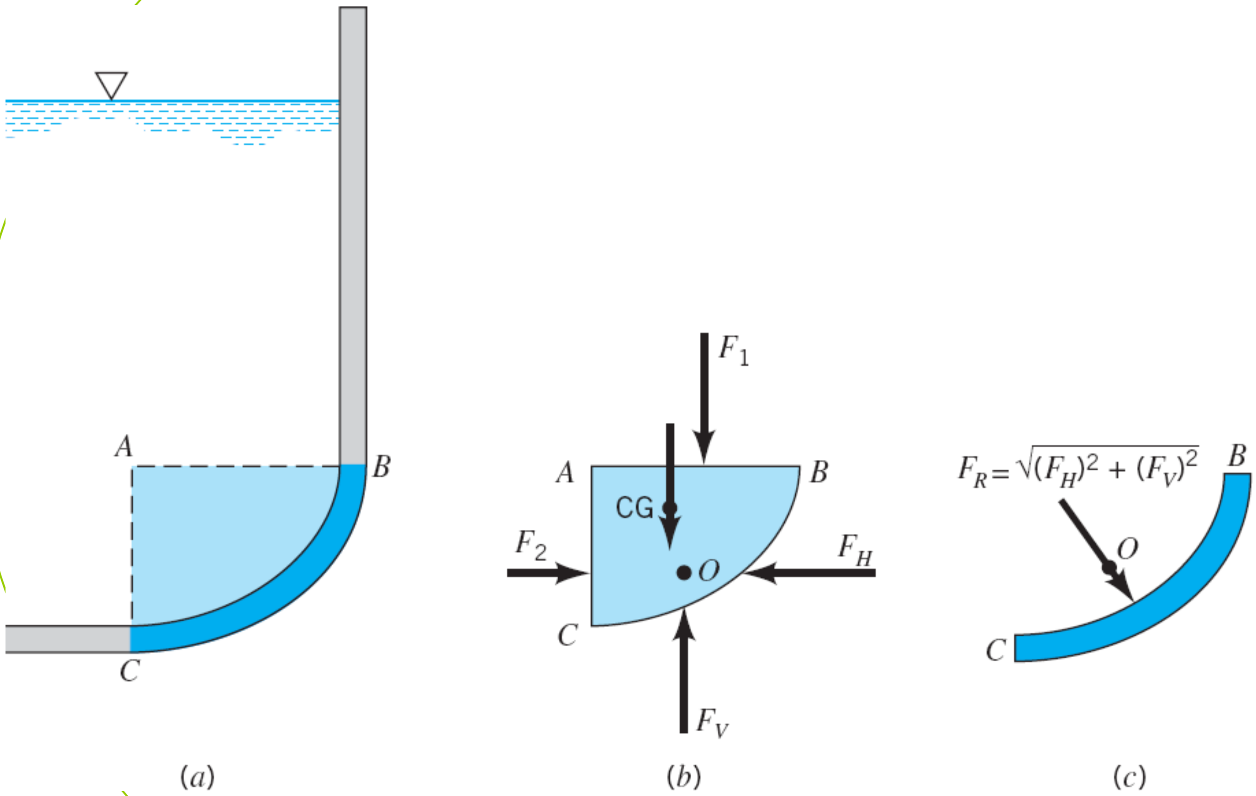
Find the magnitude and line of action of the hydrostatic force acting on surface AB

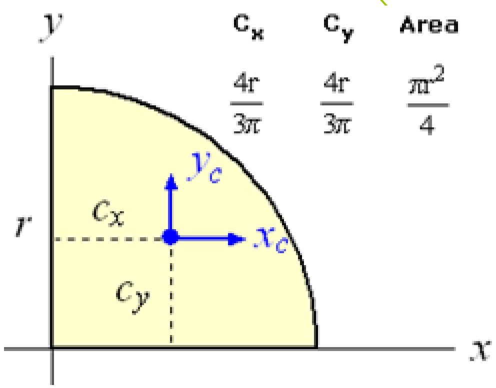


Important Questions to Ask

1. What is the shape of the curve?
2. How deep is the curved surface?
3. Where does the curve intersect straight surfaces?
4. What is the radius of the curve?

Hydrostatic Forces on Curved Surfaces



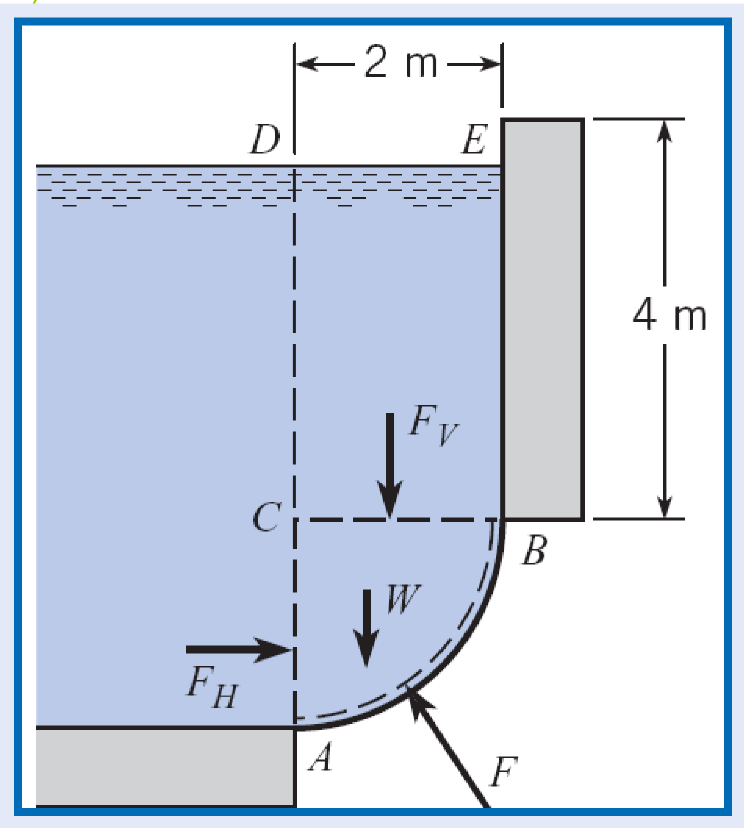


The centroid of the quadrant

A free-body diagram of a suitable volume of fluid can be used to determine the resultant force acting on a curved surface.

Hydrostatic forces on Curved surfaces.

Find the magnitude and line of action of the hydrostatic force acting on surface AB



Forces acting on the fluid element

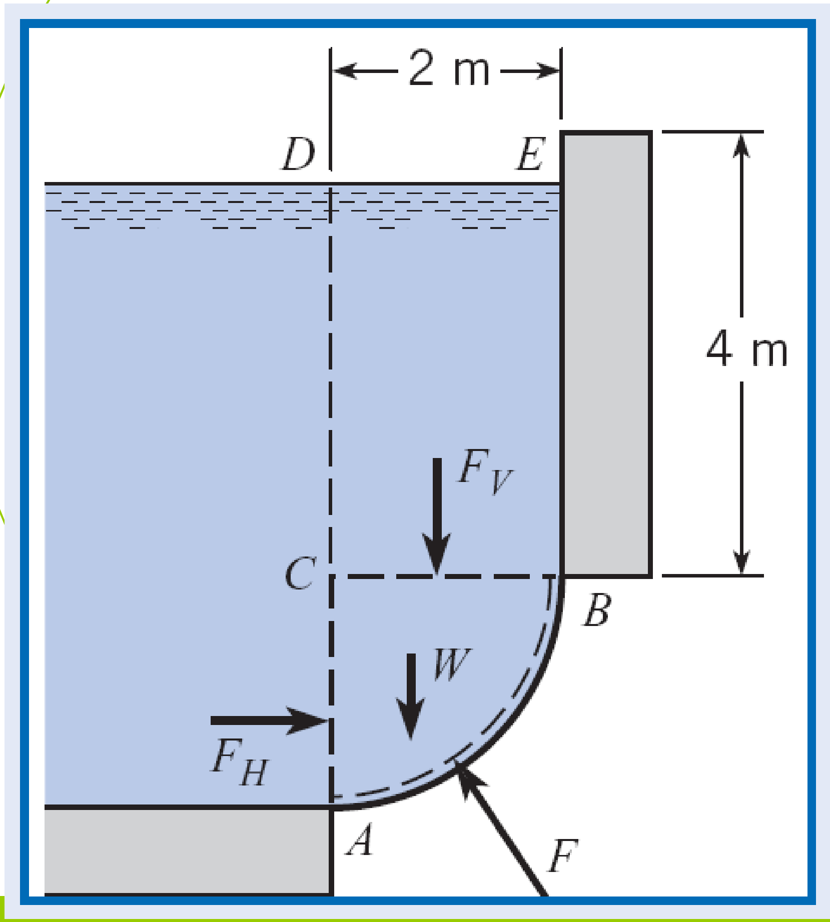
1. **F_V** : Force on the fluid element due to the weight of water above C.
2. **F_H** : Force on the fluid element due to horizontal hydrostatic forces on AC
3. **W** : Weight of the water in fluid element ABC
4. **F** : The force that counters all other forces

- F has a horizontal component: Fx

- F has a vertical component: Fy

Hydrostatic forces on Curved surfaces

Find the magnitude and line of action of the hydrostatic force acting on surface AB , - Given: Surface AB with a width of 1 m

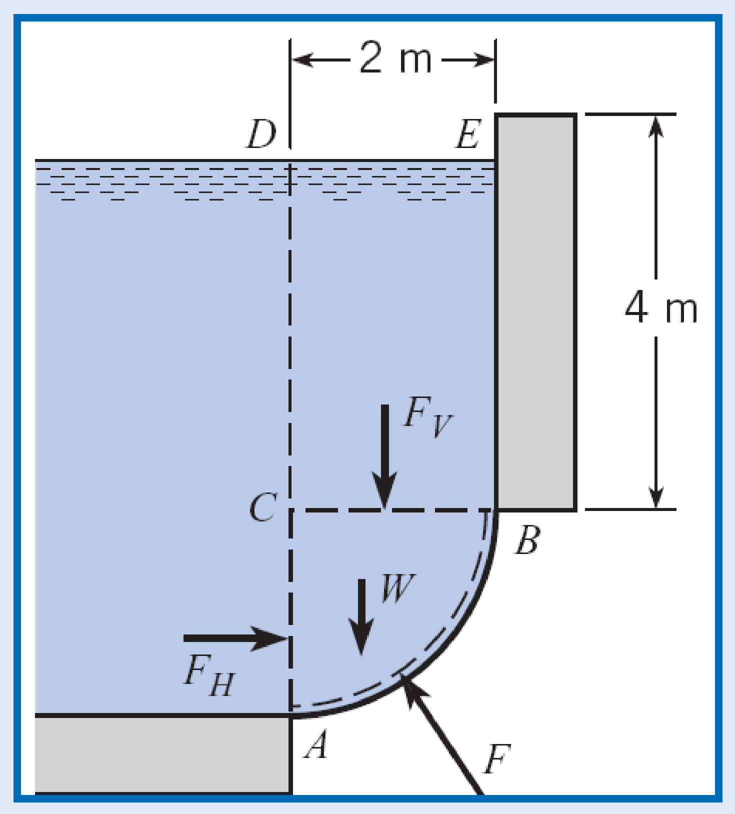


Problem Solving Preparation

1. By inspection, curve is a ¼ circle.
2. The depth to the beginning of the curve (4 m depth to B)
3. The curve radius (2 m horizontal curve projection distance = curve radius)
4. Label relevant points:
   - BCDE is water above fluid element defined by the curve
   - ABC is the fluid element defined by the curve

Example 3.11: Hydrostatic forces on Curved surfaces

Find F_v, F_H, W, F_x, F_y, F, Line of action for F_H & F_v
Given: Surface AB goes 1 m into the paper



F_x= F_H = (5 x 9810) (2 x 1) = 98.1 kN
Pres. at the cenroid AC side area

F_y= W + F_v

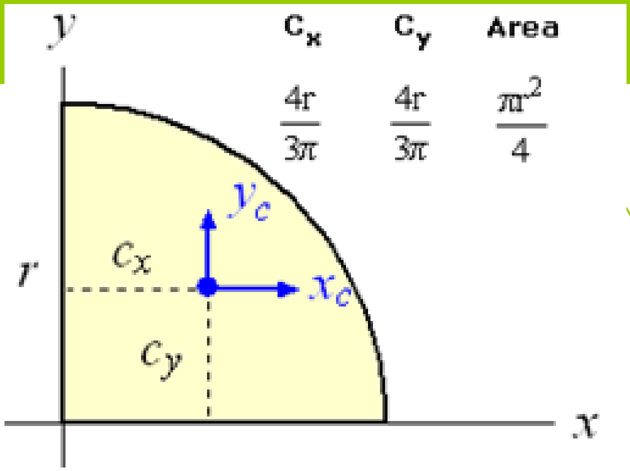
F_v= 9810 x 4 x 2 x 1 = 78.5 kN

W= γV_{ABC}= 9810 (1/4 x π r2) 1 = 30.8 kN

Fy= 78.5 + 30.8 = 109.3 kN

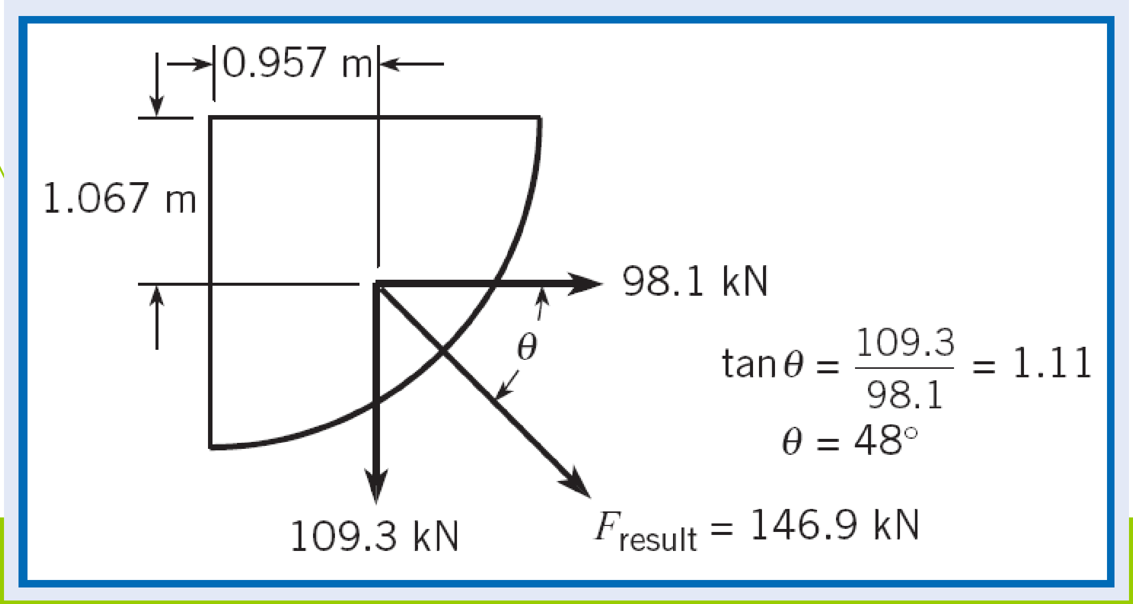
The hydrostatic force acting on AB is equal and opposite to the force *F shown*

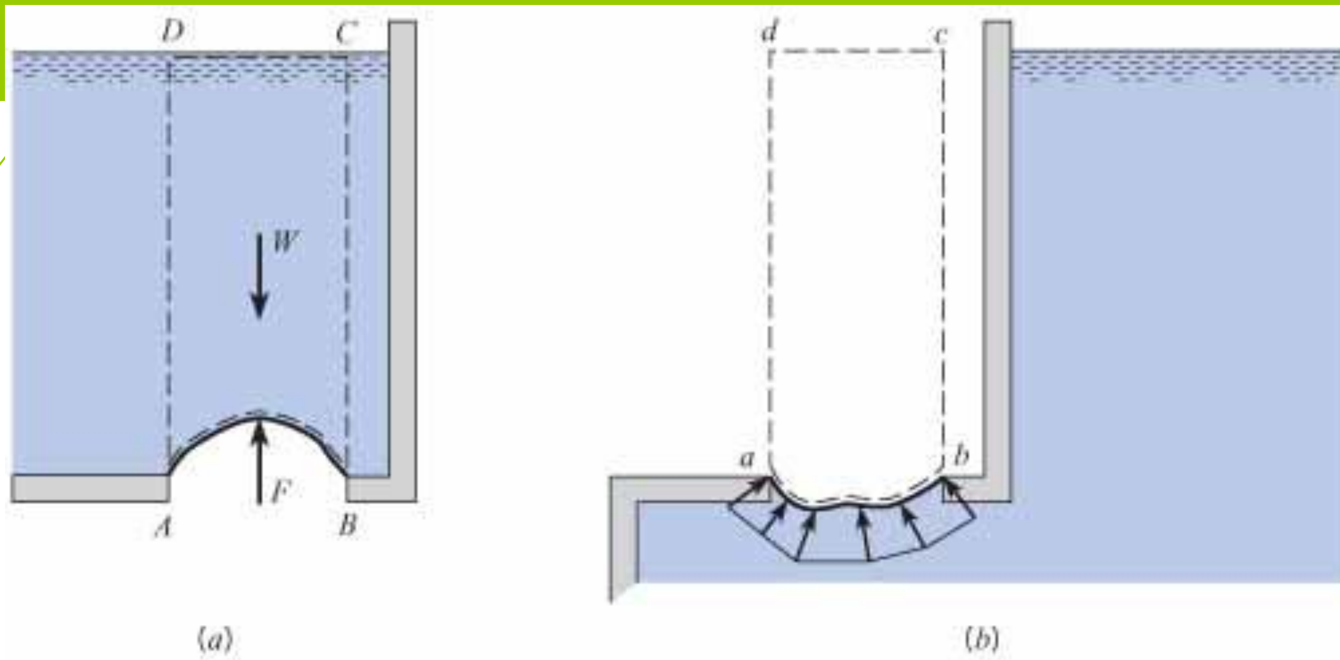
The centroid of the quadrant



The centroid of the quadrant

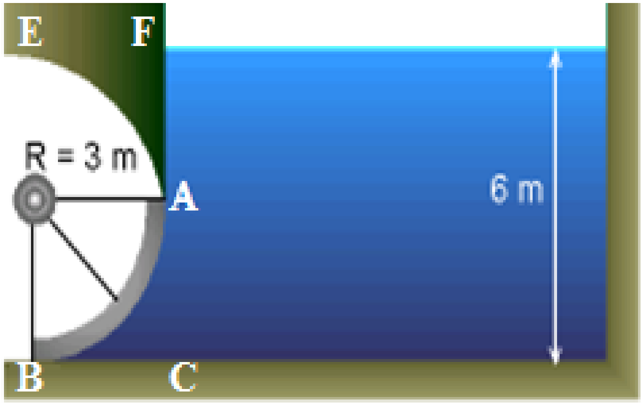
Location of the resultant force





If the region above the surface, volume *abcd, were filled with the same* liquid, the pressure acting at each point on the upper surface of *ab would equal the pressure acting at each point* on the lower surface. In other words, there would be no net force on the surface

The arc has a radius of 3 meters and the water level is at 6 meters. The spillway gate is 8 meters wide.
What is the magnitude and the line of action of the resultant force exerted on the circular surface AB by the fluid?



$F_{Rx} = \rho g h_c A_{AC}$

F_{Rx} =***(1000 kg/m³) (9.8 m/s²) (4.5 m) (3 m) (8 m)***

F_{Rx} **= *1,058 kN* ,** ***Note that h_c is the vertical distance to the centroid of plane area AC.***

the y-component of the resultant force is the weight of the water directly above the curved surface (i.e., imaginary volume ABEF).

