LECTURE NOTES

ON

FLUID MECHANICS-1

II B. Tech I semester

Definition of Stress

Consider a small area δA on the surface of a body (Fig. 1.1). The force acting on this area is δF This force can be resolved into **two perpendicular components**

The component of force acting normal to the area called **normal** force and is denoted by δF_n

The component of force acting along the plane of area is called **tangential** force and is denoted by δF_t

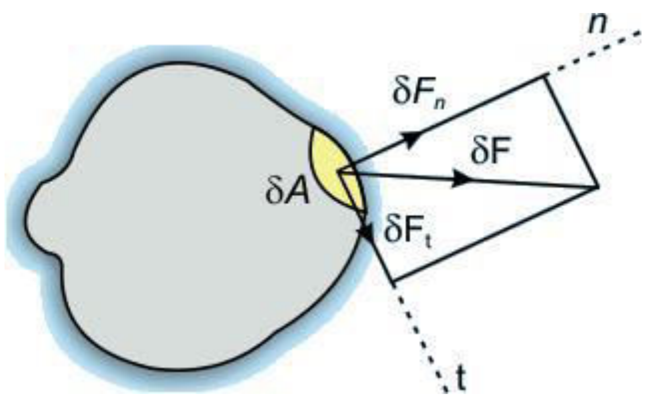


Fig 1.1 Normal and Tangential Forces on a surface

When they are expressed as force per unit area they are called as **normal stress** and **tangential stress** respectively**.** The tangential stress is alsocalled shear stress

The normal stress

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

And shear stress

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

Definition of Fluid

A fluid is a substance that **deforms continuously** in the face of tangential or shear stress, **irrespective of the magnitude of shear stress** .This continuous deformation under the application of shear stress constitutes a flow .In this connection fluid can also be defined as the **state of matter that cannot sustain any shear stress.**

Example : Consider Fig 1.2

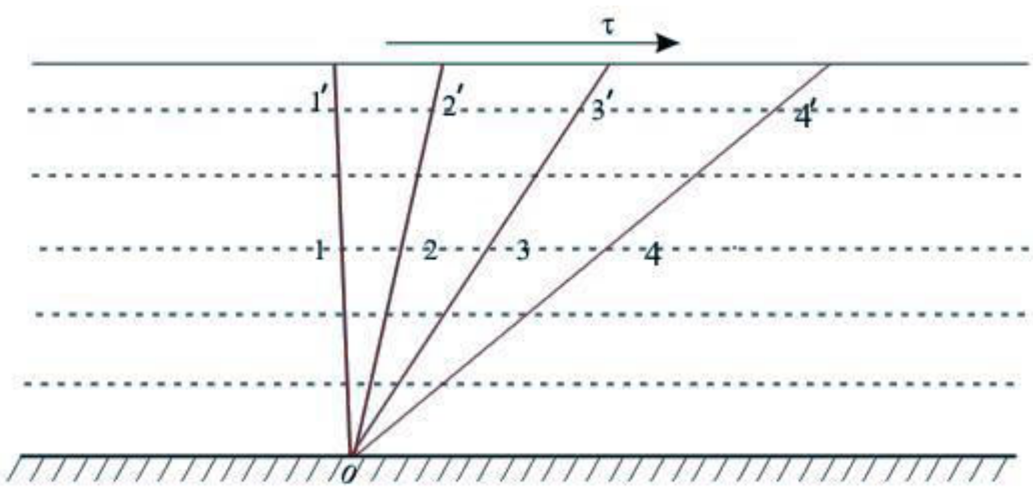


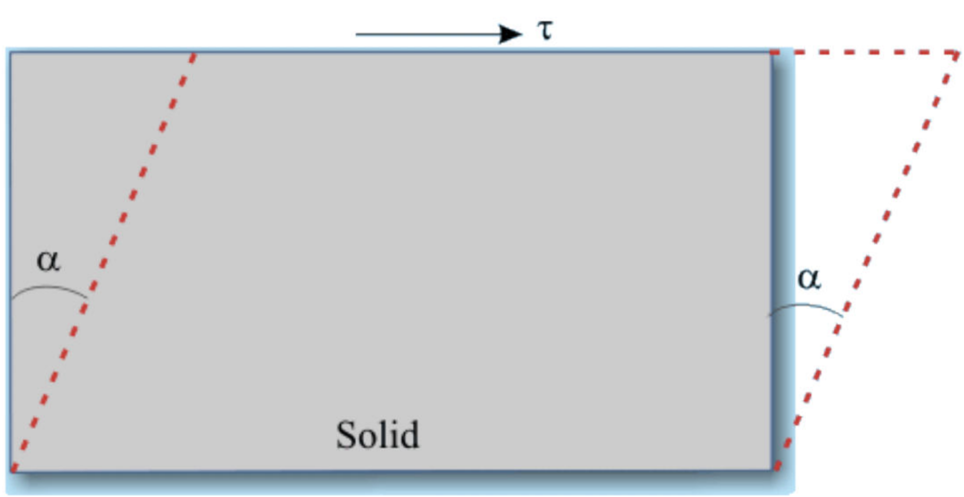
Fig 1.2 Shear stress on a fluid body

If a shear stress τ is applied at any location in a fluid, the element 011' which is initially at rest, will move to 022', then to 033'. Further, it moves to 044' and continues to move in a similar fashion.

In other words, the **tangential stress in a fluid body depends on velocity of deformation and vanishes as this velocity approaches zero. A good example is**Newton'sparallel plate experiment **where dependence of shear force on the velocity of deformation was established.**

Distinction Between Solid and Fluid

Solid	Fluid
▪ More Compact Structure	▪ Less Compact Structure
▪ Attractive Forces between the	▪ Attractive Forces between the



molecules

are larger therefore more closely packed

Solids can resist tangential stresses in static condition

Whenever a solid is subjected to shear stress

It undergoes a definite deformationα or breaks

α is proportional to shear stress upto some limiting condition

Solid may regain partly or fully its original shape when the tangential stress is removed

molecules

are smaller therefore more loosely packed

Fluids cannot resist tangential stresses in static condition.

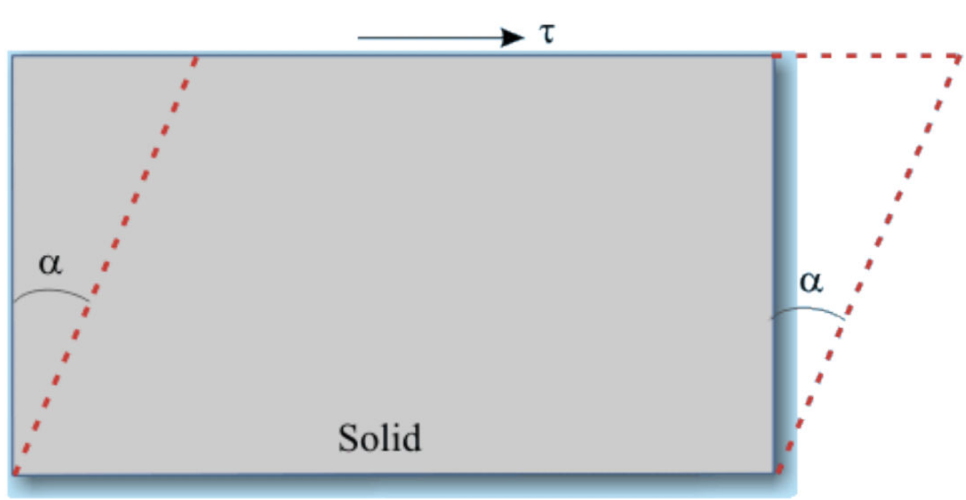
Whenever a fluid is subjected to shear stress

No fixed deformation

Continious deformation takes place

until the shear stress is applied

A fluid can never regain its original shape, once it has been distorded by the shear stress



$\nu = \frac{}{\rho}$

Fluid Properties :

Characteristics of a continuous fluid which are independent of the motion of the fluid are called basic properties of the fluid. Some of the basic properties are as discussed below.

Viscosity (μ) :

Viscosity is a fluid property whose effect is understood when the fluid is in motion. In a flow of fluid, when the fluid elements move with different velocities, each element will feel some resistance due to fluid friction within the elements. Therefore, shear stresses can be identified between the fluid elements with different velocities.The relationship between the shear stress and the velocity field was given by Sir Isaac Newton. Consider a flow (Fig. 1.5) in which all fluid particles are moving in the same direction in such a way that the fluid layers move parallel with different velocities.

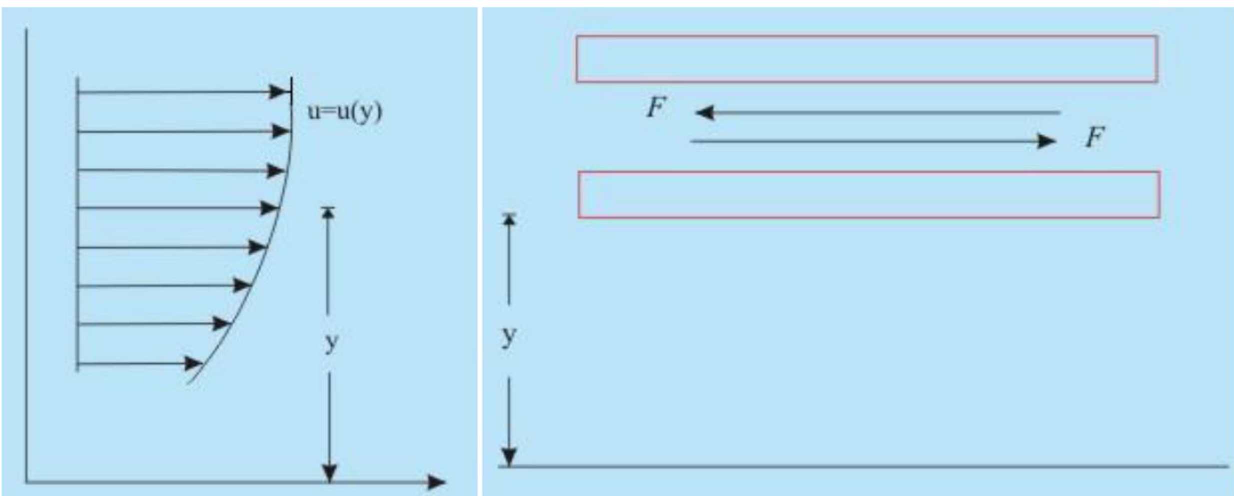


Fig 1.5 Parallel flow of a fluid

Fig 1.6 Two adjacent layers of a moving fluid.

The upper layer, which is moving faster, tries to draw the lower slowly moving layer along with it by means of a force *F* along the direction of flow on this layer. Similarly, the lower layer tries to retard the upper one, according to Newton's third law, with an equal and opposite force *F* on it (Figure 1.6).
Such a fluid flow where x-direction velocities, for example, change with y-coordinate is called **shear flow** of the fluid.
Thus, the dragging effect of one layer on the other is experienced by a tangential force *F* on the respective layers. If *F* acts over an area of contact *A*, then the shear stress τ is defined as

$$= F/A$$

Viscosity (μ) :

Newton postulated that τ is proportional to the quantity Δu/ Δy where Δy is the distance of separation of the two layers and Δu is the difference in their velocities.
In the limiting case of , Δu / Δy equals du/dy, the velocity gradient at a point in a direction perpendicular to the direction of the motion of the layer.
According to Newton τ and du/dy bears the relation

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

where, the constant of proportionality μ is known as the **coefficient of viscosity** or simply viscosity which is a property of the fluid and depends on its state. Sign of τdepends upon the sign of du/dy. For the profile shown in Fig. 1.5, du/dy is positive everywhere and hence, τ is positive. Both the velocity and stress are considered positive in the positive direction of the coordinate parallel to them.

Equation

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

defining the viscosity of a fluid, is known as Newton's law of viscosity. Common fluids, viz. water, air, mercury obey Newton's law of viscosity and are known asNewtonian fluids.Other classes of fluids, viz. paints, different polymer solution, blood do not obey the typical linear relationship, of τ and du/dy and are known as **non-Newtonian fluids**.

In non-newtonian fluids viscosity itself may be a function of deformation rate as you will study in the next lecture.

Causes of Viscosity

The causes of viscosity in a fluid are possibly attributed to two factors:
intermolecular force of cohesionmolecular momentum exchange
Due to strong cohesive forces between the molecules, any layer in a moving fluid tries to drag the adjacent layer to move with an equal speed and thus produces the effect of viscosity as discussed earlier. Since cohesion decreases with temperature, the liquid viscosity does likewise.

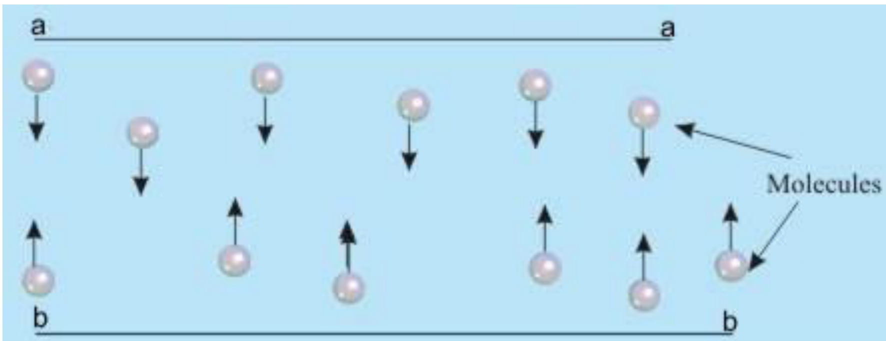


Fig 1.7 Movement of fluid molecules between two adjacent moving layers

Molecules from layer aa in course of continous thermal agitation migrate into layer bb
Momentum from the migrant molecules from layer aa is stored by molecules of layer bb by way of collisionThus layer bb as a whole is speeded up
Molecules from the lower layer bb arrive at aa and tend to retard the layer aa
Every such migration of molecules causes forces of acceleration or deceleration to drag the layers so as to oppose the differences in velocity between the layers and produce the effect of viscosity.

Causes of Viscosity - contd from previous slide...

As the random molecular motion increases with a rise in temperature, the viscosity also increases accordingly. Except for very special cases (e.g., at very high pressure) the viscosity of both liquids and gases ceases to be a function of pressure.

For Newtonian fluids, the coefficient of viscosity depends strongly on temperature but varies very little with pressure.For liquids, molecular motion is less significant than the forces of cohesion, thus **viscosity of liquids decrease with increase in temperature.**For gases,molecular motion is more significant than the cohesive forces, thus **viscosity of gases increase with increase in temperature.**

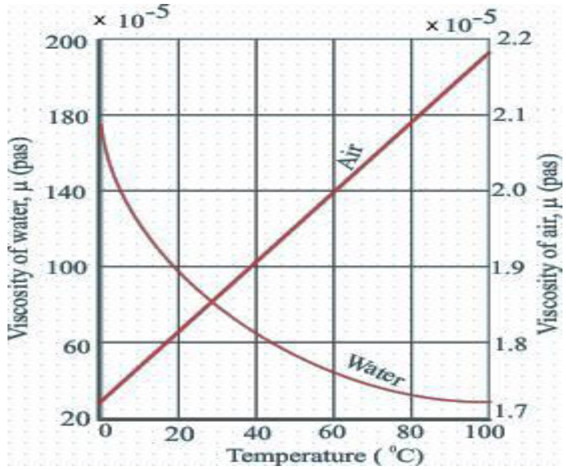


Fig 1.8: Change of Viscosity of Water and Air under 1 atm

No-slip Condition of Viscous Fluids

It has been established through experimental observations that the relative velocity between the solid surface and the adjacent fluid particles is zero whenever a viscous fluid flows over a solid surface. This is known as no-slip condition

This behavior of no-slip at the solid surface is not same as the wetting of surfaces by the fluids. For example, mercury flowing in a stationary glass tube will not wet the surface, but will have zero velocity at the wall of the tube.

The wetting property results from surface tension, whereas the no-slip condition is a consequence of fluid viscosity

Ideal Fluid

Consider a hypothetical fluid having a zero viscosity ($\mu = 0$). Such a fluid is called an ideal fluid and the resulting motion is called as **ideal** or **inviscid flow**. **In an ideal flow, there is no existence of shear force because of vanishing viscosity.**

$$\tau = \mu \frac{du}{dy} = 0 \qquad \text{since } \mu = 0$$

All the **fluids in reality have viscosity** ($\mu > 0$) and hence they are termed as real fluid and their motion is known as viscous flow.Under certain situations of very high velocity flow of viscous fluids, an accurate analysis of flow field away from a solid surface can be made from the ideal flow theory.

Non-Newtonian Fluids

There are certain fluids where the linear relationship between the shear stress and the deformation rate (velocity gradient in parallel flow) as expressed by the $\tau = \mu \frac{du}{dy}$ is not valid. For these fluids the viscosity varies with rate of deformation.Due to the deviation from Newton's law of viscosity they are commonly termed as **non-Newtonian fluids**. Figure 2.1 shows the class of fluid for which this relationship is nonlinear.

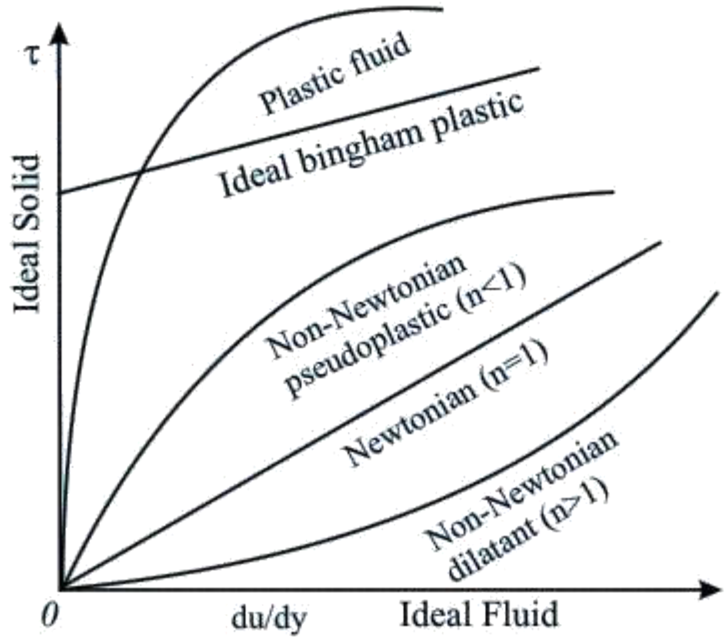


Figure 2.1 Shear stress and deformation rate relationship of different fluids

The abscissa in Fig. 2.1 represents the behaviour of ideal fluids since for the ideal fluids the resistance to shearing deformation rate is always zero, and hence they exhibit zero shear stress under any condition of flow.The ordinate represents the ideal solid for there is no deformation rate under any loading condition.The Newtonian fluids behave according to the law that shear stress is linearlyproportional to velocity gradient or rate of shear strain $\tau = \mu \, du/dy$. Thus for these fluids, the plot of shear stress against velocity gradient is a straight line through the origin. The slope of the line determines the viscosity.The non-Newtonian fluids are further classified as pseudo-plastic, dilatant and Bingham plastic.

Compressibility

Compressibility of any substance is the measure of its change in volume under the action of external forces.The normal compressive stress on any fluid element at rest is known as hydrostatic pressure p and arises as a result of innumerable molecular collisions in the entire fluid.

The degree of compressibility of a substance is characterized by the **bulk modulus of elasticity** E defined as

$$E = \lim_{\Delta\forall \to 0} \left(\frac{-\Delta p}{\Delta\forall \;/\; \forall} \right)$$

(2.3)

Where $\forall$ and Δp are the changes in the volume and pressure respectively, and $\forall$ is the initial volume. The negative sign (-sign) is included to make E positive, since increase inpressure would decrease the volume i.e for $\Delta p>0$, $\forall<0$) in volume.For a given mass of a substance, the change in its volume and density satisfies the relation

$$= 0, \qquad \rho \forall) = 0$$

$$\frac{\Delta\forall}{\forall} = -\frac{\Delta\rho}{\rho} \tag{2.4}$$

using $$E = \lim_{\Delta\forall\to 0}\left(\frac{-\Delta p}{\Delta\forall\ /\forall}\right) \quad \& \quad \frac{\Delta\forall}{\forall} = -\frac{\Delta\rho}{\rho}$$

we get

$$E = \lim_{\Delta\rho\to 0}\left(\frac{\Delta p}{\Delta\rho/\rho}\right) = \rho\frac{dp}{d\rho} \tag{2.5}$$

Values of *E* for liquids are very high as compared with those of gases (except at very high pressures). Therefore, liquids are usually termed as incompressible fluids though, in fact, no substance is theoretically incompressible with a value of *E* as ∞ .

For example, the bulk modulus of elasticity for water and air at atmospheric pressure are approximately 2 x 10^6 kN/m ^{2} and 101 kN/m ^{2} respectively. It indicates that air is about 20,000 times more compressible than water. Hence water can be treated as incompressible.
For gases another characteristic parameter, known as compressibility *K,* is usually defined , it is the reciprocal of *E*

$$K = \frac{1}{E} = \frac{1}{\rho}\left(\frac{d\rho}{dp}\right) = -\frac{1}{\forall}\left(\frac{d\forall}{dp}\right) \tag{2.6}$$

K is often expressed in terms of specific volume $\forall$.

For any gaseous substance, a change in pressure is generally associated with a change in volume and a change in temperature simultaneously. A **functional relationship between the pressure, volume and temperature at any equilibrium state is known as thermodynamic equation of state for the gas.**

For an ideal gas, the thermodynamic equation of state is given by

$$p = \rho RT$$

where *T* is the temperature in absolute thermodynamic or gas temperature scale (which are, in fact, identical), and *R* is known as the characteristic gas constant, the value of which depends upon a particular gas. However, this equation is also valid for the real gases which are thermodynamically far from their liquid phase. For air, the value of R is 287 J/kg K.

K and E generally depend on the nature of process

Distinction between an Incompressible and a Compressible Flow

In order to know, if it is necessary to take into account the compressibility of gases in fluid flow problems, we need to consider whether the change in pressure brought about by the fluid motion causes large change in volume or density.
Using Bernoulli's equation

$p + (1/2)\rho V^2$ = constant (*V* being the velocity of flow), change in pressure, Δp, in a flow field, is of the order of $(1/2)\rho V^2$ (dynamic head).

Invoking this relationship into

$$E = \lim_{\delta\rho \to 0}\left(\frac{\Delta p}{\Delta\rho/\rho}\right) = \rho\frac{dp}{d\rho}$$

we get ,

$$\frac{\Delta\rho}{\rho} \approx \frac{1}{2}\frac{\rho V^2}{E}$$

So if Δρ/ρ is very small, the flow of gases can be treated as incompressible with a good degree of approximation.

According to Laplace's equation, the velocity of sound is given

$$\frac{\Delta\rho}{\rho} \approx \frac{1}{2}\frac{V^2}{a^2} \approx \frac{1}{2}Ma^2$$

where, Ma is the ratio of the velocity of flow to the acoustic velocity in the flowing medium at the condition and is known as **Mach number**. So we can conclude that the compressibility of gas in a flow can be neglected if $\Delta\rho/\rho$ is considerably smaller than unity, i.e. $(1/2)Ma^2 << 1$.

