Unit - I

Fluid Properties and Fluid Statics

Objective: The student will be able to

- understand the concept of fluid and its properties, manometry, hydrostatic forces acting on different surfaces and also problem solving techniques.

Contents:

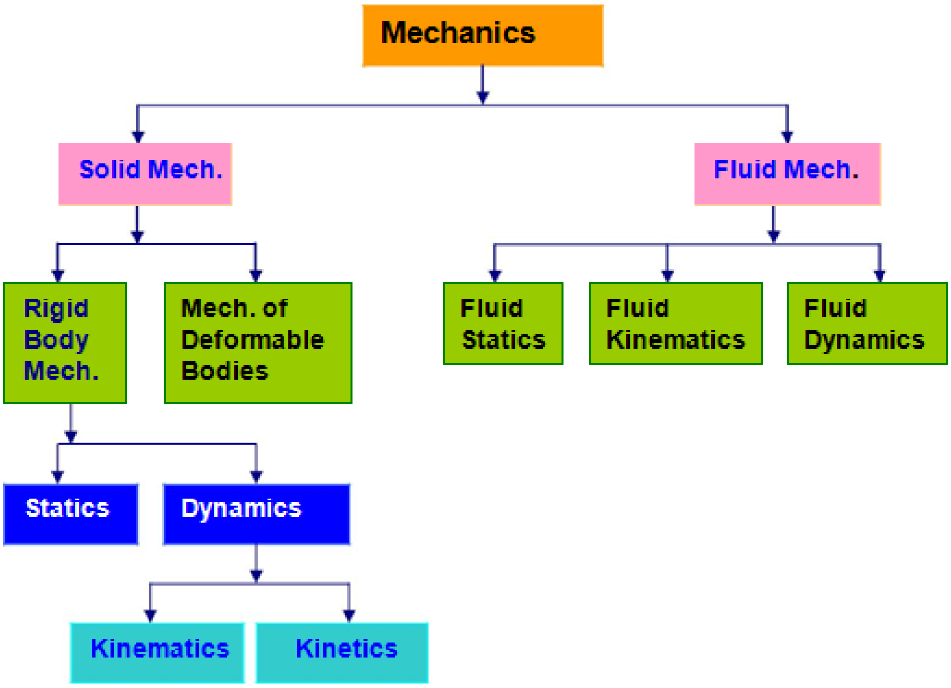
Physical properties of fluids: specific gravity, viscosity and its significance, surface tension, capillarity, vapor pressure. Atmospheric, gauge and vacuum pressure – measurement of pressure. Manometers- Piezometer, U-tube, inverted and differential manometers. Pascal's law, hydrostatic law.

Buoyancy and floatation: Meta center, stability of floating bodies and Submerged bodies. Calculation of metacenter height. Stability analysis and applications.

References:

1. Robert W. Fox, Alan T. McDonald, Philip J. Pritchard, Introduction to FLuid Mechanics, 8th Edition, John Wiley & Sons, Inc.,
2. Frank M. White, Fluid Mechanics, 7th Edition, Tata McGraw Hill.
3. Yunus A. Cengel, John M. Cimbala, Fluid Mechanics
4. S. K. Som, G. Biswas, Introduction to Fluid Mechanics and Fluid Machines, Revised Second Edition, Tata McGraw Hill Book Publishing Company Limited.

Introduction to Fluids Mechanics

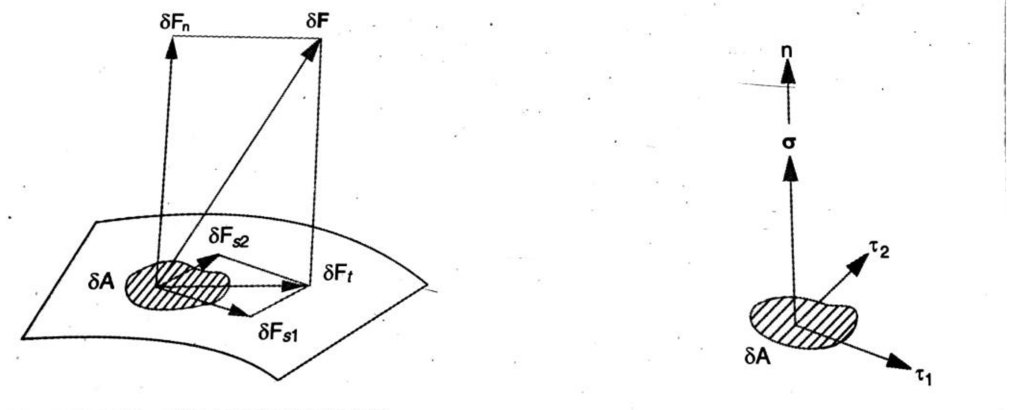


Fluid mechanics is the study of fluids either in motion (fluid *dynamics*) or at rest (fluid *statics*) and the interaction of fluids with solids or other fluids at the boundaries. Both gases and liquids are classified as fluids, and the number of fluid engineering applications is enormous: breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes, missiles, icebergs, engines, filters, jets, and sprinklers, to name a few. The study of fluid mechanics is categorized as:

- *Hydrodynamics*: It deals with the study of the motion of fluids that are practically incompressible (such as liquids, especially water, and gases at low speeds).
- *Hydraulics*: It deals with the liquid flows in pipes and open channels.
- *Gas dynamics*: It deals with the flow of fluids that undergo significant density changes, such as the flow of gases through nozzles at high speeds.
- *Aerodynamics*: It deals with the flow of gases (especially air) over bodies such as aircraft, rockets, and automobiles at high or low speeds.

Definition of stress Let us consider a small element δA on the surface of a body upon which a force is acting as δF

The force can be resolved into two components, δF_n and δF_t.



(a) NORMAL AND SHEAR FORCES (b) NORMAL AND SHEAR STRESSES

δF_t =Normal force acting on the area

δF_t = horizontal force to the area, i.e., (tangential force or shear force)

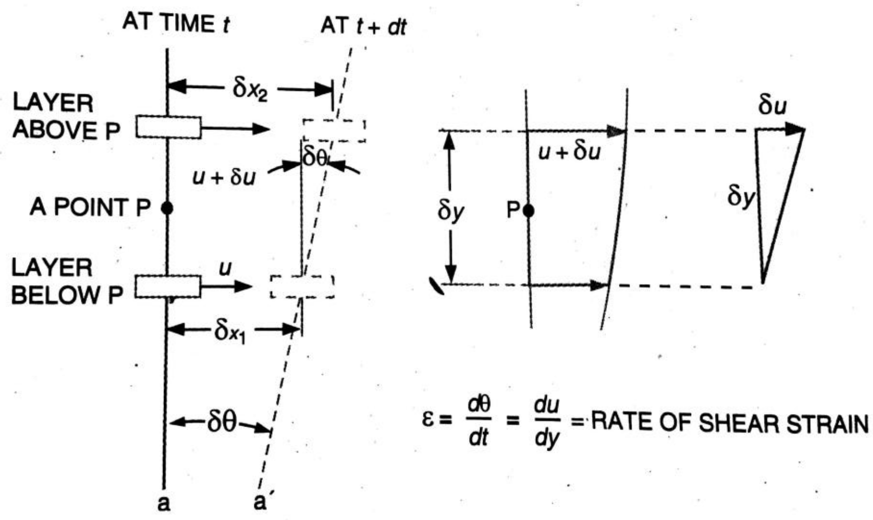
Normal force $\sigma = \lim_{\delta A \to 0} \left(\frac{\delta F_n}{\delta A} \right)$

Tangential or sheer stress $\tau = \lim_{\delta A \to 0} \left(\frac{\delta F_t}{\delta A} \right)$

Rate of strain

The rate of strain at appoint in a fluid refers to the rate of shearing of the adjacent layer at that point This may be interpreted in terms of the rate of change of angle θ of a line a-a joining the

identified fluid elements in layer just above the just the point P. Consider the changes that takes place over an interval of time δt.



This element in the layer below P, moving with a velocity u, translates a distance δx_1 to the Dotted position such that $\delta x_1 = u.\delta t$

The element above P moving with a velocity ($u+\delta u$) translates a distance δx_2 to the dotted position such that $\delta x_2 = (u+\delta u).\delta t$

The line element a-a therefore, shifts to a new position a'-a' such that the inclination $\delta\theta$

$$\delta\theta \cong \tan\delta\theta = \frac{\delta x_2 - \delta x_1}{\delta y} = \frac{(u+\delta u).\delta t - u.\delta t}{\delta y}$$

The rate of shear strain is given by $\varepsilon = \dfrac{d\theta}{dt} = \lim_{\delta t \to 0} \dfrac{\delta\theta}{\delta t} = \dfrac{du}{dt}$

Solids and Fluids:

- **Solid**: A solid can resist a shear stress by a static deflection; The amount of deflection is proportional to the magnitude of applied stress upto some limiting condition. When the applied shear force is constant, a solid stops deforming.

  The molecules of a solid are more closely packed. Hence the force of attraction between the molecules of solid is more larger than that of a fluid.

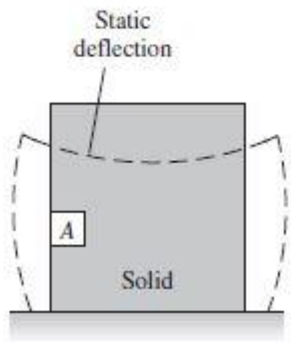


Fig. 1 Deformation in a solid

- **Fluid**: Any shear stress applied to a fluid, no matter how small, will result in motion of that fluid. The fluid moves and deforms continuously as long as the shear stress is applied. When constant shear force is applied, the fluid continues to deform and stops at certain strain rate. In fluids the applied stress is proportional to rate of deformation. A fluid may be either liquid or gas.
  - ✓ **Liquid**: A liquid is composed of relatively close-packed molecules with strong cohesive forces which tends to retain its volume and will form a free surface in a gravitational field if unconfined from above.

    Examples: water, oil, mercury, gasoline, alcohol etc.
  - ✓ **Gas**: A gas, expands until it encounters the walls of the container and fills the entire available space. This is because the gas molecules are widely spaced, and the cohesive forces between them are very small. A gas has no definite volume, and when left to itself without confinement, it forms an atmosphere that is essentially hydrostatic.

    Examples: air, helium, hydrogen, steam etc.,

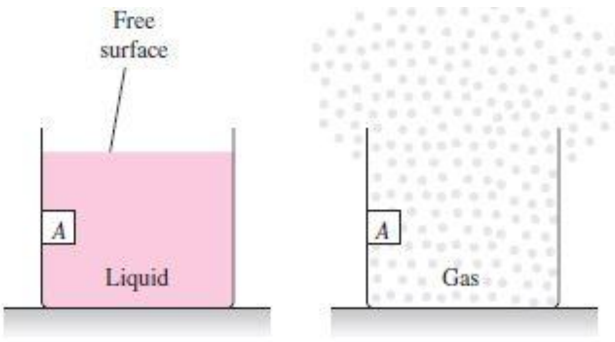


Fig. 2 Liquid and Gas

Solids can resist tangential stress under static conditions whereas fluids can do it only under dynamic conditions.

Basic Fluid Properties:

Characteristics of a fluid which are independent of motion are called basic properties.

1. **Density (ρ):** Density of a fluid is its mass per unit volume.

   Units: kg/m^3 in SI system

   The density of most gases is proportional to pressure and inversely proportional to temperature. Liquids and solids, on the other hand, are essentially incompressible substances, and the variation of their density with pressure is usually negligible. Thus liquid flows are analytically treated as incompressible.

   The heaviest common liquid is Mercury with density, $\rho = 13,580\, {kg}/{m^3}$ and the lightest gas in Hydrogen with density, $\rho = 0.0838\, {kg}/{m^3}$.

2. **Specific weight** (ω): It is the weight of the fluid to the volume occupied with it.

   Specific weight, $\omega = \rho \times g = \frac{m \times g}{v}\, {N}/{m^3}$

3. **Specific volume (V):** It is the volume occupied by the unit mass of fluid and is the reciprocal of density.

   Units: m^3/kg in SI system

4. **Specific gravity (G):** It is the ratio of density of fluid to the density of standard fluid.

   It is also referred as relative density.

   In general, water is the standard fluid for liquids while air is for gases.

   For liquids, $G = \frac{\rho_{liquid}}{\rho_{water}}$ For gases, $G = \frac{\rho_{gas}}{\rho_{air}}$

5. **State relations for liquids:**

Liquids are nearly incompressible and have a single, reasonably constant specific heat. Thus an idealized state relation for a liquid is

$$\rho \approx \text{constant} \quad c_p \approx c_v \approx \text{constant} \quad dh \approx c_p dT$$

The density of a liquid usually decreases slightly with temperature and increases moderately with pressure. If we neglect the temperature effect, an empirical pressure–density relation for a liquid is given as

$$\frac{p}{p_a} \approx (B+1)\left(\frac{\rho}{\rho_a}\right)^n - B$$

where B and n are dimensionless parameters that vary slightly with temperature and p_a and ρ_a are standard atmospheric values. Water can be fitted approximately to the values B=3000 and n=7.

Viscosity: It is a quantitative measure of a fluid's resistance to flow. It determines the strain rate in a fluid that is generated by a given applied shear stress. This property is due to the cohesive forces of attraction between the fluid molecules.

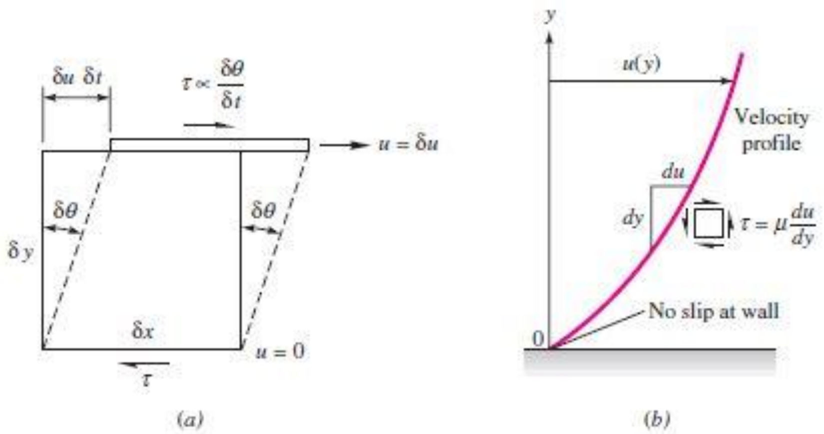


Fig. 3(a) Fluid element straining at rate $\frac{d\theta}{dt}$

Fig. 3(b) Newtonian shear distribution in a shear layer near a wall.

Consider a fluid element sheared in one plane by a single shear stress τ, as in fig. 3(a). The shear strain angle $\delta\theta$ will continuously grow with time as long as the stress τ is maintained, the upper surface moving at speed δu larger than the lower. Such common fluids as water, oil, and air show a linear relation between applied shear and resulting strain rate:

$$\tau \ \alpha \ \frac{\delta\theta}{\delta t}$$

From fig. 3(a), we can write, $\tan\delta\theta = \frac{\delta u \delta t}{\delta y}$

In the limit of infinitesimal changes, this becomes a relation between shear strain rate and velocity gradient:

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

Then, the applied shear is also proportional to the velocity gradient for the common linear fluids. The constant of proportionality is the viscosity coefficient μ:

$$\tau = \mu \frac{d\theta}{dt} = \mu \frac{du}{dy}$$

μ - absolute viscosity or dynamic viscosity.

Unit for Absolute viscosity: N-sec/m2 or Pa-sec

Common unit is poise. 1 poise = 0.1 pa-sec.

From the fig. 3(b), it has to be observed that at the solid boundary, i.e. y=0, the fluid velocity is zero indicating the no-slip condition where the velocity gradient is zero. With increase in the distance from the solid boundary, the velocity increases gradually i.e. velocity gradient increases.

Kinematic viscosity: It is the ratio of dynamic viscosity to the density of the fluid.

$$\gamma = \frac{\mu}{\rho}$$

Units: It is expressed in m^2/sec or cm^2/sec.

1 cm^2/sec = 1 stoke

Parameter	**Liquids**	**Gases**
Pressure	For liquids, both the dynamic and kinematic viscosities are practically independent of pressure, and any small variation with pressure is usually disregarded, except at extremely high pressures.	For gases, dynamic viscosity is independent of pressure (at low to moderate pressures), but not for kinematic viscosity since the density of a gas is proportional to its pressure.
Temperature	The viscosity of liquids decreases with temperature. This is because in a liquid the molecules possess more energy at higher temperatures, and they can oppose the large cohesive intermolecular forces more strongly.	the viscosity of gases increases with temperature. This is because, in a gas the intermolecular forces are negligible, making gas molecules moving randomly at higher velocities at high temperatures. This results in

	As a result, the energized liquid molecules can move more freely.	more molecular collisions per unit volume per unit time and therefore in greater resistance to flow.
Governing equations	Viscosity is given as: $\ln\left(\frac{\mu}{\mu_0}\right) \approx a + b\left(\frac{T_0}{T}\right) + c\left(\frac{T_0}{T}\right)^2$	Viscosity is given as: $\frac{\mu}{\mu_0} \approx \begin{cases} \left(\frac{T_0}{T}\right)^2 \text{.......} Power\ law \\ \frac{\left(\frac{T_0}{T}\right)^{3/2}(T_0+s)}{(T+s)} \text{..} Sutherland\ law \end{cases}$

- ***<u>Newtonian fluids</u>***: These are the fluids for which the rate of deformation is proportional to the shear stress.
  Examples: Air, water, kerosene
- ***<u>Non-Newtonian fluids</u>***: These are the fluids for which the rate of deformation is not directly proportional to the shear stress i.e. $\tau = k\left(\frac{du}{dy}\right)^n$, where k - consistency index, n - flow behavior index. This equation can be rewritten as
  $\tau = k\left(\frac{du}{dy}\right)^n = k\left(\frac{du}{dy}\right)^{n-1}\left(\frac{du}{dy}\right) = \eta\left(\frac{du}{dy}\right)$, where apparent viscosity, $\eta = k\left(\frac{du}{dy}\right)^{n-1}$.
  Non-Newtonian fluids are classified as time independent and time dependent fluids.
  <u>Time independent fluids</u>
  1. *Pseudoplastic fluids:* Fluids in which the apparent viscosity decreases with increasing deformation rate (n<1).
     Examples: Polymer solutions, colloidal suspensions, paper pulp in water, gelatine, blood, milk
  2. *Dilatant fluids*: Fluids in which the apparent viscosity increases with increasing deformation rate (n>1).
     Examples: Suspensions of starch, sugar in starch

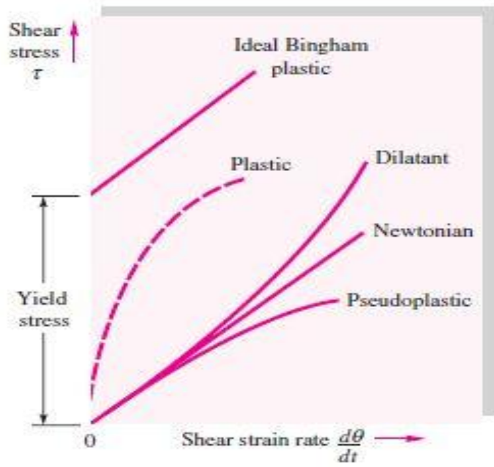


Fig. 4 Stress vs Strain rate

  3. *Bingham plastic*: It is a fluid which behaves as a solid until minimum yield stress

(τ_y) is exceeded and subsequently exhibits a linear relationship between stress and rate of deformation. The relationship is given as $\tau = \tau_y + \mu_p \left(\frac{du}{dy} \right)$

Examples: Clay suspensions, drilling muds

Time dependent fluids

1. *Thixotropic fluids*: Fluids whose apparent viscosity decreases with time under a constant applied stress.
   Examples: Most of the paints
2. *Rheopectic fluids*: Fluids whose apparent viscosity increases with time under a constant applied stress.
3. *Viscoelastic fluids*: These fluids partially return to their original shape after deformation, when the applied stress is released.

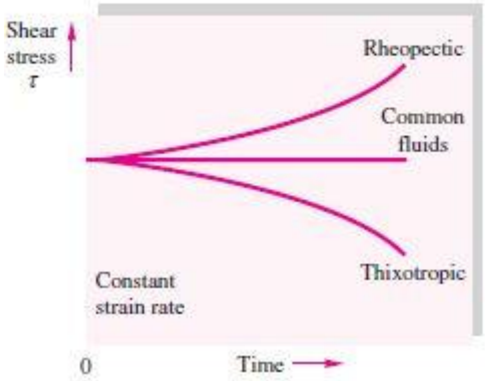


Fig. 5 Shear stress vs time

Surface Tension

Molecules deep within the liquid repel each other because of their close packing. Molecules at the surface are less dense and attract each other. Since half of the neighboring molecules are missing, the mechanical effect is that the surface is in tension. This tensile force acts parallel to the surface and towards the interior of the liquid. The magnitude of this force per unit length is called surface *tension,* σ_s .

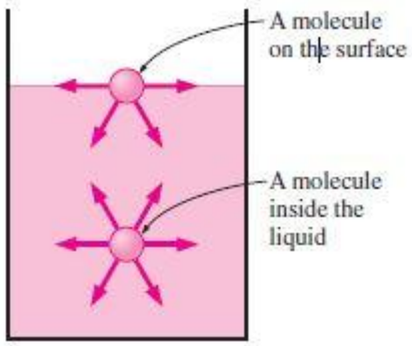


Fig. 6 Attractive forces in liquid

Units: In SI system unit is N/m.

This force (surface tension) is balanced by the repulsive forces from the molecules below the surface that are being compressed. The resulting compression effect causes the liquid to minimize its surface area. This is the reason for the tendency of the liquid droplets to attain a spherical shape, which has the minimum surface area for a given volume.

This effect is also called ***surface energy*** expressed in J/m^2, which represents the stretching work that needs to be done to increase the surface area of the liquid by a unit amount.

Applications: Surface tension determines the size of the liquid droplets that form.

1. *Pressure inside a cylindrical jet:*

   Consider a jet of water exiting as a cylinder with radius 'R' and length 'L'. When one half of the cylinder is considered, as shown in the fig. 7, two forces can be observed. One force is surface tension acting along the circumference and another due to the internal pressure (Δp in excess of atmosphere pressure).

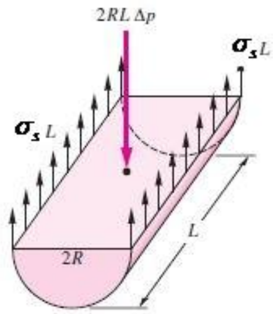
   

   Fig. 7 Section of liquid cylinder

   Surface tension force along the length of the cylinder, $F_t = \sigma_s(L+L) = 2L\sigma_s$

   Force due to internal pressure, $F_p = \Delta p(L \times 2R) = 2LR(\Delta p)$

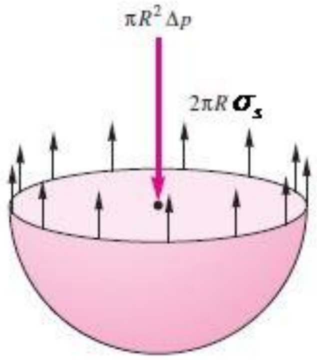
   For equilibrium, both forces should be same i.e. $F_p = F_t \Rightarrow 2LR(\Delta p) = 2L\sigma_S$

   Excess pressure inside a cylindrical jet, $\Delta p = \dfrac{\sigma_S}{R}$

2. *Pressure inside a spherical droplet:*

   Consider a spherical droplet of diameter 'R' subject to internal pressure (Δp in excess of atmosphere pressure) balanced by surface tension acting along the circumference.

   Surface tension force along the circumference, $F_t = \sigma_s(2\pi R)$

   Force due to internal pressure, $F_p = \Delta p(\pi R^2)$

   Fig. 8 Section of spherical droplet

   For equilibrium, both forces should be same i.e. $F_p = F_t \Rightarrow \Delta p(\pi R^2) = (2\pi R)\sigma_S$

   Excess pressure inside a spherical droplet, $\Delta p = \dfrac{2\sigma_S}{R}$

3. *Pressure inside a soap bubble:*

   In the case of a soap bubble, it has two interfaces with air; one at inner surface and another at outer surface. Hence the excess pressure is given as

   $$(\Delta p)_{\substack{soap\\bubble}} \approx 2(\Delta p)_{\substack{spherical\\droplet}} = \frac{4\sigma_S}{R}$$

Bulk Modulus and Compressibility

Bulk modulus, K is defined as the ratio of the change in pressure to the rate of change of volume due to the change in pressure or the ratio of change in pressure to the volumetric strain.

$$K = \frac{-dp}{\left(\frac{dv}{v}\right)}$$

where dp is the change in pressure causing a change in volume dv when the original volume was v. The unit is the same as that of pressure.

Also, we can express volumetric strain in terms of relative change in density, dv/v = – dρ/ρ.

Bulk modulus can also be expressed in terms of change of density as $K = \frac{-dp}{\left(\frac{\Delta v}{v}\right)} = \frac{dp}{\left(\frac{d\rho}{\rho}\right)}$.