$F_{Ry} = \rho g \, Vol_{ABEF} = \rho g \, (Vol_{ADEF} + Vol_{ABD})$

F_{Ry} = (1000 kg/m³) (9.8 m/s²) [(3 m) (3 m) (8 m) + (π 3²/4) m² (8 m)]

F_{Ry} = 1,260 kN

Hence, the resultant force is given by

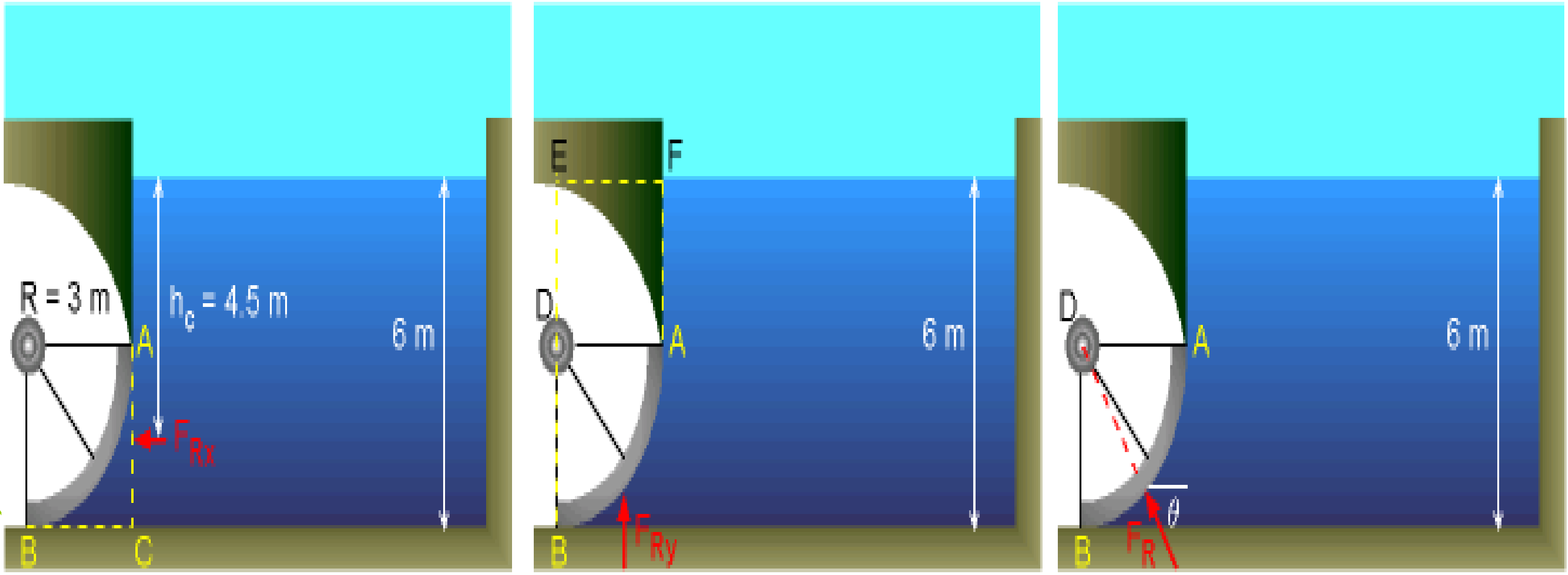
$F_R = (F_{Rx}^2 + F_{Ry}^2)^{0.5}$ = [(1,058 kN)² + (1,260 kN)²]$^{0.5}$

F_R = 1,645 kN

And the angle θ is given by

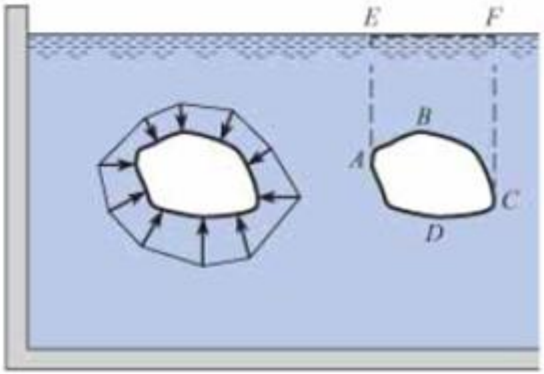
$\theta = \tan^{-1} (F_{Ry} / F_{Rx})$

$= \tan^{-1}$ (1,260 kN / 1,058 kN) = 50°

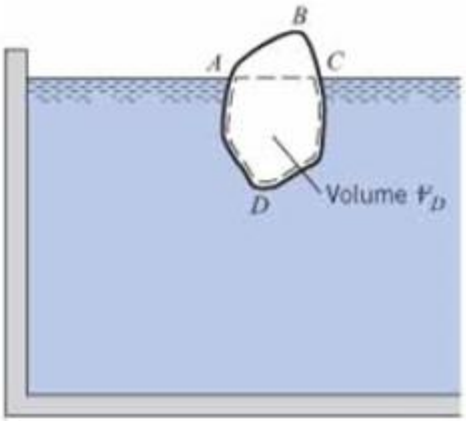

R = 3 m
h_c = 4.5 m
A
F_Rx
6 m
B
C
E
F
D
A
6 m
F_Ry
B
D
A
6 m
θ
F_R
B

Archimedes' Principle

The resultant fluid force acting on a body that is completely Submerged or floating in a fluid is called the ***buoyant force.***



Two views of a body immersed in a liquid.



A body partially submerged in a liquid.

Buoyancy is due to the fluid displaced by a body $F_B = \gamma \forall_D$

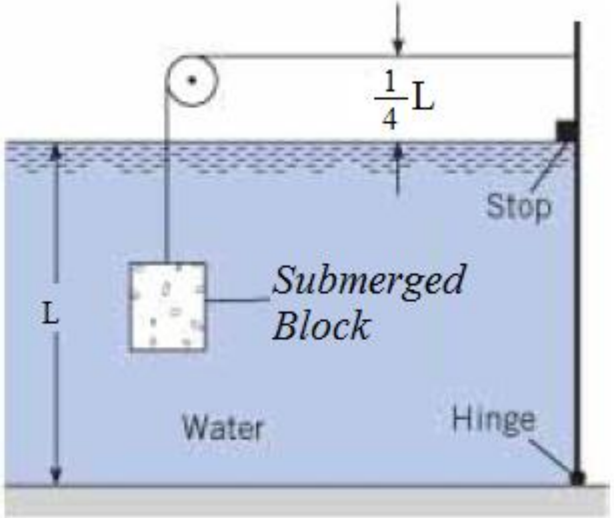
V_D is the displaced or "**Submerged** "Volume

What is the minimum volume [in m³] of submerged alloy block shown (S=2.9) needed to keep the gate (1 m wide) in a closed position? let L = 2 m. *Note the hinge at the bottom of the gate, ignore reactions at the stop.*

$$F_h = \gamma\, h_v\, Area = \gamma \frac{L}{2}\left(L * w\right) = \gamma w \frac{L^2}{2}$$

$$y_{cp} = \bar{y} + \frac{I}{\bar{y} A} = \frac{L}{2} + \frac{\frac{1}{12} w h^3}{(L/2)(L*w)}$$

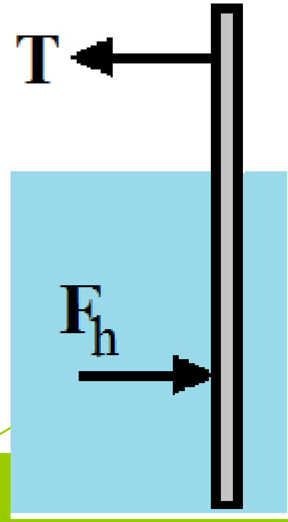
$$y_{cp} = L/2 + \frac{\frac{1}{12} w (L)^3}{(L/2)(L*w)} = \frac{L}{2} + \frac{L}{6} = \frac{2}{3} L$$



$$\sum M_{hinge} = 0$$

$$T * (5L/4) - F_h\left(L - y_{cp}\right) = 0 \Rightarrow T = F_h\left(L - y_{cp}\right)/(5L/4)$$

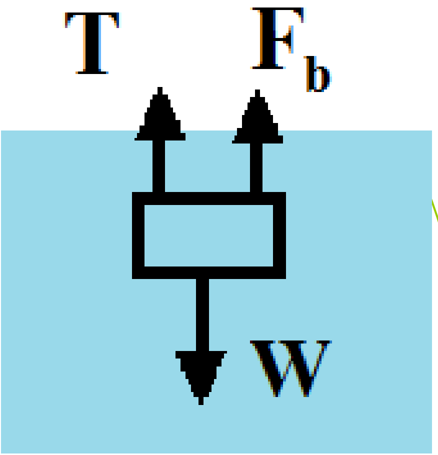
$$T = (\gamma w L^2/2)\left(L - \frac{2}{3}L\right)/(5L/4) = 2\gamma w L^2/15$$



$$T = mg - F_b = \gamma_{block} Vol - \gamma_{water} Vol$$

$$Volume = \frac{T}{\gamma_{block} - \gamma_{water}} = \frac{T}{\gamma_{water}(S_{block} - 1)}$$

$$Volume = \frac{T}{\gamma_{water}(S_{block} - 1)} = \frac{2\gamma w L^2 / 15}{\gamma_{water}(S_{block} - 1)}$$

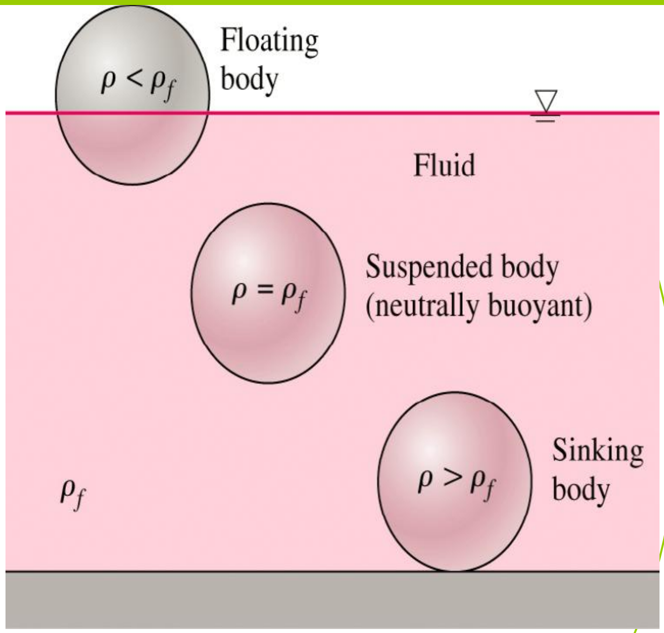


Stability of Immersed and Floating Bodies

- Buoyancy force F_B is equal only to

The displaced volume * specific weight

- 1.$\rho_{body} < \rho_{fluid}$: Floating body
- 2.$\rho_{body} = \rho_{fluid}$: Neutrally buoyant
- 3.$\rho_{body} > \rho_{fluid}$::Sinking body

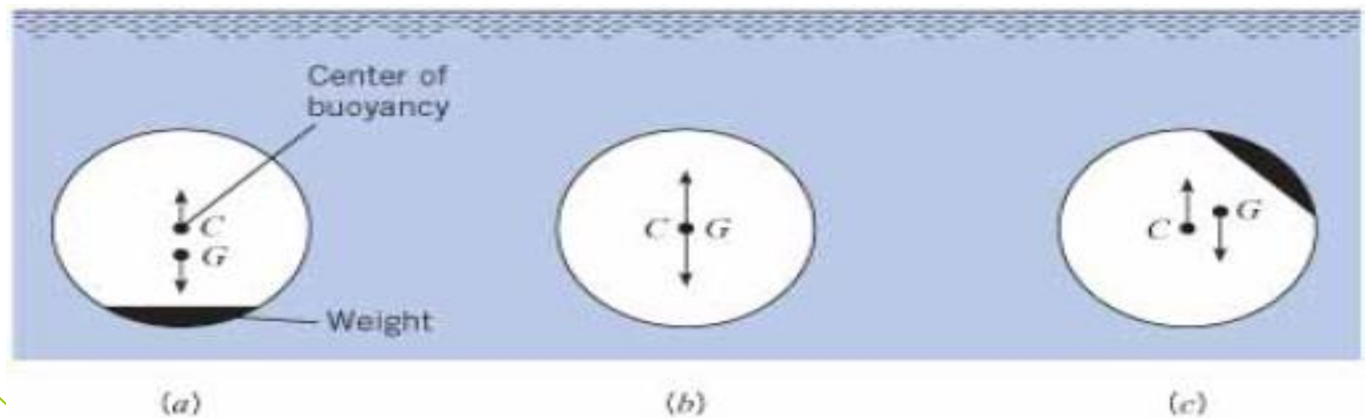


Immersed Bodies: Rotational stability

- Rotational stability of immersed bodies depends upon relative location of *center of gravity* (G) and *center of buoyancy(C)*

1. G below *C*: stable 2. *G* above *C*: unstable

3. *G* coincides with *C*: neutrally stable.



(a) *Stable.* (b) *Neutral.* (c) *Unstable.*

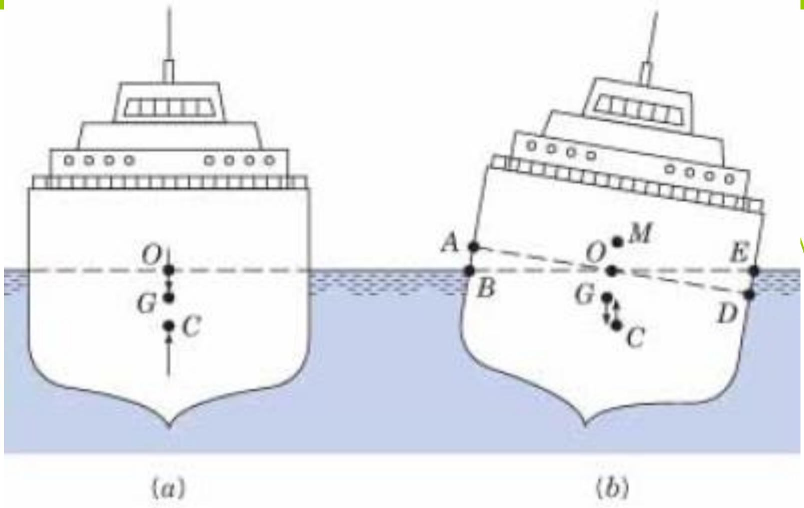
Conditions of stability for immersed bodies.

Floating Bodies

The center of gravity *G* is above the center of buoyancy *C*.

The buoyant volume changes due to the side motion "inclination" of the ship

➔Center of buoyancy changes and produces moment that makes the ship stable.

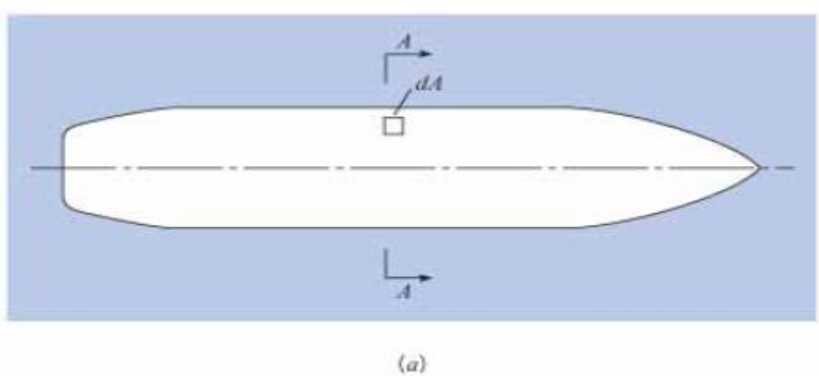


Ship stability relations.

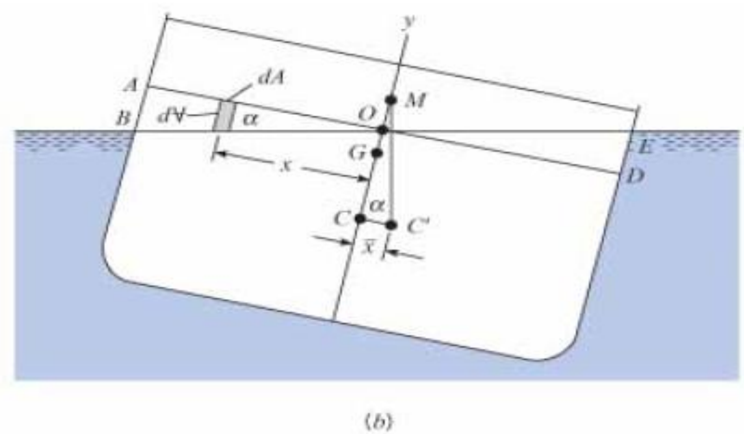
The point of intersection of the lines of action of the buoyant force before and after heel is called the *metacenter M.*

The distance *GM* is called the *metacentric height*.

If *GM* is positive—that is, if *M* is above *G*—the ship is stable; however, if *GM* is negative, the ship is unstable.



(a) Plan view of ship at waterline.



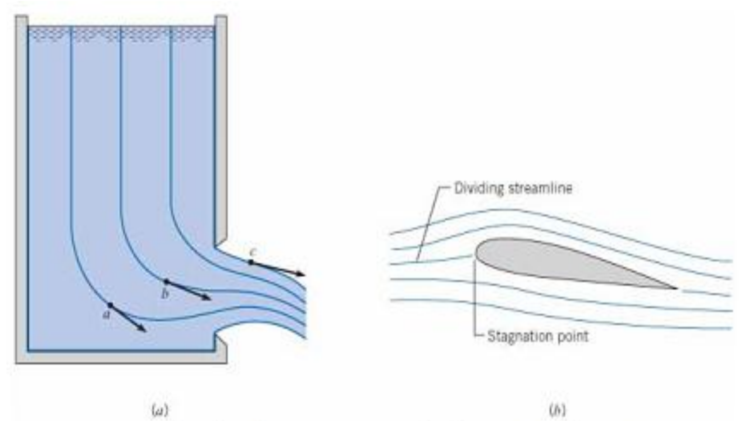
(b) Section A-A of ship.

Chapter 4
Flowing Fluids and Pressure Variation

4.1 Descriptions of Fluid Motion
Streamlines and Flow Patterns

Streamlines: lines that show the flow direction, group of these line is called flow pattern.
streamline is defined as a line drawn through the flow field in such a manner that the local velocity vector is tangent to the streamline at every point along the line at that instant.

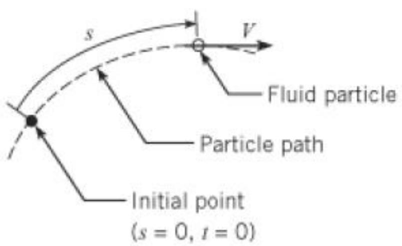
Water flowing through a slot in the side of a tank Fig(a). The velocity vectors have been sketched at three different locations: *a*, *b*, and *c*. The streamlines, are tangent to the velocity vectors at these points. Also, the velocities are parallel to the wall in the wall region, so the streamlines adjacent to the wall follow the contour of the wall. The generation of a flow pattern is a very effective way of illustrating the geometric features of the flow field.



Flow through an opening in a tank and over an airfoil section.

Whenever flow occurs around a body, part of it will go to one side and part to the other as shown in Fig. (*b*) for flow over an airfoil section. The streamline that follows the flow division (that divides on the upstream side and joins again on the downstream side) is called the *dividing streamline.* At the location where the dividing streamline intersects the body, the velocity will be zero with respect to the body. This is the *stagnation point.*

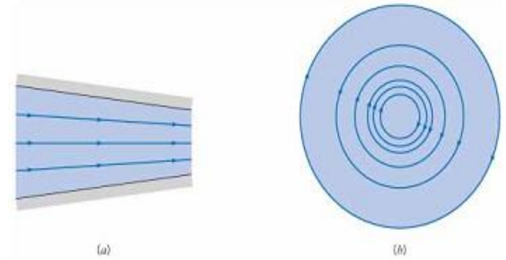
The fluid velocity can be expressed in the form $V = V(s, t)$, where *s* is the distance traveled by a fluid particle along a path, and *t* is the time. In a *uniform flow*, the velocity does not change along a fluid path; that $\frac{\partial V}{\partial s} = 0$, It follows that in uniform flow the fluid paths are straight and parallel.
In *nonuniform flow*, the velocity changes along a fluid path, so $\frac{\partial V}{\partial s} \neq 0$,



Fluid particle moving along a pathline.


Uniform flow in a pipe.

For the converging duct in Fig. (a), the magnitude of the velocity increases as the duct converges, so the flow is nonuniform. For the vortex flow shown in Fig. (*b*), the magnitude of the velocity does not change along the fluid path, but the direction does, so the flow is nonuniform.



Flow patterns for nonuniform flow. (a) Converging flow. (b) Vortex flow.

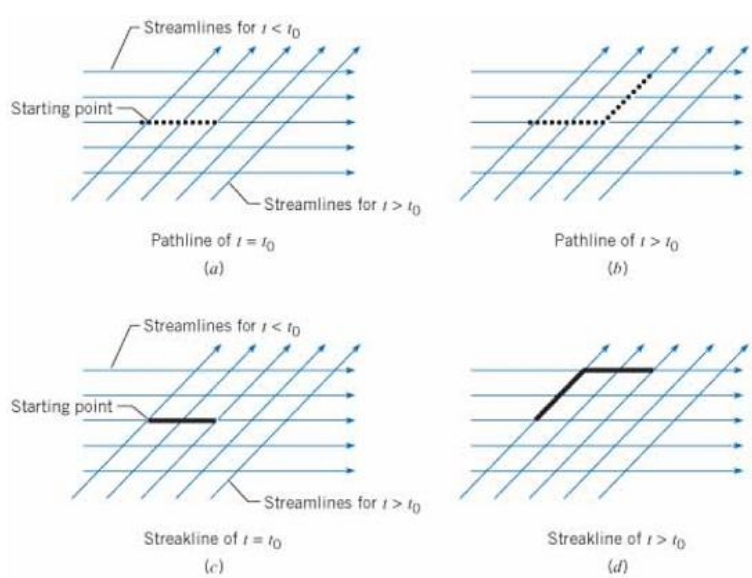
Flows can be either steady or unsteady. In a *steady flow* the velocity at a given point on a fluid path does not change with time: $\frac{\partial V}{\partial t} = 0$.

The flow in a pipe, would be an example of steady flow if there was no change in velocity with time. An *unsteady flow* exists if $\frac{\partial V}{\partial t} \neq 0$. If the flow in the pipe changed with time due to a valve opening or closing, the flow would be unsteady; that is, the velocity at any point selected on a fluid path would be increasing or decreasing with time. Although unsteady, the flow would still be uniform.steadiness or unsteadiness of the flow because the streamlines are only an instantaneous representation of the flow field.

Pathlines and Streaklines

Two other approaches used to visualize flow fields; namely, the pathline and streakline.
The *pathline* simply is the path of a fluid particle as it moves through the flow field
For an example of a pathline, consider a two-dimensional flow that initially has horizontal streamlines as shown, At a given time, t_0, the flow instantly changes direction, and the flow moves upward to the right at 45° with no further change. The flow is unsteady because the velocity at a point changes with time. A fluid particle is tracked from the starting point, and up to time t_0, the pathline is the horizontal line segment shown on Fig. *a*. After time t_0, the particle continues to follow the streamline and moves up the right as shown in Fig. b. Both line segments constitute the pathline.

The *streakline* is the line generated by a tracer fluid, such as a dye, continuously injected into the flow field at the starting point. Up to time t_0, the dye will form a line segment as shown in Fig.c. Up to this time, there is no difference between the pathline and the streakline. Now the flow changes directions, and the initial horizontal dye line is transported, in whole, in the upward 45° direction. After t_0 the dye continues to be injected and forms a new line segment along the new streamline, resulting in the streakline shown in Fig. d



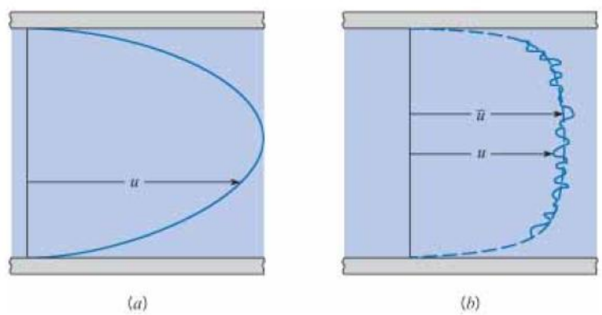
Streamlines, pathlines, and streakline for an unsteady flow field.

Obviously, the pathline and streakline are very different. In general, neither pathlines nor streaklines represent streamlines in an unsteady flow. Both the pathline and streakline provide a history of the flow field, and the streamlines indicate the current flow pattern.
In steady flow the pathline, streakline, and streamline are coincident if they pass through the same point.