In other words, if the flow velocity is small as compared to the local acoustic velocity, compressibility of gases can be neglected. **Considering a maximum relative change in density of 5 per cent as the criterion of an incompressible flow, the upper limit of Mach number becomes approximately 0.33**. In the case of air at standard pressure and temperature, the acoustic velocity is about 335.28 m/s. Hence a Mach number of 0.33 corresponds to a velocity of about 110 m/s. Therefore flow of air up to a velocity of 110 m/s under standard condition can be considered as incompressible flow.

Surface Tension of Liquids

The phenomenon of surface tension arises due to the two kinds of intermolecular forces

Cohesion :

The force of attraction between the molecules of a liquid by virtue of which they are bound to each other to remain as one assemblage of particles is known as the force of cohesion. This property enables the liquid to resist tensile stress.

Adhesion :

The force of attraction between unlike molecules, i.e. between the molecules of different liquids or between the molecules of a liquid and those of a solid body when they are in contact with each other, is known as the force of adhesion. This force enables two different liquids to adhere to each other or a liquid to adhere to a solid body or surface.

A and B experience equal force of cohesion in all directions, C experiences a net force interior of the liquid The net force is maximum for D since it is at surface

Work is done on each molecule arriving at surface against the action of an inward force. Thus mechanical work is performed in creating a free surface or in increasing the area of the surface. Therefore, a surface requires mechanical energy for its formation and the existence of a free surface implies the presence of stored mechanical energy known as free surface energy. Any system tries to attain the condition of stable equilibrium with its potential energy as minimum. Thus a quantity of liquid will adjust its shape until its surface area and consequently its free surface energy is a minimum.

The magnitude of surface tension is defined as the tensile force acting across imaginary short and straight elemental line divided by the length of the line.

The dimensional formula is F/L or MT^{-2} . It is usually expressed in N/m in SI units.

Surface tension is a binary property of the liquid and gas or two liquids which are in contact with each other and defines the interface. It decreases slightly with increasing temperature. The surface tension of water in contact with air at 20°C is about 0.073 N/m.

It is due to surface tension that a curved liquid interface in equilibrium results in a greater pressure at the concave side of the surface than that at its convex side.

Capillarity

The interplay of the forces of cohesion and adhesion explains the phenomenon of capillarity. When a liquid is in contact with a solid, if the forces of adhesion between the molecules of the liquid and the solid are greater than the forces of cohesion among the liquid molecules themselves, the liquid molecules crowd towards the solid surface. The area of contact between the liquid and solid increases and the liquid thus wets the solid surface. The reverse phenomenon takes place when the force of cohesion is greater than the force of adhesion. These adhesion and cohesion properties result in the phenomenon of capillarity by which a liquid either rises or falls in a tube dipped into the liquid depending upon whether the force of adhesion is more than that of cohesion or not (Fig.2.4) The angle θ as shown in Fig. 2.4, is the area wetting contact angle made by the interface with the solid surface.

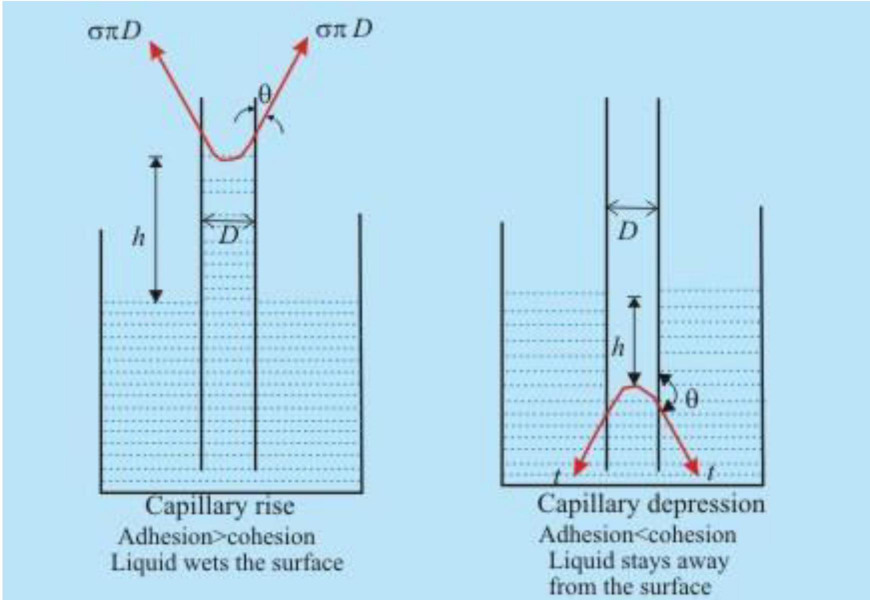


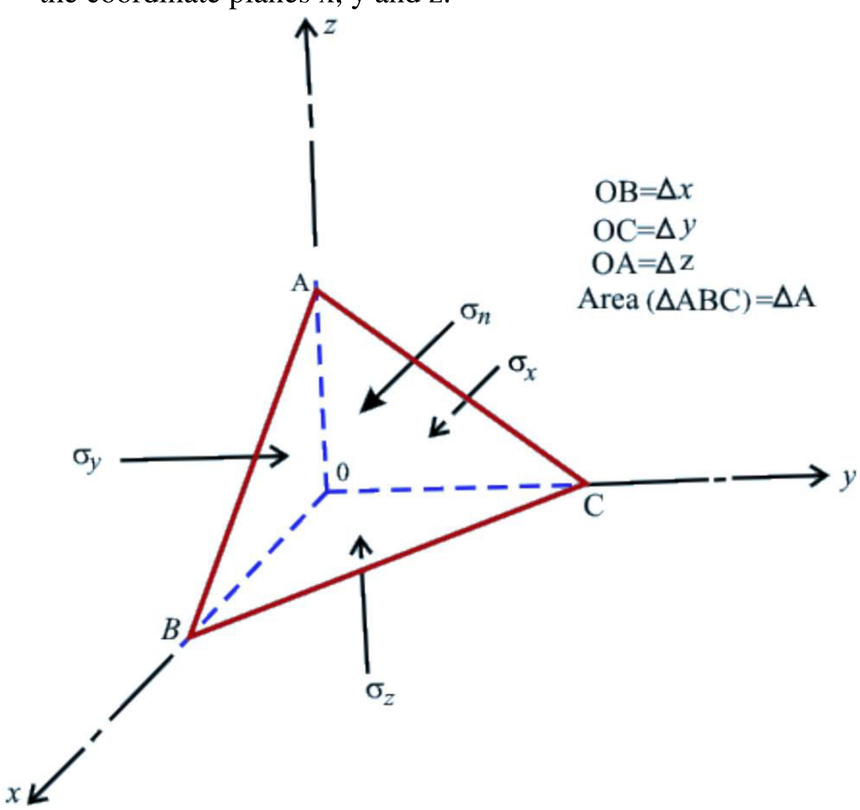
Fig 2.4 Phenomenon of Capillarity

For pure water in contact with air in a clean glass tube, the capillary rise takes place with $\theta = 0$. Mercury causes capillary depression with an angle of contact of about 130^0 in a clean glass in contact with air. Since *h* varies inversely with D as found from Eq.

($h = \dfrac{4\sigma\cos\theta}{\rho g D}$), an appreciable capillary rise or depression is observed in tubes of small diameter only

Normal Stress in a Stationary Fluid

Consider a stationary fluid element of tetrahedral shape with three of its faces coinciding with the coordinate planes x, y and z.



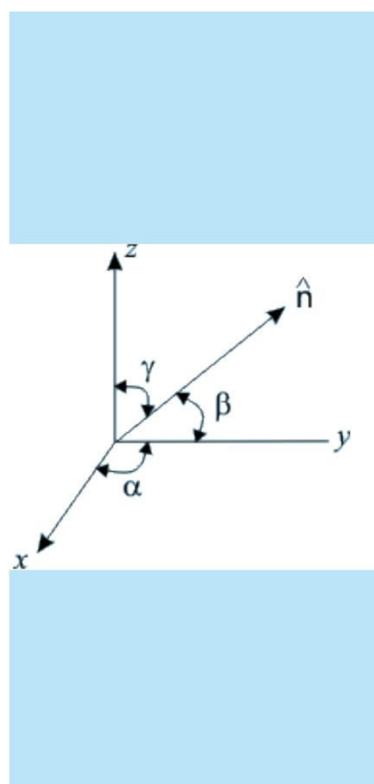


Fig 3.1 State of Stress in a Fluid Element at Rest

Since a fluid element at rest can develop neither shear stress nor tensile stress, the normal stresses acting on different faces are compressive in nature.

Suppose, ΣF_x, ΣF_y and ΣF_z are the net forces acting on the fluid element in positive x,y and z directions respectively. The direction cosines of the normal to the inclined plane of an area ΔA are cos α, cos β and cos .Considering gravity as the only source of external body force, acting in the -ve z direction, the equations of static equlibrium for the tetrahedronal fluid element can be written as

$$\sum F_x = \sigma_x \left(\frac{\Delta y \Delta z}{2}\right) - \sigma_n \Delta A \cos\alpha = 0 \qquad (3.1)$$

$$\sum F_y = \sigma_y \left(\frac{\Delta x \Delta z}{2}\right) - \sigma_n \Delta A \cos\beta = 0 \qquad (3.2)$$

$$\sum F_z = \sigma_z \left(\frac{\Delta y \Delta x}{2}\right) - \sigma_n \Delta A \cos\gamma - \frac{\rho g}{6}(\Delta x \Delta y \Delta z) = 0 \qquad (3.3)$$

where $\left(\frac{\Delta x\, \Delta y\, \Delta z}{6}\right)$ = Volume of tetrahedral fluid element

Pascal's Law of Hydrostatics

Pascal's Law

The normal stresses at any point in a fluid element at rest are directed towards the point from all directions and they are of the equal magnitude.

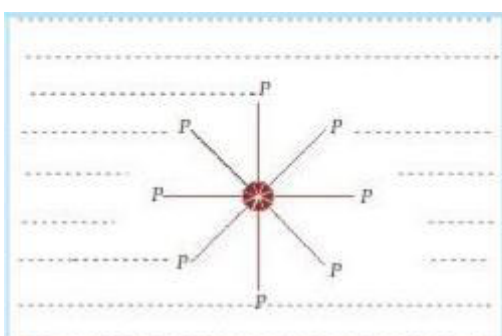


Fig 3.2 State of normal stress at a point in a fluid body at rest

Derivation:

$$\Delta A \cos \gamma = \left(\frac{\Delta x \Delta y}{2}\right)$$

The inclined plane area is related to the fluid elements (refer to Fig 3.1) as follows

$$\Delta A \cos \alpha = \left(\frac{\Delta y \Delta z}{2}\right)$$

Substituting above values in equation 3.1- 3.3 we get

$$\sigma_x = \sigma_y = \sigma_z = \sigma_n$$

Conclusion:

The state of normal stress at any point in a fluid element at rest is same and directed towards the point from all directions. These stresses are denoted by a scalar quantity p defined as the hydrostatic or thermodynamic pressure.

Using "+" sign for the tensile stress the above equation can be written in terms of pressure as

$$\sigma_x = \sigma_y = \sigma_z = -p$$

Units and scales of Pressure Measurement

Pascal (N/m^2) is the unit of pressure .

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

The absolute pressure: It is the difference between the value of the pressure and the absolute zero pressure.

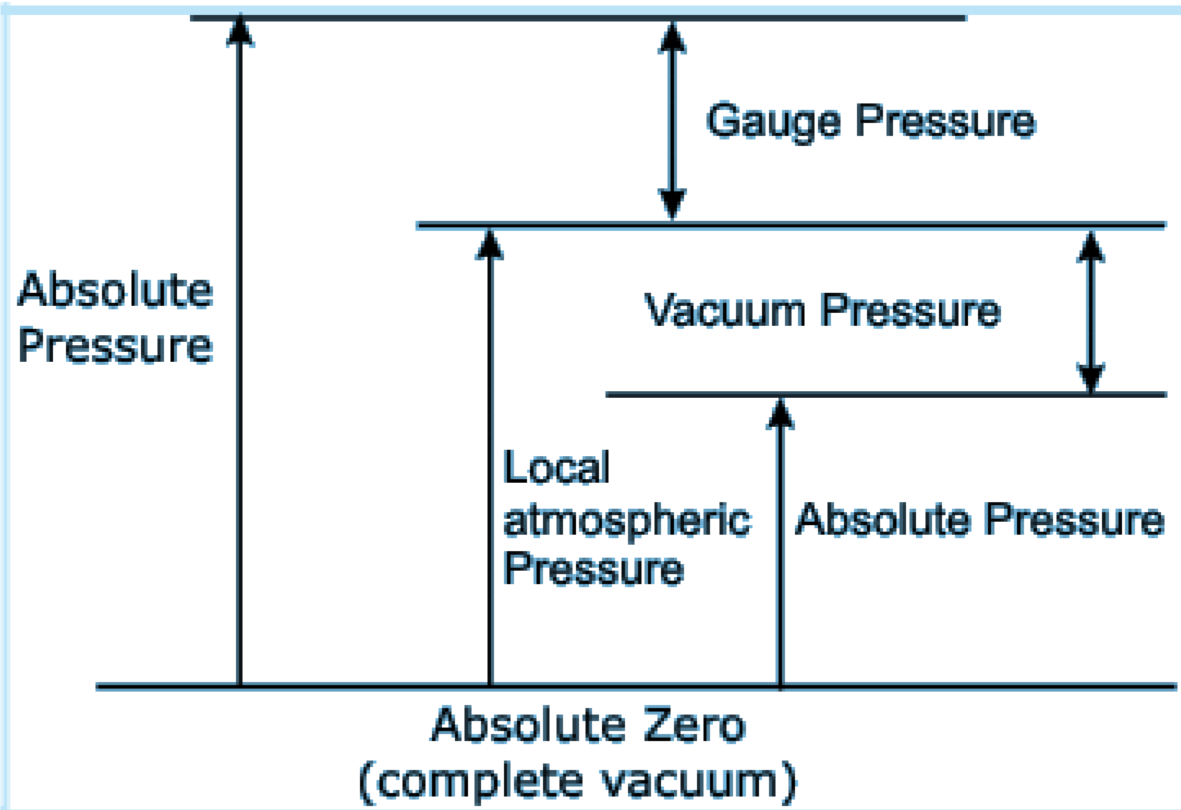
Gauge pressure: It is the diference between the value of the pressure and the local atmospheric pressure(p_{atm})

$$p_{abs} = p - 0 = p$$

$$p_{gauge} = p - p_{atm}$$

Vacuum Pressure: If $p<p_{atm}$ then the gauge pressure (p_{gauge}) becomes negative and is called the vacuum pressure.But one should always remember that hydrostatic pressure is always compressive in nature

The Scale of Pressure



At sea-level, the international standard atmosphere has been chosen as P_{atm} = 101.32 kN/m^2

Hydrostatic Thrusts on Submerged Plane Surface

Due to the existence of hydrostatic pressure in a fluid mass, a normal force is exerted on any part of a solid surface which is in contact with a fluid. The individual forces distributed over an area give rise to a resultant force.

Plane Surfaces

Consider a plane surface of arbitrary shape wholly submerged in a liquid so that the plane of the surface makes an angle θ with the free surface of the liquid. We will assume the case where the surface shown in the figure below is subjected to hydrostatic pressure on one side and atmospheric pressure on the other side.

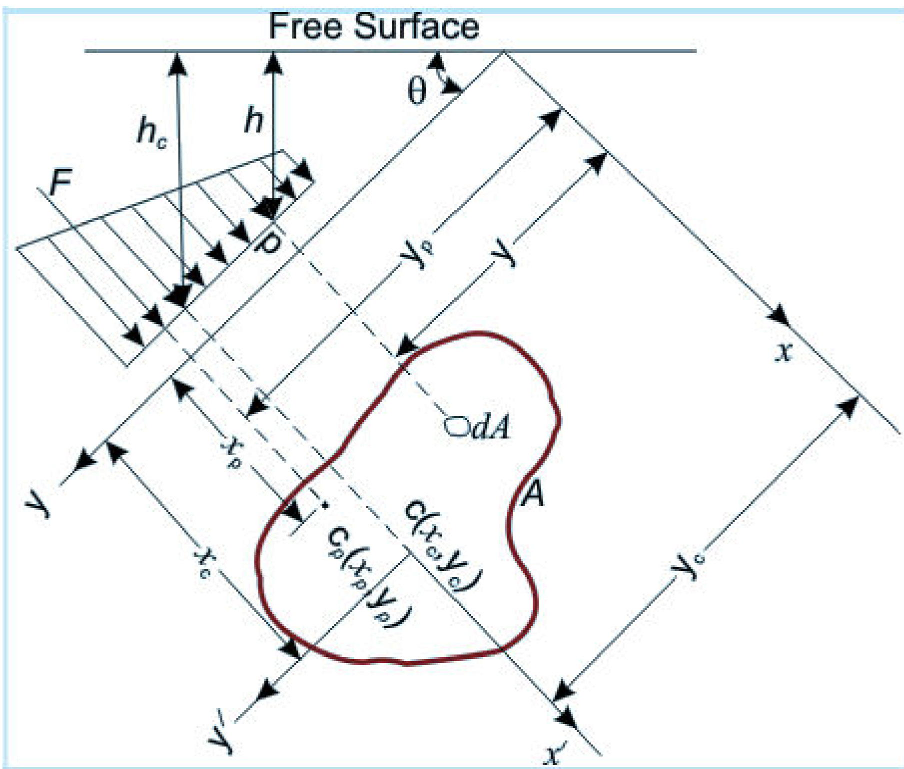


Fig 5.1 Hydrostatic Thrust on Submerged Inclined Plane Surface

Let p denotes the gauge pressure on an elemental area dA. The resultant force F on the area A is therefore

$$F = \iint_A p\, dA \qquad (5.1)$$

According to Eq (3.16a) Eq (5.1) reduces to

$$F = \rho g \iint h dA = \rho g \sin\theta \iint y dA$$

Where **h is the vertical depth of the elemental area dA from the free surface** and the distance y is measured from the x-axis, the line of intersection between the extension of the inclined plane and the free surface (Fig. 5.1). The ordinate of the centre of area of the plane surface A is defined as

Hence from Eqs (5.2) and (5.3), we get

$$F = \rho g y_c \sin\theta A = \rho g h_c A$$

where $h_c (= y_c \sin\theta)$ is the vertical depth (from free surface) of centre c of area .

Equation (5.4) implies that the hydrostatic thrust on an inclined plane is equal to the pressure at its centroid times the total area of the surface, i.e., the force that would have been experienced by the surface if placed horizontally at a depth h_c from the free surface (Fig. 5.2).

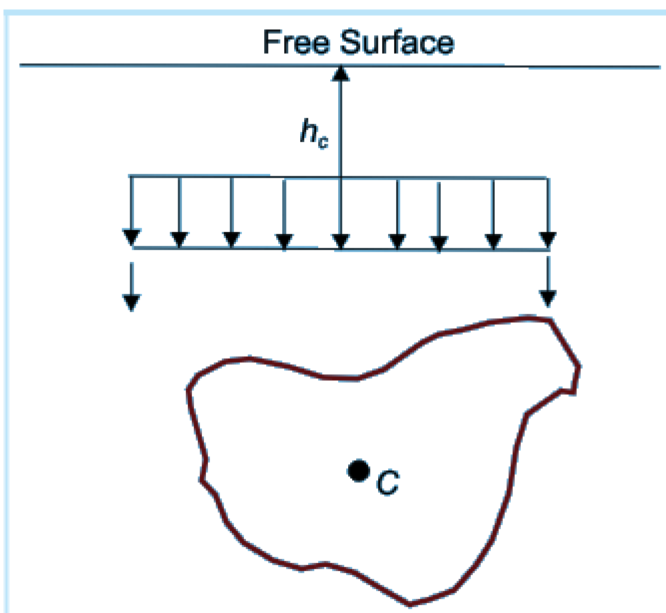


Fig 5.2 Hydrostatic Thrust on Submerged Horizontal Plane Surface

The point of action of the resultant force on the plane surface is called the centre of pressure c_p.

Let x_p and y_p be the distances of the centre of pressure from the y and x axes respectively. Equating the moment of the resultant force about the x axis to the summation of the moments of the component forces, we have

$$y_p F = \int y dF = \rho g \sin\theta \iint y^2 dA \tag{5.5}$$

Solving for y_p from Eq. (5.5) and replacing F from Eq. (5.2), we can write

$$y_p = \frac{\iint_A y^2 dA}{\iint_A y dA} \qquad 5.6)$$

In the same manner, the x coordinate of the centre of pressure can be obtained by taking moment about the y-axis. Therefore,

$$x_p F = \int x dF = \rho g \sin\theta \iint xy dA$$

From which,

$$x_p = \frac{\iint_A xy dA}{\iint_A y dA} \tag{5.7}$$

The two double integrals in the numerators of Eqs (5.6) and (5.7) are the moment of inertia about the x-axis $I_{xx}$and the product of inertia I_{xy} about x and y axis of the plane area respectively. By applying the theorem of parallel axis

$$I_{xx} = \iint y^2 dA = I_{x'x'} + Ay_c^2 \tag{5.8}$$

$$I_{xy} = \iint xy dA = I_{x'y'} + Ax_c y_c \tag{5.9}$$

where, $I_{x'x'}$ and $I_{x'y'}$ are the moment of inertia and the product of inertia of the surface about the centroidal axes $(x'-y'), x_c$, and y_c are the coordinates of the center c of the area with respect to x-y axes.With the help of Eqs (5.8), (5.9) and (5.3), Eqs (5.6) and (5.7) can be written as

$$y_p = \frac{I_{x'x'}}{Ay_c} + y_c$$

$$x_p = \frac{I_{x'y'}}{Ay_c} + x_c \tag{5.10a}$$

The first term on the right hand side of the Eq. (5.10a) is always positive. Hence, the centre of pressure is always at a higher depth from the free surface than that at which the centre of area lies. This is obvious because of the typical variation of hydrostatic pressure with the depth from the free surface. When the plane area is symmetrical about the y' axis, $I_{x'y'} = 0$, and $x_p = x_c$

Hydrostatic Thrusts on Submerged Curved Surfaces

On a curved surface, the direction of the normal changes from point to point, and hence the pressure forces on individual elemental surfaces differ in their directions. Therefore, a scalar summation of them cannot be made. Instead, the resultant thrusts in certain directions are to be determined and these forces may then be combined vector ally. An arbitrary submerged curved surface is shown in Fig. 5.3. A rectangular Cartesian coordinate system is introduced whose xy plane coincides with the free surface of the liquid and z-axis is directed downward below the x - y plane.

