J.B. INSTITUTE OF ENGINEERING AND TECHNOLOGY

(UGC AUTONOMOUS)

Bhaskar Nagar, Moinabad Mandal, R.R. District, Hyderabad -500075

DEPARTMENT OF CIVIL ENGINEERING

FLUID MECHANICS

LECTURE NOTESR18

B. TECH
II YEAR - I SEM (Sec- A&B)
Academic Year 2020-21

Prepared & Compiled by

MR.Anil kumar Butty

ASSISTANT PROFESSOR, DEPARTMENT OF CE

J.B.I.E.T

Bhaskar Nagar, Yenkapally(V), Moinabad(M),
Ranga Reddy(D), Hyderabad - 500 075, Telangana, India

UNIT I

PROPERTIES OF FLUIDS AND FLUID STATICS

Introduction to Fluid Mechanics

Definition of a fluid

A fluid is defined as a substance that deforms continuously under the action of a shear stress, however small magnitude present. It means that a fluid deforms under very small shear stress, but a solid may not deform under that magnitude of the shear stress.

Fig.L-1.1a: Deformation of solid under a constant shear force

By contrast a solid deforms when a constant shear stress is applied, but its deformation does not continue with increasing time. In Fig.L1.1, deformation pattern of a solid and a fluid under the action of constant shear force is illustrated. We explain in detail here deformation behaviour of a solid and a fluid under the action of a shear force.

In Fig.L1.1, a shear force ***F*** is applied to the upper plate to which the solid has been bonded, a shear stress resulted by the force equals to $\tau = \frac{F}{A}$, where *A* is the contact area of the upper plate. We know that in the case of the solid block the deformation is proportional to the shear stress *t* provided the elastic limit of the solid material is not exceeded.

When a fluid is placed between the plates, the deformation of the fluid element is illustrated in Fig.L1.3. We can observe the fact that the deformation of the fluid element continues to increase as long as the force is applied. The fluid particles in direct contact with the plates move with the same speed of the plates. This can be interpreted that there is no slip at the boundary. This fluid behavior has been verified in numerous experiments with various kinds of fluid and boundary material.

In short, ***a fluid continues in motion under the application of a shear stress and can not sustain any shear stress when at rest.***

Fluid as a continuum

In the definition of the fluid the molecular structure of the fluid was not mentioned. As we know the fluids are composed of molecules in constant motions. For a liquid, molecules are closely spaced compared with that of a gas. In most engineering applications the average or macroscopic effects of a large number of molecules is considered. We thus do not concern about the behavior of individual molecules. The fluid is treated as an infinitely divisible substance, a continuum at which the properties of the fluid are considered as a continuous (smooth) function of the space variables and time.

To illustrate the concept of fluid as a continuum consider fluid density as a fluid property at a small region. Density is defined as mass of the fluid molecules per unit volume. Thus the mean density within the small region **C** could be equal to mass of fluid molecules per unit volume. When the small region **C** occupies space which is larger than the cube of molecular spacing, the number of the molecules will remain constant. This is the limiting volume $\delta v'$ above which the effect of molecular variations on fluid properties is negligible.

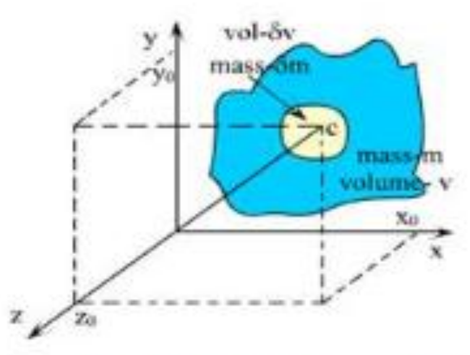


Fig. L-1.2(a):Small region in fluid domain

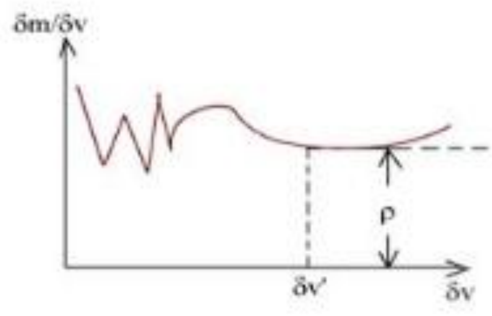


Fig. L-1.2(b): Variation of density with respect to volume of the region

The density of the fluid is defined as

$$\rho = \lim_{\delta v \to \delta v'} \frac{\delta m}{\delta v}$$

Note that the limiting volume $\delta v'$ is about $10^{-9}\,mm^3$ for all liquids and for gases at atmospheric temperature. Within the given limiting value, air at the standard condition has approximately 3×10^7 molecules. It justifies in defining a nearly constant density in a region which is larger than the limiting volume.

In conclusion, since most of the engineering problems deal with fluids at a dimension which is larger than the limiting volume, the assumption of fluid as a continuum is valid. For example the fluid density is defined as a function of space (for Cartesian coordinate system, x, y, and z) and time (t) by $\rho = \rho(x, y, z, t)$. This simplification helps to use the differential calculus for solving fluid problems.

Properties of fluid

Some of the basic properties of fluids are discussed below-

Density : As we stated earlier the density of a substance is its mass per unit volume. In fluid mechanic it is expressed in three different ways-

Mass density r is the mass of the fluid per unit volume (given by Eq.L1.1)

Unit- kg/m^3
Dimension- ML^{-3}
Typical values: water- *1000* *kg/* m^3
Air- $1.23\,kg/m^3$ at standard pressure and temperature (STP)

Specific weight, w: - As we express a mass *M* has a weight ***W=Mg*** . The specific weight of the fluid can be defined similarly as its weight per unit volume.

$$w = \rho g \qquad \text{L-2.1}$$

Unit: N/m^3
Dimension: $ML^{-2}T^{-2}$

Typical values; water- $9.810 N/m^3$
Air- $12.07 N/m^3$ (STP)

Relative density (Specific gravity), ***S*** :-

Specific gravity is the ratio of fluid density (specific weight) to the fluid density (specific weight) of a standard reference fluid. For liquids water at $4^0 C$ is considered as standard fluid.

$$S_{liquid} = \frac{\rho_{liquid}}{\rho_{water\ at\ 4^0C}} \qquad \text{L-2.2}$$

Similarly for gases air at specific temperature and pressure is considered as a standard reference fluid.

$$S_{gas} = \frac{\rho_{gas}}{\rho_{gas\ at\ STP}} \qquad \text{L-2.3}$$

Units: pure number having no units.

Dimension:- $M^oL^oT^o$

Typical vales : - Mercury- 13.6

Water-1

Specific volume v_s : - Specific volume of a fluid is mean volume per unit mass *i.e.* the reciprocal of mass density.

$$v_s = \frac{1}{\rho} \qquad \text{L-2.4}$$

Units:- m^3/kg

Dimension: $M^{-1}L^3$

Typical values: - Water - $10^{-3} m^3/kg$

Air- $1.23 \times 10^{-3} m^3/kg$

Viscosity

In section L1 definition of a fluid says that under the action of a shear stress a fluid continuously deforms, and the shear strain results with time due to the deformation. Viscosity is a fluid property, which determines the relationship between the fluid strain rate and the applied shear stress. It can be noted that in fluid flows, shear strain rate is considered, not shear strain as commonly used in solid mechanics. Viscosity can be inferred as a quantative measure of a fluid's resistance to the flow. For example moving an object through air requires very less force compared to water. This means that air has low viscosity than water.

Let us consider a fluid element placed between two infinite plates as shown in fig (Fig-2.1). The upper plate moves at a constant velocity δu under the action of constant shear force δF. The shear stress, *t* is expressed as

$$\tau = \lim_{\delta A \to 0} \frac{\delta F}{\delta A} = \frac{dF}{dA}$$

where, δA is the area of contact of the fluid element with the top plate. Under the action of shear force the fluid element is deformed from position *ABCD* at time *t* to position AB'C'D' at time $t + \delta t$ (fig-L2.1). The shear strain rate is given by

Shear strain rate $= \lim_{\delta t \to 0} \frac{\delta \alpha}{\delta t} = \frac{d\alpha}{dt}$ L2.6

Where α is the angular deformation

From the geometry of the figure, we can define

For small $\delta\alpha$, $\tan \delta\alpha = \frac{\delta u \ \delta t}{\delta y}$

Therefore,

$$\frac{\delta \alpha}{\delta t} = \frac{\delta u}{\delta y}$$

The limit of both side of the equality gives $\frac{d\alpha}{dt} = \frac{du}{dy}$ L-2.5

The above expression relates shear strain rate to velocity gradient along the *y* -axis.

Newton's Viscosity Law

Sir Isaac Newton conducted many experimental studies on various fluids to determine relationship between shear stress and the shear strain rate. The experimental finding showed that a linear relation between them is applicable for common fluids such as water, oil, and air. The relation is

$$\tau \; \alpha \; \frac{d\alpha}{dt}$$

Substituting the relation gives in equation(L-2.5)

$$\tau \, \alpha \, \frac{du}{dy} \qquad \text{L-2.6}$$

Introducing the constant of proportionality

$$\tau = \mu \frac{du}{dy}$$

where μ is called absolute or dynamic viscosity. Dimensions and units for μ are $ML^{-1}T^{-1}$ and $N-s/m^2$, respectively. [In the absolute metric system basic unit of co-efficient of viscosity is called poise. 1 poise = $N-s/m^2$]

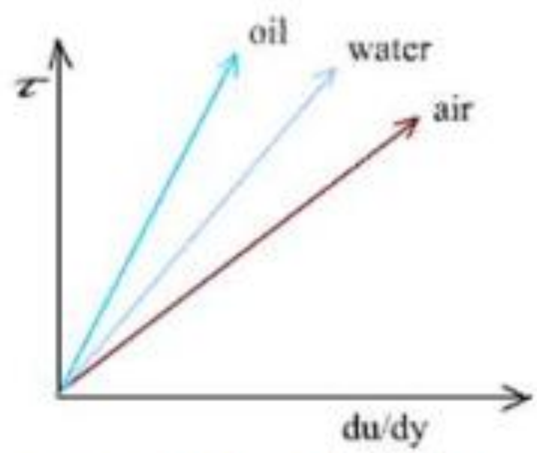


Fig.L-2.2: Relationship between shear stress and velocity gradient of Newtonian fluids

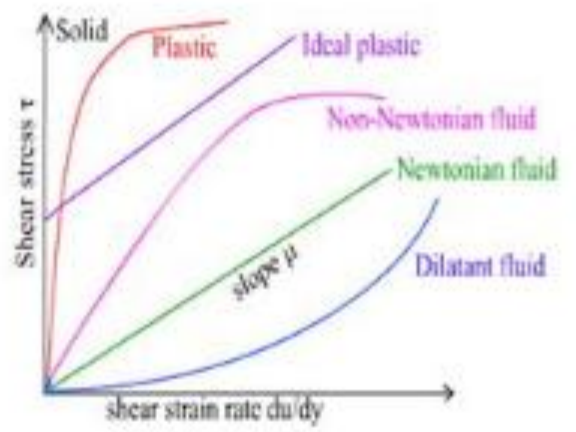


Fig.L-2.3: Relationship between shear stress and shear strain rate of diferent fluids

Typical relationships for common fluids are illustrated in Fig-L2.3.

The fluids that follow the linear relationship given in equation (L-2.7) are called Newtonian fluids.

Kinematic viscosity ν

Kinematic viscosity is defined as the ratio of dynamic viscosity to mass density

$$\nu = \frac{\mu}{\rho} \qquad \text{L-2.8}$$

Units: m^2/s

Dimension: L^2T^{-1}

Typical values: water $1.14\times10^{-6} m^2s^{-1}$ air $1.46\times10^{-5} m^2/s$

Non - Newtonian fluids

Fluids in which shear stress is not linearly related to the rate of shear strain are non– Newtonian fluids. Examples are paints, blot, polymeric solution, etc. Instead of the dynamic viscosity apparent viscosity, μ_{ap} which is the slope of shear stress versus shear strain rate curve, is used for these types of fluid.

Based on the behavior of μ_{ap}, non-Newtonian fluids are broadly classified into the following groups –

a. *Pseudo plastics* (shear thinning fluids): μ_{ap} decreases with increasing shear strain rate. For example polymer solutions, colloidal suspensions, latex paints, pseudo plastic.

b. *Dilatants* (shear thickening fluids) μ_{ap} increases with increasing shear strain rate.

   Examples: Suspension of starch and quick sand (mixture of water and sand).

c. *Plastics* : Fluids that can sustain finite shear stress without any deformation, but once shear stress exceeds the finite stress τ_y, they flow like a fluid. The relation between the shear stress and the resulting shear strain is given by

$$\tau = \tau_y + \mu_{ap}\left(\frac{du}{dy}\right)^n \qquad \text{L-2.9}$$

Fluids with $n = 1$ are called Bingham plastic. some examples are clay suspensions, tooth paste and fly ash.

d. *Thixotropic* fluid(Fig. L-2.4): μ_{ap} decreases with time under a constant applied shear stress.

Example: Ink, crude oils.

e. *Rheopectic fluid* : μ_{ap} increases with increasing time.

Example: some typical liquid-solid suspensions.

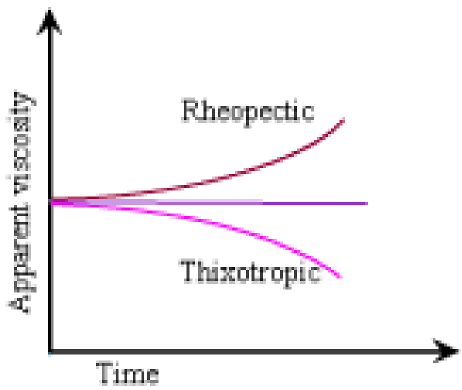
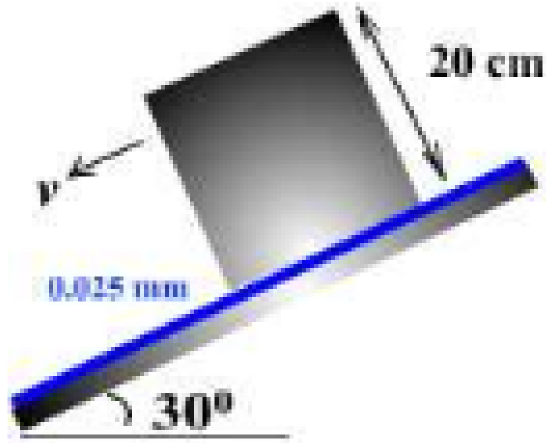


Fig. L-2.4: Thixotropic and Rheopectic fluids

Example

As shown in the figure a cubical block of 20 cm side and of 20 kg weight is allowed to slide down along a plane inclined at 30^0 to the horizontal on which there is a film of oil having viscosity 2.16×10^{-3} N-s/m^2 .What will be the terminal velocity of the block if the film thickness is 0.025mm?



Given data : Weight = 20 kg

Block dimension = 20x20x20 cm^3

Driving force along the plane $F = W \sin 30^0 = 98.1N$

Shear force $\tau = F/A = 2452.5N/m^2$

Contact area, $A = 0.2 \times 0.2 m^2$

Also, $\tau = \mu \frac{dv}{dy}$

Answer: 28.38m/s.

Example

If the equation of a velocity profile over a plate is $v = 5y\ 2 + y$ (where v is the velocity in m/s) determine the shear stress at $y = 0$ and at $y = 7.5cm$. Given the viscosity of the liquid is 8.35 poise.

Solution

Given Data: Velocity profile $v = 5y^2 + y$

$\mu = 8.35 poise$

$$\text{Velocity gradient, } \frac{dv}{dy} = 10y + 1$$

$$\tau = \mu.\frac{dv}{dy} = \mu.(10y+1)$$

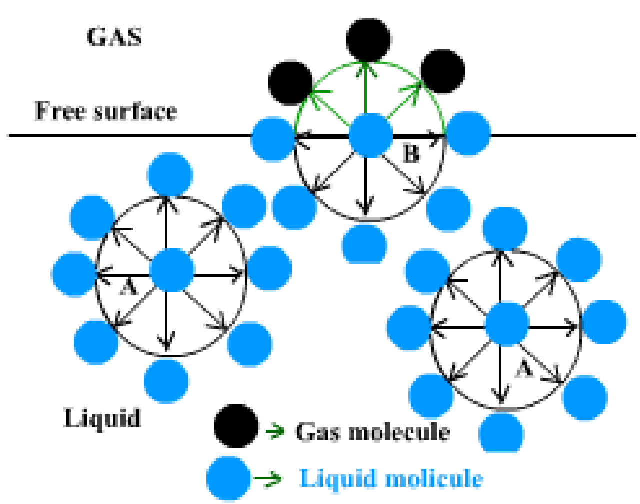
Substituting *y = 0 and y =0.075* on the above equation, we get shear stress at respective depths.

Answer: 0.835 ; $1.46\, N/m^2$

Surface tension and Capillarity

Surface tension

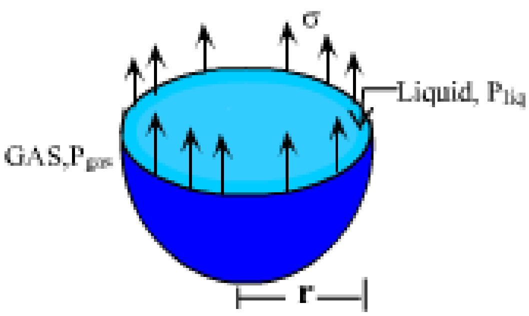
In this section we will discuss about a fluid property which occurs at the interfaces of a liquid and gas or at the interface of two immiscible liquids. As shown in Fig (L - 3.1) the liquid molecules- 'A' is under the action of molecular attraction between like molecules (cohesion). However the molecule ‘B' close to the interface is subject to molecular attractions between both like and unlike molecules (adhesion). As a result the cohesive forces cancel for liquid molecule 'A'. But at the interface of molecule 'B' the cohesive forces exceed the adhesive force of the gas. The corresponding net force acts on the interface; the interface is at a state of tension similar to a stretched elastic membrane. As explained, the corresponding net force is referred to as surface tension, δ. In short it is apparent tensile stresses which acts at the interface of two immiscible fluids.



Dimension: MT^{-2}
Unit: N/m
Typical values: Water $0.074\,N/m$ at 20° C with air.

Note that surface tension decreases with the liquid temperature because intermolecular cohesive forces decreases. At the critical temperature of a fluid surface tension becomes zero; i.e. the boundary between the fluids vanishes.

Pressure difference at the interface



Surface tension on a droplet

In order to study the effect of surface tension on the pressure difference across a curved interface, consider a small spherical droplet of a fluid at rest.

Since the droplet is small the hydrostatic pressure variations become negligible. The droplet is divided into two halves as shown in Fig.L-3.2. Since the droplet is at rest, the sum of the forces acting at the interface in any direction will be zero. Note that the only forces acting at the interface are pressure and surface tension. Equilibrium of forces gives

$$\left(P_{liq} - P_{gas}\right)\pi r^2 = \sigma\left(2\pi r\right) \qquad \text{L - 3.1}$$

Solving for the pressure difference and then denoting $\Delta P = P_{liq} - P_{gas}$ we can rewrite equation (L- 3.1) as

$$\Delta P = \frac{2\sigma}{r}$$

Contact angle and welting

As shown in fig. a liquid contacts a solid surface. The line at which liquid gas and solid meet is called the contact line. At the contact line the net surface tension depending upon all three materials - liquid, gas, and solid is evident in the contact angle, θ_C. A force balance on the contact line yields:

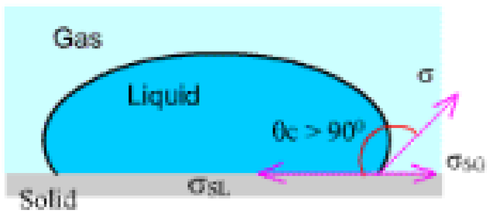


Fig : L-3.3: Contact line for wetting condition

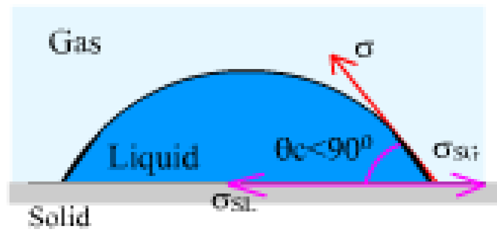


Fig : L-3.4: Contact line for non-wetting condition

$$\sigma_{gas} - \sigma_{solid} = \sigma \cos \theta_c$$

here σ_{gas} is the surface tension of the gas-solid interface, σ_{solid} is the surface tension of solid-liquid interface, and σ is the surface tension of liquid-gas interface.

Typical values:

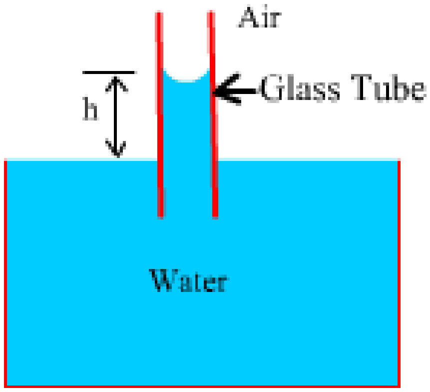
$\theta_C \approx 0^\circ$ for air-water- glass interface

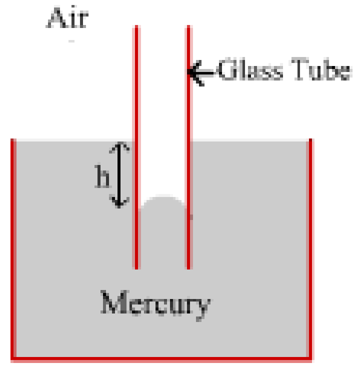
$\theta_C \approx 140^\circ$ for air-mercury–glass interface

If the contact angle $\theta_c < 90^\circ$ the liquid is said to wet the solid. Otherwise, the solid surface is not wetted by the liquid, when $\theta_c > 90^\circ$.

Capillarity

If a thin tube, open at the both ends, is inserted vertically in to a liquid, which wets the tube, the liquid will rise in the tube (fig : L -3.4). If the liquid does not wet the tube it will be depressed below the level of free surface outside. Such a phenomenon of rise or fall of the liquid surface relative to the adjacent level of the fluid is called capillarity. If θ_c is the angle of contact between liquid and solid, *d* is the tube diameter, we can determine the capillary rise or depression, *h* by equating force balance in the z-direction (shown in Fig : L-3.5), taking into account surface tension, gravity and pressure. Since the column of fluid is at rest, the sum of all of forces acting on the fluid column is zero.





The pressure acting on the top curved interface in the tube is atmospheric, the pressure acting on the bottom of the liquid column is at atmospheric pressure because the lines of constant pressure in a liquid at rest are horizontal and the tube is open.

Upward force due to surface tension $= \sigma \cos\theta_c \pi d$

Weight of the liquid column $= \rho g . \pi \frac{d^2}{4} h$

Thus equating these two forces we find

$$\sigma \cos\theta_c \pi d = \rho g \pi \frac{d^2}{4} h$$

The expression for *h* becomes

$$h = \frac{4\sigma \cos\theta_c}{\rho g d} \qquad \text{L -3.2}$$

Typical values of capillary rise are

a. Capillary rise is approximately 4.5 mm for water in a glass tube of 5 mm diameter.
b. Capillary depression is approximately - 1.5 mm (depression) for mercury in the same tube.
c. Capillary action causes a serious source of error in reading the levels of the liquid in small pressure measuring tubes. Therefore the diameter of the measuring tubes should be large enough so that errors due to the capillary rise should be very less. Besides this,

capillary action causes the movement of liquids to penetrate cracks even when there is no significant pressure difference acting to move the fluids in to the cracks.

d. In figure (Fig : L - 3.6), a two-dimensional model for the capillary rise of a liquid in a crack width, *b*, is illustrated. The height of the capillary rise can also be computed by equating force balance as explained in the previous section.