Laminar and Turbulent Flow

Laminar flow is a well-ordered state of flow in which adjacent fluid layers move smoothly with respect to each other. A typical laminar flow would be the flow of honey or thick syrup from a pitcher. Laminar flow in a pipe has a smooth, parabolic velocity distribution.
Turbulent flow is an unsteady flow characterized by intense cross-stream mixing. For example, the flow in the wake of a ship is turbulent. The eddies "الدوامات" observed in the wake cause intense mixing. The transport of smoke from a smoke stack on a windy day also exemplifies a turbulent flow. The mixing is apparent as the plume widens and disperses.

An instantaneous velocity profile for turbulent flow in a pipe is shown in Fig. (b). A near uniform velocity distribution occurs across the pipe because the high-velocity fluid at the pipe center is transported by turbulent eddies across the pipe to the low-velocity region near the wall. Because the flow is unsteady, the velocity at any point in the pipe fluctuates with time.



Laminar and turbulent flow in a straight pipe. (a) Laminar flow. (b) Turbulent flo

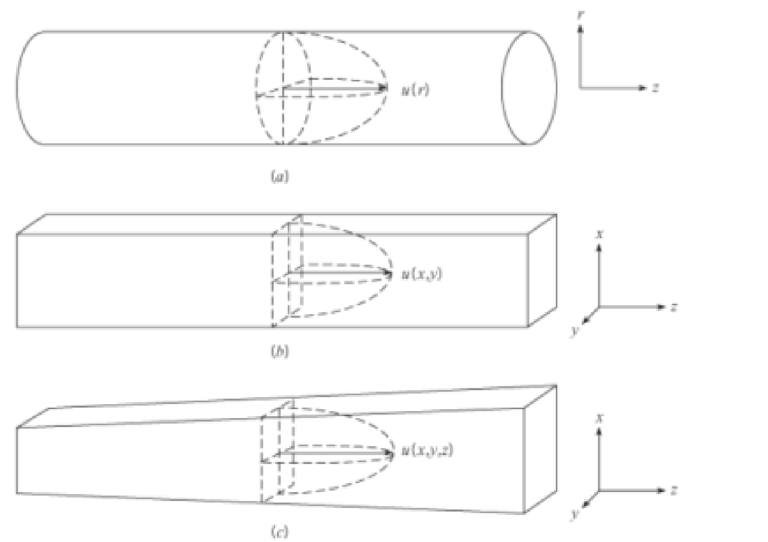
The standard approach to treating turbulent flow is to represent the velocity as a time-averaged average value plus a fluctuating quantity, $u = \bar{u} + u'$. The time-averaged value is designated $\bar{u}$ by in Fig. (*b*). The fluctuation velocity is the difference between the local velocity and the averaged velocity. A turbulent flow is often designated as "steady" if the time-averaged velocity is unchanging with time.
In general, laminar pipe flows are associated with low velocities and turbulent flows with high velocities. Laminar flows can occur in small tubes, highly viscous flows, or flows with low velocities, but turbulent flows are, by far, the most common.

One-Dimensional and Multi-Dimensional Flows

The dimensionality of a flow field is characterized by the number of spatial dimensions needed to describe the velocity field. Fig. 4.8*a* shows the velocity distribution for an axisymmetric flow in a circular duct. The flow is uniform, or fully developed, so the velocity does not change in the flow direction (z). The velocity depends on only one dimension, namely the radius r, so the flow is one-dimensional. Fig. 4.8*b* shows the velocity distribution for uniform flow in a square duct. In this case the velocity depends on two dimensions, namely x and y, so the flow is two-dimensional. Figure 4.8*c* also shows the velocity distribution for the flow in a square duct but the duct cross-sectional area is expanding in the flow direction so the velocity will be dependent on *z* as well as *x* and *y*. This flow is three-dimensional.

Another good example of three-dimensional flow is turbulence, because the velocity components at any one time depend on the three coordinate directions. For example, the velocity component *u* at a given time depends on *x*, *y*, and *z*; that is, *u*(*x*,*y*,*z*). Turbulent flow is unsteady, so the velocity components also depend on time.

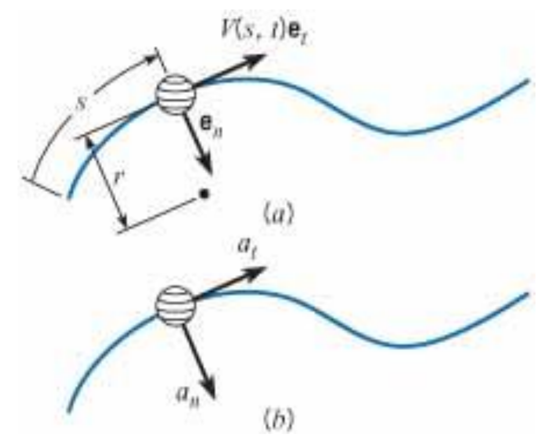
Another definition frequently used in fluid mechanics is quasi–one–dimensional flow. By this definition it is assumed that there is only one component of velocity in the flow direction and that the velocity profiles are uniformly distributed; that is, constant velocity across the duct cross section.



Flow dimensionality, (a) one-dimensional flow, (b) two-dimensional flow, and (c) three-dimensional flow.

4.2 Acceleration

Acceleration of a fluid particle as it moves along a pathline, as shown in Fig (a). is the rate of change of the particle's velocity with time. The local velocity of the fluid particle depends on the distance traveled, *s*, and time, *t*. The local radius of curvature of the pathline is *r*. The components of the acceleration vector are shown in Fig.(b). The normal component of acceleration a_n will be present anytime a fluid particle is moving on a curved path (i.e., centripetal acceleration). The tangential component of acceleration a_t will be present if the particle is changing speed.



Particle moving on a pathline. (a) Velocity. (b) Acceleration

$$V = V(s,t)\boldsymbol{e_t}$$

where *V*(*s*, *t*) is the speed of the particle, which can vary with distance along the pathline, *s*, and time, *t*. The direction of the velocity vector is given by a unit vector $\boldsymbol{e_t}$.

Using the definition of acceleration,

$$a = \frac{dV}{dt} = \left(\frac{dV}{dt}\right)\boldsymbol{e_t} + V\left(\frac{d\boldsymbol{e_t}}{dt}\right)$$

$$\frac{dV(s,t)}{dt} = \left(\frac{dV}{ds}\right)\left(\frac{ds}{dt}\right) + \frac{\partial V}{\partial t}$$

$$\frac{dV(s,t)}{dt} = V\left(\frac{\partial V}{\partial s}\right) + \frac{\partial V}{\partial t}$$

The derivative of the unit vector $d\boldsymbol{e_t}/dt$ is nonzero because the direction of the unit vector changes with time as the particle moves along the pathline. The derivative is

$$\frac{d\boldsymbol{e_t}}{dt} = \frac{V}{r}\boldsymbol{e_n}$$

where **e***n* is the unit vector perpendicular to the pathline and pointing inward toward the center of curvature.

$$a = \left(V\left(\frac{dV}{ds}\right) + \frac{\partial V}{\partial t}\right)\boldsymbol{e_t} + \left(\frac{V^2}{r}\right)\boldsymbol{e_n}$$

Convective, Local, and Centripetal Acceleration

The acceleration component along a pathline depends on two terms. The variation of velocity with time at a point on the pathline, namely $\partial V / \partial t$, is called the *local acceleration*. In steady flow the local acceleration is zero. The other term, $V\partial V / \partial s$, depends on the variation of velocity along the pathline and is called the *convective acceleration*. In a uniform flow, the convective acceleration is zero. The acceleration with magnitude V^2 / r, which is normal to the pathline and directed toward the center of rotation, is the *centripetal acceleration.*

Problems 4.22, & 4.23

Liquid flows through this two-dimensional slot with a velocity of $V = 2(q_0/b)(t/t_0)$, where q_0 and t_0 are reference values. What will be the local and convective acceleration at $x = 2B$ and $y = 0$ in terms of B, t, t_0, and q_0?

$$V = 2\left(\frac{q0}{b}\right)\left(\frac{t}{t0}\right), \quad \mathrm{b} = \mathrm{B} - \frac{\mathrm{x}}{8}$$

$$V = 2\left(\frac{q0}{\mathrm{B}-\frac{\mathrm{x}}{8}}\right)\left(\frac{t}{t0}\right) \rightarrow a_l = \frac{\partial V}{\partial t} = 2\left(\frac{q0}{\mathrm{B}-\frac{\mathrm{x}}{8}}\right)\left(\frac{1}{t0}\right)$$

@x=2B , $a_l = \frac{\partial V}{\partial t} = 2\left(\frac{q0}{\mathrm{B}-\frac{2\mathrm{B}}{8}}\right)\left(\frac{1}{t0}\right) = \frac{8}{3}\left(\frac{q0}{\mathrm{B}t0}\right)$

$$a_c = V\partial V/\partial x$$

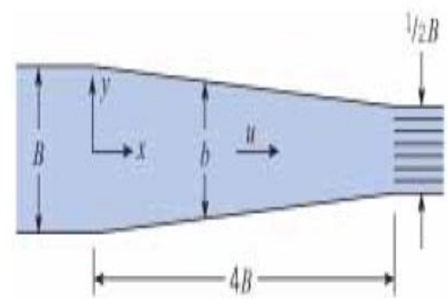
The width varies as $b = B - x/8$

$$\begin{aligned} V &= (q_0/t_0)2t(B - x/8)^{-1} \\ \partial V/\partial x &= (q_0/t_0)2t(1/8)(B - x/8)^{-2} \\ a_c &= V\partial V/\partial x = V(q_0/t_0)^2 4t^2(1/8)/(B - (1/8)x)^{-3} \end{aligned}$$

At $x = 2B$

$$a_c = (1/2)(q_0/t_0)^2 t^2/((3/4)B)^3$$

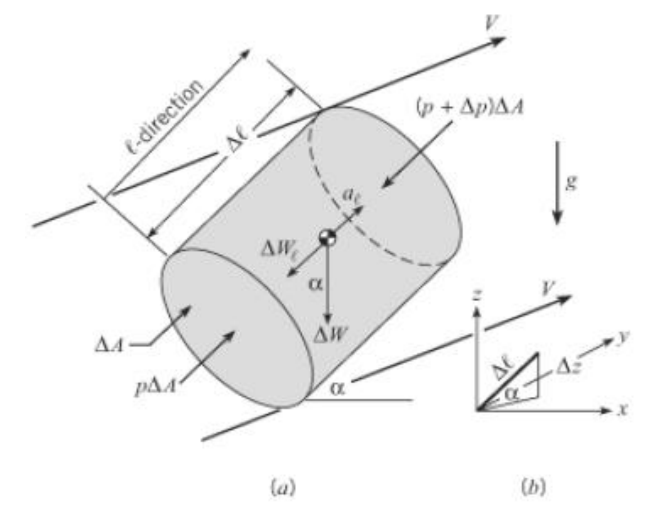
$$\boxed{a_c = 32/27(q_0/t_0)^2 t^2/B^3}$$



PROBLEMS 4.22, 4.23

4.3 Euler's Equation

Consider the cylindrical element in Fig.(*a)* oriented in an arbitrary direction ℓ with cross-sectional area ΔA in a flowing fluid. The element is oriented at an angle α with respect to the horizontal plane (the *x-y* plane) as shown in Fig.(b). The element has been isolated from the flow field and can be treated as a "free body" where the presence of the surrounding fluid is replaced by pressure forces acting on the element. Assume that the viscous forces are zero. Here the element is being accelerated in the ℓ-direction. Note that the coordinate axis *z* is vertically upward and that the pressure varies along the length of the element. Applying Newton's second law in the ℓ-direction results in



Free-body diagram for fluid element accelerating the ℓ-direction. (a) Fluid element. (b) Orientation of element in coordinate system.

$$\sum F_l = m\, a_l$$
$$F_{pressure} + F_{gravity} = m\, a_l$$

The mass of the fluid element is $m = \rho \Delta A \Delta l$

The net force due to pressure in the ℓ-direction is $F_{pressure} = p\Delta A - (p + \Delta p)\Delta A = -\Delta p \Delta A$

Any pressure forces acting on the side of the cylindrical element will not contribute to a force in the ℓ-direction.

The force due to gravity is the component of weight in the ℓ-direction

$$F_{gravity} = -\Delta W_l = -\Delta W \ \sin\alpha$$

where the minus sign occurs because the component of weight is in the negative ℓ-direction.

From the diagram in Fig.(b). $\sin\alpha = \Delta z/\Delta l$, so

$$F_{gravity} = -\Delta W \frac{\Delta z}{\Delta l}$$

Also $\Delta W = \gamma \Delta l \Delta A$ substitute back to get

$$-\Delta p \Delta A - \gamma \Delta l \Delta A \frac{\Delta z}{\Delta l} = \rho \Delta l \Delta A\, a_l$$

Divide by $\Delta l \Delta A$

$$-\frac{\Delta p}{\Delta l} - \gamma \frac{\Delta z}{\Delta l} = \rho a_l$$

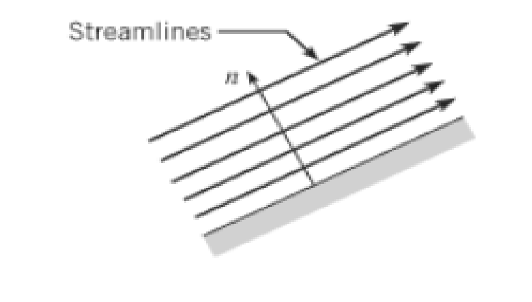
Taking the limit as Δl approaches zero (element shrinks to a point) leads to the differential equation for acceleration in the ℓ-direction,

$$-\frac{\partial p}{\partial l} - \gamma \frac{\partial z}{\partial l} = \rho a_l$$
$$-\frac{\partial}{\partial l}(p + \gamma z) = \rho a_l$$

Euler's equation for motion of a fluid. It shows that the acceleration is equal to the change in piezometric pressure with distance, and the minus sign means that the acceleration is in the direction of decreasing piezometric pressure.

In a static body of fluid, Euler's equation reduces to the hydrostatic differential equation. In a static fluid, there are no viscous stresses, which is a condition required in the derivation of Euler's equation. Also there is no motion, so the acceleration is zero in all directions. Thus, Euler's equation reduces to $\partial/\partial\ell(p + \gamma z) = 0$.

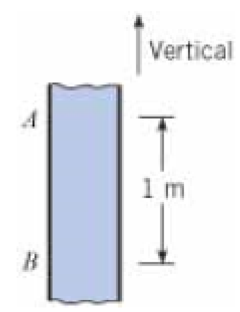
Euler's equation can be applied to find the pressure distribution across streamlines in rectilinear flow. Consider the flow with parallel streamlines adjacent a wall shown in Fig. 4.12. In the direction normal to the wall, the n direction, the acceleration is zero. Applying Euler's equation in the n direction gives $\partial/\partial n(p + \gamma z) = 0$, so the piezometric pressure is constant in the normal direction.



Normal direction to parallel stream surfaces.

Problem 4.29

The hypothetical liquid in the tube shown in the figure has zero viscosity and a specific weight of 10 kN/m^3. If $p_{B-}p_A$ is equal to 12 kPa, one can conclude that the liquid in the tube is being accelerated (a) upward, (b) downward, or (c) neither: acceleration = 0.



PROBLEM 4.29

$$\rho a_z = -\frac{\partial}{\partial z}(p + \gamma z) ==> a_z = -\frac{1}{\rho}\frac{\partial}{\partial z}(p + \gamma z)$$

At B ➔ $z_B = 0, \quad p = p_B$

$z_A = 1, \quad p = p_A$

$$\frac{\partial p}{\partial z} = \frac{\Delta p}{\Delta z} = \frac{p_{B-}p_A}{z_{B-}z_A} = \frac{12{,}000}{0 - 1} = -12{,}000 Pa$$

$$a_z = -\frac{1}{\rho}\frac{\partial}{\partial z}(p + \gamma z)$$

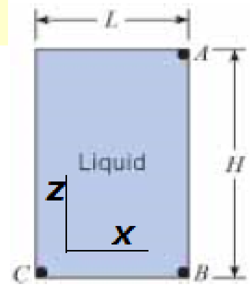
$\gamma = g\rho$ ➔ $\rho = \gamma/g$

$$a_l = -\frac{1}{\rho}\left(\frac{\partial}{\partial z}p + \gamma\right) = -g\left(\frac{1}{\gamma}\frac{\Delta p}{\Delta l} + 1\right)$$

$$a_l = -g\left(\frac{1}{\gamma}(-12{,}000) + 1\right) = g\left(\frac{12{,}000}{\gamma} - 1\right)$$

Since a_l has a positive value then the liquid in the tube is being accelerated upward

P4.38 **The closed tank shown, which is full of liquid, is accelerated downward at (2*g/3) and to the right at 1g. Here L = 2.5 m, H = 3 m, and the liquid has a specific gravity of 1.3. Determine p_C - p_A and p_B - p_A***



Euler's equation in z direction

$$\begin{aligned} dp/dz + \gamma &= -\rho a_z \\ dp/dz &= -\rho(g + a_z) \end{aligned}$$

a_z = ***-2/3 g = -6.54 m/s²***

$$\begin{aligned} dp/dz &= -1.3 \times 1,000(9.81 - 6.54) \\ &= -4,251 \text{ N/m}^3 \end{aligned}$$

$$(p_B - p_A)/(z_B - z_A) = \frac{dp}{dz}$$

$$(p_B - p_A)/(-H) = -4.251$$

$$H = 3$$

$$(p_B - p_A) = (-4.251)(-3)$$

$$(p_B - p_A) = 12.753 kPa$$

Euler's equation in x-direction

$$\begin{aligned} -\frac{dp}{dx} &= \rho a_x \\ p_C - p_B &= \rho a_x L \\ &= 1.3 \times 1,000 \times 9.81 \times 2.5 \\ &= 31,882 \text{ Pa} \end{aligned}$$

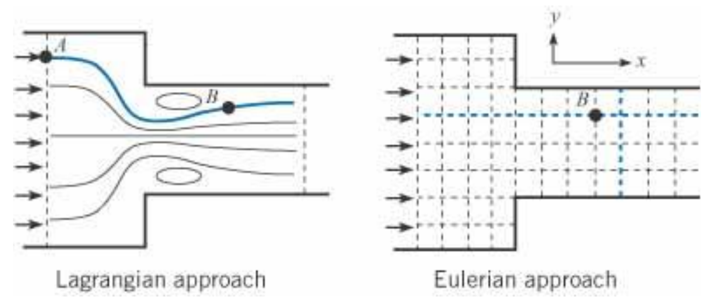
$$p_C - p_A = p_C - p_B + (p_B - p_A)$$

$$\begin{aligned} p_C - p_A &= 31,882 + 12,753 \\ &= 44,635 \text{ Pa} \end{aligned}$$

$$\boxed{p_C - p_A = 44.63 \text{ kPa}}$$

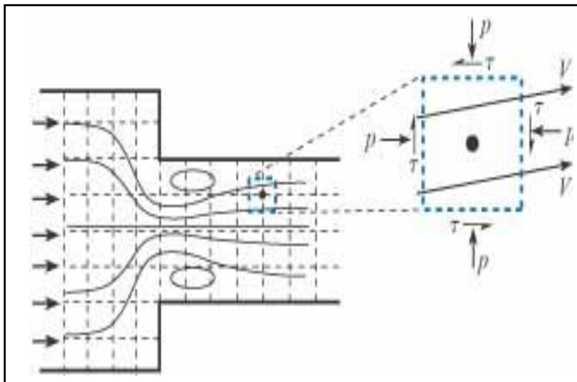
Chapter 5 Control Volume Approach and Continuity Equation

Lagrangian and Eulerian Approach



The Lagrangian and Eulerian approaches for quantifying the flow field in sudden contraction.

To evaluate the pressure and velocities at arbitrary locations in a flow field.
The flow into a sudden contraction, It is desired to evaluate the pressure at point *B*. The pressure and velocity are known at the inlet.
Lagrangian approach: locate the pathline that starts at the inlet, point *A*, and passes through point *B*. along this pathline,
The pressure changes with velocity according to Euler's equation. Integrating Euler's equation from *A* to *B* would yield the pressure at point *B*. If the flow is steady, the Bernoulli equation could be used between the two points.
It is an enormous task to keep track of all the pathlines required to evaluate the flow properties at a given point in the flow field. Besides in unsteady flows, where different pathlines will pass through the same point at different times, problem becomes compounded.
Eulerian approach: Solving the fluid flow equations to yield the flow properties at any point in the field. Develop a solution to the flow field that provides the flow properties at any point. Thus if the pressure were available as a function of location, $p(x,y)$, then the pressure at point *B* would simply be obtained by substituting in the values of the coordinates at that point.



A control volume in a flow field

The Eulerian approach ➔ the basic equations must be recast in Eulerian form.
In solid body mechanics, the fundamental equations are developed using the "free body" concept in which an element in the field is isolated and the effect of the surroundings is replaced by forces acting on the surface of element.
Same approach is used in fluid mechanics as shown in Fig. The volume enclosing a point is identified as a "control volume." The effects of the surroundings are replaced by forces due to pressure and shear stress acting on the surface of the control volume.

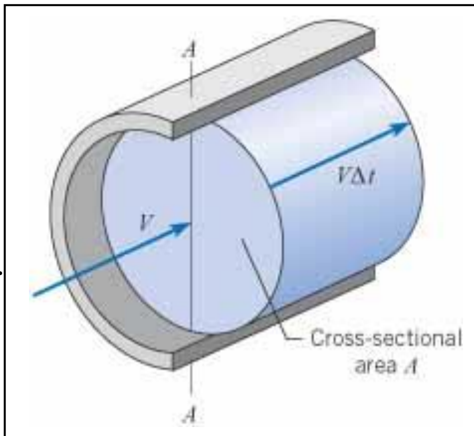
In addition to the forces like those applied to the "free body," there is a flow through the control volume that has to be taken into account.

5.1 Rate of Flow

Discharge

The *discharge*, *Q*, often called the *volume flow rate*, is the volume of fluid that passes through an area per unit time. For example, when filling the gas tank of an automobile, the discharge or volume flow rate would be the gallons per minute flowing through the nozzle. Typical units for discharge are ft3/s (cfs), ft3/min (cfm), gpm, m3/s, and L/s.
The discharge "volume flow rate" in a pipe is related to the flow velocity and cross-sectional area.