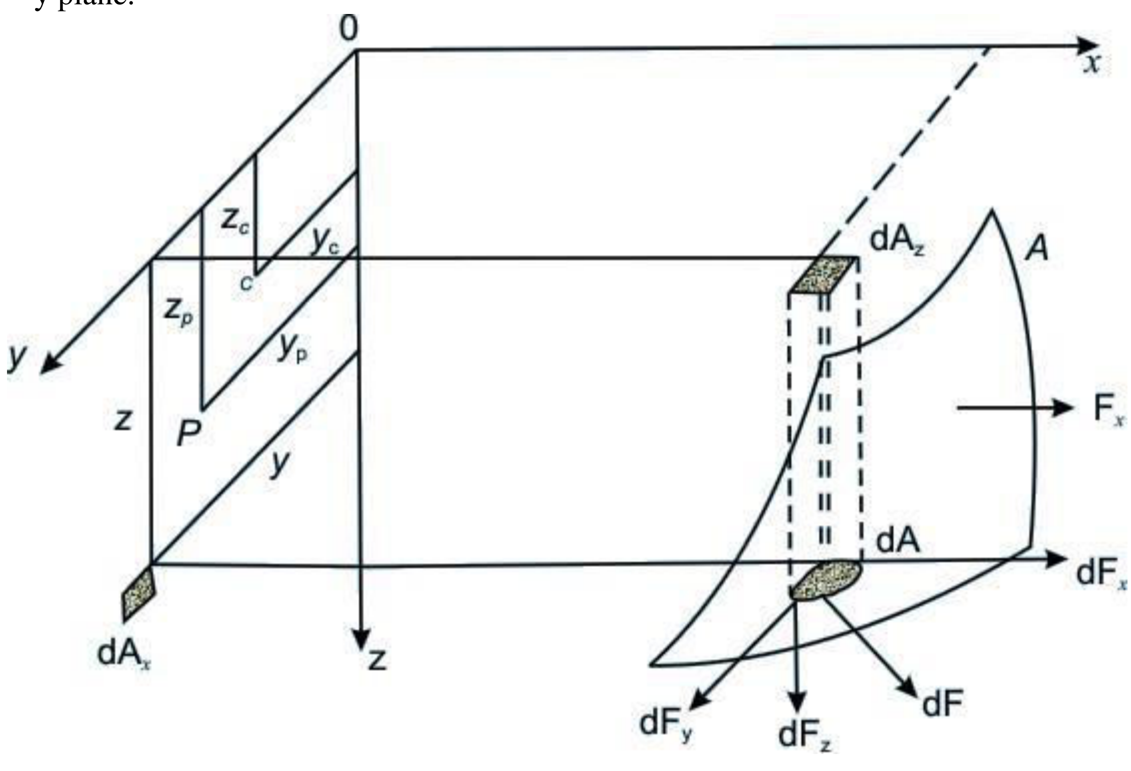


Fig 5.3 Hydrostatic thrust on a Submerged Curved Surface

Consider an elemental area dA at a depth z from the surface of the liquid. The hydrostatic force on the elemental area dA is

$$dF = \rho g z dA$$

and the force acts in a direction normal to the area dA. The components of the force dF in x, y and z directions are

$$dF_x = l dF = l \rho g z dA \qquad (5.12a)$$

$$dF_y = m dF = m \rho g z dA$$

$$dF_z = n dF = n \rho g z dA \qquad (5.12b)$$

(5.13c)

Where *l*, m and n are the direction cosines of the normal to dA. The components of the surface element dA projected on yz, xz and xy planes are, respectively

$$dA_x = l\, dA$$

$$dA_z = n\, dA$$

Substituting Eqs (5.13a-5.13c) into (5.12) we can write

$$dF_x = \rho g z dA_x$$

$$dF_y = \rho g z dA_y$$

$$dF_z = \rho g z dA_z$$

Therefore, the components of the total hydrostatic force along the coordinate axes are

$$F_x = \iint \rho g z dA_x = \rho g z_c A_x$$

$$F_y = \iint \rho g z dA_y = \rho g z_c A_y \qquad (5.15a)$$

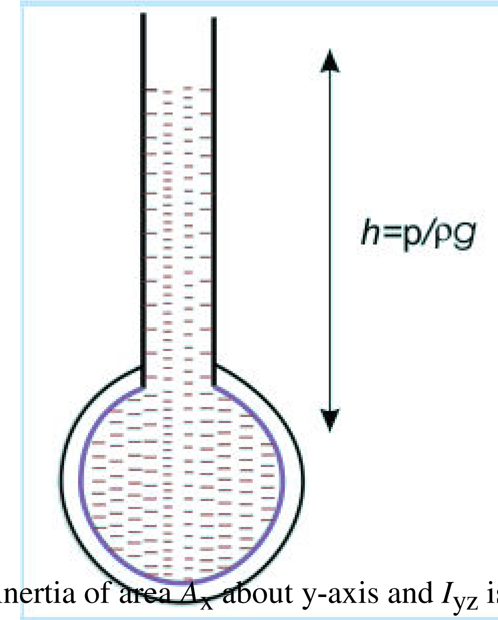
(5.15b)

$$F_z = \iint \rho g z dA_z \qquad (5.15c)$$

where z_c is the z coordinate of the centroid of area A_x and A_y (the projected areas of curved surface on yz and xz plane respectively). If z_p and y_p are taken to be the coordinates of the point of action of F_x on the projected area A_x on yz plane, , we can write

$$z_p = \frac{1}{A_x z_c} \iint z^2 dA_x = \frac{I_{yy}}{A_x z_c}$$

$$y_p = \frac{1}{A_x z_c} \iint yz dA_x = \frac{I_{yz}}{A_x z_c}$$



where I_{yy} is the moment of inertia of area A_x about y-axis and I_{yz} is the product of inertia of A_x with respect to axes y and z. In the similar fashion, $z_p^{'}$ and $x_p^{'}$ the coordinates of the point of action of the force F_y on area A_y, can be written as

where I_{xx} is the moment of inertia of area A_y about x axis and I_{xz} is the product of inertia of A_y about the axes x and z.We can conclude from Eqs (5.15), (5.16) and (5.17) that for a curved surface, the component of hydrostatic force in a horizontal direction is equal to the hydrostatic force on the projected plane surface perpendicular to that direction and acts through the centre of pressure of the projected area. From Eq. (5.15c), the vertical component of the hydrostatic force on the curved surface can be written as

$$F_z = \rho g \iint z \, dA_z = \rho g \forall \qquad (5.18)$$

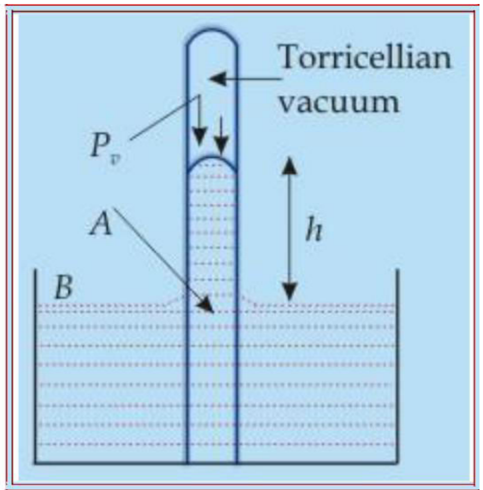
where $\forall$ is the volume of the body of liquid within the region extending vertically above the submerged surface to the free surfgace of the liquid. Therefore, the vertical component of hydrostatic force on a submerged curved surface is equal to the weight of the liquid volume vertically above the solid surface of the liquid and acts through the center of gravity of the liquid in that volume.

Piezometer Tube

The direct proportional relation between gauge pressure and the height h for a fluid of constant density enables the pressure to be simply visualized in terms of the vertical height, $h = p/\rho g$.

The height h is termed as pressure head corresponding to pressure p. For a liquid without a free surface in a closed pipe, the pressure head $p/\rho g$ at a point corresponds to the vertical height above the point to which a free surface would rise, if a small tube of sufficient length and open to atmosphere is connected to the pipe

Such a tube is called a piezometer tube, and the height h is the measure of the gauge pressure of the fluid in the pipe. If such a piezometer tube of sufficient length were closed at the top and the space above the liquid surface were a perfect vacuum, the height of the column would then



Click to play the Demonstration

Fig 4.3 A Simple Barometer

correspond to the absolute pressure of the liquid at the base. This principle is used in the well known mercury barometer to determine the local atmospheric pressure.

The Barometer

Barometer is used to determine the local atmospheric pressure. Mercury is employed in the barometer because its density is sufficiently high for a relative short column to be obtained. and also because it has very small vapour pressure at normal temperature. High density scales down the pressure head(h) to repesent same magnitude of pressure in a tube of smaller height.

A Simple Barometer

Even if the air is completely absent, a perfect vacuum at the top of the tube is never possible. The space would be occupied by the mercury vapour and the pressure would equal to the vapour pressure of mercury at its existing temperature. This almost vacuum condition above the mercury in the barometer is known as Torricellian vacuum.The pressure at A equal to that at B (Fig. 4.3) which is the atmospheric pressure p_{atm} since A and B lie on the same horizontal plane. Therefore, we can write

$$p_B = p_{atm} = p_v + \rho g h$$

The vapour pressure of mercury p_V, can normally be neglected in comparison to p_{atm}.

At 20^0C,P_V is only 0.16 p_{atm}, where p_{atm} =1.0132 X10^5 Pa at sea level. Then we get from Eq. (4.1)

$$h = p_{atm} / \rho g = \frac{1.0132 \times 10^5 \ N/m^2}{(13560 \, kg/m^3)(9.81 N/Kg)} = 0.752 m \ of \ \ Hg$$

For accuracy, small corrections are necessary to allow for the variation of □□with temperature. the thermal expansion of the scale (usually made of brass). and surface tension effects. If water was used instead of mercury, the corresponding height of the column would be about 10.4 m provided that a perfect vacuum could be achieved above the water. However, the vapour pressure of water at ordinary temperature is appreciable and so the actual height at, say, 15°C would be about 180 mm less than this value. Moreover. with a tube smaller in diameter than about 15 mm, surface tension effects become significant.

Manometers for measuring Gauge and Vacuum Pressure

Manometers are devices in which columns of a suitable liquid are used to measure the difference in pressure between two points or between a certain point and the atmosphere.Manometer is needed for measuring large gauge pressures. It is basically the modified form of the piezometric tube. A common type manometer is like a transparent "U-tube" as shown in Fig. 4.4.

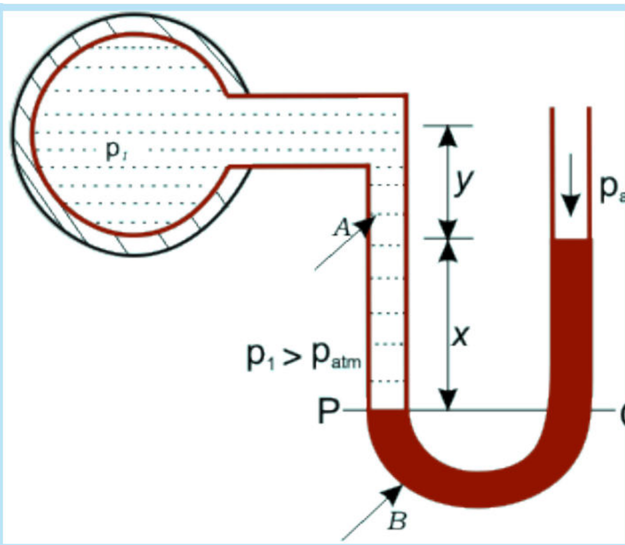


Fig 4.4 A simple manometer to measure gauge pressure

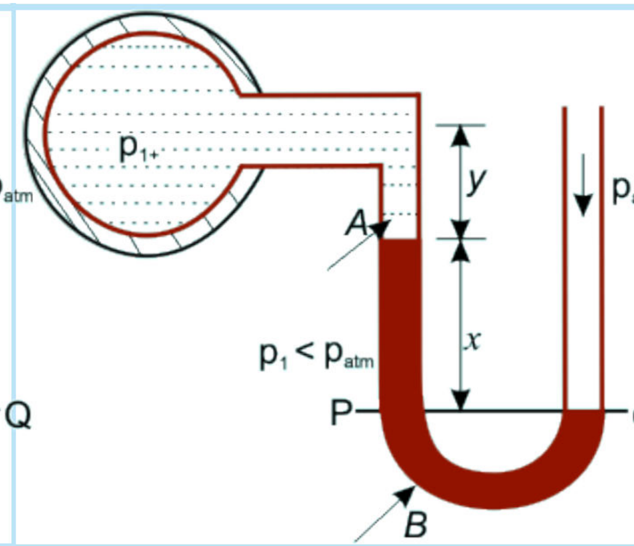


Fig 4.5 A simple manometer to measure vacuum pressure

One of the ends is connected to a pipe or a container having a fluid (A) whose pressure is to be measured while the other end is open to atmosphere. The lower part of the U-tube contains a liquid immiscible with the fluid A and is of greater density than that of A. This fluid is called the manometric fluid.

The pressures at two points *P* and *Q* (Fig. 4.4) in a horizontal plane within the continuous expanse of same fluid (the liquid B in this case) must be equal. Then equating the pressures at *P* and *Q* in terms of the heights of the fluids above those points, with the aid of the fundamental equation of hydrostatics (Eq 3.16), we have

$$p_1 + \rho_A g(y + x) = p_{atm} + \rho_B g x$$

Hence,

$$p_1 - p_{atm} = (\rho_B - \rho_A) g x - \rho_A g y$$

where p_1 is the absolute pressure of the fluid A in the pipe or container at its centre line, and p_{atm} is the local atmospheric pressure. 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 Fig. 4.5.

Manometers to measure Pressure Difference

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

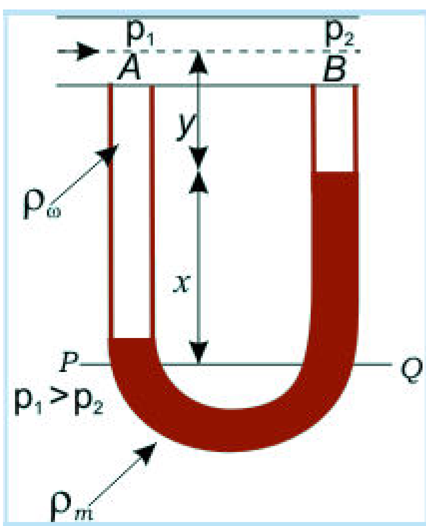


Fig 4.6 Manometer measuring pressure difference

The axis of each connecting tube at A and B should be perpendicular to the direction of flow and also for the edges of the connections to be smooth. Applying the principle of hydrostatics at P and Q we have,

$$p_1 + (y + x)\rho_w g = p_2 + y\,\rho_w g + \rho_m g x$$

$$p_1 - p_2 = (\rho_m - \rho_w) g x$$

where, ρ_m is the density of manometric fluid and ρ_w is the density of the working fluid flowing through the pipe.We can express the difference of pressure in terms of the difference of heads (height of the working fluid at equilibrium).

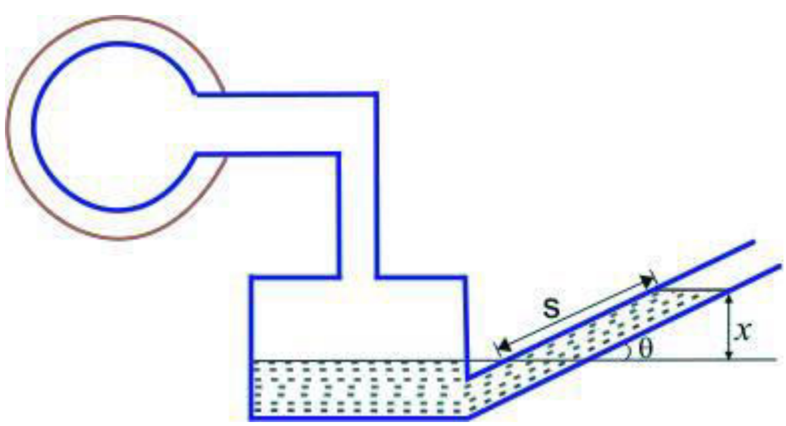
$$h_1 - h_2 = \frac{p_1 - p_2}{\rho_w g} = \left(\frac{\rho_m}{\rho_w} - 1\right) x \qquad (4.4)$$

Inclined Tube Manometer

For accurate measurement of small pressure differences by an ordinary u-tube manometer, it is essential that the ratio $_{m}$ $_{w}$ should be close to unity. □ /□ This is not possible if the working fluid is a gas; also having a manometric liquid of density very close to that of the working liquid and giving at the same time a well defined meniscus at the interface is not always possible. For this purpose, an inclined tube manometer is used.

If the transparent tube of a manometer, instead of being vertical, is set at an angle θ to the horizontal (Fig. 4.7), then a pressure difference corresponding to a vertical difference of along the slope.

levels x gives a movement of the meniscus $s = x/\sin$□



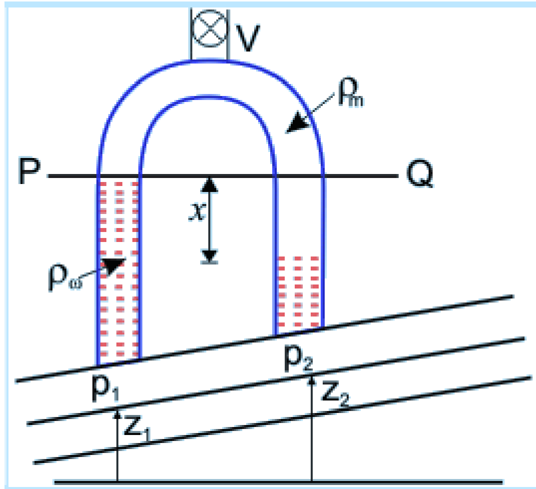


Fig 4.7 An Inclined Tube Manometer

If θ is small, a considerable magnifications of the movement of the meniscus may be achieved. Angles less than 5^0 are not usually satisfactory, because it becomes difficult to determine the exact position of the meniscus.
One limb is usually made very much greater in cross-section than the other. When a pressure difference is applied across the manometer, the movement of the liquid surface in the wider limb is practically negligible compared to that occurring in the narrower limb. If the level of the surface in the wider limb is assumed constant, the displacement of the meniscus in the narrower limb needs only to be measured, and therefore only this limb is required to be transparent.

Inverted Tube Manometer

For the measurement of small pressure differences in liquids, an inverted U-tube manometer is usedHere $\rho_m < \rho_w$ and the line PQ is taken at the level of the higher meniscus to equate the pressures at P and Q from the principle of hydrostatics. It may be written that

$$p_1^* - p_2^* = (\rho_w - \rho_m)gx$$

where p^* represents the **piezometric pressure**, $p + \rho gz$ (z being the vertical height of the point concerned from any reference datum). In case of a horizontal pipe ($z_1 = z_2$) the difference inpiezometric pressure becomes equal to the difference in the static pressure. If $(\rho_w - \rho_m)$ issufficiently small, a large value of x may be obtained for a small value of $p_1^* - p_2^*$. Air is used asthe manometric fluid. Therefore, ρ_m is negligible compared with ρ_w and hence,

$$p_1^* - p_2^* \approx \rho_w gx \qquad (4.5)$$

Air may be pumped through a valve V at the top of the manometer until the liquid menisci are at a suitable level.

Micromanometer

When an additional gauge liquid is used in a U-tube manometer, a large difference in meniscus levels may be obtained for a very small pressure difference.

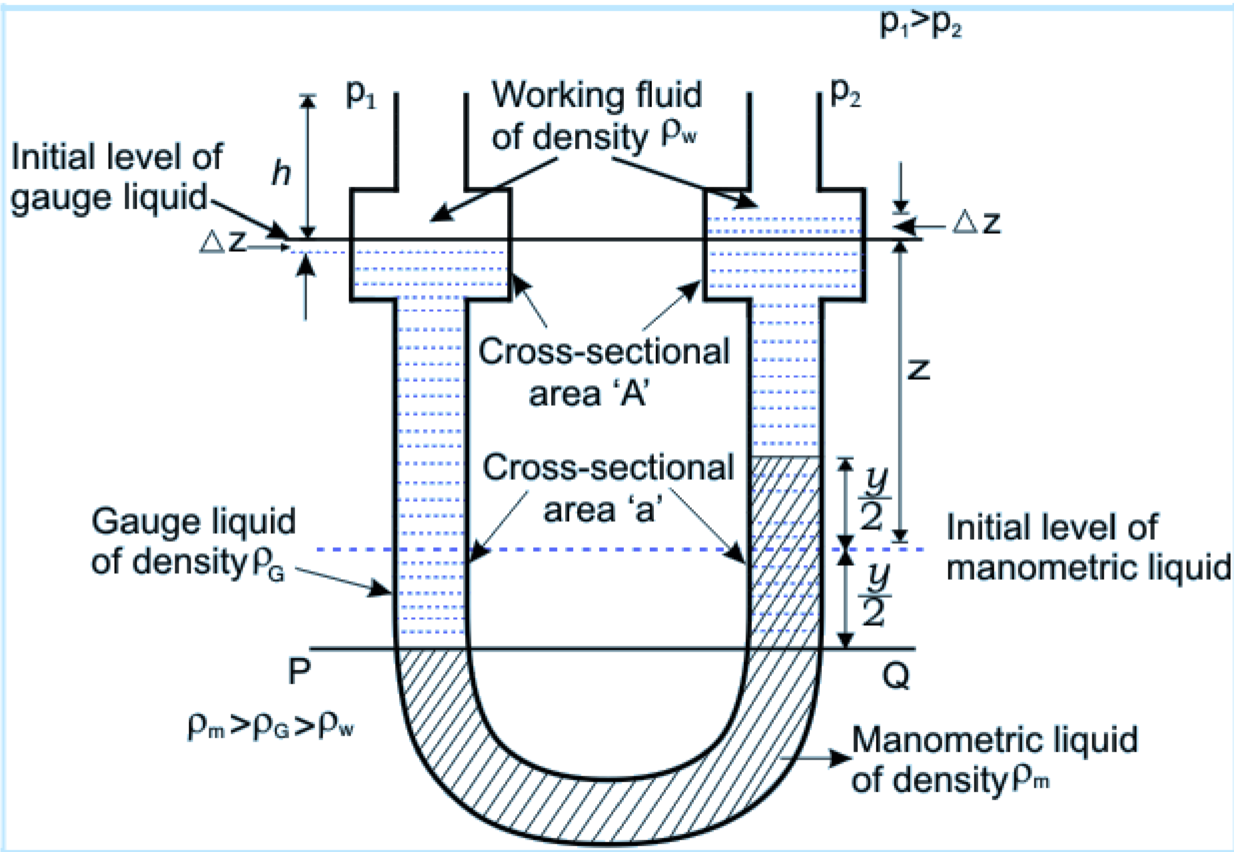


Fig 4.9 A Micromanometer

The equation of hydrostatic equilibrium at PQ can be written as

$$p_1 + \rho_w g(h + \Delta z) + \rho_G g\left(z - \Delta z + \frac{y}{2}\right) = p_2 + \rho_w g(h - \Delta z) + \rho_G g\left(z + \Delta z - \frac{y}{2}\right) + \rho_m g y$$

where ρ_w, ρ_G and ρ_m are the densities of working fluid, gauge liquid and manometric liquid respectively.From continuity of gauge liquid,