Capillary rise, $$h = \frac{2\sigma \cos\theta_c}{b\rho g} \qquad \text{L-3.3}$$

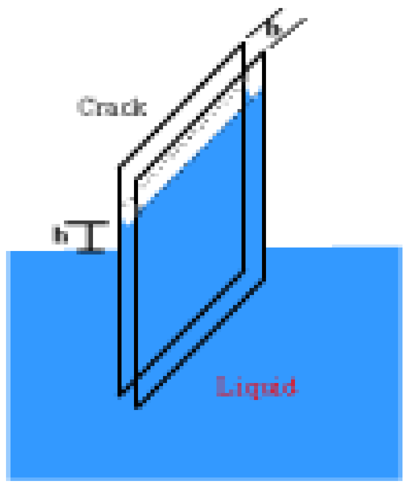


Fig. L-3.6: Capillary rise in a Crack

Vapour Pressure

Since the molecules of a liquid are in constant motion, some of the molecules in the surface layer having sufficient energy will escape from the liquid surface, and then changes from liquid state to gas state. If the space above the liquid is confined and the number of the molecules of the liquid striking the liquid surface and condensing is equal to the number of liquid molecules at any time interval becomes equal, an equilibrium exists. These molecules exerts of partial pressure on the liquid surface known as vapour pressure of the liquid, because degree of molecular activity increases with increasing temperature. The vapour pressure increases with temperature. Boiling occurs when the pressure above a liquid becomes equal to or less then the vapour pressure of the liquid. It means that boiling of water may occur at room temperature if the pressure is reduced sufficiently.

For example water will boil at 60 ° C temperature if the pressure is reduced to 0.2 atm.

Cavitation

In many fluid problems, areas of low pressure can occur locally. If the pressure in such areas is equal to or less then the vapour pressure, the liquid evaporates and forms a cloud of vapour bubbles. This phenomenon is called cavitation. This cloud of vapour bubbles is swept in to an area of high pressure zone by the flowing liquid. Under the high pressure the bubbles collapses. If this phenomenon occurs in contact with a solid surface, the high pressure developed by collapsing bubbles can erode the material from the solid surface and small cavities may be formed on the surface.

The cavitation affects the performance of hydraulic machines such as pumps, turbines and propellers.

Compressibility and the bulk modulus of elasticity

When a fluid is subjected to a pressure increase the volume of the fluid decreases. The relationship between the change of pressure and volume is linear for many fluids. This relationship may be defined by a proportionality constant called bulk modulus.

Consider a fluid occupying a volume *V* in the piston and cylinder arrangement shown in figure. If the pressure on the fluid increase from *p* to $p + \delta p$ due to the piston movement as a result the volume is decreased by ΔV. We can express the bulk modulus of elasticity

$$k = -\frac{\delta p}{\delta v / v} \qquad \text{L - 4.1}$$

The negative sign indicates the volume decreases as pressure increases. As in the limit as $\delta p \rightarrow 0$ then

$$k = -\frac{dp}{dv / v} \qquad \text{L - 4.2}$$

Since $-\frac{dv}{v} = \frac{dp}{p}$ the equation can be rearranged as

$$k = \frac{dp}{d\rho / \rho} \qquad \text{L - 4.3}$$

Dimension :- $ML^{-1}T^{-2}$

Unit :- N/m^2

Typical values:-

Air - 1.03 x 10 5 N/m^2

water $2.05\times10^{9}\ N/m^{2}$ at standard temperature and pressure as compared to that of Mild steel $2.06\times10^{11}\ N/m^{2}$.

The above typical values show that the air is about 20,000 times more compressible than water while water is about 100 times more compressible than mild steel.

Basic Equations

To analysis of any fluid problem, the knowledge of the basic laws governing the fluid flows is required. The basic laws, applicable to any fluid flow, are:

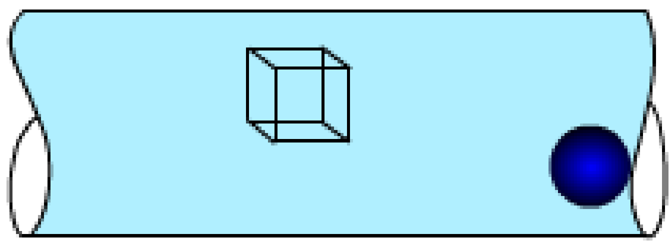
a. Conservation of mass. (Continuity)
b. Linear momentum. (Newton 's second law of motion)
c. Conservation of energy (First law of Thermodynamics)

Besides these governing equations, we need the state relations like $\rho = \rho(P,T)$ and appropriate boundary conditions at solid surface, interfaces, inlets and exits. Note that all basic laws are not always required to any one problem. These basic laws, as similar in solid mechanics and thermodynamics, are to be reformulated in suitable forms so that they can be easily applied to solve wide variety of fluid problems.

System and control volume

A system refers to a fixed, identifiable quantity of mass which is separated from its surrounding by its boundaries. The boundary surface may vary with time however no mass crosses the system boundary. In fluid mechanics an infinitesimal lump of fluid is considered as a system and is referred as a fluid element or a particle. Since a fluid particle has larger dimension than the limiting volume (refer to section fluid as a continuum). The continuum concept for the flow analysis is valid.

control volume is a fixed, identifiable region in space through which fluid flows. The boundary of the control volume is called control surface. The fluid mass in a control volume may vary with time. The shape and size of the control volume may be arbitrary.



System and control volume

When a fluid is at rest, the fluid exerts a force normal to a solid boundary or any imaginary plane drawn through the fluid. Since the force may vary within the region of interest, we conveniently define the force in terms of the pressure, P, of the fluid. The pressure is defined as the *force per unit area.*

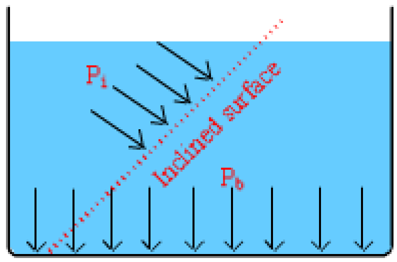


Fig : L - 6.1: Pressure variation at the bottom surface P_b and at the inclined surface P_i

In Fig : L - 6.1 pressure variation of a fluid at different locations is illustrated.

Commonly the pressure changes from point to point. We can define the pressure at a point as

$$P = \lim_{\delta A \to 0} \frac{\delta F}{\delta A} = \frac{dF}{dA} \qquad \text{L - 6.1}$$

where dA is the area on which the force dF acts. It is a scalar field and varies spatially and temporally as given $P = P\,(x, y, z, t)$

Pascal's Law : Pressure at a point

The Pascal's law states that *the pressure at a point in a fluid at rest is the same in all directions* . Let us prove this law by considering the equilibrium of a small fluid element shown in Fig : L - 6.2

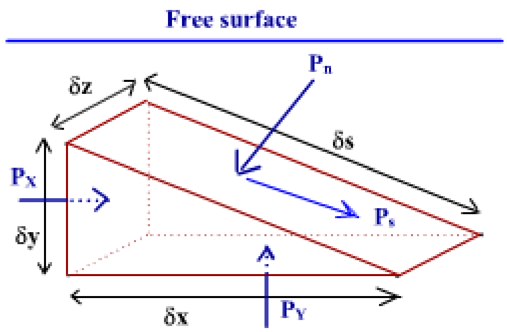


Fig : L -6.2: A fluid element with force components

Since the fluid is at rest, there will be no shearing stress on the faces of the element.

The equilibrium of the fluid element implies that sum of the forces in any direction must be zero. For the x-direction:

Force due to P_x is $P_x \cdot \delta y \cdot \delta z$

Component of force due to P_n

$$= -P_n \cdot \delta n \cdot \delta z \cdot \frac{\delta y}{\delta n}$$
$$= -P_n \cdot \delta y \cdot \delta z$$

Summing the forces we get,

$$P_x \cdot \delta y \cdot \delta z - P_n \cdot \delta y \cdot \delta z = 0$$
$$\text{then } P_x = P_n$$

Similarly in the y-direction, we can equate the forces as given below

Force due to P_y = $P_y \cdot \delta x \cdot \delta z$

Component of force due to P_n

$$= -P_n \cdot \delta n \cdot \delta z \cdot \frac{\delta x}{\delta n}$$
$$= -P_n \cdot \delta x \cdot \delta z$$

$$\text{Weight of the fluid element} = -\text{Specific weight} \times \text{volume of the element}$$
$$= -\rho \cdot g \cdot \frac{1}{2} \cdot \delta x \cdot \delta y \cdot \delta z$$

The negative sign indicates that weight of the fluid element acts in opposite direction of the z-direction.

Summing the forces yields

$$P_y \cdot \delta n \cdot \delta z - P_n \cdot \delta x \cdot \delta z - \frac{1}{2} \cdot \rho \cdot g \cdot \delta x \cdot \delta y \cdot \delta z = 0$$

Since the volume of the fluids $\delta x \cdot \delta y \cdot \delta z$ is very small, the weight of the element is negligible in comparison with other force terms. So the above Equation becomes

$P_y = P_n$

Hence, $P_n = P_x = P_y$

Similar relation can be derived for the z-axis direction.

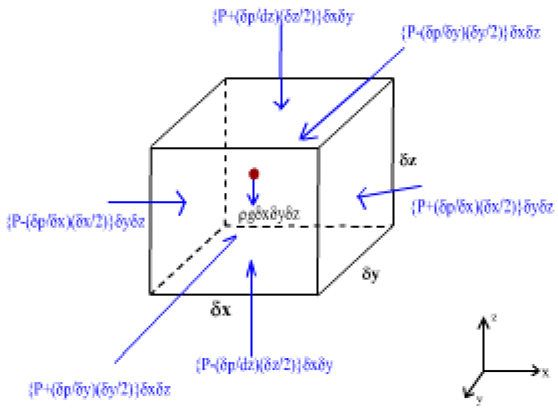
This law is valid for the cases of fluid flow where shear stresses do not exist. The cases are

a. Fluid at rest.
b. No relative motion exists between different fluid layers. For example, fluid at a constant linear acceleration in a container.
c. Ideal fluid flow where viscous force is negligible.

Basic equations of fluid statics

An equation representing pressure field $P = P(x, y, z)$ within fluid at rest is derived in this section. Since the fluid is at rest, we can define the pressure field in terms of space dimensions (x, y and z) only.

Consider a fluid element of rectangular parellopiped shape(Fig : L - 7.1) within a large fluid region which is at rest. The forces acting on the element are body and surface forces.



Body force: The body force due to gravity is

$$d\overline{F}_B = \rho \cdot g \cdot \delta x \cdot \delta y \cdot \delta z \qquad \text{L -7.1}$$

Where $\delta x \cdot \delta y \cdot \delta z$ is the volume of the element.

Surface force: The pressure at the center of the element is assumed to be *P* (x, y, z). Using Taylor series expansion the pressure at point $\left(x, y-\frac{\delta y}{2}, z\right)$ on the surface can be expressed as

$$P\left(x, y-\frac{\delta y}{2}, z\right) = P(x,y,z) + \frac{\delta p}{\delta y}\left(-\frac{\delta y}{2}\right) + \frac{1}{2!}\frac{\partial^2 p}{\partial y^2}\left(-\frac{\delta y}{2}\right)^2 + \ldots\ldots\ldots \qquad \text{L -7.2}$$

When $\delta y \to 0$, only the first two terms become significant. The above equation becomes

$$P\left(x, y-\frac{\delta y}{2}, z\right) = P(x,y,z) + \frac{\delta p}{\delta y}\left(-\frac{\delta y}{2}\right) \qquad \text{L - 7.3}$$

Similarly, pressures at the center of all the faces can be derived in terms of *P (x, y, z)* and its gradient.

Note that surface areas of the faces are very small. The center pressure of the face represents the average pressure on that face.
The surface force acting on the element in the y-direction is

$$dF_y = \left\{P + \frac{\delta P}{\delta y}\left\{-\frac{\delta y}{2}\right\}\right\}\delta x \cdot \delta y - \left\{P + \frac{\delta P}{\delta y}\left\{\frac{\delta y}{2}\right\}\right\}\delta x \cdot \delta z$$

$$= -\frac{\delta P}{\delta y} \cdot \delta x \cdot \delta y \cdot \delta z \qquad \text{L -7.4}$$

Similarly the surface forces on the other two directions (x and z) will be

$$dF_x = -\frac{\delta P}{\delta x} \cdot \delta x \cdot \delta y \cdot \delta z$$

$$dF_z = -\frac{\delta P}{\delta z} \cdot \delta x \cdot \delta y \cdot \delta z$$

The surface force which is the vectorical sum of the force scalar components

$$dF_s = -\left(\frac{\delta p}{\delta x}\hat{i} + \frac{\delta p}{\delta y}\hat{j} + \frac{\delta p}{\delta z}\hat{k}\right)(\delta x \cdot \delta y \cdot \delta z)$$

$$= -\nabla p \cdot \delta x \cdot \delta y \cdot \delta z \qquad \text{L - 7.5}$$

The total force acting on the fluid is

$$d\,\overline{F} = d\,\overline{F_S} + d\,\overline{F_B}$$

$$= \left(-\nabla p + \rho\vec{g}\right)(\delta x \cdot \delta y \cdot \delta z) \qquad \text{L - 7.6}$$

The total force per unit volume is

$$\frac{dF}{\delta x \cdot \delta y \cdot \delta z} = -\nabla p + \rho\vec{g}$$

For a static fluid, *dF=0* .

Then, $\left(-\nabla p + \rho\vec{g}\right) = 0$ L -7.7

$$\begin{bmatrix}\text{Net pressure force}\\ \text{per unit volume}\\ \text{at a point}\end{bmatrix} + \begin{bmatrix}\text{Body force}\\ \text{per unit volume}\\ \text{at a point}\end{bmatrix} = 0$$

If acceleration due to gravity $\vec{g}$ is expressed as $\vec{g} = g_x\hat{i} + g_y\hat{j} + g_z\hat{k}$, the components of Eq(L- 7.8) in the x, y and z directions are

$$-\frac{\delta p}{\delta z} + \rho g_z = 0$$

$$-\frac{\delta p}{\delta x} + \rho g_x = 0$$

$$-\frac{\delta p}{\delta y} + \rho g_y = 0$$

The above equations are the basic equation for a fluid at rest.

Simplifications of the Basic Equations

If the gravity $\vec{g}$ is aligned with one of the co-ordinate axis, for example *z*- axis, then

$$g_x = 0$$

$$g_y = 0$$

$$g_z = -g$$

The component equations are reduced to

$$\frac{\delta p}{\delta x} = 0$$

$$\frac{\delta p}{\delta y} = 0 \qquad \text{L-7.9}$$

$$\frac{\delta p}{\delta z} = -\rho g$$

Under this assumption, the pressure *P* depends on z only. Therefore, total derivative can be used instead of the partial derivative.

$$\frac{dp}{dz} = -\rho g$$

This simplification is valid under the following restrictions

a. Static fluid

b. Gravity is the only body force.

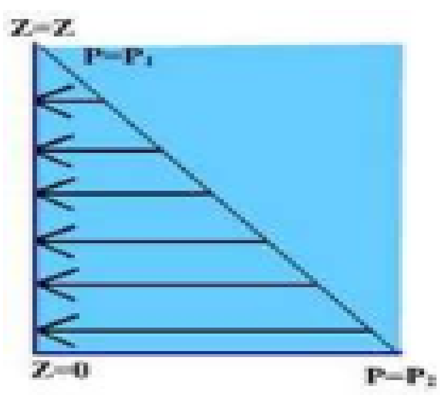
c. The z-axis is vertical and upward.

Pressure variations in an incompressible fluid at rest

In some fluid problems, fluids may be considered homogenous and incompressible *i.e*. density ρ is constant. Integrating the equation (L -7.10) with condition given in figure (Fig : L - 7.2), we have

$$\int_{P_1}^{P_2} dp = \int_{0}^{z} -\rho g \cdot dz$$

$$P_2 - P_1 = -\rho g z$$



Pressure variation in an incompressible fluid

This indicates that the pressure increases linearly from the free surface in an incompressible static fluid as illustrated by the linear distribution in the above figure.

Scales of pressure measurement

Fluid pressures can be measured with reference to any arbitrary datum. The common datum are

1. Absolute zero pressure.
2. Local atmospheric pressure

When absolute zero (complete vacuum) is used as a datum, the pressure difference is called an absolute pressure, P_{abs} .

When the pressure difference is measured either above or below local atmospheric pressure, P_{local}, as a datum, it is called the gauge pressure. Local atmospheric pressure can be measured by mercury barometer.

At sea level, under normal conditions, the atmospheric pressure is approximately 101.043 kPa.

As illustrated in figure(Fig : L -7.2),

When $P_{abs} < P_{local}$

$$P_{gauge} = P_{local} - P_{abs} \qquad \text{L - 7.12}$$

Note that if the absolute pressure is below the local pressure then the pressure difference is known as vacuum suction pressure.

Example 1 :

Convert a pressure head of 10 m of water column to kerosene of specific gravity 0.8 and carbon-tetra-chloride of specific gravity of 1.62.

Solution

Given data:

Height of water column, $h_1 = 10$ m

Specific gravity of water $s_1 = 1.0$

Specific gravity of kerosene $s_2 = 0.8$

Specific gravity of carbon-tetra-chloride, $s_3 = 1.62$

For the equivalent water head

Weight of the water column = Weight of the kerosene column.

So, $\square\ g\ h_1\ s_1 = \square\ g\ h_2\ s_2 = \square\ g\ h_3\ s_3$

Answer:- 12.5 m and 6.17 m.

Example 2

Determine (a) the gauge pressure and (b) The absolute pressure of water at a depth of 9 m from the surface.

Solution

Given data:

Depth of water = 9 m

the density of water = 998.2 kg/m^3

And acceleration due to gravity = 9.81 m/s^2

Thus the pressure at that depth due to the overlying water is P = r gh = 88.131 kN/m^2

Case a) as already discussed, gauge pressure is the pressure above the normal atmospheric pressure.

Thus, the gauge pressure at that depth = 88.131 kN/m^2

Case b) The standard atmospheric pressure is 101.213 kN/m^2

Thus, the absolute pressure as P_{abs} = 88.131+101.213 = 189.344 kN/m^2

Answer: 88.131 kN/m^2 ; 101.213 kN/m^2

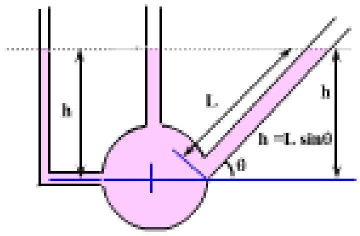
Manometers: Pressure Measuring Devices

Manometers are simple devices that employ liquid columns for measuring pressure difference between two points.

In Figure(L 8.1), some of the commonly used manometers are shown.

In all the cases, a tube is attached to a point where the pressure difference is to be measured and its other end left open to the atmosphere. If the pressure at the point *P* is higher than the local atmospheric pressure the liquid will rise in the tube. Since the column of the liquid in the tube is at rest, the liquid pressure *P* must be balanced by the hydrostatic pressure due to the column of liquid and the superimposed atmospheric pressure, P_{atm} .

$$P = \rho g h + P_{atm}$$



Simple Manometer

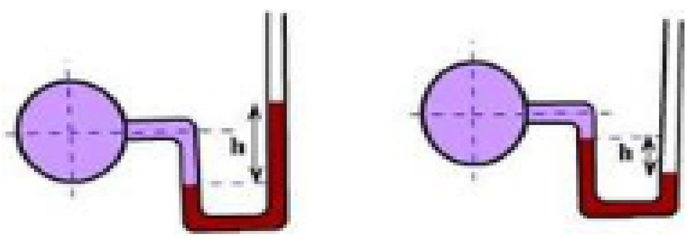
This simplest form of manometer is called a *Piezometer* . It may be inadequate if the pressure difference is either very small or large.

U - Tube Manometer

In (Fig : L -8.2), a manometer with two vertical limbs forms a U-shaped measuring tube. A liquid of different density □□ is used as a manometric fluid. We may recall the Pascal's law which states that the pressure on a horizontal plane in a continuous fluid at rest is the same. Applying this equality of pressure at points B and C on the plane gives

$$P + \rho g h = P_{atm} + \rho_1 g h_1$$

$$P - P_{atm} = \rho_1 g h_1 - \rho g h$$



U-tube Manometer

Inclined Manometer

A manometer with an inclined tube arrangement helps to amplify the pressure reading, especially in low press range. A typical arrangement of the same is shown in Fig. L-8.3.

The pressure at O is

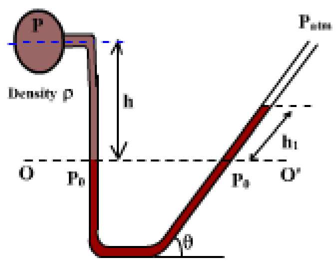
$$P_0 = P + \rho g h$$

The pressure at O is

$$P_{0'} = P_{atm} + \rho_1 g h_1 \sin\theta$$

Equating the pressures, we have

$$P_0 - P_{atm} = \rho_1 g h_1 \sin\theta - \rho g h$$



Inclined Manometer

At the same pressure difference, Equations (1) and (2) indicate that inclined tube manometer amplifies the length of measurement by $\frac{1}{\sin\theta}$, which is the primary advantage of such type of manometer.

Differential Manometers

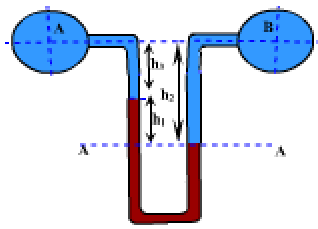
Differential Manometers measure difference of pressure between two points in a fluid system and cannot measure the actual pressures at any point in the system.

Some of the common types of differential manometers are

a. Upright U-Tube manometer
b. Inverted U-Tube manometer
c. Inclined Differential manometer
d. Micro manometer

Upright U-Tube manometer:

As shown in Fig. : L-8.4, an upright U-tube manometer is connected between points A and B. The difference of pressure between the points may be calculated by balancing pressure in a horizontal plane, the lowest interface A-A is used for this case.



Upright U-tube Manometer

$$P_A + \rho_1 g h_1 + \rho_3 g h_3 = P_B + \rho_2 g h_2$$

or

$$P_A - P_B = \rho_2 g h_2 - \rho_1 g h_1 - \rho_3 g h_3$$
$$= (\rho_2 h_2 - \rho_1 h_1 - \rho_3 h_3)\, g$$

Inverted U-Tube manometer:

The manometer fluid used in this type of manometer is lighter than the working fluids. Thus the height difference in two limbs is enhanced. This is therefore suitable for measurement of small pressure difference in liquids. For the configurations given in Fig. L-8.1.

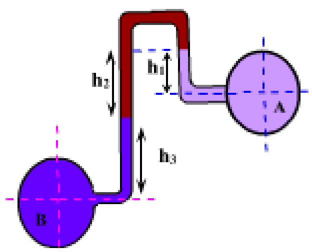


Fig. L-8.5 Inverted Manometer

$$P_A - \rho_1 g h_1 = P_B - \rho_2 g h_2 - \rho_3 g h_3$$ Or $$P_A - P_B = (\rho_1 h_1 - \rho_2 h_2 - \rho_3 h_3)\, g$$

If the two points A and B are at the same level and the same fluid is used, then $P_1 = P_2 = P$ and $h_2 + h_3 = h_1$.