Consider the idealized flow of fluid in a pipe as shown in Fig. in which the velocity is constant across the pipe section. Suppose a marker is injected over the cross section at section *A-A* for a period of time Δt. The fluid that passes *A-A* in time ,*t* is represented by the marked volume. The length of the marked volume is $V\Delta t$ so the volume is $\Delta\forall = V\Delta t A$, where A is the cross-sectional area of the pipe. The volume flow per unit time past *A-A* is



Volume of fluid in flow with uniform velocity distribution that passes section A-A in time Δt

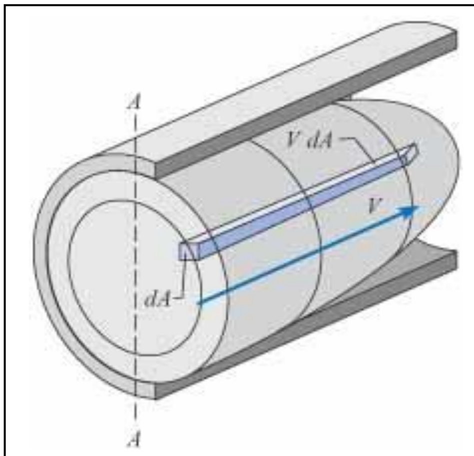
$$\frac{\Delta\forall}{\Delta t} = VA$$

Taking the limit as $\Delta t \rightarrow 0$ gives

$$Q = \lim_{\Delta t \to 0} \frac{\Delta\forall}{\Delta t} = VA$$

The discharge given by Eq. is based on a constant flow velocity over the cross-sectional area.

In general, the velocity varies across the section such as shown in Fig. The volume flow rate through a differential area of the section is *V dA*, and the total volume flow rate is obtained by integration over the entire cross-section



Volume of fluid that passes section A-A in time Δt

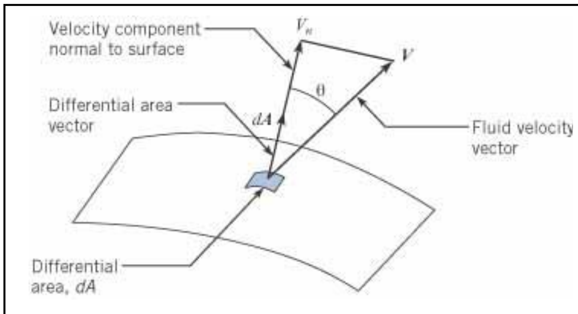
$$Q = \int_A V dA$$

$$\bar{V} = Q/A$$

Laminar flows in circular pipes, the velocity profile is parabolic the mean velocity is half the centerline velocity

Turbulent pipe flow, the time-averaged velocity profile is nearly uniformly distributed across the pipe, so the mean velocity is fairly close to the velocity at the pipe center.

The volume flow rate equation can be generalized by using the concept of the dot product. The flow velocity vector is not normal to the surface but is oriented at an angle θ with respect to the direction that is normal to the surface. The only component of velocity that contributes to the flow through the differential area *dA* is the component normal to the area, V_n. The differential discharge through area *dA*



Velocity vector oriented at angle θ with respect to normal.

$$dQ = V_n dA$$

If the vector, ***dA***, is defined with magnitude equal to the differential area, *dA*, and direction normal to the surface, then $V_n\, dA = |\boldsymbol{V}| \cos\theta\, dA = \boldsymbol{V}.d\boldsymbol{A}$ where $\boldsymbol{V} \,.\, d\boldsymbol{A}$ is the dot product of the two vectors. Thus a more general equation for the discharge or volume flow rate through a surface *A* is

$$Q = \int_A V.dA$$

If the velocity is constant over the area and the area is a planar surface, then the discharge is given as $Q = V.A$

If, in addition, the velocity and area vectors are aligned, then $Q = VA$

Mass Flow Rate

The *mass flow rate*, $\dot{m}$, is the mass of fluid passing through a cross-sectional area per unit time. The common units for mass flow rate are kg/s, lbm/s, and slugs/s. Using the same approach as for volume flow rate, the mass of the fluid in the marked volume in is $\Delta m = \rho \Delta \forall$, where ρ is the average density. The *mass flow rate equation* is

$$\dot{m} = \lim_{\Delta t \to 0} \frac{\Delta m}{\Delta t} = \rho \lim_{\Delta t \to 0} \frac{\Delta \forall}{\Delta t} = \rho Q = \rho V A$$

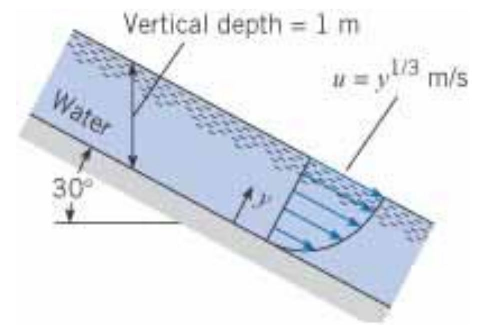
$$\dot{m} = \int_A \rho V . dA$$

Also if the velocity vector is aligned with the area vector, such as integrating over the cross-sectional area of a pipe, the mass flow equation reduces to

$$\dot{m} = \int_A \rho V dA$$

$$Q = \bar{V} A = \frac{\dot{m}}{\rho} = \int_A V . dA$$

P 5.18 The rectangular channel shown is 1.5 m wide. What is the discharge in the channel?



Problem 5.18

$$Q = \int_A u \, dA$$

$$u = y^{1/3}$$

$$dA = width * dy = 1.5 \, dy$$

$$depth = vertical \ depth \times \cos 30 = \cos 30$$

$$Q = \int_0^{\cos 30} 1.5 \, y^{1/3} \, dy$$

$$Q = \left(\frac{3}{2}\right)\left(\frac{3}{4}\right) y^{4/3}\Big|_0^{cos30}$$

$$Q = \frac{9}{8}(cos30)^{4/3} = 0.93 \, m^3/s$$

P 5.10 An aircraft engine test pipe is capable of providing a flow rate of 200 kg/s at altitude conditions corresponding to an absolute pressure of 50 kPa and a temperature of -18°C. The velocity of air through the duct attached to the engine is 240 m/s. Calculate the diameter of the duct.

$$Q = \bar{V} A = \frac{\dot{m}}{\rho}$$

$$\rho = \frac{p}{RT} = \frac{50{,}000}{287 \times 255} = 0.6832 kg/s$$

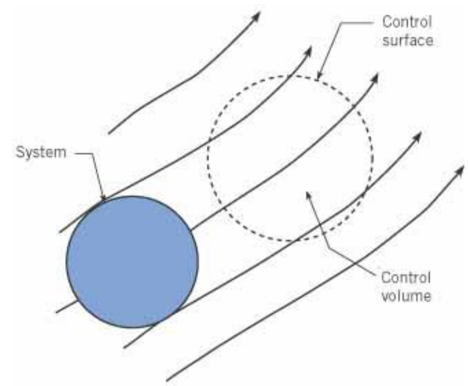
$$Q = \frac{\dot{m}}{\rho} = \frac{200}{0.6832} = 292.74\ m^3/s$$

$$A = \frac{Q}{\overline{V}} = \frac{\pi}{4} d^2$$

$$d = \left(\frac{4}{\pi} \frac{Q}{\overline{V}}\right)^{1/2} = \left(\frac{4}{\pi} \times \frac{292.74}{240}\right)^{1/2} = 1.25\, m$$

5.2 Control Volume Approach

A *system* is a continuous mass of fluid that always contains the same matter. The shape of the system may change with time, but the mass is constant since it always consists of the same matter. The fundamental equations, such as Newton's second law and the first law of thermodynamics, apply to a system.
A *control volume* is volume located in space and through which matter can pass. The system can pass through the control volume.
Fluid mass enters and leaves the control volume through the control surface. The control volume can deform with time as well as move and rotate in space and the mass in the control volume can change with time.



System, control surface, and control volume in a flow field.

Intensive and Extensive Properties

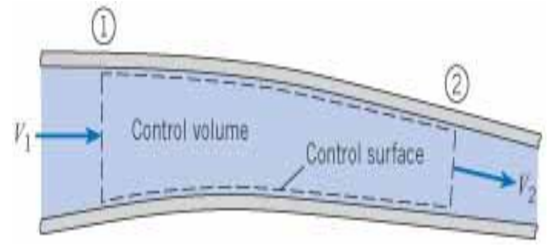
An *extensive property* is any property that depends on the amount of matter present. The extensive properties of a system include mass, *m*, momentum, *m***v** (where **v** is velocity), and energy, *E*. Another example of an extensive property is weight because the weight is *mg*.
An *intensive property* is any property that is independent of the amount of matter present. Examples of intensive properties include pressure and temperature. Many intensive properties are obtained by dividing the extensive property by the mass present. The intensive property for momentum is velocity **v**, and for energy is *e*, the energy per unit mass. The intensive property for weight is *g*.
In this section an equation for a general extensive property, *B*, will be developed. The corresponding intensive property will be *b*.

$$B_{CV} = \int_{CV} b dm = \int_{CV} b \rho d\forall$$

$dm, d\forall$: are the differential mass and differential volume, respectively, and the integral is carried out over the control volume.

Property Transport Across the Control Surface

When fluid flows across a control surface, properties such as mass, momentum, and energy are transported with the fluid either into or out of the control volume.



Flow through control volume in a duct

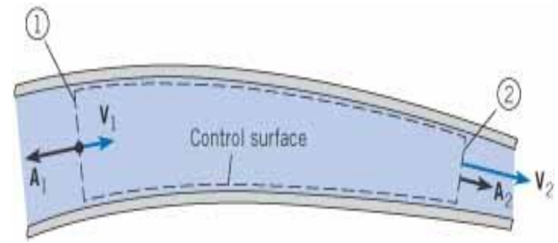
Consider the flow through the control volume in the duct in the fig. If the velocity is uniformly distributed across the control surface, the mass flow rate through each cross section is given by

$\dot{m}_1 = \rho_1 V_1 A_1$ and $\dot{m}_2 = \rho_2 V_2 A_2$

The net mass flow rate out of the control volume *[the outflow rate minus the inflow rate]* is

$$\text{net mass outflow rate} = \dot{m}_2 - \dot{m}_1 = \rho_2 V_2 A_2 - \rho_1 V_1 A_1$$

The same control volume is shown in Fig. with each control surface area represented by a vector, **A**, oriented outward from the Control volume and with magnitude equal to the cross-sectional area. The velocity is represented by a vector, **V**. Taking the dot product of the velocity and area vectors at both stations gives



Control surfaces represented by area vectors and velocities by velocity vectors.

$V_1 . A_1 = -V_1 A_1$ and $V_2 . A_2 = V_2 A_2$

Since at station 1 the velocity and area have the opposite directions while at station 2 the velocity and area vectors are in the same direction. Now the net mass outflow rate can be written as

$$\text{net mass outflow rate} = \rho_2 V_2 A_2 - \rho_1 V_1 A_1 = \rho_2 \boldsymbol{V_2} . \boldsymbol{A_2} - \rho_1 \boldsymbol{V_1} . \boldsymbol{A_1} = \sum_{CS} \rho \ \boldsymbol{V} \ . \boldsymbol{A}$$

if the dot product ρ$\boldsymbol{V} \cdot \boldsymbol{A}$ is summed for all flows into and out of the control volume, the result is the net mass flow rate out of the control volume, or the net mass efflux. If the summation is positive, the net mass flow rate is out of the control volume. If it is negative, the net mass flow rate is into the control volume. If the inflow and outflow rates are equal, then

$$\sum_{CS} \rho \ \boldsymbol{V} \ . \boldsymbol{A} \ = 0$$

In a similar manner, to obtain the net rate of flow of an extensive property *B* out of the control volume, the mass flow rate is multiplied by the intensive property *b*:

$$\dot{B}_{net} = \sum_{CS} b\rho \ \boldsymbol{V} \ . \boldsymbol{A}$$

$$\dot{B}_{net} = \left\{\frac{B}{mass}\right\}\left\{\frac{mass}{time}\right\} = \left\{\frac{B}{time}\right\}$$

This Equation is applicable for all flows where the properties are uniformly distributed across the area. If the properties vary across a flow section, then it becomes necessary to integrate across the section to obtain the rate of flow. A more general expression for the net rate of flow of the extensive property from the control volume is thus

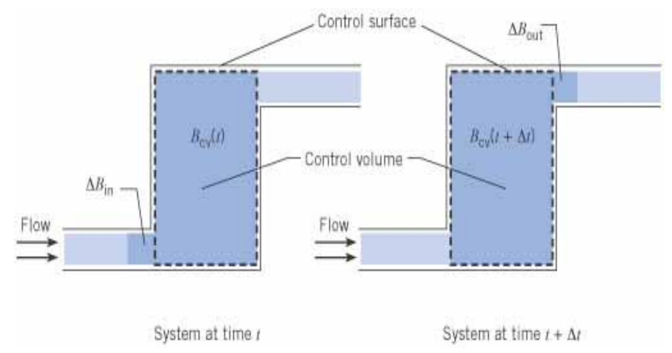
$$\dot{B}_{net} = \int_{cs} b\rho \ \boldsymbol{V} \ . d\boldsymbol{A}$$

Reynolds Transport Theorem

$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{CV} b\rho d\forall + \int_{cs} b\rho\ \boldsymbol{V}\ .d\boldsymbol{A}$$

$$\begin{Bmatrix} \text{Rate of change of} \\ \text{propety } B \text{ in a system} \end{Bmatrix} = \begin{Bmatrix} \text{Rate of change of} \\ \text{propety } B \text{ in } CV \end{Bmatrix} + \begin{Bmatrix} \text{Net outflow of property } B \\ \text{Through } CS \end{Bmatrix}$$

The left side of the equation is the Lagrangian form; that is, the rate of change of property *B* evaluated moving with the system. The right side is the Eulerian form; that is, the change of property *B* evaluated in the control volume and the flux measured at the control surface.
This equation applies at the instant the system occupies the control volume and provides the connection between the Lagrangian and Eulerian descriptions of fluid flow.
The application of this equation is called the *control volume approach.* The velocity ***V*** is always measured with respect to the control surface because it relates to the mass flux across the surface.



Progression of a system through a control volume.

A simplified form of the Reynolds transport theorem can be written if the mass crossing the control surface occurs through a number of inlet and outlet ports, and the velocity, density and intensive property *b* are uniformly distributed (constant) across each port. Then

$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{CV} b\rho d\forall + \sum_{CS} b\rho\ \boldsymbol{V}\ .\boldsymbol{A}$$

where the summation is carried out for each port crossing the control surface
An alternative form can be written in terms of the mass flow rates:

$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{CV} b\rho d\forall + \sum_{CS} \dot{m}_o\, b_o - \sum_{CS} \dot{m}_i\, b_i$$

where the subscripts *i* and *o* refer to the inlet and outlet ports, respectively, located on the control surface. This form of the equation does not require that the velocity and density be uniformly distributed across each inlet and outlet port, but the property *b* must be.

5.3 Continuity Equation

The continuity equation derives from the conservation of mass, which, in Lagrangian form, simply states that the mass of the system is constant. $m_{sys} = constant$
The Eulerian form is derived by applying the Reynolds transport theorem. In this case the extensive property of the system is its mass, $B_{cv} = m_{sys}$. The corresponding value for *b* is the mass per unit mass $b_{sys} = \frac{m_{sys}}{m_{sys}} = 1$

General Form of the Continuity Equation

$$\frac{dm_{sys}}{dt} = \frac{d}{dt}\int_{CV} \rho d\forall + \int_{cs} \rho\ \boldsymbol{V}\ .d\boldsymbol{A}$$

However, $dm_{sys}/dt = 0$, so the general, or integral, form of the *continuity equation* is

$$\frac{d}{dt}\int_{CV} \rho d\forall + \int_{cs} \rho \ \boldsymbol{V} \ . d\boldsymbol{A} = 0$$

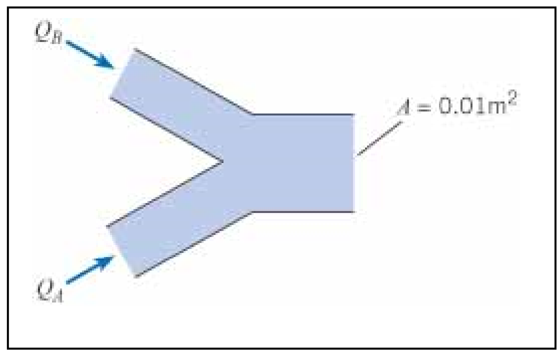
$$\begin{Bmatrix} The\ accumlation\ rate \\ of\ mass\ in\ CV \end{Bmatrix} + \begin{Bmatrix} The\ net\ outflow\ rate \\ of\ mass\ through\ \ CS \end{Bmatrix} = 0$$

If the mass crosses the control surface through a number of inlet and exit ports, the continuity equation simplifies to

$$\frac{d}{dt} m_{CV} + \sum_{cs} \dot{m}_o - \sum_{cs} \dot{m}_i = 0$$

where m_{cv} is the mass of fluid in the control volume. Note that the two summation terms represent the net mass outflow through the control surface

Problem 5.46 Two streams discharge into a pipe as shown. The flows are incompressible. The volume flow rate of stream *A* into the pipe is given by $QA = 0.02t$ m^3/s and that of stream *B* by $QB = 0.008t^2$ m^3/s, where *t* is in seconds. The exit area of the pipe is 0.01 m2. Find the velocity and acceleration of the flow at the exit at $t = 1$s.



solution

Continuity principle

$Q_{exit} = Q_A + Q_B$

$V_{exit} = (1/A_{exit})(Q_A + Q_B)$

$= (1/0.01\ \text{m}^2)(.02t\ \text{m}^3/\text{s} + 0.008t^2\ \text{m}^3/\text{s})$

$V_{exit} = 2t$ m/s $+ 0.8t^2$ m/s

Then at t=1sec

$V_{exit} = 2.8$ m/s

Accelaration

$a_{exit} = \partial V/\partial t + V\partial V/\partial x$, Since V varies with time, but not with position, this becomes

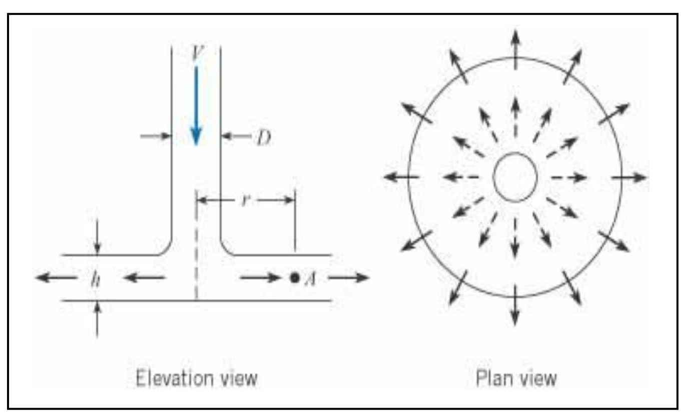
$a_{exit} = \partial V/\partial t = 2+1.6t$

$a_{exit} = 2+1.6 = 3.6$ m/s^2

Problem 5.47

Air discharges downward in the pipe and then outward between the parallel disks. Assuming negligible density change in the air, derive a formula for the acceleration of air at point *A*, which is a distance *r* from the center of the disks.

If $D = 10$ cm $h = 1$ cm, $r = 20$ cm and the discharge is given as $Q = Q_0(t/t_0)$, where $Q_0 = 0.1$ m^3/s and $t_0 = 1$ s. For the, what will be the acceleration at point *A* when $t = 2$ s and $t = 3$ s?