The negative sign indicates that if the pressure is increases, then the volume of the substance decreases.

This definition can be applied to liquids as such, without any modifications. In the case of gases, the value of compressibility will depend on the process law for the change of volume and will be different for different processes.

The bulk modulus for liquids depends on both pressure and temperature. The value increases with pressure as dv will be lower at higher pressures for the same value of dp. With temperature the bulk modulus of liquids generally increases, reaches a maximum and then decreases. For water the maximum is at about 50°C. The value is in the range of 2000 MN/m^2.

Pressure

Pressure is a measure of force distribution over any surface associated with the force. Pressure is a surface phenomenon and it can be physically visualized or calculated only if the surface over which it acts is specified. ***Pressure may be defined as the force acting along the normal direction on unit area of the surface.***

$$P = \frac{dF}{dA}$$

The force dF in the normal direction on the area dA due to the pressure P is dF = P dA.

The unit of pressure in the SI system is N/m^2, called as Pascal (Pa). As the magnitude is small kN/m^2 (kPa) and MN/m^2 (Mpa) are more popularly used. The atmospheric pressure is approximately 10^5 N/m^2 and is designated as ‘‘bar’’. This is also a popular unit of pressure. In the metric system the popular unit of pressure is kg-f/cm^2. This is approximately equal to the atmospheric pressure or 1 bar.

Pascal's Law

A fluid element when isolated from the surroundings experiences two types of external forces: (a) Body force and (b) Surface force

Body force: These forces act throughout the body of the fluid element and are distributed over the entire mass or the volume of the element.

Ex: Gravitational force, Magnetic force etc.,

Surface force: It includes all the forces exerted on the fluid element by its surroundings through direct contact at the surface.

Ex: Normal force and shear force

For a fluid element at rest, shear force is zero (shear stress = 0) and hence only normal stresses exist.

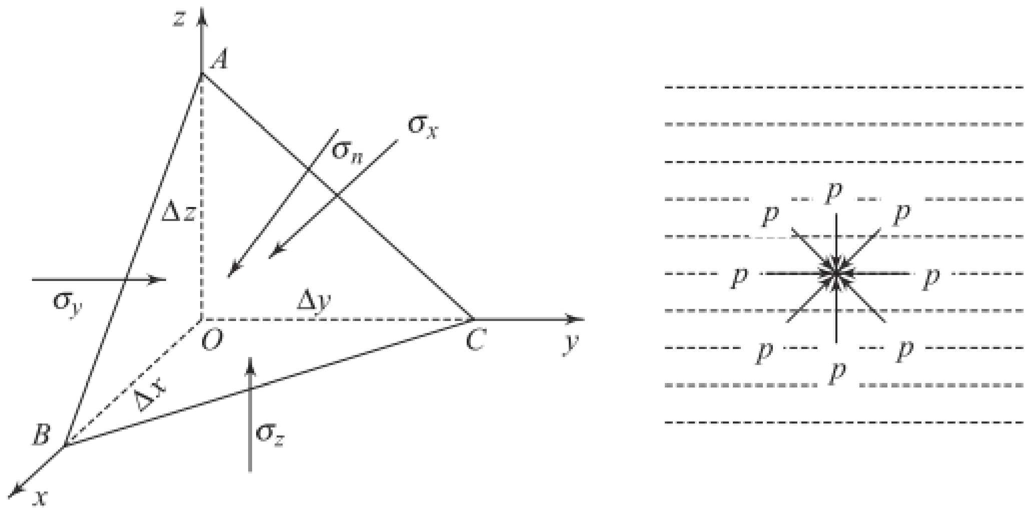


Fig. 9 State of stress on a fluid element at rest, state of normal stress on a fluid element at rest

Consider a stationary fluid element of tetrahedronal shape with three of its faces coinciding with the coordinate planes as shown in the figure. Hence the nature of normal stress acting on the fluid element will be of compressive in nature. Considering gravity as the only external body force, the equations of static equilibrium for the tetrahedronal fluid element are written as:

$$\Sigma F_x = \sigma_x \left(\frac{\Delta y\ \Delta z}{2}\right) - \sigma_n\ \Delta A\ \cos\ \alpha = 0$$
$$\Sigma F_y = \sigma_y \left(\frac{\Delta x\ \Delta z}{2}\right) - \sigma_n\ \Delta A\ \cos\ \beta = 0$$
$$\Sigma F_z = \sigma_z \left(\frac{\Delta x\ \Delta y}{2}\right) - \sigma_n\ \Delta A\ \cos\ \gamma - \frac{\rho g}{6}(\Delta x\ \Delta y\ \Delta z) = 0$$

Where $\sum F_X, \sum F_Y\ and\ \sum F_Z$ are the net forces acting on the fluid element in positive X, Y and Z directions respectively, cosα, cosβ and cosγ are the direction cosines of the normal to the plane of area ΔA.

Therefore,

$\Delta A \cos\alpha = (\Delta y.\Delta z)/2$

$\Delta A \cos\beta = (\Delta x.\Delta z)/2$

$\Delta A \cos\gamma = (\Delta x.\Delta y)/2$.

Substituting these equations in force equilibrium equations gives,

$$\sigma_x = \sigma_y = \sigma_z = \sigma$$

This equation concludes that the normal stress at any point in a fluid at rest are directed towards the point from all directions and are of equal magnitude. These stresses are defined as hydrostatic pressure or thermodynamics pressure. This is known as Pascal's law of hydrostatics. With convention of the positive sign for the tensile stress, the above statement can be analytically expressed as

$$\sigma_x = \sigma_y = \sigma_z = -P$$

Pressure Measurement

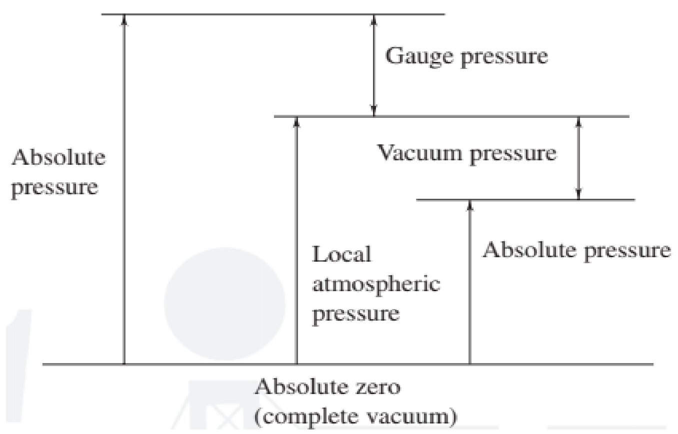


Fig. Scale of Pressure

- Pressure is usually expressed with reference to either absolute zero pressure (complete vacuum) or local atmospheric pressure.

- The absolute pressure is the pressure expressed as a difference between its value and the absolute zero pressure.
- When the pressure is expressed as a difference between its value and the local atmospheric pressure, it is known as gauge pressure.

$$P_{gauge} = P - P_{atm}$$

- If the pressure P is less than the local atmospheric pressure, the gauge pressure becomes negative and is called vacuum pressure.
- At sea level, the international standard atmospheric pressure is given as 101.325kN/m^2.

Pressure is generally measured using a sensing element which is exposed on one side to the pressure to be measured and on the other side to the surrounding atmospheric pressure or other reference pressure.

Piezometer

It is used to measure the gauge pressure i.e. system pressure greater than the local atmospheric pressure. It contains a vertical tube with an opening at the top to atmosphere. Thus when connected to a pipe in which some fluid is flowing, there occurs a rise in the liquid level in the vertical tube called as pressure head.

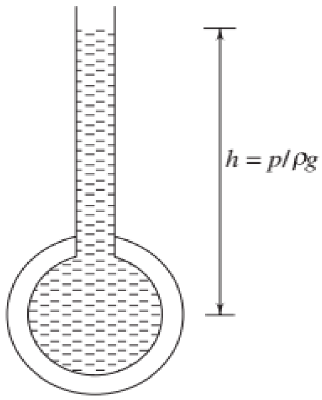


Fig. Piezometer

From the Hydrostatic law, the Pressure at a point 'A' is

P_A = External pressure + Pressure due to rise of liquid level in the tube

$P_A = Patm + \rho gh$

Gauge pressure = $P_A - Patm = \rho gh$

Barometer

It is used to measure the atmospheric pressure. Mercury is used as the working fluid because of its very small vapour pressure at normal temperature and its high density for a relatively short column to be obtained.

When an inverted tube is dipped into a vessel containing fluid like mercury, there occurs a rise in level in the tube above the surface level in the vessel, as shown in the figure.

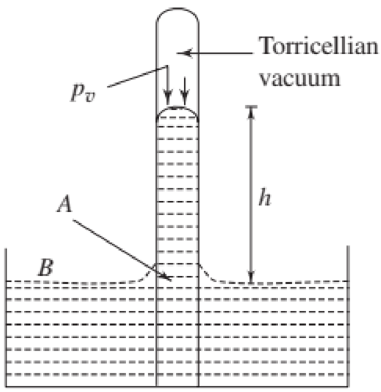


Fig. Barometer

At the level above the mercury in the tube, vacuum is created thus forming mercury vapour having pressure equal to the vapour pressure of mercury at its existing temperature.

From the figure, we can observe that points A and B lie on the same horizontal plane. Therefore we can write, $P_B = P_{atm} = Pv + \rho gh$.

The vapour pressure will be small in comparison to atmospheric pressure and hence it is neglected. Hence $P_B = P_{atm} = \rho gh$.

Bourdon Tube Pressure Gauge:

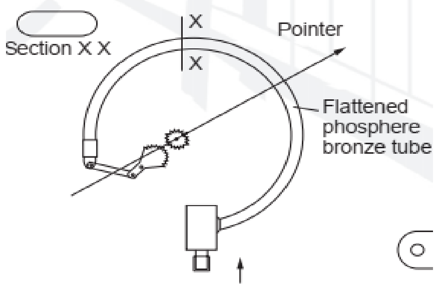


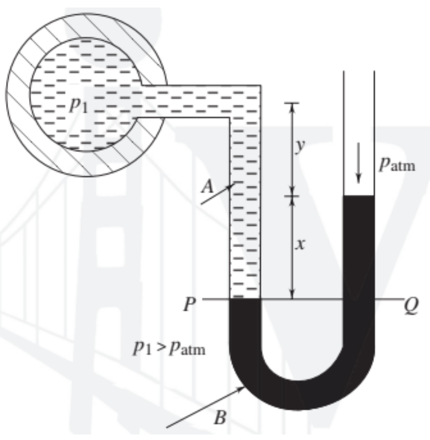
Fig. 9 Bourdon Tube

In the Borden gauge a tube of elliptical section bent into circular shape is exposed on the inside to the pressure to be measured and on the outside to atmospheric pressure. The tube will tend to straighten under pressure. The end of the tube will move due to this action and will actuate through linkages the indicating pointer in proportion to the Pressure Gauge pressure. Vacuum also can be measured by such a gauge. Under vacuum the tube will tend to bend further inwards and as in the case of pressure, will actuate the pointer to indicate the vacuum pressure. The scale is obtained by calibration with known pressure source.

Manometers: Manometer is a device to measure pressure or mostly difference in pressure using a column of liquid to balance the pressure. It is a basic instrument and is used extensively in flow measurement. It needs no calibration. Very low pressures can be measured using micromanometers.

The basic principle of operation of manometers is that at the same level in continuous fluid at rest, the pressure is the same. The pressure due to a constant density liquid (ρ) column if height h is equal to ρgh.

Simple U tube Manometer



It is a transparent 'U' tube as shown in figure. One of its ends is connected to a pipe or a container having fluid (A) whose pressure is to be measured while the other end is open to atmosphere. The lower part of the tube contains a liquid immiscible with the fluid A and is of greater than that of A. This fluid is called manometric fluid. The pressure at two points P and Q in a horizontal plane within the continuous expanse of the same fluid must be equal.

Fig. 10 Manometer to measure gauge pressure

Pressure at the point P = Pressure at the point Q

$$P_1 + \rho_A g\left(y + x\right) = P_{atm} + \rho_B gx$$

$$P_1 - P_{atm} = gx\left(\rho_B - \rho_A\right) - \rho_A gy$$

When the pressure of the fluid in the container is lower than the atmospheric pressure, the liquid levels in the manometer would be adjusted as shown in the figure. Hence it becomes

Pressure at the point P = Pressure at the point Q

$$P_1 + \rho_A gy + \rho_B gx = P_{atm}$$

$$P_{atm} - P_1 = -\rho_A gy - \rho_B gx$$

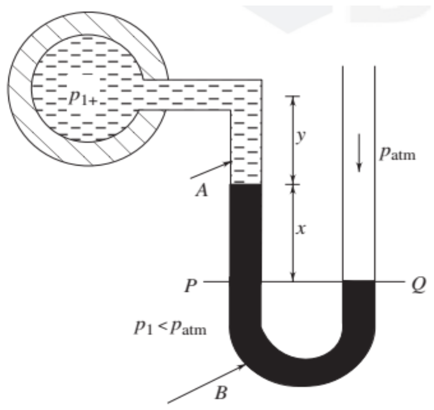


Fig. 11 Manometer measuring vacuum

Differential U tube manometer

A manometer is frequently used to measure the pressure difference, in course of flow, across a restriction in a horizontal pipe as shown in figure.

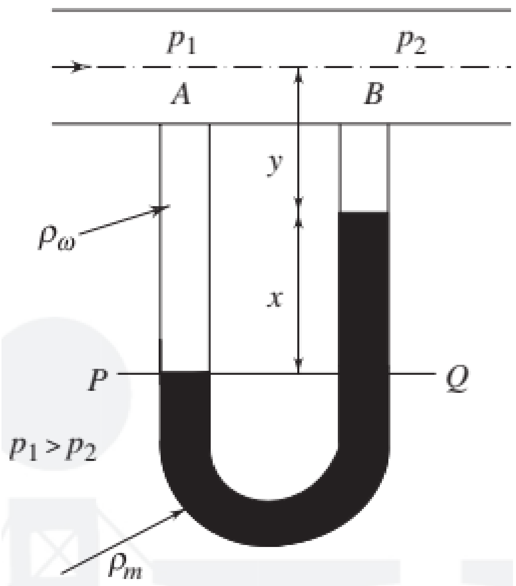


Fig. 12 Manometer measuring pressure differential

It is important that the axis of each connecting tube at A and B to be perpendicular to the direction of the flow and also for the edges of the connection to be smooth. From the principle of hydrostatics, Pressure at the point P = Pressure at the point Q

$$P_1 + \rho_w g\left(y + x\right) = P_2 + \rho_w gy + \rho_m gx$$

$$P_1 - P_2 = gx\left(\rho_m - \rho_w\right)$$

Where ρm - density of manometric fluid and ρw is the density of working fluid.

Other configurations of manometers are:

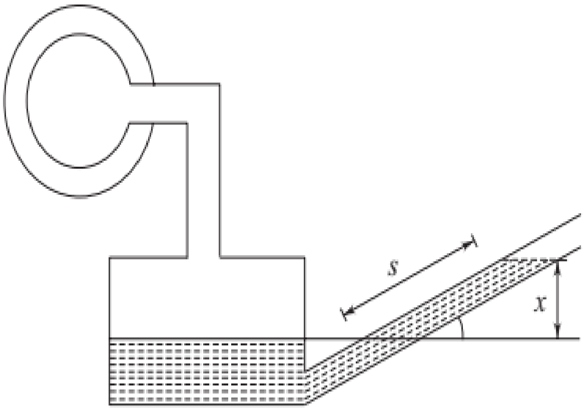


Fig. 13 Inclined tube manometer

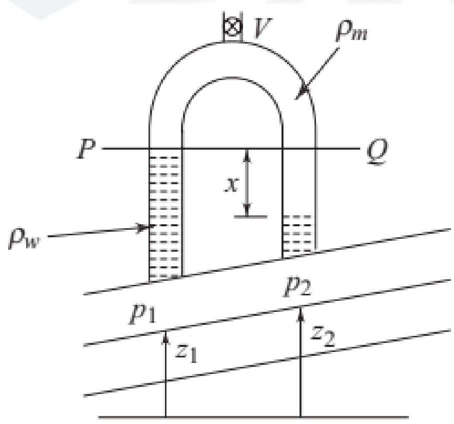


Fig. 14 Inverted tube manometer

FLUID MECHANICS

UNIT-II

FLUID KINEMATICS

Introduction

Fluid Kinematics gives the geometry of fluid motion. It is a branch of fluid mechanics, which describes the fluid motion, and it's consequences without consideration of the nature of forces causing the motion. Fluid kinematics is the study of velocity as a function of space and time in the flow field. From velocity, pressure variations and hence, forces acting on the fluid can be determined.

Velocity field

Velocity at a given point is defined as the instantaneous velocity of the fluid particle, which at a given instant is passing through the point. It is represented by V=V(x,y,z,t). Vectorially, V=ui+vj+wk where u,v,w are three scalar components of velocity in x,y and z directions and (t) is the time. Velocity is a vector quantity and velocity field is a vector field.

Velocity and acceleration

Let V= Resultant velocity at any point in a fluid flow. (u,v,w) are the velocity components in x,y and z directions which are functions of space coordinates and time.

u=u(x,y,z,t) ; v=v(x,y,z,t) ;w=w(x,y,z,t).

Resultant velocity=V=ui+vj+wk

$|V| = (u^2+v^2+w^2)^{1/2}$

Let a_x, a_y, a_z are the total accelerations in the x,y,z directions respectively $a_{x} =$

$[du/dt] = [\partial u/\partial x][\partial x/\partial t] + [\partial u/\partial y][\partial y/\partial t] + [\partial u/\partial z][\partial z/\partial t] + [\partial u/\partial t]$

$a_{x} = [du/dt] = u[\partial u/\partial x] + v[\partial u/\partial y] + w[\partial u/\partial z] + [\partial u/\partial t]$

Similarly,

$a_{y} = [dv/dt] = u[\partial v/\partial x] + v[\partial v/\partial y] + w[\partial v/\partial z] + [\partial v/\partial t]$

$a_{z} = [dw/dt] = u[\partial w/\partial x] + v[\partial w/\partial y] + w[\partial w/\partial z] + [\partial w/\partial t]$

1. Convective Acceleration Terms – The first three terms in the expressions for a_x, a_y, a_z. Convective acceleration is defined as the rate of change of velocity due to change of position of the fluid particles in a flow field

2. Local Acceleration Terms- The 4th term, $[\partial (\ )/ \partial t]$ in the expressions for a_x, a_y, a_z. Local or temporal acceleration is the rate of change of velocity with respect to time at a given point in a flow field.

Material or Substantial Acceleration = Convective Acceleration + Local or Temporal Acceleration

In a steady flow, temporal or local acceleration is zero. In uniform flow, convective acceleration is zero.

For steady flow, $[\partial u/ \partial t]= [\partial v/ \partial t]= [\partial w/ \partial t]= 0$

$a_{x=}$ $[du/dt]= u[\partial u/\partial x]+v[\partial u/\partial y]+w[\partial u/\partial z]$

$a_{y=}$ $[dv/dt]= u[\partial v/\partial x]+v[\partial v/\partial y]+w[\partial v/\partial z]$

$a_{z=}$ $[dw/dt]= [\partial w/\partial x]+v[\partial w/\partial y]+w[\partial w/\partial z]$

Acceleration Vector=$a_x i+a_y j+a_z k$; $|A|=[a_x^2+a_Y^2+a_z^2]^{1/2}$

Method of describing fluid motion

Two methods of describing the fluid motion are: (a) Lagrangian method and (b) Eularian method.

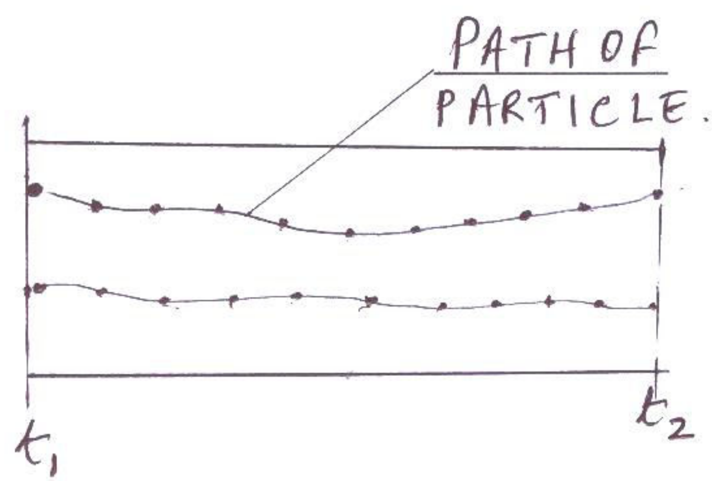


Fig. Lagrangian method

A single fluid particle is followed during its motion and its velocity, acceleration etc. are described with respect to time. Fluid motion is described by tracing the kinematics behavior of each and every individual particle constituting the flow. We follow individual fluid particle as it moves through the flow. The particle is identified by its position at some instant and the time elapsed since that instant. We identify and follow small, fixed masses of fluid. To describe the fluid flow where there is a relative motion, we need to follow many particles and to resolve details of the flow; we need a large number of particles. Therefore, Langrangian method is very difficult and not widely used in Fluid Mechanics.

Eularian method

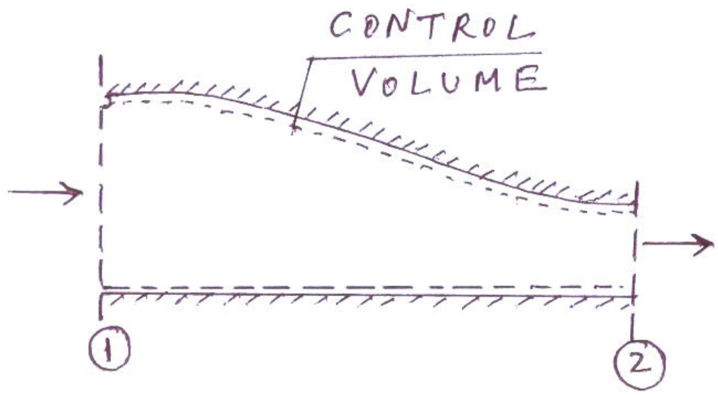


Fig. Eulerian Method

The velocity, acceleration, pressure etc. are described at a point or at a section as a function of time. This method commonly used in Fluid Mechanics. We look for field description, for Ex.; seek the velocity and its variation with time at each and every location in a flow field. Ex., V=V(x,y,z,t). This is also called control volume approach. We draw an imaginary box around a fluid system. The box can be large or small, and it can be stationary or in motion.

Types of fluid flow

1. Steady and Un-steady flows
2. Uniform and Non-uniform flows
3. Laminar and Turbulent flows
4. Compressible and Incompressible flows
5. Rotational and Irrotational flows
6. One, Two and Three dimensional flows

1. **Steady and unsteady flow**

Steady flow is the type of flow in which the various flow parameters and fluid properties at any point do not change with time. In a steady flow, any property may vary from point to point in the field, but all properties remain constant with time at every point.$[\partial V/\partial t]_{x,y,z} = 0$; $[\partial p/\partial t]_{x,y,z} = 0$. Ex.: V=V(x,y,z); p=p(x,y,z) . Time is a criterion.

Unsteady flow is the type of flow in which the various flow parameters and fluid properties at any point change with time. $[\partial V/\partial t]_{x,y,z} \neq 0$; $[\partial p/\partial t]_{x,y,z} \neq 0$, Eg.:V=V(x,y,z,t), p=p(x,y,z,t) or V=V(t), p=p(t) . Time is a criterion

2. **Uniform and non-uniform flows**

Uniform Flow is the type of flow in which velocity and other flow parameters at any instant of time do not change with respect to space. Eg., V=V(x) indicates that the flow is uniform in 'y' and 'z' axis. V=V (t) indicates that the flow is uniform in 'x', 'y' and 'z' directions. Space is a criterion.

Uniform flow field is used to describe a flow in which the magnitude and direction of the velocity vector are constant, i.e., independent of all space coordinates throughout the entire flow field (as opposed to uniform flow at a cross section). That is, $[\partial V/\partial s]_{t=constant} = 0$, that is 'V' has unique value in entire flow field

Non-uniform flow is the type of flow in which velocity and other flow parameters at any instant change with respect to space.
$[\partial V/\partial s]_{t=constant}$ is not equal to zero. Distance or space is a criterion

3. **Laminar and turbulent flows**

Laminar Flow is a type of flow in which the fluid particles move along well-defined paths or stream-lines. The fluid particles move in laminas or layers gliding smoothly over one another. The behavior of fluid particles in motion is a criterion.

Turbulent Flow is a type of flow in which the fluid particles move in zigzag way in the flow field. Fluid particles move randomly from one layer to another. Reynolds number is a criterion. We can assume that for a flow in pipe, for Reynolds No. less than 2000, the flow is laminar; between 2000-4000, the flow is transitional; and greater than 4000, the flow is turbulent.