$$A\Delta z = a\frac{y}{2} \tag{4.6}$$

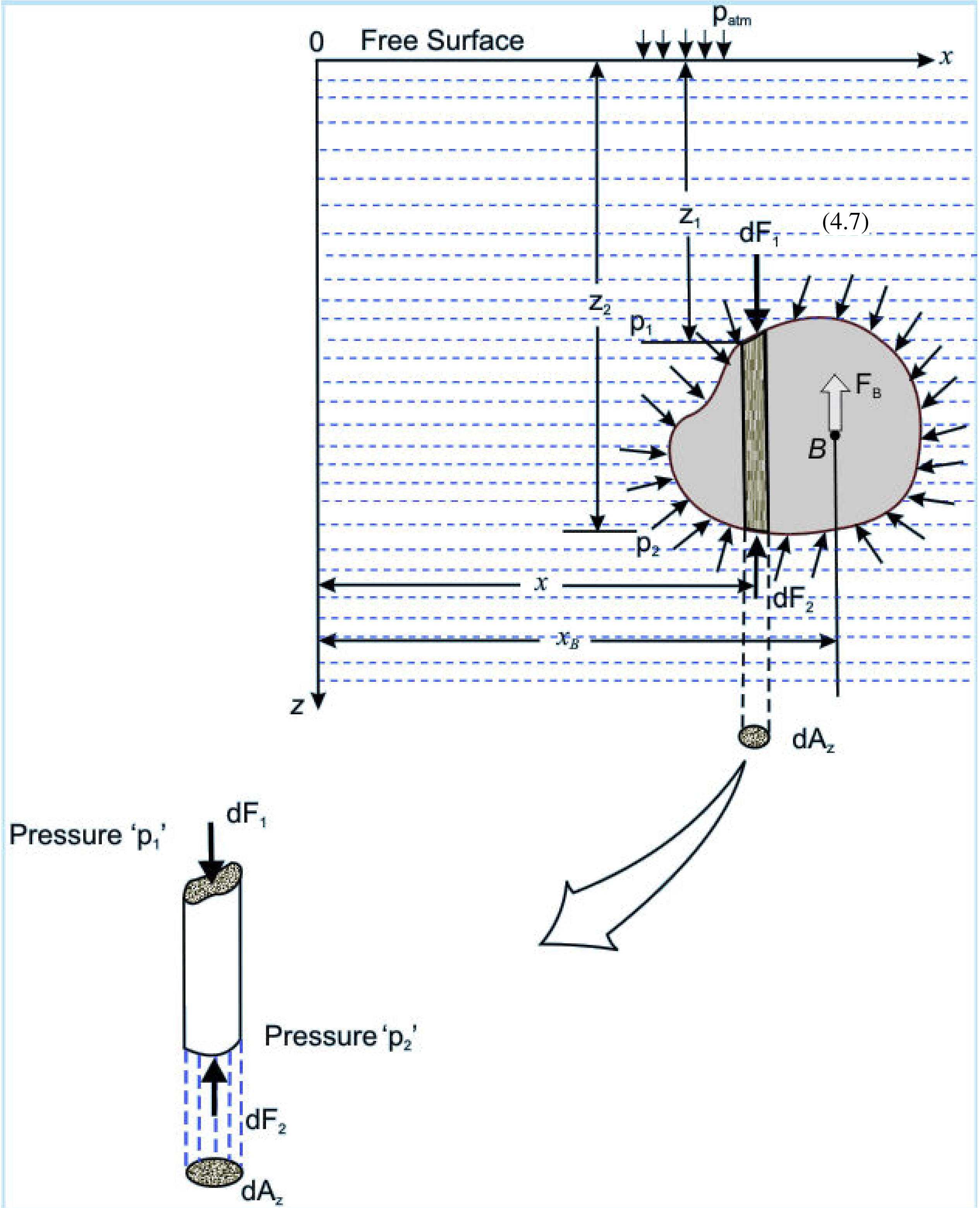


Fig 5.4 Buoyant Force on a Submerged Body

$(\rho_m \approx \rho_G)$

If a is very small compared to A $p_1 - p_2 \approx (\rho_m - \rho_G) gy$

With a suitable choice for the manometric and gauge liquids so that their densities are

Close a reasonable value of y may be achieved for a small pressure difference.

Buoyancy

When a body is either wholly or partially immersed in a fluid, a lift is generated due to the net vertical component of hydrostatic pressure forces experienced by the body.This lift is called the buoyant force and the phenomenon is called buoyancyConsider a solid body of arbitrary shape completely submerged in a homogeneous liquid as shown in Fig. 5.4. Hydrostatic pressure forces act on the entire surface of the body.
To calculate the vertical component of the resultant hydrostatic force, the body is considered to be divided into a number of elementary vertical prisms. The vertical forces acting on the two ends of such a prism of cross-section dA_z (Fig. 5.4) are respectively

$$dF_1 = (p_{atm} + p_1) dA_z = (p_{atm} + \rho g z_1) dA_z$$

Therefore, the buoyant force (the net vertically upward force) acting on the elemental prism of volume $d\forall$ is -

$$dF_B = dF_2 - dF_1 = \rho g (z_2 - z_1) dA_z = \rho g d\forall$$

Hence the buoyant force F_B on the entire submerged body is obtained as

$$F_B = \iiint_{\forall} \rho g d\forall = \rho g \forall$$

Where $\forall$ **is the total volume of the submerged body.** The line of action of the force F_B can be found by taking moment of the force with respect to z-axis. Thus

$$x_B F_B = \int x dF_B \qquad (5.21)$$

Substituting for dF_B and F_B from Eqs (5.19c) and (5.20) respectively into Eq. (5.21), the **x coordinate of the center of the buoyancy is** obtained as

$$x_B = \frac{1}{\forall}\iiint_{\forall} x\,d\forall \qquad (5.22)$$

which is **the centroid of the displaced volume**. It is found from Eq. (5.20) that the buoyant force F_B equals to the weight of liquid displaced by the submerged body of volume $\forall$. This phenomenon was discovered by Archimedes and is known as the Archimedes principle.

ARCHIMEDES PRINCIPLE

The buoyant force on a submerged body

The Archimedes principle states that the buoyant force on a submerged body is equal to the weight of liquid displaced by the body, and acts vertically upward through the centroid of the displaced volume.Thus the net weight of the submerged body, (the net vertical downward force experienced by it) is reduced from its actual weight by an amount that equals the buoyant force.

The buoyant force on a partially immersed body

According to Archimedes principle, the buoyant force of a partially immersed body is equal to the weight of the displaced liquid.

Therefore the buoyant force depends upon the density of the fluid and the submerged volume of the body.

For a floating body in static equilibrium and in the absence of any other external force, the buoyant force must balance the weight of the body.

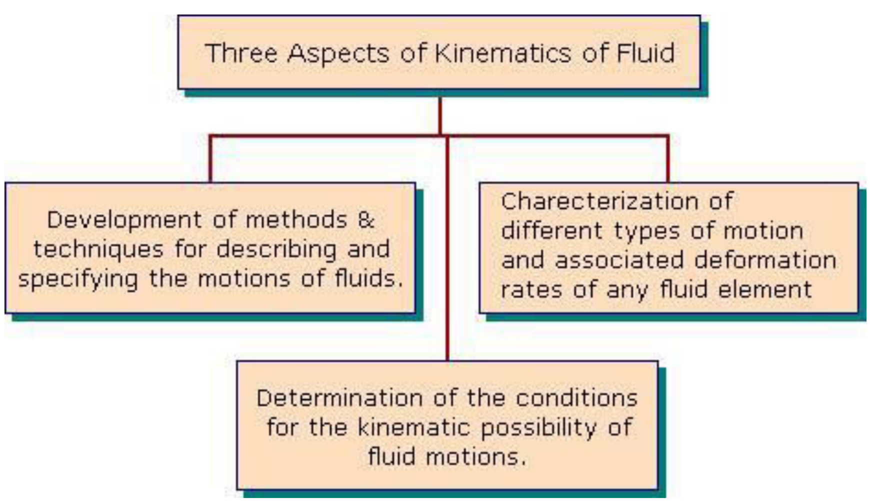
KINEMATICS OF FLUID

Introduction

Kinematics is the geometry of Motion.

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

The subject has **three main aspects:**



Scalar and Vector Fields

Scalar: Scalar is a quantity which can be expressed by a single number representing its **magnitude Example**: mass, density and temperature

Scalar Field

If at every point in a region, a scalar function has a defined value, the region is called a **scalar field**.**Example**: Temperature distribution in a rod.

Vector: Vector is a quantity which is specified by **both magnitude and direction**.

Example: Force, Velocity and Displacement.

Vector Field

If at every point in a region, a vector function has a defined value, the region is called a **vector field**.

Example: velocity field of a flowing fluid **.**

Flow Field

The region in which the flow parameters i.e. velocity, pressure etc. are defined at each and every point at any instant of time is called a **flow field**.

Thus a flow field would be specified by the velocities at different points in the region at different times.

Variation of Flow Parameters in Time and Space

Hydrodynamic parameters like pressure and density along with flow velocity may vary from one point to another and also from one instant to another at a fixed point.

According to **type of variations**, categorizing the flow:

Steady and Unsteady Flow

Steady Flow

A steady flow is defined as a flow in which the various hydrodynamic parameters and fluid properties at any point do not change with time.

In **Eulerian approach**, a steady flow is described as, $\vec{V} = V(\vec{S})$

and $\vec{a} = a(\vec{S})$

Implications: Velocity and acceleration are functions of space coordinates only.In a steady flow, the hydrodynamic parameters may vary with location, but the spatial distribution of these parameters remain invariant with time. In the **Lagrangian approach**,Time is inherent in describing the trajectory of any particle. In steady flow, the velocities of all particles passing through any fixed point at different times will be same. Describing velocity as a function of time for a given particle will show the velocities at different points through which the particle has passed providing the information of velocity as a function of spatial location as described by **Eulerian method**. Therefore, the Euclidian and Lagrangian approaches of describing fluid motion become identical under this situation.

Unsteady Flow

An unsteady Flow is defined as a flow in which the hydrodynamic parameters and fluid properties changes with time.

Uniform and Non-uniform Flows

Uniform Flow

The flow is defined as uniform flow when in the flow field the **velocity and other hydrodynamic parameters do not change from point to point at any instant of time.**

For a uniform flow, the velocity is a function of time only, which can be expressed in Eulerian description as $\vec{V} = V(t)$

Implication:

For a uniform flow, there will be no spatial distribution of hydrodynamic and other parameters.

Any hydrodynamic parameter will have a unique value in the entire field, irrespective of whether it changes with time − **unsteady uniform flow**

OR

does not change with time − **steady uniform flow.**

Thus ,steadiness of flow and uniformity of flow does not necessarily go together.

Non-Uniform Flow

When the **velocity and other hydrodynamic parameters changes from one point to another** the flow is defined as **non-uniform**.

Important points:

1.For a non-uniform flow, the changes with position may be found either in the direction of flow or in directions perpendicular to it.

2.Non-uniformity in a direction perpendicular to the flow is always encountered near solid boundaries past which the fluid flows.

Reason: All fluids possess **viscosity** which reduces the relative velocity (of the fluid w.r.t. to the wall) to zero at a solid boundary. This is known as **no-slip condition.**

Four possible combinations

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

Streamlines

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

Description: In the **Eulerian** method, the velocity vector is defined as a function of time and space coordinates. If for a fixed instant of time, a **space curve** is drawn so that it is **tangent** everywhere to the **velocity** vector, then this curve is called a **Streamline**. Therefore, the Eulerian method gives a series of instantaneous streamlines of the state of motion (Fig. 7.2a).

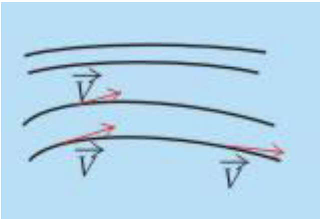


Fig 7.2a Streamlines

Alternative Definition:

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

Comments:

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

In a **steady flow**, the orientation or the pattern of streamlines will be **fixed**.
From the above definition of streamline, it can be written as

$$\vec{V} \times d\vec{S} = 0$$

Description of the terms:

$d\vec{S}$ is the length of an infinitesimal line segment along a streamline at a point .
$\vec{V}$ is the instantaneous velocity vector.
The above expression therefore represents the **differential equation of a streamline**. In a cartesian coordinate-system, representing

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

the above equation (Equation 7.3) may be simplified as

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

Stream tube:

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

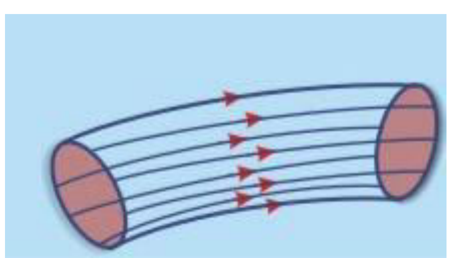


Fig 7.2b Stream Tube

Properties of Stream tube:

The stream-tube is bounded on all sides by streamlines.Fluid velocity does not exist across a streamline, no fluid may enter or leave a stream-tube except through its ends.The entire flow in a flow field may be imagined to be composed of flows through stream-tubes arranged in some arbitrary positions.

Path Lines

Definition: A path line is the trajectory of a fluid particle of **fixed identity** as defined by Eq. (6.1).

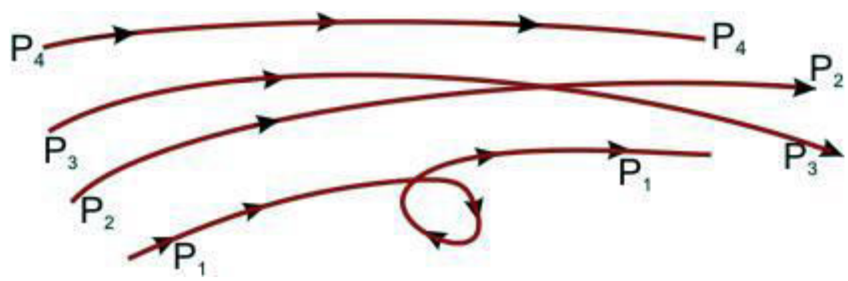


Fig 7.3 Path lines

family of path lines represents the **trajectories of different particles**, say, P_1, P_2, P_3, etc. (Fig. 7.3).

Differences between Path Line and Stream Line

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

Note: In a steady flow path lines are identical tostreamlines as the Eulerian and Lagrangian versionsbecome the same.

Streak Lines

Definition: A streak line is the locus of the temporary locations of all particles that have passed though a fixed point in the flow field at any instant of time.

Features of a Streak Line:

While a path line refers to the identity of a fluid particle, a streak line is specified by a fixed point in the flow field.It is of particular interest in experimental flow visualization.
Example: If dye is injected into a liquid at a fixed point in the flow field, then at a later time t, the dye will indicate the end points of the path lines of particles which have passed through the injection point.
The equation of a streak line at time t can be derived by the **Lagrangian method**.If a fluid particle $(\vec{S}_o)$ passes through a fixed point $(\vec{S}_1)$ in course of time t, then the Lagrangian method of description gives the equation

$$S(\vec{S}_o, t) = \vec{S}_1$$

If the positions $(\vec{S}_o)$ of the particles which have passed through the fixed point $(\vec{S}_1)$ are determined, then **a streak line** can be drawn through these points.

Equation: The equation of the streak line at a time t is given by

$$\vec{S} = f(\vec{S}_o, t)$$

Substituting Eq. (7.5) into Eq. (7.6) we get the **final form of equation of the streak line**,

$$\vec{S} = f\left[F(\vec{S}_1, t), t\right]$$

One, Two and Three Dimensional Flows

Fluid flow is **three-dimensional** in nature.
This means that the flow parameters like velocity, pressure and so on vary in all the three coordinate directions.Sometimes simplification is made in the analysis of different fluid flow problems by:Selecting the appropriate coordinate directions so that appreciable variation of the hydro dynamic parameters take place in only two directions or even in only one.

One-dimensional flow

All the flow parameters may be expressed as functions of time and one space coordinate only. The single space coordinate is usually the distance measured along the centre-line (not necessarily straight) in which the fluid is flowing.
Example: the flow in a pipe is considered one-dimensional when variations of pressure and velocity occur along the length of the pipe, but any variation over the cross-section is assumed negligible.In reality, flow is never one-dimensional because **viscosity** causes the velocity to decrease to zero at the solid boundaries.

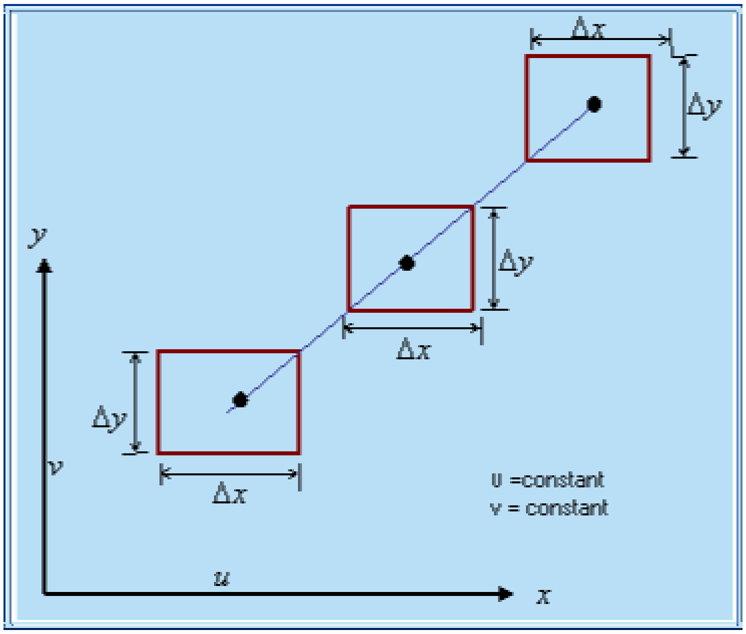


Fig 8.1 Fluid Element in pure translation

If however, the **non uniformity of the actual flow is not too great**, valuable results may often be obtained from a "**one dimensional analysis**".

The **average values** of the flow parameters at any given section (perpendicular to the flow) are assumed to be applied to the entire flow at that section.

Two-dimensional flow

All the flow parameters are functions of time and two space coordinates (say x and y).

No variation in z direction.The same streamline patterns are found in all planes perpendicular to z direction at any instant.

Three dimensional flow

The hydrodynamic parameters are functions of three space coordinates and time.

Translation of a Fluid Element

The movement of a fluid element in space has three distinct features simultaneously.

Translation

Rate of deformation

Rotation.

Figure 7.4 shows the picture of a pure translation in absence of rotation and deformation of a fluid element in a two-dimensional flow described by a rectangular cartesian coordinate system.

In absence of deformation and rotation,

There will be no change in the length of the sides of the fluid element.

There will be no change in the included angles made by the sides of the fluid element.

The sides are displaced in parallel direction.

This is possible when the flow velocities u (the x component velocity) and v (the y component velocity) are neither a function of x nor of y, i.e., the flow field is totally**uniform**.If a component of flow velocity becomes the function of only **one space coordinate** along which that velocity component is defined.

For example,

if $u = u(x)$ and $v = v(y)$, the fluid element ABCD suffers a change in its linear dimensions along with translation

there is no change in the included angle by the sides as shown in Fig. 7.5.

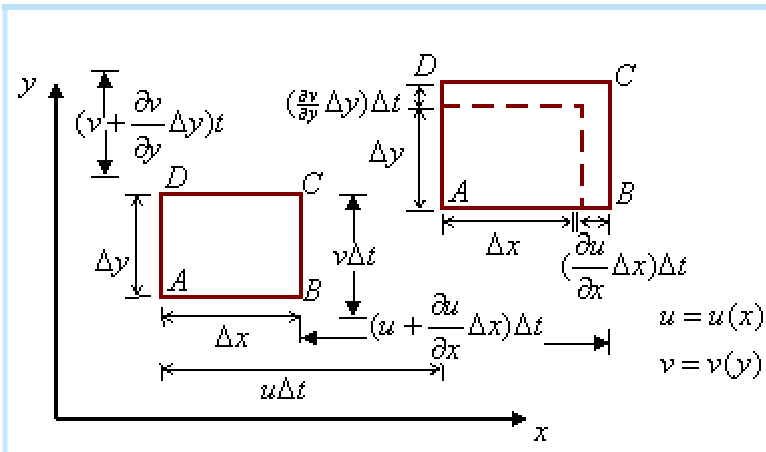


Fig 8.2 Fluid Element in Translation with Continuous Linear Deformation

The relative displacement of point B with respect to point A per unit time in x direction is

$$\frac{\partial u}{\partial x}\Delta x$$

Similarly, the relative displacement of D with respect to A per unit time in y direction is

$$\frac{\partial v}{\partial y}\Delta y$$

Translation with Linear Deformations

Observations from the figure:

Since u is not a function of y and v is not a function of x

All points on the linear element AD move with same velocity in the x direction.

All points on the linear element AB move with the same velocity in y direction.
Hence the sides move parallel from their initial position without changing the included angle. This situation is referred to as **translation with linear deformation**.

Strain rate:

The changes in lengths along the coordinate axes per unit time per unit original lengths are defined as the **components of linear deformation or strain rate in the respective directions**.

Therefore, **linear strain rate** component in the x direction

$$\dot{\varepsilon}_{xx} = \frac{\partial u}{\partial x}$$

and, linear strain rate component in y direction

$$\dot{\varepsilon}_{yy} = \frac{\partial v}{\partial y}$$

Rate of Deformation in the Fluid Element

Let us consider both the velocity component u and v are functions of x and y, i.e.,

$$u = u(x,y)$$

$$v = v(x,y)$$

Figure 8.3 represent the above condition

Observations from the figure: Point B has a relative displacement in y direction with respect to the point A. Point D has a relative displacement in x direction with respect to point A. The included angle between AB and AD changes. The fluid element suffers a continuous angular deformation along with the linear deformations in course of its motion.

Rate of Angular deformation:

The rate of angular deformation is defined as the **rate of change of angle** between the linear segments AB and AD which were initially perpendicular to each other.

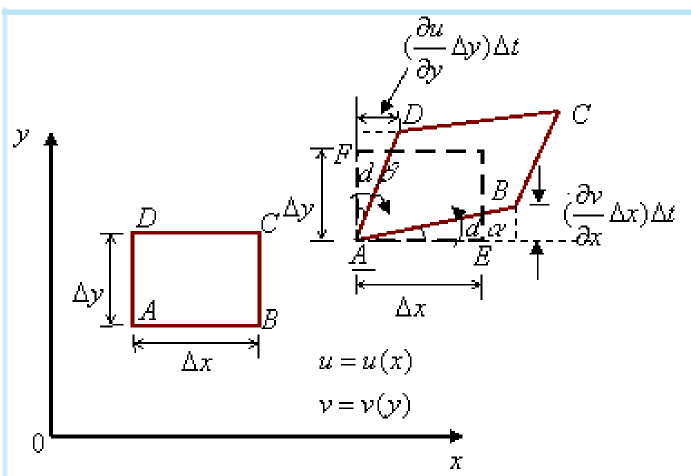


Fig 8.3 Fluid element in translation with simultaneous linear and angular deformation rates

From the above figure rate of angular deformation,

$$\dot{\gamma}_{xy} = \left(\frac{d\alpha}{dt} + \frac{d\beta}{dt}\right)$$

From the geometry

$$d\alpha = \frac{\partial v}{\partial x}dt \qquad (8.2a)$$

$$d\beta = \frac{\partial u}{\partial y}dt \qquad (8.2b)$$

Hence, $$d\beta = \lim_{\Delta t \to 0}\left(\frac{\frac{\partial u}{\partial y}\Delta y\,\Delta t}{\Delta y\left(1+\frac{\partial v}{\partial y}\Delta t\right)}\right) = \frac{\partial u}{\partial y}dt$$

$$\frac{d\alpha}{dt} + \frac{d\beta}{dt} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) \qquad (8.3)$$

Finally $$\dot{\gamma}_{xy} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)$$

Rotation

Figure 8.3 represent the situation of rotation

Observations from the figure:

The transverse displacement of B with respect to A and the lateral displacement of D with respect to A (Fig. 8.3) can be considered as the rotations of the linear segments AB and AD about A.This brings the concept of rotation in a flow field.