The above equation becomes $P_A - P_B = (\rho_1 - \rho_3)\, h_3 g$

Inclined Differential Manometer

In this type of manometer a narrow tube is connected to a reservoir at an inclination. The cross section of the reservoir is larger than that of the tube. Fluctuations in the reservoir may be ignored. As shown in Fig.L-8.6, the initial liquid level in both the reservoir and the tube is at o-o. The application of the differential pressure liquid level of the reservoir drops by Δh, whereas h is the rising level in the tube. Therefore

$$P_A = P_B + (h + \Delta h)\rho g$$

Since the volume of liquid displaced in the reservoir equals to the volume of liquid in the tube, we can define

$$A \cdot \Delta h = a \cdot L$$

Where 'A' and 'a' are the cross sectional areas of the reservoir and the tube respectively. Then the equation becomes $P_A - P_B = (h + \frac{a}{A}L)\rho g$

In practice, the reservoir area is much larger than that of the tube; the ratio $\frac{a}{A}$ is negligible and the above equation is reduced to $P_A - P_B = \rho g L \sin\theta$; $h = L \sin\theta$

Micro manometer:

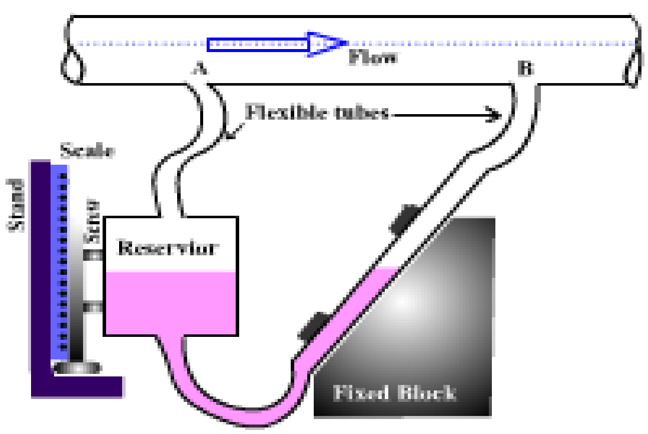


Fig. L-8.6: Micro manometer

A typical micro-manometer tube arrangement as shown in fig has a reservoir which can be moved up and down by means of micrometer screw. A flexible tube is connected between point A and the reservoir. Another flexible tube connecting point B and the other end of the reservoir is placed on an inclined surface. A reference mark 'R' is provided on the inclined portion of the tube. Before application of the pressure, the level of the reservoir is moved so as to coincide this level with the reference mark. When a pressure difference is applied, the liquid levels will be disturbed. The micrometer arrangement is then adjusted to vary the reservoir level so as to coincide with the reference. The extent of movement of the micrometer screw gives the pressure difference between the two points A and B.

Example 1:

Two pipes on the same elevation convey water and oil of specific gravity 0.88 respectively. They are connected by a U-tube manometer with the manometric liquid having a specific gravity of 1.25. If the manometric liquid in the limb connecting the water pipe is 2 m higher than the other find the pressure difference in two pipes.

Solution :

Given data:

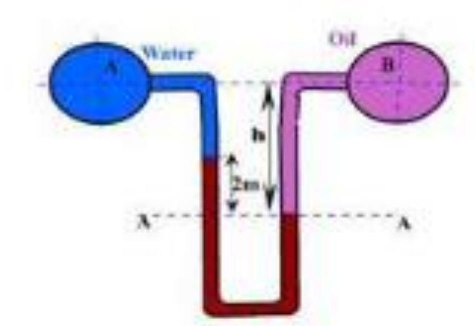
Height difference = 2 m

Specific gravity of oil s = 0.88

Specific gravity of manometric liquid s = 1.25

Equating pressure head at section (A-A)

$$P_A + 2\times 1.25\rho_W\ g + (h-2)\rho_W\ g = P_B + h\times 0.88\rho_W\ g$$



h = 5 m

Fig. Ex1

Substituing h = 5 m and density of water 998.2 kg/m^3 we have $P_A - P_B = 10791$

Example 2:

A two liquid double column enlarged-ends manometer is used to measure pressure difference between two points. The basins are partially filled with liquid of specific gravity 0.75 and the lower portion of U-tube is filled with mercury of specific gravity 13.6. The diameter of the basin is 20 times higher than that of the U-tube. Find the pressure difference if the U-tube reading is 25 mm and the liquid in the pipe has a specific weight of 0.475 N/m^3.

Solution:

Given data: U-tube reading 25 mm
Specific gravity of liquid in the basin 0.75
Specific gravity of Mercury in the U-tube13.6
As the volume displaced is constant we have,

$$Y = 25\frac{a}{A} = 25 \times \frac{1}{20^2}$$

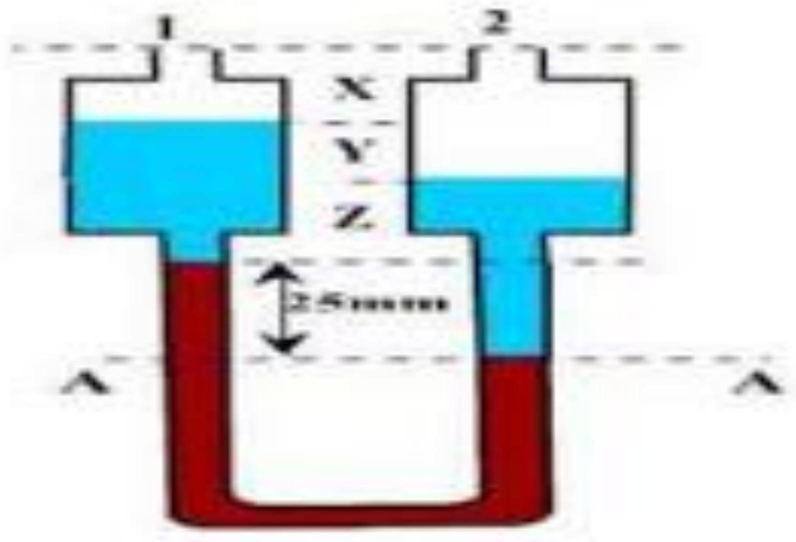


Fig. Ex 2

Equating pressure head at (A--A)

$$P_1 + X\frac{0.475}{1000}\rho_w g + (Z+Y)\rho_w g \times 0.75 + 25 \times 13.6\rho_w g$$

$$= P_2 + (X+Y)\frac{0.475}{1000}\rho_w g + (Z+25)\times 0.75\rho_w g$$

Put the value of Y while X and Z cancel out.
Answer: 31.51 kPa

Example 3:

As shown in figure water flows through pipe A and B. The pressure difference of these two points is to be measured by multiple tube manometers. Oil with specific gravity 0.88 is in the upper portion of inverted U-tube and mercury in the bottom of both bends. Determine the pressure difference.

Solution

Given data: Specific gravity of the oil in the inverted tube 0.88
Specific gravity of Mercury in the U-tube13.6

Calculate the Pressure difference between each two point as follow
$P_2 - P_1 = h \rho g = h S \rho_w g$

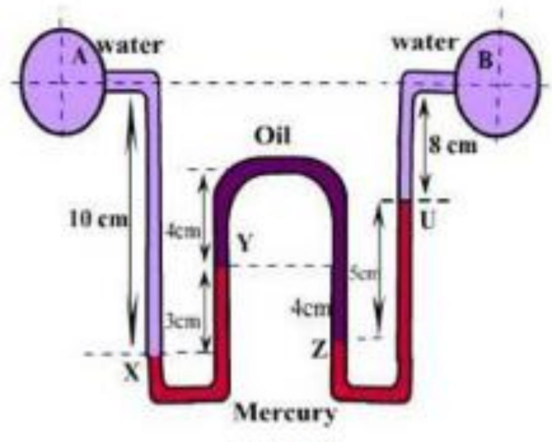


Fig. Ex3

Start from one and i.e. P_A or P_B

$$\text{Now, } P_X = P_A + 10\rho_w g$$
$$\text{Similarly, } P_Y = P_X - 3\times 13.6\rho_w g$$
$$P_Z = P_Y + 4\times 0.88\rho_w g$$
$$P_U = P_Z - 5\times 13.6\rho_w g$$
$$P_B = P_U - 8\rho_w g$$

Rearranging and summing all these equations we have $P_A - P_B = 103.28\ \square_w\ g$

Example 4:

A manometer connected to a pipe indicates a negative gauge pressure of 70 mm of mercury . What is the pressure in the pipe in N/m^2 ?

Solution :

Given data:

Manometer pressure- 70 mm of mercury (Negative gauge pressure)

A pressure of 70 mm of Mercury, P = r gh = 9.322 kN/m^2

Also we know the gauge pressure is the pressure above the atmosphere.

Thus a negative gauge pressure of 70 mm of mercury indicates the absolute pressure of

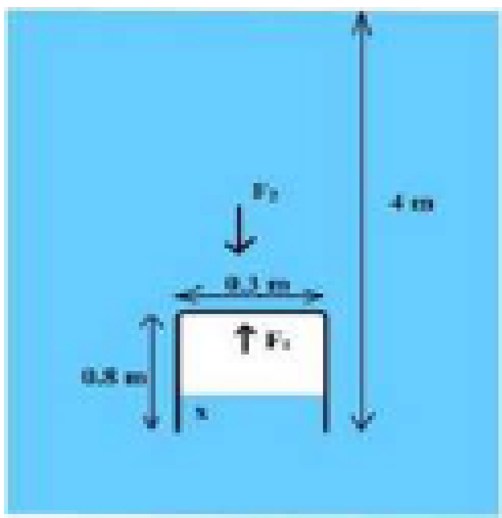
$P_{abs} = 101.213 + (-9.322) = 91.819\ kN/m^2$

Answer: $91.819\ kN/m^2$

Example 5:

An empty cylindrical bucket with negligible thickness and weight is forced with its open end first into water until its lower edge is 4m below the water level. If the diameter and length of the bucket are 0.3m and 0.8m respectively and the trapped water remains at constant temperature. What would be the force required to hold the bucket in that position atmospheric pressure being $1.03\ N/cm^2$

Solution :



Let, the water rises a height *x* in the bucket

By applying the Boyle's Law at constant temperature we have

$$p_1 \times (0.3)^2 \times \frac{\pi}{4} \times (0.8 - x) = p_{atm} \times (0.3)^2 \frac{\pi}{4} \times 0.8$$

Also, Downward pressure ion the bucket, $p_1 = p_{atm} + (4 - x) \times 9810$

Solve for, p_1 and x.

$$p_1 = 6.46 \times 10^4\ N/m^2$$

$$x = 0.610m$$

Total upward force exerted by the trapped water $F_1 = p_1 \times \frac{\pi}{4} \times 0.3^2 = 4.57 \times 10^3\ N/m^2$

Downward force due to the overlying water and the Atmospheric Pressure

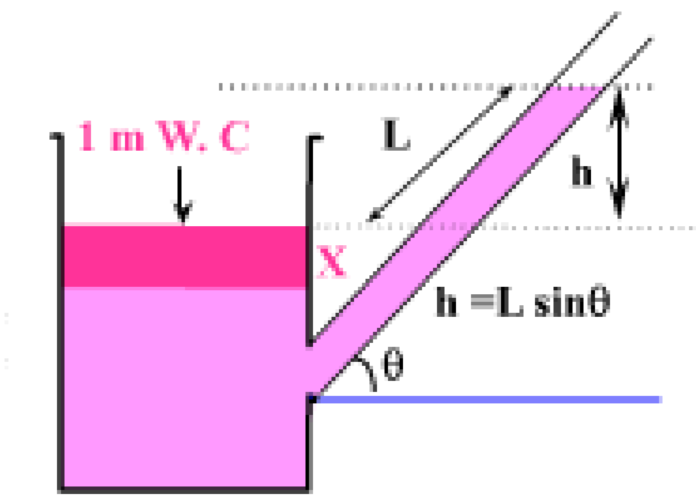
$$F_2 = [1.03\times10^4 + 9810\times(4-0.8)]\times\frac{\pi}{4}\times0.3_{2}$$

Answer: 1.62KN

Example 6:

A pipe connected with a tank (diameter 3 m) has an inclination of □ with the horizontal and the diameter of the pipe is 20 cm. Determine the angle ? which will give a deflection of 5 m in the pipe for a gauge pressure of 1 m water in the tank. Liquid in the tank has a specific gravity of 0.88.

Solution :



Given data:

Diameter of tank = 3 m
Diameter of tube = 20 cm
Deflection in the pipe, L = 5 m
From the figure shown
h = L sin□□□
If X m fall of liquid in the tank rises L m in the tube. (Note that the volume displaced is the same in the tank is equal to the volume displaced in the pipe)

$$x\pi\frac{3^2}{4} = L\pi\frac{0.2^2}{4}$$

$$\text{or} \quad x = \frac{0.04L}{9}.$$

Difference of head = x + h = L sin q + 0.04 L/9

And $$\left\{L\sin\theta + \frac{0.04L}{9}\right\} \times 0.88 = 1$$

Substitute L = 5m in the above equation.

Answer: □ = 12.87 0

Hydrostatic force on submerged surfaces

Introduction

Designing of any hydraulic structure, which retains a significant amount of liquid, needs to calculate the total force caused by the retaining liquid on the surface of the structure. Other critical components of the force such as the direction and the line of action need to be addressed. In this module the resultant force acting on a submerged surface is derived.

Hydrostatic force on a plane submerged surface

Shown in Fig.L-9.1 is a plane surface of arbitrary shape fully submerged in a uniform liquid. Since there can be no shear force in a static liquid, the hydrostatic force must act normal to the surface.

Consider an element of area $d\bar{A}$ on the upper surface. The pressure force acting on the element is

$$d\bar{F} = -Pd\bar{A}$$

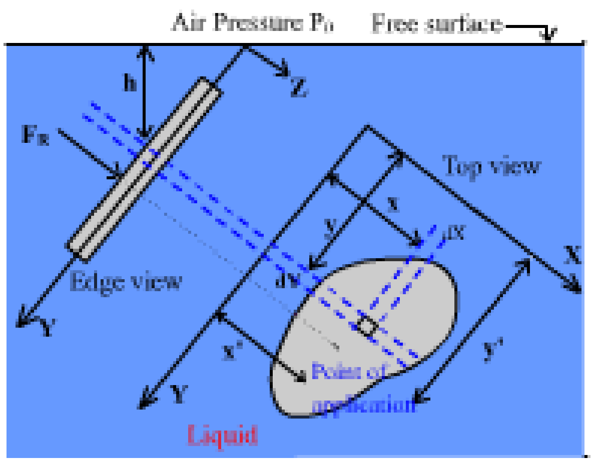


Fig : L - 9.1: Hydrostatic force and center of pressure on an inclined surface

Note that the direction of $d\bar{A}$ is normal to the surface area and the negative sign shows that the pressure force $d\bar{F}$ acts against the surface. The total hydrostatic force on the surface can be computed by integrating the infinitesimal forces over the entire surface area.

$$F = \int_A -P \cdot d\bar{A}$$

If *h* is the depth of the element, from the horizontal free surface as given in Equation (L2.9) becomes

$$\frac{dP}{dh} = \rho g = w \qquad \text{L-9.1}$$

If the fluid density ρ is constant and P 0 is the atmospheric pressure at the free surface, integration of the above equation can be carried out to determine the pressure at the element as given below

$$\begin{aligned} P &= P_0 + \int_0^h w\,dh \\ &= P_0 + wh \end{aligned} \qquad \text{L-9.2}$$

Total hydrostatic force acting on the surface is

$$\begin{aligned} F &= \int_A P \cdot d\bar{A} \\ &= \int_A \left(P_0 + wh\right) \cdot d\bar{A} \\ &= \int_A \left(P_0 + w \cdot y \sin\theta\right) \cdot d\bar{A} \\ &= P_0 A + w \cdot \sin\theta \int_A y \cdot d\bar{A} \end{aligned} \qquad \text{L-9.3}$$

The integral $\int_A y \cdot d\bar{A}$ is the first moment of the surface area about the x-axis.

If y_c is the y coordinate of the centroid of the area, we can express

$$\int_A y \cdot d\bar{A} = y_c \cdot A \qquad \text{L-9.4}$$

in which *A* is the total area of the submerged plane.

Thus

$$F = P_0 \cdot A + w \sin\theta \cdot (y_c A)$$
$$= P_c A \qquad \text{L-9.5}$$

This equation says that the total hydrostatic force on a submerged plane surface equals to the pressure at the centroid of the area times the submerged area of the surface and acts normal to it

Centre of Pressure (CP)

The point of action of total hydrostatic force on the submerged surface is called the Centre of Pressure (CP). To find the co-ordinates of CP, we know that the moment of the resultant force about any axis must be equal to the moment of distributed force about the same axis. Referring to Fig. L-9.2, we can equate the moments about the x-axis.

$$Y_{cp} F = \int_A y \cdot P \cdot dA \qquad \text{L-9.6}$$

Neglecting the atmospheric pressure ($P_0 = 0$) and substituting $F = w \sin\theta \cdot y_c A$, $P = wh$ and $h = y \sin\theta$,

We get $$Y_{cp} \cdot w \sin\theta \cdot y_c A = w \sin\theta \int_A y^2 \cdot dA$$

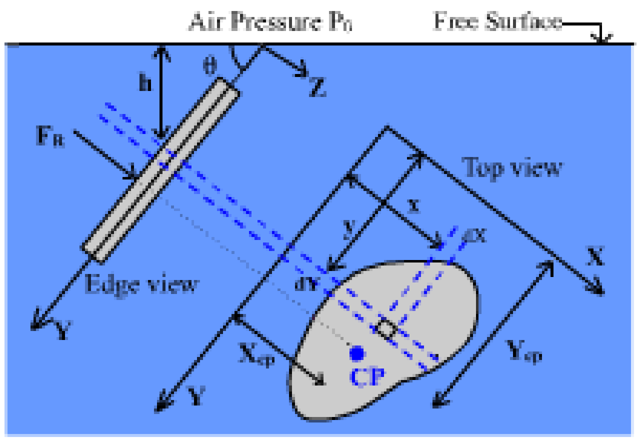


Fig. L-9.2 : Centre of pressure

We get $$Y_{cp} \cdot w\sin\theta \cdot y_c A = w\sin\theta \int_A y^2 \cdot dA$$

$$Y_{cp} = \frac{\int_A y^2 \cdot dA}{y_c A}$$

$$= \frac{\int_A y^2 \cdot dA}{\int y \cdot dA}$$

$$= \frac{\text{second moment of area about '0'}}{\text{first moment of area about '0'}}$$

From parallel-axis theorem

$$I_{xx} = I_{xc} + A \cdot y_c^{\,2}$$

Where I_{xc} is the second moment of the area about the centroidal axis.

$$Y_{cp} = \frac{I_{xc} + A \cdot y_c^{\,2}}{A \cdot y_c}$$

$$= \frac{I_{xc}}{A \cdot y_c} + y_c$$

L-9.8

This equation indicates that the centre of the pressure is always below the centroid of the submerged plane. Similarly, the derivation of x_{cp} can be carried out

Hydrostatic force on a Curved Submerged surface

On a curved submerged surface as shown in Fig. L-9.3, the direction of the hydrostatic pressure being normal to the surface varies from point to point. Consider an elementary area $d\bar{A}$ in the curved submerged surface in a fluid at rest. The pressure force acting on the element is

$$d\vec{F} = Pd\vec{A}$$

The total hydrostatic force can be computed as

$$\vec{F} = \int_A -Pd\vec{A}$$

Note that since the direction of the pressure varies along the curved surface, we cannot integrate the above integral as it was carried out in the previous section. The force vector $\overline{F}$ is expressed in terms of its scalar components as

$$\overline{F} = F_x\hat{i} + F_y\hat{j} + F_z\hat{k}$$

in which $F_x, F_y \text{ and } F_z$ represent the scalar components of F in the x , y and z directions respectively.

For computing the component of the force in the x-direction, the dot product of the force and the unit vector ($\hat{i}$ i) gives

$$F_x = \int d\overline{F}\cdot\hat{i}$$
$$= \int_A -PdA\cdot\hat{i}$$
$$= -\int_A PdA_x$$

Where dA_x is the area projection of the curved element on a plane perpendicular to the x-axis. This integral means that each component of the force on a curved surface is equal to the force on the plane area formed by projection of the curved surface into a plane normal to the component. The magnitude of the force component in the vertical direction (z direction)

$$F_z = \int_{A_z} PdA_z$$

Since $P = P_0 + wh$ and neglecting P_0, we can write

$$F_z = \int_{A_z} wh\cdot dA_z$$
$$= \int wd\nu$$

in which is the weight of liquid above the element surface. This integral shows that the z-component of the force (vertical component) equals to the weight of liquid between the submerged surface and the free surface. The line of action of the component passes through the centre of gravity of the volume of liquid between the free surface and the submerged surface

Example 1 :

A vertical gate of 5 m height and 3 m wide closes a tunnel running full with water. The pressure at the bottom of the gate is 195 kN/m ^{2} . Determine the total pressure on the gate and position of the centre of the pressure.

Solution

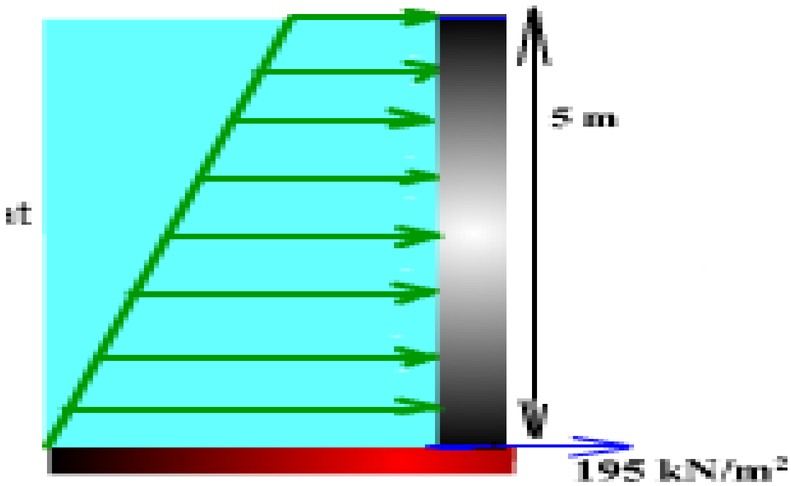


Fig. Ex1

Given data: Area of the gate = 5x3 = 15 m ^{2}

The equivalent height of water which gives a pressure intensity of 195 kN/m^2 at the bottom.

h = P/w =19.87m.

Total force $F = wA\bar{x}$.

And $\bar{x} = 19.87 - 2.5 = 17.37 m$

Centre of Pressure $\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$ $[I_G = bd^3/12]$

Answer: 2.56MN and 17.49 m

Example 2 :

A vertical rectangular gate of 4m x 2m is hinged at a point 0.25 m below the centre of gravity of the gate. If the total depth of water is 7 m what horizontal force must be applied at the bottom to keep the gate closed?

Solution

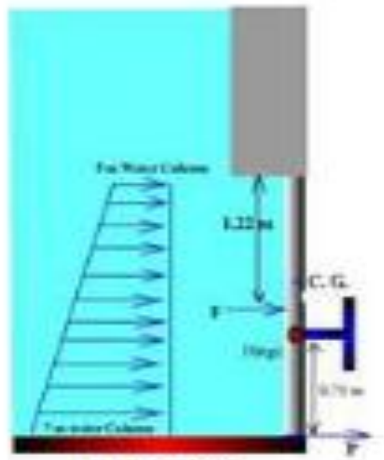


Fig. Ex2

Given data: Area of the gate = 4x2 = 8 m^2

Depth of the water = 7 m

Hydrostatic force on the gate

$$F = wA\bar{x} \qquad \bar{x} = 5+1 = 6 \text{ m}$$
$$= 4.7 \times 10^5 N$$

$$\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}} = 6.22m$$

Taking moments about the hinge we get, $F \times 0.03 = P \times 0.75$

Answer: 18.8 kN.

Buoyancy

Introduction

In our common experience we know that wooden objects float on water, but a small needle of iron sinks into water. This means that a fluid exerts an upward force on a body which is immersed fully or partially in it. The upward force that tends to lift the body is called the buoyant force, F_b.

The buoyant force acting on floating and submerged objects can be estimated by employing hydrostatic principle.

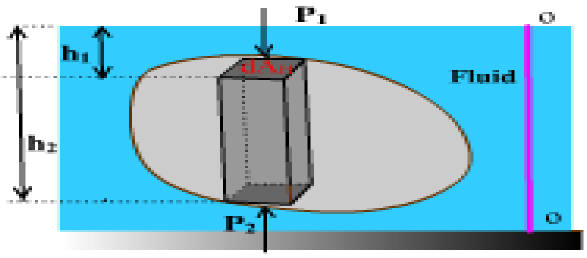


Fig L 10.1 : Buoyant force

With reference to figure(L- 10.1), consider a fluid element of area dA_H . The net upward force acting on the fluid element is

$$dF_B = (P_2 - P_1)dA_H$$
$$= w(h_2 - h_1)dA_H$$

The total upward buoyant force becomes

$$F_B = \int w(h_2 - h_1)dA_H = w(\text{volume of the body})$$ L-10.2

This result shows that the buoyant force acting on the object is equal to the weight of the fluid it displaces.