4. **Compressible and incompressible flows**

Incompressible Flow is a type of flow in which the density (ρ) is constant in the flow field. This assumption is valid for flow Mach numbers with in 0.25. Mach number is used as a criterion. Mach Number is the ratio of flow velocity to velocity of sound waves in the fluid medium

Compressible Flow is the type of flow in which the density of the fluid changes in the flow field. Density is not constant in the flow field. Classification of flow based on Mach number is given below:

M < 0.25 – Low speed

M < unity – Subsonic

M around unity – Transonic

M > unity – Supersonic

M > > unity, (say 7) – Hypersonic

5. Rotational and ir rotational flows

Rotational flow is the type of flow in which the fluid particles while flowing along stream-lines also rotate about their own axis.

Ir-rotational flow is the type of flow in which the fluid particles while flowing along stream-lines do not rotate about their own axis.

6. One,two and three dimensional flows

The number of space dimensions needed to define the flow field completely governs dimensionality of flow field. Flow is classified as one, two and three-dimensional depending upon the number of space co-ordinates required to specify the velocity fields.

One-dimensional flow is the type of flow in which flow parameters such as velocity is a function of time and one space coordinate only.

For Ex., V=V(x,t) – 1-D, unsteady ; V=V(x) – 1-D, steady

Two-dimensional flow is the type of flow in which flow parameters describing the flow vary in two space coordinates and time.

For Ex., V=V(x,y,t) – 2-D, unsteady; V=V(x,y) – 2-D, steady

Three-dimensional flow is the type of flow in which the flow parameters describing the flow vary in three space coordinates and time.

For Ex., V=V(x,y,z,t) – 3-D, unsteady ; V=V(x,y,z) – 3D, steady

Flow patterns

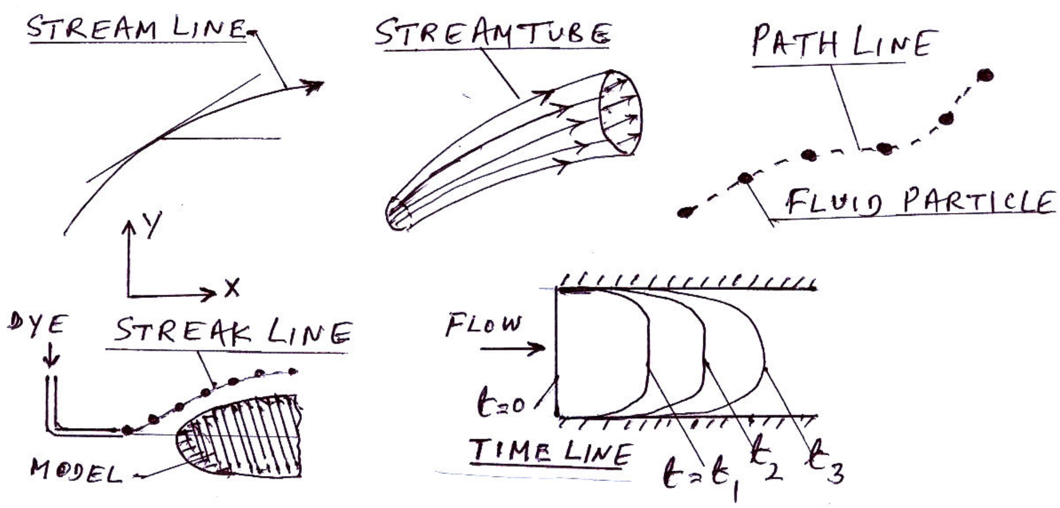


Fig. Flow Patterns

Fluid Mechanics is a visual subject. Patterns of flow can be visualized in several ways. Basic types of line patterns used to visualize flow are streamline, path line, streak line and time line.

(a) **Stream line** is a line, which is everywhere tangent to the velocity vector at a given instant.

(b) **Path line** is the actual path traversed by a given particle.

(c) **Streak line** is the locus of particles that have earlier passed through a prescribed point.

(d) Time line is a set of fluid particles that form a line at a given instant.

Streamline is convenient to calculate mathematically. Other three lines are easier to obtain experimentally. Streamlines are difficult to generate experimentally. Streamlines and Time lines are instantaneous lines. Path lines and streak lines are generated by passage of time. In a steady flow situation, streamlines, path lines and streak lines are identical. In Fluid Mechanics, the most common mathematical result for flow visualization is the streamline pattern – It is a common method of flow pattern presentation.

Streamlines are everywhere tangent to the local velocity vector. For a stream line, (dx/u) = (dy/v) = (dz/w). Stream tube is formed by a closed collection of streamlines. Fluid within the stream tube is confined there because flow cannot cross streamlines. Stream tube walls need not be solid, but may be fluid surfaces

Continuity Equation

Rate of flow or discharge (Q) is the volume of fluid flowing per second. For incompressible fluids flowing across a section,

Volume flow rate, Q= A×V m^3/s where A=cross sectional area and V= average velocity.

For compressible fluids, rate of flow is expressed as mass of fluid flowing across a section per second.

Mass flow rate (m) =(ρAV) kg/s where ρ = density.

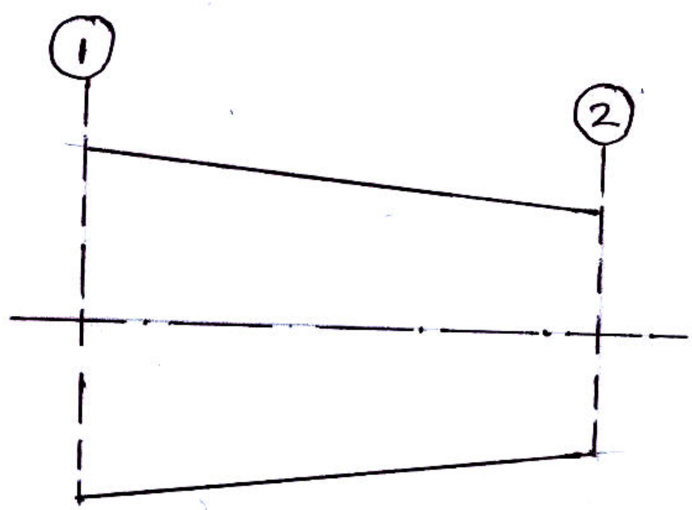


Fig. Continuity Equation

Continuity equation is based on Law of Conservation of Mass. For a fluid flowing through a pipe, in a steady flow, the quantity of fluid flowing per second at all cross-sections is a constant.

Let v_1=average velocity at section [1],

ρ_1=density of fluid at [1],

A_1=area of flow at [1];

Let v_2, ρ_2, A_2 be corresponding values at section [2].

Rate of flow at section [1]= $\rho_1 A_1 v_1$

Rate of flow at section [2]= $\rho_2 A_2 v_2$

$\rho_1 A_1 v_1 = \rho_2 A_2 v_2$

This equation is applicable to steady compressible or incompressible fluid flows and is called Continuity Equation. If the fluid is incompressible, $\rho_1 = \rho_2$ and the continuity equation reduces to $A_1 v_1 = A_2 v_2$

For steady, one dimensional flow with one inlet and one outlet,

$\rho_1 A_1 v_1 - \rho_2 A_2 v_2 = 0$

For control volume with N inlets and outlets

$\sum_{i=1}^{N} (\rho_i A_i v_i) = 0$ where inflows are positive and outflows are negative .

Velocities are normal to the areas. This is the continuity equation for steady one dimensional flow through a fixed control volume

When density is constant, $\sum_{i=1}^{N} (A_i v_i) = 0$

Continuity equation in 3-dimensions

(Differential form, Cartesian co-ordinates)

Consider infinitesimal control volume as shown of dimensions dx, dy and dz in x,y,and z directions

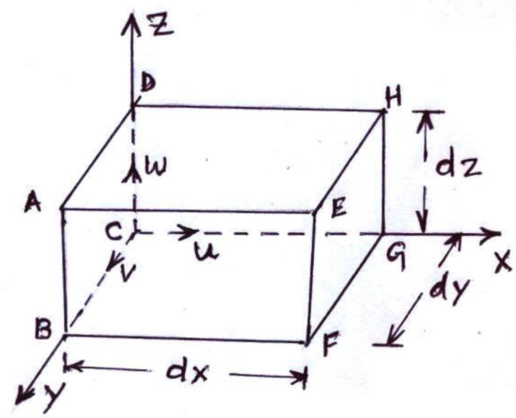


Fig. Continuity Equation in Three Dimensions

u,v,w are the velocities in x,y,z directions.

Mass of fluid entering the face ABCD =

Density ×velocity in x-direction ×Area ABCD = $\rho u dy.dz$

Mass of fluid leaving the face EFGH= $\rho u dy.dz + [\partial (\rho u dy.dz)/ \partial x](dx)$

Therefore, net rate of mass efflux in x-direction= $-[\partial (\rho u dy.dz)/ \partial x](dx) =$

$- [\partial (\rho u)/ \partial x](dxdydz)$

Similarly, the net rate of mass efflux in

y-direction= $- [\partial (\rho v)/ \partial y](dxdydz)$ z-

direction= $- [\partial (\rho w)/ \partial z](dxdydz)$

The rate of accumulation of mass within the control volume $=\partial (\rho dV)/ \partial t = \rho\partial/\partial t\ (dV)$

where dV=Volume of the element=dxdydz and dV is invarient with time.

From conservation of mass, the net rate of efflux = Rate of accumulation of mass within the control volume.

$- [\partial(\rho u)/\partial x + \partial(\rho v)/\partial y + \partial(\rho w)/\partial z](dxdydz) = \rho\partial/\partial t(dxdydz)$ OR

$\rho\partial/\partial t + \partial(\rho u)/\partial x + \partial(\rho v)/\partial y + \partial(\rho w)/\partial z = 0$

This is the continuity equation applicable for

(a) Steady and unsteady flows

(b) Uniform and non-uniform flows

(c) Compressible and incompressible flows.

For steady flows, $(\partial/\partial t) = 0$ and $[\partial(\rho u)/\partial x + \partial(\rho v)/\partial y + \partial(\rho w)/\partial z] = 0$

If the fluid is incompressible, ρ= constant $[\partial u/\partial x + \partial v/\partial y + \partial w/\partial z] = 0$

This is the continuity equation for 3-D flows.

For 2-D flows, w=0 and $[\partial u/\partial x + \partial v/\partial y] = 0$

Velocity potential and stream function

Velocity Potential Function is a Scalar Function of space and time co-ordinates such that its negative derivatives with respect to any direction give the fluid velocity in that direction.

$\Phi = \Phi(x,y,z)$ for steady flow.

$u = -(\partial\Phi/\partial x)$; $v = -(\partial\Phi/\partial y)$; $w = -(\partial\Phi/\partial z)$ where u,v,w are the components of velocity in x,y and z directions.

In cylindrical co-ordinates, the velocity potential function is given by $u_r = (\partial\Phi/\partial r)$, $u_\theta = (1/r)(\partial\Phi/\partial\theta)$

The continuity equation for an incompressible flow in steady state is

$(\partial u/\partial x + \partial v/\partial y + \partial w/\partial z) = 0$

Substituting for u, v and w and simplifying,

$(\partial^2\Phi/\partial x^2 + \partial^2\Phi/\partial y^2 + \partial^2\Phi/\partial z^2) = 0$

Which is a Laplace Equation. For 2-D Flow, $(\partial^2\Phi/\partial x^2 + \partial^2\Phi/\partial y^2) = 0$

If any function satisfies Laplace equation, it corresponds to some case of steady incompressible fluid flow.

Ir rotational flow and velocity potential

Assumption of Ir-rotational flow leads to the existence of velocity potential. Consider the rotation of the fluid particle about an axis parallel to z-axis. The rotation component is defined as the average angular velocity of two infinitesimal linear segments that are mutually perpendicular to each other and to the axis of rotation.

Consider two-line segments δx, δy. The particle at P(x,y) has velocity components u,v in the x-y plane.

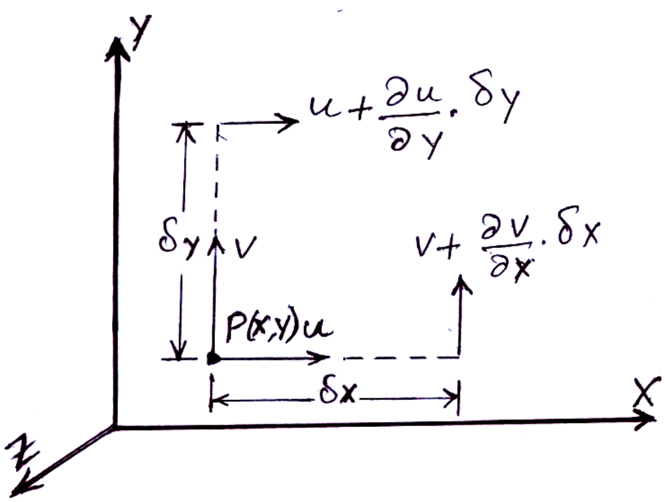


Fig. Rotation of a fluid partical.

The angular velocities of δx and δy are sought.

The angular velocity of (δx) is $\{[v + (\partial v/\partial x)\ \delta x - v] / \delta x\} = (\partial v/\partial x)$ rad/sec

The angular velocity of (δy) is $-\{[u + (\partial u/\partial y)\ \delta y - u] / \delta y\} = -(\partial u/\partial y)$ rad/sec

Counter clockwise direction is taken positive. Hence, by definition, rotation component (ω_z) is $\omega_z = 1/2\ \{(\partial v/\partial x) - (\partial u/\partial y)\}$. The other two components are

$\omega_x = 1/2\ \{(\partial w/\partial y) -$

$(\partial v/\partial z)\}$ $\omega_y = 1/2$

$\{(\partial u/\partial z) - (\partial w/\partial x)\}$

The rotation vector = $\omega = i\omega_x + j\omega_y + k\omega_z$.

The vorticity vector(Ω) is defined as twice the rotation vector = 2ω

Properties of potential function

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

$\omega_x = 1/2\ \{(\partial w/\partial y) - (\partial v/\partial z)\}$

$\omega_y = 1/2\{(\partial u/\partial z) - \partial w/\partial x)\}$;

Substituting $u = -(\partial \Phi/\partial x)$;

$v = -(\partial \Phi/\partial y)$;

$w = -(\partial \Phi/\partial z)$;

we get $\omega_z = 1/2\ \{(\partial/\partial x)(-\partial \Phi/\partial y) - (\partial/\partial y)(-\partial \Phi/\partial x)\}$

$= ½\{-(\partial^2\Phi/\partial x\partial y) + (\partial^2\Phi/\partial y\partial x)\} = 0$ since Φ is a continuous function.

Similarly, $\omega_x=0$ and $\omega_y=0$

All rotational components are zero and the flow is irrotational.– Therefore, irrotational flow is also called as Potential Flow.

If the velocity potential (Φ) exists, the flow should be irrotational. If velocity potential function satisfies Laplace Equation, It represents the possible case of steady, incompressible, irrotational flow. Assumption of a velocity potential is equivalent to the assumption of irrotational flow.

Laplace equation has several solutions depending upon boundary conditions. If

Φ_1 and Φ_2 are both solutions, $\Phi_1 + \Phi_2$ is also a solution

$\nabla^2(\Phi_1)=0$, $\nabla^2(\Phi_2)=0$, $\nabla^2(\Phi_1 + \Phi_2)=0$

Also if Φ_1 is a solution, $C\Phi_1$ is also a solution (where C=Constant)

Stream function (ψ)

Stream Function is defined as the scalar function of space and time such that its partial derivative with respect to any direction gives the velocity component at right angles to that direction. Stream function is defined only for two dimensional flows and 3-D flows with axial symmetry.

$(\partial\psi/\partial x) = v$;

$(\partial\psi/\partial y) = -u$

In Cylindrical coordinates

, $u_r = (1/r)\ (\partial\psi/\partial\theta)$ and

$u_\theta = (\partial\psi/\partial r)$

Continuity equation for 2-D flow is $(\partial u/\partial x) + (\partial v/\partial y)$

$=0\ (\partial/\partial x)\ (-\partial\psi/\partial y) + (\partial/\partial y)\ (\partial\psi/\partial x) = 0$

$-(\partial^2\psi/\partial x\partial y) + (\partial^2\psi/\partial y\partial x) = 0;$

Therefore, continuity equation is satisfied. Hence, the existence of (ψ) means a possible case of fluid flow. The flow may be rotational or irrotational. The rotational component are:

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

$\omega_z = 1/2\ \{(\partial/\partial x)(\partial\psi/\partial x) - (\partial/\partial y)(-$

$\partial\psi/\partial y)\}$ $\omega_z = ½\{(\partial^2 \psi/\partial x^2) + (\partial^2\psi/\partial y^2)\}$

For irrotational flow, $\omega_z = 0$. Hence for 2-D flow, $(\partial^2 \psi/\partial x^2) + (\partial^2\psi/\partial y^2) = 0$ which is a Laplace equation.

Properties of stream function

1.If the Stream Function (ψ) exists, it is a possible case of fluid flow, which may be rotational or irrotational.

2.If Stream Function satisfies Laplace Equation, it is a possible case of an irrotational flow.

Equi-potential & constant stream function lines

On an equi-potential line, the velocity potential is constant, Φ=constant or $d(\Phi)=0$.

$\Phi = \Phi(x,y)$ for steady flow.

$d(\Phi) = (\partial\Phi/\partial x)\, dx + (\partial\Phi/\partial y)\, dy$.

$d(\Phi) = -u\, dx - v\, dy = -(u\, dx + v\, dy) = 0$.

For equi-potential line, $u\, dx + v\, dy = 0$

Therefore, $(dy/dx) = -(u/v)$ which is a slope of equi-potential lines

For lines of constant stream Function,

ψ = Constant or $d(\psi)=0$, $\psi = \psi(x,y)$

$d(\psi) = (\partial\psi/\partial x)\, dx + (\partial\psi/\partial y)\, dy = vdx -$

udy Since $(\partial\psi/\partial x) = v$; $(\partial\psi/\partial y) = -u$

Therefore, $(dy/dx) = (v/u)$ = slope of the constant stream function line. This is the slope of the stream line.

The product of the slope of the equi-potential line and the slope of the constant stream function line (or stream Line) at the point of intersection = -1.

Thus, equi-potential lines and streamlines are orthogonal at all points of intersection.

Flow net

A grid obtained by drawing a series of equi-potential lines and streamlines is called a Flow Net. A Flow Net is an important tool in analyzing two-dimensional ir-rotational flow problems.

<u>Examples of flow nets</u>

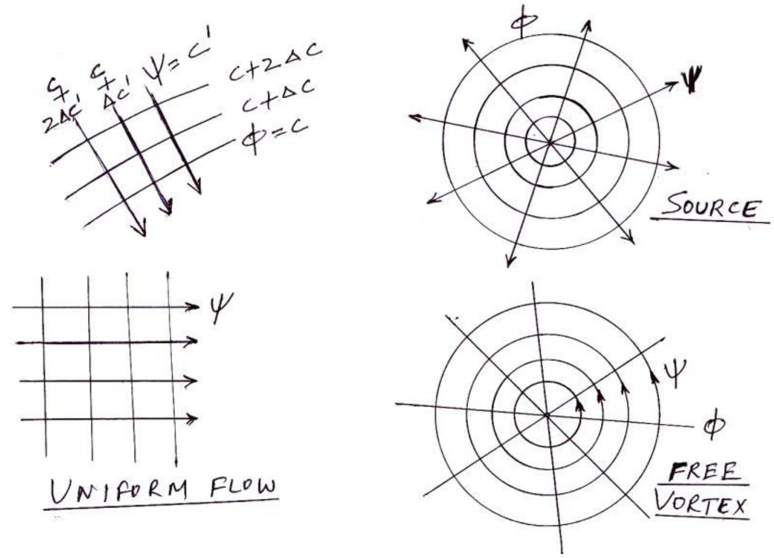


Fig. Flow Nets

Examples: Uniform flow, Line source and sink, Line vortex

Two-dimensional doublet – a limiting case of a line source approaching a line sink

<u>Relationship between stream function and velocity Potential</u>

$u = -(\partial \Phi/\partial x)$, $v = -(\partial \Phi/\partial y)$

$u = -(\partial \psi/\partial y)$, $v = (\partial \psi/\partial x)$; Therefore, -

$(\partial \Phi/\partial x) = -(\partial \psi/\partial y)$ and $-(\partial \Phi/\partial y) = (\partial \psi/\partial x)$

Hence, $(\partial \Phi/\partial x) = (\partial \psi/\partial y)$ and $(\partial \Phi/\partial y) = -(\partial \psi/\partial x)$

Types of motion

A Fluid particle while moving in a fluid may undergo any one or a combination of the following four types of displacements:

1. Linear or pure translation
2. Linear deformation
3. Angular deformation
4. Rotation.

(1) Linear Translation is defined as the movement of fluid element in which fluid element moves from one position to another bodily – Two axes ab & cd and a'b'& c'd' are parallel (Fig. 8a)

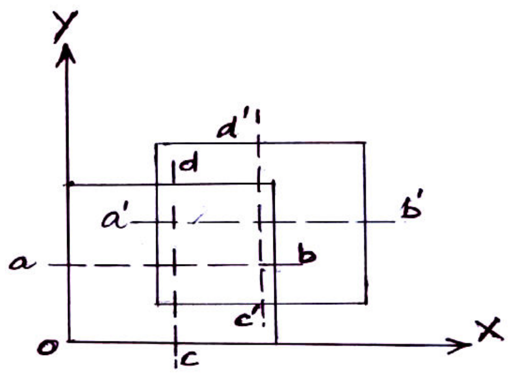


Fig. Linear translation.

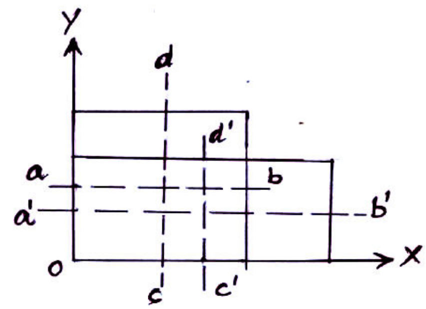


Fig. Linear deformation.

(2) Linear deformation is defined as deformation of fluid element in linear direction – axes are parallel, but length changes.

(2) Angular deformation, also called shear deformation is defined as the average change in the angle contained by two adjacent sides. The angular deformation or shear strain rate = $\frac{1}{2}(\Delta\theta_1 + \Delta\theta_2) = \frac{1}{2}(\partial v/\partial x + \partial u/\partial y)$

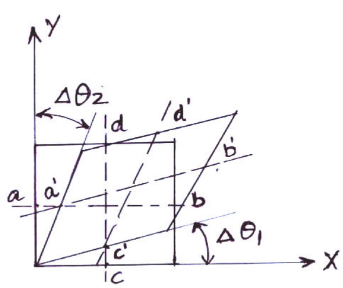


Fig Angular deformation

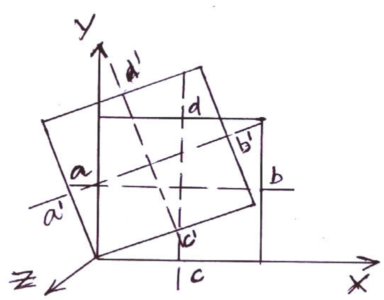


Fig. Rotation

(4) Rotation is defined as the movement of the fluid element in such a way that both its axes (horizontal as well as vertical) rotate in the same direction. Rotational components are:

Fluid Mechanics
Unit-III
Fluid Dynamics

- Fluid Dynamics is the branch of fluid science that deals with the study of fluid motion and the associated external forces and internal resistances.
- Based on Newton's second law, it is possible to derive certain differential equations, called equations of motion or momentum conservation, which completely define the Dynamic behavior of the body.

TYPES OF FORCES IN FLUID MOTION:

The different forces acting on the fluid mass in motion may be classified as follows;

i. Body forces or Volume forces (Weights, Gravity Force $\mathbf{F_g}$, compressibility or elastic force $\mathbf{F_e}$)

ii. Surface forces (Pressure forces $\mathbf{F_p}$, Shear Forces, viscous force $\mathbf{F_v}$, Turbulent Force $\mathbf{F_t}$)

iii. Linear Forces (Capillary or surface tension forces $\mathbf{F_s}$)

If a certain mass of fluid in motion is influenced by all the above mentioned forces, then according to newtons second law, the following equations of motion can be written.

$M \times a = F_g + F_e + F_p + F_v + F_t + F_s$ -------------------------------- eq. (1)

Where M = Mass of the fluid in flow

a= Acceleration of the flow

Further by resolving the various forces in x, y & z directions, we get

$M \times a_x = F_{gx} + F_{ex} + F_{px} + F_{vx} + F_{tx} + F_{sx}$

$M \times a_y = F_{gy} + F_{ey} + F_{py} + F_{vy} + F_{t\,y} + F_{sy}$

$M \times a_z = F_{gz} + F_{ez} + F_{pz} + F_{vz} + F_{tz} + F_{sz}$

Reynold's equation of motion:

In most of the problems of fluids in motion, the surface tension forces and compressibility forces are not significant. Hence these forces can be neglected and eq. (1) becomes

$M \times a = F_g + F_p + F_v + F_t$ -- eq. (2)

The above eq is known as Reynold's equation of motion.