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

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

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

and $\dfrac{d\beta}{dt}$ respectively.

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

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

or

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

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

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

Special case : Situation of pure Rotation

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

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

Observation:

Vortex line:

If tangent to an imaginary line at a point lying on it is in the direction of the Vorticity vector at that point , the line is a **vortex line**.For an **irrotational** flow, vorticity components are zero.The **general equation** of the **vortex line** can be written as,

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

In a rectangular cartesian cartesian coordinate system, it becomes

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

where, $\Omega_x = 2\omega_x$

$$\Omega_y = 2\omega_y$$

$$\Omega_z = 2\omega_z$$

Vorticity components as vectors:

The vorticity is actually an **anti symmetric tensor** and its three distinct elements transform like the components of a vector in cartesian coordinates.This is the reason for which the vorticity components can be treated as vectors.

Existence of Flow

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

principles of conservation of momentum and energy.

System

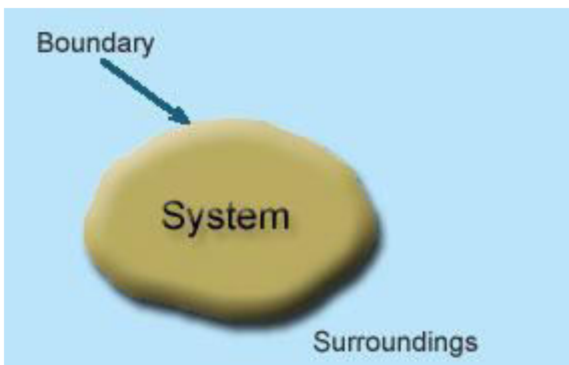


Fig 9.1 System and Surroundings

Definition System: A quantity of matter in space which is analyzed during a problem.

Surroundings: Everything external to the system.

System Boundary: A separation present between system and surrounding.
Classification of the system boundary:-

Real solid boundary

Imaginary boundary

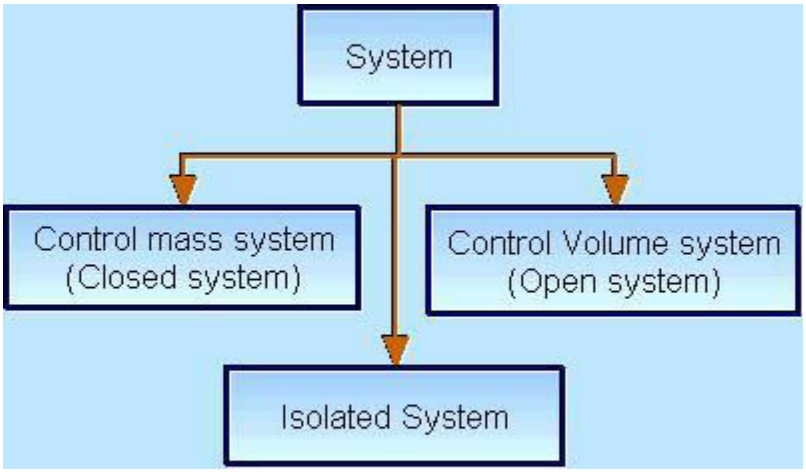
The system boundary may be further classified as:-

Fixed boundary or Control Mass System

Moving boundary or Control Volume System

The choice of boundary depends on the problem being analyzed.

Classification of Systems



Types of System

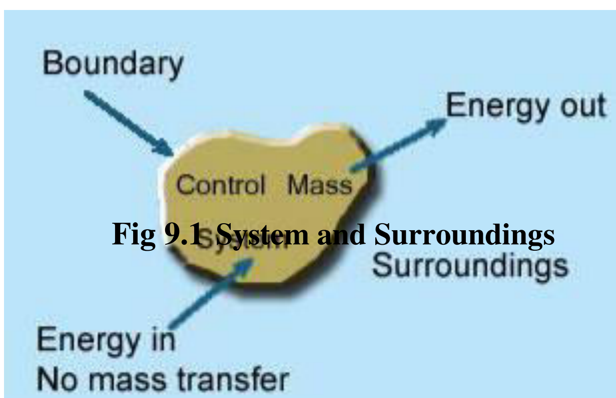


Fig 9.1 System and Surroundings

Fig 9.2 A Control Mass System or Closed System

Control Mass System (Closed System)

Its a system of **fixed mass** with **fixed identity**.
This type of system is usually referred to as **"closed system"**.
There is no mass transfer across the system boundary.
Energy transfer may take place into or out of the system.

[Click to play the Demonstration]

Control Volume System (Open System)

Its a system of **fixed volume**.
This type of system is usually referred to as **"open system"** or a **"control volume"**
Mass transfer can take place across a control volume.
Energy transfer may also occur into or out of the system.
A control volume can be seen as a fixed region across which mass and energy transfers are studied.
Control Surface- Its the boundary of a control volume across which the transfer of both mass and energy takes place.
The mass of a control volume (open system) may or may not be fixed.
When the net influx of mass across the control surface equals zero then the mass of the system is fixed and vice-versa.
The identity of mass in a control volume always changes unlike the case for a control mass system (closed system).
Most of the engineering devices, in general, represent an open system or control volume.
Example:-

Heat exchanger - Fluid enters and leaves the system continuously with the transfer of heat across the system boundary.
Pump - A continuous flow of fluid takes place through the system with a transfer of mechanical energy from the surroundings to the system.

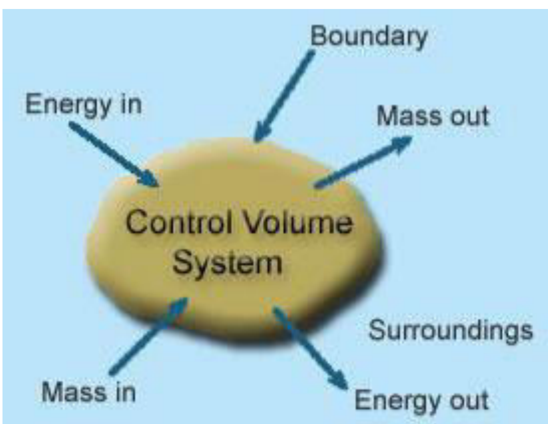


Fig 9.3 A Control Volume System or Open System

[Click to play the Demonstration]

Isolated System

Its a system of **fixed mass** with *same* **identity and fixed energy**.

No interaction of mass or energy takes place between the system and the surroundings. In more informal words an isolated system is like a closed shop amidst a busy market.

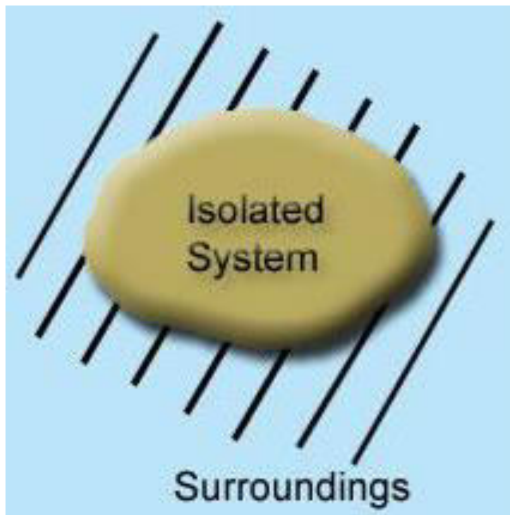


Fig 9.4 An Isolated System

Conservation of Mass - The Continuity Equation

Law of conservation of mass

The law states that *mass can neither be created nor be destroyed*. Conservation of mass is inherent to a control mass system (closed system)

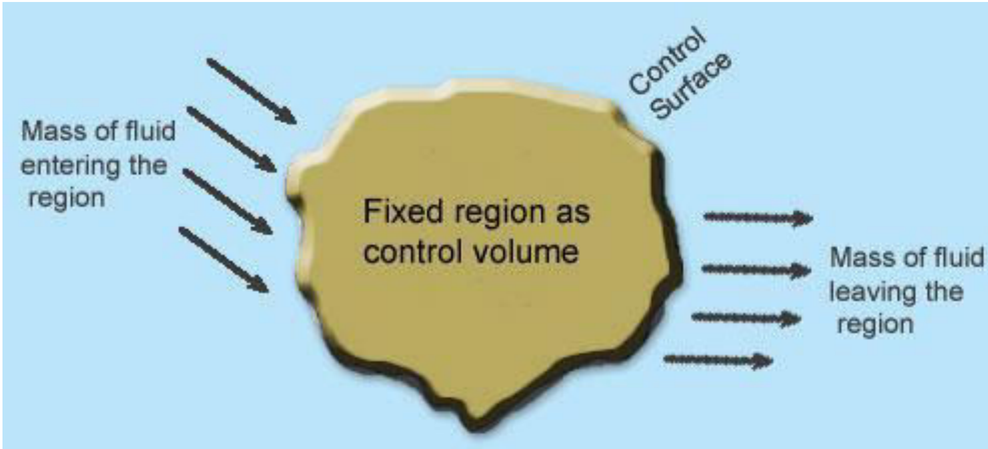
The mathematical expression for the above law is stated as:

$\Delta m/\Delta t = 0$, where m = mass of the system

For a control volume (Fig.9.5), the principle of conservation of mass is stated as Rate at which mass enters = Rate at which mass leaves the region + Rate of accumulation of mass in the region

Continuity equation

The above statement expressed analytically in terms of velocity and density field of a flow is known as the **equation of continuity**.



[Click to play the Demonstration]()

Continuity Equation - Differential Form

Derivation

The point at which the continuity equation has to be derived, is enclosed by an elementary control volume.The influx, efflux and the rate of accumulation of mass is calculated across each surface within the control volume

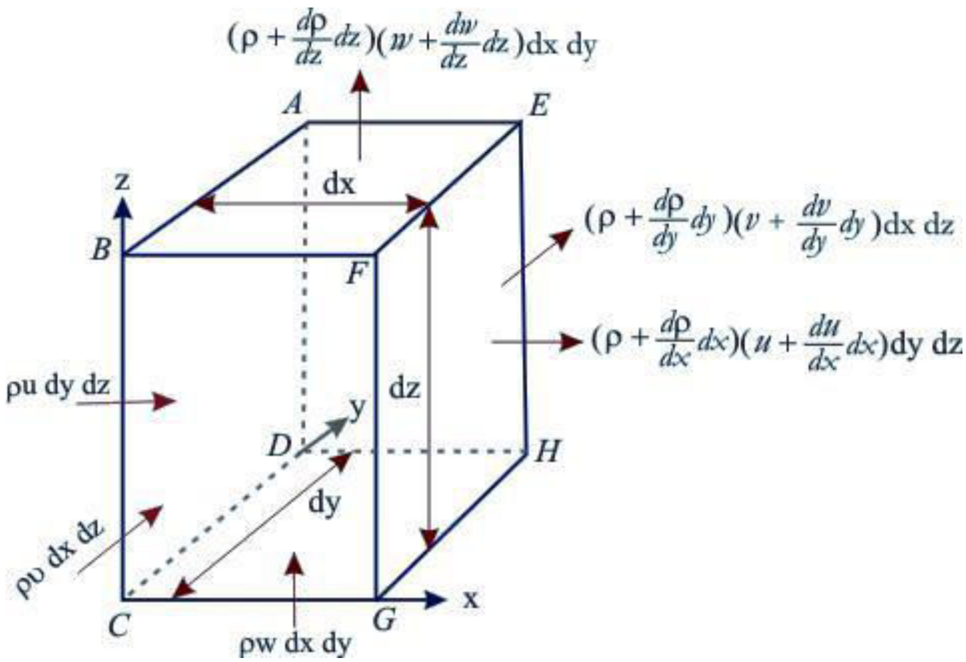


Fig 9.6 A Control Volume Appropriate to a Rectangular Cartesian Coordinate System

Consider a rectangular parallelopiped in the above figure as the control volume in a rectangular

cartesian frame of coordinate axes.Net efflux of mass along x -axis must be the excess outflow over inflow across faces normal to x -axis.Let the fluid enter across one of such faces ABCD with a velocity u and a density ρ.The velocity and density with which the fluid will leave the face EFGH will

be $u+\frac{\partial u}{\partial x}dx$ and $\rho+\frac{\partial \rho}{\partial x}dx$ respectively (neglecting the higher order terms in δx).

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

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

(neglecting the higher order terms in dx)

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

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

Rate of accumulation for a point in a flow field

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

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

$$\rho udydz+\rho vdxdz+\rho wdxdy=\frac{\partial \rho}{\partial t}d\forall+(\rho+\frac{\partial \rho}{\partial x}dx)(u+\frac{\partial u}{\partial x}dx)dydz$$
$$+(\rho+\frac{\partial \rho}{\partial y}dy)(v+\frac{\partial v}{\partial y}dy)dxdz+(\rho+\frac{\partial \rho}{\partial z}dz)(w+\frac{\partial w}{\partial z}dz)dxdy$$

Transferring everything to right side

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

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

Continuity Equation - Vector Form

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

The continuity equation can be written in a vector form as

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

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

where $\vec{V}=u\hat{i}+v\hat{j}+w\hat{k}$ is the velocity of the point

In case of a **steady flow**,
Hence Eq. (9.3) becomes

$$\nabla.(\rho\vec{V})=0 \tag{9.4}$$

In a rectangular cartesian coordinate system

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0 \tag{9.5}$$

Equation (9.4) or (9.5) represents the **continuity equation for a steady flow**.
In case of an incompressible flow,
ρ = constant

Hence,

$$\frac{\partial \rho}{\partial t}=0$$

$$\nabla.(\rho\vec{V})=\rho\nabla.(\vec{V})$$

Therefore, the **continuity equation for an incompressible flow** becomes

$$\nabla.(\vec{V}) = 0$$

$$\text{or,} \quad \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

In cylindrical polar coordinates reduces to

$$\frac{1}{R}\frac{\partial}{\partial R}(R^2 V_R) + \frac{1}{\sin\varphi}\frac{\partial V_\theta}{\partial \theta} + \frac{1}{\sin\varphi}\frac{\partial (V\varphi \sin\varphi)}{\partial \varphi} = 0$$

can be written in terms of the strain rate components as

$$\dot{\varepsilon}_{xx} + \dot{\varepsilon}_{yy} + \dot{\varepsilon}_{zz} = 0$$

Continuity Equation - A Closed System Approach

We know that the conservation of mass is inherent to the definition of a closed system as Dm/Dt = 0 (where m is the mass of the closed system).

However, the general form of continuity can be derived from the basic equation of mass conservation of a system.

Derivation :-

Let us consider an elemental closed system of volume V and density ρ.

$$\frac{Dm}{Dt} = 0 \Rightarrow \frac{D}{Dt}(\rho \Delta \forall) = 0$$

$$\Rightarrow \Delta \forall \frac{D\rho}{Dt} + \rho \frac{D(\Delta \forall)}{Dt} = 0$$

$$\Rightarrow \frac{D\rho}{Dt} + \frac{\rho}{\Delta \forall}\frac{D(\Delta \forall)}{Dt} = 0$$

$$\Rightarrow \frac{\partial \rho}{\partial t} + u\frac{\partial \rho}{\partial x} + v\frac{\partial \rho}{\partial y} + w\frac{\partial \rho}{\partial z} + \frac{\rho}{\Delta \forall}\frac{D\Delta \forall}{Dt} = 0$$

Now $\frac{1}{\Delta\forall}\frac{D\Delta\forall}{Dt}=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}$ (dilation per unit volume)

$$\Rightarrow \frac{\partial \rho}{\partial t}+u\frac{\partial \rho}{\partial x}+v\frac{\partial \rho}{\partial y}+w\frac{\partial \rho}{\partial z}+\rho\frac{\partial u}{\partial x}+\rho\frac{\partial v}{\partial y}+\rho\frac{\partial w}{\partial z}=0$$

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

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

In vector notation we can write this as

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

Stream Function

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

$u = u(x, y, t)$

$v = v(x, y, t)$

$w = 0$

The equation of continuity is

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (10.1)$$

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

$$u=\frac{\partial \psi}{\partial y}$$

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

so that it automatically satisfies the equation of continuity (Eq. (10.1)), then the function is known as stream function.

Note that for a **steady flow, ψ is a function of two variables x and y only.**

$\psi(x, y)$ = constant (10.3)

Constancy of ψ on a Streamline

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

$$d\psi = \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy = -vdx + udy$$

The equation of a streamline is given by

It follows that dψ = 0 on a streamline.This implies the value of ψ is constant along a streamline. Therefore, the equation of a streamline can be expressed in terms of stream function as

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

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

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

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

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

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

Stream function for an irrotational flow

In case of a two-dimensional irrotational flow

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

Physical Significance of Stream Funtion ψ

Figure 10.1 illustrates a two dimensional flow.

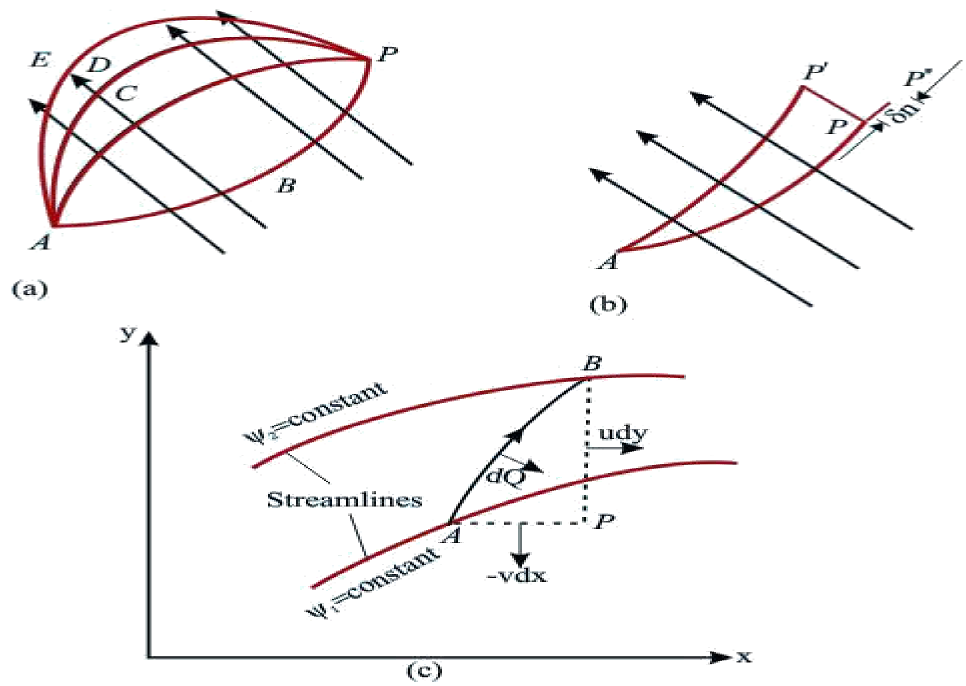


Fig 10.1 Physical Interpretation of Stream Function

Let A be a fixed point, whereas P be any point in the plane of the flow. The points A and P are joined by the arbitrary lines ABP and ACP. For an incompressible steady flow, the volume flow rate across ABP into the space ABPCA (considering a unit width in a direction perpendicular to the plane of the flow) must be equal to that across ACP. A number of different paths connecting A and P (ADP, AEP,...) may be imagined but the volume flow rate across all the paths would be the same. This implies that the **rate of flow across any curve between A and P depends only on the end points A and P.**

Since A is fixed, the rate of flow across ABP, ACP, ADP, AEP (any path connecting A and P) is a function only of the position P. This function is known as the**stream function ψ.**

The value of ψ at P represents the volume flow rate across any line joining P to A.

The value of ψ at A is made arbitrarily zero. If a point P' is considered (Fig. 10.1b),PP' being along a streamline, then the rate of flow across the curve joining A to P' must be the same as across AP, since, by the definition of a streamline, there is no flow across PP'

The value of ψ thus remains same at P' and P. Since P' was taken as any point on the streamline through P, it follows that ψ is constant along a streamline. Thus the flow may be represented by a series of streamlines at equal increments of ψ.

In fig (10.1c) moving from A to B net flow going past the curve AB is

$$\int dQ = \int_A^B (udy - vdx)$$

$$= \int_A^B (\psi_y dy + \psi_x dx) \qquad \left[\text{since } u = \psi_y \text{ and } v = -\psi_x\right]$$

$$\int dQ = \int_A^B d\psi$$

$$\therefore Q = \int_A^B d\psi = \psi_2 - \psi_1$$

The stream function, in a polar coordinate system is defined as

$$V_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \quad \text{and} \quad V_\theta = -\frac{\partial \psi}{\partial r}$$

The expressions for V_r and V_θ in terms of the stream function automatically satisfy the equation of continuity given by

tream Function in Three Dimensional and Compressible Flow

$$\frac{\partial}{\partial r}(V_r r) + \frac{\partial}{\partial \theta}(V_\theta) = 0$$

Stream Function in Three Dimensional Flow

In case of a three dimensional flow, it is not possible to draw a streamline with a single stream function.

An axially symmetric three dimensional flow is similar to the two-dimensional case in a sense that the flow field is the same in every plane containing the axis of symmetry.

The equation of continuity in the cylindrical polar coordinate system for an incompressible flow is given by the following equation

$$\frac{\partial V_r}{\partial r} + \frac{V_r}{r} + \frac{1}{r}\frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} = 0$$

For an axially symmetric flow (the axis r = 0 being the axis of symmetry), the term $\frac{1}{r}\frac{\partial V_\theta}{\partial \theta}$ =0 ,and simplified equation is satisfied by functions defined as

$$rV_r = -\frac{\partial \psi}{\partial z}, \qquad rV_z = \frac{\partial \psi}{\partial r}$$

The function ψ , defined by the Eq.(10.4) in case of a three dimensional flow with an axial symmetry, is called the **stokes stream function.**

Stream Function in Compressible Flow

For compressible flow, stream function is related to mass flow rate instead of volume flow rate because of the extra density term in the continuity equation (unlike incompressible flow)

The continuity equation for a steady two-dimensional compressible flow is given by

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

Hence a stream function ψ is defined which will satisfy the above equation of continuity as

$$\rho u=\rho_o\frac{\partial\psi}{\partial y}$$

$$\rho v=-\rho_o\frac{\partial\psi}{\partial x} \qquad \text{[where } \rho_0 \text{ is a reference density]} \qquad (10.5)$$

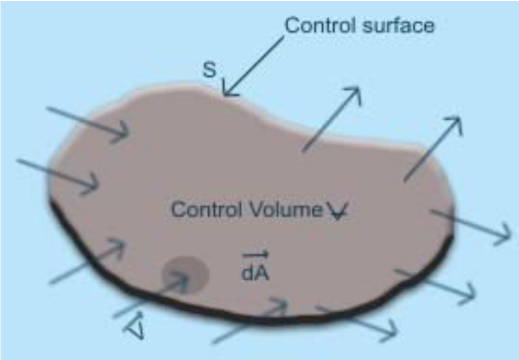
ρ_0 is used to retain the unit of ψ same as that in the case of an incompressible flow. Physically, the difference in stream function between any two streamlines multiplied by the reference density ρ_0 will give the mass flow rate through the passage of unit width formed by the streamlines

Continuity Equation: Integral Form

Let us consider a control volume ∀ bounded by the control surface S. The efflux of mass across the control surface S is given by

$$\iint_S \rho\vec{V}.d\vec{A}$$

where $\vec{V}$ is the velocity vector at an elemental area(which is treated as a vector by considering its positive direction along the normal drawn outward from the surface).