Center of Buoyancy

The line of action of the buoyant force on the object is called the center of buoyancy. To find the centre of buoyancy, moments about an axis OO can be taken and equated to the moment of the resultant forces. The equation gives the distance to the centeroid to the object volume.

The centeroid of the displaced volume of fluid is the centre of buoyancy, which, is applicable for both submerged and floating objects. This principle is known as the Archimedes principle which states:

“A body immersed in a fluid experiences a vertical buoyant force which is equal to the weight of the fluid displaced by the body and the buoyant force acts upward through the centroid of the displaced volume".

Buoyant force in a layered fluid

As shown in figure (L-10.2) an object floats at an interface between two immiscible fluids of density ρ_1 and ρ_2 .

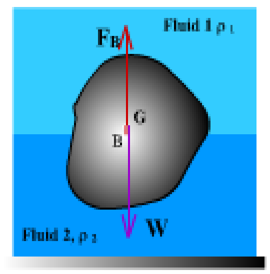


Fig. L-10.2: Buoyant force in a layered fluid

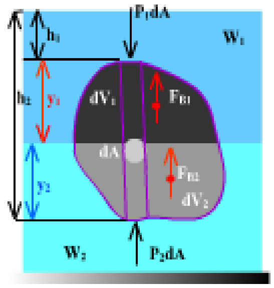


Fig. L-10.3: Element with hydrostatics forces

Considering the element shown in Figure L-10.3, the buoyant force F_B is

$$F_B = \int dF_B = \int \rho_1 g dV_1 + \int \rho_2 g dV_2$$
$$= \sum_1^n \rho_i g \left(\text{displaced volume}\right)_i \qquad \text{L-10.3}$$

where dV_1 and dV_2 are the volumes of fluid element submerged in fluid 1 and 2 respectively. The centre of buoyancy can be estimated by summing moments of the buoyant forces in each fluid volume displaced.

Buoyant force on a floating body

When a body is partially submerged in a liquid, with the remainder in contact with air (as shown in figure), the buoyant force of the body can also be computed using equation (L-10.3). Since the specific weight of the air (11.8 N/m^3) is negligible as compared with the specific weight of the liquid (for example specific weight of water is 9800 kN/m^3),we can neglect the weight of displaced air. Hence, equation (L-10.3) becomes

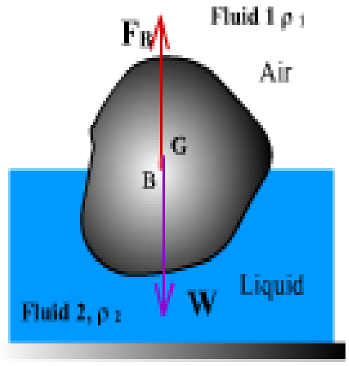


Fig. L-10.4: Partially submerged body

$F_B = \rho g$ (Displaced volume of the submerged liquid)

= The weight of the liquid displaced by the body.

The buoyant force acts at the centre of the buoyancy which coincides with the centeroid of the volume of liquid displaced.

Example 1:

A large iceberg floating in sea water is of cubical shape and its specific gravity is 0.9 If 20 cm proportion of the iceberg is above the sea surface, determine the volume of the iceberg if specific gravity of sea water is 1.025.

Solution:

Let the side of the cubical iceberg be *h*.

Total volume of the iceberg = h^3

volume of the submerged portion is = $(h - 20) \times h^2$

Now,
For flotation, weight of the iceberg = weight of the displaced water

$$(h-20)\times h^2 \times 1.025 \times w = h^3 \times 0.9 \times w$$
$$or,\ h = 164\ cm$$

The side of the iceberg is 164 cm.

Thus the volume of the iceberg is $4.41m^3$

Answer: $4.41m^3$

Stability

Introduction

Floating or submerged bodies such as boats, ships etc. are sometime acted upon by certain external forces. Some of the common external forces are wind and wave action, pressure due to river current, pressure due to maneuvering a floating object in a curved path, etc. These external forces cause a small displacement to the body which may overturn it. If a floating or submerged body, under action of small displacement due to any external force, is overturn and then capsized, the body is said to be in unstable. Otherwise, after imposing such a displacement the body restores its original position and this body is said to be in stable equilibrium. Therefore, in the design of the floating/submerged bodies the stability analysis is one of major criteria.

Stability of a Submerged body

Consider a body fully submerged in a fluid in the case shown in figure (Fig. L-11.1) of which the center of gravity (CG) of the body is below the centre of buoyancy. When a small angular displacement is applied a moment will generate and restore the body to its original position; the body is stable.

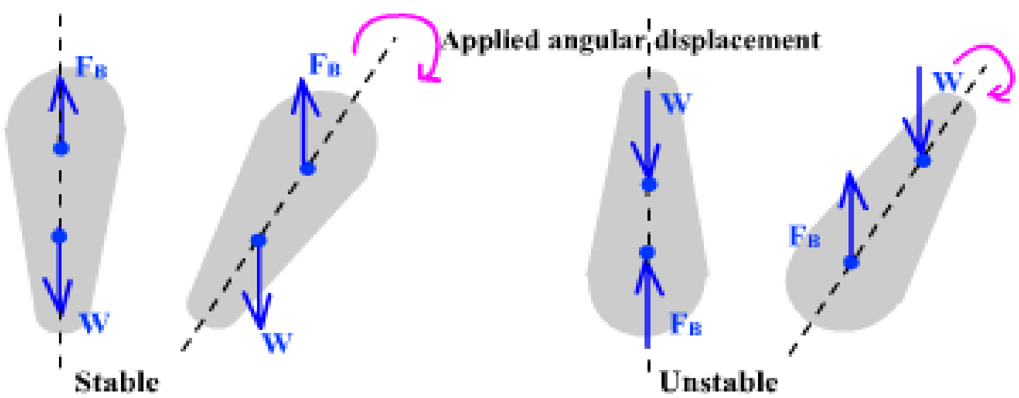


Fig. L-11.1 Fig. L-11.2

However if the CG is above the centre of buoyancy an overturning moment rotates the body away from its original position and thus the body is unstable (see Fig L-11.2). Note that as the body is fully submerged, the shape of the displaced fluid remains the same when the body is tilted. Therefore the centre of buoyancy in a submerged body remains unchanged.

Stability of a floating body

A body floating in equilibrium ($F_B = W$) is displaced through an angular displacement θ. The weight of the fluid W continues to act through G. But the shape of immersed volume of liquid changes and the centre of buoyancy relative to body moves from B to B 1 . Since the buoyant force F_B and the weight *W* are not in the same straight line, a turning movement proportional to ' $W x \theta$ ' is produced.

In figure (Fig. L-11.2) the moment is a restoring moment and makes the body stable. In figure (Fig. L-11.2) an overturning moment is produced. The point ' *M* ' at which the line of action of the new buoyant force intersects the original vertical through the *CG* of the body, is called the metacentre. The restoring moment

$$= W.x = W.\overline{GM}.\theta$$

Provided θ is small; $\sin\theta = \theta$ (in radians).

The distance *GM* is called the metacentric height. We can observe in figure that

Stable equilibrium: when *M* lies above *G* , a restoring moment is produced. Metacentric height *GM* is positive.

Unstable equilibrium: When *M* lies below *G* an overturning moment is produced and the metacentric height *GM* is negative.

Natural equilibrium: If *M* coincides with *G* neither restoring nor overturning moment is produced and *GM* is zero.

Determination of Meta-centric Height

Experimental method

The metacentric height of a floating body can be determined in an experimental set up with a movable load arrangement. Because of the movement of the load, the floating object is tilted with angle θ for its new equilibrium position. The measurement of θ is used to compute the metacentric height by equating the overturning moment and restoring moment at the new tilted position.

The overturning moment due to the movement of load *P* for a known distance, *x*, is $= P.x$

The restoring moment is $= W.\overline{GM}\theta$

For equilibrium in the tilted position, the restoring moment must equal to the overturning moment. Equating the same yields

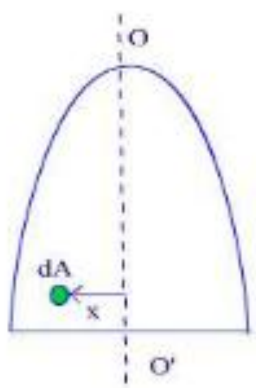
$$P.x = W.\overline{GM}.\theta$$

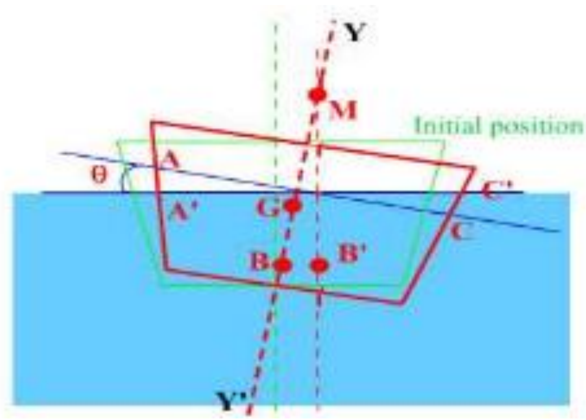
The metacentric height becomes

$$\overline{GM} = \frac{P.x}{W.\theta}$$

And the true metacentric height is the value of $\overline{GM}$ as $\theta \to 0$. This may be determined by plotting a graph between the calculated value of $\overline{GM}$ for various θ values and the angle θ.

Theoretical method:





For a floating object of known shape such as a ship or boat determination of meta-centric height can be calculated as follows.

The initial equilibrium position of the object has its centre of Buoyancy, B, and the original water line is AC. When the object is tilted through a small angle θ the center of buoyancy will move to new position B'. As a result, there will be change in the shape of displaced fluid. In the new position $A'C'$ is the waterline. The small wedge OCC' is submerged and the wedge OAA'

is uncovered. Since the vertical equilibrium is not disturbed, the total weight of fluid displaced remains unchanged.

Weight of wedge OAA' = Weight of wedge OCC'.

In the waterline plan a small area, *da* at a distance *x* from the axis of rotation *OO* uncover the volume of the fluid is equal to $DD'xda = x\theta da$

Integrating over the whole wedge and multiplying by the specific weight *w* of the liquid,

Weight of wedge $0AA' = \int_{OAA'} w\theta xda$

Similarly,

Weight of wedge $OCC' = \int_{OCC'} w\theta xda$

Equating Equations ( ) and (),

$$W\theta \int_{OAA'} xda = W\theta \int_{OCC} xda$$

$$\int xda = 0$$

in which, this integral represents the first moment of the area of the waterline plane about *OO* , therefore the axis *OO* must pass through the centeroid of the waterline plane.

Computation of the Meta-centric Height

Refer to Figure(), the distance $\overline{BM}$ is

$$BM = {BB'}/{\theta}$$

The distance BB' is calculated by taking moment about the centroidal axis YY'.

$$BB'wv_{A'ECCO} = \int_{AA'ECO} xwdv + \int_{OCC'} xwdv - \int_{OAA'} xwdv$$

The integral $\int_{AA'ECO} xwdv$ equals to zero, because YY' axis symmetrically divides the submerged portion $AA'ECO$.

At a distance *x* , $dv = Lx \tan\theta . dx$

Substituting it into the above equation gives

$$BB'V_{AECCO} = 0 + \int_{OCC} xLx\tan\theta dx - \int_{OAA'} xL(-x\tan\theta)dx$$
$$= \tan\theta \int_{waterline} x^2 dA_{waterline}$$
$$= \tan\theta \, I_O$$

Where I_0 is the second moment of area of water line plane about OO'. Thus,

Distance

$$\overline{BM} = BB'/\theta$$
$$= \frac{I_o \tan\theta}{\theta . V_{AECCO}}$$
$$= \frac{I_o}{V_{AECCO}}$$

Since,

$$\overline{BM} = \overline{GM} + \overline{BG}$$
$$\overline{GM} = \frac{I_o}{V_{submerged}} - \overline{BG}$$

Example:

A large icebergе, floating in seawater, is of cubical shape and its average specific gravity is 0.9. If a 20-cm -high proportion of the iceberg is above the surface of the water, determine the volume of the iceberg if the specific gravity of the seawater is 1.025.

Solution:

Let the side of the cubical iceberg is *h*.

Then volume of the submerged portion is = (*h* -20) x h^2

Total volume of the iceberg = h^3

Now,

For flotation, weight of the iceberg = weight of the displaced water

$$(h-20)\times h^2 \times 1.025 = h^3 \times 0.9$$
$$or,\ h = 164$$

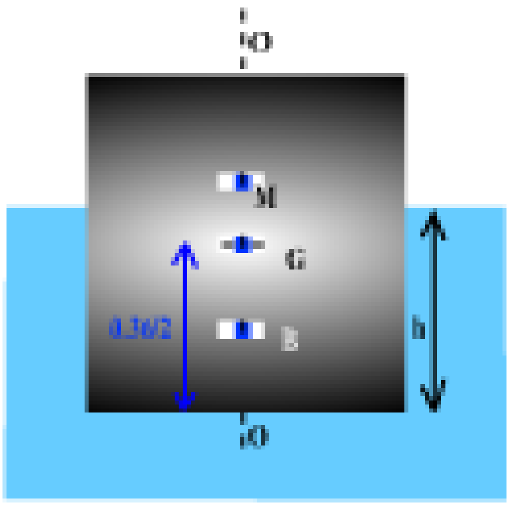
So, the side of the iceberg is 164 cm.

Thus the volume of the iceberg is $4.41 m^3$

Example

A log of wood of 1296 cm 2 cross section (square) with specific gravity 0.8 floats in water. Now if one of its edges is depressed to cause the log roll, find the period of roll.

Solution



Let, *h* be the depth of immersion and *L* be the length (perpendicular to the page)

Since the section is square its dimension should be 0.36 m x 0.36 m
For flotation
Weight of water displaced = Weight of the log

$$L \times 0.1296 \times 0.8 = h \times 0.36 \times L$$

Then, *h* = 0.288 m.

$$\overline{BG} = \frac{0.36}{2} - \frac{h}{2} = 0.036$$

$$\overline{BM} = \frac{I_0}{V_{submerged}} = \frac{\frac{1}{12} \times L \times 0.36^3}{0.36 \times 0.288 \times L} = 0.0375$$

$$\overline{GM} = \left(\overline{BM} - \overline{BG}\right) = 0.0015\ m$$

Time period, $T = \frac{2}{\pi}\sqrt{\frac{K_{G^2}}{GM}}$. and we have, $K_{G^2} = \frac{0.36^2}{12}$

Answer: 5.38 second

Example

To find the metacentre of a ship of 10,000 tonnes a weight of 55 tonnes is placed at a distance of 6 m from the longitudinal centre plane to cause a heel through an angle of 3^0. What is the metacentre height? Hence find the angle of heel and its direction when the ship is moving ahead and 2.8 MW is being transmitted by a single propeller shaft at the rate of 90 rpm.

Solution

Given data: Weight of the ship, W = 10 7 kg

Angle of heel ? = 3^0

Distance of the weight X = 6 m

Weight placed *w* = 5.5 x 10 4

Meta-centric height

$$h = \frac{w.X}{W\ \tan\theta}$$

$$= 0.629\ \text{m}.$$

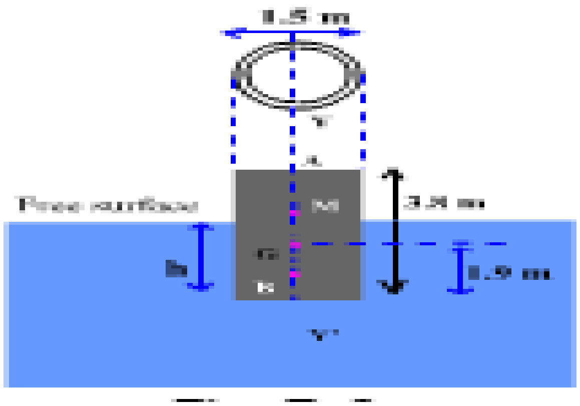
$$\text{Torque transmitted - T=P}/\omega = 2.97 \times 10^5 N - m.$$

$$\omega.h.\tan\theta' = T.$$

Answer:- 0.629 m and 0.27^0.

Example

A hollow cylinder closed in both end, of outside diameter 1.5 m and length of 3.8 m and specific weight 75 kN per cubic meter floats just in stable equilibrium condition. Find the thickness of the cylinder if the sea water has a specific weight of 10 kN per cubic meter.



Solution

Given data : Outside diameter 1.5 m
Length L = 3.8 m
Specific weight 75 kN/m ^{3}
Let the thickness t and immersion depth *h* .
For flotation
Weight of water displaced = weight of the cylinder

$$\frac{\pi}{4}(1.5^2 \times h) \times 10 = \left[\pi\{1.5 \times t\}\,3.8 + 2 \times \frac{\pi}{4} \times 1.5^2 \times t\right] \times 75$$

Assuming the thickness is very small compared to the diameter

h = 91 t

$$\overline{BM} = \frac{I_0}{V_{submerged}} = \frac{1.545 \times 10^{-3}}{t} \qquad \text{as we have } I_0 = \frac{\pi}{64}\ 1.5^4$$

$$\overline{BG} = \left[\frac{L}{2} - h\right] = [\frac{3.8}{2} - \frac{91}{2}t]$$

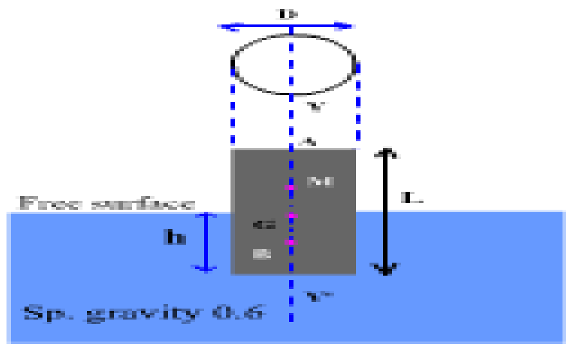
For the cylinder to be in equilibrium $\overline{BM} = \overline{BG}$

Solving for *t* we have *t* = 0.0409 *or* 0.000829m

Answer:- t = 0.83 mm

Example

A wooden cylinder of length L and diameter D is to be floated in stable equilibrium on a liquid keeping its axis vertical. What should be the relation between L and D if the specific gravity of liquid and that of the wood are 0.6 and 0.8 respectively?



Solution

Given data: Specific gravity of liquid = 0.6
Specific gravity of liquid = 0.8
If the depth of immersion is h
Weight of water displaced = weight of the cylinder

$$\frac{\pi}{4}D^2L \times 0.6 = \frac{\pi}{4}D^2h \times 0.8$$

The depth of immersion $h = \frac{3}{4}L$.

Height of centre of pressure from bottom x = $\frac{h}{2} = \frac{3}{8}L$

Then, $\overline{BM} = I/V = D^2/12L$

$$\overline{BG} = (\overline{OG} - \overline{OB}) = \frac{L}{8}$$

For Stable equilibrium

$$\overline{BM} > \overline{BG}$$

$$or \quad \frac{D^2}{12L} > \frac{L}{8}$$

Answer: L < 0.817D.

UNIT II

FLUID KINEMATICS

Introduction

Kinematics is the geometry of Motion.

Kinematics of fluid describes the fluid motion and its consequences without consideration of the nature of forces causing the motion.

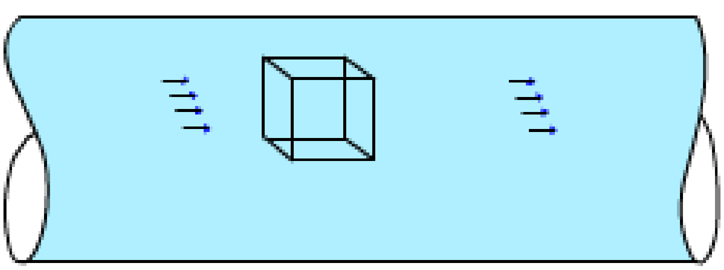
The fluid kinematics deals with description of the motion of the fluids without reference to the force causing the motion.

Thus it is emphasized to know how fluid flows and how to describe fluid motion. This concept helps us to simplify the complex nature of a real fluid flow.

When a fluid is in motion, individual particles in the fluid move at different velocities. Moreover at different instants fluid particles change their positions. In order to analyze the flow behavior, a function of space and time, we follow one of the following approaches

1. Lagarangian approach
2. Eularian approach

In the Lagarangian approach a fluid particle of fixed mass is selected. We follow the fluid particle during the course of motion with time



The fluid particles may change their shape, size and state as they move. As mass of fluid particles remains constant throughout the motion, the basic laws of mechanics can be applied to them at all times. The task of following large number of fluid particles is quite difficult. Therefore this approach is limited to some special applications for example re-entry of a spaceship into the earth's atmosphere and flow measurement system based on particle imagery.

In the Eularian method a finite region through which fluid flows in and out is used. Here we do not keep track position and velocity of fluid particles of definite mass. But, within the region, the field variables which are continuous functions of space dimensions (x , y , z) and time (t), are defined to describe the flow. These field variables may be scalar field variables, vector field variables and tensor quantities. For example, pressure is one of the scalar fields. Sometimes this finite region is referred as control volume or flow domain.

For example the pressure field 'P' is a scalar field variable and defined as

$$P = P(x, y, z, t)$$

Velocity field, a vector field, is defined as $\vec{V} = \vec{V}(x, y, z, t)$

Similarly shear stress τ is a tensor field variable and defined as

$$\tau = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix}$$

Note that we have defined the fluid flow as a three dimensional flow in a Cartesian co-ordinates system.

Advantages of Lagrangian Method:
1. Since motion and trajectory of each fluid particle is known, its history can be traced.
2. Since particles are identified at the start and traced throughout their motion, conservation of mass is inherent.

Disadvantages of Lagrangian Method:
1. The solution of the equations presents appreciable mathematical difficulties except

certain special cases and therefore, the method is rarely suitable for practical applications.

Types of Fluid Flow

Uniform and Non-uniform flow : If the velocity at given instant is the same in both magnitude and direction throughout the flow domain, the flow is described as uniform.

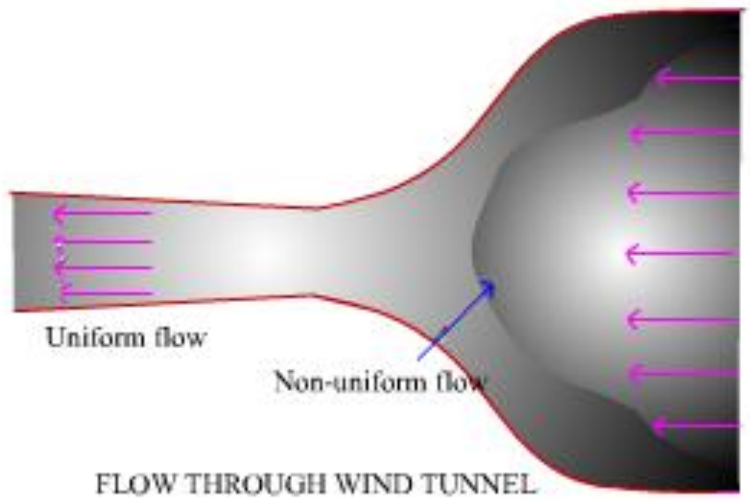


Fig. L-16.2 : Uniform and Non-uniform flow.

Mathematically the velocity field is defined as $\vec{V} = \vec{V}(t)$, independent to space dimensions (x , y , z).

When the velocity changes from point to point it is said to be non-uniform flow. Fig. shows uniform flow in test section of a well designed wind tunnel and describing non uniform velocity region at the entrance.