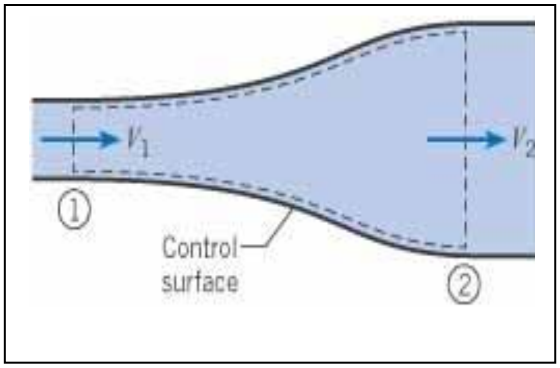
$Q = Q_0(t/t_0)$, $Q_0 = 0.1$ m^3/s and $t_0 = 1$ s
$Q=(0.1\ t)$m^3/s
$V_r = Q/A = Q/(2\pi rh)= Q_0(t/t_0)/(2\pi rh)$
$a_c = V_r\,\partial V_r/\partial r$
$= (Q/(2\pi rh))(-1)(Q)/(2\pi r^2h)$
$a_c = -(Q_0(t/t_0))^2/[r^3\{2\pi h\}^2]$

$a_l = \partial V/\partial t = \partial/\partial t(Q/(2\pi rh))$
$a_l = \partial/\partial t[Q_0(t/t_0)/(2\pi rh)]= (Q_0/t_0)/(2\pi rh)$

At t = 2s, Q = 0.2 m^3/s,
$a_c = -(0.2)^2/[(0.2)^3\{2\pi(0.01)\}^2] = -1266$ m/s^2,
$a_l = (0.1/1)/(2\pi \times 0.20 \times 0.01) = 7.958$ m/s^2
$a = a_{l+}\ a_c = 7.958-1266=1258$ m/s^2

At t = 3s, Q = 0.3 m^3/s,
$a_c = -(0.3)^2/[(0.2)\{2\pi(0.2)(0.01)\}^2] = -2850$ m/s^2,
$a_l = (0.1/1)/(2\pi \times 0.20 \times 0.01) = 7.958$ m/s^2
$a = a_{l+}\ a_c = 7.958-2850= -2842$ m/s^2

Continuity Equation for Flow in a Pipe



Position a control volume inside a pipe,
Mass enters through station 1 and exits through station 2. The control volume is fixed to the pipe walls, and its volume is constant. If the flow is steady, then m_{cv} is constant so the mass flow formulation of the continuity equation reduces to

$$\dot{m}_1 = \dot{m}_2$$

For flow with a uniform velocity and density distribution, the continuity equation for steady flow in a pipe is

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

If the flow is incompressible, then

$$V_1 A_1 = V_2 A_2 \ \rightarrow\ Q_1 = Q_2$$

If there are more than two ports, then the general form of the continuity equation for steady flow:

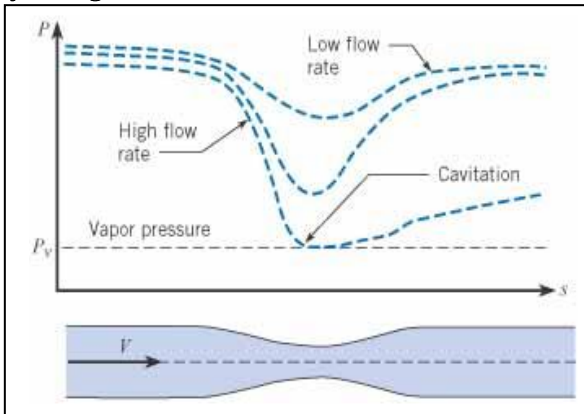
$$\sum_{cs} \dot{m}_i = \sum_{cs} \dot{m}_o$$

If the flow is incompressible

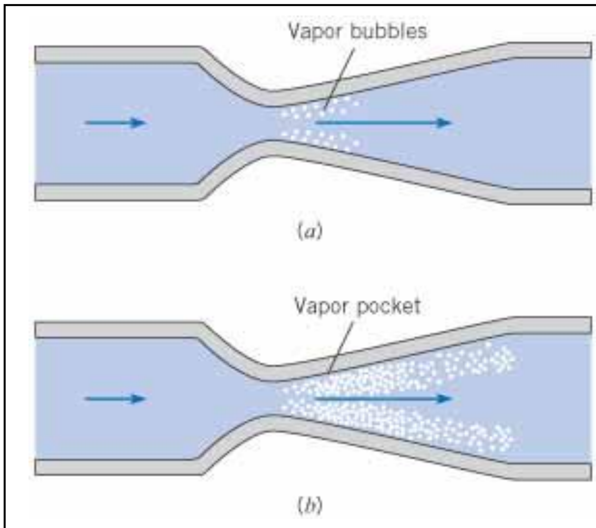
$$\sum_{cs} Q_i = \sum_{cs} Q_o$$

5.4 Cavitation

Cavitation is the phenomenon that occurs when the fluid pressure is reduced to the local vapor pressure and boiling occurs. Under such conditions vapor bubbles form in the liquid, grow, and then collapse, producing shock waves, noise, and dynamic effects that lead to decreased equipment performance and, frequently, equipment failure. Engineers are often concerned about the possibility of cavitation, and they must design flow systems to avoid potential problems.
Cavitation can also be beneficial. Cavitation is responsible for the effectiveness of ultrasonic cleaning. Supercavitating torpedoes have been developed in which a large bubble envelops the torpedo, significantly reducing the contact area with the water and leading to significantly faster speeds. Cavitation plays a medical role in shock wave lithotripsy for the destruction of kidney stones.
Cavitation typically occurs at locations where the velocity is high.
The pipe area decreases ➔ velocity increases [*according to the continuity equation]* ➔ the pressure decreases *[the Bernoulli equation]*.
For low flow rates ➔ small drop in pressure at the restriction➔ water remains well above the vapor pressure and boiling does not occur.
As the flow rate increases, the pressure at the restriction becomes progressively lower until a flow rate is reached where the pressure is equal to the vapor pressure as shown in Fig. At this point, the liquid boils to form bubbles and cavitation ensues. Cavitation can also be affected by the presence of contaminant gases, turbulence and viscosity.
The formation of vapor bubbles at the restriction is shown in Fig. *a*. The vapor bubbles form and then collapse as they move into a region of higher pressure and are swept downstream with the flow. When the flow velocity is increased further, the minimum pressure is still the local vapor pressure, but the zone of bubble formation is extended as shown in Fig. *b*. In this case, the entire vapor pocket may intermittently grow and collapse, producing serious vibration problems. Severe damage that occurred on a centrifugal pump impeller and serious erosion produced by cavitation in a spillway tunnel of Hoover Dam. Cavitation should be avoided or minimized by proper design of equipment and structures and by proper operational procedures.
Experimental studies reveal that very high pressure, as high as 800 MPa develops in the vicinity of the bubbles when they collapse➔ damage to boundaries such as pipewalls, pump impellers, valve casings, and dam slipway floors.



Flow through pipe restriction: variation of pressure for three different flow rates.



Formation of vapor bubbles in the process of cavitation. (a) Cavitation. (b) Cavitation—higher flow rate.

Chapter 6 Momentum Equation

6.1 Momentum Equation: Derivation

When forces act on a particle, the particle accelerates according to Newton's second law

$$\sum F = ma$$

$$\sum F = m\frac{d(\mathbf{v})}{dt} = \frac{d(m\mathbf{v})}{dt}$$

The law can also be formulated for a system composed of a group of particles, for example, a fluid system. In this case, the law may be written as

$$\sum F = \frac{d(\mathbf{Mom}_{sys})}{dt}$$

The term $\mathbf{Mom}_{sys}$ denotes the total momentum of all mass comprising the system.

The above equation is a Lagrangian equation. To derive an Eulerian equation, the Reynolds transport theorem

$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{CV} b\rho d\forall + \int_{CS} b\rho \;\; \boldsymbol{V} \;\; .d\boldsymbol{A}$$

Where $\boldsymbol{V}$ is fluid velocity relative to the control surface at the location where the flow crosses the surface. The extensive property *B*sys becomes the momentum of the system: $B = \mathbf{Mom}_{sys}$. The corresponding intensive property *b* becomes the momentum per unit mass within the system. The momentum of any fluid particle of mass *m* in the system is $m\boldsymbol{v}$, and so $b = (m\boldsymbol{v}) / m = \boldsymbol{v}$.

The velocity $\boldsymbol{v}$ must be relative to an inertial reference frame, that is, a frame that does not rotate and can either be stationary or moving at a constant velocity. Substituting for *B*sys and *b*

$$\frac{d(\mathbf{Mom}_{sys})}{dt} = \frac{d}{dt}\int_{CV} \mathbf{v}\rho d\forall + \int_{CS} \mathbf{v}\rho \;\; \boldsymbol{V} \;\; .d\boldsymbol{A}$$

Combining Eqs. above gives the *integral form of the momentum equation*:

$$\sum F = \frac{d}{dt}\int_{CV} \mathbf{v}\rho d\forall + \int_{CS} \mathbf{v}\rho \;\; \boldsymbol{V} \;\; .d\boldsymbol{A}$$

This equation can be expressed in words as

$$\begin{Bmatrix} sum\ of\ forces\ acting \\ on\ the\ matter\ on\ the \\ control\ volume \end{Bmatrix} = \begin{Bmatrix} time\ rate\ of\ change \\ of\ momentum\ in \\ control\ volume \end{Bmatrix} + \begin{Bmatrix} net\ outflow\ rate\ of \\ momentum\ through \\ control\ \ surface \end{Bmatrix}$$

It is important to remember that the momentum equation is a vector equation; that is, there is a direction associated with the each term in the equation.

If the flow crossing the control surface occurs through a series of inlet and outlet ports and if the velocity $\boldsymbol{v}$ is uniformly distributed across each port, then a simplified form of the Reynolds transport theorem, can be used, and the momentum equation becomes

$$\sum F = \frac{d}{dt}\int_{CV} \mathbf{v}\rho d\forall + \sum_{CS} \dot{m}_o\mathbf{v_o} - \sum_{CS} \dot{m}_i\mathbf{v_i}$$

where the subscripts *o* and *i* refer to the outlet and inlet ports, respectively. This form of the momentum equation will be identified as the *vector form*. Notice that the product of $\dot{m}\mathbf{v}$

corresponds to the mass per unit time times velocity, or momentum per unit time, which has the same units as force.

As long as $\boldsymbol{v}$ is uniformly distributed across control surface, Eq. above applies to any control volume, including one that is moving, deforming, or both. In all cases, is the rate at which mass is passing across the control surface, and $\boldsymbol{v}$ is velocity evaluated at the control surface with respect to the inertial reference frame that is selected.

$$x - direction : \sum F_x = \frac{d}{dt}\int_{CV} \boldsymbol{v_x}\rho d\forall + \sum_{CS} \dot{m}_o \boldsymbol{v_{ox}} - \sum_{CS} \dot{m}_i \boldsymbol{v_{ix}}$$

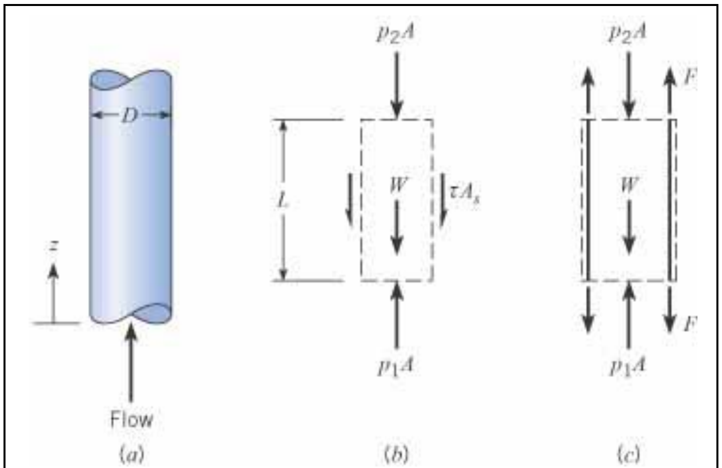
$$y - direction : \sum F_y = \frac{d}{dt}\int_{CV} \boldsymbol{v_y}\rho d\forall + \sum_{CS} \dot{m}_o \boldsymbol{v_{oy}} - \sum_{CS} \dot{m}_i \boldsymbol{v_{iy}}$$

$$z - direction : \sum F_z = \frac{d}{dt}\int_{CV} \boldsymbol{v_z}\rho d\forall + \sum_{CS} \dot{m}_o \boldsymbol{v_{oz}} - \sum_{CS} \dot{m}_i \boldsymbol{v_{iz}}$$

where the subscripts x, y, and z refer to the force and velocity components in the coordinate directions. These equations will be identified as the *component form of the momentum equation*. When velocity $\boldsymbol{v}$ varies across the control surface, the general form of the momentum equation must always be used.

6.2 Momentum Equation: Interpretation

Force Terms



Forces associated with flow in a pipe: (a) pipe schematic, (b) control volume situated inside the pipe, and (c) control volume surrounding the pipe.

Consider flow inside a vertical pipe (Fig a) One possible control volume is a cylinder with diameter D and length L located just inside the pipe wall. As shown in (Fig. b), the fluid within the control volume has been isolated from its surroundings, and the effect of the surroundings are shown as forces. The effect of the wall is replaced by a force equal to the shear stress (τ) times the pipe surface area ($A_s = \pi DL$). The force due to pressure is given by pressure (p) times the section area ($A=\pi D^2/4$) and always acts toward the control surface (a compressive force). The weight of the fluid is given by $W = \gamma(\pi D^2/4)L$. Thus, the net force acting in the z-direction is given by

$$\sum F_z = (p_1 - p_2)\frac{\pi}{4}\mathrm{D}^2 - \tau(\pi \mathrm{DL}) - \gamma\frac{\pi}{4}\mathrm{D}^2 L$$

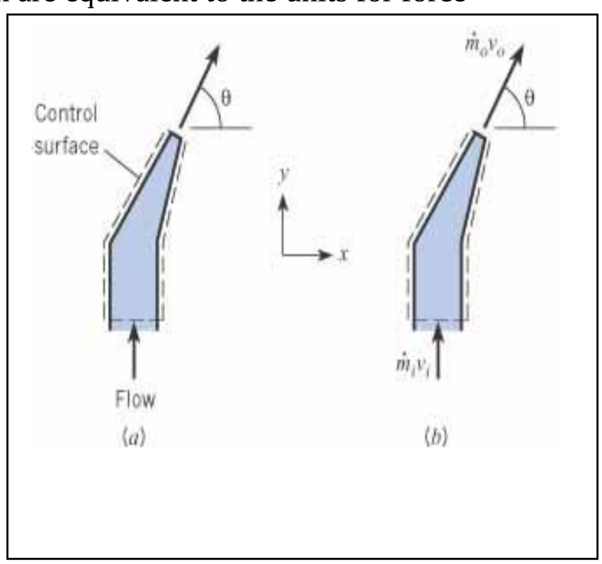
Another possible control volume has a length L and a diameter that is larger than the pipe's outside diameter. As shown in (Fig. *c)*, this control volume cuts through the pipe wall. Comparing (Figs. *b* and *c)* shows that the pressure forces are the same. However, in (Fig. c), there is no force associated with shear stress, but there are two new forces, $F1$ and $F2$, which represent the forces due to the pipe wall. Also, the weight of matter within the control volume now includes the weight of the fluid and the pipe wall (W_p). The net z-direction force is

$$\sum F_z = (p_1 - p_2)\frac{\pi}{4}\mathrm{D}^2 - \mathrm{F}_1 + \mathrm{F}_2 - (\mathrm{W_p} + \gamma\frac{\pi}{4}\mathrm{D}^2 L)$$

The choice of control volume depends on what information being sought.
To relate the pressure change between sections to wall shear stress?
To find the tensile force carried by the pipe wall?
The sketches shown in (Figs. b and c) are identified as *force diagrams* (FD). A force diagram shows the forces acting on the matter contained within a control volume. A force diagram is equivalent to a free-body diagram at the instant in time when the momentum equation is applied. In Fig. b, the force of gravity (weight) acts on each mass element in the control volume (with the resultant force acting at the mass center). A force that acts on mass elements within the body is defined as a *body force*. A body force can act at a distance without any physical contact. Examples of body forces include gravitational, electrostatic, and magnetic forces.
Except for the body force (weight), all forces shown in (Figs. b and c) are *surface forces*. A *surface force* is defined as a force that requires physical contact, meaning that surface forces act at the control surface.

Momentum Accumulation

The term $\frac{d}{dt}\int_{CV} \mathbf{v}\rho d\forall$ represents the rate at which the momentum of the material inside the control volume is changing with time. In particular, the mass of a volume element in the control volume is $\boldsymbol{\rho d}\forall$. so the product $\mathbf{v}\rho d\forall$ is the momentum of a volume element. Integrating over the control volume gives total momentum of the material in the control volume. Taking the time derivative gives the rate at which the momentum is changing. This term may be described as the net rate of momentum accumulation, and it will be referred to it as the *momentum accumulation* term. The units are momentum per unit time, which are equivalent to the units for force



In many problems, the momentum accumulation is zero. For example, consider steady flow through the control volume surrounding the nozzle shown in Fig. The fluid inside the control volume has momentum because it is moving. However, the velocity and density at each point do not change with time, so the total momentum in the control volume is constant, and the momentum accumulation term is zero. The evaluation of the momentum accumulation term is completed by considering the structural elements (i.e., the nozzle walls). Since the structural elements are stationary, there is no momentum change, so the momentum accumulation rate is zero.
In summary, the momentum of the material inside a control volume is evaluated by integrating the momentum of each volume element over the control volume. If the momentum in each differential volume is constant with time (e.g., steady flow, a stationary structural part), the momentum accumulation rate is zero.

Momentum Diagram

The momentum terms on the right side of momentum Eq. may be visualized with a *momentum diagram* (MD). The momentum diagram is created by sketching a control volume and then drawing a vector to represent the momentum accumulation term and a vector to represent momentum flow at each section where mass crosses the control surface.

Although the momentum diagram applies to the integral form of the momentum principle, the diagram takes on a simple form when the velocity **v** is uniformly distributed across each inlet and outlet port. For example, consider steady flow through the nozzle shown above. For the control volume indicated, the momentum accumulation term is zero, and this vector is omitted from the diagram. If the velocity is assumed to be uniform across the inlet and exit sections, the outlet momentum flow is given by $\dot{m}_o\mathbf{v_o}$and the inlet momentum flow is given by $\dot{m}_i\mathbf{v_i}$. To evaluate the momentum flow, one can use the diagram to see that

$$\sum_{cs}\dot{m}_o\mathbf{v_o} = (\dot{m}_o v_o cos\theta)\mathbf{i} + (\dot{m}_o v_o sin\theta)\mathbf{j}$$

And

$$\sum_{cs}\dot{m}_i\mathbf{v_i} = (\dot{m}_i v_i)\mathbf{j}$$

Recognizing that $\dot{m}_o = \dot{m}_i = \dot{m}$, the above equations can be combined to show that the net outward flow of momentum is

$$\sum_{cs}\dot{m}_o\mathbf{v_o} - \sum_{cs}\dot{m}_i\mathbf{v_i} = (\dot{m}v_o cos\theta)\mathbf{i} + (\dot{m}v_o sin\theta - \dot{m}v_i)\mathbf{j}$$

Systematic Approach

Problem Setup

• Select an appropriate control volume. Sketch the control volume and coordinate axes. Select an inertial reference frame.

• Identify governing equations. This will include either the vector or component form of the momentum equation. Other equations, such as the Bernoulli equation and/or the continuity equation, may be needed.

Force Analysis and Diagram

• Sketch body force(s) (usually only gravitational force) on the force diagram.

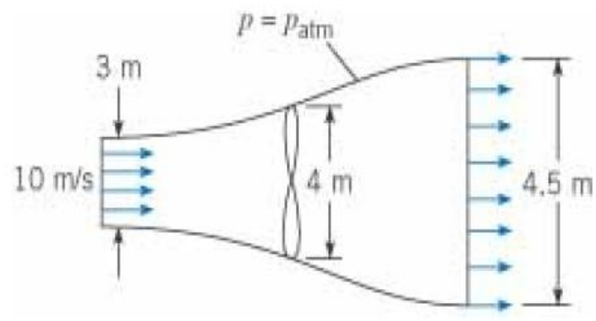
• Sketch surface forces on the force diagram; these are forces caused by pressure distribution, shear stress distribution, and supports and structures.

Momentum Analysis and Diagram

• Evaluate the momentum accumulation term. If the flow is steady and other materials in the control volume are stationary, the momentum accumulation is zero. Otherwise, the momentum accumulation term is evaluated by integration, and an appropriate vector is added to the momentum diagram.

• Sketch momentum flow vectors on the momentum diagram. For uniform velocity, each vector is$\dot{m}\boldsymbol{v}$.

Q6.77) A windmill is operating in a 10 m/s wind `that has a density of 1.2 kg/m^3. The diameter of the windmill is 4 m. The constant-pressure (atmospheric) streamline has a diameter of 3 m upstream of the windmill and 4.5 m downstream. Assume that the velocity distributions are uniform and the air is incompressible.
Determine the thrust on the mill.



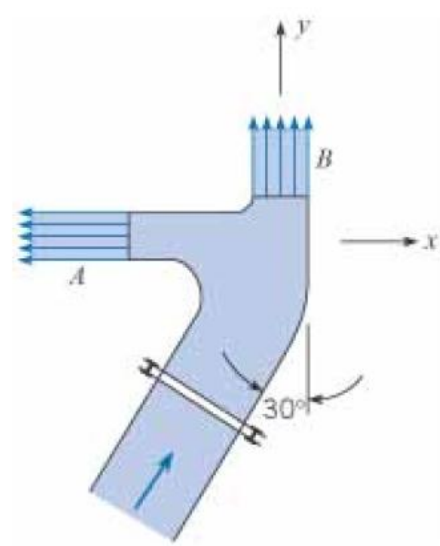
PROBLEM 6.77

Continuity principle
$Q_1=Q_2$ since density is constant
$V_1 A_1=V_2 A_2$
$V_2= 10 \times (3/4.5)^2= 4.44$ m/s

<u>Momentum principle</u> (x-direction)

$$\begin{aligned} \sum F_x &= \dot{m}(v_2 - v_1) \\ F_x &= \dot{m}(v_2 - v_1) \\ &= (1.2)(\pi/4 \times 3^3)(10)(4.44 - 10) \\ F_x &= -472.0 \text{ N (acting to the left)} \\ & \boxed{T = 472 \text{ N (acting to the right)}} \end{aligned}$$

Q6.64) This “double” nozzle discharges water (at 10°C) into the atmosphere at a rate of 0.50 m^3/s. If the nozzle is lying in a horizontal plane, what *x*-component of force acting through the flange bolts is required to holdthe nozzle in place? *Note:* Assume irrotational flow, and assume the water speed in each jet to be the same.
Jet *A* is 10 cm in diameter, jet *B* is 12 cm in diameter, and the pipe is 30 cm in diameter.



PROBLEMS 6.63, 6.64

solution
$V_A = V_B$ **given**
$Q_{total} = Q_A + Q_B$ for incompressible flow
$Q_{total} = V_A A_A + V_B A_B$
$V_A = V_B = Q_{total}/(A_A + A_B)$
$= 0.5/(\pi \times 0.05 \times 0.05 + \pi \times 0.06 \times 0.06) = 26.1$ m/s
$V_1 = 0.5/(\pi \times 0.15 \times 0.15) = 7.07$ m/s

Bernoulli equation {between 1 & A [same as 1 & B] }
$p_1 = (1000/2)(26.12 - 7.072) = 315, 612$ Pa

Momentum principle (x-direction)

$$\sum F_x = \dot{m}_o v_{ox} - m_i v_{ix}$$

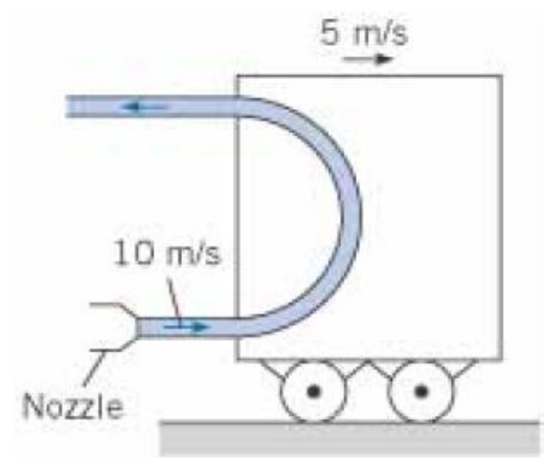
$$F_x + p_1 A_1 \sin 30 = -\dot{m} v_A - \dot{m} v_i \sin 30$$

$$F_x = -315,612 \times \pi \times 0.15^2 \times \sin 30^\circ - 26.1 \times 1,000 \times 26.1$$

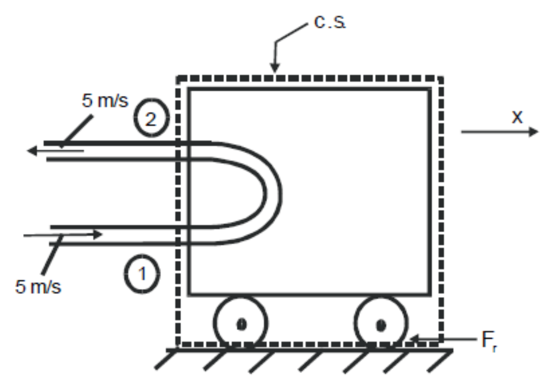
$$\times \pi \times 0.05^2 - 7.07 \times 1000 \times 0.5 \sin 30^\circ = -18,270 \text{ N} = \boxed{\text{-18.27 kN}}$$

Q 6.86

A cart is moving along a track at a constant velocity of 5 m/s as shown. Water (ρ = 1000 kg/m3) issues from a nozzle at 10 m/s and is deflected through 180° by a vane on the cart. The cross-sectional area of the nozzle is 0.0012 m2. Calculate the resistive force on the cart.