Navier – stokes equation of motion:

Further for laminar or viscous flows, turbulent forces also become less significant. Hence eq. (2) becomes

$M \times a = F_g + F_p + F_v$ --- eq. (3)

The above eq is known as Navier – stokes equation of motion.

Euler's equation of motion:

Further if the viscous forces are also of little significance, then these forces may also be neglected. Hence eq. (3) becomes

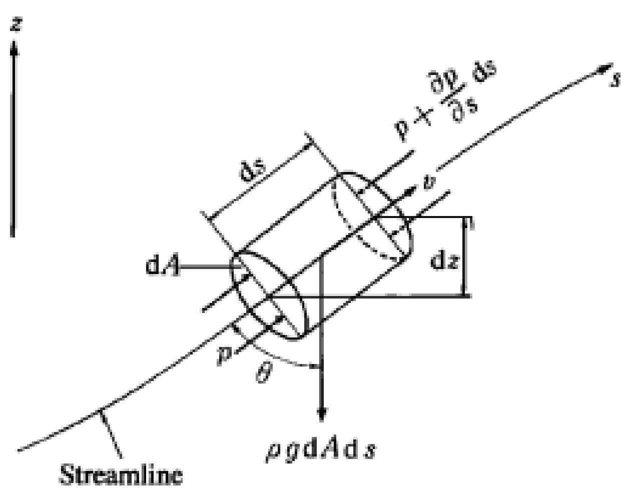
$M \times a = F_g + F_p$ --- eq. (4)

The above equation is known as Euler's equation of motion.

EULER'S EQUATION FOR 1-D FLOW:

General flows are three dimensional, but many of them may be studied as if they are one dimensional. For example, whenever a flow in a tube is considered, if it is studied in terms of mean velocity, it is a one-dimensional flow which is studied very simply. Let us apply the principle of conservation of momentum to the infinitesimal, cylindrical element of fluid having the cross-sectional area dA and length ds which lay along the streamline

- We also assume that the flow is steady, it is that all local time derivatives are equal zero, $\partial v / \partial t = 0$.
- We also assume that flow is invicid (ideal).



Around a cylindrical element of fluid having the cross-sectional area 'dA' and length 'ds' is considered.

- Let p be the pressure acting on the lower face, and pressure {p+ dp} acts on the upper face a distance ‘ds’ away.
- The gravitational force acting on this element is its weight, [p.g.dA.ds]. Applying Newton’s second law of motion to this element, the resultant force acting on it, and producing acceleration along the streamline is the force due to the pressure difference across the streamline and the component of any other external force (in this case only the gravitational force) along the streamline. Therefore the following equation is obtained:
- We know that the external forces tending to accelerate the fluid element in the direction of the streamline

$$= P.dA - (P + dP)dA$$
$$= -dP.dA$$

- **(1)**
- We also know that the weight of the fluid element,

$$dW = \rho g.dA.ds$$

- From the geometry of the figure, we find that the component of the weight of the fluid element in the direction of flow,

$$= \rho g.dA.ds \cos\theta$$
$$= \rho g.dA.ds \frac{dz}{ds}$$
$$= \rho g.dA.dz$$

- **(2)**
- ∴ Mass of the fluid element = $\rho.dA.ds$
- We see that the acceleration of the fluid element

$$\frac{dv}{dt} = \frac{dv}{ds} \times \frac{ds}{dt} = v\frac{dv}{ds}$$

- **(3)**
- Now, as per Newton's second law of motion, we know that Force = Mass *Acceleration

$$\Rightarrow (-dp.dA) - (\rho g.dA.dz) = \rho.dA.ds \times v.\frac{dv}{ds}$$

- Dividing both sides by $(-\rho dA)$

$$\Rightarrow \frac{dP}{\rho} + g.dz = -v.dv$$

- or,

$$\Rightarrow \frac{dP}{\rho} + g.dz + v.dv = 0$$

- **(4)**
- This is the required Euler's equation for motion as in the form of a differential equation.

BERNOLLIES EQUATION FROM EULER’S EQUATION

- Integrating the above equation,

$$\frac{1}{\rho}\int dP + \int g.dz + v.dv = Constant$$
$$\Rightarrow \frac{P}{\rho} + gz + \frac{v^2}{2} = Constant$$
$$\Rightarrow P + wz + \frac{wv^2}{2g} = Constant$$
$$\Rightarrow \frac{P}{w} + Z + \frac{v^2}{2g} = Constant$$

- or in other words,

$$\frac{P_1}{w_1} + Z_1 + \frac{v_1^2}{2g} = \frac{P_2}{w_2} + Z_2 + \frac{v_2^2}{2g}$$

MOMENTUM EQUATION:

It is based on law of conservation of momentum, which states that the net force acting on the fluid mass is equal to the change in momentum of flow per unit time in that direction .

The force acting on fluid mass ‘m’ is given by Newton’s second law

F = m×a

But a = dv/dt

Therefore F = m [dv/dt]

⇨ F = [d(m.v)/dt]------------------ (1) momentum principle

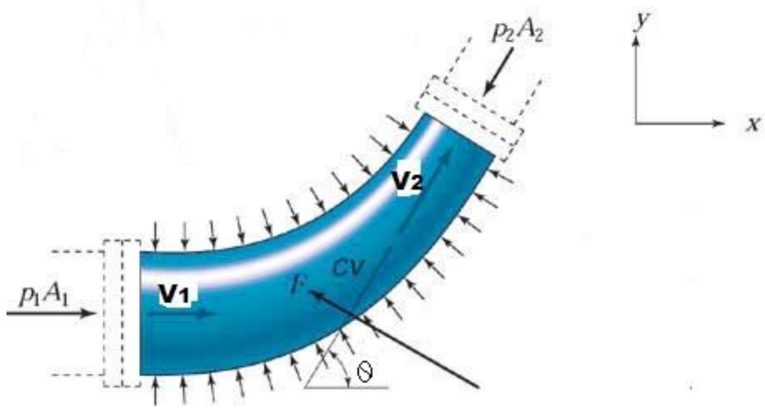
⇨ F×dt = d(m.v)------------------- (2) Impulse – momentum equation

Impulse momentum equation states that the impulse of a force F acting on a fluid mass m in a short interval of time dt is equal to the change of momentum d (m.v) in the direction of force.

Application:

The Impulse momentum equation is used to determine the resultanyt force exerted by flowing fluid on a pipe bend.

Consider a pipe bend with two sections 1 and 2 as shown in figure.



Let V_1 and V_2 be the velocities at inlet and outlet respectively. P_1 and P_2 be the pressures at inlet and outlet respectively.

The continuity equation yields $\rho\, V_1\, A_1 \;=\; \rho\, V_2\, A_2 \;=\; \dot{m}$

Carrying out a force balance in x-direction, we have

$$p_1\, A_1 \;-\; p_2\, A_2\, \cos\theta \;+\; F_x \;=\; \rho V_2\, A_2\, V_2\, \cos\theta \;-\; \rho\, V_1\, A_1\, V_1$$

$$= \dot{m}\, V_2\, \cos\theta \;-\; \dot{m}\, V_1 \;=\; \dot{m}\, (V_2\, \cos\theta \;-V_1)$$

In the y-direction we have,

$$0 \;-\; p_2\, A_2\, \sin\theta \;+\; F_y \;=\; \rho V_2\, A_2\, V_2\, \sin\theta \;-\; 0$$

giving $$p_2\, A_2\, \sin\theta \;+\; F_y \;=\; \dot{m}\, V_2\, \sin\theta$$

Thus the force components acting on the bend are

$$F_x \;=\; p_2\, A_2\, \cos\theta \;-\; p_1\, A_1 \;-\; \dot{m}\, (V_2\, \cos\theta \;-\; V_1)$$

$$F_y \;= -p_2\, A_2\, \sin\theta \;+\; \dot{m}\, V_2\, \sin\theta$$

Unit – V

Fluid Mechanics
Laminar flow

Course Objectives: To introduce the principles of viscous flow ,shear stress and velocity distribution in a laminar flow

Course Outcomes:

- Students will be able to understand the shear stress and velocity distribution
- Apply the concept of viscous flow in a pipe and plates
- Analyze the shear stress and velocity distribution in viscous flow field

UNIT-V

Concept of Laminar Flow Laminar Fully Developed Pipe Flow Hagen Poiseuille flow laminar flow through inclined pipes Laminar flow through annulus Flow between parallel plates – Both plates at Rest L5.07 Flow between parallel plates ,one Plate is moving and other at rest L5.08 Introduction to turbulent flows L5.09 Moody Chart

RELATION BETWEEN SHEAR AND PRESSURE GRADIENTS IN LAMINAR FLOW

Consider a free body of fluid having the form of an elementary parallelopiped of length δx ,width δz,Thickness δy as shown in fig 5.1.On account of relative motion between different layers of fluid,the Velocity distribution is non-uniform.Thus the fluid layer abcd is moving at a higher velocity than the Layer below it and hence the layer abcd exerts a shear stress in the positive direction on the lower layer. the lower layer on the other hand, exerts an equal and opposite shear stress on the layer abcd. Similarly it can be seen that a shear stress is exerted on the layer a'b'c'd' by the layer above it in the positive x-direction. The magnitudes of shear stresses ion the layers abcd and a'b'c'd' will be different. Thus if τ represents the shear on the layer abcd then the shear stress on the layer a'b'c'd' is equal to$\left(\tau + \frac{\partial \tau}{\partial y}\right) \delta y$

For two-dimensional steady flow there will be no shear stresses on the vertical faces abb'a'. Thus the only forces acting on the parallelepiped in the direction of flow x will be the pressure and shear forces as shown in Fig 5.1. The net shear force acting on the parallelepiped

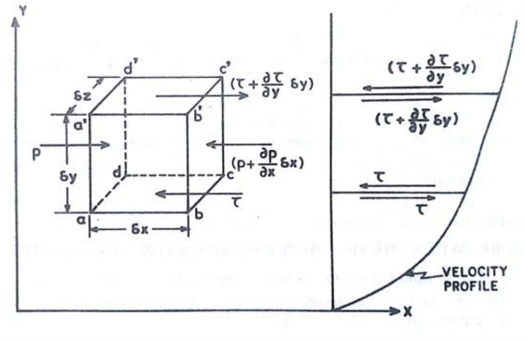


Fig 5.1 Forces on a fluid element in laminar flow

$$\left[\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right)\delta x \delta z - \tau \delta x \delta z\right] = \frac{\partial \tau}{\partial y}\delta x \delta y \delta z.$$

If the pressure intensity on face add 'a' is p, and since there exists a pressure gradient in the direction of flow, the pressure intensity on the face bcc 'b' will be $\left(p + \frac{\partial p}{\partial x}\delta x\right)$.The net pressure force acting on the parallelepiped

$$= \left[p\delta y \delta z - \left(p + \frac{\partial p}{\partial x}\delta x\right)\delta y \delta z\right] = -\left(\frac{\partial p}{\delta x}\right)\delta x \delta y \delta z.$$

For steady and uniform flow, there being no acceleration in the direction of motion, the sum of these forces in the x-direction must be equal to zero. Thus

$$\left(\frac{\partial \tau}{\partial y}\right)\delta x \delta y \delta z - \left(\frac{\partial p}{\partial x}\right)\delta x \delta y, \delta z = 0$$

$$\text{or } \frac{\partial \tau}{\partial y} = \frac{\partial p}{\partial x} \qquad \ldots(5.1)$$

Equation 5.1 indicates that in a steady uniform laminar flow the pressure gradient in the direction of flow is equal to the shear stress gradient in the normal direction. Further for steady uniform flow, since acceleration is absent, it is apparent that the pressure gradient $(\partial p/\partial y)$ is independent of y and the shear stress gradiant $(\partial \tau/\partial y)$ is independent of x.

Moreover, for viscous fluids according to Newton's law of viscosity $\tau = \mu(\partial v/\partial y)$.Thus substituting for τ in equation 5.1 the following differential equation for laminar flow is obtained

$$\mu \frac{\delta^2 v}{\delta y^2} = \frac{\partial p}{\partial x} \qquad \ldots(5.2)$$

By integrating equation 5.1 or 5.2 the problems of steady uniform laminar flow can be analysed.

STEADY LAMINAR FLOW IN CIRCULAR PIPES-HAGEN-POISEUILLE LAW:

Fig:3.2: shows a horizontal circular pipe having a laminar flow of fluid through it .Consider two sections 1 and 2 of this pipe L distance apart .let p1 and p2 be the average intensities of pressure acting at two sections 1 and 2 respectively and let V be the mean velocity of flow of fluid in the pipe.

Applying Bernoulli's equation between the two sections 1 and 2,

$$\frac{p_1}{w} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{v_2^2}{2g} + z_2 + h_f$$

Where h_f is the loss of head due to resistance to flow between the sections 1 and 2. Since $V_1 = V_2 = V$ and $Z_1 = Z_2$,

Loss of head = $h_f = (p_1/w - p_2/w)$

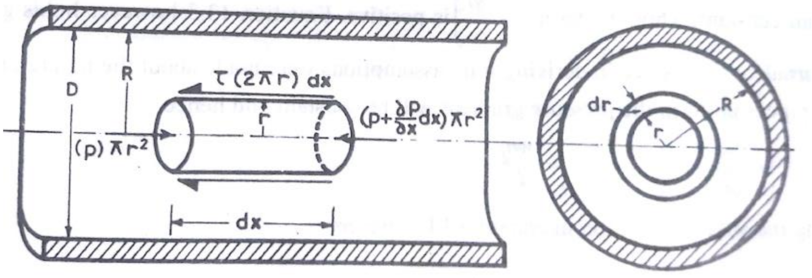


Fig 5.2 Laminar flow through a circular pipe

i.e., pressure gradient must exist in the direction of flow in order to overcome the resistance to flow and the loss of head is proportional to pressure drop (p_1-p_2) occurring in the length L of the pipe.

A small concentric cylindrical fluid element of radius r and length dx is chosen as free body as shown in fig. 5.2. Since the flow is steady and the size of the cross section does not change every particle of fluid moves without acceleration. Therefore the summation of forces on the free body in the direction of motion must be equal to zero. The forces acting on the fluid element in the direction of motion are normal pressure forces over the end areas and shear forces over the curved surface of the cylinder. If p is the pressure intensity on the left face then as indicated above since a pressure gradient must exist in the direction of flow in order to overcome the shear resistance, the pressure intensity on the right face will be (p+$\left(\frac{\partial p}{\partial x}\right) dx$).The shear stress τ on the periphery of the cylinder will be acting in the direction opposite to that of the flow of fluid and on account of symmetry its magnitude will be constant on whole of the periphery. The total pressure forces on the left and the right faces of the cylinder are (p)πr^2 and (p+$\left(\frac{\partial p}{\partial x}\right) dx$) πr^2 respectively and total shear force acting on the periphery of the cylindrical element is $\tau(2\pi r)dx$. Since the summation of all these forces in the x-direction must equal zero.

$$(p)\pi r^2 - \left(p+\left(\frac{\partial p}{\partial x}\right) dx\right)\pi r^2 - \tau(2\pi r)dx = 0$$

Or $$\left(-\left(\frac{\partial p}{\partial x}\right) dx\right) \pi r^2 - \tau(2\pi r)dx = 0$$

Dividing the above equation by the volume of the element (πr^2)dx and further simplifying it, we get

$$\tau = \frac{\partial p}{\partial x}\frac{r}{2} \qquad \text{....(5.3)}$$

Equation 5.3 shows that the shear stress τ varies linearly along the radius of the pipe (Fig.5.3) . At the centre of the pipe since r=0, the shear stress τ is zero and at the pipe wall, since r=R the maximum value of stress τ_0 is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} \qquad \text{...(5.4)}$$

The negative sign for the pressure gradient $\left(\frac{\partial p}{\partial x}\right)$ indicates a decrease of fluid pressure in the direction of flow.The pressure must decrease in the direction of flow in a horizontal pipe because pressure force is the only means available to compensate for the resistance to the flow, the potential and kinetic energies remain constant. Thus the term $\left(-\frac{\partial p}{\partial x}\right)$ is positive. Equation 5.3 however holds good for both laminar and turbulent flow, since in deriving it no assumptions were made about the nature of the flow.

Since the pipe is uniform the pressure gradient will be constant and hence

$$\left(-\frac{\partial p}{\partial x}\right) = \left(\frac{p_1 - p_2}{L}\right) = \frac{Wh_f}{L}$$

Introducing the above equation in 13.3,we get

$$\tau = \frac{wh_f}{2L} r \qquad \text{..... (5.5)}$$

In laminar flow the shear stress τ is entirely due to viscous action and therefore it may be evaluated by using Newton's law of viscosity, according to which $\tau = \mu\left(\frac{\partial v}{\partial y}\right)$;in which v is the velocity of flow at a distance y from the pipe wall.Since in the present case $y = (R - r)$, it follows that $dy = -dr$ and the expression for τ becomes

$$\tau = -\mu\frac{dv}{dr} \qquad \text{.....(5.6)}$$

In which the negative sign is on account of particular choice of coordinates and predicts mathematically that velocity v decreases as the radius r increases .

Substituting the value of τ from the equation 5.6 in equation 5.3,we have

$$-\mu\frac{\partial v}{\partial r} = -\frac{\partial p}{\partial x}\frac{r}{2}$$

or

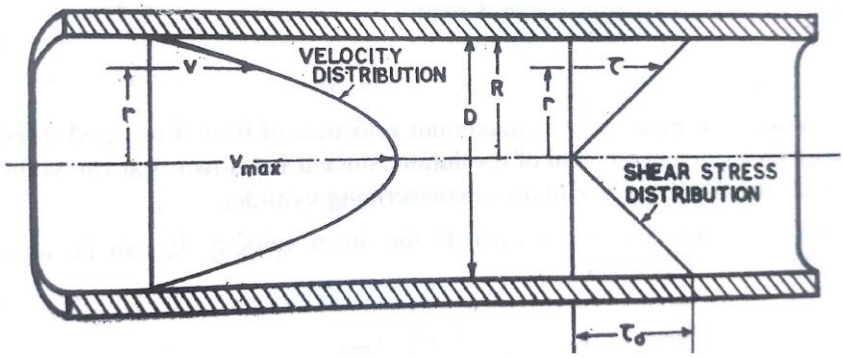
$$\frac{\partial v}{\partial r} = \frac{1}{\mu}\frac{\partial p}{\partial x}\frac{r}{2} \qquad \text{.....(5.7)}$$

For the steady uniform flow since the drop in the pressure depends only in the distance x and independent of r,the pressure gradient$(\partial p|\partial x)$ in the direction of flow must have a constant value. The integration of equation 5.7with respect to r then yields the expression for the velocity distribution for lminar flow in a circular pipes as

$$v = \frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)r^2 + C \qquad \text{.....(5.8)}$$

The constant of integration C can be evaluated from the boundary conditions that at $r = R$,the velocity $v = 0$.

$C= -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)R^2$



Equation 5.3 becomes

$$v = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2 - r^2) \qquad \text{. ...(5.9)}$$

Fig 5.3 Velocity and shear stress distribution for laminar flow in circular pipe

Equation 5.9 shows that in laminar flow through circular pipes, velocity of flow varies parabolically and the surface of velocity distribution is a paraboloid of revolution as shown in the fig. 5.3. The maximum velocity v_{max} occurs at the axis of the pipe and has the magnitude of

$$v = v_{max}[1 - (r^2|R^2)] \qquad \text{...5.10}$$

The discharge Q passing through any cross section of circular pipe can be obtained by integrating a small discharge passing through an elementary ring of thickness dr, considered at a radial distance r, out of the cross sectional area of the pipe, as shown in the fig 5.2.

Since $\qquad dQ = v(2\pi r)dr$

$$= \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2 - r^2)(2\pi r)dr \qquad \text{....5.11}$$

By integrating on both sides, we get

$$Q = \frac{\pi}{2\mu}\left(-\frac{\partial p}{\partial x}\right)\int_0^R (R^2 - r^2)r dr$$

$$Q = \frac{\pi}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^4 \qquad \text{....5.12}$$

If D is the diameter of the pipe, then equation 13.12 expressed in terms of D becomes

$$Q = \frac{\pi}{128\mu}\left(-\frac{\partial p}{\partial x}\right)D^4 \qquad \text{....5.13}$$

The mean velocity of flow V which is equal to (Q/A) or ($Q/\pi R^2$), is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^2 = \frac{1}{32\mu}\left(-\frac{\partial p}{\partial x}\right)D^2 \qquad \text{....5.14}$$

Comparison of equations 13.10 and13.14 reveals that

$$V = \frac{1}{2} v_{max} \qquad \ldots 5.15$$

The relation between the mean and the maximum velocities of flow, indicated in equation 13.15, also follows from the geometry of a paraboloid of revolution, since it is known that the volume of a paraboloid of revolution is equal to half the volume of the circumscribing cylinder.

The point where the local velocity is equal to the mean velocity V, can be located by combining equations 5.1 and 5.15. Thus

$$V = v_{max}\left(1 - \left(\frac{r^2}{R}\right)\right) = V = \frac{1}{2} v_{max}$$

or $$r = \frac{1}{\sqrt{2}} R = 0.707R \qquad \ldots 5.16$$

That is the mean velocity of flow V occurs at a radial distance of (0.707R) from the centre of the pipe. Rearranging equation 5.14 to solve for pressure drop, we have

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{32\mu V}{D^2}$$

Or $$\frac{P_1 - P_2}{L} = \frac{32\mu V}{D^2}$$

Or $$(P_1 - P_2) = \frac{32\mu VL}{D^2} = \frac{128\mu QL}{\pi D^4} \qquad \ldots 5.17$$

Equation 5.17 is known as Hagen-poiseuille equation for laminar flow in the circular pipes. It was first determined experimentally by a German Engineer G.H.L. Hagen in 1839 and almost simultaneously but independently by J.L.M. poiseuille, a French physician in 1840. The striking feature of Hagen-poiseuille equation is that it involves no empirically coefficient or experimental factor of any kind, except for the physical properties such as viscosity and specific weight of the flowing fluid.

Further if h_f represents the drop in pressure head, then

$$h_f = \frac{P_1 - P_2}{w} = \frac{32\mu NL}{wD^2}$$

Or $$h_f = \frac{64}{R_e}\left(\frac{LV^2}{2gd}\right) = \left(\frac{128\mu QL}{w\pi D^4}\right) \qquad \ldots 5.18$$

Where Reynolds number $R_e = \left(\rho DV/\mu\right)$ and w is the specific weight of the flowing fluid. It may again be noted that the drop in pressure head h_f is nothing but the head loss over length L of the pipe, which varies directly as the power of the velocity of flow of fluid and inversely as the square of the diameter of the pipe.

Several investigators have verified the validity of the Hagen-poiseuille equation, and hence it is often employed for experimental determination of the fluid viscosity. From 5.17, we have

$$\mu = \frac{(P_1 - P_2)D^2}{32VL} = \frac{(P_1 - P_2)D^4\pi}{128QL} \qquad \ldots 5.19$$

The pressure drop (p_1 - p_2) in a fixed length L of a pipe of diameter D for any given rate of laminar flow can be measured. Thus all the quantities on the right hand side of equation 5.19 will be known, from which the valve of the coefficient of viscosity μ of the fluid can be obtained.

The loss of head due to frictional resistance in a long straight pipe of length L and diameter D may also be expressed by Darcy – Weisbach equation as

$$h_f = \frac{P_1 - P_2}{w} = f\frac{L}{D}\frac{V^2}{2g} \qquad \ldots 5.20$$

Where f is friction factor and V is mean velocity of flow. Equating the two values of h_f given by equation 5.18 and 5.20, obtain

$$\frac{flV^2}{2gD} = \frac{32\mu VL}{wD^2}$$

or

$$f = \frac{64\mu}{\rho VD} = \frac{64}{Re} \qquad \ldots 5.21$$

LAMINAR FLOW BETWEEN PARALLEL PLATES-BOTH ARE AT REST

Consider laminar flow of fluid between to fixed parallel flat plates located at a distance B apart, as shown in the figure 5.6. A small rectangular element of the fluid of length δx is considered as free body as shown in fig.5 .6. let the lower face of the element be at a distance y from the lower plate and here let the velocity be v. At the upper face of the element which is at a distance of $(y + \delta y)$ from the lower plate, let the velocity be $(v + \delta v)$.

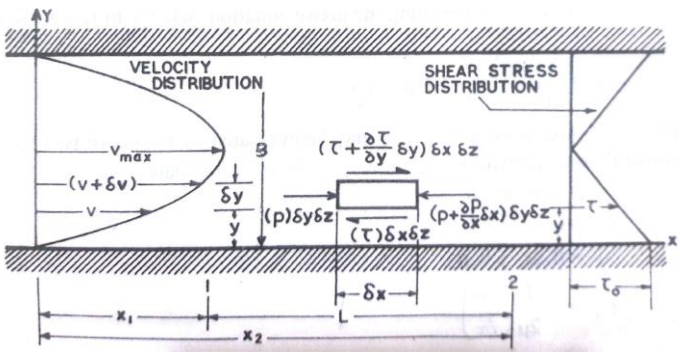


Fig 5.6 Laminar flow between two fixed parallel flat plates

If δv is positive, the faster moving fluid just above the upper face of the element exerts a forward force on the upper face. Similarly, the slower moving fluid adjacent to the lower face to retard its motion.