Steady and unsteady flows

The flow in which the field variables don't vary with time is said to be steady flow. For steady flow,

$$\frac{\delta V}{\delta t} = 0 \qquad \text{Or} \qquad \vec{V} = \vec{V}(x, y, z)$$

It means that the field variables are independent of time. This assumption simplifies the fluid problem to a great extent. Generally, many engineering flow devices and systems are designed to operate them during a peak steady flow condition.

If the field variables in a fluid region vary with time the flow is said to be unsteady flow.

$$\frac{\delta V}{\delta t} \neq 0 \qquad \vec{V} = \vec{V}(x, y, z, t)$$

Four possible combinations

Type	Example
1. Steady Uniform flow	Flow at constant rate through a duct of uniform cross-section (The region close to the walls of the duct is disregarded)
2. Steady non-uniform flow	Flow at constant rate through a duct of non-uniform cross-section (tapering pipe)
3. Unsteady Uniform flow	Flow at varying rates through a long straight pipe of uniform cross-section. (Again the region close to the walls is ignored.)
4. Unsteady non-uniform flow	Flow at varying rates through a duct of non-uniform cross-section.

One, two and three dimensional flows

Although fluid flow generally occurs in three dimensions in which the velocity field vary with three space co-ordinates and time. But, in some problem we may use one or two space components to describe the velocity field. For example consider a steady flow through a long straight pipe of constant cross-section. The velocity distributions shown in figure are independent of co-ordinate x and θ and a function of r only. Thus the flow field is one dimensional.

Fig. L-16.3

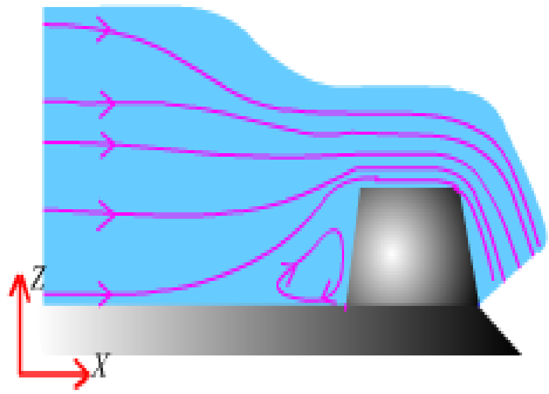


Fig. L-16.4

But in the case of flow over a weir of constant cross-section (), we can use two co-ordinate system x and z in defining the velocity field. So, this flow is a case of two dimensional flow. The reduction of independent space variable in a fluid flow problem makes it simpler to solve.

Laminar and Turbulent flow

In fluid flows, there are two distinct fluid behaviors experimentally observed. These behaviors were first observed by Sir Osborne Reynolds. He carried out a simple experiment in which water was discharged through a small glass tube from a large tank (the schematic of the experiment shown in Fig.). A colour dye was injected at the entrance of the tube and the rate of flow could be regulated by a valve at the out let.

When the water flowed at low velocity, it was found that the die moved in a straight line. This clearly showed that the particles of water moved in parallel lines. This type of flow is called laminar flow, in which the particles of fluid moves along smooth paths in layers. There is no exchange of momentum from fluid particles of one layer to the fluid particles of another layer.

This type of flow mainly occurs in high viscous fluid flows at low velocity, for example, oil flows at low velocity. Fig. shows the steady velocity profile for a typical laminar flow.

When the water flowed at high velocity, it was found that the dye colour was diffused over the whole cross section. This could be interpreted that the particles of fluid moved in very irregular paths, causing an exchange of momentum from one fluid particle to another. This type of flow is known as turbulent flow. The time variation of velocity at a point for the turbulent flow is shown in Fig

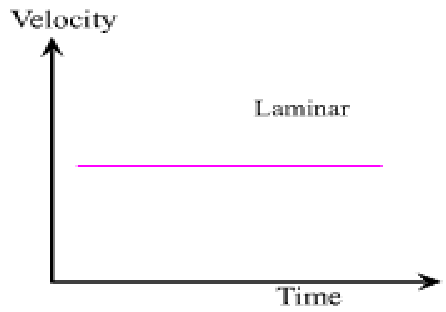


Fig-L16.5 : Velocity profile for laminar flow

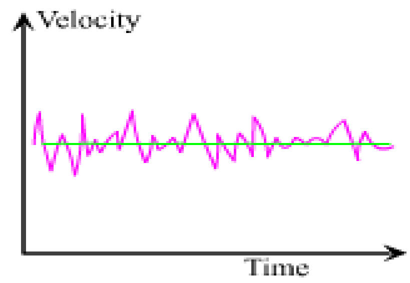


Fig. L-16.6 : Velocity profile for turbulent flow

It means that the flow is characterized by continuous random fluctuations in the magnitude and the direction of velocity of the fluid particles.

Velocity Field

Consider a uniform stream flow passing through a solid cylinder (Fig.). The typical velocities at different locations within the fluid domain vary from position to position at a particular time t . At different time instants this velocity distribution may change. Keeping this observation in mind, the velocity within a flow domain can be represented as function of position (x , y , z) and time t .

In the Cartesian co-ordinates the variation of velocity can be represented as a vector $\vec{V} = u\hat{i} + v\hat{j} + w\hat{k}$ where u , v , w are the velocity scalar components in x , y and z directions respectively.

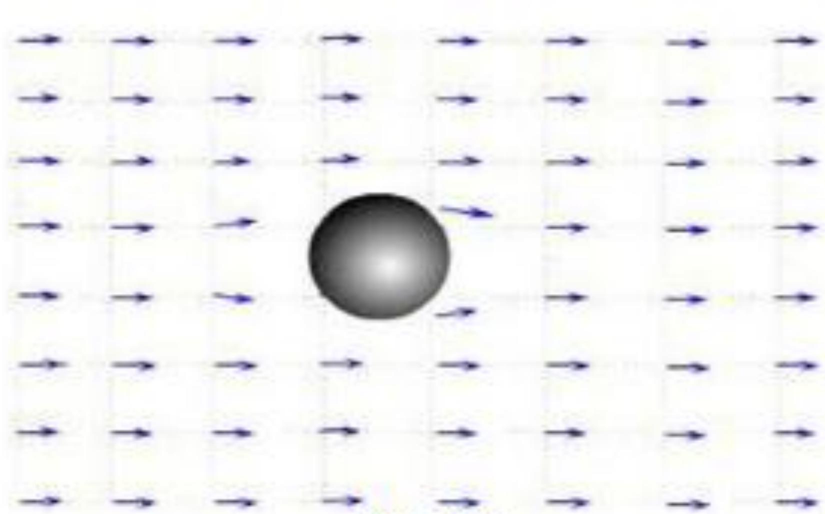


Fig. L-17.1

The scalar components *u* , *v* and *w* are dependent functions of position and time. Mathematically we can express them as

$$u = u(x, y, z, t)$$
$$v = v(x, y, z, t)$$
$$w = w(x, y, z, t)$$

This type of continuous function distribution with position and time for velocity is known as velocity field. It is based on the Eularian description of the flow. We also can represent the Lagrangian description of velocity field.

Let a fluid particle exactly positioned at point A moving to another point A' during time interval Δt. The velocity of the fluid particle is the same as the local velocity at that point as obtained from the Eulerian description

At time t , $\vec{V}$ particle at x , y , z $(t) = \vec{V}(x, y, z, t)$

At time $t + \Delta t$, $\vec{V}$ particle at x', y', z' $(t + \Delta t) = \vec{V}(x', y', z', t + \Delta t)$

This means that instead of describing the motion of the fluid flow using the Lagrangian description, the use of Eularian description makes the fluid flow problems quite easier to solve. Besides this difficult, the complete description of a fluid flow using the Lagrangian description requires to keep track over a large number of fluid particles and their movements with time. Thus, more computation is required in the Lagrangian description.

The Acceleration field

At given position A, the acceleration of a fluid particle is the time derivative of the particle's velocity.

Acceleration of a fluid particle: $\vec{a}_{particle} = \frac{d\vec{V}_{particle}}{dt}$

Since the particle velocity is a function of four independent variables (x , y , z and t), we can express the particle velocity in terms of the position of the particle as given below

$$\vec{a}_{particle} = \frac{d\vec{V}_{particle}}{dt} = \frac{d\vec{V}\left(x_{particle}, y_{particle}, z_{particle}\right)}{dt}$$

Applying chain rule, we get

$$\vec{a}_{particle} = \frac{\delta\vec{V}}{\delta t}\cdot\frac{dt}{dt} + \frac{\delta\vec{V}}{\delta x_{particle}}\frac{dx_{particle}}{dt} + \frac{\delta\vec{V}}{\delta y}\frac{dy_{particle}}{dt} + \frac{\delta\vec{V}}{\delta z}\frac{dz_{particle}}{dt}$$

Where δ and d are the partial derivative operator and total derivative operator respectively.

The time rate of change of the particle in the x -direction equals to the x -component of velocity vector, u . Therefore

$$\frac{dx_{particle}}{dt} = u$$

Similarly, $\frac{dy_{particle}}{dt} = v$

$$\frac{dz_{particle}}{dt} = w$$

As discussed earlier the position vector of the fluid particle (x particle , y particle , z particle) in the Lagranian description is the same as the position vector (x , y , z) in the Eulerian frame at time t and the acceleration of the fluid particle, which occupied the position (x , y , z) is equal to $a(x, y, z, t)$ in the Eularian description.

Therefore, the acceleration is defined by

$$\vec{a}_{(x,y,z,t)} = \frac{\delta\vec{V}}{\delta t} + u\frac{\delta\vec{V}}{\delta x} + v\frac{\delta\vec{V}}{\delta y} + w\frac{\delta\vec{V}}{\delta z}$$

in vector form

$$\vec{a}_{(x,y,z,t)} = \underbrace{\frac{\delta\vec{V}}{\delta t}}_{\text{(local accelaration)}} + \underbrace{\left(\vec{V}.\nabla\right)\vec{V}}_{\text{(convective accelartion)}}$$

where ∇ is the gradient operator.

The first term of the right hand side of equation represents the time rate of change of velocity field at the position of the fluid particle at time t. This acceleration component is also independent to the change of the particle position and is referred as the local acceleration. However the term $\left(\vec{V}.\nabla\right)\vec{V}$ accounts for the affect of the change of the velocity at various positions in this field. This rate of change of velocity because of changing position in the field is called the convective acceleration.

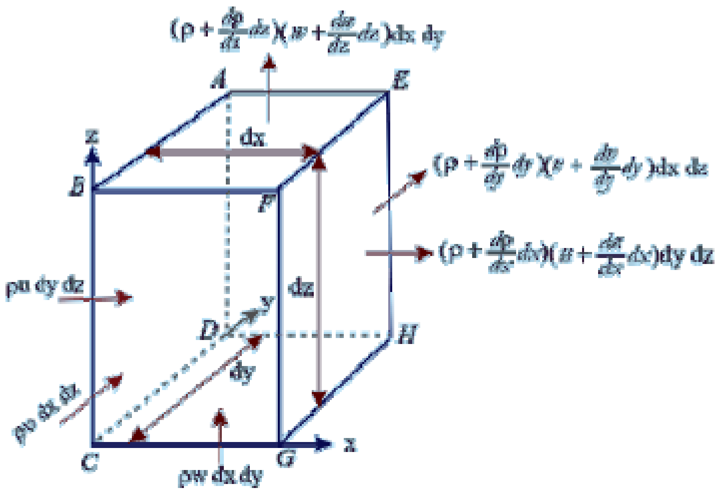
Type of Flow	Material Acceleration	
	Temporal	Convective
1. Steady Uniform flow	0	0
2. Steady non-uniform flow	0	exists
3. Unsteady Uniform flow	exists	0
4. Unsteady non-uniform flow	exists	exists

Continuity Equation - Differential Form

Derivation

1. The point at which the continuity equation has to be derived, is enclosed by an elementary control volume.

2. The influx, efflux and the rate of accumulation of mass is calculated across each surface within the control volume.



A Control Volume Appropriate to a Rectangular Cartesian coordinate system

Consider a rectangular parallelopiped in the above figure as the control volume in a rectangular cartesian frame of coordinate axes.

 Net efflux of mass along x -axis must be the excess outflow over inflow across faces normal to x -axis.

 Let the fluid enter across one of such faces ABCD with a velocity u and a density ρ.The velocity and density with which the fluid will leave the face EFGH will be $u+\frac{\partial u}{\partial x}dx$ and respectively (neglecting the higher order terms in δx).

 Therefore, the rate of mass entering the control volume through face ABCD = $\rho u\, dy\, dz$.

 The rate of mass leaving the control volume through face EFGH will be

$$=\left(\rho+\frac{\partial \rho}{\partial x}dx\right)\left(u+\frac{\partial u}{\partial x}dx\right)dy\,dz$$

$$=\left(\rho u+\frac{\partial}{\partial x}(\rho u)dx\right)dy\,dz$$

(neglecting the higher order terms in dx)

Similarly influx and efflux take place in all y and z directions also.

Rate of accumulation for a point in a flow field

$$\frac{\partial m}{\partial t} = \frac{\partial}{\partial t}\rho(d\forall) = \frac{\partial \rho}{\partial t}d\forall$$

Using, Rate of influx = Rate of Accumulation + Rate of Efflux

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

Transferring everything to right side

$$0 = \left[\left(\rho\frac{\partial u}{\partial x} + u\frac{\partial \rho}{\partial x}\right) + \left(\rho\frac{\partial v}{\partial y} + v\frac{\partial \rho}{\partial y}\right) + \left(\rho\frac{\partial w}{\partial z} + w\frac{\partial \rho}{\partial z}\right)\right]dx dy dz + \left(\frac{\partial \rho}{\partial t}\right)d\forall$$

$$\Rightarrow \left[\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w)\right]d\forall = 0$$

This is the Equation of Continuity for a compressible fluid in a rectangular Cartesian coordinate system.

Continuity Equation - Vector Form

The continuity equation can be written in a vector form as

$$\frac{\partial \rho}{\partial t} + \left(\frac{\partial}{\partial x}\hat{i} + \frac{\partial}{\partial y}\hat{j} + \frac{\partial}{\partial z}\hat{k}\right).\left[\rho u\hat{i} - \rho v\hat{j} + \rho w\hat{k}\right] = 0$$

or,

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{V}) = 0$$

Streamlines, Pathlines and Streakline

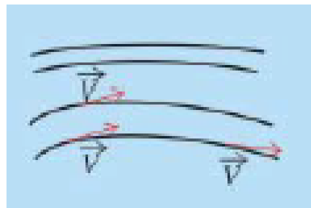
Streamlines

Definition: Streamlines are the Geometrical representation of the of the flow velocity.

Description:

- In the **Eulerian** method, the velocity vector is defined as a function of time and space coordinates.
- If for a fixed instant of time, a **space curve** is drawn so that it is **tangent** everywhere to the **velocity** vector, then this curve is called a **Streamline**.

Therefore, the Eulerian method gives a series of instantaneous streamlines of the state of motion.



Streamlines

Alternative Definition: A streamline at any instant can be defined as an imaginary curve or line in the flow field so that the tangent to the curve at any point represents the direction of the **instantaneous velocity** at that point.

In an **unsteady flow** where the velocity vector changes with time, the pattern of streamlines also **changes from instant to instant.**

In a **steady flow**, the orientation or the pattern of streamlines will be **fixed**.

From the above definition of streamline, it can be written as $\vec{V} \times d\vec{S} = 0$

1. $d\vec{S}$ is the length of an infinitesimal line segment along a streamline at a point .
2. $\vec{V}$ is the instantaneous velocity vector.

The above expression therefore represents the **differential equation of a streamline**. In a cartesian coordinate-system, representing

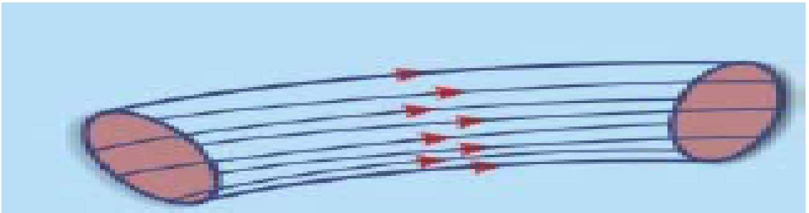
$$d\vec{S} = \vec{i}dx + \vec{j}dy + \vec{k}dz \qquad \vec{V} = \vec{i}u + \vec{j}v + \vec{k}w$$

the above equation may be simplified as

$$\frac{dx}{u} = \frac{dy}{v} = \frac{dz}{w}$$

Stream tube:

A bundle of neighboring streamlines may be imagined to form a passage through which the fluid flows. This passage is known as a **stream-tube**.



Properties of Stream tube:

1. The stream-tube is bounded on all sides by streamlines.

2. Fluid velocity does not exist across a streamline, no fluid may enter or leave a stream-tube except through its ends.

3. The entire flow in a flow field may be imagined to be composed of flows through streamtubes arranged in some arbitrary positions

Path Lines

Definition: A path line is the trajectory of a fluid particle of **fixed identity**

Path lines

A family of path lines represents the **trajectories of different particles**, say, P1, P 2, P3, etc.

Differences between Path Line and Stream Line

Path Line	Stream Line
This refers to a path followed by a fluid particle over a period of time.	This is an imaginary curve in a flow field for a fixed instant of time, tangent to which gives the instantaneous velocity at that point .
Two path lines can intersect each other as or a single path line can form a loop as different particles or even same particle can arrive at the same point at different instants of time.	Two stream lines can never intersect each other, as the instantaneous velocity vector at any given point is unique.

In a steady flow **path lines** are **identical** to **streamlines** as the **Eulerian and Lagrangian** versions become the **same**.

Vorticity

Definition: The vorticity Ω in its simplest form is defined as a vector which is equal to **two times the rotation vector**

$$\vec{\Omega} = 2\vec{\omega} = \nabla \times \vec{V}$$

For an **irrotational** flow, vorticity components are zero.

Vortex line:

If tangent to an imaginary line at a point lying on it is in the direction of the Vorticity vector at that point , the line is a **vortex line**.

The **general equation** of the **vortex line** can be written as,

$$\vec{\Omega} \times d\vec{s} = 0$$

In a rectangular cartesian cartesian coordinate system, it becomes

$$\frac{dx}{\Omega_x} = \frac{dy}{\Omega_y} = \frac{dz}{\Omega_z}$$

where,

$$\Omega_x = 2\omega_x$$

$$\Omega_y = 2\omega_y$$

$$\Omega_z = 2\omega_z$$

Vorticity components as vectors:

The vorticity is actually an **anti symmetric tensor** and its three distinct elements transform like the components of a vector in cartesian coordinates.

This is the reason for which the vorticity components can be treated as vectors.

Existence of Flow

- A fluid must obey the law of conservation of mass in course of its flow as it is a material body.
- For a Velocity field to exist in a fluid continuum, the velocity components must obey the **mass conservation principle**.
- Velocity components which follow the mass conservation principle are said to constitute a possible fluid flow
- Velocity components violating this principle, are said to describe an impossible flow.
- The existence of a physically possible flow field is verified from the principle of conservation of mass.

The detailed discussion on this is deferred to the next chapter along with the discussion on principles of**conservation of momentum and energy**.

Definition of rotation at a point:

The rotation at a point is defined as the **arithmetic mean** of the **angular velocities** of two perpendicular linear segments meeting at that point.

Example: The angular velocities of AB and AD about A are

$\frac{d\alpha}{dt}$ and $\frac{d\beta}{dt}$ respectively.

Considering the **anticlockwise direction as positive**, the rotation at A can be written as,

$$\omega_z = \frac{1}{2}\left(\frac{d\alpha}{dt} - \frac{d\beta}{dt}\right)$$

or

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

The suffix z in ω represents the rotation about z-axis.

When $u = u\ (x,\ y)$ and $v = v\ (x,\ y)$ the **rotation and angular deformation of a fluid element exist simultaneously**.

Special case : Situation of pure Rotation

$\frac{\partial v}{\partial x} = -\frac{\partial u}{\partial y}$, $\dot{\gamma}_{xy} = 0$ and $\omega_z = \frac{\partial v}{\partial x} = -\frac{\partial u}{\partial y}$

Observation:

- The linear segments AB and AD move with the same angular velocity (both in magnitude and direction).
- The included angle between them remains the same and no angular deformation takes place. This situation is known as **pure rotation.**

UNIT III

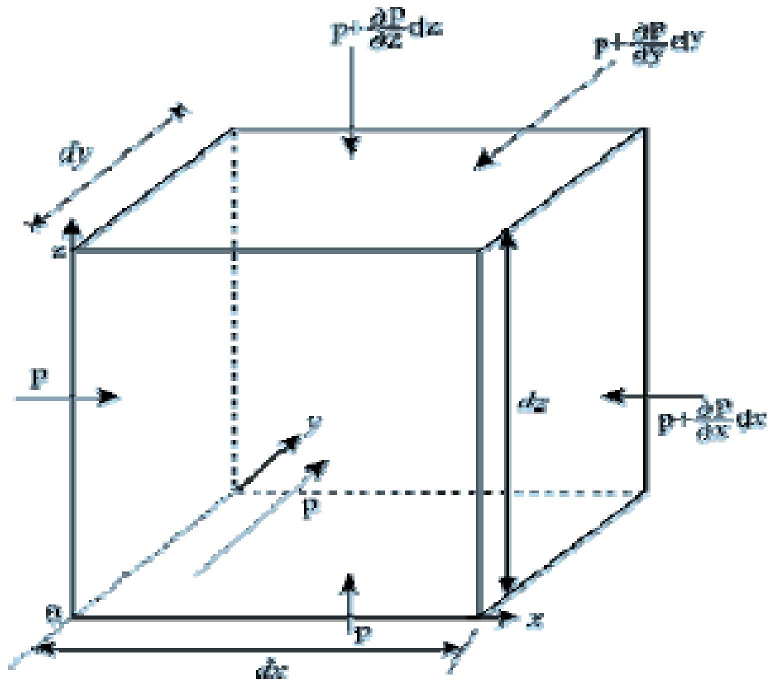
FLUID DYNAMICS

Euler and Navier Stokes Equation:

Euler's Equation: The Equation of Motion of an Ideal Fluid

Using the Newton's second law of motion the relationship between the velocity and pressure field for a flow of an inviscid fluid can be derived. The resulting equation, in its differential form, is known as Euler's Equation. The equation is first derived by the scientist Euler.

Derivation:
Let us consider an elementary parallelopiped of fluid element as a control mass system in a frame of rectangular cartesian coordinate axes as shown in Fig.. The external forces acting on a fluid element are the body forces and the surface forces.