PROBLEM 6.86



Velocity analysis

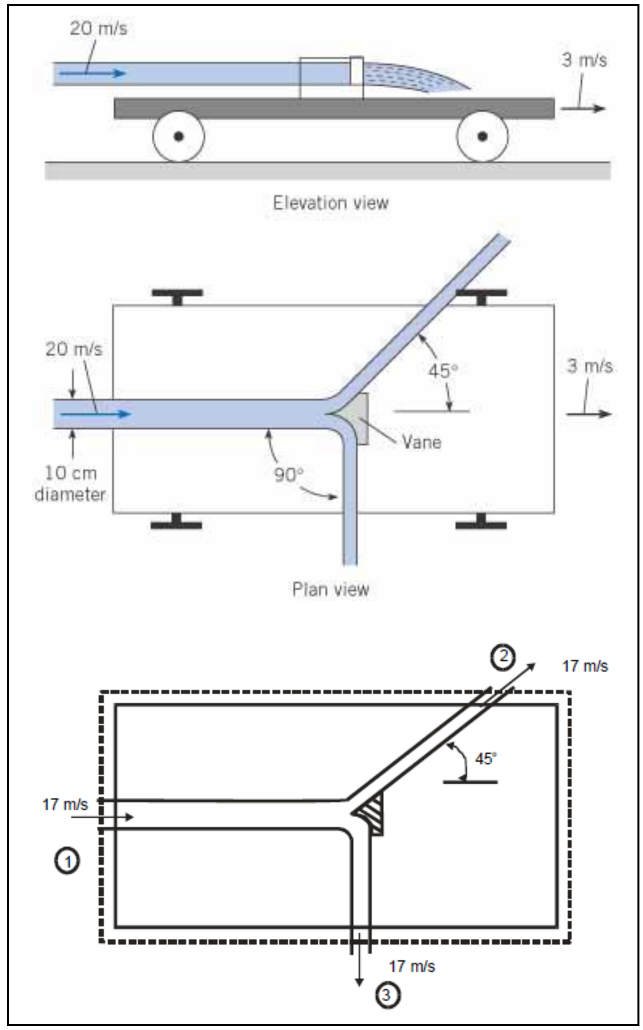
$$\begin{aligned} V_1 &= v_1 = v_2 = 5 \text{ m/s} \\ \dot{m} &= pA_1V_1 \\ &= (1000)(0.0012)(5) \\ &= 6 \text{ kg/s} \end{aligned}$$

Momentum principle (x-direction)

$$\begin{aligned} \sum F_x &= \dot{m}(v_2 - v_1) \\ -F_r &= 6(-5-5) = -60 \text{ N} \end{aligned}$$

$$\boxed{F_r = 60 \text{ N (acting to the left)}}$$

6.30 A vane on this moving cart deflects a 10 cm water (ρ = 1000 kg/m3) jet as shown. The initial speed of the water in the jet is 20 m/s, and the cart moves at a speed of 3 m/s. If the vane splits the jet so that half goes one way and half the other, what force is exerted on the vane by the jet?



Momentum principle (x-direction)

$$F_x = \dot{m}_2 v_{2x} - \dot{m}_1 v_1$$
$$F_x = (17^2 \cos 45^\circ)(1000)(\pi/4)(0.1^2)/2 - (17)(1000)(17)(\pi/4)(0.1^2)$$
$$= +802 - 2270 = -1470 \text{ N}$$

Momentum principle (y-direction)

$$F_y = \dot{m}_2 v_{2y} - \dot{m} v_{3y}$$
$$= (17)(1,000)(\sin 45^\circ)(17)(\pi/4)(0.1^2)/2 - (17)^2(1000)(\pi/4)(0.1^2)/2$$
$$= -333 \text{ N}$$

$$\boxed{\text{F(water on vane)} = (1470\text{i} + 333\text{j}) \text{ N}}$$

Equation of motion of a rocket

$$\sum F_z = p_e A_e - W - D$$

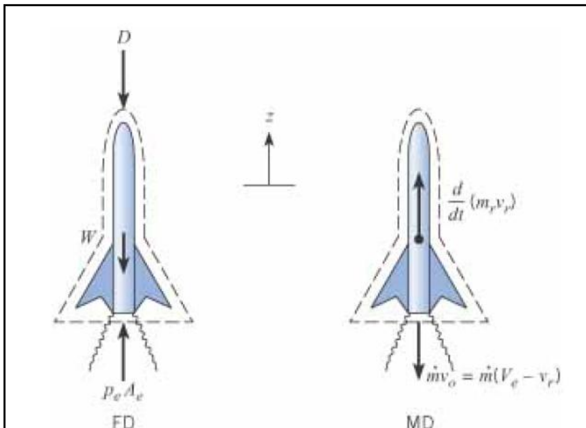
$$m_r \frac{dv_r}{dt} = T - D - W$$

T: thrust of the rocket, the sum of the momentum outflow and the pressure force at the nozzle exit. Neglecting the drag and weight, the equation of motion reduces to

$$T = m_r \frac{dv_r}{dt}$$

$$\dot{m}_r = m_i - \dot{m} \ \ t$$

where m_i is the initial rocket mass and t is the time from ignition



Force and momentum diagrams for rocket.

$$v_{bo} = \frac{T}{\dot{m}} \ln \frac{m_i}{m_f}$$

Where $\boldsymbol{v}_{bo}$ is the burnout velocity and mf is the final (or payload) mass. The ratio $T/\dot{m}_i$is known as the specific impulse, I_{sp}, and has units of velocity.

Q6.89 It is common practice in rocket trajectory analyses to neglect the body-force term and drag, so the velocity at burnout is given by $v_{bo} = \frac{T}{\lambda}\ln(M_i/M_f)$

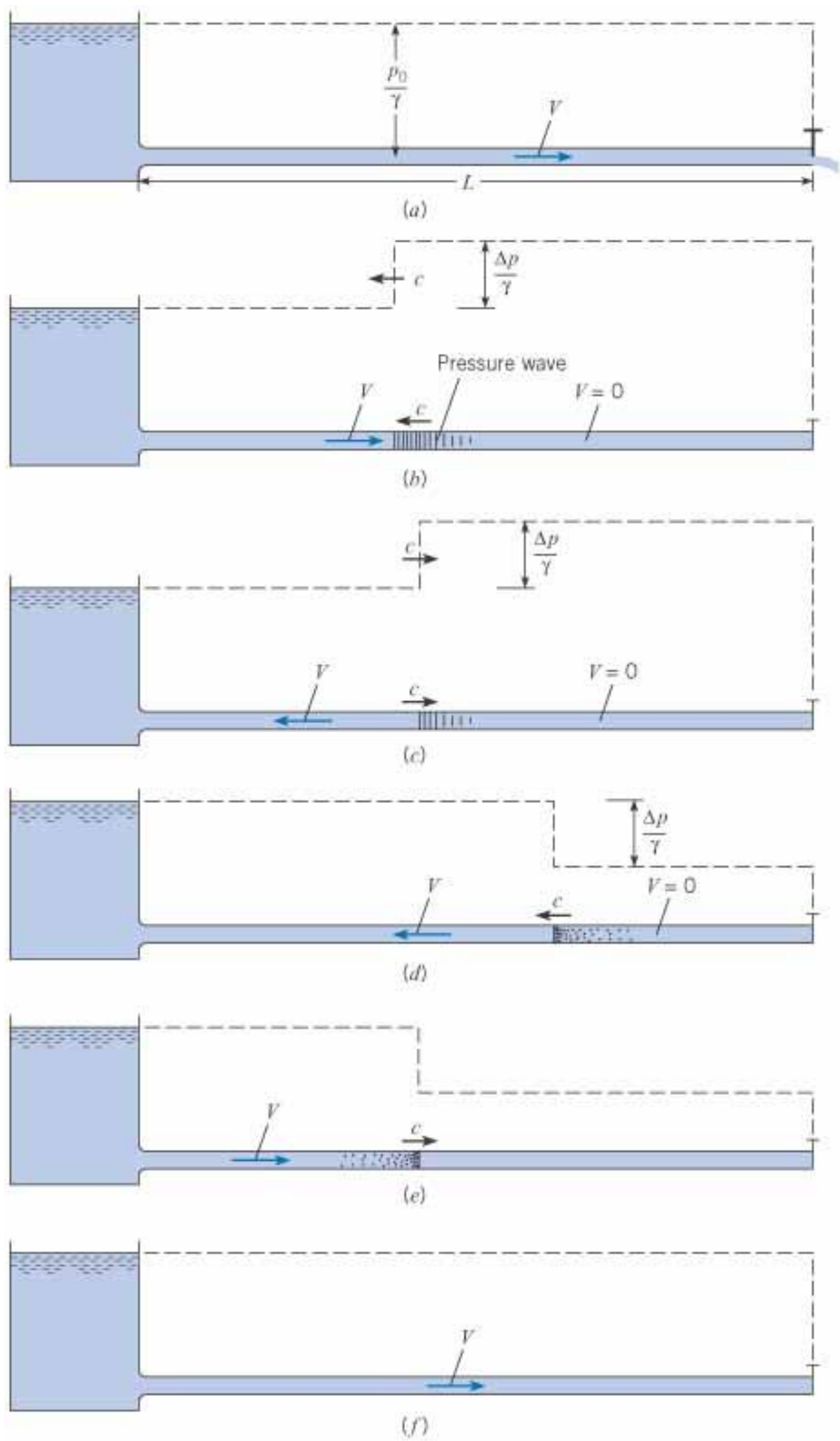
Assuming a thrust-to-mass-flow ratio of 3000 N.s/kg and a final mass of 50 kg, calculate the initial mass needed to establish the rocket in an earth orbit at a velocity of 7200 m/s.

$$v_{bo} = \frac{T}{\lambda}\ln(M_i/M_f)$$
$$7200 = 3000\ln(M_i/50)$$
$$\ln(M_i/50) = 7200/3000 = 72/30 = 2.4$$
$$\frac{M_i}{50} = \exp(2.4)$$
$$M_i = 50\exp(2.4) = 550.2\ kg$$

Water Hammer: Physical Description

Whenever a valve is closed in a pipe, a positive pressure wave is created upstream of the valve and travels up the pipe at the speed of sound. If the pressure is greater than the existing steady-state pressure. This pressure wave may be great enough to cause pipe failure. Therefore, a basic understanding of this process, which is called *water hammer,* is necessary for the proper design and operation of such systems.

Consider flow in the pipe shown in Fig. 6.7. Initially the valve at the end of the pipe is only partially open (Fig. 6.7*a*); consequently, an initial velocity *V* and initial pressure p_0 exist in the pipe. At time $t = 0$ it is assumed that the valve is instantaneously closed, thus creating a pressure increase behind the valve and a pressure wave that travels from the valve toward the reservoir at the speed of sound, *c*. All the water between the pressure wave and the upper end of the pipe will have the initial velocity *V*, but all the water on the other side of the pressure wave (between the wave and the valve) will be at rest. This condition is shown in Fig. 6.7*b*. Once the pressure wave reaches the upper end of the pipe (after time $t = L/c$), it can be visualized that all of the water in the pipe will be under a pressure $p_0 + \Delta p$; however, the pressure in the reservoir at the end of the pipe is only p_0. This imbalance of pressure at the reservoir end causes the water to flow from the pipe back into the reservoir with a velocity *V*. Thus a new pressure wave is formed that travels toward the valve end of the pipe (Fig. 6.7*c*), and the pressure on the reservoir side of the wave is reduced to p_0. When this wave finally reaches the valve, all the water in the pipe is flowing toward the reservoir with a velocity *V*. This condition is only momentary, however, because the closed valve prevents any sustained flow.



Water hammer process.
(a) Initial condition. *(b) Condition during time $0 < t < L/c$.*
(c) Condition during time $L/c < t < 2L/c$. *(d) Condition during time $2L/c < t < 3L/c$.*
(e) Condition during time $3L/c < t < 4L/c$.

Next, during time $2L/c < t < 3L/c$, a rarefied wave of pressure ($p < p0$) travels up to the reservoir, as shown in Fig. 6.7*d*. When the wave reaches the reservoir, all the water in the pipe has a pressure less than that in the reservoir. This imbalance of pressure causes flow to be established again in the entire pipe, as shown in Fig. 6.7*f*, and the condition is exactly the same as in the initial condition (Fig. 6.7*a*). Hence the process will repeat itself in a periodic manner.
From this description, it may be seen that the pressure in the pipe immediately upstream of the valve will be alternately high and low, as shown in Fig. 6.7*a.* A similar observation for the pressure at the midpoint of the pipe reveals a more complex variation of pressure with time, as shown in Fig. 6.8*b*. Obviously, a valve cannot be closed instantaneously, and viscous effects, which were neglected here, will have a damping effect on the process. Therefore, a more realistic pressure–time trace for the point just upstream of the valve is given in Fig.
6.8*c*. The finite time of closure erases the sharp discontinuities in the pressure trace that were present in Fig. 6.8*a.* However, it should be noted that the maximum pressure developed at the valve will be virtually the same as for instantaneous closure if the time of closure is less than $2L/c$ That is, the change in pressure will be the same for a given change in velocity unless the negative wave from the reservoir mitigates the positive pressure, and it takes a time $2L/c$ before this negative wave can reach the valve. The value $2L/c$ is called the *critical time of closure* and is given the symbol *tc*.

Chapter 7 The Energy Equation

7.1 Energy, Work, and Power

When matter has *energy*, the matter can be used to do work. A fluid can have several forms of energy. For example a fluid jet has kinetic energy, water behind a dam has gravitational potential energy, and hot steam has thermal energy.

Work is force acting through a distance when the force is parallel to the direction of motion

Work = Force x distance =Torque x Angular displacement

Machine is any device that transmits or modifies energy, typically to perform or assist in a human task.

a *turbine* is a machine that is used to extract energy from a flowing fluid.

a *pump* is a machine that is used to provide energy to a flowing fluid.

Work and energy both have the same primary dimensions, and the same units, and both characterize an amount or quantity

Power: Amount of work per unite time.

$$P = \lim_{\Delta t \to 0} \frac{\Delta W}{\Delta t} = \dot{W}$$

$P = \lim_{\Delta t \to 0} \frac{F \Delta x}{\Delta t} = FV \ , for\ moving\ body$, V: velocity

$P = \lim_{\Delta t \to 0} \frac{T \Delta \theta}{\Delta t} = T\omega \ \ , for\ rotating\ shaft$, ω: Angular speed

$$P = \dot{W} = FV = T\omega$$

Watt, kWatt, Horsepower

1 Horsepower (Hp) = 746 Watt or 1 Kw = 1.34 Hp

7.2 Energy Equation: General Form

$$\dot{Q} - \dot{W} = \frac{dE}{dt}$$

$$\left\{\begin{matrix} Net\ rate\ of \\ Thermal\ Energy \\ entering\ system \end{matrix}\right\} - \left\{\begin{matrix} Net\ rate\ at\ which \\ system\ works\ on \\ environment \end{matrix}\right\} = \left\{\begin{matrix} rate\ of\ change \\ of\ Energy\ of\ the \\ matter\ within\ system \end{matrix}\right\}$$

$$\dot{Q} - \dot{W} = \frac{d}{dt} \int_{CV} \left(\frac{V^2}{2} + gz + u\right) \rho \ \ d\forall + \int_{CS} \left(\frac{V^2}{2} + gz + u\right) \rho \ \ \boldsymbol{V} \ \ .d\boldsymbol{A}$$

u: internal energy

Shaft and Flow Work

Work = flow work + shaft work

$$\dot{W}_{shaft} = \dot{W}_{Turbine} - \dot{W}_{pump} = \dot{W}_t - \dot{W}_p$$

$$\dot{W}_{flow} = \int_{CS} \left(\frac{p}{\rho}\right) \rho \ \ \boldsymbol{V} \ \ .d\boldsymbol{A}$$

$$\dot{Q} - \dot{W} = \frac{d}{dt}\int_{CV}\left(\frac{V^2}{2} + gz + u\right)\rho \quad d\forall + \sum_{CS}\dot{m}_o\left(\frac{V_o^2}{2} + gz_o + h_o\right) - \sum_{CS}\dot{m}_i\left(\frac{V_i^2}{2} + gz_i + h_i\right)$$

Where h = u+p/ρ : enthalpy

7.3 Energy Equation: Pipe Flow

Kinetic Energy Correction Factor

$$\alpha = \frac{\text{actual KE / time that crosses a section}}{\text{KE / time by assuming a uniform velocity distribution}} = \frac{\int_A \frac{\rho V^3 dA}{2}}{\frac{\rho \bar{V}^3 A}{2}}$$

$$\alpha = \frac{1}{A}\int_A \left(\frac{V}{\bar{V}}\right)^3 dA$$

In most cases, α takes on a value of 1 or 2. When the velocity profile in a pipe is uniformly distributed, then α = 1. When flow is laminar, the velocity distribution is parabolic and α = 2. When flow is turbulent, the velocity profile is plug-like and α ≈ 1.05. For turbulent flow it is common practice to let α = 1.

A Simplified Form of the Energy Equation

$$\left(\frac{p_1}{\gamma} + z_1 + \alpha_1\frac{\bar{V}_1^2}{2g}\right) + h_p = \left(\frac{p_2}{\gamma} + z_2 + \alpha_2\frac{\bar{V}_2^2}{2g}\right) + h_t + h_L$$

$$h_p, h_t, h_L: pump\ head, turbine\ head\ and\ \ losses\ head\ respectively.$$

$$\begin{pmatrix}\text{head}\\ \text{carried by}\\ \text{flow into the cv}\end{pmatrix} + \begin{pmatrix}\text{head}\\ \text{added by}\\ \text{pumps}\end{pmatrix} = \begin{pmatrix}\text{head}\\ \text{carried by}\\ \text{flow out of the cv}\end{pmatrix} + \begin{pmatrix}\text{head}\\ \text{extracted by}\\ \text{turbines}\end{pmatrix} + \begin{pmatrix}\text{head}\\ \text{loss due to}\\ \text{viscous effects}\end{pmatrix}$$

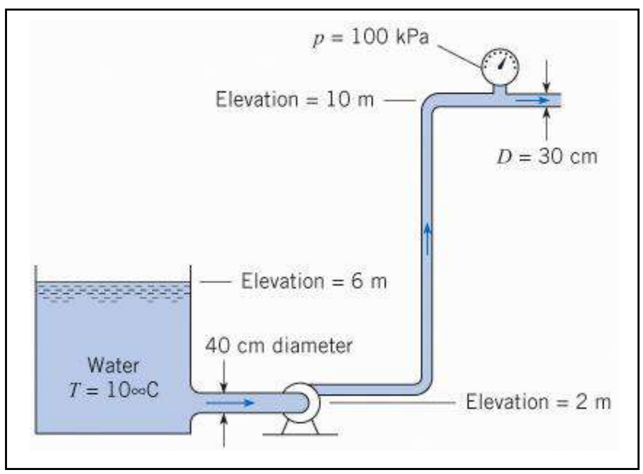
$$\text{head} \approx \frac{\text{energy / time or work / time}}{\text{weight / time of flowing fluid}}$$

$$h_p = \frac{\dot{W}_p}{\dot{m}g}, h_t = \frac{\dot{W}_t}{\dot{m}g}$$

$$h_L = \frac{f(L/D)V^2}{2g}, f: friction\ factor$$

Example

Water (10°C) is flowing at a rate of 0.35 m3/s, and it is assumed that hL= $2V^2/2g$ **from the reservoir to the gage, where V is the velocity in the 30-cm pipe.** What power must the pump supply? Assume α= 1.0 at all locations



$$\begin{aligned} V &= \frac{Q}{A} \\ &= \frac{0.35}{(\pi/4)\times(0.3\,\mathrm{m})^2} \\ &= 4.95\ \mathrm{m/s} \\ \frac{V_2^2}{2g} &= 1.250\ \mathrm{m} \end{aligned}$$

Energy equation (locate 1 on the reservoir surface; locate 2 at the pressure gage)

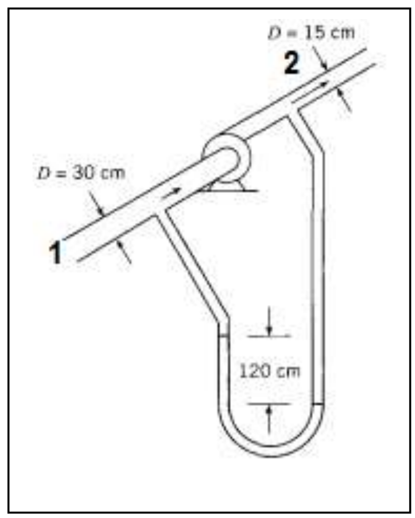
$$\begin{aligned} 0+0+6\,\mathrm{m}+h_p &= \frac{100000\,\mathrm{Pa}}{9810\,\mathrm{N/m^3}} + 1.25\,\mathrm{m} + 10\,\mathrm{m} + 2.0\,(1.25\,\mathrm{m}) \\ h_p &= 17.94\,\mathrm{m} \end{aligned}$$

Power equation:

$$\begin{aligned} P &= Q\gamma h_p \\ &= \left(0.35\,\mathrm{m^3/s}\right)\left(9810\,\mathrm{N/m^3}\right)(17.94\,\mathrm{m}) \\ & \boxed{P = 61.6\,\mathrm{kW}} \end{aligned}$$

Example

In the pump test shown, the rate of flow is 0.16 m^3/s of oil (S = 0.88). Calculate the horsepower that the pump supplies to the oil if there is a differential reading of 120 cm of mercury in the U-tube manometer. Assume α= 1.0 at all locations.