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

Fig 10.2 A Control Volume for the Derivation of Continuity Equation (integral form)

The rate of mass accumulation within the control volume becomes

$$\frac{\partial}{\partial t} \iiint_V \rho . d\forall$$

where d $\forall$ is an elemental volume, ρ is the density and $\forall$ is the total volume bounded by the control surface S. Hence, the continuity equation becomes (according to the statement given by Eq. (9.1))

$$\frac{\partial}{\partial t} \iiint_V \rho . d\forall + \iint_S \rho \vec{V} . d\vec{A} = 0 \qquad (10.6)$$

(10.6)

The second term of the Eq. (10.6) can be converted into a volume integral by the use of the Gauss divergence theorem as

$$\iint_S \rho \vec{V} . d\vec{A} = \iiint_V \nabla.(\rho \vec{V}) d\forall$$

Since the volume $\forall$ does not change with time, the sequence of differentiation and integration in the first term of Eq.(10.6) can be interchanged.

Therefore Eq. (10.6) can be written as

$$\iiint_V \left[\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{V}) \right] d\forall = 0$$

Equation (10.7) is valid for any arbitrary control volume irrespective of its shape and size. So we can write

Definition

System: A quantity of matter in space which is analyzed during a problem.
Surroundings: Everything external to the system.
System Boundary: A separation present between system and surrounding.
Classification of the system boundary:-

Real solid boundary
Imaginary boundary

The system boundary may be further classified as:-

Fixed boundary or Control Mass System

Moving boundary or Control Volume System
The choice of boundary depends on the problem being analyzed.

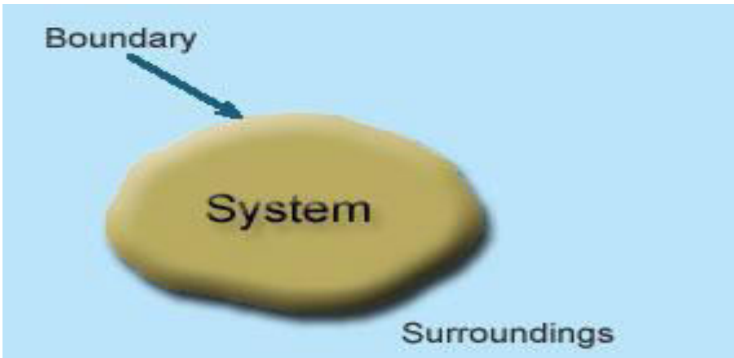


Fig 9.1   System and Surroundings

Types of System

Control Mass System (Closed System)

Its a system of **fixed mass** with **fixed identity**.
This type of system is usually referred to as **"closed system"**.
There is no mass transfer across the system boundary.
Energy transfer may take place into or out of the system.

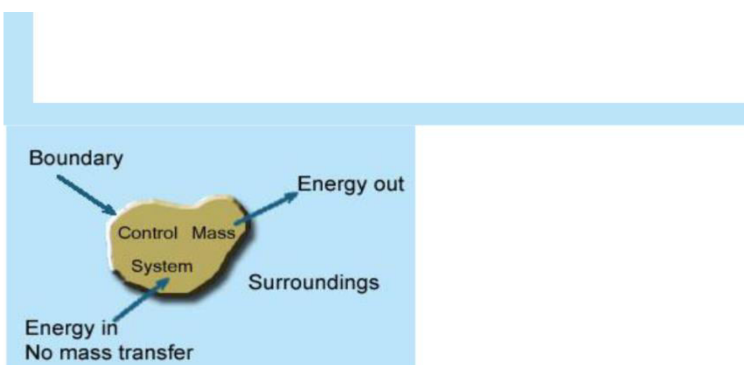


Fig 9.2A Control Mass System or Closed System

[Click to play the Demonstration]

Control Volume System (Open System)

Its a system of **fixed volume**

This type of system is usually referred to as **"open system"** or a **"control volume"**Mass transfer can take place across a control volume.

Energy transfer may also occur into or out of the system.

A control volume can be seen as a fixed region across which mass and energy transfers are studied.

Control Surface- Its the boundary of a control volume across which the transfer of both mass and energy takes place.

The mass of a control volume (open system) may or may not be fixed.

When the net influx of mass across the control surface equals zero then the mass of the system is fixed and vice-versa.

The identity of mass in a control volume always changes unlike the case for a control mass system (closed system).

Most of the engineering devices, in general, represent an open system or control volume.

Example:-

Heat exchanger - Fluid enters and leaves the system continuously with the transfer of heat across the system boundary.

Pump - A continuous flow of fluid takes place through the system with a transfer of mechanical energy from the surroundings to the system.

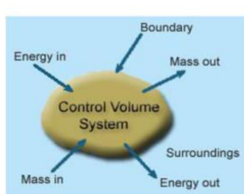


Fig 9.3 A Control Volume System or Open System

Isolated System

Its a system of **fixed mass** with *same* **identity and fixed energy**.

No interaction of mass or energy takes place between the system and the surroundings.In more informal wordan isolated system is like aclosed shop amidst a busy market

Fig 9.4 An Isolated Syste

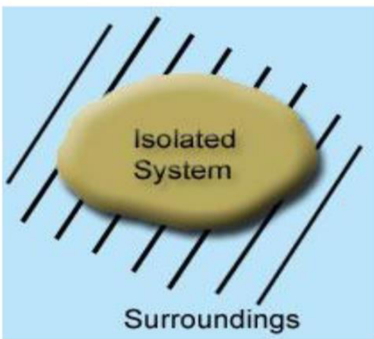
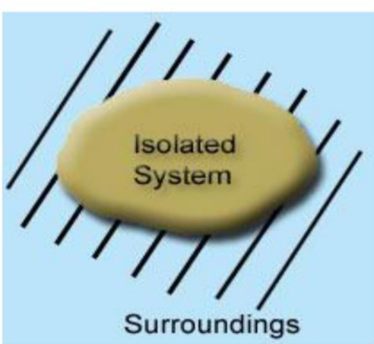


Fig 9.4 An Isolated Syste



Conservation of Momentum: Momentum Theorem

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

Newton's Second Law of Motion

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

Reynolds Transport Theorem

A study of fluid flow by the Eulerian approach requires a mathematical modeling for a control volume either in differential or in integral form. Therefore the physical statements of the principle of conservation of mass, momentum and energy with reference to a control volume become necessary.

This is done by invoking a theorem known as the Reynolds transport theorem which relates the control volume concept with that of a control mass system in terms of a general property of the system.

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

Equation of Reynolds Transport Theorem

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

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

In this equation

N - flow property which is transported- intensive value of the flow property

Analysis Of Finite Control Volumes - the application of momentum theorem

We'll see the application of momentum theorem in some practical cases of inertial and non-inertial control volumes.

Inertial Control Volumes

Applications of momentum theorem for an inertial control volume are described with reference to three distinct types of practical problems, namelyForces acting due to internal flows through expanding or reducing pipe bends.Forces on stationary and moving vanes due to impingement of fluid jets.Jet propulsion of ship and aircraft moving with uniform velocity.

Non-inertial Control Volume

A good example of non-inertial control volume is a rocket engine which works on the principle of jet propulsion.We shalll discuss each example seperately in the following slides.

Eulers Equation along a Streamline

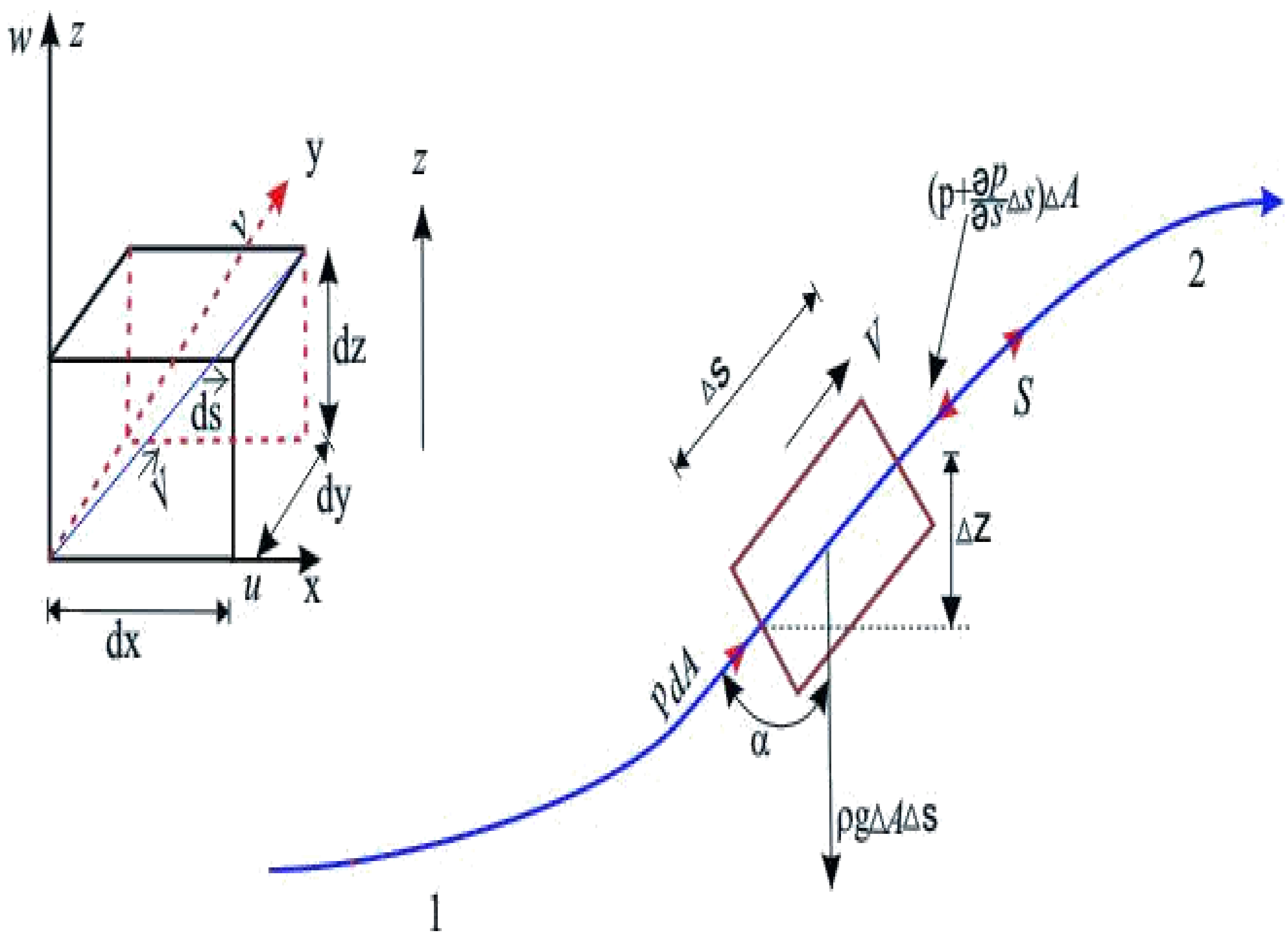


Fig 12.3 Force Balance on a Moving Element Along a Streamline

Derivation

Euler's equation along a streamline is derived by applying Newton's second law of motion to a fluid element moving along a streamline. Considering gravity as the only body force component acting vertically downward (Fig. 12.3), the net external force acting on the fluid element along the directions can be written as

$$F_s = -\frac{\partial p}{\partial s}\Delta s\Delta A - \rho\Delta s\Delta A g\cos\alpha \qquad (12.8)$$

where ΔA is the cross-sectional area of the fluid element. By the application of Newton's second law of motion in s direction, we get

$$\rho\Delta s\Delta A\frac{DV}{Dt} = -\frac{\partial p}{\partial s}\Delta s\Delta A - \rho\Delta s\Delta A g\cos\alpha \qquad (12.9)$$

From geometry we get

$$\cos\alpha = \lim_{\Delta s\to 0}\frac{\Delta z}{\Delta s} = \frac{dz}{ds}$$

Hence, the final form of Eq. (12.9) becomes

$$\rho\frac{DV}{Dt} = -\frac{\partial p}{\partial s} - \rho g\frac{dz}{ds}$$

$$\frac{\partial V}{\partial t} + V\frac{\partial V}{\partial s} = -\frac{1}{\rho}\frac{\partial p}{\partial s} - g\frac{dz}{ds}$$

Equation (12.10) is the Euler's equation along a streamline.

Let us consider $d\vec{s}$ along the streamline so that

$$d\vec{s} = \vec{i}dx + \vec{j}dy + \vec{k}dz$$

Again, we can write from Fig. 12.3

$$\frac{dx}{ds} = \frac{u}{V},\quad \frac{dy}{ds} = \frac{v}{V}\quad \text{and}\quad \frac{dz}{ds} = \frac{w}{V}$$

The equation of a streamline is given by

$$\vec{V}\times d\vec{S}=0$$

$$\begin{vmatrix} i & j & k \\ u & v & w \\ dx & dy & dz \end{vmatrix}=0$$

$$\rho\left(\frac{ds}{V}\cdot\frac{\partial}{\partial t}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)+\frac{\partial}{\partial x}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)dx+\frac{\partial}{\partial y}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)dy+\frac{\partial}{\partial z}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)dz\right)$$

which finally leads to

$$\rho\left(\frac{ds}{V}\cdot\frac{\partial}{\partial t}\left(\frac{V^2}{2}\right)+\frac{\partial}{\partial x}\left(\frac{V^2}{2}\right)dx+\frac{\partial}{\partial y}\left(\frac{V^2}{2}\right)dy+\frac{\partial}{\partial z}\left(\frac{V^2}{2}\right)dz\right)$$

$$udy=vdx;\qquad udz=wdx;\qquad vdz=wdy$$

Multiplying Eqs (12.7a), (12.7b) and (12.7c) by dx, dy and dz respectively and then substituting the above mentioned equalities, we get

$$\rho\left(u\frac{\partial u}{\partial t}\cdot\frac{ds}{V}+u\frac{\partial u}{\partial x}dx+u\frac{\partial u}{\partial y}dy+u\frac{\partial u}{\partial z}dz\right)=-\frac{\partial p}{\partial x}dx+X_x dx$$

$$\rho\left(v\frac{\partial v}{\partial t}\cdot\frac{ds}{V}+v\frac{\partial v}{\partial x}dx+v\frac{\partial v}{\partial y}dy+v\frac{\partial v}{\partial z}dz\right)=-\frac{\partial p}{\partial y}dy+X_y dy$$

$$\rho\left(w\frac{\partial w}{\partial t}\cdot\frac{ds}{V}+w\frac{\partial w}{\partial x}dx+w\frac{\partial w}{\partial y}dy+w\frac{\partial w}{\partial z}dz\right)=-\frac{\partial p}{\partial z}dy+X_z dz$$

Adding these three equations, we can write

$$\rho\left[\frac{\partial V}{\partial t}+V\left(\frac{\partial V}{\partial x}\cdot\frac{dx}{ds}+\frac{\partial V}{\partial y}\cdot\frac{dy}{ds}+\frac{\partial V}{\partial z}\cdot\frac{dz}{ds}\right)\right]=-\left(\frac{\partial p}{\partial x}\cdot\frac{dx}{ds}+\frac{\partial p}{\partial y}\cdot\frac{dy}{ds}+\frac{\partial p}{\partial z}\cdot\frac{dz}{ds}\right)-\rho g\frac{dz}{ds}$$

$$\boxed{\rho\left[\frac{\partial V}{\partial t}+V\frac{\partial V}{\partial s}\right]=-\frac{\partial p}{\partial s}-\rho g\frac{dz}{ds}}$$

Hence,

This is the more popular form of Euler's equation because the velocity vector in a flow field is always directed along the streamline.

A Control Volume Approach for the Derivation of Euler's Equation

Euler's equations of motion can also be derived by the use of the momentum theorem for a control volume.

Derivation

In a fixed x, y, z axes (the rectangular cartesian coordinate system), the parallelopiped which was taken earlier as a control mass system is now considered We can define the velocity vector $\vec{V}$ and the body force per unit volume $\rho\vec{X}$ asas a control volume (Fig. 12.4).

$$\vec{V} = \vec{i}u + \vec{j}v + \vec{k}w$$

$$\rho\vec{X} = \vec{i}\rho X_x + \vec{j}\rho X_y + \vec{k}\rho X_z$$

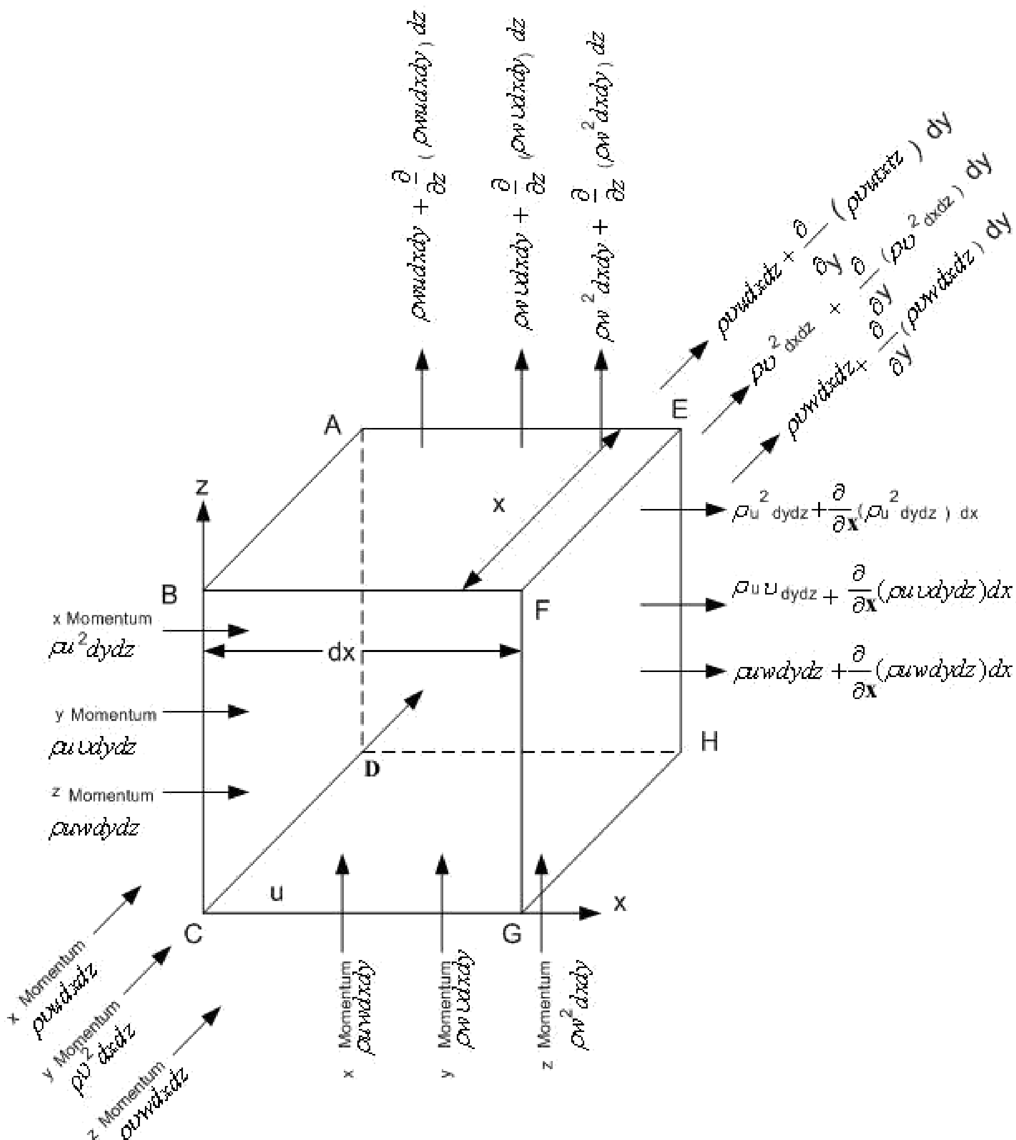


Fig 12.4 A Control Volume used for the derivation of Euler's Equation

The rate of x momentum influx to the control volume through the face ABCD is equal to ρu^2 dy dz. The rate of x momentum efflux from the control volume through the face EFGH

equals $\rho u^2 dy\, dz + \dfrac{\partial}{\partial x}\left(\rho u^2 dy\, dz\right)dx$

Therefore the rate of net efflux of x momentum from the control volume due to the faces

perpendicular to the x direction (faces ABCD and EFGH) = $\dfrac{\partial}{\partial x}(\rho u^2)\mathrm{d}\forall$ where, $\mathrm{d}\forall$, the elemental volume = dx dy dz.

Similarly,

The rate of net efflux of x momentum due to the faces perpendicular to the y direction

(face BCGF and ADHE) = $\dfrac{\partial}{\partial y}(\rho uv)\mathrm{d}\forall$

The rate of net efflux of x momentum due to the faces perpendicular to the z direction

(faces DCGH and ABFE) = $\dfrac{\partial}{\partial z}(\rho uw)\mathrm{d}\forall$

Hence, the net rate of x momentum efflux from the control volume becomes

$$\left[\frac{\partial}{\partial z}(\rho u^2)+\frac{\partial}{\partial z}(\rho uv)+\frac{\partial}{\partial z}(\rho uw)\right]\mathrm{d}\forall$$

The time rate of increase in x momentum in the control volume can be written as

$\dfrac{\partial}{\partial t}(\rho u\mathrm{d}\forall)=\dfrac{\partial}{\partial t}(\rho u)\mathrm{d}\forall$ (Since, $\mathrm{d}\forall$, by the definition of control volume, is invariant with time)
Applying the principle of momentum conservation to a control volume (Eq. 4.28b), we get

$$\frac{\partial}{\partial t}(\rho u)+\frac{\partial}{\partial x}(\rho u^2)+\frac{\partial}{\partial y}(\rho uv)+\frac{\partial}{\partial z}(\rho uw)=\rho X_x-\frac{\partial p}{\partial x} \qquad (12.11a)$$

The equations in other directions y and z can be obtained in a similar way by considering the y momentum and z momentum fluxes through the control volume as

$$\frac{\partial}{\partial t}(\rho v)+\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho v^2)+\frac{\partial}{\partial z}(\rho vw)=\rho X_y-\frac{\partial p}{\partial y} \qquad (12.11b)$$

$$\frac{\partial}{\partial t}(\rho w)+\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial y}(\rho vw)+\frac{\partial}{\partial z}(\rho w^2)=\rho X_z-\frac{\partial p}{\partial z} \qquad (12.11c)$$

The typical form of Euler's equations given by Eqs (12.11a), (12.11b) and (12.11c) are known as the conservative forms.