A Fluid Element appropriate to a Cartesian Coordinate System

used for the derivation of Euler's Equation

Let Xx, Xy, Xz be the components of body forces acting per unit mass of the fluid element along the coordinate axes x, y and z respectively. The body forces arise due to external force fields like gravity, electromagnetic field, etc., and therefore, the detailed description of Xx, Xy and Xz are provided by the laws of physics describing the force fields. The surface forces for an inviscid fluid will be the pressure forces acting on different surfaces as shown in Fig. Therefore, the net forces acting on the fluid element along x, y and z directions can be written as

$$F_x = \rho X_x dx\, dy\, dz + p\, dy\, dz - \left(p + \frac{\partial p}{\partial x} dx\right) dy\, dz = \left(\rho X_x - \frac{\partial p}{\partial x}\right) dx\, dy\, dz$$

$$F_y = \rho X_y dx\, dy\, dz + p\, dx\, dz - \left(p + \frac{\partial p}{\partial y} dy\right) dx\, dz = \left(\rho X_y - \frac{\partial p}{\partial y}\right) dx\, dy\, dz$$

$$F_z = \rho X_z dx\, dy\, dz + p\, dy\, dx - \left(p + \frac{\partial p}{\partial z} dz\right) dx\, dz = \left(\rho X_z - \frac{\partial p}{\partial z}\right) dx\, dy\, dz$$

Since each component of the force can be expressed as the rate of change of momentum in the respective directions, we have

$$\frac{D}{Dt}(\rho\, dx\, dy\, dz\, u) = \left(\rho X_x - \frac{\partial p}{\partial x}\right) dxdydz$$

$$\frac{D}{Dt}(\rho\, dx\, dy\, dz\, v) = \left(\rho X_y - \frac{\partial p}{\partial y}\right) dxdydz$$

$$\frac{D}{Dt}(\rho\, dx\, dy\, dz\, w) = \left(\rho X_z - \frac{\partial p}{\partial z}\right) dxdydz$$

s the mass of a control mass system does not change with time, $\rho dxdydz$ is constant with time and can be taken common. Therefore we can write as

$$\frac{Du}{Dt} = X_x - \frac{1}{\rho}\frac{\partial p}{\partial x}$$

$$\frac{Dv}{Dt} = X_y - \frac{1}{\rho}\frac{\partial p}{\partial y}$$

$$\frac{Dw}{Dt} = X_z - \frac{1}{\rho}\frac{\partial p}{\partial z}$$

Expanding the material accelerations in Eqs in terms of their respective temporal and convective components, we get

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = X_x - \frac{1}{\rho}\frac{\partial p}{\partial x}$$

$$\frac{\partial v}{\partial t} - u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = X_y - \frac{1}{\rho}\frac{\partial p}{\partial y}$$

$$\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = X_z - \frac{1}{\rho}\frac{\partial p}{\partial z}$$

$$\frac{D\vec{V}}{Dt} = -\frac{\nabla p}{\rho} + \vec{X}$$

$$\frac{\partial \vec{V}}{\partial t} + (\vec{V}.\nabla)\vec{V} = \vec{X} - \frac{1}{\rho}\nabla p$$

Derivation of Bernoulli's Equation for Inviscid and Viscous Flow Field

Bernoulli's Equation

Energy Equation of an ideal Flow along a Streamline

Euler's equation (the equation of motion of an inviscid fluid) along a stream line for a steady flow with gravity as the only body force can be written as

$$V\frac{dV}{ds}=-\frac{1}{\rho}\frac{dp}{ds}-g\frac{dz}{ds}$$

Application of a force through a distance ds along the streamline would physically imply work interaction. Therefore an equation for conservation of energy along a streamline can be obtained by integrating the above Eq. with respect to ds as

$$\int V\frac{dV}{ds}ds=-\int\frac{1}{\rho}\frac{dp}{ds}ds-\int g\frac{dz}{ds}ds$$

$$\text{or,}\quad \frac{V^2}{2}+\int\frac{dp}{\rho}+gz=C$$

Where C is a constant along a streamline. In case of an incompressible flow, above Eq. can be written as

$$\frac{p}{\rho}+\frac{V^2}{2}+gz=C$$

Pressure head + Velocity head + Potential head =Total head (total energy per unit weight).

Bernoulli's Equation with Head Loss

The derivation of mechanical energy equation for a real fluid depends much on the information about the frictional work done by a moving fluid element and is excluded from the scope of the book. However, in many practical situations, problems related to real fluids can be analysed with the help of a modified form of Bernoulli's equation as

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2+h_f$$

where, hf represents the frictional work done (the work done against the fluid friction) per unit weight of a fluid element while moving from a station 1 to 2 along a streamline in the direction of flow. The term hf is usually referred to as head loss between 1 and 2, since it amounts to the loss in total mechanical energy per unit weight between points 1 and 2 on a streamline due to the effect of fluid friction or viscosity. It physically signifies that the difference in the total mechanical energy between stations 1 and 2 is dissipated into intermolecular or thermal energy and is expressed as loss of head hf in above Eq. The term head loss, is conventionally symbolized as hL instead of hf in dealing with practical problems. For an inviscid flow hL = 0, and the total mechanical energy is constant along a streamline.

Bernoulli's Equation In Irrotational Flow

$$\frac{p}{\rho}+\frac{V^2}{2}+gz=C$$

- This equation was obtained by integrating the Euler’s equation (the equation of motion) with respect to a displacement '**ds'** along a streamline. Thus, the value of C in the above equation is constant only along a streamline and should essentially vary from streamline to streamline.
- The equation can be used to define relation between flow variables at point B on the streamline and at point A, along the same streamline. So, in order to apply this equation, one should have knowledge of velocity field beforehand. This is one of the limitations of application of Bernoulli's equation.

Irrotationality of flow field

Under some special condition, the constant C becomes invariant from streamline to streamline and the Bernoulli’s equation is applicable with same value of C to the entire flow field. The typical condition is the irrotationality of flow field.

Momentum Equation in Integral Form:

Conservation of Momentum: Momentum Theorem

In Newtonian mechanics, the conservation of momentum is defined by Newton’s second law of motion.

Newton’s Second Law of Motion

- The rate of change of momentum of a body is proportional to the impressed action and takes place in the direction of the impressed action.
- If a force acts on the body ,linear momentum is implied.
- If a torque (moment) acts on the body,angular momentum is implied.

Reynolds Transport Theorem

A study of fluid flow by the Eulerian approach requires a mathematical modeling for a control volume either in differential or in integral form. Therefore the physical statements of the principle of conservation of mass, momentum and energy with reference to a control volume become necessary. This is done by invoking a theorem known as the Reynolds transport theorem which relates the control volume concept with that of a control mass system in terms of a general property of the system.

Statement of Reynolds Transport Theorem

The theorem states that "the time rate of increase of property N within a control mass system is equal to the time rate of increase of property N within the control volume plus the net rate of efflux of the property N across the control surface”.

Equation of Reynolds Transport Theorem

After deriving Reynolds Transport Theorem according to the above statement we get

$$\left(\frac{dN}{dt}\right)_{CMS} = \frac{\partial}{\partial t}\iiint_{CV} \eta\rho\, dV + \iint_{CS} \eta\rho\vec{V}.d\vec{A}$$

In this equation
N - flow property which is transported
η - intensive value of the flow property
Application of the Reynolds Transport Theorem to Conservation of Mass and Momentum

Angular Momentum Equation in Integral Form:

Angular Momentum
The angular momentum or moment of momentum theorem is also derived from below Eq in consideration of the property N as the angular momentum and accordingly η as the angular momentum per unit mass. Thus,

$$\frac{d}{dt}(A_{CMS}) = \frac{\partial}{\partial t}\iiint_{CV} \rho(\vec{r}\times\vec{V}_r)dV + \iint_{CS}(\vec{r}\times\vec{V}_r)\rho\vec{V}_r.d\vec{A})$$

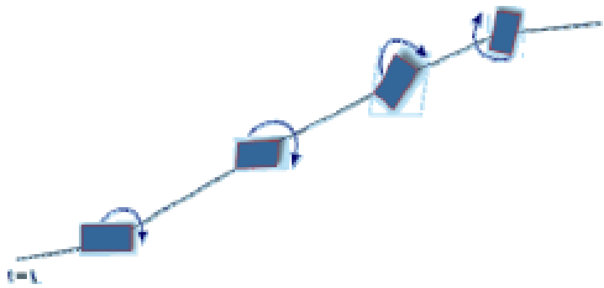
where
Control mass system is the **angular momentum of the control mass system**. . It has to be noted that the origin for the angular momentum is the origin of the position vector

Laplace Equation:

Potential Flow Theory
Let us imagine a pathline of a fluid particle. Rate of spin of the particle is **ω**z . The flow in which this spin is zero throughout is known as irrotationalflow.

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

For irrotational flows,



Pathline of a Fluid Particle

Velocity Potential and Stream Function

Since for irrotational flows . $\nabla \times \vec{V} = 0$.
the velocity for an irrotational flow, can be expressed as the gradient of a scalar function called

$$\vec{V} = \nabla \phi$$

the velocity potential, denoted by Φ
Combination of above eq'ns yields

$$\nabla^2 \phi = 0$$

Laplace equation

For irrotational flows

$$\vec{\omega} = 0$$

For two-dimensional case

$$\omega = \frac{1}{2}\left[\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right] = 0$$

$$\Rightarrow \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0$$

$$-\left(\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}\right) = 0 \qquad \left[u = \frac{\partial \psi}{\partial y}; v = -\frac{\partial \psi}{\partial x}\right]$$

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

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

$$\nabla^2 \psi = 0$$

which is again Laplace's equation.

From the above Eq. we see that **an inviscid, incompressible, irrotational flow is governed by Laplace's equation.**

A complicated flow pattern for an inviscid, incompressible, irrotational flow can be synthesized by adding together a number of elementary flows (provided they are also inviscid, incompressible and irrotational)----- **The Superposition Principle**

Stream Function

Let us consider a two-dimensional incompressible flow parallel to the x - y plane in a rectangular cartesian coordinate system. The flow field in this case is defined by

$$u = u(x, y, t)$$
$$v = v(x, y, t)$$
$$w = 0$$

The equation of continuity is

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

If a function $\psi(x, y, t)$ is defined in the manner

$$u = \frac{\partial \psi}{\partial y}$$
$$v = -\frac{\partial \psi}{\partial x}$$

so that it automatically satisfies the equation of continuity , then the function is known as stream function.
Note that for a **steady flow, ψ is a function of two variables x and y only.**

Constancy of ψ on a Streamline

Since ψ is a point function, it has a value at every point in the flow field. Thus a change in the stream function ψ can be written as

$$d\psi = \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy = -v\,dx + u\,dy$$

The equation of a streamline is given by

$$\frac{u}{dx} = \frac{v}{dy} \qquad \text{or} \qquad u\,dy - v\,dx = 0 \text{(since tangent } dy/dx \text{ equals the velocity } v/u\text{)}$$

It follows that dψ = 0 on a streamline.This implies the value of ψ is constant along a streamline.

Therefore, the equation of a streamline can be expressed in terms of stream function as

$\psi(x, y)$ = constant

Once the function ψ is known, streamline can be drawn by joining the same values of ψ in the flow field.

Stream function for an irrotational flow

In case of a two-dimensional irrotational flow

$$\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \qquad \Rightarrow \frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial x}\right) - \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial y}\right) = 0$$

$$\Rightarrow -\frac{\partial^2 \psi}{\partial x^2} - \frac{\partial^2 \psi}{\partial y^2} = 0 \qquad \Rightarrow \frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

$$\Rightarrow \psi_{xx} + \psi_{yy} = 0$$

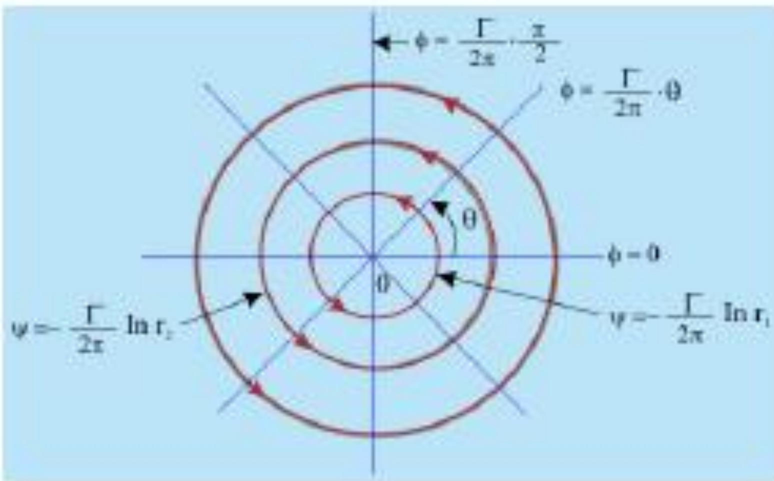
$$\Rightarrow \nabla^2 \psi = 0$$

Conclusion drawn: For an irrotational flow, stream function satisfies the Laplace's equation

Concept of Circulation in a Free Vortex Flow

Free Vortex Flow

- Fluid particles move in circles about a point.
- The only non-trivial velocity component is tangential.
- This tangential speed varies with radius r so that same circulation is maintained.
- Thus,all the **streamlines are concentric circles** about a given point where the velocity along each streamline is inversely proportional to the distance from the centre. This **flow is necessarily irrotational.**



Flownet for a vortex (free vortex)

Lift and Drag for Flow Past a Cylinder without Circulation

Pressure in the Cylinder Surface

Pressure becomes uniform at large distances from the cylinder (where the influence of doublet is small).

Let us imagine the pressure p0 is known as well as uniform velocity U_0 .

We can apply Bernoulli's equation between infinity and the points on the boundary of the cylinder.

Neglecting the variation of potential energy between the aforesaid point at infinity and any point on the surface of the cylinder, we can write

$$\frac{p_0}{\rho g} + \frac{U_0^2}{2g} = \frac{p_b}{\rho g} + \frac{U_b^2}{2g}$$

where the subscript b represents the surface on the cylinder.

Since fluid cannot penetrate the solid boundary, the velocity **Ub should be only in the transverse direction** , or in other words, only vθ component of velocity is present on the streamline ψ = 0 .

Thus at $r = \left(\frac{\Lambda}{U_0}\right)^{1/2}$

$$U_b = v_\theta \Big|_{r=\left(\frac{\Lambda}{U_0}\right)^{1/2}} = -\frac{1}{2}\frac{\partial \psi}{\partial r}\Big|_{r=\left(\frac{\Lambda}{U_0}\right)^{1/2}} = -2U_0 \sin\theta$$

$$p_b = \rho g \left[\frac{U_0^2}{2g} + \frac{p_0}{\rho g} - \frac{(2U_0 \sin\theta)^2}{2g}\right]$$

UNIT - IV

Boundary Layer theory

Navier Stokes Equation in Vector Form:

A general way of deriving the Navier-Stokes equations from the basic laws of physics.

 Consider a general flow field as represented in Fig. 4.1.

 Imagine a closed **control volume,** within the flow field. The control volume is **fixed in space** and the fluid is moving through it. The control volume occupies reasonably large finite region of the flow field.

 A **control surface , A_0** is defined as the surface which **bounds** the volume .

 According to **Reynolds transport theorem, "**The rate of change of momentum for a **system** equals the sum of the rate of change of momentum inside the **control volume** and the rate of **efflux** of momentum across the **control surface"**.

 The rate of change of momentum for a system (in our case, the control volume boundary and the system boundary are same) **is equal to the net external force acting on it.**

Now, we shall transform these statements into equation by accounting for each term,

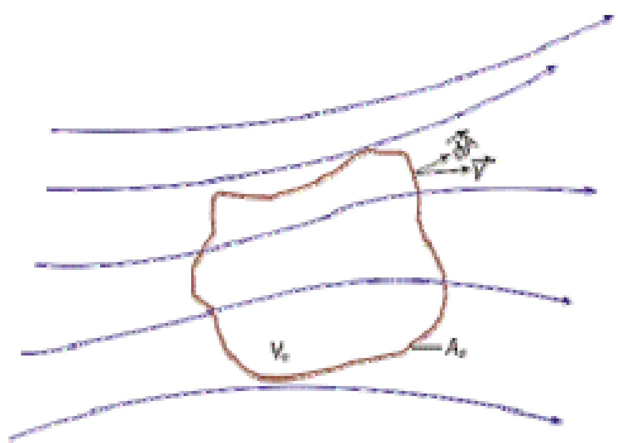


FIG 4.1 Finite control volume fixed in space with the fluid moving through it

- **Rate of change of momentum inside the control volume**

$$= \frac{\partial}{\partial t}\int\int\int_{V_0} \rho\vec{V}\,dV$$

$$= \int\int\int_{V_0} \frac{\partial}{\partial t}(\rho\vec{V})\,dV$$

(since t is independent of space variable)

- **Rate of efflux of momentum through control surface**

$$\int\int_{A_0} \rho\vec{V}(\vec{V}.d\vec{A}) = \int\int_{A_0} \rho\vec{V}\vec{V}.\hat{n}\,dA$$

$$= \int\int\int_{V_0} \left(\vec{V}(\nabla.\rho\vec{V}) + \rho\vec{V}.\nabla\vec{V}\right)dV$$

- **Surface force acting on the control volume**

$$= \int\int_{A_0} d\vec{A}.\sigma$$

(σ is symmetric stress tensor)

$$= \int\int\int_{V_0} (\nabla.\sigma)\,dV$$

- **Body force acting on the control volume**

$$\int\int\int_{V_0} \rho\vec{f}_b\,dV$$

$\vec{f}_b$ in Eq. (25.4) is the body force per unit mass.

- **Finally, we get,**

$$\int\int\int_{V_0} \left(\frac{\partial}{\partial t}(\rho\vec{V}) + \vec{V}(\nabla.\rho\vec{V}) + \rho\vec{V}.\nabla\vec{V}\right)dV$$
$$= \int\int\int_{V_0} \left(\nabla.\sigma + \rho\vec{f}_b\right)dV$$

or

or, $\rho\frac{\partial\vec{V}}{\partial t} + \vec{V}\frac{\partial\rho}{\partial t} + \rho\vec{V}.\nabla\vec{V} + \vec{V}(\nabla.\rho\vec{V}) = \nabla.\sigma + \rho\vec{f}_b$

or $\rho\left(\frac{\partial\vec{V}}{\partial t} + \vec{V}.\nabla\vec{V}\right) + \vec{V}\left(\frac{\partial\rho}{\partial t} + \nabla.\rho\vec{V}\right) = \nabla.\sigma + \rho\vec{f}_b$

We know that $\frac{\partial \rho}{\partial t} + \nabla\cdot\rho\vec{V} = 0$ is the general form of **mass conservation equation** (popularly known as the **continuity equation**), valid for both **compressible** and **incompressible** flows.

Exact Solutions to Navier Stokes Equations:

Consider a class of flow termed as parallel flow in which **only one velocity term is nontrivial** and all the fluid particles move in one direction only.

- We choose x to be the direction along which all fluid particles travel , i.e. $u \neq 0, v = w = 0$. Invoking this in continuity equation, we get

$$\frac{\partial u}{\partial x} + \overset{0}{\frac{\partial v}{\partial y}} + \overset{0}{\frac{\partial w}{\partial z}} = 0$$

$$\frac{\partial u}{\partial x} = 0 \text{ which means } u = u(y,z,t)$$

- Now. Navier-Stokes equations for incompressible flow become

$$\frac{\partial u}{\partial t} + u\overset{0}{\frac{\partial u}{\partial x}} + v\overset{0}{\frac{\partial u}{\partial y}} + w\overset{0}{\frac{\partial u}{\partial z}} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left[\overset{0}{\frac{\partial^2 u}{\partial x^2}} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

$$\overset{0}{\frac{\partial v}{\partial t}} + u\overset{0}{\frac{\partial v}{\partial x}} + v\overset{0}{\frac{\partial v}{\partial y}} + w\overset{0}{\frac{\partial v}{\partial z}} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left[\overset{0}{\frac{\partial^2 v}{\partial x^2}} + \overset{0}{\frac{\partial^2 v}{\partial y^2}} + \overset{0}{\frac{\partial^2 v}{\partial z^2}}\right]$$

$$\cancelto{0}{\frac{\partial v}{\partial t}} + u\cancelto{0}{\frac{\partial v}{\partial x}} + v\cancelto{0}{\frac{\partial v}{\partial y}} + w\cancelto{0}{\frac{\partial v}{\partial z}} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left[\cancelto{0}{\frac{\partial^2 v}{\partial x^2}} + \cancelto{0}{\frac{\partial^2 v}{\partial y^2}} + \cancelto{0}{\frac{\partial^2 v}{\partial z^2}}\right]$$

So, we obtain

$$\frac{\partial p}{\partial y} = \frac{\partial p}{\partial z} = 0 \text{ which means } p = p(x) \text{ alone}$$

and $$\frac{\partial u}{\partial t} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left[\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

Couette Flow:

Couette flow is the flow between **two parallel plates**. Here, one plate is at **rest** and the other is **moving** with a velocity U . Let us assume the plates are infinitely large in z direction, so the z **dependence is not there**.

The **governing equation** is

$$\frac{dp}{dx} = \mu\frac{d^2u}{dy^2}$$

flow is independent of any variation in z-direction.

The **boundary conditions** are ---(i)**At y = 0, u = 0** (ii)**At y = h, u = U.**

Boundary Layer Concept:

Introduction

 The **boundary layer** of a flowing fluid is **the thin layer close to the wall**
 In a flow field, **viscous stresses are very prominent within this layer**.
 Although the layer is thin, it is very important to know the details of flow within it.
 The **main-flow velocity** within this layer **tends to zero** while approaching the wall **(noslip condition)**.
 Also the gradient of this velocity component in a direction normal to the surface is large as compared to the gradient in the streamwise direction.

Boundary Layer Properties:

Boundary Layer Equations

 In 1904, **Ludwig Prandtl,** the well known German scientist, introduced the concept of boundary layer and **derived the equations for boundary layer flow** by correct reduction of Navier-Stokes equations.

He hypothesized that **for fluids having relatively small viscosity**, **the effect of internal friction in the fluid is significant only in a narrow region surrounding solid boundaries or bodies over which the fluid flows.**
 Thus, close to the body is the boundary layer where **shear stresses exert an increasingly larger effect** on the fluid **as one moves from free stream towards the solid boundary.**

 However, **outside the boundary layer where the effect of the shear stresses on the flow is small compared to values inside the boundary layer (since the velocity gradient is negligible)**,---------

1. the fluid particles experience **no vorticity** and therefore**,**
2. the flow is similar to a **potential flow.**

 Hence, the ***surface*** **at the boundary layer interface** is a rather fictitious one, that **divides rotational and irrotational flow.** Fig 28.1 shows Prandtl's model regarding boundary layer flow.
 Hence with the exception of the immediate vicinity of the surface, the flow is frictionless (inviscid) and the velocity is U (the potential velocity).

In the region, very near to the surface (in the thin layer), there is friction in the flow which signifies that the fluid is retarded until it adheres to the surface (**no-slip condition).**
 The transition of the mainstream velocity from zero at the surface (with respect to the surface) to full magnitude takes place across the boundary layer.

About the boundary layer

 Boundary layer **thickness is** which is a **function of** the coordinate direction ***x*** .
 The thickness is considered to be **very small compared to the characteristic length *L*** of the domain.
 In the normal direction, **within this thin layer**, the gradient **is very large compared to** the gradient in the flow direction **.**

Now we take up the Navier-Stokes equations for : steady, two dimensional, laminar, incompressible flows.
Considering the Navier-Stokes equations together with the equation of continuity, the following dimensional form is obtained.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right]$$

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

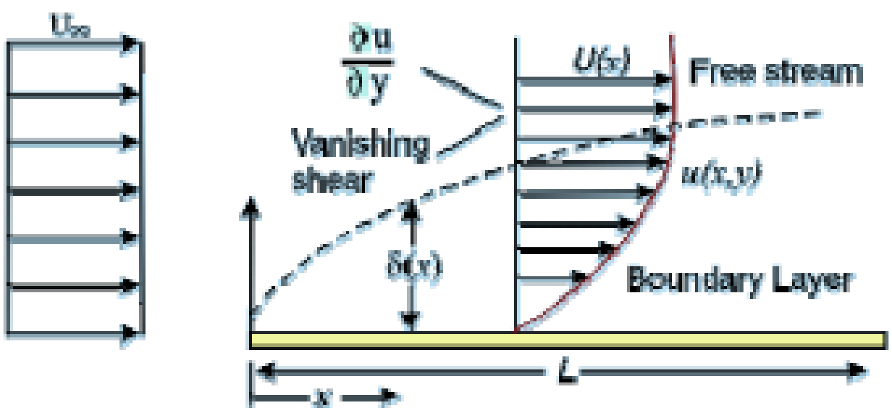


Fig 28.1 Boundary layer and Free Stream for Flow Over a flat plate

- u - velocity component along x direction.
- v - velocity component along y direction
- p - static pressure
- ρ - density.
- μ - dynamic viscosity of the fluid
- The equations are now non-dimensionalised.
- The l**ength and the velocity scales** are chosen as ***L* and respectively.**
- The non-dimensional variables are:

$$u^* = \frac{u}{U_\infty}, v^* = \frac{v}{U_\infty}, p^* = \frac{p}{\rho U_\infty^2}$$

$$x^* = \frac{x}{L}, y^* = \frac{y}{L}$$

Derivation of Prandtl Bounday Layer Equation:

Order of Magnitude Analysis

- Let us examine what happens to the *u* velocity as we go across the boundary layer.

At the **wall** the *u* velocity is **zero** [with respect to the wall and absolute zero for a stationary wall (which is normally implied if not stated otherwise)].