Solution

$$V_1 = \frac{Q}{A_1} = \frac{0.16}{\left(\frac{\pi}{4}\right)0.3^2} = 2.26$$

Similarly

$$V_2 = \frac{Q}{A_2} = \frac{0.16}{\left(\frac{\pi}{4}\right)0.15^2} = 9.04$$

Manometer equation

$$\left(\frac{p_2}{\gamma}+z_2\right)-\left(\frac{p_1}{\gamma}+z_1\right) = \frac{h_m(S_m-S_o)}{S_o} = \frac{1.2(13.76-0.88)}{0.88} = 17.5636\,m$$

Energy equation reduces to

$$\left(\frac{p_1}{\gamma}+z_1+\alpha_1\frac{\bar{V}_1^2}{2g}\right)+h_p = \left(\frac{p_2}{\gamma}+z_2+\alpha_2\frac{\bar{V}_2^2}{2g}\right)$$

$$h_p = \left(\frac{p_2}{\gamma}+z_2\right)-\left(\frac{p_1}{\gamma}+z_1\right)+\left(\frac{\bar{V}_2^2}{2g}-\frac{\bar{V}_1^2}{2g}\right) = \left(17.5636+\frac{(9.04)^2-(2.26)^2}{2*9.81}\right) = 21.5m$$

$$Power = h_p Q\gamma = 21.5*0.16*9.81*880 = \mathbf{29.7kW}$$

Power = 39.81 HP

7.4 Power Equation

$$Power = h \quad Q\gamma$$
$$Turbine\ Power = \dot{W}_T = h_T Q\gamma$$
$$Pump\ Power = \dot{W}_p = h_p Q\gamma$$

Efficiency the ratio of power output to power input

$$\eta = \frac{power\ out\ from\ a\ machine\ \text{or system}}{power\ input\ to\ a\ machine\ \text{or system}} = \frac{P_{out}}{P_{in}}$$

Mechanical efficiency of the pump is η_p, the power output delivered by the pump to the flow is

$$\dot{W}_p = \eta_p \dot{W}_s$$

Where $\dot{W}_s$ power supplied to pump, usually by a rotating shaft that is connected to a motor.
For a turbine, the output power $\dot{W}_s$ is usually delivered by a rotating shaft to a generator
Mechanical efficiency of the turbine is η_T, the output power supplied by the turbine is

$$\dot{W}_s = \eta_T \dot{W}_T$$

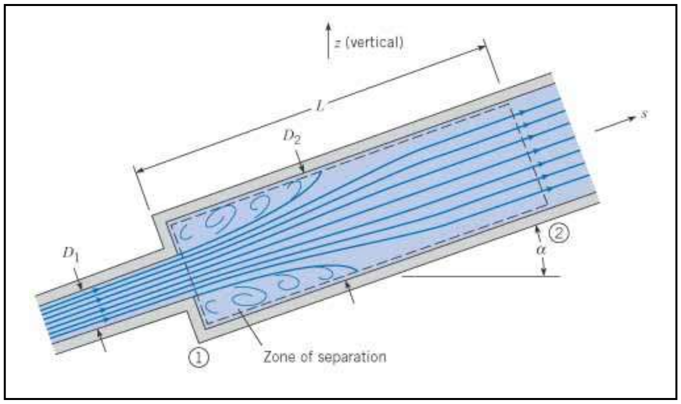
where $\dot{W}_T$is the power input to the turbine from the flow.

7.5 Contrasting the Bernoulli Equation and the Energy Equation

The Bernoulli equation and the energy equation are derived in different ways.
The Bernoulli equation was derived by applying Newton's second law to a particle and then integrating the resulting equation along a streamline. The energy equation was derived by starting with the first law of thermodynamics and then using the Reynolds transport theorem. Consequently, the Bernoulli equation involves only mechanical energy, whereas the energy equation includes both mechanical and thermal energy.
The Bernoulli equation is applied by selecting two points on a streamline and then equating terms at these points:
In addition, these two points can be anywhere in the flow field for the special case of irrotational flow. The energy equation is applied by selecting an inlet section and an outlet section in a pipe and then equating terms as they apply to the pipe.
The two equations have different assumptions. The Bernoulli equation applies to steady, incompressible, and inviscid flow. The energy equation applies to steady, viscous, incompressible flow in a pipe with additional energy being added through a pump or extracted through a turbine.
Under special circumstances the energy equation can be reduced to the Bernoulli equation. If the flow is inviscid, there is no head loss; that is, $h_L = 0$. If the "pipe" is regarded as a small stream tube enclosing a streamline, then $\alpha = 1$. There is no pump or turbine along a streamline, so $h_p = h_t = 0$. In this case the energy equation is identical to the Bernoulli equation. Note that the energy equation cannot be developed starting with the Bernoulli equation.

7.6 Transitions
Abrupt Expansion

An *abrupt or sudden expansion* in a pipe or duct is a change from a smaller section area to a larger section area.



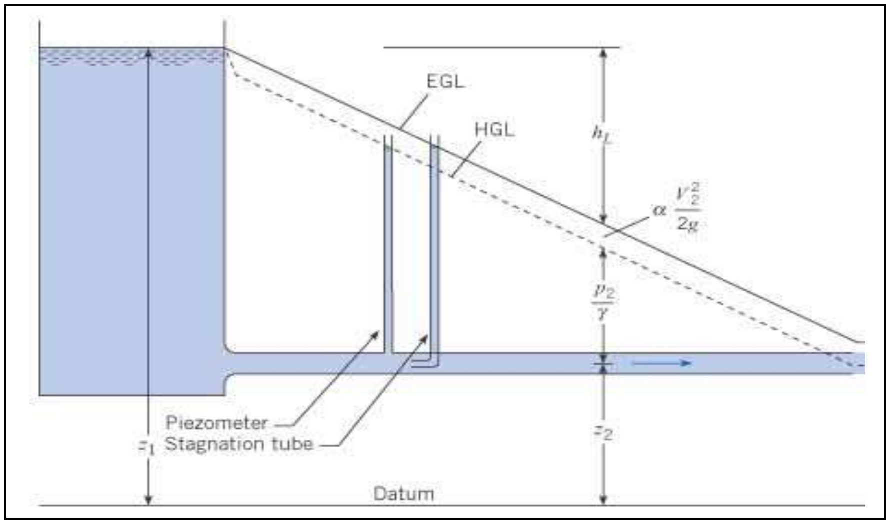
7.7 Hydraulic and Energy Grade Lines

$$\text{EGL} = \begin{pmatrix}\text{velocity}\\ \text{head}\end{pmatrix} + \begin{pmatrix}\text{pressure}\\ \text{head}\end{pmatrix} + \begin{pmatrix}\text{elevation}\\ \text{head}\end{pmatrix}$$

$$\text{EGL} = \alpha \frac{V^2}{2g} + \frac{P}{\gamma} + z = \begin{pmatrix}\text{total}\\ \text{head}\end{pmatrix}$$

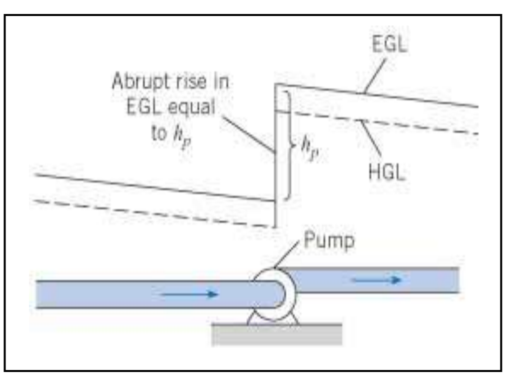
$$\text{HGL} = \begin{pmatrix}\text{pressure}\\ \text{head}\end{pmatrix} + \begin{pmatrix}\text{elevation}\\ \text{head}\end{pmatrix}$$

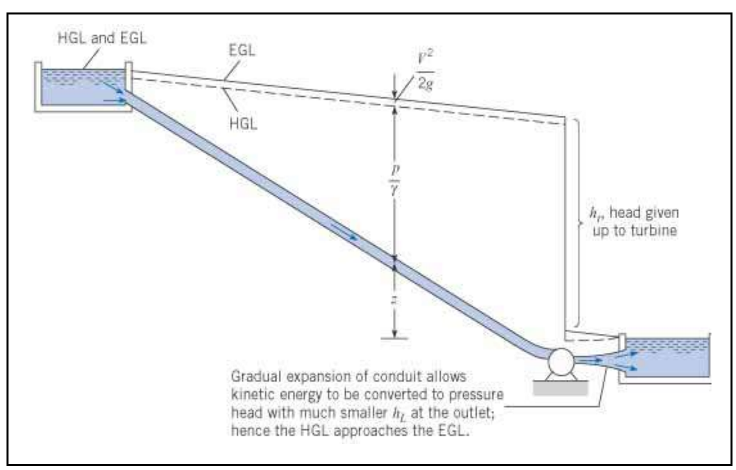
$$\text{HGL} = \frac{P}{\gamma} + z = \begin{pmatrix}\text{piezometric}\\ \text{head}\end{pmatrix}$$

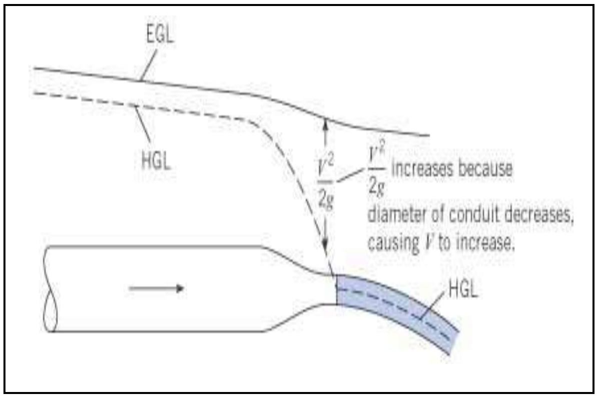


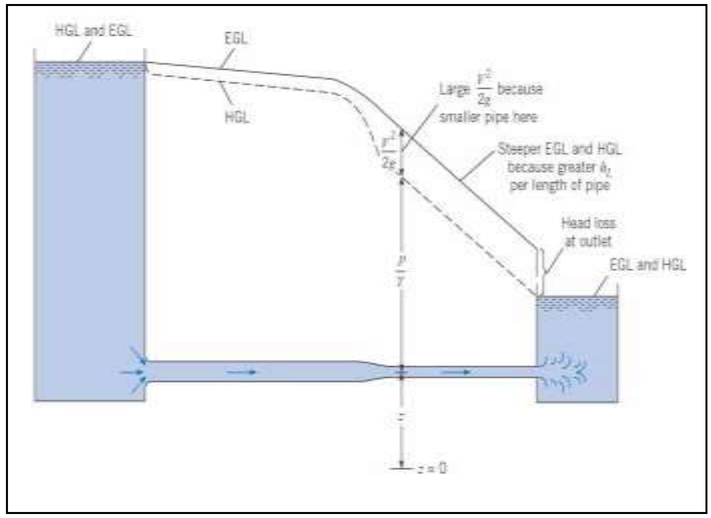
Tips for Drawing HGLs and EGLs

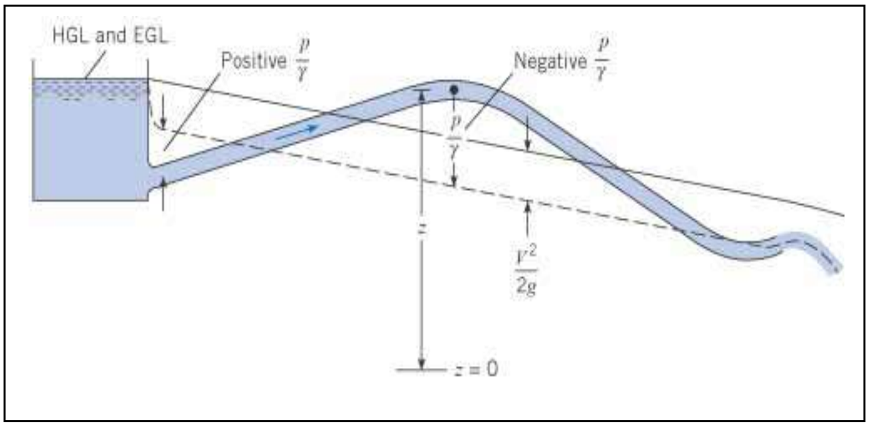
1. In a lake or reservoir, the HGL and EGL will coincide with the liquid surface. Also, both the HGL and EGL will indicate piezometric head. For example, see Fig. 7.7.
2. A pump causes an abrupt rise in the EGL and HGL by adding energy to the flow. For example, see Fig. 7.8.
3. For steady flow in a Pipe of constant diameter and wall roughness, the slope (7*hL*/7*L*) of the EGL and the HGL will be constant. For example, see Fig. 7.7
4. Locate the HGL below the EGL by a distance of the velocity head (α*V*2/2*g*).
5. Height of the EGL decreases in the flow direction unless a pump is present.
6. A turbine causes an abrupt drop in the EGL and HGL by removing energy from the flow. For example, see Fig. 7.9.
7. Power generated by a turbine can be increased by using a gradual expansion at the turbine outlet. As shown in Fig. 7.9, the expansion converts kinetic energy to pressure. If the outlet to a reservoir is an abrupt expansion, as in Fig. 7.11, this kinetic energy is lost.
8. When a pipe discharges into the atmosphere the HGL is coincident with the system because $p/\gamma = 0$ at these points. For example, in Figures 7.10 and 7.12, the HGL in the liquid jet is drawn through the jet itself.
9. When a flow passage changes diameter, the distance between the EGL and the HGL will change (see Fig.7.10 and Fig. 7.11) because velocity changes. In addition, the slope on the EGL will change because the head loss per length will be larger in the conduit with the larger velocity
10. If the HGL falls below the pipe, then p/γ is negative, indicating subatmospheric pressure (see Fig. 7.12) and a potential location of cavitation.






EGL
HGL
$\frac{V^2}{2g}$
$\frac{V^2}{2g}$ increases because diameter of conduit decreases, causing V to increase.
HGL


HGL and EGL
EGL
HGL
Large $\frac{V^2}{2g}$ because smaller pipe here
$\frac{V^2}{2g}$
Steeper EGL and HGL because greater h_L per length of pipe
Head loss at outlet
EGL and HGL
$\frac{p}{\gamma}$
z
$z = 0$


HGL and EGL
Positive $\frac{p}{\gamma}$
Negative $\frac{p}{\gamma}$
$\frac{p}{\gamma}$
z
$\frac{V^2}{2g}$
$z = 0$

Chapter 8 Dimensional Analysis

enormous savings in time and money

$F = f(L,U,\rho,\mu)$, find dimensional groups using Pi-therorem.

Solution

Write the function and count variables:

$F = f(L,U,\rho,\mu)$

There are five variables (n= 5)

List dimensions of each variable.

F	L	U	ρ	μ
$\{MLT^{-2}\}$	$\{L\}$	$\{LT^{-1}\}$	$\{ML^{-3}\}$	$\{ML^{-1}T^{-1}\}$

Find j. No variable contains the dimension θ, and so j is less than or equal to 3 (*MLT*). We inspect the list and see that L, U, and ρ cannot form a pi group because only ρ contains mass and only U contains time. **Therefore *j* does equal 3, and *n* - *j* =5 -3 =2 = *k*.**

The pi theorem guarantees for this problem ➔ two independent dimensionless groups.

Select repeating j variables. The group L, U, ρ we found in step 3 will do fine.

Combine L, U, ρ with one additional variable, in sequence, to find the two pi products.

First add force to find Π_1. You may select *any* exponent on this additional term as you please, to place it in the numerator or denominator to any power. Since F is the output, or dependent variable, we select it to appear to the first power in the numerator:

$$\Pi_1 = L^a U^b \rho^c F = L^a (LT^{-1})^b (MT^{-3})^c (MLT^{-2}) = M^0 L^0 T^0$$

Equate exponents:

Length: $a + b - 3c + 1 = 0$

Mass: $c + 1 = 0$

Time: $-b - 2 = 0$

We can solve explicitly for $b=-2$, $c=-1$ *and* $a=-2$

Therefore

$$\Pi_1 = L^{-2} U^{-2} \rho^{-1} F = \frac{F}{\rho L^2 U^2} = C_f$$

Finally, add viscosity to L, U, and ρ to find Π_2

$$\Pi_2 = L^a U^b \rho^c \mu = L^a (LT^{-1})^b (MT^{-3})^c (ML^{-1}T^{-1}) = M^0 L^0 T^0$$

a+b-3c-1=0

c+1=0

-2b -1 =0

a=b=c=-1

$$\Pi_2 = \frac{\mu}{L^1 U^1 \rho^1}$$

Or

$$\Pi_2 = \frac{\rho U L}{\mu} = Re$$

$$\frac{F}{\rho L^2 U^2} = f\left(\frac{\rho U L}{\mu}\right)$$

Example

At low velocities (laminar flow), the volume flow *Q* through a small-bore tube is a function only of the tube radius *R*, the fluid viscosity μ, and the pressure drop per unit tube length *dp*/*dx*. Using the pi theorem, find an appropriate dimensionless relationship.

Solution

Write the given relation and count variables:

$$Q = f(R, \mu, dp/dx)$$

Four variables (*n* = 4)

Q	R	μ	dp/dx
L^3T^{-1}	L	$ML^{-1}T^{-1}$	$ML^{-2}T^{-2}$

There are three primary dimensions (*M*, *L*, *T*), hence *j* ≤3.

By trial and error we determine that *R*, μ, and *dp*/*dx* cannot be combined into a pi group. Then *j*=3, and *n* - *j* = 4 - 3 =1.

There is only *one* pi group, which we find by combining *Q* in a power product with the other three

$$\Pi_1 = R^a\mu^b\left(\frac{dp}{dx}\right)^c Q = L^a(ML^{-1}T^{-1})^b(ML^{-2}T^{-2})^c(L^3T^{-1}) = M^0L^0T^0$$

Equate exponents:

Mass : b + c = 0

Length : a – b - 2c + 3 = 0

Time : – b - 2c – 1 = 0

Solving simultaneously, we obtain *a*=-4, *b*= 1, *c* =-1. Then

$$\Pi_1 = R^{-4}\mu^1\left(\frac{dp}{dx}\right)^{-1} Q$$

$$\Pi_1 = \frac{\mu Q}{R^4\left(\frac{dp}{dx}\right)} = \text{constant}$$

8.4 Common π-Groups

Pressure coefficient

$$C_p = \frac{p - p_0}{\frac{1}{2}\rho U^2}$$

Shear stress coefficient “friction”

$$C_f = \frac{\tau}{\frac{1}{2}\rho U^2}$$

Force coefficient

$$C_F = \frac{F}{\frac{1}{2}\rho U^2 L^2}$$

F =Drag ➔ Drag coefficient , F=Lift ➔ Lift coefficient

The general functional form for all the π-groups is $C_p, C_f, C_F = f(Re, M, We, Fr)$

$$Re = \frac{\rho UL}{\mu}, Fr = \frac{U}{\sqrt{gL}}, M = \frac{U}{\sqrt{E_v/\rho}}, We = \frac{\rho LU^2}{\sigma}$$

Table 8.3 COMMON Π-GROUPS

π-Group	Symbol	Name	Ratio
$\frac{p-p_0}{(\rho V^2)/2}$	C_p	Pressure coefficient	$\frac{\text{Pressure differences}}{\text{Kinetic pressure}}$
$\frac{\tau}{(\rho V^2)/2}$	c_f	Shear-stress coefficient	$\frac{\text{Shear stress}}{\text{Kinetic pressure}}$
$\frac{F}{(\rho V^2 L^2)/2}$	C_F	Force coefficient	$\frac{\text{Force}}{\text{Kinetic force}}$
$\frac{\rho L V}{\mu}$	Re	Reynolds number	$\frac{\text{Kinetic force}}{\text{Viscous force}}$
$\frac{V}{c}$	M	Mach number	$\frac{\text{Kinetic force}}{\text{Compressive force}}$
$\frac{\rho L V^2}{\sigma}$	We	Weber number	$\frac{\text{Kinetic force}}{\text{Surface-tension force}}$
$\frac{V}{\sqrt{gL}}$	Fr	Froude number	$\frac{\text{Kinetic force}}{\text{Gravitational force}}$

8.5 Similitude

Geometric Similitude

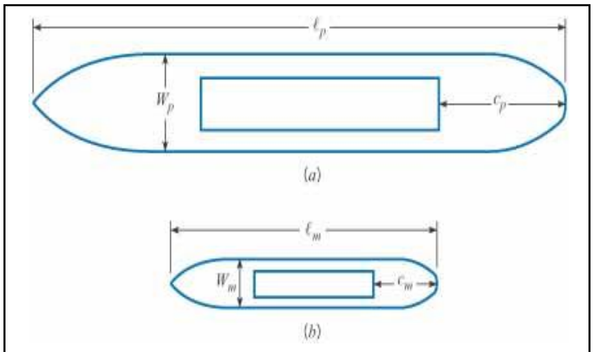
Geometric similitude means that the model is an exact geometric replica of the prototype. Consequently, if a 1:10 scale model is specified, all linear dimensions of the model must be 1 / 10 of those of the prototype

if the model and prototype are geometrically similar, the following equalities hold:

$$\frac{C_m}{C_p} = \frac{l_m}{l_p} = \frac{w_m}{w_p} = L_r$$

L_r: scale ratio, hence

$$A_r = (L_r)^2 \text{ and } \frac{\forall_m}{\forall_p} = (L_r)^3$$



(a) Prototype. (b) Model.

Dynamic Similitude

Dynamic similitude means that the forces that act on corresponding masses in the model and prototype are in the same ratio (F_m/F_p = constant) throughout the entire flow field.