Bernoulli's Equation

Energy Equation of an ideal Flow along a Streamline

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

$$V\frac{dV}{ds}=-\frac{1}{\rho}\frac{dp}{ds}-g\frac{dz}{ds} \qquad (13.6)$$

Application of a force through a distance ds along the streamline would physically imply work interaction. Therefore an equation for conservation of energy along a streamline can be obtained by integrating the Eq. (13.6) with respect to ds as

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

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

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

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

The Eqs (13.7) and (13.8) are based on the assumption that no work or heat interaction between a fluid element and the surrounding takes place. The first term of the Eq. (13.8) represents the flow work per unit mass, the second term represents the kinetic energy per unit mass and the third term represents the potential energy per unit mass. Therefore the sum of three terms in the left hand side of Eq. (13.8) can be considered as the total mechanical energy per unit mass which remains constant along a streamline for a steady inviscid and incompressible flow of fluid. Hence the Eq. (13.8) is also known as **Mechanical energy equation**.

$$\frac{P}{\rho g}+\frac{V^2}{2g}+z=C_1(\text{constant}) \qquad (13.9)$$

In a fluid flow, the energy per unit weight is termed as head. Accordingly, equation 13.9 can be interpreted as

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

Bernoulli's Equation with Head Loss

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

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2+h_f \qquad (13.10)$$

where, h_f represents the frictional work done (the work done against the fluid friction) per unit weight of a fluid element while moving from a station 1 to 2 along a streamline in the direction of flow. The term h_f is usually referred to as head loss between 1 and 2, since it amounts to the loss in total mechanical energy per unit weight between points 1 and 2 on a streamline due to the effect of fluid friction or viscosity. It physically signifies that the difference in the total mechanical energy between stations 1 and 2 is dissipated into intermolecular or thermal energy and is expressed as loss of head h_f in Eq. (13.10). The term head loss, is conventionally symbolized as h_L instead of h_f in dealing with practical problems. For an inviscid flow $h_L = 0$, and the total mechanical energy is constant along a streamline.

Bernoulli's Equation In Irrotational Flow

In the previous lecture (lecture 13) we have obtained Bernoulli's equation

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

This equation was obtained by integrating the Euler's equation (the equation of motion) with respect to a displacement '**ds**' along a streamline. Thus, the value of C in the above equation is constant only along a streamline and should essentially vary from streamline to streamline.

The equation can be used to define relation between flow variables at point B on the streamline and at point A, along the same streamline. So, in order to apply this equation, one should have knowledge of velocity field beforehand. This is one of the limitations of application of Bernoulli's equation.

Irrotationality of flow field

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

Proof:

Let us consider a steady two dimensional flow of an ideal fluid in a rectangular Cartesian coordinate system. The velocity field is given by

$$\vec{V} = \vec{i}u + \vec{j}v$$

hence the condition of irrotationality is

$$\nabla \times \vec{V} = \left\{ \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right\} = 0$$

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

The steady state Euler's equation can be written as

$$\rho \left\{ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right\} = -\frac{\partial p}{\partial x} \qquad (14.2a)$$

$$\rho \left\{ u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right\} = -\frac{\partial p}{\partial y} \; -\rho g$$

We consider the y-axis to be vertical and directed positive upward. From the condition of irrotationality given by the Eq. (14.1), we substitute $\frac{\partial v}{\partial x}$ in place of $\frac{\partial u}{\partial y}$ in the Eq. 14.2a and $\frac{\partial u}{\partial y}$ in place of $\frac{\partial v}{\partial x}$ in the Eq. 14.2b. This results in

$$\left\{ u \frac{\partial u}{\partial x} + v \frac{\partial v}{\partial x} \right\} = -\frac{1}{\rho} \frac{\partial p}{\partial x}$$

(14.3a)

$$\left\{u\frac{\partial u}{\partial y}+v\frac{\partial v}{\partial y}\right\}=-\frac{1}{\rho}\frac{\partial p}{\partial y}\quad -g \tag{14.3b}$$

Now multiplying Eq.(14.3a) by 'dx' and Eq.(14.3b) by 'dy' and then adding these two equations we have

$$u\left\{\frac{\partial u}{\partial x}dx+\frac{\partial u}{\partial y}dy\right\}+v\left\{\frac{\partial v}{\partial x}dx+\frac{\partial v}{\partial y}dy\right\}=-\frac{1}{\rho}\left\{\frac{\partial p}{\partial x}dx+\frac{\partial p}{\partial y}dy\right\}-gdy \tag{14.4}$$

The Eq. (14.4) can be physically interpreted as the equation of conservation of energy for an arbitrary displacement

$d\vec{r}=\vec{i}dx+\vec{j}dy$. Since, u, v and p are functions of x and y, we can write

$$du=\frac{\partial u}{\partial x}dx+\frac{\partial u}{\partial y}dy \tag{14.5a}$$

$$dv=\frac{\partial v}{\partial x}dx+\frac{\partial v}{\partial y}dy \tag{14.5b}$$

$$dp=\frac{\partial p}{\partial x}dx+\frac{\partial p}{\partial y}dy \tag{14.5c}$$

With the help of Eqs (14.5a), (14.5b), and (14.5c), the Eq. (14.4) can be written as

$$udu+vdv=-\frac{1}{\rho}dp-gdy$$

$$d\left\{\frac{u^2}{2}\right\}+d\left\{\frac{v^2}{2}\right\}=-\frac{1}{\rho}dp-gdy$$

$$d\left\{\frac{u^2+v^2}{2}\right\}=-\frac{1}{\rho}dp-gdy$$

$$d\left\{\frac{V^2}{2}\right\}=-\frac{1}{\rho}dp-gdy$$

The integration of Eq. 14.6 results in

$$\int \frac{dp}{\rho} + \frac{V^2}{2} + gy = C \qquad (14.7a)$$

For an **incompressible flow**

The constant C in Eqs (14.7a) and (14.7b) has the same value in the entire flow field, since no restriction was made in the choice of dr which was considered as an arbitrary displacement in evaluating the work.

Note: In deriving Eq. (13.8) the displacement **ds** was considered along a streamline. Therefore, the total mechanical energy remains constant everywhere in an inviscid and irrotational flow, while it is constant only along a streamline for an inviscid but rotational flow.

The equation of motion for the flow of an inviscid fluid can be written in a vector form as

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

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

where $\vec{X}$ is the body force vector per unit mass

Plane Circular Vortex Flows

Plane circular vortex flows are defined as flows where streamlines are concentric circles. Therefore, with respect to a polar coordinate system with the centre of the circles as the origin or pole, the velocity field can be described as

$$V_\theta \neq 0 \qquad\qquad V_r = 0$$

where V_θ and V_r are the tangential and radial component of velocity respectively.

The equation of continuity for a two dimensional incompressible flow in a polar coordinate system is

$$\frac{\partial V_r}{\partial r} + \frac{V_r}{r} + \frac{1}{r}\frac{\partial V_\theta}{\partial \theta} = 0$$

which for a plane circular vortex flow gives $\frac{\partial V_\theta}{\partial \theta} = 0$ i.e. V_θ is not a function of θ. Hence, V_θ is a function of r only.We can write for the variation of total mechanical energy with radius as

$$\frac{\partial H}{\partial r} = \frac{V_\theta}{g}\left(\frac{dV_\theta}{dr} + \frac{V_\theta}{r}\right)$$

Free Vortex Flows

Free vortex flows are the plane circular vortex flows where the total mechanical energy remains constant in the entire flow field. There is neither any addition nor any destruction of energy in the flow field.

Therefore, the total mechanical energy does not vary from streamline to streamline. Hence from Eq. (14.8), we have,

$$\frac{\partial H}{\partial r} = \frac{V_\theta}{g}\left(\frac{dV_\theta}{dr} + \frac{V_\theta}{r}\right) = 0$$

$$\frac{1}{r}\left[\frac{d}{dr}(V_\theta r)\right] = 0 \qquad (14.9)$$

Integration of Eq 14.9 gives

$$V_\theta = \frac{C}{r} \qquad (14.10)$$

The Eq. (14.10) describes the velocity field in a free vortex flow, where C is a constant in the entire flow field. The vorticity in a polar coordinate system is defined by -

$$\Omega = \frac{\partial V_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta} + \frac{V_\theta}{r}$$

In case of vortex flows, it can be written as

$$\Omega = \frac{dV_\theta}{dr} + \frac{V_\theta}{r}$$

For a free vortex flow, described by Eq. (14.10),Ω becomes zero. Therefore we conclude that a free vortex flow is irrotational, and hence, it is also referred to as**irrotational vortex**.
It has been shown before that the total mechanical energy remains same throughout in an irrotational flow field. Therefore, irrotationality is a direct consequence of the constancy of total mechanical energy in the entire flow field and vice versa.
The interesting feature in a free vortex flow is that as $r \to 0,\ V_\theta \to \infty$ [Eq. (14.10)]. It mathematically signifies a point of singularity at r = 0 which, in practice, is impossible. In fact, the definition of a free vortex flow cannot be extended as r = 0 is approached.
In a real fluid, friction becomes dominant as **r→0** and so a fluid in this central region tends to rotate as a solid body. Therefore, the singularity at r = 0 does not render the theory of irrotational vortex useless, since, in practical problems, our concern is with conditions away from the central core.

Pressure Distribution in a Free Vortex Flow

Pressure distribution in a vortex flow is usually found out by integrating the equation of motion in the r direction. The equation of motion in the radial direction for a vortex flow can be written as

$$\frac{1}{\rho}\frac{dp}{dr} = \frac{V_\theta^2}{r} - g\cos\theta$$

$$\frac{1}{\rho}\frac{dp}{dr} = \frac{V_\theta^2}{r} - g\frac{dz}{dr} \qquad (14.11)$$

(14.12)

Integrating Eq. (14.12) with respect to dr, and considering the flow to be incompressible we have,

$$\frac{p}{\rho} = \int \frac{V_\theta^2}{r}dr - gz + A \qquad (14.13)$$

For a free vortex flow,

$$V_\theta = \frac{C}{r}$$

Hence Eq. 14.13 becomes

$$\frac{p}{\rho} = -\frac{C^2}{2r^2} - gz + A \qquad (14.14)$$

If the pressure at some radius **r = r$_a$,** is known to be the atmospheric pressure p_{atm} then equation (14.14) can be written as

$$\frac{p - p_{atm}}{\rho} = \frac{C^2}{2}\left(\frac{1}{r_a^2} - \frac{1}{r^2}\right) - g(z - z_a)$$

$$= \frac{\left(V_\theta^2\right)_{r=r_a}}{2} - \frac{V_\theta^2}{2} - g(z - z_a)$$

where z and z_a are the vertical elevations (measured from any arbitrary datum) at r and r_a.

Equation (14.15) can also be derived by a straight forward application of Bernoulli's equation between any two points at $r = r_a$ and r = r.

In a free vortex flow total mechanical energy remains constant. There is neither any energy interaction between an outside source and the flow, nor is there any dissipation of mechanical energy within the flow. The fluid rotates by virtue of some rotation previously imparted to it or because of some internal action.

Some examples are a whirlpool in a river, the rotatory flow that often arises in a shallow vessel when liquid flows out through a hole in the bottom (as is often seen when water flows out from a bathtub or a wash basin), and flow in a centrifugal pump case just outside the impeller.

Cylindrical Free Vortex

A cylindrical free vortex motion is conceived in a cylindrical coordinate system with axis z directing vertically upwards (Fig. 14.1), where at each horizontal cross-section, there exists a planar free vortex motion with tangential velocity given by Eq. (14.10).

The total energy at any point remains constant and can be written as

$$\frac{p}{\rho} + \frac{C^2}{2r^2} + gz = H\,(Cons.) \qquad (14.16)$$

The pressure distribution along the radius can be found from Eq. (14.16) by considering z as constant; again, for any constant pressure p, values of z, determining a surface of equal pressure, can also be found from Eq. (14.16).

If p is measured in gauge pressure, then the value of z, where p = 0 determines the free surface (Fig. 14.1), if one exists.

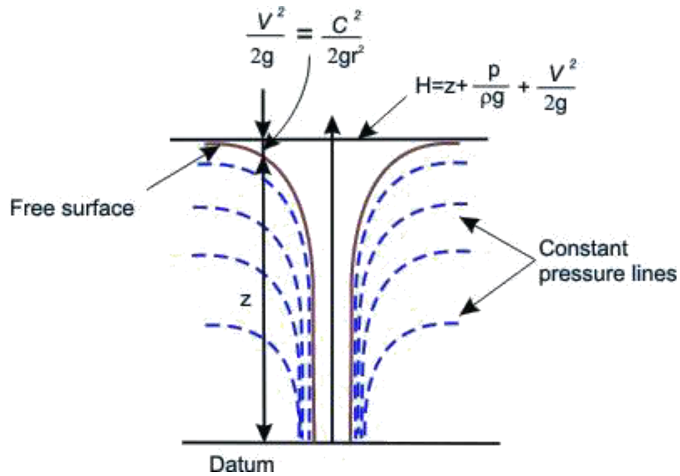


Fig 14.1 Cylindrical Free Vortex

Forced Vortex Flows

Flows where streamlines are concentric circles and the tangential velocity is directly proportional to the radius of curvature are known as **plane circular forced vortex flows**. The flow field is described in a polar coordinate system as,

$$V_\theta = \omega r$$

and $V_r = 0$

All fluid particles rotate with the same angular velocity ω like a solid body. Hence a forced vortex flow is termed as a **solid body rotation**.The vorticity Ω for the flow field can be calculated as Therefore, a forced vortex motion is not irrotational; rather it is a rotational flow with a constant vorticity 2ω. Equation (14.8) is used to determine the distribution of mechanical energy across the radius as

$$\Omega = \frac{\partial V_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta} + \frac{V_\theta}{r}$$

$$= \omega - \mathbf{0} + \omega = \mathbf{2}\omega$$

Integrating the equation between the two radii on the same horizontal plane, we have,

$$H_2 - H_1 = \frac{\omega^2}{g}(r_2^2 - r_1^2) \qquad (14.18)$$

Thus, we see from Eq. (14.18) that the total head (total energy per unit weight) increases with an increase in radius. The total mechanical energy at any point is the sum of kinetic energy, flow work or pressure energy, and the potential energy.
Therefore the difference in total head between any two points in the same horizontal plane can be written as,

$$H_2 - H_1 = \left[\frac{p_2}{\rho g} - \frac{p_1}{\rho g}\right] + \left[\frac{V_2^2}{2g} - \frac{V_1^2}{2g}\right]$$

$$= \frac{p_2}{\rho g} - \frac{p_1}{\rho g} + \frac{\omega^2}{2g}(r_2^2 - r_1^2)$$

Substituting this expression of H_2-H_1 in Eq. (14.18), we get

$$\frac{p_2 - p_1}{\rho} = \frac{\omega^2}{2}(r_2^2 - r_1^2)$$

The same equation can also be obtained by integrating the equation of motion in a radial direction as

$$\int_1^2 \frac{1}{\rho}\frac{dp}{dr}dr = \int_1^2 \frac{V_\theta^2}{r}dr = \omega^2 \int_1^1 r\,dr$$

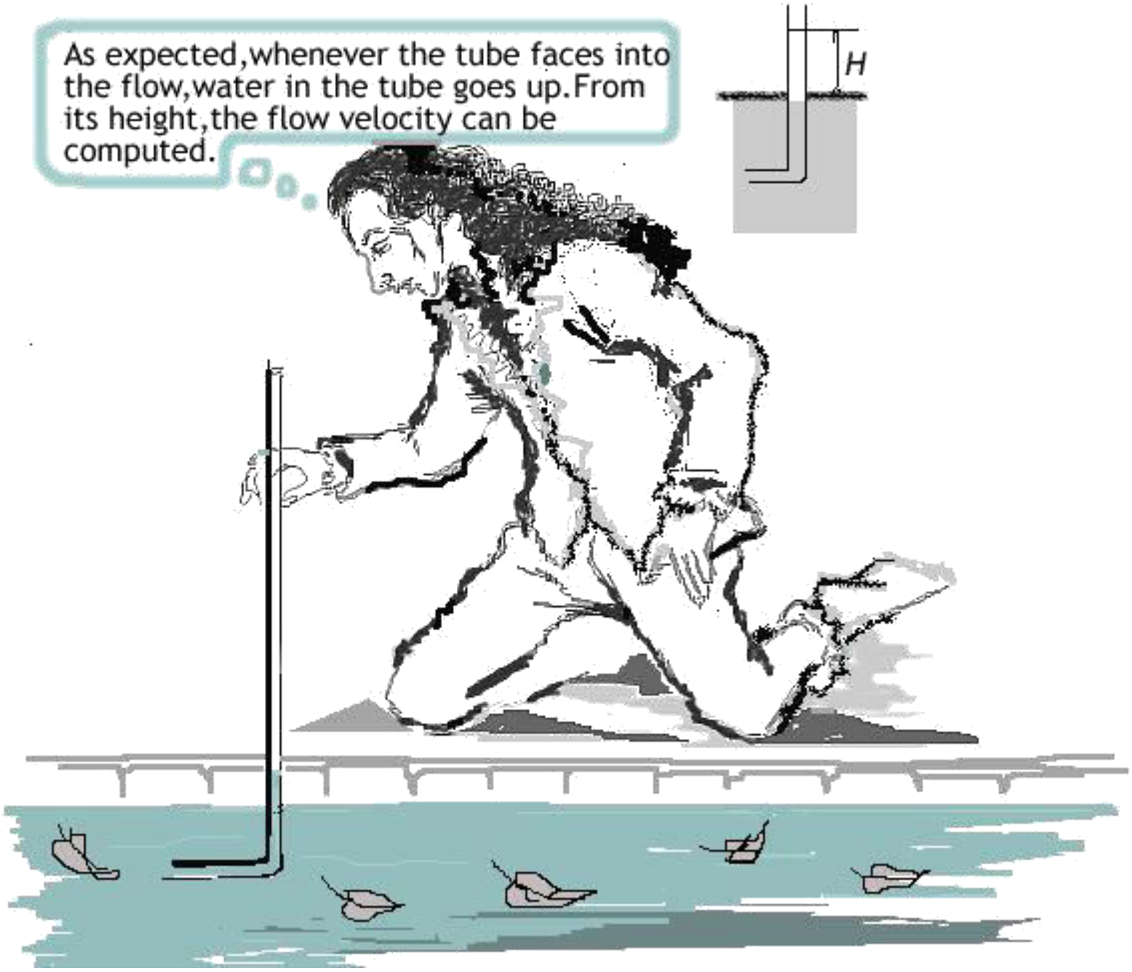
$$\frac{p_2 - p_1}{\rho} = \frac{\omega^2}{2}(r_2^2 - r_1^2)$$

Measurement Of Flow Rate Through Pipe

Flow rate through a pipe is usually measured by providing a coaxial area contraction within the pipe and by recording the pressure drop across the contraction. Therefore the determination of the flow rate from the measurement of pressure drop depends on the straight forward application of Bernoulli's equation.

Three different flow meters operate on this principle.

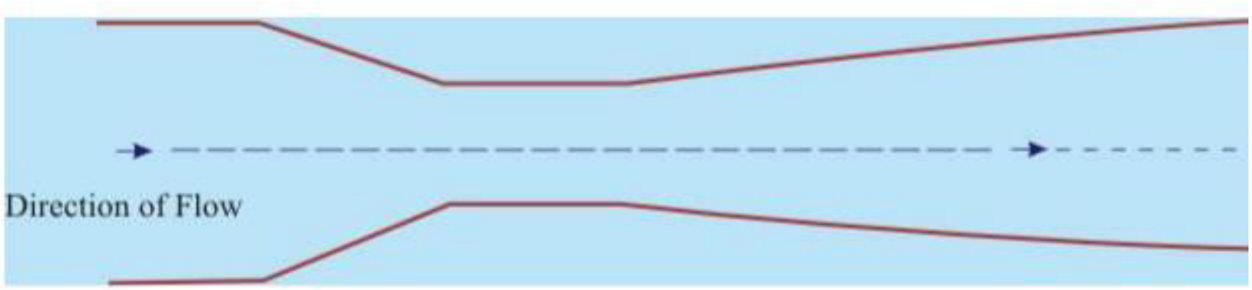
Venturimeter
Orificemeter
Flow nozzle.



Venturimeter

Construction: A venturimeter is essentially a short pipe (Fig. 15.1) consisting of two conical parts with a short portion of uniform cross-section in between. This short portion has the minimum area and is known as the throat. The two conical portions have the same base diameter, but one is having a shorter length with a larger cone angle while the other is having a larger length with a smaller cone angle.

Fig 15.1 A Venturimeter



Working:

The venturimeter is always used in a way that the upstream part oftheflowtakesplacthroughthe short conical portion while the downstream part of the flow through thelong one.This ensures a rapid converging passage and a gradual diverging passage in the direction of flow to avoid the loss of energy due to separation. In course of a flow through the converging part, the velocity increases in the direction of flow according to the principle of continuity, while the pressure decreases according to Bernoulli's theorem.

The velocity reaches its maximum value and pressure reaches its minimum value at the throat. Subsequently, a decrease in the velocity and an increase in the pressure takes place in course of flow through the divergent part. This typical variation of fluid velocity and pressure by allowing it to flow through such a constricted convergent-divergent passage was first demonstrated by an Italian scientist Giovanni Battista Venturi in 1797.

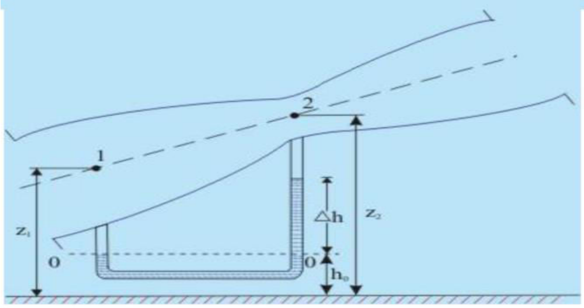


Fig 15.2 Measurement of Flow by a Venturimeter

Figure 15.2 shows that a venturimeter is inserted in an inclined pipe line in a vertical plane to measure the flow rate through the pipe. Let us consider a steady, ideal and one dimensional (along the axis of the venturi meter) flow of fluid. Under this situation, the velocity and pressure at any section will be uniform.

Let the velocity and pressure at the inlet (Sec. 1) are V_1 and p_1 respectively, while those at the throat (Sec. 2) are V_2 and p_2. Now, applying Bernoulli's equation between Secs 1 and 2, we get

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \tag{15.1}$$

$$\frac{V_2^2-V_1^2}{2g}=\frac{p_1-p_2}{2g}+z_1-z_2 \tag{15.2}$$

where ρ is the density of fluid flowing through the venturimeter.

From continuity,

$$V_1A_1=V_2A_2 \tag{15.3}$$

where A_1 and A_2 are the cross-sectional areas of the venturi meter at its throat and inlet respectively.