The value of *u* on the **inviscid side**, that is on the free stream side beyond the boundary layer is ***U*.**

For the case of external flow over a flat plate, this ***U*** is equal to .

- Based on the above, we can identify the following scales for the boundary layer variables:

Variable	*Dimensional scale*	*Non-dimensional scale*
u	U_∞	1
x	L	1
y	δ	$\varepsilon = \delta / L$

The **symbol describes a value much smaller than 1.**

Now we analyse equations 28.4 - 28.6, and look at the order of magnitude of each individual term

8.4 Blasius Solution:

Blasius Flow Over A Flat Plate

- The classical problem considered by H. Blasius was
  1. Two-dimensional, steady, incompressible flow over a flat plate at zero angle of incidence with respect to the uniform stream of velocity U_∞.
  2. The fluid extends to infinity in all directions from the plate.

The physical problem is already illustrated in Fig. 28.1

- Blasius wanted to determine
  (a) the velocity field solely within the boundary layer,
  (b) the boundary layer thickness (δ),
  (c) the shear stress distribution on the plate, and
  (d) the drag force on the plate.
- The Prandtl boundary layer equations in the case under consideration are

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\nu\frac{\partial^2 u}{\partial y^2} \qquad (28.15)$$

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

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

The boundary conditions are

$$at\ y=0,\ u=v=0 \qquad (28.16)$$

$$at\quad y=\infty,\quad u=U_\infty$$

- Note that the substitution of the term $-\frac{1}{\rho}\frac{dp}{dx}$ in the original boundary layer momentum equation in terms of the free stream velocity produces $U_\infty\frac{dU_\infty}{dx}$ which is equal to zero.
- Hence the governing Eq. (28.15) does not contain any pressure-gradient term.

- However, the characteristic parameters of this problem are U_∞, ν, x, y that is, $u = u(U_\infty, \nu, x, y)$
- This relation has five variables U_∞, ν, x, y.
- It involves two dimensions, length and time.
- Thus it can be reduced to a dimensionless relation in terms of (5-2) =3 quantities (**Buckingham Pi Theorem**)
- Thus a similarity variables can be used to find the solution
- Such flow fields are called self-similar flow field .

Law of Similarity for Boundary Layer Flows

- It states that the *u* component of velocity with two velocity profiles of *u(x,y)* at different *x* locations differ only by scale factors in *u* and *y* .
- Therefore, the velocity profiles *u(x,y)* at all values of *x* can be made congruent if they are plotted in coordinates which have been made dimensionless with reference to the scale factors.
- The local free stream velocity *U(x)* at section *x* is an obvious scale factor for *u*, because the dimensionless *u(x)* varies between zero and unity with *y* at all sections.
- The scale factor for *y* , denoted by *g(x)* , is proportional to the local boundary layer thickness so that *y* itself varies between zero and unity.
- Velocity at two arbitrary *x* locations, namely x_1 and x_2 should satisfy the equation

$$\frac{u[x_1,(y/g(x_1))]}{U(x_1)} = \frac{u[x_2,(y/g(x_2))]}{U(x_2)} \qquad (28.17)$$

- Now, for Blasius flow, it is possible to identify g(x) with the boundary layers thickness δ we know

$$g = \frac{\delta}{L} \sim \frac{1}{\sqrt{\mathrm{Re}_L}}$$

Thus in terms of x we get

$$\frac{\delta}{x} \sim \frac{1}{\sqrt{\frac{U_\infty x}{\nu}}}$$

$$\delta \sim \sqrt{\frac{\nu x}{U_\infty}}$$

Turbulent Boundary Layer over Flat Plate:

Derivation of Governing Equations for Turbulent Flow

- For **incompressible flows**, the Navier-Stokes equations can be rearranged in the form

$$\rho\left[\frac{\partial u}{\partial t}+\frac{\partial (u^2)}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}\right]=-\frac{\partial p}{\partial x}+\mu\nabla^2 u \qquad (33.1a)$$

$$\rho\left[\frac{\partial v}{\partial t}+\frac{\partial (uv)}{\partial x}+\frac{\partial (v^2)}{\partial y}+\frac{\partial (vw)}{\partial z}\right]=-\frac{\partial p}{\partial y}+\mu\nabla^2 v \qquad (33.1b)$$

$$\rho\left[\frac{\partial w}{\partial t}+\frac{\partial (uw)}{\partial x}+\frac{\partial (vw)}{\partial y}+\frac{\partial (w^2)}{\partial z}\right]=-\frac{\partial p}{\partial z}+\mu\nabla^2 w \qquad (33.1c)$$

and

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad (33.2)$$

- Express the velocity components and pressure in terms of time-mean values and corresponding fluctuations. In **continuity equation**, this substitution and subsequent time averaging will lead to

$$\overline{\frac{\partial(\bar{u}+u')}{\partial x}}+\overline{\frac{\partial(\bar{v}+v')}{\partial y}}+\overline{\frac{\partial(\bar{w}+w')}{\partial z}}=0$$

or,

$$\left(\frac{\partial \bar{u}}{\partial x}+\frac{\partial \bar{v}}{\partial y}+\frac{\partial \bar{w}}{\partial z}\right)+\left(\frac{\partial \overline{u'}}{\partial x}+\frac{\partial \overline{v'}}{\partial y}+\frac{\partial \overline{w'}}{\partial z}\right)=0$$

Since, $\frac{\partial \overline{u'}}{\partial x} = \frac{\partial \overline{v'}}{\partial y} = \frac{\partial \overline{w'}}{\partial z} = 0$

We can write $\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0$ (33.3a)

From Eqs (33.3a) and (33.2), we obtain

$$\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} = 0 \quad (33.3b)$$

- It is evident that **the time-averaged velocity components** and **the fluctuating velocity components**, each satisfy the continuity equation for incompressible flow.
- Imagine a two-dimensional flow in which the turbulent components are independent of the z -direction. Eventually, Eq.(33.3b) tends to

$$\frac{\partial u'}{\partial x} = -\frac{\partial v'}{\partial y} \quad (33.4)$$

On the basis of condition (33.4), it is postulated that if at an instant there is an increase in u' in the x -direction, it will be followed by an increase in v' in the negative y -direction. In other words, $\overline{u'v'}$ **is non-zero and negative**. (see Figure 33.2)

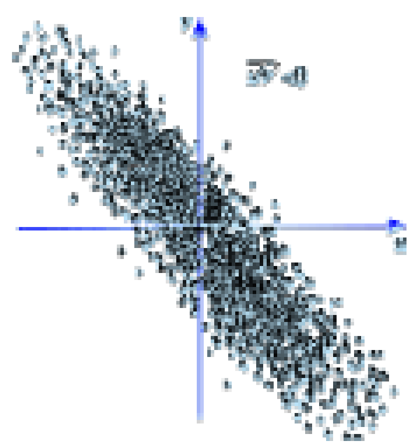


Fig 33.2 Each dot represents uv pair at an instant

c), we obtain expressions in terms of mean and fluctuating components. Now, forming time averages and considering the rules of averaging we discern the following. The terms which are linear, such as $\frac{\partial u'}{\partial t}$ and $\frac{\partial^2 u'}{\partial x^2}$ vanish when they are averaged [from (32.6)]. The same is true for the mixed terms like $\bar{u}\cdot u'$, or $\bar{u}\cdot v'$, but the quadratic terms in the fluctuating components remain in the equations. After averaging, they form $\overline{u'^2}$, $\overline{u'v'}$ etc.

- If we perform the aforesaid exercise on the x-momentum equation, we obtain

$$\rho\left\{\frac{\partial \bar{u}}{\partial t}+\cancelto{0}{\frac{\partial \overline{u'}}{\partial t}}+\frac{\partial\left(\bar{u}^2+\overline{u'^2}\right)}{\partial x}+\frac{\partial\left(\overline{u\cdot v}+\overline{u'v'}\right)}{\partial y}+\frac{\partial\left(\bar{u}\cdot\bar{w}+\overline{u'w'}\right)}{\partial z}\right\}$$

$$=-\frac{\partial \bar{p}}{\partial x}-\cancelto{0}{\frac{\partial \overline{p'}}{\partial x}}+\mu\left[\frac{\partial^2 \bar{u}}{\partial x^2}+\frac{\partial^2 \bar{u}}{\partial y^2}+\frac{\partial^2 \bar{u}}{\partial z^2}+\left(\cancelto{0}{\frac{\partial^2 \overline{u'}}{\partial x^2}}+\cancelto{0}{\frac{\partial^2 \overline{u'}}{\partial y^2}}+\cancelto{0}{\frac{\partial^2 \overline{u'}}{\partial z^2}}\right)\right]$$

using rules of time averages,

$$\frac{\partial \overline{u'}}{\partial t}=0,\ \frac{\partial \overline{p'}}{\partial x}=0,\ \frac{\partial^2 \overline{u'}}{\partial x^2}=\frac{\partial^2 \overline{u'}}{\partial y^2}=\frac{\partial^2 \overline{u'}}{\partial z^2}=0$$

We obtain

$$\rho\left\{\frac{\partial \bar{u}}{\partial t}+\frac{\partial\left(\bar{u}^2\right)}{\partial x}+\frac{\partial\left(\bar{u}\cdot\bar{v}\right)}{\partial y}+\frac{\partial\left(\bar{u}\cdot\bar{w}\right)}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial x}+\mu\nabla^2\bar{u}-\rho\left[\frac{\partial}{\partial x}\overline{u'^2}+\frac{\partial}{\partial y}\overline{u'v'}+\frac{\partial}{\partial z}\overline{u'w'}\right]$$

- Introducing simplifications arising out of continuity Eq. (33.3a), we shall obtain.

$$\rho\left\{\frac{\partial \bar{u}}{\partial t}+\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}+\bar{w}\frac{\partial \bar{u}}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial x}+\mu\nabla^2\bar{u}-\rho\left[\frac{\partial}{\partial x}\overline{u'^2}+\frac{\partial}{\partial y}\overline{u'v'}+\frac{\partial}{\partial z}\overline{u'w'}\right]$$

- Performing a similar treatment on y and z momentum equations, finally we obtain the momentum equations in the form.

In x direction,

$$\rho\left\{\frac{\partial \bar{u}}{\partial t}+\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}+\bar{w}\frac{\partial \bar{u}}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial x}+\mu\nabla^2\bar{u}-\rho\left[\frac{\partial}{\partial x}\overline{u'^2}+\frac{\partial}{\partial y}\overline{u'v'}+\frac{\partial}{\partial z}\overline{u'w'}\right] \quad (33.5a)$$

In y direction,

$$\rho\left\{\frac{\partial \bar{v}}{\partial t}+\bar{u}\frac{\partial \bar{v}}{\partial x}+\bar{v}\frac{\partial \bar{v}}{\partial y}+\bar{w}\frac{\partial \bar{v}}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial y}+\mu\nabla^2\bar{v}-\rho\left[\frac{\partial}{\partial x}\overline{u'v'}+\frac{\partial}{\partial y}\overline{v'^2}+\frac{\partial}{\partial z}\overline{v'w'}\right] \quad (33.5b)$$

In z direction,

$$\rho\left\{\frac{\partial \bar{w}}{\partial t}+\bar{u}\frac{\partial \bar{w}}{\partial x}-\bar{v}\frac{\partial \bar{w}}{\partial y}+\bar{w}\frac{\partial \bar{w}}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial z}+\mu\nabla^2\bar{w}-\rho\left[\frac{\partial}{\partial x}\overline{u'w'}+\frac{\partial}{\partial y}\overline{v'w'}+\frac{\partial}{\partial z}\overline{w'^2}\right] \quad (33.5c)$$

- Comments on the governing equation :
  1. The left hand side of Eqs (33.5a)-(33.5c) are essentially similar to the steady-state Navier-Stokes equations if the velocity components u, v and w are replaced by $\bar{u}$, $\bar{v}$ and $\bar{w}$.
  2. The same argument holds good for the first two terms on the right hand side of Eqs (33.5a)-(33.5c).
  3. However, the equations contain some additional terms which depend on turbulent fluctuations of the stream. **These additional terms can be interpreted as components of a stress tensor.**
- Now, the resultant surface force per unit area due to these terms may be considered as

In x direction,

$$\rho\left\{\frac{\partial \bar{u}}{\partial t}+\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}+\bar{w}\frac{\partial \bar{u}}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial x}+\mu\nabla^2\bar{u}+\left[\frac{\partial}{\partial x}\sigma'_{xx}+\frac{\partial}{\partial y}\tau'_{yx}+\frac{\partial}{\partial z}\tau'_{zx}\right] \quad (33.6a)$$

In y direction,

$$\rho\left\{\frac{\partial \bar{v}}{\partial t}+\bar{u}\frac{\partial \bar{v}}{\partial x}+\bar{v}\frac{\partial \bar{v}}{\partial y}+\bar{w}\frac{\partial \bar{v}}{\partial z}\right\}=-\frac{\partial \bar{p}}{\partial y}+\mu\nabla^2\bar{v}+\left[\frac{\partial}{\partial x}\tau'_{xy}+\frac{\partial}{\partial y}\sigma'_{yy}+\frac{\partial}{\partial z}\tau'_{zy}\right] \quad (33.6b)$$

In z direction,

$$\rho\left(\frac{\partial \bar{w}}{\partial t}+\bar{u}\frac{\partial \bar{w}}{\partial x}+\bar{v}\frac{\partial \bar{w}}{\partial y}+\bar{w}\frac{\partial \bar{w}}{\partial z}\right)=-\frac{\partial \bar{p}}{\partial z}+\mu\nabla^2\bar{w}+\left[\frac{\partial}{\partial x}\tau'_{xz}+\frac{\partial}{\partial y}\tau'_{yz}+\frac{\partial}{\partial z}\sigma'_{zz}\right] \quad (33.6c)$$

- Comparing Eqs (33.5) and (33.6), we can write

$$\begin{bmatrix}\sigma'_{xx} & \tau'_{xy} & \tau'_{xz}\\ \tau'_{xy} & \sigma'_{yy} & \tau'_{yz}\\ \tau'_{xz} & \tau'_{yz} & \sigma'_{zz}\end{bmatrix}=-\rho\begin{bmatrix}\overline{u'^2} & \overline{u'v'} & \overline{u'w'}\\ \overline{u'v'} & \overline{v'^2} & \overline{v'w'}\\ \overline{u'w'} & \overline{v'w'} & \overline{w'^2}\end{bmatrix} \quad (33.7)$$

- It can be said that the mean velocity components of turbulent flow satisfy the same Navier-Stokes equations of laminar flow. However, for the turbulent flow, the laminar stresses must be increased by additional stresses which are given by the stress tensor (33.7). These additional stresses are known as **apparent stresses of turbulent flow** or **Reynolds stresses** . Since turbulence is considered as eddying motion and the aforesaid additional stresses are added to the viscous stresses due to mean motion in order to explain the complete stress field, it is often said that the apparent stresses are caused by eddy viscosity . The total stresses are now

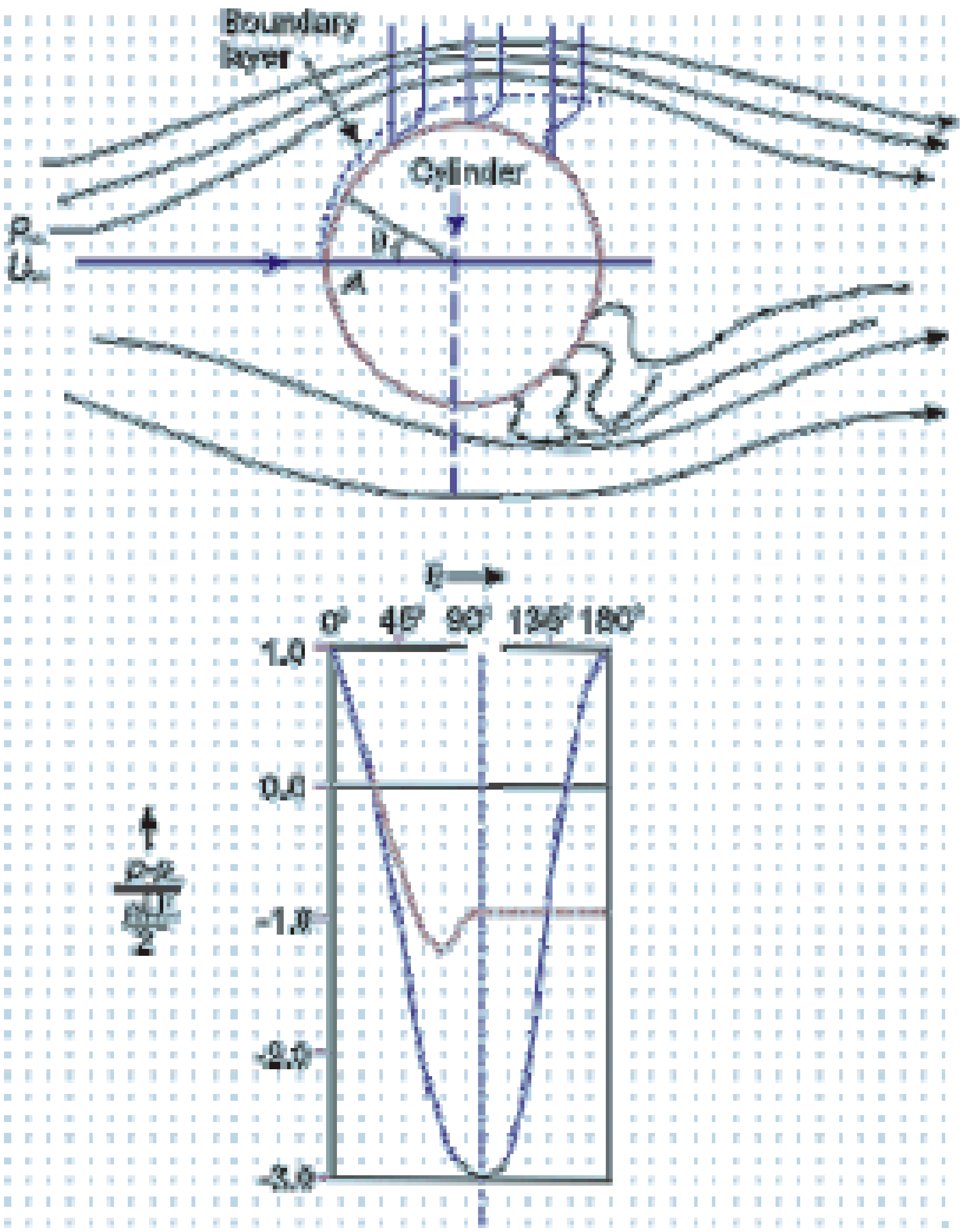
$$\left.\begin{aligned}\sigma_{xx}&=-\bar{p}+2\mu\frac{\partial \bar{u}}{\partial x}-\rho\overline{u'^2}\\ \tau_{xy}&=\mu\left(\frac{\partial \bar{u}}{\partial y}+\frac{\partial \bar{v}}{\partial x}\right)-\rho\overline{u'v'}\end{aligned}\right] \quad (33.8)$$

and so on. The apparent stresses are much larger than the viscous components, and the viscous stresses can even be dropped in many actual calculations.

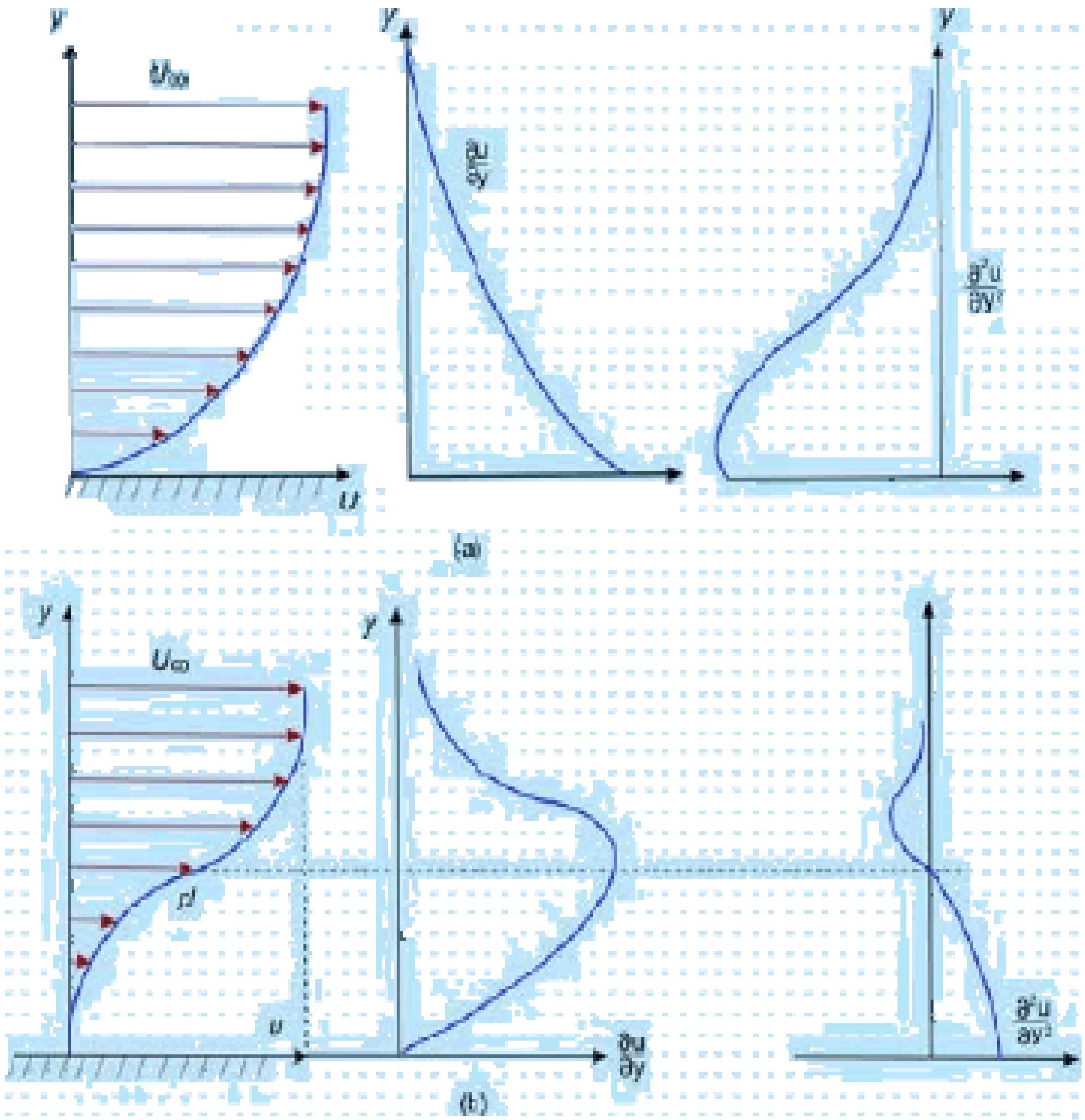
Boundary Layer Control:

Seperation of Boundary Layer

- It has been observed that the **flow is reversed at the vicinity of the wall** under certain conditions.
- The phenomenon is termed as **separation of boundary layer**.
- Separation takes place **due to excessive momentum loss near the wall in a boundary layer trying to move downstream against increasing pressure, i.e., $\frac{dp}{dx} > 0$, which is called *adverse pressure gradient.***
- Figure 29.2 shows the flow past a circular cylinder, in an infinite medium.
    1. Up to $\theta = 90^\circ$, the flow area is like a constricted passage and the flow behaviour is like that of a nozzle.
    2. Beyond $\theta = 90^\circ$ the flow area is diverged, therefore, the flow behaviour is much similar to a diffuser



Flow separation and formation of wake behind a circular cylinder



Velocity distribution within a boundary layer

(a) Favourable pressure gradient, $\frac{dp}{dx} < 0$

(b) adverse pressure gradient $\frac{dp}{dx} > 0$

Let us reconsider the flow past a circular cylinder and continue our **discussion on the wake behind a cylinder.** The pressure distribution which was shown by the firm line in Fig. 21.5 is obtained from the potential flow theory. However. somewhere near **(in experiments it has been observed to be at) . the boundary layer detaches itself from the wall.**

2. Meanwhile, **pressure in the wake remains close to separation-point-pressure** since the eddies (formed as a consequence of the retarded layers being carried together with the upper layer through the action of shear) cannot convert rotational kinetic energy into pressure head. The actual pressure distribution is shown by the dotted line in Fig. 29.3.