$i.e.,$ it exerts a backward force on the lower face. Thus there are shear stresses of magnitude τ on the lower face and $\left(\tau + \frac{\partial \tau}{\partial x}\delta y\right)$ on the upper face of the element in the directions as shown in the fig.13.6. in order to balance the shearing forces in the fluid a pressure gradient in the direction of the flow must be maintained .thus if p is the pressure intensity at the left face of the element, then $\left(p + \frac{\partial p}{\partial x}\delta x\right)$ will be the pressure intensity on the right face of the element.

If the width of the element in the direction perpendicular to the paper is δz , the total force acting on the element towards the right is

$$\left[p - \left(p + \frac{\partial p}{\partial x}\delta x\right)\right]\partial y \partial z + \left[\left(\tau + \frac{\partial \tau}{\partial x}\delta y\right) - \tau\right]\partial x \partial z$$

But for steady and uniform flow, there is no acceleration and hence this total force must be equal to zero.

$$-\frac{\partial p}{\partial x}\delta x \delta y \delta z + \frac{\partial \tau}{\partial x}\delta x \delta y \delta z = 0$$

Dividing by the volume of the element$(\delta x\ \delta y\ \delta z)$, we get

$$\frac{\partial p}{\partial x} = \frac{\partial \tau}{\partial x}$$

Which is same as the equation 13.1 derived above. Again according to newton's law of viscosity for laminar flow the shear stress is $\tau = \mu \frac{\partial v}{\partial x}$. Hence by substitution in the above equation the following differential equation for laminar flow is obtained, which is same as equation 13.2.

$$\frac{\partial p}{\partial x} = \mu \frac{\partial^2 v}{\partial y^2}$$

Since $\left(\frac{\partial p}{\partial x}\right)$ is independent of y, integrating the above equation twice with respect to y gives

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + c_1 y + c_2 \qquad \text{.....13.34}$$

The two constants of integration and may be evaluated by means of two boundary conditions. There being no slip of fluid at two solid boundary surfaces of the plates,

$$v = 0\ at\ y = 0$$

$$\therefore \quad c_2 = 0$$

$$v = 0\ at\ y = B$$

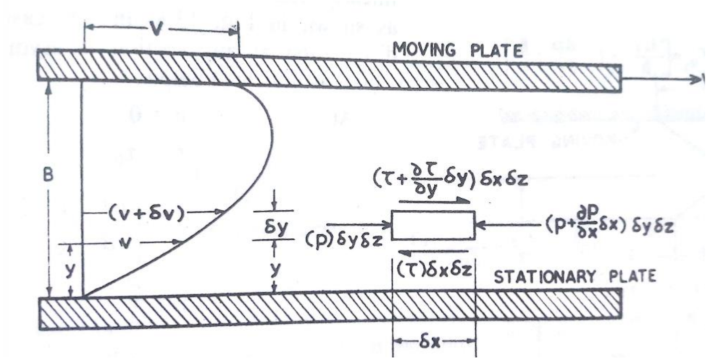
$$\therefore \quad c_1 = -\frac{B}{2\mu}\left(\frac{\partial P}{\partial x}\right)$$

Introducing these values in equation 13.34. the following for the velocity distribution is obtained

$$v = \frac{1}{2\pi}\left(\frac{\partial p}{\partial x}\right)(By - y^2) \qquad \text{.....13.35}$$

$v = V \, at \, y = B$

$$C_1 = \frac{V}{B} - \frac{1}{2\mu}\left(\frac{\partial P}{\partial X}\right) B$$



Hence substituting the values of C_1 and C_2 in equation 13.34, it yields the following equation for the velocity distribution for Couette flow

$$v = \frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)$$

It is thus observed that the velocity distribution in Couette flow depends on both V and $\left(\frac{\partial p}{\partial x}\right)$.However, the pressure gradient $\left(\frac{\partial p}{\partial x}\right)$ in this case may be either positive or negative.

In particular case when $\left(\frac{\partial p}{\partial x}\right)$ equals zero, there is no pressure gradient in the direction of flow, then, we have

$$v = V\frac{y}{B}$$

Which indicates that the velocity distribution is linear. This particular case is known as simple(or plain)Couette flow or simple shear flow.Now, if V equals zero,equation 13.44 reduces to equation 13.35. Thus it may be stated that the general case of Couette flow is a superposition of the simple Couette flow over a laminar flow between two fixed parallel flat plates.

The distribution of shear stress across any section for the case of Couette flow may also be determined by substituting equation 13.44 into Newton's law of viscosity. Thus

$$\tau = \mu\frac{\partial v}{\partial y}$$

Or $\tau = \mu\frac{\partial}{\partial y}\left[\frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)\right]$

Or $\tau = \mu\frac{V}{B} + \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right)$

Evidently in this case also the shear stress varies linearly with the distance from the boundary. However, as shown in Fig 13.8, in this case the shear stress distribution at any section is asymmetrical with the values as indicated below.

At y=0

$\tau = \tau_{01}$

$= \left[\frac{\mu V}{B} + \left(-\frac{\partial P}{\partial X}\right)\frac{B}{2}\right]$

At $Y=\frac{B}{2}$

$\tau = \frac{\mu V}{B}$

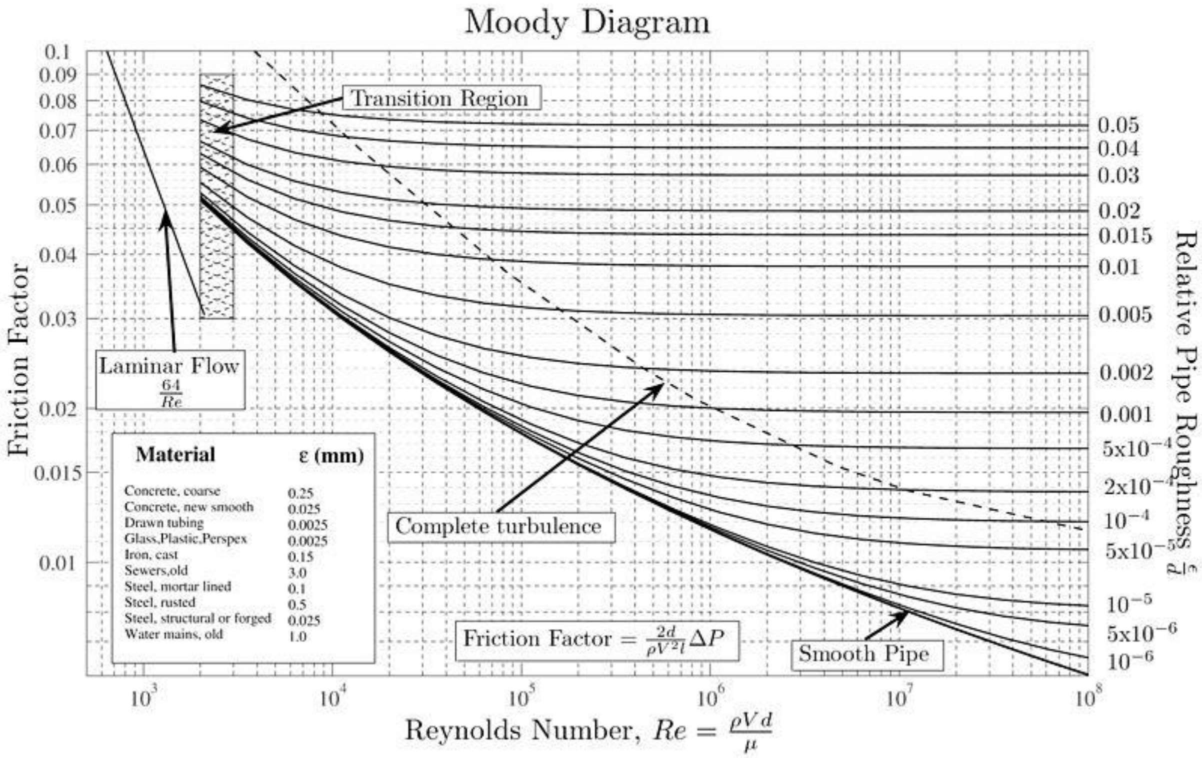
At $y = \left[\frac{B}{2} + \frac{\mu V}{B}\left(-\frac{1}{\partial P/\partial X}\right)\right]$

$\tau = 0$

At y=B

$\tau = \tau_{02} = \left[\frac{\mu V}{B} - \left(\frac{\partial P}{\partial X}\right)\frac{B}{2}\right]$

Moody Diagram



Fluid Mechanics

Unit –IV

Closed Conduit Flow and measurement of flow

Learning Objectives:

To introduce the laws of fluid friction and measurement of flow

Syllabus:

Closed Conduit Flow

Reynolds experiment, laws of fluid friction Darcy weishbach equation, minor losses in a pipes , The Hydraulic Grade Lin and *Energy Line, pipes in series and pipes in parallel ,siphon*

Measurement of flow: Pitot tube, venture-meter, orifice-meter, flow nozzles, turbine flow meter , Notches and weirs

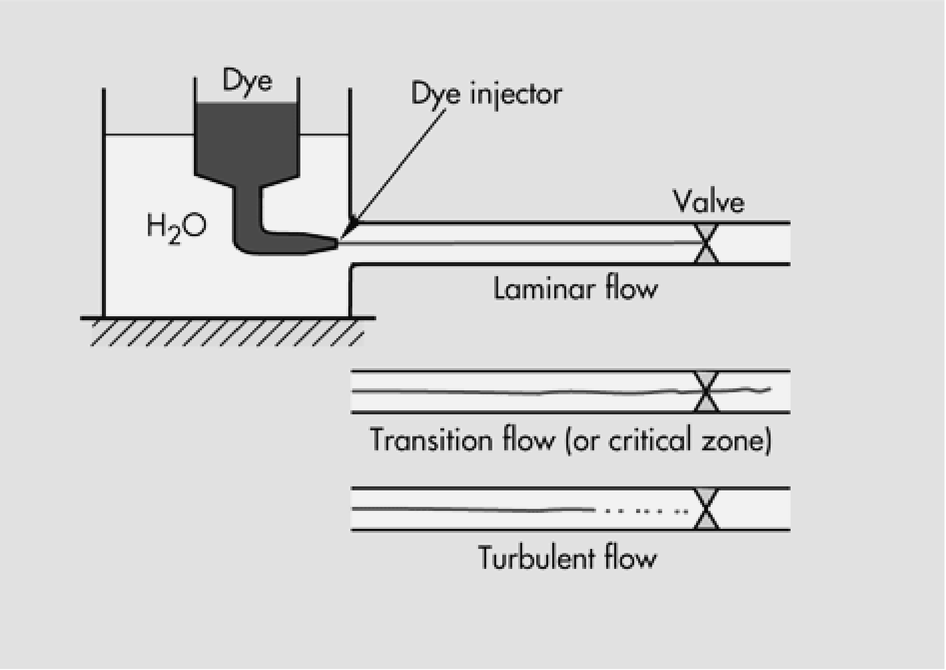
Learning Outcomes: At the end of the unit, the student will be able to

- Understand the different major losses and minor losses in pipes flow.
- Understand the different flow measurement devices.
- Analyze the various losses occurring when the fluid flowing in a pipe flow

Reynolds' Experiment

The Reynolds number is the ratio of inertial forces to viscous forces within a fluid which is subject to relative internal movement due to different fluid velocities.

The **Reynolds number** (**Re**) is an important dimensionless quantity in fluid mechanics that is used to help predict flow patterns in different fluid flow situations. It is widely used in many applications ranging from liquid flow in a pipe to the passage of air over an aircraft.



Apparatus :

1. A Constant head tank filled with water
2. A small tank containing dye
3. A horizontal glass tube provided with a bell mouthed entrance.

Laminar flow	: $Re < 2000$
Transitional flow	: $2000 < Re < 4000$
Turbulent flow	: $Re > 4000$

Procedure:

- The water was made to flow through the glass tube into the atmosphere and the velocity of flow was varied by adjusting valve.
- The liquid dye was introduced into the flow at the bell mouth through a small tube.

- We will be looking here at the flow of real fluid in pipes – ***real*** meaning a fluid that possesses viscosity hence looses energy due to friction as fluid particles interact with one another and the pipe wall.
- shear stress induced in a fluid flowing near a boundary is given by Newton's law of viscosity:

$$\zeta \alpha \mu \frac{du}{dy}$$

This tells us that the shear stress, τ, in a fluid is proportional to the velocity gradient –the rate of change of velocity across the fluid path.

For a "Newtonian" fluid we can write: $\zeta = \mu \frac{du}{dy}$

flow can be classified into one of two types, **laminar** or **turbulent** flow

The non-dimensional number, the Reynolds number, Re, is used to determine which type of flow occurs:

$$\text{Re} = \frac{\rho v D}{\mu}$$

Laminar flow	: Re < 2000
Transitional flow	: 2000 < Re < 4000
Turbulent flow	: Re > 4000

Loss of energy

When water flows in a pipe , it experiences some resistance to its motion ,due to which its velocity and ultimately the head of water available is reduced.

Energy losses in pipe flow

When a fluid is flowing through a pipe, the fluid experiences some resistance due to which some of the energy of the fluid is lost

Energy losses	
Major energy loss (due to friction)	**Minor energy losses** a. Head loss due to Sudden expansion b. Head loss due to Sudden contraction c. Head loss due to an abstraction in the pipe d. Head loss due to the entrance to a pipe. e. Head loss due to the entrance to a pipe. f. Head loss due to the exit of the pipe. g. Head loss due to Bend in pipe h. Head loss due to various pipe fittings

Darcy weisbach Equation

Consider a uniform horizontal pipe , having steady flow

$P_{1=}$ Pressure intensity at section 1-1

V_1= Velocity of flow at section 1-1,

L= Length of the pipe between section 1-1, and 2-2,

d= diameter of pipe,

f'= frictional resistance per unit wetted area per unit velocity,

h_f = loss of head due to friction

P_2 ,V_2 = are values of pressure intensity and velocity section 2-2

$$\frac{p_1}{\rho g}+\frac{v_1^{\,2}}{2g}+z_1=\frac{p_2}{\rho g}+\frac{v_2^{\,2}}{2g}+z_2+h_f$$

$$z_1=z_2(horizontal-pipe)$$

$$and$$

$$v_1=v_2(\text{diameter.}-\text{uniform})$$

$$\frac{p_1}{\rho g}=\frac{P_2}{\rho g}+h_f$$

$$h_f=\frac{p_1}{\rho g}-\frac{P_2}{\rho g}$$

Frictional resistance =(Frictional resistance per unit wetted area unit velocity x wetted area x velocity^2)

$F_1 = f' \times \Pi dL \times V^2$

$= f' \times \mathrm{P} \times \mathrm{L} \times V^2$

The pressure force acting between sections 1-1 and 2-2

At section 1-1 = $P_1 A$

At section 2-2 = $P_2 A$

Frictional force acting against the direction of force =F1

Resolving horizontal forces =

$P_1 A - P_2 A - F_1 = 0$

$(P_1 - P_2)\,\mathrm{A} = F_1$

$(P_1 - P_2)\,\mathrm{A} = f' \times \Pi dL \times V^2$

$(P_1 - P_2)\,\mathrm{A} = f' \times \mathrm{P} \times \mathrm{L} \times V^2$

$$(P_1 - P_2) = \frac{f' \times \mathrm{P} \times \mathrm{L} \times V^2}{A}$$

$P_1 - P_2 = \rho g h_f$

$$\rho g h_f = \frac{f' \times \mathrm{P} \times \mathrm{L} \times V^2}{A} \left(\because \frac{P}{A} = \frac{4}{d} \right)$$

$$h_f = \frac{4f}{2g} \frac{\mathrm{L} \times V^2}{d}$$

$$h_f = \frac{f'}{\rho g} \times \frac{4LV^2}{d} \qquad \left(\because \frac{f'}{\rho} = \frac{f}{2} \right)$$

f=coefficient of friction $h_f = \dfrac{4f \times \mathrm{L} \times V^2}{2gd}$

f is known as friction factor

Loss of energy due to friction

Darcy-Weishbach equation: Head loss due to friction,

L: length of the pipe

V: mean velocity of the flow

d: diameter of the pipe

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

f is the friction factor for fully developed laminar flow:

$$f = \frac{\mathrm{R_e}}{64}(\mathrm{Re} < 2000)$$

$$\mathrm{Re} = \frac{\rho v D}{\mu}$$

C_f is the skin friction coefficient or Fanning's friction factor.

Minor Losses

$$1.\textit{Head loss due to Sudden expansion}\left(h_e = \frac{(v_1 - v_2)^2}{2g} \right)$$

$$2.\textit{Head loss due to Sudden contraction}\left(h_c = \frac{v_2{}^2}{2g}\left(\frac{1}{C_c} - 1 \right)^2 \right)$$

$$3.\textit{Head loss due to an abstraction in the pipe}\left(h_{abs} = \left(\frac{A}{C_c(A-a)} - 1 \right)^2 \frac{v^2}{2g} \right)$$

$$4.\textit{Head loss due to the entrance to a pipe}\left(h_f = 0.5\frac{v^2}{2g} \right)$$

$$5.\textit{Head loss due to the exit of the pipe}\left(h_o = \frac{v^2}{2g} \right)$$

$$6.\textit{Head loss due to Bend in pipe}\left(h_b = k\frac{v^2}{2g} \right)$$

$$7.\textit{Head loss due to various pipe fittings}\left(h_{fiiting} = k\frac{v^2}{2g} \right)$$

The Hydraulic Grade Line

The Hydraulic Grade Line is a line representing the total head available to the fluid - minus the velocity head and can be expressed as:

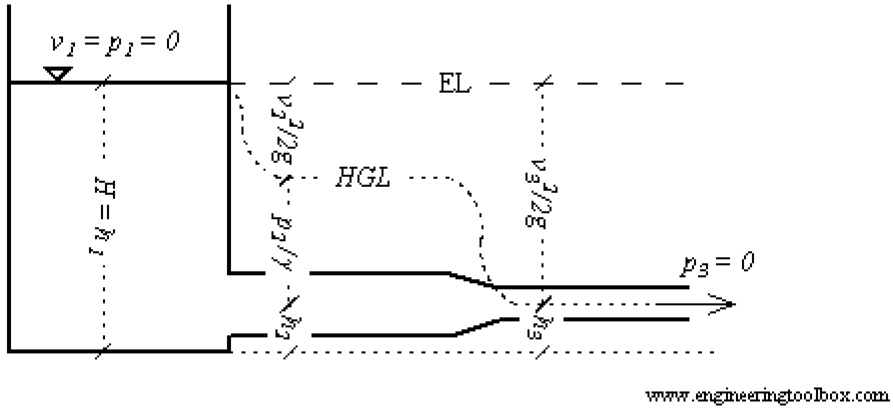
$$HGL = \frac{p}{\gamma} + h$$

HGL = Hydraulic Grade Line (m)

The Energy Line

The Energy Line is a line that represent the total head available to the fluid and can be expressed as:

$$TEL = H = \frac{p}{\gamma} + \frac{v^2}{2g} + h = constant\ along\ a\ streamline$$

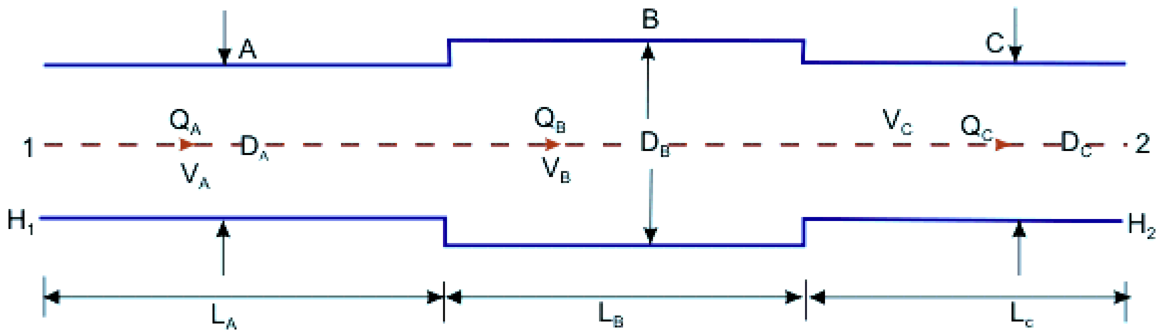


Flow Through Branched Pipes

In several practical situations, flow takes place under a given head through different pipes jointed together either in series or in parallel or in a combination of both of them.

Pipes in Series

- If a pipeline is joined to one or more pipelines in continuation, these are said to constitute pipes in series. A typical example of pipes in series is shown in Fig. 36.1. Here three pipes *A*, *B* and *C* are joined in series.



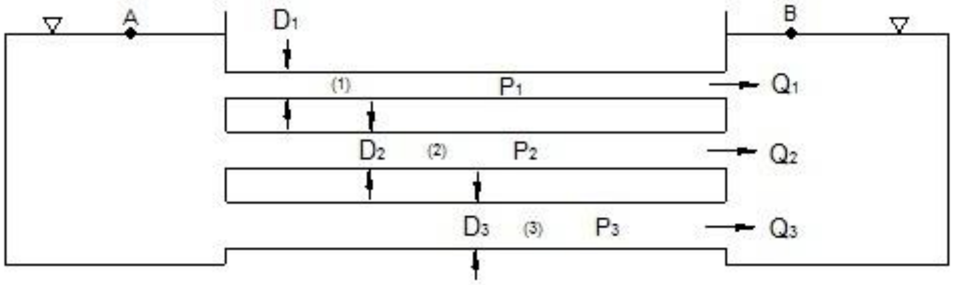
In this case, rate of flow *Q* remains same in each pipe. Hence,

$Q_A = Q_B = Q_C = Q$

Pipes in Parallel

Pipes that are in parallel will experience the same pressure loss across each pipe. In addition, each pipe will experience a different flow rate of fluid going through it. Refer to equation 1 to calculate the different flow rates in each pipe when the same type of fluid is flowing through each pipe.

$$Q = Q_1 + Q_2 \cdots Q_n$$



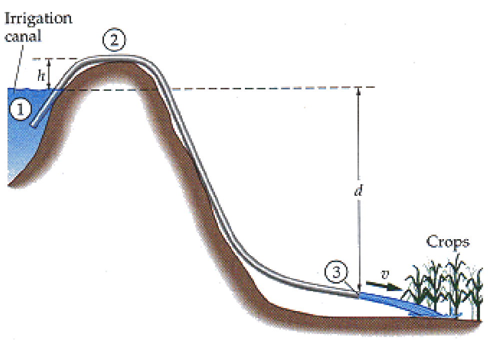
Siphon

The word **siphon** is used to refer to a wide variety of devices that involve the flow of liquids through tubes

Siphon tubes are a basic implement used in irrigation to transfer water over a barrier (such as the bank of a raised irrigation canal), using the siphon principle

Operation

- The simplest siphon tubes are operated by simply filling the tube with water (by immersion in the canal, or other means), keeping one end in the canal and with the other end sealed, placing it in the area to be irrigated. The seal can then be removed and the water will siphon transferring the water from the submerged higher end to the lower end.
- The tubes of up to 150mm diameter and several meters long, this is all easily undertaken by one person without any other tools. An effective seal is produced using a hand - with a rapid enough action there will be sufficient water in the tube to start the siphon effect.
- Larger capacity siphon tubes may have mechanical means of sealing the ends, and allow for a pump to be used to prime the tube.



Use

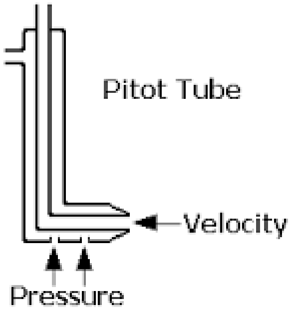
- They can be used to either flood, or drain, as required.

Benefits

- Inexpensive,
- It do not require any engineering to put into place (unlike say a sluice gate embedded in the bank of the canal).
- Easily relocated from one bay or field to another as the required water level is reached.

Pitot tube :

A **pitot tube** is a pressure measurement instrument used to measure fluid flow velocity.



Theory of operation

- The basic pitot tube consists of a tube pointing directly into the fluid flow. As this tube contains fluid, a pressure can be measured; the moving fluid is brought to rest (stagnates) as there is no outlet to allow flow to continue.
- This pressure is the stagnation pressure of the fluid, also known as the total pressure or (particularly in aviation) the pitot pressure.
- The measured stagnation pressure cannot itself be used to determine the fluid flow velocity (airspeed in aviation). However, Bernoulli's equation states:

stagnation pressure = static pressure + dynamic pressure

$$p_t = p_s + \left(\frac{\rho u^2}{2}\right)$$

Solving that for flow velocity:

$u = \sqrt{\dfrac{2(p_t - p_s)}{\rho}}$ (it can be treated as incompressible)

$u = flow\ velocity\ to\ be\ measured\ \ in\ m/s;$

$P_s = stagnation\ or\ total\ pressure\ in\ pascals$

$P_t = static\ pressure\ in\ pascals;$

$\rho = density - in - kg/m^3 kg/m^3.$

Venturimeter Principle

A **venturimeter** is a device used for measuring the rate of a flow of a fluid flowing through a pipe.