1. The Froude number for the model must be equal to the Froude number for the prototype to have the same ratio of forces on the model and the prototype.

$$Fr_m = \frac{U_m}{\sqrt{gL_m}} = \frac{U_p}{\sqrt{gL_p}} = Fr_p$$

2. The Reynolds number for the model must be equal to the Reynolds number for the prototype to have the same ratio of forces on the model and the prototype.

$$Re_m = \frac{\rho_m U_m L_m}{\mu_m} = \frac{\rho_p U_p L_p}{\mu_p} = Re_p$$

3. Mach Number
4. Weber Number

Q 8.30) The drag on a submarine moving below the free surface is to be determined by a test on a 1/15 scale model in a water tunnel. The velocity of the prototype in sea water ($\rho = 1015$ kg/m^3, $\nu = 1.4 \times 10^{-6}$ m^2/s) is 2 m/s. The test is done in pure water at 20°C. Determine the speed of the water in the water tunnel for dynamic similitude and the ratio of the drag force on the model to the drag force on the prototype.

Solution

Dynamic similarity is achieved when the Reynolds numbers are the same

Drag coefficient on prototype & model will be the same $C_D = \frac{F_D}{\frac{1}{2}\rho U^2 L^2}$

Match Reynolds number

$$\begin{aligned} \mathrm{Re}_m &= \mathrm{Re}_p \\ V_m &= \frac{L_p}{L_m}\frac{\nu_m}{\nu_p}V_p \\ V_m &= 15 \times \frac{1 \times 10^{-6}}{1.4 \times 10^{-6}} \times 2 = \boxed{21.4 \text{ m/s}} \end{aligned}$$

The ratio of the drag force on the model to that on the prototype is

$$\begin{aligned} \frac{F_{D,m}}{F_{D,p}} &= \frac{\rho_m}{\rho_p}\left(\frac{V_m}{V_p}\right)^2\left(\frac{l_m}{l_p}\right)^2 \\ &= \frac{998}{1015}\left(\frac{21.4}{2}\right)^2\left(\frac{1}{15}\right)^2 \\ &= \boxed{0.500} \end{aligned}$$

8.33 A large venturi meter is calibrated by means of a 1/10 scale model using the prototype liquid. What is the discharge ratio *Qm/Qp* for dynamic similarity? If a pressure difference of 300 kPa is measured across ports in the model for a given discharge, what pressure difference will occur between similar ports in the prototype for dynamically similar conditions?

Match Reynolds Number.
Equate pressure coefficient.

Match Reynolds number

$$\begin{aligned} \mathrm{Re}_m &= \mathrm{Re}_p \\ V_m L_m/\nu_m &= V_p L_p/\nu_p \\ V_m/V_p &= (L_p/L_m)(\nu_m/\nu_p) \end{aligned}$$

Multiply both sides of Eq. (1) by $A_m/A_p = L_m^2/L_p^2$:

$$\begin{aligned} (V_m A_m)/(V_p A_p) &= (L_p/L_m) \times (1) \times L_m^2/L_p^2 \\ Q_m/Q_p &= L_m/L_p \\ Q_m/Q_p &= \boxed{1/10} \\ C_{p_m} &= C_{p_p} \\ (\Delta p/\rho V^2)_m &= (\Delta p/\rho V^2)_p \\ \Delta p_p &= \Delta p_m(\rho_p/\rho_m)(V_p/V_m)^2 \\ &= \Delta p_m(1)(L_m/L_p)^2 \\ &= 300 \times (1/10)^2 = \boxed{3.0 \text{ kPa}} \end{aligned}$$

Chapter 10 Flow in Conduits

10.1 Classifying Flow
Laminar Flow and Turbulent Flow

$Re = \frac{\rho U D}{\mu} = \frac{UD}{\nu}, \quad \nu = \frac{\mu}{\rho}$	$Re \le 2000$	Laminar flow
	$2000 \le Re \le 3000$	Unpredictable
	$Re \ge 3000$	Turbulent flow

$$Re = \frac{\rho U D}{\mu} = \frac{4Q}{\pi D \nu} = \frac{4\dot{m}}{\pi D \mu}$$

Near entrance: undeveloped "developing" flow
In developing flow, the wall shear stress is changing. In fully developed flow, the wall shear stress is constant

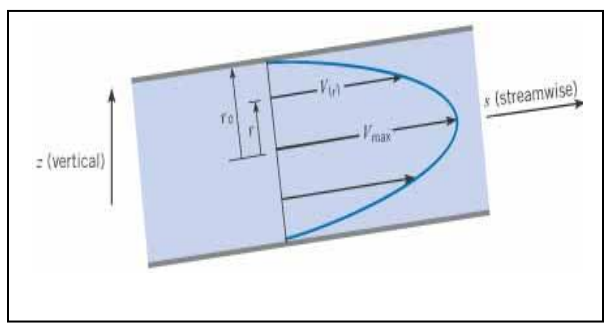
10.3 Pipe Head Loss
Combined (Total) Head Loss

Total Head Loss = pipe head loss + component head loss
Component head loss is associated with flow through devices such as valves, bends, and tees.
Pipe head loss is associated with fully developed flow in conduits, and it is caused by shear stresses that act on the flowing fluid.
The Darcy-Weisbach equation, the flow should be fully developed and steady

$$\mathrm{h}_f = f \frac{L}{D}\frac{V^2}{2\mathrm{g}}$$

$$f = \frac{4\tau_0}{\rho V^2/2} \approx \frac{shear\ stress\ acting\ onwall}{Kinetic\ pressure}$$

10.5 Laminar Flow in a Round Tube



For laminar flow

$$V(r) = V_{max}(1 - (r/r_0)^2)$$

$$V_{max} = -\left(\frac{r_0^2}{4\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right)$$

r_0: radius of pipe, Δh is the change in piezometric head over a length ΔL of conduit.

$$\bar{V} = -\left(\frac{D^2}{32\mu}\right)\left(\frac{\gamma \Delta h}{\Delta L}\right) = \frac{V_{max}}{2}$$

$$h_f = f \frac{L}{D}\frac{V^2}{2\mathrm{g}} \text{ where } f = \frac{64}{Re}$$

10.6 Turbulent Flow and the Moody Diagram

Turbulent flow is a flow regime in which the movement of fluid particles is chaotic, eddying, and unsteady, with significant movement of particles in directions transverse to the flow direction. Because of the chaotic motion of fluid particles, turbulent flow produces high levels of mixing and has a velocity profile that is more uniform or flatter than the corresponding laminar velocity profile. Engineers and scientists model turbulent flow by using an empirical approach. Because the complex nature of turbulent flow has prevented researchers from establishing a mathematical solution of general utility

Equations for the Velocity Distribution

$$\frac{V(r)}{V_{max}} = \left(\frac{r_0 - r}{r_0}\right)^m$$

Re	4×10^3	2.3×10^4	1.1×10^5	1.1×10^6	3.2×10^6
m	$\frac{1}{6.0}$	$\frac{1}{6.6}$	$\frac{1}{7.0}$	$\frac{1}{8.8}$	$\frac{1}{10.0}$
u_{max}/V	1.26	1.24	1.22	1.18	1.16

the turbulent boundary-layer equations

$$\frac{V(r)}{V^*} = 2.44 ln \frac{\left(V^*(r_0 - r)\right)}{\nu} + 5.56$$

$$V^*: shear\ veocity = \sqrt{\tau_0/\rho}$$

Equations for the Friction Factor, *f*

the resistance coefficient for turbulent flow in tubes that have smooth walls

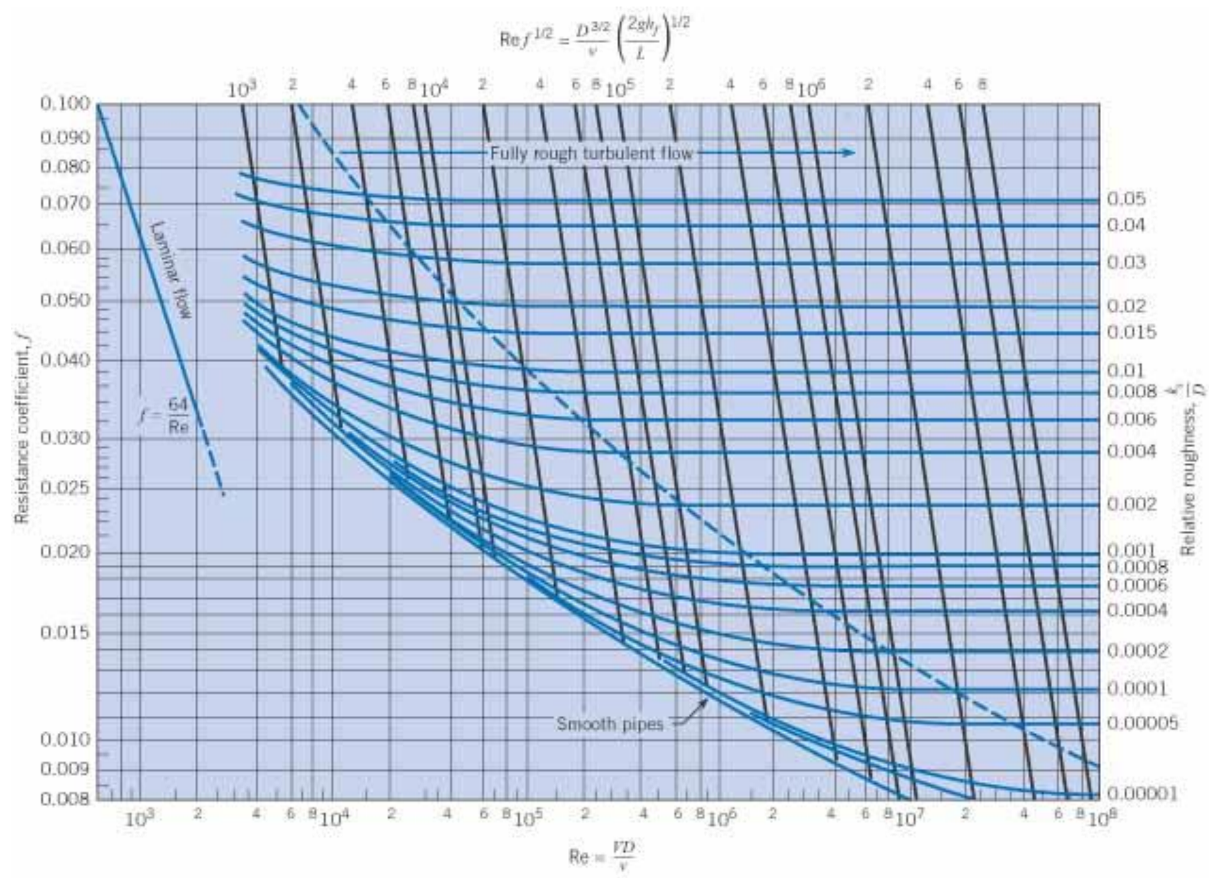
$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}(Re\sqrt{f}) - 0.8$$

Table 10.3 EFFECTS OF WALL ROUGHNESS

Type of Flow	Parameter Ranges		Influence of Parameters on f
Laminar Flow	Re < 2000	NA	f depends on Reynolds number f is independent of wall roughness (k_s/D)
Turbulent Flow, Smooth Tube	Re > 3000	$\left(\frac{k_s}{D}\right)Re < 10$	f depends on Reynolds number f is independent of wall roughness (k_s/D)
Transitional Turbulent Flow	Re > 3000	$10 < \left(\frac{k_s}{D}\right)Re < 1000$	f depends on Reynolds number f depends on wall roughness (k_s/D)
Fully Rough Turbulent Flow	Re > 3000	$\left(\frac{k_s}{D}\right)Re > 1000$	f is independent of Reynolds number f depends on wall roughness (k_s/D)

Moody Diagram

Boundary Material	k_s, Millimeters	k_s, Inches
Glass, plastic	Smooth	Smooth
Copper or brass tubing	0.0015	6×10^{-5}
Wrought iron, steel	0.046	0.002
Asphalted cast iron	0.12	0.005
Galvanized iron	0.15	0.006
Cast iron	0.26	0.010
Concrete	0.3 to 3.0	0.012–0.12
Riveted steel	0.9–9	0.035–0.35
Rubber pipe (straight)	0.025	0.001



To provide a more convenient solution to some types of problems, the top of the Moody diagram presents a scale based on the parameter Re *f*1/2. This parameter is useful when *hf* and *ks*/*D* are known but the velocity *V* is not.

In the Moody diagram, Fig., the variable *ks* denotes the *equivalent sand roughness.* That is, a pipe that has the same resistance characteristics at high Re values as a sand-roughened pipe is said to have a roughness equivalent to that of the sand-roughened pipe. Table 10.4 gives the equivalent sand roughness for various kinds of pipes. This table can be used to calculate the relative roughness for a given pipe diameter, which, in turn, is used in Fig."Moody chart" to find the friction factor

Using the Darcy-Weisbach equation and the definition of Reynolds number

$$Re\ f^{1/2} = \frac{D^{\frac{3}{2}}}{\upsilon}\left(\frac{2gh_f}{\mathrm{L}}\right)^{1/2}$$

In the Moody diagram, curves of constant $Re\ f^{1/2}$ are plotted using heavy black lines that slant from the left to right.

$$h_f = f\frac{L}{D}\frac{V^2}{2\mathrm{g}}$$

When using computers to carry out pipe-flow calculations, it is much more convenient to have an equation for the friction factor as a function of Reynolds number and relative roughness

$$f = \frac{0.25}{\left[\log_{10}\left(\frac{k_s}{3.7D} + \frac{5.74}{Re^{0.9}}\right)\right]}$$

It is reported that this equation predicts friction factors that differ by less than 3% from those on the Moody diagram for $4 \times 10^3 < Re < 10^8$ and $10^{-5} < ks/D < 2 \times 10^{-2}$.

10.8 Combined Head Loss

The Minor Loss Coefficient, *K*

When fluid flows through a component such as a partially open value or a bend in a pipe, viscous effects cause the flowing fluid to lose mechanical energy.

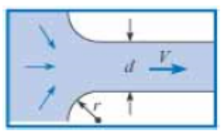
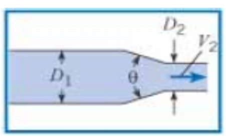
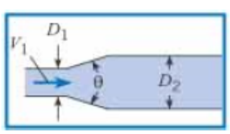
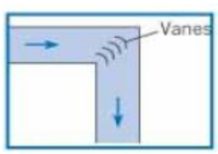
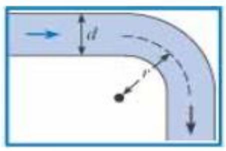
$$K = \frac{\Delta h}{V^2/2g} = \frac{\Delta p}{(\rho V^2)/2}$$

Δh: drop in piezometric head that is caused by a component.

Δp: drop in pressure that is caused by a component

V: Mean velocity

$$h_L = K\frac{V^2}{2g}$$

Description	Sketch	Additional Data		K
Pipe entrance		r/d		K_e
$h_L = K_e V^2/2g$		0.0		0.50
		0.1		0.12
		>0.2		0.03
Contraction			K_C	K_C
$h_L = K_C V_2^2/2g$		D_2/D_1	θ = 60°	θ = 180°
		0.00	0.08	0.50
		0.20	0.08	0.49
		0.40	0.07	0.42
		0.60	0.06	0.27
		0.80	0.06	0.20
		0.90	0.06	0.10
Expansion			K_E	K_E
		D_1/D_2	θ = 20°	θ = 180°
		0.00		1.00
$h_L = K_E V_1^2/2g$		0.20	0.30	0.87
		0.40	0.25	0.70
		0.60	0.15	0.41
		0.80	0.10	0.15
90° miter bend		Without vanes		$K_b = 1.1$
		With vanes		$K_b = 0.2$
90° smooth bend		r/d		
		1	$K_b = 0.35$	
		2	0.19	
		4	0.16	
		6	0.21	

Threaded pipe fittings	Globe valve—wide open	$K_v = 10.0$
	Angle valve—wide open	$K_v = 5.0$
	Gate valve—wide open	$K_v = 0.2$
	Gate valve—half open	$K_v = 5.6$
	Return bend	$K_b = 2.2$
	Tee	
	Straight-through flow	$K_t = 0.4$
	Side-outlet flow	$K_t = 1.8$
	90° elbow	$K_b = 0.9$
	45° elbow	$K_b = 0.4$

Combined Head Loss Equation

Total head loss = {Pipe head loss} + {Component head loss}

$$h_L = \sum_{\text{Pipes}} f \frac{L}{D}\frac{V^2}{2g} + \sum_{\text{compoents}} \mathrm{K}\frac{V^2}{2g}$$

10.9 Non-round Conduits

$$h_L = f\frac{L}{D_h}\frac{V^2}{2g}$$

$$D_h = \frac{4 \times cross\ section\ area}{wetted\ perimeter}$$

For rectangular cross section: $L \times w$, area = $L \times w$, perimeter = 2L+2w=2(L+w)

$$D_h = \frac{4(L \times w)}{2(L+w)} = \frac{2Lw}{L+w}$$

The head loss per kilometer of 20 cm asphalted cast-iron pipe is 12.2 m. What is the flow rate of water through the pipe?

1. Compute the parameter $D^{3/2}\sqrt{2gh_f/L}/\nu$.

$$D^{3/2}\sqrt{\frac{2gh_f/L}{\nu}} = (0.20\text{ m})^{3/2} \times \frac{[2(9.81\text{ m/s}^2)(12.2\text{ m}/1000\text{ m})]^{1/2}}{1.0\times10^{-6}\text{ m}^2/\text{s}} = 4.38\times10^4$$

2. Determine resistance coefficient.

   - Relative roughness:

   $$k_s/D = (0.00012\text{ m})/(0.2\text{ m}) = 0.0006$$

   - Look up f on the Moody diagram for

   $$D^{3/2}\sqrt{2gh_f/L}/\nu = 4.4\times10^4 \text{ and } k_s/D = 0.0006:$$
   $$f = 0.019$$

3. Find V using the Darcy-Weisbach equation.

$$h_f = f\left(\frac{L}{D}\right)\left(\frac{V^2}{2g}\right)$$

$$12.2\text{ m} = 0.019\left(\frac{1000\text{ m}}{0.20\text{ m}}\right)\left(\frac{V^2}{2(9.81\text{ m/s}^2)}\right)$$

$$V = 1.59\text{ m/s}$$

4. Use flow rate equation to find discharge.

$$Q = VA = (1.59\text{ m/s})(\pi/4)(0.2\text{ m})^2 = \boxed{0.05\text{ m}^3/\text{s}}$$

Q 10.72 A heat exchanger consists of a closed system with a series of parallel tubes connected by 180° elbows as shown in the figure. There are a total of 14 return elbows. The pipe diameter is 2 cm, and the total pipe length is 10 m. The head loss coefficient for each return elbow is 2.2. The tube is copper. Water with an average temperature of 40°C flows through the system with a mean velocity of 10 m/s. Find the power required to operate the pump if the pump is 80% efficient.

Find: Power required to operate pump.

Properties: From Table A.5 $\nu = 6.58 \times 10^{-7}$ m^2/s.
From Table 10.2 $k_s = 0.0015$ mm.

ANALYSIS

Reynolds number

$$\text{Re} = \frac{0.02 \times 10}{6.58 \times 10^{-7}} = 3.04 \times 10^5$$

Flow rate equation

$$Q = \frac{\pi}{4} \times 0.02^2 \times 10 = 0.00314 \text{ m}^3/\text{s}$$

Relative roughness (copper tubing)

$$\frac{k_s}{D} = \frac{1.5 \times 10^{-3} \text{ mm}}{20 \text{ mm}} = 7.5 \times 10^{-5}$$

Resistance coefficient (from Moody diagram)

$$f = 0.0155$$

Energy equation

$$\begin{aligned} h_p &= \frac{V^2}{2g}(f\frac{L}{D} + \sum K_L) \\ &= \frac{10^2}{2 \times 9.81}(0.0155 \times \frac{10 \text{ m}}{0.02 \text{ m}} + 14 \times 2.2) = 196 \text{ m} \end{aligned}$$

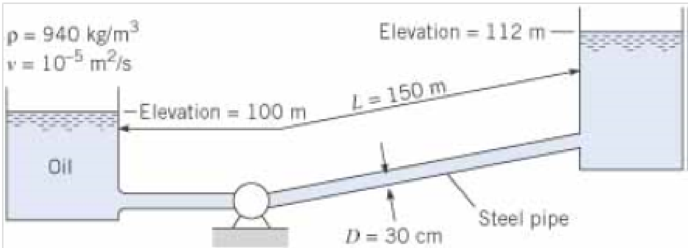
Power equation

$$\begin{aligned} P &= \frac{\gamma Q h_p}{\eta} \\ &= \frac{9732 \times 0.00314 \times 196}{0.8} \\ &= 7487 \text{ W} \end{aligned}$$

$$\boxed{P = 7.49 \text{ kW}}$$

Q10.66 What power must the pump supply to the system to pump the oil from the lower reservoir to the upper reservoir at a rate of $0.20 m^3/s$? Sketch the HGL and the EGL for the system.

From Table 10.2 k_s = 0.046 mm



Energy equation

$$
\begin{aligned}
p_1/\gamma + \alpha_1 V_1^2/2g + z_1 + h_p &= p_2/\gamma + \alpha_2 V_2^2/2g + z_2 + \sum h_L \\
100 + h_p &= 112 + V^2/2g(K_e + fL/D + K_E) \\
h_p &= 12 + (V^2/2g)(0.03 + fL/D + 1)
\end{aligned}
$$

Flow rate equation

$$
\begin{aligned}
V &= Q/A \\
&= 0.20/((\pi/4) \times 0.30^2) \\
&= 2.83 \text{ m/s} \\
V^2/2g &= 0.408 \text{ m}
\end{aligned}
$$

Reynolds number

$$
\begin{aligned}
\text{Re} &= VD/\nu \\
&= 2.83 \times 0.30/(10^{-5}) \\
&= 8.5 \times 10^4 \\
k_s/D &= 4.6 \times 10^{-5}/0.3 \\
&= 1.5 \times 10^{-4}
\end{aligned}
$$

Resistance coefficient (from the Moody diagram, Fig. 10.8)

$$f = 0.019$$

Then
$$
\begin{aligned}
h_p &= 12 + 0.408(0.03 + (0.019 \times 150/0.3) + 1.0) \\
&= 16.3 \text{ m}
\end{aligned}
$$

Power equation

$$
\begin{aligned}
P &= Q\gamma h_p \\
&= 0.20 \times (940 \times 9.81) \times 16.3 = 2.67 \times 10^4 \text{ W} \\
&= \boxed{30.1 \text{ kW}}
\end{aligned}
$$