3. Since the **wake zone pressure is less than that of the forward stagnation point** (pressure at point A in Fig. 29.3), the cylinder experiences a drag force which is basically attributed to the pressure difference.

The drag force, brought about by the pressure difference is known as *form drag* whereas the shear stress at the wall gives rise to *skin friction drag*.
Generally, these two drag forces together are responsible for resultant drag on a body

Control Of Boundary Layer Separation –

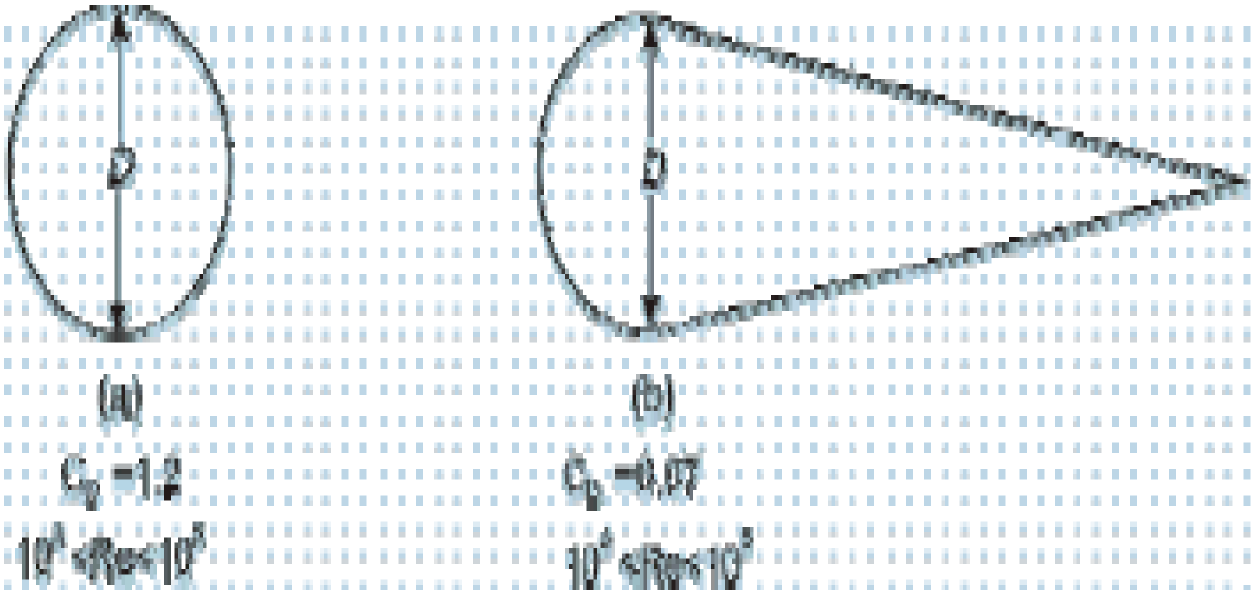
 The total drag on a body is attributed to form drag and skin friction drag. In some flow configurations, the contribution of form drag becomes significant.

 In order **to reduce the form drag, the boundary layer separation** should be **prevented or delayed** so that **better pressure recovery takes place** and the form drag is reduced considerably. There are some popular methods for this purpose which are stated as follows.

i. By giving the profile of the body a streamlined shape

1. This has an elongated shape in the rear part to reduce the magnitude of the pressure gradient.

2. The optimum contour for a streamlined body is the one for which the wake zone is very narrow and the form drag is minimum.



Reduction of drag coefficient (CD) by giving the profile a streamlined shape

The injection of fluid through porous wall can also control the boundary

layer separation. This is generally accomplished by blowing high energy fluid particles tangentially from the location where separation would have taken place otherwise.

1. The **injection of fluid promotes turbulence**

2. This **increases skin friction.** But the **form drag is reduced** considerably due to suppression of flow separation

3. The reduction in form drag is quite significant and **increase in skin friction drag can be ignored.**

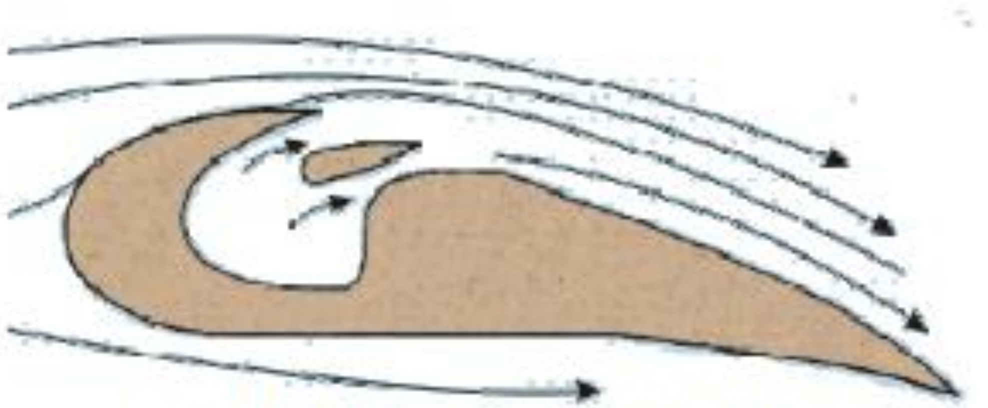
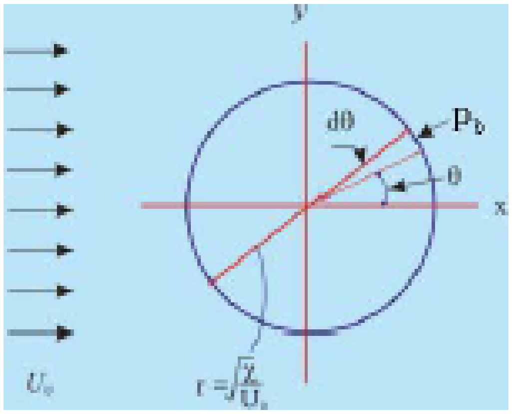


Fig. 31.3 Boundary layer control by blowing

Lift and Drag

Lift :force acting on the cylinder (per unit length) in the direction normal to uniform flow.

Drag: force acting on the cylinder (per unit length) in the direction parallel to uniform flow.



Calculation of Drag in a Cylinder

The drag is calculated by integrating the force components arising out of pressure, in the x direction on the boundary. Referring to Fig.22.4, the drag force can be written as

$$D = -\int_0^{2\pi} p_s \cos\theta r d\theta \qquad r d\theta = ds \rightarrow$$ infinitesimal length on the circumference

Since, $r = \left(\frac{\Gamma}{U_0}\right)^{1/2}$

$$D = -\int_0^{2\pi} p_s \cos\theta \left(\frac{\Gamma}{U_0}\right)^{1/2} d\theta$$

or,

$$D = -\int_0^{2\pi} \rho g \left(\frac{\Gamma}{U_0}\right)^{1/2} \left[\frac{U_0^2}{2g} + \frac{p_0}{\rho g} - \frac{(2U_0 \sin\theta)^2}{2g}\right] \cos\theta d\theta$$

$$D = -\int_0^{2\pi} \left[p_0 + \frac{\rho U_0^2}{2}\left(1 - 4U_0 \sin^2\theta\right)\right]\left(\frac{\Gamma}{U_0}\right)^{1/2} \cos\theta d\theta$$

Similarly, the lift force may be calculated as

$$L = -\int_0^{2\pi} p_s \sin\theta \left(\frac{\Gamma}{U_0}\right)^{1/2} d\theta$$

However, in reality, the cylinder will always experience some drag force. This contradiction between the inviscid flow result and the experiment is usually known as D 'Almbert paradox.

UNIT V

Closed Conduit Flow

- Energy equation
- EGL and HGL
- Head loss
  - – major losses
  - – minor losses
- Non circular conduits

Conservation of Energy

- Kinetic, potential, and thermal energy

h_p = head supplied by a pump

h_t = head given to a turbine

h_L = head loss between sections 1 and 2

Cross section 2 is downstream from cross section 1!

$$\frac{p_1}{\gamma} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_t + h_L$$

Energy Equation Assumptions

- Pressure is __hydrostatic__ in both cross sections
  - pressure changes are due to elevation only $p = \gamma h$
- section is drawn perpendicular to the streamlines (otherwise the __kinetic__ energy term is incorrect)
- Constant __density__ at the cross section
- __Steady__ flow

$$\frac{p_1}{\gamma} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_t + h_L$$

Bernoulli Equation Assumption

- __Frictionless__ (viscosity can't be a significant parameter!)
- Along a __streamline__
- __Steady__ flow
- Constant __density__

$$z + \frac{V^2}{2g} + \frac{p}{\gamma} = const$$

287

Pipe Flow: Review

- We have the control volume energy equation for pipe flow.
- We need to be able to predict the head loss term.
- How do we predict head loss? Dimensional analysis.

$$\frac{p_1}{\gamma} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_p = \frac{p_2}{\gamma} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_t + h_L$$

Pipe Flow Energy Losses

$$\frac{p_1}{\gamma} + \alpha_1 \cancel{\frac{V_1^2}{2g}} + \cancel{z_1} + \cancel{h_p} = \frac{p_2}{\gamma} + \alpha_2 \cancel{\frac{V_2^2}{2g}} + \cancel{z_2} + \cancel{h_t} + h_l$$

$$h_f = -\frac{\Delta p}{\gamma}$$ Horizontal pipe

$$f = \left(C_p \frac{D}{L}\right) = \text{function of} \left(\frac{e}{D}, \text{Re}\right)$$ Dimensional Analysis

$$C_p = \frac{-2\Delta p}{\rho V^2} \qquad C_p = \frac{2gh_f}{V^2}$$

$$f = \frac{2gh_f}{V^2}\frac{D}{L} \qquad h_f = f\frac{L}{D}\frac{V^2}{2g}$$ Darcy-Weisbach equation

Friction Factor: Major losses

- Laminar flow
  - Hagen-Poiseuille
- Turbulent (Smooth, Transition, Rough)
  - Colebrook Formula
  - Moody diagram
  - Swamee-Jain

290

Laminar Flow Friction Factor

$$V = \frac{\gamma D^2 h_l}{32\mu L}$$

Hagen-Poiseuille

$$h_f = \frac{32mLV}{r\,gD^2} \qquad h_f = \frac{128mLQ}{pr\,gD^4}$$

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

Darcy-Weisbach

$$\frac{32mLV}{r\,gD^2} = f\frac{L}{D}\frac{V^2}{2g}$$

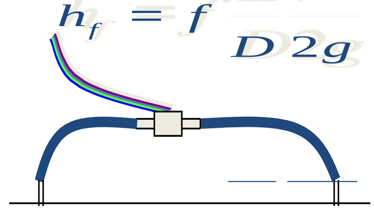
$$f = \frac{64m}{r\,VD} = \frac{64}{\text{Re}}$$

Slope of -1 on log-log plot

Turbulent Pipe Flow Head Loss

- Proportional to the length of the pipe
- Proportional to the square of the velocity (almost)
- Increases with surface roughness
- Is a function of density and viscosity
- Is independent of pressure

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



Smooth, Transition, Rough Turbulent Flow

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

- Hydraulically smooth pipe law (von Karman, 1930)

$$\frac{1}{\sqrt{f}} = 2\log\left(\frac{\mathrm{Re}\sqrt{f}}{2.51}\right)$$

- Rough pipe law (von Karman, 1930)

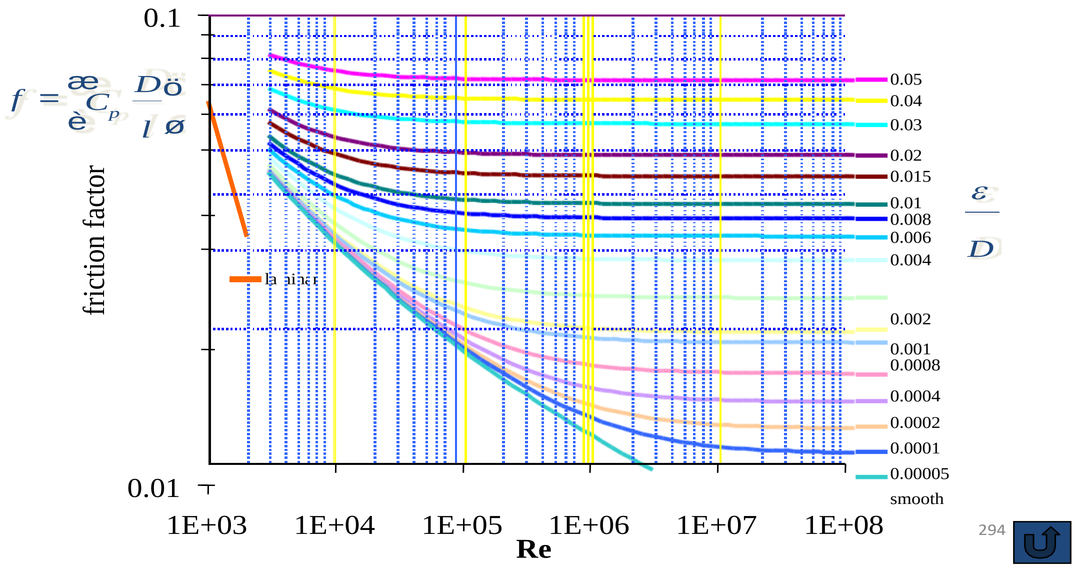
$$\frac{1}{\sqrt{f}} = 2\log\left(\frac{3.7D}{e}\right)$$

- Transition function for both smooth and rough pipe laws (Colebrook)

$$\frac{1}{\sqrt{f}} = -2\log\left(\frac{e/D}{3.7} + \frac{2.51}{\mathrm{Re}\sqrt{f}}\right)$$

(used to draw the Moody diagram)

Moody Diagram



Pipe roughness

pipe material	pipe roughness ε (mm)
glass, drawn brass, copper	0.0015
commercial steel or wrought iron	0.045
asphalted cast iron	0.12
galvanized iron	0.15
cast iron	0.26
concrete	0.18-0.6
rivet steel	0.9-9.0
corrugated metal	45
PVC	0.12

Exponential Friction Formulas

- Commonly used in commercial and industrial settings

$$h_f = \frac{RLQ^n}{D^m}$$

- Only applicable over __range of data__ collected
- Hazen-Williams exponential friction formula

$$R = \begin{cases} \dfrac{4.727}{C^n} & \text{USC units} \\ \dfrac{10.675}{C^n} & \text{SI units} \end{cases}$$

$$h_f = \frac{10.675L}{D^{4.8704}}\left(\frac{Q}{C}\right)^{1.852} \quad \text{SI units}$$

C = Hazen-Williams coefficient

296

Head loss:
Hazen-Williams Coefficient

C	Condition
150	PVC
140	Extremely smooth, straight pipes; asbestos cement
130	Very smooth pipes; concrete; new cast iron
120	Wood stave; new welded steel
110	Vitrified clay; new riveted steel
100	Cast iron after years of use
95	Riveted steel after years of use
60-80	Old pipes in bad condition

297

Hazen-Williams

vs

Darcy-Weisbach

$$h_f = \frac{10.675L}{D^{4.8704}}\left(\frac{Q}{C}\right)^{1.852} \quad \text{SI units}$$

$$h_f = f\frac{8}{p^2 g}\frac{LQ^2}{D^5}$$

- Both equations are empirical
- Darcy-Weisbach is rationally based, dimensionally correct, and preferred .

- Hazen-Williams can be considered valid only over the range of gathered data.
- Hazen-Williams can't be extended to other fluids without further experimentation.

Head Loss: Minor Losses

- Head loss due to outlet, inlet, bends, elbows, valves, pipe size changes
- Losses due to expansions are greater than losses due to contractions
- Losses can be minimized by gradual transitions

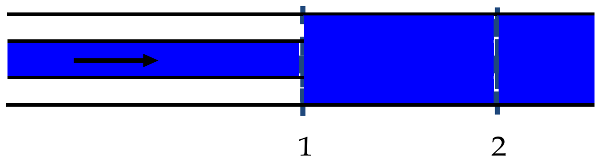
Minor Losses

- Most minor losses can not be obtained analytically, so they must be measured
- Minor losses are often expressed as a loss coefficient, K, times the velocity head.

$$C_p = f(\text{geometry}, \text{Re})$$

$$C_p = \frac{-2\Delta p}{\rho V^2} \qquad C_p = \frac{2gh_l}{V^2} \qquad h_l = C_p \frac{V^2}{2g} \qquad h = K\frac{V^2}{2g}$$

Head Loss due to Sudden Expansion: Conservation of Energy



$$\frac{p_1}{\gamma_1} + z_1 + \alpha_1 \frac{V_1^2}{2g} + H_p = \frac{p_2}{\gamma_2} + z_2 + \alpha_2 \frac{V_2^2}{2g} + H_t + h_l$$

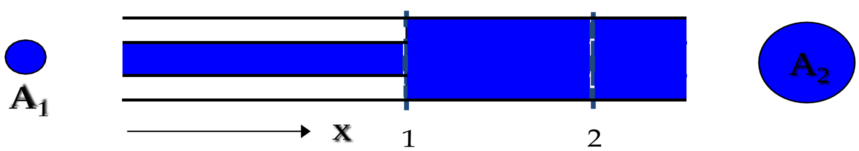
$$\frac{p_1 - p_2}{\gamma} = \frac{V_2^2 - V_1^2}{2g} + h_l$$

$z_1 = z_2$

$$h_l = \frac{p_1 - p_2}{\gamma} + \frac{V_1^2 - V_2^2}{2g}$$

What is $p_1 - p_2$?

Head Loss due to Sudden Expansion: Conservation of Momentum



$$\mathbf{M}_1 + \mathbf{M}_2 = \mathbf{W} + \mathbf{F}_{p_1} + \mathbf{F}_{p_2} + \mathbf{F}_{ss}$$

Apply in direction of flow

$$M_{1x} + M_{2x} = F_{p_{1x}} + F_{p_{2x}}$$

Neglect surface shear

$$M_{1x} = -\rho V_1^2 A_1 \qquad M_{2x} = \rho V_2^2 A_2$$

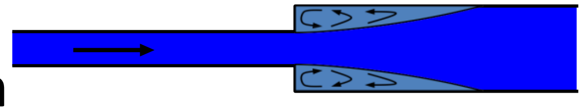
$$-\rho V_1^2 A_1 + \rho V_2^2 A_2 = p_1 A_2 - p_2 A_2$$

$$\frac{p_1 - p_2}{\gamma} = \frac{V_2^2 - V_1^2 \frac{A_2}{A_1}}{g}$$

Pressure is applied over all of section 1.
Momentum is transferred over area corresponding to upstream pipe diameter.
V_1 is velocity upstream.

Divide by $(A_2 \gamma)$

Head Loss due to Sudden Expansion



Energy
$$h_l = \frac{p_1 - p_2}{\gamma} + \frac{V_1^2 - V_2^2}{2g}$$

Mass
$$\frac{A_1}{A_2} = \frac{V_2}{V_1}$$

Momentum
$$\frac{p_1 - p_2}{\gamma} = \frac{V_2^2 - V_1^2 \frac{A_1}{A_2}}{g}$$

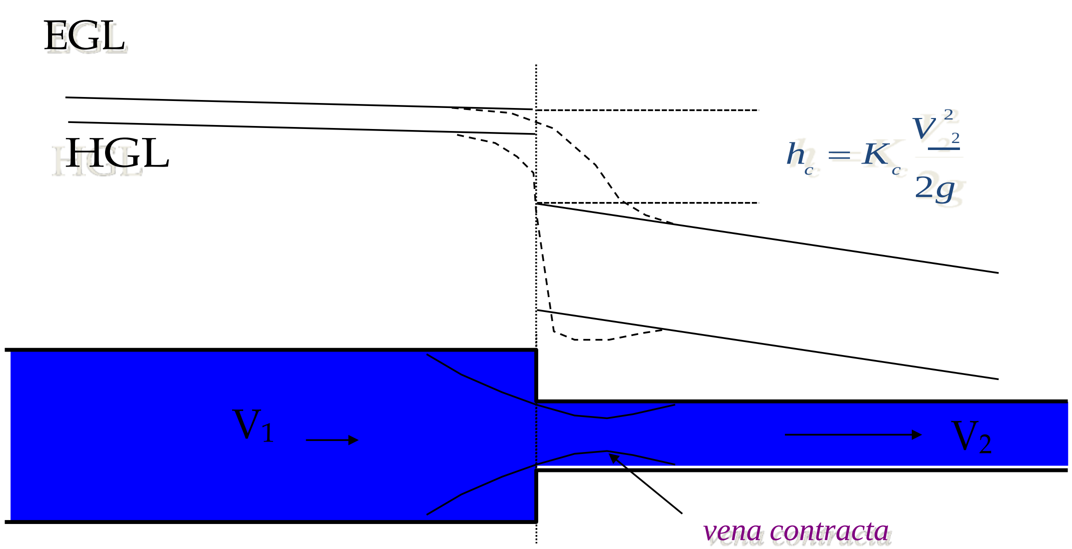
$$h_l = \frac{V_2^2 - V_1^2 \frac{V_2}{V_1}}{g} + \frac{V_1^2 - V_2^2}{2g}$$

$$h_l = \frac{V_2^2 - 2V_1V_2 + V_1^2}{2g}$$

$$h_l = \frac{(V_1 - V_2)^2}{2g} \qquad h_l = \frac{V_1^2}{2g}\left(1 - \frac{A_1}{A_2}\right)^2 \qquad K = \left(1 - \frac{A_1}{A_2}\right)^2$$

303

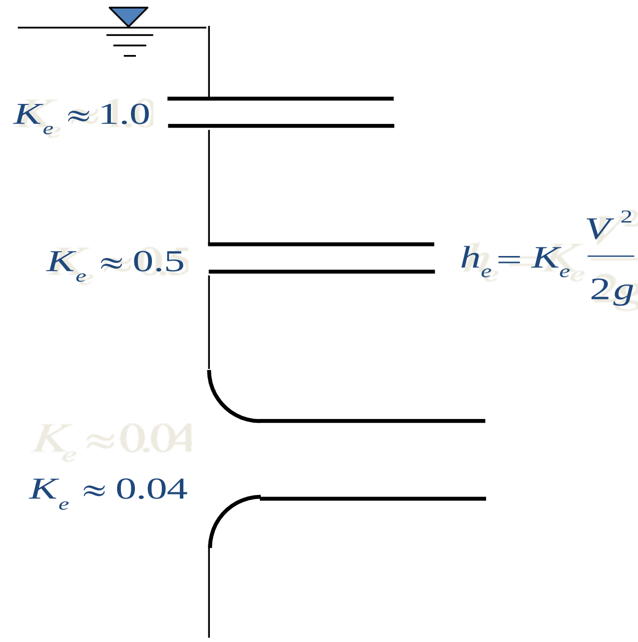
Contraction



losses are reduced with a gradual contraction

Entrance Losses

- Losses can be reduced by accelerating the flow gradually and eliminating the *vena contracta*



Head Loss in Valves

- Function of valve type and valve position
- The complex flow path through valves often results in high head loss
- What is the maximum value that K_v can have?

$$h_v = K_v \frac{V^2}{2g}$$

Non-Circular Conduits: Hydraulic Radius Concept

- A is cross sectional area
- P is wetted perimeter
- R_h is the "Hydraulic Radius" (Area/Perimeter)
- Don't confuse with radius!

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

$$R_h = \frac{A}{P} = \frac{\frac{\pi}{4} D^2}{\pi D} = \frac{D}{4}$$

For a pipe

$$D = 4R_h$$

$$h_f = f \frac{L}{4R_h} \frac{V^2}{2g}$$

We can use Moody diagram or Swamee Jain with D = 4R!

311