It consists of three parts.

- A short converging part
- Throat
- Diverging part

- It is based on the principle of Bernoulli's equation.
- Inside of the venturimeter pressure difference is created by reducing the cross-sectional area of the flow passage.
- The pressure difference is measured by using a differential U-tube manometer.
- This pressure difference helps in the determination of rate of flow of fluid or discharge through the pipe line. As the inlet area of the venturi is large than at the throat, the velocity at the throat increases resulting in decrease of pressure. By this, a pressure difference is created between the inlet and the throat of the venturi.

Let

D1 and D2 — Diameter at inlet and throat respectively

P1and P2 — Pressure at inlet and throat

V1 and V2 — Velocity at inlet ad throat

A1 and A2 – Area of cross section of inlet and throat

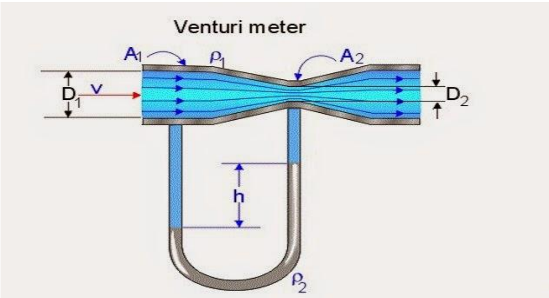
By applying Bernoulli's equation

$$\frac{P_1}{\rho g} + \frac{v^2{}_1}{2g} + Z_1 = \frac{P_2}{\rho g} + \frac{v^2{}_2}{2g} + Z_2$$

As pipe is horizontal z1=z2

$$\frac{P_1}{\rho g} + \frac{v^2{}_1}{2g} = \frac{P_2}{\rho g} + \frac{v^2{}_2}{2g}$$

$$\frac{P_1 - P_2}{\rho g} = \frac{v^2{}_2}{2g} - \frac{v^2{}_1}{2g}$$



$$h = \frac{P_1 - P_2}{\rho g}$$

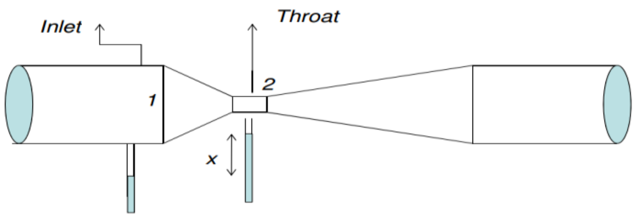
From the continuity equation at sections 1 and 2, we obtain

$$A_1 v_1 = A_2 v_2$$

$$h = \frac{V_2^2}{2g}\left(\frac{A_1^2 - A_2^2}{A_1^2}\right)$$

$$Q = A_1 \ v_1 = A_2 \ v_2$$

$$v_2 = \frac{A_1}{\sqrt{A_1^2 - A_2^2}}\sqrt{2gh}$$



$$Q = A_1 \ v_1 = A_2 \ v_2$$

$$Q_{actual} = \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}}\sqrt{2gh}$$

Orifice meter

- it is a device used for measuring the rate of flow of a fluid flowing through a pipe.
- It is a cheaper device as compared to venturimeter.
- This also works on the same principle as that of venturimeter.
- It consists of flat circular plate which has a circular hole, in concentric with the pipe. This is called orifice.
- The diameter of orifice is generally 0.5 times the diameter of the pipe (D), although it may vary from 0.4 to 0.8 times the pipe diameter.

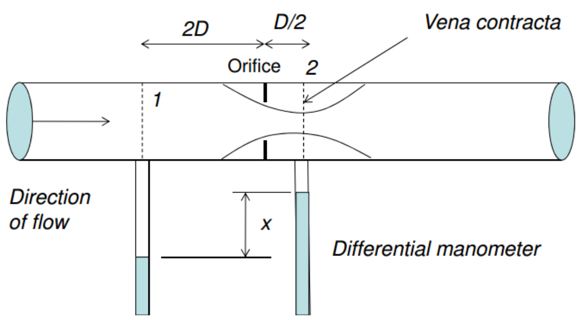
Let

D1 and D2 — Diameter at inlet and throat respectively

P1and P2 — Pressure at inlet and throat

V1 and V2 — Velocity at inlet ad throat

A1 and A2 – Area of cross section of inlet and throat



<u>By applying Bernoulli's equation</u>

$$\frac{P_1}{\rho g} + \frac{v^2_1}{2g} + Z_1 = \frac{P_2}{\rho g} + \frac{v^2_2}{2g} + Z_2$$

As pipe is horizontal z1=z2

$$\frac{P_1}{\rho g} + \frac{v^2_1}{2g} = \frac{P_2}{\rho g} + \frac{v^2_2}{2g}$$

$$\frac{P_1 - P_2}{\rho g} = \frac{v^2{}_2}{2g} - \frac{v^2{}_1}{2g}$$

$$\left(\frac{P_1}{\rho g} + Z_1\right) - \left(\frac{P_2}{\rho g} + Z_2\right) = \left(\frac{v^2{}_2}{2g} - \frac{v^2{}_1}{2g}\right)$$

$$h = \left(\frac{v^2{}_2}{2g} - \frac{v^2{}_1}{2g}\right)$$

$$v_2 = \sqrt{2gh + v_1{}^2}$$

Let A_0 is the area of the orifice.

Coefficient of contraction, $C_c = \frac{A_2}{A_0}$

By continuity equation, we have

$$Q = A_1 \ v_1 = A_2 \ \mathrm{v}_2$$

$$v_1 = \frac{A_0 C_c}{A_1} V_2$$

$$v_2 = \sqrt{2gh + \frac{A_0{}^2 C_c{}^2 V_2{}^2}{A_1^2}}$$

$$v_2 = \frac{\sqrt{2gh}}{\sqrt{1 - \frac{A_0{}^2 C_c{}^2}{A_1^2}}}$$

Discharge,

$$Q = A_2 \ \mathrm{v}_2 = C_c A_0 \mathrm{v}_2$$

$$Q = \frac{C_c A_0 \sqrt{2gh}}{\sqrt{1 - \frac{A_0{}^2 C_c{}^2}{A_1^2}}}$$

C_d is the co-efficient of discharge for orifice meter, which is defined as

$$C_c = C_d \frac{\sqrt{1 - \frac{A_0{}^2}{A_1^2}}}{\sqrt{1 - \frac{A_0{}^2 C_c{}^2}{A_1^2}}}$$

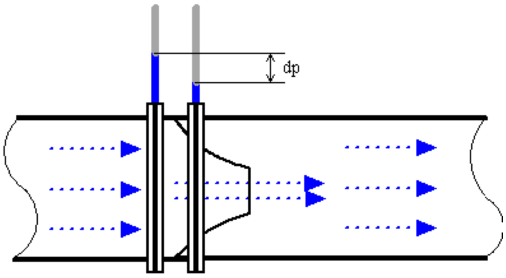
$$C_d = C_c \frac{\sqrt{1 - \frac{A_0^2 C_c^2}{A_1^2}}}{\sqrt{1 - \frac{A_0^2}{A_1^2}}}$$

$$Q = C_d \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}} \sqrt{2gh}$$

The coefficient of discharge of the orifice meter is much smaller than that of a venturimeter.

Flow nozzle

The flow nozzle is used for high velocity flow measurement where erosion or cavitation could wear or damage an orifice plate.



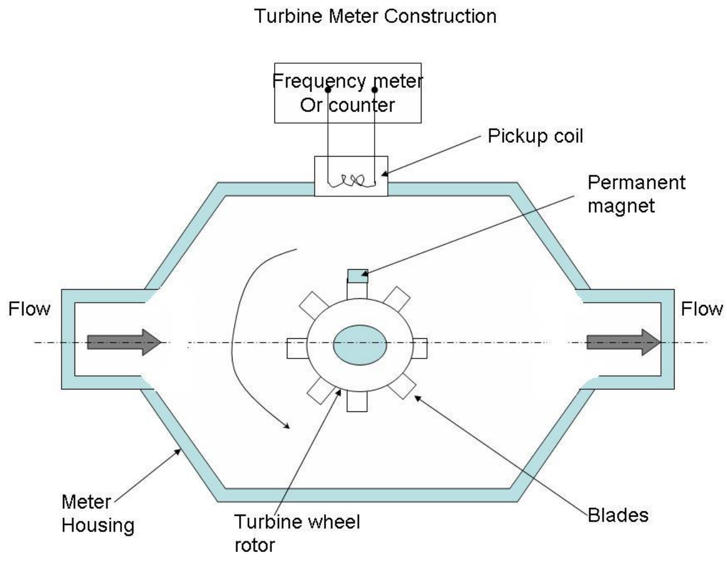
- The flow nozzle is recommended for both clean and dirty liquids
- The range ability is *4 to 1*
- The relative pressure loss is *medium*
- Typical accuracy is *1-2%* of full range
- Required upstream pipe length is *10 to 30* diameters
- The viscosity effect *high*
- The relative cost is *medium*

Discharge Coefficient - c_d				
Diameter Ratio $d = D_2 / D_1$	**Reynolds Number - *Re***			
	10^4	10^5	10^6	10^7
0.2	0.968	0.988	0.994	0.995
0.4	0.957	0.984	0.993	0.995
0.6	0.95	0.981	0.992	0.995
0.8	0.94	0.978	0.991	0.995

How Turbine Flowmeters Work

- Turbine flowmeters use the mechanical energy of the fluid to rotate a "pinwheel" (rotor) in the flow stream. Blades on the rotor are angled to transform energy from the flow stream into rotational energy. The rotor shaft spins on bearings.

- When the fluid moves faster, the rotor spins proportionally faster. Turbine flowmeters now constitute 7% of the world market. Shaft rotation can be sensed mechanically or by detecting the movement of the blades. Blade movement is often detected magnetically, with each blade or embedded piece of metal generating a pulse.
- Turbine flowmeter sensors are typically located external to the flowing stream to avoid material of construction constraints that would result if wetted sensors were used. When the fluid moves faster, more pulses are generated. The transmitter processes the pulse signal to determine the flow of the fluid. Transmitters and sensing systems are available to sense flow in both the forward and reverse flow directions



Notch

A notch may be defined as an opening in one side of a tank or a reservoir, like a large orifice, with the upstream liquid level below the top edge of the opening.

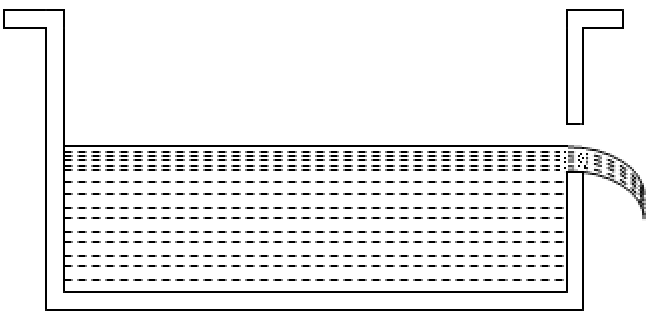


Fig : A Notch

Since the top edge of the notch above the liquid level serves no purpose, therefore a notch may have only the bottom edge and sides.

The bottom edge, over which the liquid flows, is known as **sill** or **crest** of the notch and the sheet of liquid flowing over a notch (or a weir) is known as **nappe** or **vein**.

A notch is, usually made of a metallic plate and is used to measure the discharge of liquids.

Weir

A weir is a barrier across a river designed to alter the flow characteristics. In most cases weirs take the form of a horizontal barrier across the width of a river that pools water behind it whilst still allowing it to flow steadily over the top

It is used to prevent flooding, measure discharge and help render rivers navigable. In some places the crest of an overflow spillway on a large dam may also be called a weir.

Ex:,

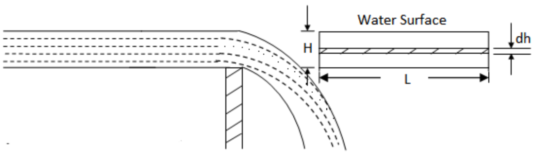


Fig : Rectangular weir

Dimensional Analysis and Model Similitude

Content:

- ✓ Introduction to Dimensional Analysis
- ✓ Dimensional homogeneity
- ✓ Methods of dimensional analysis
- ✓ Use of dimensional analysis in presenting experimental data
- ✓ Model investigation
- ✓ Similitude - types of similarities
- ✓ Dimensionless numbers and their significance

References:

1. Robert W. Fox, Alan T. McDonald, Philip J. Pritchard, Introduction to Fluid Mechanics, 8th Edition, John Wiley & Sons, Inc.,
2. Frank M. White, Fluid Mechanics, 7th Edition, Tata McGraw Hill.
3. Yunus A. Cengel, John M. Cimbala, Fluid Mechanics
4. S. K. Som, G. Biswas, Introduction to Fluid Mechanics and Fluid Machines, Revised Second Edition, Tata McGraw Hill Book Publishing Company Limited.

Introduction:

Experiments that result in large data sets, might be reduced to a single set of curves or even a single curve, when suitably non-dimensionalized. The technique for doing this is dimensional analysis. It is also effective in theoretical studies. Most practical fluid flow problems are too complex, both geometrically and physically, to be solved analytically. They must be tested by experiment or approximated by computational fluid dynamics (CFD). These results are typically reported as experimental or numerical data points and smoothed curves. Such data have much more generality if they are expressed in compact, economic form. This is the motivation for dimensional analysis.

Dimension: It is a measure of physical quantity without numerical values.

Primary dimension: These are the fundamental dimensions and are seven in number - mass, length, time, temperature, electric current, amount of light and amount of matter.

S. No	Dimension	Symbol	SI Unit
1	Mass	M	kg (kilogram)
2	Length	L	m (meter)
3	Time	T	s (second)
4	Temperature	θ	K (Kelvin)
5	Electric current	I	A (ampere)
6	Amount of light	C	cd (candela)
7	Amount of matter	N	mol (mole)

Derived dimensions: These are the dimensions expressed in terms of fundamental dimensions.

S. No	Parameter	Formula	Unit	Dimension
1	Area	A = L x L	m^2	L^2
2	Volume	V = L x L x L	m^3	L^3

3	Velocity	$v = {ds}/{dt}$	m/s	LT^{-1}
4	Acceleration	$a = {dv}/{dt}$	m/s^2	LT^{-2}
5	Force	F = ma	N (newton)	MLT^{-2}
6	Torque	T = F.s	N-m	ML^2T^{-2}
7	Power	P=Tω	N-m/s	ML^2T^{-3}
8	Pressure, Stress	$p = F/A$	N/m^2	$ML^{-1}T^{-2}$
9	Density	$\rho = m/V$	kg/m^3	ML^{-3}
10	Absolute viscosity	$\mu = \tau \left(du/dy \right)^{-1}$	$N\text{-}s/m^2$	$ML^{-1}T^{-1}$
10	Kinematic viscosity	$\gamma = \mu/\rho$	m^2/s	L^2T^{-1}

Dimensional Homogeneity: It imposes conditions on the quantities involved in a physical problem and plays a central role in dimensional analysis.

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

Example: Consider velocity of a freely falling body, $v = u + at$.

For initial velocity, (u) – dimension is LT^{-1}

For instantaneous velocity, (v) – dimension is LT^{-1}

For the term (at) – dimension is $[LT^{-2}][T] = LT^{-1}$

All the terms in the above equation are having same dimensions. Hence the equation is dimensionally correct and is said to be homogeneous in dimensions.

Dimensional variables: These are the dimensional quantities that change or vary in the problem.

Non-dimensional variables: These are the quantities that change or vary in the equation but have no dimensions.

Purposes of dimensional analysis

It is a method of determining the dimensionless parameters by a mathematical technique. It is helpful:

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

Advantages of Dimensional Analysis:

i. Increases the insight about the relationships between key variables.

ii. Reduces the number of parameters in the problem.

iii. Extrapolation to untested values of one or more of the dimensional parameters is possible.

Non-dimensionalization of equations:

The law of dimensional homogeneity guarantees that every additive term in an equation has the same dimensions. It follows that if we divide each term in the equation by a collection of variables and constants whose product has those same dimensions, the equation is rendered non-dimensional. If, in addition, the non-dimensional terms in the equation are of order unity, the equation is called normalized.

Example: ***Strain*** is a non-dimensional parameter defined as change in length per unit length.

$$\varepsilon = \Delta L / L .$$

Here strain has no units and dimensions.

There are two approaches in dimensional analysis: Buckingham-π theorem and Rayleigh method.

Buckingham-π theorem or Method of Repeating Variables:

"If there are 'n' variables in a dimensionally homogeneous equation and if these variables contain 'm' primary dimensions, then the variables can be grouped into (m-n) non-dimensional parameters".

Mathematically: Physical variables are related as $f\left(x_1, x_2, x_3, ...x_n\right) = 0$.

Where x_1, x_2, x_3, ….x_n are the dimensional physical quantities like velocity, density, pressure, area etc., pertinent to a particular physical phenomenon, then the same phenomenon can be described by (m-n) dimensionless Pi-terms as $\phi = \left(\pi_1, \pi_2, \pi_3, ...\pi_{m-n}\right)$

Where, m represents the fundamental dimensions such as mass, length, and time.

Out of the physical variables, one has to select any three variables which contain all the fundamental units of M, L, and T. These forms non-dimensional groups when a geometric property (such as length), a fluid property (such as density) and a flow characteristic (such as velocity) are chosen to represent such groups. These variables are known as repeated variables.

The major limitation of the method of repeating variables is that it cannot predict the exact mathematical form of the equation.

Example: Show by the use of Buckingham's Pi-theorem, that the velocity through an orifice is given by $v = \sqrt{2gH}\, f\left(\frac{D}{H}, \frac{\mu}{\rho v H}, \frac{\sigma}{\rho v^2 H}\right)$, where H is the head causing the flow, D is the diameter of the orifice, μ is the coefficient of viscosity, ρ is the density of the fluid, σ is the surface tension and g is the gravitational acceleration.

Solution: The functional relationship between physical variables is $f\left(D, H, v, g, \sigma, \rho, \mu\right) = 0$.

Number of physical variables, n = 7

Number of fundamental dimensions, m = 3

Number of Pi-terms needed = 4 ($\left(\pi_1, \pi_2, \pi_3, \pi_4\right)$

Consider, repeating variables as ρ, v, H.

First Pi-term: $\pi_1 = \rho^{a1} v^{b1} H^{c1} D$

Writing in dimensional form, we get $M^0 L^0 T^0 = \left[ML^{-3}\right]^{a1} \left[LT^{-1}\right]^{b1} \left[L\right]^{c1} \left[L\right]$

On simplification, $M^0 L^0 T^0 = \left[M\right]^{a1} \left[L\right]^{-3a1+b1+c1+1} \left[T\right]^{b1}$

Equating the exponents of M, L and T respectively, yields, $a1 = 0, b1 = 0, c1 = -1$.

Hence, $\pi_1 = H^{-1}D = \frac{D}{H}$

<u>Second Pi-term:</u> $\pi_2 = \rho^{a2}v^{b2}H^{c2}g$

Writing in dimensional form, we get $M^0L^0T^0 = \left[ML^{-3}\right]^{a2}\left[LT^{-1}\right]^{b2}\left[L\right]^{c2}\left[LT^{-2}\right]$

On simplification, $M^0L^0T^0 = \left[M\right]^{a2}\left[L\right]^{-3a2+b2+c2+1}\left[T\right]^{-b2-2}$

Equating the exponents of M, L and T respectively, yields, $a2 = 0, b2 = -2, c2 = 1$

Hence, $\pi_2 = v^{-2}Hg = \frac{gH}{v^2}$

<u>Third Pi-term:</u> $\pi_3 = \rho^{a3}v^{b3}H^{c3}\mu$

Writing in dimensional form, we get $M^0L^0T^0 = \left[ML^{-3}\right]^{a3}\left[LT^{-1}\right]^{b3}\left[L\right]^{c3}\left[ML^{-1}T^{-1}\right]$

On simplification, $M^0L^0T^0 = \left[M\right]^{a3+1}\left[L\right]^{-3a3+b3+c3-1}\left[T\right]^{-b3-1}$

Equating the exponents of M, L and T respectively, yields, $a3 = -1, b3 = -1, c3 = -1$

Hence, $\pi_3 = \rho^{-1}v^{-1}H^{-1}\mu = \frac{\mu}{\rho v H}$

<u>Fourth Pi-term:</u> $\pi_4 = \rho^{a4}v^{b4}H^{c4}\sigma$

Writing in dimensional form, we get $M^0L^0T^0 = \left[ML^{-3}\right]^{a4}\left[LT^{-1}\right]^{b4}\left[L\right]^{c4}\left[MT^{-2}\right]$

On simplification, $M^0L^0T^0 = \left[M\right]^{a4+1}\left[L\right]^{-3a4+b4+c4}\left[T\right]^{-b4-2}$

Equating the exponents of M, L and T respectively, yields, $a4 = -1, b4 = -2, c4 = -1$

Hence, $\pi_4 = \rho^{-1}v^{-2}H^{-1}\sigma = \frac{\sigma}{\rho v^2 H}$

The functional relationship can be written as $\phi\left(\pi_1, \pi_2, \pi_3, \pi_4\right) = 0$

or $\frac{v^2}{gH} = \phi\left(\frac{D}{H}, \frac{\mu}{\rho v H}, \frac{\sigma}{\rho v^2 H}\right) \Rightarrow \frac{v^2}{gH} = \phi\left(\frac{D}{H}, \frac{\mu}{\rho v H}, \frac{\sigma}{\rho v^2 H}\right)$

Rayleigh's method:

This method involves the following steps:

1. Collect all the independent variables which are likely to influence the value of the dependent variable.
2. Write the functional relationship, i.e. if the dependent variable 'y' is some function of the independent variables $x_1, x_2, x_3, x_4, ...$, then $y = f\left(x_1, x_2, x_3, x_4, ...\right)$.
3. Write the above equation in the form: $y = K\left(x_1^a, x_2^b, x_3^c, x_4^d, ...\right)$ where K is a dimensionless coefficient to be determined either from the physical characteristics of the problem or from experiments.
4. Express each of the quantities on both sides of the above equation in the fundamental units M, L, and T.
5. Utilize the principle of dimensional homogeneity to obtain a set of simultaneous equations involving the exponents a, b, c, …

6. Solve the simultaneous equations for a, b, c, etc., that helps to group the variables into recognized dimensionless parameters.
7. Substitute the value of exponents in the main equation, and form the non-dimensional parameters by grouping the variables with like exponents.

Example: Show that the resistance F to the motion of a sphere of diameter D moving with a uniform velocity C through a real fluid of density ρ and viscosity μ is given by: $F = \rho D^2 V^2 f\left(\frac{\mu}{VD\rho}\right)$.

Use this result to explain how dimensional analysis results in the simplification of experimental data.

Solution: Let the resistive force is assumed to have a functional relationship as $F = f\left(D, V, \rho, \mu\right)$.

It can be written as $F = K\left(D^a \times V^b \times \rho^c \times \mu^d\right)$, where K is a dimensionless coefficient.

Writing dimensions for all the terms in the above equation

$$\left[MLT^{-2}\right] = [1][L]^a\left[LT^{-1}\right]^b\left[\mathrm{ML^{-3}}\right]^c\left[\mathrm{ML^{-1}T^{-1}}\right]^d$$

Since K is a constant, its dimension is taken as unity.

$$\left[MLT^{-2}\right] = [L]^{a+b-3c-d}[T]^{-b-d}[\mathrm{M}]^{c+d}$$

For dimensional homogeneity, exponents of M, L, and T are equated on both sides.

Corresponding equations are:

For M: $c + d = 1 \Rightarrow c = 1 - d$

For T: $-b - d = -2 \Rightarrow b = 2 - d$

For L: $a + b - 3c - d = 1 \Rightarrow a = 2 - d$

Substituting the expressions for a, b, and c in the above equation gives

$$F = K\left(D^a \times V^b \times \rho^c \times \mu^d\right) = K\left(D^{2-d} \times V^{2-d} \times \rho^{1-d} \times \mu^d\right)$$

On simplification, we get

$$F = KD^2V^2\rho\left(\frac{\mu}{DV\rho}\right)^d \Rightarrow F = D^2V^2\rho f\left(\frac{\mu}{DV\rho}\right)$$

Principle of Similarity

In many cases in real-life engineering, the equations are either not known or too difficult to solve; oftentimes experimentation is the only method of obtaining reliable information. In most experiments, to save time and money, tests are performed on a geometrically scaled model, rather than on the full-scale prototype. This concept of Physical similarity is used:

- To apply the results taken from tests under one set of conditions to another set of conditions.
- To predict the influence of a large number of independent operating variables on the performance of a system from an experiment with a limited number of operating variables.

Prototype and Model:

Prototype is the full size structure employed in the actual engineering design. It operates under the actual working conditions.

Model is a small scale replica of the prototype, by which the characteristics of other similar systems can be studied. A model can be even larger or same size of the prototype depending on the need and purpose.

Example: The performance of a carburetor and a wrist watch is made on a large scale model.

Types of Similarity:

There are three necessary conditions for complete similarity between a model and a prototype.

i. Geometric Similarity - The model must be the same shape as the prototype, but may be scaled by some constant scale factor.

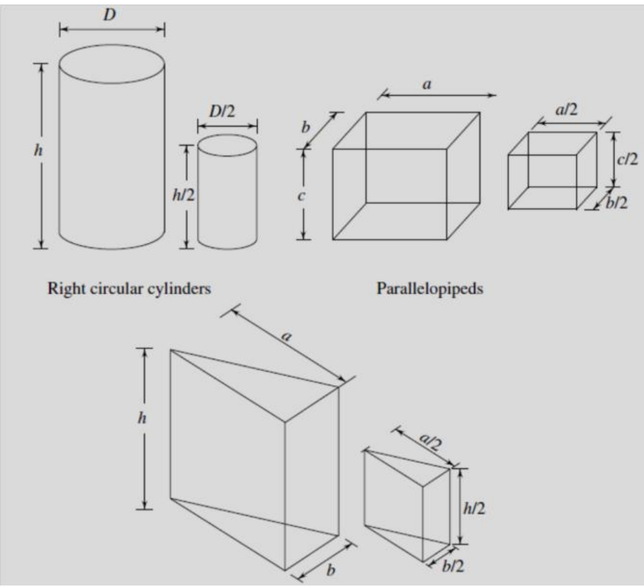


Fig. Geometrically similar objects

In geometrically similar systems, the ratio of any length in one system to the corresponding length in another system is the same everywhere. This ratio is known as scale factor. Therefore, geometrically similar objects are similar in shapes, i.e. proportionate in their physical dimensions, but differ in size.

ii. Kinematic similarity – It means that the velocity at any point in the model flow must be proportional (by a constant scale factor) to the velocity at the corresponding point in the prototype flow.

iii. Dynamic similarity – It is achieved when all forces in the model flow scale by a constant factor to corresponding forces in the prototype flow. (force-scale equivalence)

In other words, the ratio of magnitudes of any two forces in one system must be the same as the magnitude ratio of the corresponding forces in other systems.

In a system involving the flow of fluid, different forces will act on the fluid element due to different reasons. They are

Viscous forces	-	due to viscosity	- F_v
Pressure force	-	due to difference in pressure	- F_p
Gravity force	-	due to gravitational attraction	- F_g
Capillary force	-	due to surface tension	- F_s
Compressibility force	-	due to elasticity	- F_e

According to the Newton's law, resultant of all these forces will cause the acceleration of the fluid element. Hence, $F_r = F_v + F_p + F_g + F_s + F_e$.

From the D'Alemberts principle, the inertia force is equal and opposite to the resultant force i.e. $F_r = -F_i$.

Finally, $F_i + F_v + F_p + F_g + F_s + F_e = 0$.

For dynamic similarity, the ratio of these forces has to be same for model and prototype. A fluid motion, under all such forces is characterized by the

(i) hydrodynamic parameters like pressure, velocity and acceleration due to gravity.

(ii) rheological and other physical properties of the fluid involved.

(iii) geometric dimensions of the system.

- **Inertial force:** The inertia force acting on the fluid element is equal in magnitude to the mass of the element multiplied by its acceleration.

  Mass = density x volume $\Rightarrow m = \rho l^3$

  Acceleration is the rate at which the velocity changes with time and hence proportional to characteristic velocity 'v' divided with time interval-'t' i.e. $a \propto \frac{v}{t}$.

  Also time is proportional to the ratio of length to characteristic velocity i.e. $t \propto \frac{l}{v}$.

  Hence Inertia force, $F_i \propto ma \propto \rho l^3 \frac{v^2}{l} \Rightarrow F_i \propto \rho l^2 v^2$

- **Pressure force:** It arises due to difference in pressure and is proportional to pressure difference multiplied by area.

  Pressure force, $F_p \propto \Delta p l^2$

- **Viscous force:** It arises from the shear stress in a flow of fluid.

  From Newton's law of viscosity,

  Shear stress, τ = absolute viscosity x rate of shear strain

  Rate of shear strain or velocity gradient can be written as $\frac{v}{l}$.

  Surface area can be written as l^2.

  Shear force or viscous force, F_v = shear stress x surface area

  Finally, viscous force $F_v \propto \tau A \Rightarrow F_v \propto \mu \left(\frac{v}{l}\right) l^2 \Rightarrow F_v \propto \mu v l$.

- **Gravity force:** The gravity force on a fluid element is its weight.

  Hence gravity force $F_g \propto mg \Rightarrow F_g \propto \rho l^3 g$.

- **Capillary or surface tension force:** It arises due to existence of an interface between two fluids. It acts tangential to the surface and is equal to the coefficient of surface tension σ multiplied by the length of linear element on the surface perpendicular to which the force acts.

  Hence $F_c \propto \sigma l$

- **Compressibility or Elastic force:** Elastic force comes into consideration due to the compressibility of the fluid when it is flowing. For a given change in volume, the increase in pressure is proportional to the bulk modulus of elasticity E and gives rise to a force known as elastic force.
  Hence $F_e \propto El^2$.

<u>Case – 1:</u> For the *<u>flows in which viscous forces, pressure forces and inertia forces dominate</u>*, Reynolds number and Euler number should be same for both model and prototype, for the existence of complete dynamic similarity.

Reynolds number: It is defined as the ratio of inertia force to viscous force.

$$\text{Re} = \frac{F_{inertia}}{F_{vicsous}} = \frac{F_i}{F_v} = \frac{\rho l^2 v^2}{\mu v l} = \frac{\rho v l}{\mu}$$

Here Reynold's number is the ratio of forces and has no units and dimensions.

Equality of Re is a requirement for the dynamic similarity of flows in which viscous forces are important.

Euler number: It is defined as the ratio of pressure force to inertia force.

$$Eu = \frac{F_{pressure}}{F_{inertia}} = \frac{F_p}{F_i} = \frac{\Delta p l^2}{\rho l^2 v^2} = \frac{\Delta p}{\rho v^2}$$

Thus, for dynamic similarity**,** $\frac{\rho_m v_m l_m}{\mu_m} = \frac{\rho_p v_p l_p}{\mu_p}$ and $\frac{\Delta p_m}{\rho_m v^2{}_m} = \frac{\Delta p_p}{\rho_p v^2{}_p}$.

These kinds of flows are observed in instances like (i) the full flow of fluid in a completely closed conduit (ii) flow of air past a low-speed aircraft and (iii) flow of water past a submarine deeply submerged to produce no waves on the surface.

In the case of internal flows, the characteristic length 'l' is replaced by the hydraulic diameter 'D_h' which is defined as the ratio of four times the cross-sectional area to the wetted perimeter.

$$D_h = \frac{4A_c}{P}$$

For circular sections, hydraulic diameter is equal to its diameter.

$$D_h = \frac{4A_c}{P} = \frac{4\pi D^2}{4}\frac{1}{\pi D} = D$$

For a rectangular section of width 'w' and height 'h', the hydraulic diameter is

$$D_h = \frac{4A_c}{P} = \frac{4wh}{2(w+h)} = \frac{2wh}{(w+h)}$$

<u>Case – 2:</u> For the *<u>flows in which gravity forces, pressure forces and inertia forces dominate</u>*, Euler number and Froude number should be same for both model and prototype, for the existence of complete dynamic similarity. These kinds of forces are observed where free surface flows are available.

Euler number: It is defined as the ratio of pressure force to inertia force.

$$Eu = \frac{F_{pressure}}{F_{inertia}} = \frac{F_p}{F_i} = \frac{\Delta p l^2}{\rho l^2 v^2} = \frac{\Delta p}{\rho v^2}$$

Froude number: It as the square root of the ratio of inertia force to gravitational force.

$$Fr = \sqrt{\frac{F_{inertia}}{F_{gravity}}} = \sqrt{\frac{F_i}{F_g}} = \sqrt{\frac{\rho l^2 v^2}{\rho l^3 g}} = \frac{v}{\sqrt{\lg}}$$

Thus, for dynamic similarity**,** $\frac{v_m}{\sqrt{l_m g_m}} = \frac{v_p}{\sqrt{l_p g_p}}$ and $\frac{\Delta p_m}{\rho_m v^2{}_m} = \frac{\Delta p_p}{\rho_p v^2{}_p}$.

Equality of Fr is a requirement for the dynamic similarity of flows with a free surface in which gravity forces are important.

These kinds of flows are observed in instances like (i) the flow of a liquid in an open channel (ii) wave motion caused by the passage of a ship through water and (iii) flow over wiers and spillways.

Case – 3: For the *flows in which surface tension force and inertia force dominate*, Weber number should be same for both model and prototype, for the existence of complete dynamic similarity.

Weber number: It is defined as the ratio of pressure force to inertia force.

$$Wb = \frac{F_{surface\ tension}}{F_{inertia}} = \frac{\sigma l}{\rho l^2 v^2} = \frac{\sigma}{\rho l v^2}$$

These kinds of flows are observed in instances like (i) the flow of a liquid in which capillary waves appear (ii) flows of small jets and thin sheets of liquid injected by a nozzle in air and (iii) flow of thin sheet of liquid over a solid surface.

Note:

- ✓ When fluid motions are kinematically similar, the patterns formed by the streamlines are geometrically similar at corresponding times. Since the impermeable boundaries also represent streamlines, kinematically similar flows are possible only past geometrically similar boundaries. Therefore, geometric similarity is a necessary condition for the kinematic similarity to be achieved.
- ✓ We can understand geometric similarity as length-scale equivalence and kinematic similarity as time-scale equivalence i.e similarity in motion.
- ✓ In a general flow field, complete similarity between a model and prototype is achieved only when there is geometric, kinematic, and dynamic similarity.

Dimensional analysis is useful in *all* disciplines, especially when it is necessary to design and conduct experiments.

Assignment Questions

Dimensional Analysis and Model Similitude

A. Questions testing the remembering / understanding level

I. Objective Questions:

1. Which of the following is not a dimensionless number? [ ]

   (a) the coefficient of lift (C_f) (b) the pipe friction factor (f)

   (c) the manning's coefficient (n) (d) the coefficient of discharge (C_d)

2. The principal of dimensional homogeneity implies that [ ]

(a) only the dimensional quantities may be added or subtracted

(b) only the variables with the same dimensions may be multiplied or divided

(c) only the variables with the same dimensions may be added or subtracted

(d) dimensions of the two sides of the equation may not be same

3. If there are ‘n’ variables in a particular flow situation, and these variables contain ‘m’ primary dimensions, then the number of dimensionless groups relating the variables will be [ ]

   (a) n+m (b) n-m (c) n/m (d) m/n

4. The repeated variables in dimension analysis should [ ]

   (a) for the non-dimensional parameters among themselves

   (b) not include the dependent variables

   (c) have two variables with the same dimensions

   (d) must contain jointly all the fundamental dimensions involved in the phenomenon

5. Dimensional analysis is useful in [ ]

   (a) checking the correctness of a physical equation

   (b) determining the number of variables involved in a particular phenomenon

   (c) determining the dimensionless groups from the given variables

   (d) the exact formulation of a physical phenomenon

6. The limitation of dimensional analysis is that the ____________________ has to be determined by experiments.

7. Kinematic similarity between model and prototype is [ ]

   (a) the similarity of stream line patterns (b) the similarity of discharge

   (c) the similarity of the force influencing the flow (d) the use of same model scale throughout

8. Dynamic similarity between model and prototype implies that [ ]

   (a) the forces acting at corresponding locations are same

   (b) the flow pattern is similar

   (c) there is point to point correspondence between the two systems

   (d) both the systems undergo similar rates of change of motion

9. Principles of similitude forms the basis of [ ]

   (a) performing acceptance tests (b) comparing two identical equipments

   (c) comparing the similarity between design and actual equipment

   (d) designing and testing models so that the results can be worked out for the prototype

10. Select the situation in which Reynolds number law is applicable [ ]

    (a) flow over the spillway of a dam (b) flow of blood in veins and arteries

    (c) water hammer created in penstocks (d) flow through low speed turbomachines

11. The square root of inertia force to gravity force is known as [ ]

    (a) pressure coefficient (b) Froude’s number

    (c) Weber number (d) Mach number

12. The froude's model law would not be applicable for the analysis of [ ]

    (a) pressure rise due to sudden closure of valves (b) flow over the spillway of a dam

    (c) flow of liquid jets from orifices

    (d) motion of ship in rough and turbulent seas

13. Mach number is significant in [ ]

    (a) flow of highly viscous fluids (b) motion of rockets

    (c) water waves breaking against a wall (d) motion of submarine completely in water

14. Weber number will be an important consideration in the study of [ ]

    (a) flow of water through a pipeline (b) surface wave generated in liquids

    (c) steel balls dropping through oil (d) formation of spherical rain drops

15. Match the following:

Column A	Column B
(a) Froude's number	(i) ratio of pressure force to inertia force
(b) Mach number	(ii) ratio of surface tension force to inertia force
(c) Euler number	(iii) ratio of gravity force to inertia force
(d) Weber number	(iv) ratio of elastic force to inertia force

16. With Froude number as the criterion of dynamic similarity for a certain flow situation, the scale factor for discharge in terms of length scale ratio L_r is [ ]

    (a) $L_r^{0.5}$ (b) L_r (c) $L_r^{1.5}$ (d) $L_r^{2.5}$

B. Question testing the applying/analyzing level

I. Multiple Choice Questions:

1. In the model test of a spillway, the discharge over the model was measured as $2m^3/s$. The corresponding discharge over the prototype, which is 36 times the model size, would be

   (a) 432 m^3/s (b) 176 m^3/s (c) 2592 m^3/s (d) 15552 m^3/s

2. It is proposed to model a submarine at 10 m/s by testing a 10:1 scale model. The model need to be moved with a velocity of

   (a) 1 m/s (b) 10m/s (c) 50 m/s (d) 100 m/s

3. An ocean liner 250 m log has a maximum speed of 15 m/s. To simulate the wave resistance, the towing speed of a 10 m long model should be

   (a) 3 m/s (b) 5 m/s (c) 7.5 m/s (d) 12 m/s

4. A 1/10 model of an aeroplane is tested in a variable density wind tunnel. The prototype plane is to fly at 400 kmph speed under atmospheric conditions. If the pressure used in the tunnel is 10 times the atmospheric pressure, the velocity of air in the wind tunnel should be

   (a) 150 kmph (b) 300 kmph (c) 400 kmph (d) 600 kmph

II. Problems*:*

1. In flow over a smooth flat plate, the boundary layer thickness δ is found to depend on the free stream velocity u, fluid density and viscosity and the distance x from the leading edge. Express the correlation in the form of dimensionless groups.
2. Obtain a relationship for the torque τ to rotate a disk of diameter *D* in a fluid of viscosity μ at an angular speed ω over a plate, with clearance h.
3. To study the pressure drop in flow of water through a pipe, a model of scale 1/10 is used. Determine the ratio of pressure drops between model and prototype if water is used in the model. In case air is used determine the ratio of pressure drops.
4. An aircraft fuselage has been designed for speeds of 380 kmph. To estimate power requirements the drag is to be determined. A model of 1/10 size is decided on. In order to reduce the effect of compressibility, the model is proposed to be tested at the same speed in a pressurized tunnel. Estimate the pressure required. If the drag on the model was measured as 100 N, predict the drag on the prototype.

C. Questions testing the evaluating/creating level

1. In film lubricated journal bearings, the frictional torque is found to depend on the speed of rotation, viscosity of the oil, the load on the projected area and the diameter. Evaluate dimensionless parameters for application to such bearings in general.
2. The volume flow rate of a gas through a sharp edged orifice is found to be influenced by the pressure drop, orifice diameter and density and kinematic viscosity of the gas. Using the method of dimensional analysis obtain an expression for the flow rate.
3. The capillary rise h is found to be influenced by the tube diameter D, density ρ, gravitational acceleration g and surface tension σ. Determine the dimensionless parameters for the correlation of experimental results.
4. The power developed by hydraulic machines is found to depend on the head h, flow rate Q, density ρ, speed N, runner diameter D, and acceleration due to gravity, g. Obtain suitable dimensionless parameters to correlate experimental results.
5. A water tunnel operates with a velocity of 3m/s at the test section and power required was 3.75 kW. If the tunnel is to operate with air, determine for similitude the flow velocity and the power required. Take ρ_a = 1.25 kg/m3, $\gamma_a = 14.8 \times 10^{-6}$ m^2/s, $\gamma_w = 1.14 \times 10^{-6}$ m^2/s.

Boundary layer theory

Content:

- ✓ Introduction to boundary layer theory
- ✓ Thickness of boundary layer
- ✓ Development of boundary layer along a thin flat plate and its characteristics
- ✓ Boundary layer equations
- ✓ Momentum integral equation
- ✓ Laminar Boundary layer
- ✓ Separation of boundary layer
- ✓ Control of flow separation, Stream lined body , Bluff body and its applications
- ✓ Displacement, momentum and energy thickness of boundary Layer

References:

1. Robert W. Fox, Alan T. McDonald, Philip J. Pritchard, Introduction to FLuid Mechanics, 8th Edition, John Wiley & Sons, Inc.,
2. Frank M. White, Fluid Mechanics, 7th Edition, Tata McGraw Hill.
3. Yunus A. Cengel, John M. Cimbala, Fluid Mechanics
4. S. K. Som, G. Biswas, Introduction to Fluid Mechanics and Fluid Machines, Revised Second Edition, Tata McGraw Hill Book Publishing Company Limited.

Introduction:

The boundary layer of a flowing fluid is a thin layer close to the wall. In a flow field, viscous stresses are very prominent within this layer. The main flow velocity within this layer tends to zero while approaching the wall. Also the gradient of this velocity component in a direction normal to the surface is large as compared to the gradient of this component in the stream-wise direction.

Ludwig Prandtl (1904) introduced the concept of boundary layer theory and derived the expressions for boundary layer flow. He hypothesized that "for fluids having relatively small viscosity, the effect of the internal friction in the fluid is significant only in a narrow region surrounding the solid boundaries over which the fluid flows". Thus, close to the body is boundary layer where shear stresses exert an increasingly larger effect on the fluid as one moves from free stream towards the solid boundary. Outside the boundary layer where the effect of shear stresses is small compared to values inside the boundary layer (since the velocity gradient $\partial u/\partial y$ is negligible), the fluid particles experience no vorticity, and therefore the flow is similar to potential flow. Hence the boundary layer interface is a fictious one dividing rotational and irrotational flows.

Flow over a flat plate

Consider the flow of fluid over a top surface of a flat plate in a large flow field as shown in the figure. The velocity is uniform in the flow field having a value of u_∞.

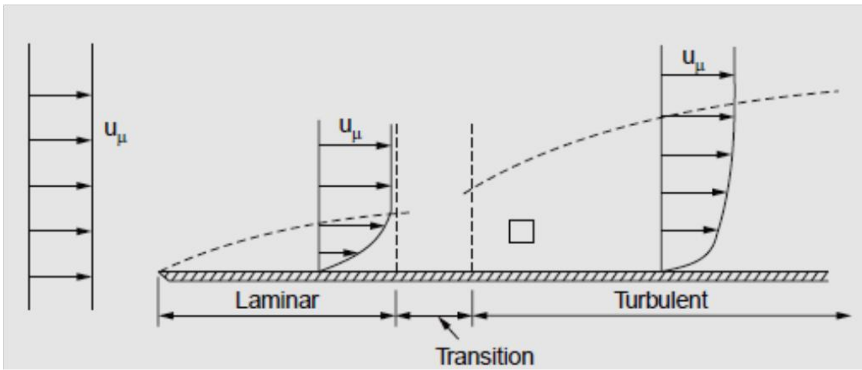


Fig. Formulation of Boundary layer over a flat surface

Real fluids have viscosity. When these fluids flow over surfaces, "no slip condition" prevails. The layer near the surface has to have the same velocity as the surface. If the surface is at rest, then this layer comes to rest. The adjacent layer is retarded to a lesser extent and this proceeds to layers more removed from the surface at rest. A velocity gradient forms leading to shear force being exerted over the layers. The velocity gradient is steepest at the interface and the shear is also highest at the interface. Work is to be done to overcome the force.

The edge of the plate at which the fluid enters over a plate is called leading edge and the edge at which the fluid leaves the plate is called trailing edge. Boundary layer begins to form from the leading edge and increases in thickness as the flow proceeds, due to the presence of viscosity. At the earlier stages the flow is regular and layers keep their position and there is no macroscopic mixing between layers. Momentum transfer resulting in the retarding force is by molecular diffusion between layers. This type of flow is called laminar flow. Viscous effects prevail over inertial effects in such a layer. Viscous forces maintain orderly flow.

The thickness is considered to be very small compared to the characteristic length L of the domain. In the normal direction, within the thin layer the velocity gradient $\frac{\partial u}{\partial y}$ is very large compared to the velocity gradient in the flow direction $\frac{\partial u}{\partial x}$.

As flow proceeds farther, inertial effects begin to prevail over viscous forces resulting in macroscopic mixing between layers. This type of flow is called turbulent flow. Higher rates of momentum transfer takes place in such a flow.

(i) Continuity equation

Consider a small element of shape rectangular parallelepiped of size 'dx' and 'dy' along x- and y- directions respectively in the boundary layer as shown in the fig. Thickness of the element is assumed to be unity.

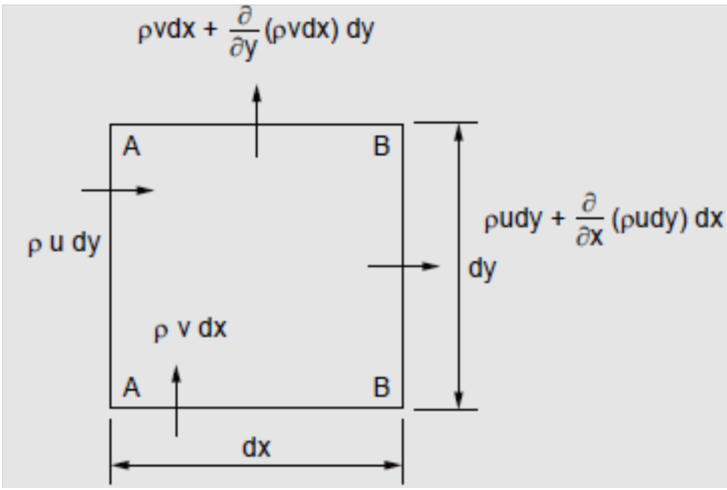


Fig. Enlarged view of element in boundary layer

Assumptions:

i. The flow is steady and incompressible (density remains constant).
ii. There is no pressure gradient in the boundary layer.

According to the principle of conservation of mass, under steady flow conditions, the net mass flow across the element should be zero. Under unsteady conditions, the net mass flow should equal the change of mass in the elemental volume considered.

Let ρ – density of the flowing fluid
u - velocity of fluid in x-direction
v – velocity of fluid in y-direction

Mass of fluid entering the face AA (x-direction) , $m_{in} = \rho udy$

Mass of fluid leaving the face BB (x-direction), $m_{out} = \rho udy + \frac{\partial(\rho udy)}{\partial x}dx$

Net flow along x-direction, $\Delta m_x = \frac{\partial(\rho udy)}{\partial x}dx = \rho\frac{\partial(u)}{\partial x}dxdy$

Mass of fluid entering the bottom face AB (y-direction) , $m_{in} = \rho vdx$

Mass of fluid leaving the top face AB (y-direction), $m_{out} = \rho vdx + \frac{\partial(\rho vdx)}{\partial y}dy$

Net flow along x-direction, $\Delta m_y = \frac{\partial(\rho vdx)}{\partial y}dy = \rho\frac{\partial(v)}{\partial y}dxdy$

Under steady state conditions the net mass flow across the element is zero.

$$\rho\frac{\partial(u)}{\partial x}dxdy + \rho\frac{\partial(v)}{\partial y}dxdy = 0 \Rightarrow \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

This equation is known as Continuity equation for steady and incompressible flow.

(ii) Momentum equation:

The equation is based on Newton’s second law of motion. The net force on the surface of the element should equal the rate of change of momentum of the fluid flowing through the element.

Consider a free body diagram of a small element of shape rectangular parallelepiped of size ‘dx’ and ‘dy’ along x- and y- directions respectively in the boundary layer as shown in the fig. Thickness of the element is assumed to be unity.

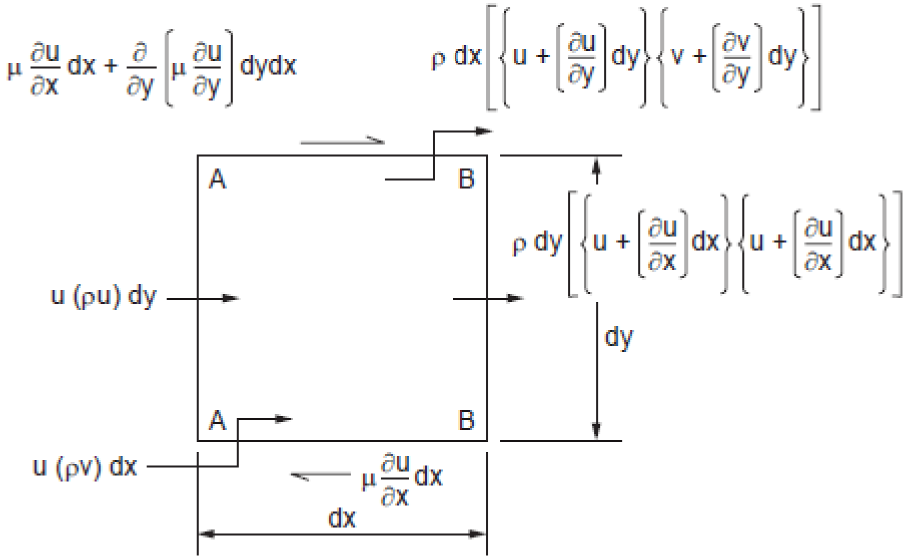


Fig. Momentum analysis

Momentum of the flow across the face AA along x-direction, $M_{AA} = (\rho u dy)u$

Momentum of the flow across the face BB along x-direction, $M_{BB} = (\rho u dy)u + \dfrac{\partial(\rho u^2 dy)}{\partial x}dx$

Net momentum exchange along x-direction, $M_x = \dfrac{\partial(\rho u^2 dy)}{\partial x}dx = \rho dxdy\left(u\dfrac{\partial u}{\partial x} + u\dfrac{\partial u}{\partial x}\right)$

Momentum of the flow across the bottom face AB, $M_{AB} = (\rho v dx)u$

Momentum of the flow across the face BB along, $M_{AB} = (\rho uvdx) + \dfrac{\partial(\rho uvdx)}{\partial y}dy$

Net momentum exchange along x-direction due to flow along y-direction,

$$M_y = \frac{\partial(\rho uvdx)}{\partial y}dy = \rho dxdy\left(u\frac{\partial v}{\partial y} + v\frac{\partial u}{\partial y}\right)$$

Net change of momentum along x-direction,

$$\rho dxdy\left(u\frac{\partial v}{\partial y} + v\frac{\partial u}{\partial y}\right) + \rho dxdy\left(u\frac{\partial u}{\partial x} + u\frac{\partial u}{\partial x}\right) = \rho dxdy\left(u\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) + \left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right)\right)$$

Also from continuity equation, $\dfrac{\partial u}{\partial x} + \dfrac{\partial v}{\partial y} = 0$

Then net change in momentum along x-direction $= \rho dxdy\left(u\dfrac{\partial u}{\partial x} + v\dfrac{\partial u}{\partial y}\right)$

Assuming that body forces and pressure forces are absent and surface force due to viscosity is present,

Shear force at the bottom surface $= \mu\dfrac{\partial u}{\partial y}dx$

Shear force at the top surface $= \mu \frac{\partial u}{\partial y} dx + \frac{\partial}{\partial y}\left(\mu \frac{\partial u}{\partial y} dx \right) dy$

Net shear force on the element $= \frac{\partial}{\partial y}\left(\mu \frac{\partial u}{\partial y} dx \right) dy = \mu dxdy \frac{\partial^2 u}{\partial y}$

Net change in momentum = net change in shear force

$$dxdy\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} \right) = \mu dxdy \frac{\partial^2 u}{\partial y^2}$$

$$\Rightarrow u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \frac{\mu}{\rho}\frac{\partial^2 u}{\partial y^2} = \gamma \frac{\partial^2 u}{\partial y^2}$$

This equation represents the momentum equation for boundary layer and 'γ' represents kinematic viscosity or momentum diffusivity. If it is a pressure induced flow $-\frac{1}{\rho}\frac{\partial P}{\partial x}$ is to be added to R.H.S of the above equation.

Note: Prandtl's Boundary layer equations

- ✓ Continuity equation: $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$
- ✓ Momentum equation: $u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \frac{\mu}{\rho}\frac{\partial^2 u}{\partial y} = \gamma \frac{\partial^2 u}{\partial y^2}$
- ✓ Pressure gradient normal to the flow direction is zero $\frac{\partial P}{\partial y} = 0$.

  This indicates that the pressure is approximately uniform across the boundary layer. The pressure at the surface is therefore equal to that at the edge of the boundary layer.

Solution for velocity profile – Laminar flow

For a flow over a flat plate, flow is said to be laminar if the Reynolds number is less than 5 x $10^{5.}$

Here both the continuity and momentum equations should be simultaneously solved to obtain the velocity profile. Boundary conditions for a given velocity profile in boundary layer

1. At the surface, i.e. y = 0, u = 0
2. At the boundary layer i.e. y = δ, u = u
3. At the boundary layer i.e. y = δ, velocity gradient is zero. $\frac{\partial u}{\partial y} = 0$

After applying the boundary conditions to the velocity profile,

The boundary layer thickness is obtained as $\frac{\delta}{x} = \frac{5}{\sqrt{\mathrm{Re}_x}}$

Shear stress is obtained as $\tau_w = \frac{0.332\rho u_\infty^2}{\sqrt{\mathrm{Re}_x}}$

Local skin friction coefficient $C_{fx} = \dfrac{\tau_w}{\left(\dfrac{1}{2}\rho u_\infty^2\right)} = \dfrac{0.332\rho u_\infty^2}{\sqrt{\text{Re}_x}}\left(\dfrac{1}{\dfrac{1}{2}\rho u_\infty^2}\right) = \dfrac{0.664}{\sqrt{\text{Re}_x}}$

Average friction coefficient, $C_f = \dfrac{1}{L}\int_0^L C_{fx}dx = \dfrac{1}{L}\int_0^L \dfrac{0.664}{\sqrt{\text{Re}_x}}dx = \dfrac{1}{L}\int_0^L \dfrac{0.664}{\sqrt{\dfrac{\rho u x}{\mu}}}dx = \dfrac{1.328}{\sqrt{\text{Re}_L}}$

Coefficient of Drag, $C_D = \dfrac{F_D}{\left(\dfrac{1}{2}\rho A U^2\right)}$, where F_D is the drag force.

Drag force, $F_D = \tau_w \times A$

Measurements of Boundary layer thickness

As the velocity in the boundary layer smoothly joins that of the outer flow, we have to decide how to define boundary layer thickness.

(a) Boundary layer thickness

The simplest situation that can be analyzed is the flow over a flat plate placed parallel to uniform flow velocity in a large flow field. The layer near the surface is retarded to rest or zero velocity. The next layer is retarded to a lower extent. This proceeds farther till the velocity equals the free stream velocity. As the distance for this condition is difficult to determine, *the boundary layer thickness is arbitrarily defined as the distance from the surface where the velocity is 0.99 times the free stream velocity*.

(b)Displacement thickness:

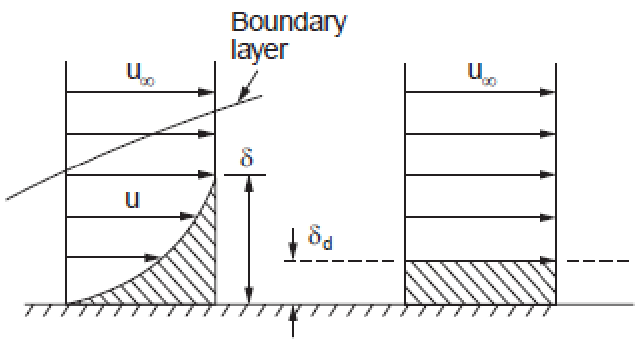


Fig. Displacement thickness

Compared to the thickness δ in free stream, the flow in the boundary layer is reduced due to the reduction in velocity which is the result of viscous forces. In the absence of the boundary layer the flow rate that would pass through the thickness δ will be higher.

The reduction in the volume per unit width is given by $\Delta Q = \int_0^\delta \rho\left(u_\infty - u\right)dy$.

If viscous forces were absent the velocity all through the thickness δ will be equal to u_∞. A thickness δ_d can be defined by equating the reduction in flow to a uniform flow with velocity u_∞ or $\rho u_\infty \delta_d$.

$$\delta_d = \int_0^\delta \frac{u_\infty - u}{u_\infty} dy = \int_0^\delta \left(1 - \frac{u}{u_\infty}\right) dy$$

This tells that displacing the boundary by a distance δ_d would pass the flow in the boundary layer at free stream velocity.

Finally displacement thickness is defined as the distance by which the wall would have to be displaced outward in a hypothetical frictionless flow so as to maintain same mass flux as in the actual flow. Mathematically, it is represented as

$$\delta_d = \int_0^\delta \left(1 - \frac{u}{u_\infty}\right) dy$$

Uses:

- This concept is used in the design of ducts, intake of air-breathing engines, wind tunnels etc. by first assuming a frictionless flow, and then enlarging the passage walls by the displacement thickness so as to allow the same flow rate.
- It is used in finding the pressure gradient $\partial P / \partial x$ at the edge of the boundary layer needed for solving the boundary layer equations.

(c) Momentum thickness

There is a reduction in momentum flow through the boundary layer as compared to the momentum flow in a thickness δ at free stream velocity.

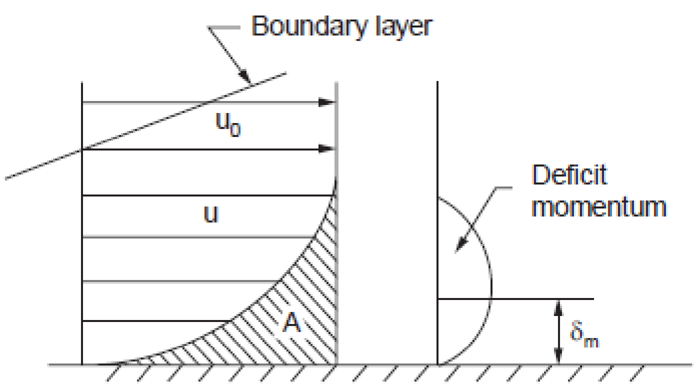


Fig. Momentum thickness

The thickness at which free stream velocity will have the same momentum flow as the deficit flow is called momentum thickness. The deficit flow at any thin layer at 'y' of thickness 'dy' is (for unit width) $\rho\,(u_\infty - u)\,dy$.

Momentum for this flow = $\rho u(u_\infty - u)dy$

The deficit momentum = $\int_0^\delta \rho u(u_\infty - u)dy$

Considering δm as momentum thickness,

$$\delta_m \rho u_\infty u_\infty = \int_0^\delta \rho u(u_\infty - u)dy \Rightarrow \delta_m = \int_0^\delta \left(\frac{u}{u_\infty} - \left(\frac{u}{u_\infty} \right)^2 \right) dy$$

Note: For a parabolic velocity profile of the form, $u = a + by + cy^2$

The displacement thickness is 1/3rd of the thickness of hydrodynamic boundary layer and that of momentum thickness is 1/7th of the thickness of hydrodynamic boundary layer.

Boundary layer separation

Boundary layer is formed in the case of flow of real fluids. Viscous forces exist in such flows. The shear stress at the wall is given by $\tau_w = \mu \left. \frac{du}{dy} \right|_{y=0}$.

The wall shear stress cannot be zero. Hence at y=0, $\frac{du}{dy} \neq 0$. This means that the velocity gradient at the wall cannot be zero.

Separation of flow is said to occur when the direction of the flow velocity near the surface is opposed to the direction of the free stream velocity, which means (du/dy) ≤ 0. Such a situation does not arise when there is no pressure gradient opposed to the flow direction, i.e., the pressure downstream of flow is higher compared to the pressure upstream. An example is subsonic diffuser. In the direction of flow the pressure increases. The increase in area along the flow causes a pressure rise.

If (dp/dx) increases to the extent that it can overcome the shear near the surface, then separation will occur. Such a pressure gradient is called adverse pressure gradient.

<u>Noteworthy Points:</u>

- In the case of incompressible flow in a nozzle a favorable pressure gradient exists. Separation will not occur in such flows.
- In the case of diverging section of a diffuser, separation can occur if the rate of area increase is large.
- In turbulent flow, the momentum near the surface is high compared to laminar flow. Hence turbulent layer is able to resist separation better than laminar layer.
- In the case of flow over spheres, cylinders, blunt bodies, airfoils etc., there is a change in flow area due to the obstruction and hence an adverse pressure gradient may be produced. *Simple analytical solutions are not available to determine exactly at what conditions separation will occur. Experimental results are used to predict such conditions.*

Assignment Questions

Dimensional Analysis and Model Similitude

A. Questions testing the remembering / understanding level

I. Objective Questions:

1. The boundary layer exists in the ____________. [ ]

   (a) Flow of real fluids (b) Flow of ideal Fluids

   (c) Flow over flat surface only (d) pipe flow only

2. Boundary layer is a thin fluid region close to the surface of a body in which [ ]

   (a) viscous force are negligible (b) velocity is uniform

   (c) inertial forces can be neglected (d) viscous forces cannot be neglected

3. The separation point 'A' is at the mid section and 'B' at trailing edge, the shear stress τ_A at A and τ_B at B are such that [ ]

   (a) $\tau_A > \tau_B$ (B) $\tau_B > \tau_A$ (c) $\tau_B = \tau_A$ (D) $\tau_B = \tau_A$ for laminar boundary layer

4. The separation of boundary layer occurs when [ ]

   (a) the flow is accelerated past a boundary (b) the fluid is ideal

   (c)the boundary layer comes at rest (d) any adverse pressure is encountered

5. Boundary layer separation occurs in [ ]

   (a) Zero pressure gradient (b) negative pressure gradient

   (c) positive pressure gradient (d) none of the above

6. In the boundary layer, the flow is: [IES-2006]

   (a) Viscous and rotational (b) Inviscid and irrotational

   (c) Inviscid and rotational (d) Viscous and irrotational

7. The hydrodynamic boundary layer thickness is defined as the distance from the surface where the [ ]

   (a) Velocity equals the local external velocity

   (b) Velocity equals the approach velocity

   (c) Momentum equals 99% of the momentum of the free stream

   (d) Velocity equals 99% of the local external velocity

8. The critical value of Reynolds number for transition from laminar to turbulent boundary layer in external flows is taken as: [ ]

   (a) 2300 (b) 4000 (c) 5×10^5 (d) 3×10^6

9. The separation of boundary layer can be prevented by [ ]

(a) Providing small divergence in a diffuser

b) providing a trip-wire ring in the laminar

(c) providing a bypass in the slotted wing

(d)Suction of the slow moving fluid by a suction slot (e) all the above

10. The thickness of boundary layer on a flat plate is proportional to [IES-19 99]

(a)x (b) $\sqrt{x}$ (c) x^2 (d) $x^{\frac{3}{2}}$

11. Momentum thickness is given by which of the following relations?

(a) $\int_0^\delta \left(1-\frac{u}{U}\right)dy$ (b) $\int_0^\delta \frac{u}{U}\left(1-\frac{u}{U}\right)dy$

(c) $\int_0^\delta \frac{u}{U}\left(1-\left(\frac{u}{U}\right)^2\right)dy$ (d) None of the above

12. The boundary layer separation occurs when [ ]

(a) $\frac{dp}{dx}<0$ (b) $\left(\frac{\partial u}{\partial y}\right)_{y=0}>0$ (c) $\left(\frac{\delta u}{\delta y}\right)_{y=0}>0$ (d) none of the above

13. Which of the following is the condition for detached flow? [ ]

(a) $\left(\frac{\partial u}{\partial y}\right)_{y=0}=0$ (b) $\left(\frac{\partial u}{\partial y}\right)_{y=0}>0$ (c) $\left(\frac{\partial u}{\partial y}\right)_{y=0}<0$ (d) None of the above

B. Question testing the applying/analyzing level

I. Multiple Choice Questions:

1. A flat plate is kept in an infinite fluid medium. The fluid has a uniform free stream velocity parallel to the plate. for the laminar boundary formed on the plate, pick the correct option matching

List-I	List-II
A. Boundary Layer thickness	1. decrease in flow direction
B. Shear stress at the plate	2.increse in the flow direction
C. Pressure gradient along the plate	3.remains unchanged

Codes:

	A	**B**	**C**
(a)	**1**	**2**	**3**
(b)	**2**	**2**	**2**
(c)	**1**	**1**	**2**
(d)	**2**	**1**	**3**

2. Match the following

List-I	List-II
A. Boundary layer thickness	1. Distance from the boundary where velocity is 99 % of uniform velocity
B. Displacement thickness	2. Distance from the boundary by which the main flow can be assumed to be shifted
C. Laminar Boundary Layer	3.Distance from the boundary where from the flow ceases to the laminar
D. Turbulent Boundary Layer	4. Region near the boundary where viscous stress is also present

Codes:

	A	B	C	D
(a)	1	2	3	4
(b)	2	1	3	4
(c)	1	2	4	3
(d)	2	1	4	3

3. If δ_1 and δ_2 are the boundary layer thickness at a point 'x' from the leading edge on a flat plate at the Reynolds's numbers 100 and 256 respectively, then the ratio of δ_1 to δ_2 will be [ ]

(a) 0.625 (b) 1.6 (c) 2.56 (d) 4.96

4. Ina laminar Boundary Layer over a flat plate the ratio of shear stress τ_1 and τ_2 at two sections 1 and 2 at distance from the leading such that x_2=4x, is [ ]

(a)1 (b) 2 (c) 4 (d)16

5. A Laminar Boundary layer has a velocity distribution $\frac{u}{U}=\frac{y}{\delta}$. The ratio of $\frac{\delta}{\delta^*}$ and $\frac{\theta}{\theta^*}$ are respectively [ ]

(a) $\frac{1}{3},\frac{1}{2}$ (b) $\frac{1}{6},\frac{1}{3}$ (c) $\frac{1}{2},\frac{1}{6}$ (d) $\frac{1}{6},\frac{1}{2}$

From above data, the shape factor is [ ]

(a) $\frac{1}{2}$ (b) $\frac{1}{3}$ (c) 2 (d) 3

6. The velocity distribution in the boundary layer is given by $\frac{u}{U}=\frac{y}{\delta}$, where is the velocity at a distance y and u=U at y= δ . Find displacement thickness [ ]

(a) $\frac{\delta}{2}$ (b) $\frac{\delta}{3}$ (c) $\frac{\delta}{5}$ (d) δ

From the above data, momentum thickness is [ ]

(a) $\frac{\delta}{6}$ (b) $\frac{\delta}{3}$ (c) $\frac{\delta}{5}$ (d) δ

From the above data, Energy thickness is __________ [ ]

(a) $\frac{\delta}{4}$ (b) $\frac{\delta}{3}$ (c) $\frac{\delta}{5}$ (d) δ

7. The thickness of the laminar boundary layer on a flat plate at a point A is 2 cm and at a point B, 1 cm downstream of A is 3 cm. What is the distance of A from the leading edge? [ ]

   (a) 0.50 m (b) 0.80 m (c)1.00 m (d) 1.25 m

8. The transition Reynolds number for flow over a flat plate is 5×10^5. What is the distance from the leading edge at which transition will occur for flow of water with a uniform velocity of 1 m/s? [For water, the kinematic viscosity, $\nu = 0.858 \times 10^{-6}$ m^2/s [IES-1994] [ ]

   (a) 1 m (b) 0.43 m (c) 43 m (d) 103 m

9. Which one of the following velocity distributions of u/u_∞ satisfies the boundary conditions for laminar flow on a flat plate? (Here u_∞ is the free stream velocity, u is velocity at any normal distance y from the flat plate, $\eta = y/\delta$ and δ is boundary layer thickness) [IES-1996] [ ]

   (a) $\eta - \eta 2$ (b) $1.5\eta - 0.5\eta 3$ (c) $3\eta - \eta 2$ (d) $\cos(\pi\eta / 2)$

10. A flat plate, 2m × 0.4 m is set parallel to a uniform stream of air (density1.2kg/m^3 and viscosity 16 centistokes) with its shorter edges along the flow. The air velocity is 30 km/h. What is the approximate estimated thickness of boundary layer at the downstream end of the plate? [IES-2004] [ ]

    (a) 1.96 mm (b) 4.38 mm (c) 13.12 mm (d) 9.51 mm

11. For linear distribution of velocity in the boundary layer on a flat plate, what is the ratio of displacement thickness (δ^*) to the boundary layer thickness (δ)? [IES-2005]

    (a) ¼ (b) 1/3 (c) ½ (d) 1/15 [ ]

12. The velocity distribution in the boundary layer is given as $\frac{u}{u_\infty} = \frac{y}{\delta}$ u / us = y / δ, where u is the velocity at a distance y from the boundary, u_∞ is the free stream velocity and δ is the boundary layer thickness at a certain distance from the leading edge of a plate. The ratio of displacement to momentum thicknesses is: [IES-2001; 2004] [ ]

    (a) 5 (b) 4 (c) 3 (d) 2

II. Problems*:*

1. Find the displacement thickness, the momentum thickness and energy thickness for the velocity distribution in the boundary layer given by $\frac{u}{U} = \frac{y}{\delta}$ where u is the velocity at a distance y from the plate and u = U at y = δ, where δ = Boundary layer thickness.

2. Find the displacement thickness, the momentum thickness and energy thickness for the velocity distribution in the boundary layer given by $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$.

3. For the velocity profile for laminar boundary layer flows given as $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$ find the expression for boundary layer thickness, shear stress and coefficient of drag in terms of Reynolds number.
4. A thin plate is moving in still atmospheric air at a velocity of 5 m/s. The length of the plate is 0.6 m and width 0.5 m. Calculate (i) the thickness of the boundary layer at the end of the plate, and (ii) drag force on one side of the plate. Take density of air as 1.24 kg/m^3 and kinematic viscosity 0.15 stokes.
5. A flat plate 1.5m x1.5m moves at 50 km/hour in stationary air of density 1.15kg/m^3. If the coefficient of drag and lift are 0.15 and 0.75 respectively, determine (i) the lift force (ii) the drag force (iii) the resultant force, and (iv) the power required to keep the plate in motion.
6. Experiments were conducted in a wind tunnel with a wind speed of 50 km/hour on a flat plate of size 2m long and 1m wide. The density of air is 1.15 kg/m^3, the coefficient of lift and drag forces are 0.75 and 0,15 respectively. Determine (i) The lift force (ii) the drag force (iii) The resultant force, (iv) the direction of resultant force and (v) the power required to keep the plate in motion.
7. An airfoil of chord length 2 m and spam 15 m has an angle of attack as 6^0. The airfoil is moving with a velocity of 80 m/sec in air whose density is 1.25 kg/m^3. Find the weight of air foil and power required to drive it. The values of coefficient of drag and lift corresponding to angle of attack are given as 0.03and o.5 respectively.

8. Find the difference in drag force exerted on a flat plate of size 2m x 2m when the plate is moving at a speed of 4m /sec normal to its plane in (i) water (ii) air of density $1.24 kg/m^3$. Coefficient of drag is given as 1.15.
9. A 7 cm diameter sphere is tested in water at 20^0C and velocity of 4m/s and has a measured drag of 8 N. What will be the velocity and drag force of a 3 m diameter weather balloon moving in air 20 ^{o} C and 1 atm under similar conditions?

C. Questions testing the evaluating/creating level

1. Air is flowing over a smooth plate with a velocity of 10m/s. The length of the plate is 1.2 m and width 0.8 m. If a laminar boundary layer exists up to a value of Re $=2\times10^5$, find the maximum distance from the leading edge up to which laminar boundary layer exists. Find the maximum thickness of laminar boundary layer if the velocity profile is given by . $\frac{u}{U}=2\left(\frac{y}{\delta}\right)-\left(\frac{y}{\delta}\right)^2$ Take kinematic viscosity of air=0.15 stokes.
2. Water is flowing over a thin smooth plate of length 4 m and width 2 m at a velocity of 1 m/s. If the Boundary layer flow changes from laminar to turbulent at a Reynolds number 5×10^5 , find i) the distance from leading edge up to which boundary layer is laminar, ii) the thickness of the boundary layer at the transition point, and iii) the drag force on one side of the plate. Take viscosity of water = 9.81×10^{-4} Ns/m^2
3. A kite weighing 7.848 N has an effective area of $0.8m^2$. It is maintained in air at an angle of 100to the horizontal. The string attached to the kite makes an angle of 45^0 to the horizontal and at this position the value of coefficient of drag and lift are 0.6 and 0.8 respectively. Find the speed of the wind and tension in the string . Take the density of air is 1.25 kg/ m^3
4. A kite 0.8m x0.8m weighing 3.924N assumes an angle of 12^0 to the horizontal. The string attached to the kite makes an angle of 45^0 to the horizontal. The pull on the string is 24.525N when wind is flowing at a speed of 30 km/hour. Find the corresponding co-efficient of drag and lift. Density of air is given as $1.25 kg/m^3$