With the help of Eq. (15.3), Eq. (15.2) can be written as

$$\frac{V_2^2}{2g}\left(1-\frac{A_2^2}{A_1^2}\right)=\left(\frac{p_1}{2g}+z_1\right)-\left(\frac{p_2}{2g}+z_2\right)$$

$$V_2=\frac{1}{\sqrt{1-\frac{A_2^2}{A_1^2}}}\sqrt{2g(h_1^*-h_2^*)}$$

where h_1^* and h_2^* are the piezometric pressure heads at sec. 1 and sec. 2 respectively, and are defined as

$$h_1^*=\frac{p_1}{\rho g}+z_1 \tag{15.5a}$$

$$h_2^*=\frac{p_2}{\rho g}+z_2 \tag{15.5b}$$

Hence, the volume flow rate through the pipe is given by

$$Q = A_2 V_2 = \frac{A_2}{\sqrt{1 - \frac{A_2^2}{A_1^2}}} \sqrt{2g(h_1^* - h_2^*)} \qquad (15.6)$$

If the pressure difference between Sections 1 and 2 is measured by a manometer as shown in Fig. 15.2, we can write

$$p_1 + \rho g(z_1 - h_o) = p_2 + \rho g(z_2 - h_o - \Delta h) + \Delta h \rho_m g$$

$$\text{or,} \quad (p_1 + \rho g z_1) - (p_2 + \rho g z_2) = (\rho_m - \rho) g \Delta h$$

$$\left(\frac{p_1}{\rho g} + z_1\right) - \left(\frac{p_2}{\rho g} + z_2\right) = \left(\frac{\rho_m}{\rho} - 1\right)\Delta h$$

$$\text{or,} \quad h_1^* - h_2^* = \left(\frac{\rho_m}{\rho} - 1\right)\Delta h \qquad (15.7)$$

where ρ_m is the density of the manometric liquid.

Equation (15.7) shows that a manometer always registers a direct reading of the difference in piezometric pressures. Now, substitution of $h_1^* - h_2^*$ from Eq. (15.7) in Eq. (15.6) gives

$$Q = \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}} \sqrt{2g(\rho_m / \rho - 1)\Delta h} \qquad (15.8)$$

If the pipe along with the venturimeter is horizontal, then $z_1 = z_2$; and hence $h_1^* - h_2^*$ becomes $h_1 - h_2$, where h_1 and h_2 are the static pressure heads

$$\left(h_1 = \frac{p_1}{\rho g}, h_2 = \frac{p_2}{\rho g}\right)$$

The manometric equation Eq. (15.7) then becomes

$$h_1 - h_2 = \left[\frac{\rho_m}{\rho} - 1\right]\Delta h$$

Therefore, it is interesting to note that the final expression of flow rate, given by Eq. (15.8), in terms of manometer deflection Δh, remains the same irrespective of whether the pipe-line along with the venturimeter connection is horizontal or not.Measured values of Δh, the difference in piezometric pressures between Secs I and 2, for a real fluid will always be greater than that assumed in case of an ideal fluid because of frictional losses in addition to the change in momentum.

$$Q_{actual} = C_d \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}} \sqrt{2g(\rho_m / \rho - 1)\Delta h}$$

Therefore, Eq. (15.8) always overestimates the actual flow rate. In order to take this into account, a multiplying factor C_d, called the coefficient of discharge, is incorporated in the Eq. (15.8) as

The **coefficient of discharge** C_d is always less than unity and is defined as

$$C_d = \frac{\text{Actual rate of discharge}}{\text{Theoretical rate of discharge}}$$

where, the theoretical discharge rate is predicted by the Eq. (15.8) with the measured value of Δh, and the actual rate of discharge is the discharge rate measured in practice. **Value of C_d for a venturimeter usually lies between 0.95 to 0.98.**

Orificemeter

Construction: An orificemeter provides a simpler and cheaper arrangement for the measurement of fow through a pipe. An orificemeter is essentially a thin circular plate with a sharp edged concentric circular hole in it.

Working:

The orifice plate, being fixed at a section of the pipe, (Fig. 15.3) creates an obstruction to the flow by providing an opening in the form of an orifice to the flow passage.

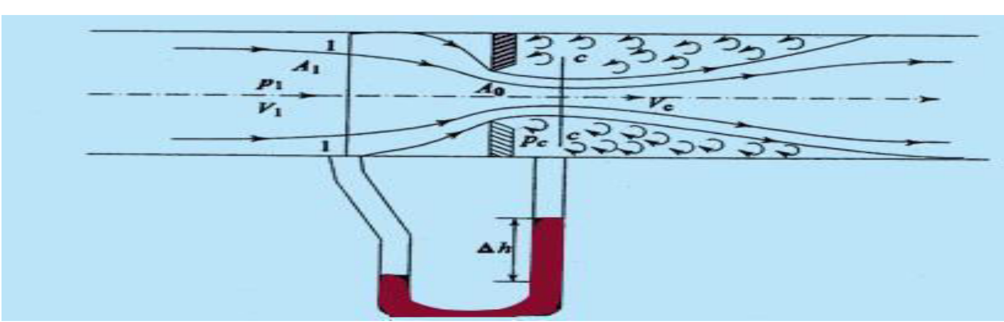


Fig 15.3 Flow through an Orificemeter

The area A_0 of the orifice is much smaller than the cross-sectional area of the pipe. The flow from an upstream section, where it is uniform, adjusts itself in such a way that it contracts until a section downstream the orifice plate is reached, where the vena contracta is formed, and then expands to fill the passage of the pipe.
One of the pressure tapings is usually provided at a distance of one diameter upstream the orifice plate where the flow is almost uniform (Sec. 1-1) and the other at a distance of half a diameter downstream the orifice plate.
Considering the fluid to be ideal and the downstream pressure taping to be at the vena contracta (Sec. c-c), we can write, by applying Bernoulli's theorem between Sec. 1-1 and Sec. c-c,

$$\frac{p_1^*}{\rho g} + \frac{V_1^2}{2g} = \frac{p_c^*}{\rho g} + \frac{V_c^2}{2g} \tag{15.10}$$

where p_1^* and p_c^* are the piezometric pressures at Sec.1-1 and c-c respectively.

From the equation of continuity,

$$V_1 A_1 = V_c A_c \tag{15.11}$$

where A_c is the area of the vena contracta.

With the help of Eq. (15.11), Eq. (15.10) can be written as,

$$V_c = \sqrt{\frac{2(p_1^* - p_c^*)}{\rho\left(1 - A_c^2 / A_1^2\right)}} \tag{15.12}$$

Correction in Velocity

Recalling the fact that the measured value of the piezometric pressure drop for a real fluid is always more due to friction than that assumed in case of an inviscid flow, a coefficient of velocity C_v (always less than 1) has to be introduced to determine the actual velocity V_c when the pressure drop $p_1^* - p_c^*$ in Eq. (15.12) is substituted by its measured value in terms of the manometer deflection **'Δh'**

$$V_c = C_v \sqrt{\frac{2\rho g(\rho_m / \rho - 1)\Delta h}{1 - A_c^2 / A_1^2}}$$

where 'Δh' is the difference in liquid levels in the manometer and ρ_m is the density of the manometric liquid.

Volumetric flow rate

$$Q = A_c V_c$$

15.14) If a **coefficient of contraction** C_c is defined as, $C_c = A_c / A_0$, where A_0 is the area of the orifice, then Eq.(15.14) can be written, with the help of Eq. (15.13),

$$Q = C_c A_0 C_v \sqrt{\frac{2g(\rho_m/\rho - 1)\Delta h}{1 - C_c^2 A_0^2 / A_1^2}}$$

$$= C_c A_0 C_v \sqrt{\frac{2g}{1 - C_c^2 A_0^2 / A_1^2}} \sqrt{(\rho_m/\rho - 1)\Delta h}$$

$$= C\sqrt{(\rho_m/\rho - 1)\Delta h}$$

The value of C depends upon the ratio of orifice to duct area, and the Reynolds number of flow.

with, $C = C_d A_0 \sqrt{\frac{2g}{1 - C_c^2 A_0^2 / A_1^2}}$, $where\,(C_d = C_v C_c)$

The main job in measuring the flow rate with the help of an orificemeter, is to find out accurately the value of C at the operating condition.
The downstream manometer connection should strictly be made to the section where the vena contracta occurs, but this is not feasible as the vena contracta is somewhat variable in position and is difficult to realize.
In practice, various positions are used for the manometer connections and C is thereby affected. **Determination of accurate values of C of an orificemeter at different operating conditions is known as calibration of the orifice meter.**

Concept and Types of Physical Similarity

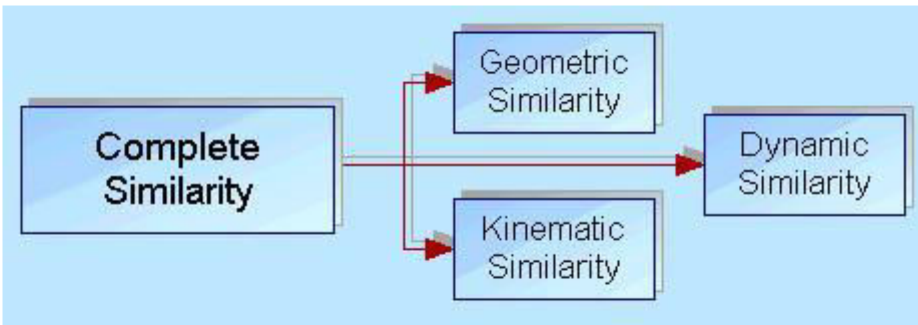
The primary and fundamental requirement for the **physical similarity** between two problems is that the **physics of the problems must be the same**.

For an example, two flows: one governed by viscous and pressure forces while the other by gravity force cannot be made physically similar. Therefore, the laws of similarity have to be sought between problems described by the same physics.

Definition of physical similarity as a general proposition.

Two systems, described by the same physics, operating under different sets of conditions are said to be physically similar in respect of certain specified physical quantities; when the ratio of corresponding magnitudes of these quantities between the two systems is the same everywhere.

In the field of mechanics, there are three types of similarities which constitute the complete similarity between problems of same kind.



Geometric Similarity : If the specified physical quantities are geometrical dimensions, the similarity is called Geometric Similarty,

Kinematic Similarity : If the quantities are related to motions, the similarity is called Kinematic Similarity

Dynamic Similarity : If the quantities refer to forces, then the similarity is termed as Dynamic Similarity.

Geometric Similarity

Geometric Similarity implies the similarity of shape such that, the **ratio of any length in one system to the corresponding length in other system is the same everywhere**.
This ratio is usually known as **scale factor**.
Therefore, geometrically similar objects are similar in their shapes, i.e., proportionate in their physical dimensions, but differ in size.In investigations of physical similarity,

- the full size or **actual scale systems** are known as **prototypes**
- the **laboratory scale systems** are referred to as **models**
- use of the same fluid with both the prototype and the model is not necessary
- model need not be necessarily smaller than the prototype. The flow of fluid through an injection nozzle or a carburettor , for example, would be more easily studied by using a model much larger than the prototype.

$$\frac{l_{1m}}{l_{1p}} = \frac{l_{2m}}{l_{2p}} = l_r$$

- the model and prototype may be of identical size, although the two may then differ in regard to other factors such as velocity, and properties of the fluid.If l_1 and l_2 are the two characteristic physical dimensions of any object, then the requirement of geometrical similarity is **model ratio**)

(The second suffices *m* and *p* refer to model and prototype respectively) where l_r is the scale factor or sometimes known as the model ratio. Figure 5.1 shows three pairs of geometrically similar objects, namely, a right circular cylinder, a parallelopiped, and a triangular prism.

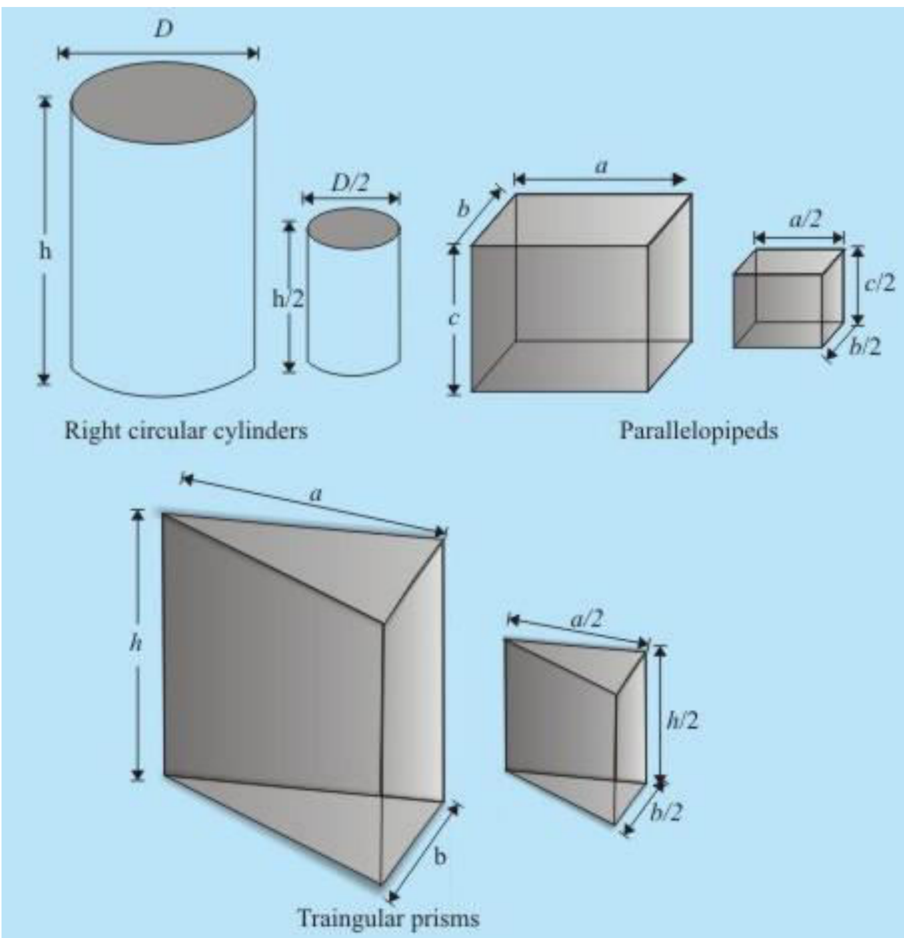


Fig 17.1 Geometrically Similar Objects

In all the above cases model ratio is 1/2

Geometric similarity is perhaps the most obvious requirement in a model system designed to correspond to a given prototype system.A perfect geometric similarity is not always easy to attain.

Problems in achieving perfect geometric similarity are:

- For a small model, the surface roughness might not be reduced according to the scale factor (unless the model surfaces can be made very much smoother than those of the prototype). If for any reason the scale factor is not the same throughout, a distorted model results.

- Sometimes it may so happen that to have a perfect geometric similarity within the available laboratory space, physics of the problem changes. For example, in case of large prototypes, such as rivers, the size of the model is limited by the available floor space of the laboratory; but if a very low scale factor is used in reducing both the horizontal and vertical lengths, this may result in a stream so shallow that surface tension has a considerable effect and, moreover, the flow may be laminar instead of turbulent. In this situation, a distorted model may be unavoidable (a lower scale factor ”for horizontal lengths while a relatively higher scale factor for vertical lengths. The extent to which perfect geometric similarity should be sought therefore depends on the problem being investigated, and the accuracy required from the solution.

Kinematic Similarity

Kinematic similarity refers to **similarity of motion.**

Since motions are described by distance and time, it implies **similarity of lengths (i.e., geometrical similarity**) and, in addition**, similarity of time intervals**.

If the corresponding lengths in the two systems are in a fixed ratio, the velocities of corresponding particles must be in a fixed ratio of magnitude of corresponding time intervals.

If the ratio of corresponding lengths, known as the **scale factor, is l_r** and **the ratio of corresponding time intervals is t_r**, then the magnitudes of corresponding**velocities are in the ratio l_r/t_r** and the magnitudes of corresponding **accelerations are in the ratio $l_r/t^2{}_r$**.

A well-known **example** of kinematic similarity is found in a planetarium. Here the galaxies of stars and planets in space are reproduced in accordance with a certain length scale and in simulating the motions of the planets, a fixed ratio of time intervals (and hence velocities and accelerations) is used.

When fluid motions are kinematically similar, the **patterns formed by 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, but not the sufficient one.

For example, geometrically similar boundaries may ensure geometrically similar streamlines in the near vicinity of the boundary but not at a distance from the boundary.

Dynamic Similarity

Dynamic similarity is the **similarity of forces** .

In dynamically similar systems, the **magnitudes of forces** at correspondingly similar points in each system are **in a fixed ratio.**

In a system involving flow of fluid, different forces due to different causes may act on a fluid element. These forces are as follows:

Viscous Force (due to viscosity)	$\vec{F}_v$
Pressure Force (due to different in pressure)	$\vec{F}_p$
Gravity Force (due to gravitational attraction)	$\vec{F}_g$
Capillary Force (due to surface tension)	$\vec{F}_c$
Compressibility Force (due to elasticity)	$\vec{F}_e$

According to Newton 's law, the resultant F_R of all these forces, will cause the acceleration of a fluid element. Hence

$$\vec{F}_R = \vec{F}_v + \vec{F}_p + \vec{F}_g + \vec{F}_c + \vec{F}_e \qquad (17.1)$$

Moreover, the **inertia force** $\vec{F}_i$ **is defined as equal and opposite to the resultant accelerating force** $\vec{F}_R$

$\vec{F}_i$ = - $\vec{F}_R$

Therefore Eq. 17.1 can be expressed as

$$\vec{F}_v + \vec{F}_p + \vec{F}_g + \vec{F}_c + \vec{F}_e + \vec{F}_i = 0$$

For dynamic similarity, the magnitude ratios of these forces have to be same for both the prototype and the model. **The inertia force $\vec{F}_i$ is usually taken as the common one to describe the ratios** as (or putting in other form we equate the the non dimensionalised forces in the two systems

A fluid motion, under all such forces is characterised by

Hydrodynamic parameters like pressure, velocity and acceleration due to gravity,
Rheological and other physical properties of the fluid involved, and
Geometrical dimensions of the system.
It is important to express the magnitudes of different forces in terms of these parameters, to know the extent of their influences on the different forces acting on a flluid element in the course of its flow.

Inertia Force $\vec{F}_i$

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

The mass of a fluid element is proportional to ρl^3 where, ρ is the density of fluid and l is the characteristic geometrical dimension of the system.

The acceleration of a fluid element in any direction is the rate at which its velocity in that direction changes with time and is therefore proportional in magnitude to some characteristic velocity *V* divided by some specified interval of time t. The time interval t is proportional to the characteristic length l divided by the characteristic velocity *V*, so that the acceleration becomes proportional to V^2/l.
The magnitude of inertia force is thus proportional to

$$\boxed{\rho l^3 V^2 / l = \rho l^2 V^2}$$

This can be written as,

$$\left|\vec{F}_i\right| \propto \rho l^2 V^2 \qquad (18.1a)$$

Viscous Force

The viscous force arises from shear stress in a flow of fluid.Therefore, we can write

Magnitude of viscous force $\vec{F}_v$ = shear stress X surface area over which the shear stress acts

Again, shear stress = μ (viscosity) X rate of shear strainwhere, rate of shear strain $\propto$ velocity gradient $\propto \frac{V}{l}$ and surface area Hence

$$\propto \mu V l$$

Pressure Force $\vec{F}_p$

The pressure force arises due to the difference of pressure in a flow field.

Hence it can be written as

$$\left|\vec{F}_p\right| \propto \Delta p . l^2$$

(where, □p is some characteristic pressure difference in the flow.)

Gravity Force $\vec{F}_g$ The gravity force on a fluid element is its

weight. Hence,

$$\left|\vec{F}_g\right| \propto p l^3 g$$

(where g is the acceleration due to gravity or weight per unit mass)

Capillary or Surface Tension Force $\vec{F}_c$

The capillary force arises due to the existence of an interface between two fluids.

The surface tension force acts tangential to a surface .
It is equal to the coefficient of surface tension σ multiplied by the length of a linear element on the surface perpendicular to which the force acts.
Therefore,

$$\left|\vec{F}_c\right| \propto \sigma l \qquad (18.1e)$$

Compressibility or Elastic Force $\vec{F}_e$

Elastic force arises due to the compressibility of the fluid in course of its flow.

For a given compression (a decrease in volume), the increase in pressure is proportional to the bulk modulus of elasticity E
This gives rise to a force known as the elastic force.

$$\Delta p \; \alpha \; E$$

Hence, for a given compression

$$\left|\vec{F}_e\right| \propto El^2 \qquad (18.1f)$$

The flow of a fluid in practice does not involve all the forces simultaneously.

Therefore, the pertinent dimensionless parameters for dynamic similarity are derived from the ratios of significant forces causing the flow.

Dynamic Similarity of Flows governed by Viscous, Pressure and Inertia Forces

The criterion of dynamic similarity for the **flows controlled by viscous, pressure and inertia forces** are derived from the ratios of the representative magnitudes of these forces with the help of Eq. (18.1a) to (18.1c) as follows:

$$\frac{\text{Viscous force}}{\text{Inertia Force}} = \frac{\left|\vec{F}_v\right|}{\left|\vec{F}_i\right|} \propto \frac{\mu V l}{\rho V^2 l^2} = \frac{\mu}{\rho V l} \qquad (18.2a)$$

$$\frac{\text{Pressure force}}{\text{Inertia Force}} = \frac{\left|\vec{F}_p\right|}{\left|\vec{F}_i\right|} \propto \frac{\Delta p l^2}{\rho V^2 l^2} = \frac{\Delta p}{\rho V^2} \qquad (18.2b)$$

The term $\rho l V/\mu$ **is known as Reynolds number, Re** after the name of the scientist who first

developed it and is thus proportional to the magnitude ratio of inertia force to viscous force .(Reynolds number plays a vital role in the analysis of fluid flow)

The term $\Delta p/\rho V^2$ **is known as Euler number, Eu** after the name of the scientist who first derived it. The dimensionless terms Re and Eu represent the critieria of dynamic similarity for the flows which are affected only by viscous, pressure and inertia forces. Such instances, for example, arethe full flow of fluid in a completely closed conduit,flow of air past a low-speed aircraft and the flow of water past a submarine deeply submerged to produce no waves on the surface.

Hence, for a complete dynamic similarity to exist between the prototype and the model for this class of flows, the Reynolds number, Re and Euler number, Eu have to be same for the two (prototype and model). Thus

$$\frac{\rho_p\, l_p V_p}{\mu_p} = \frac{\rho_m\, l_m V_m}{\mu_m}$$

$$\frac{\Delta P_p}{\rho_p V_p^2} = \frac{\Delta P_m}{\rho_m V_m^2} \qquad (18.2c)$$

(18.2d)

where, the suffix p and suffix m refer to the parameters for prototype and model respectively.

In practice, the pressure drop is the dependent variable, and hence it is compared for the two systems with the help of Eq. (18.2d), while the equality of Reynolds number (Eq. (18.2c)) along with the equalities of other parameters in relation to kinematic and geometric similarities are maintained.

The characteristic geometrical dimension l **and** the reference velocity **V** in the expression of the Reynolds number **may be any geometrical dimension and any velocity which are significant in determining the pattern of flow.**

For internal flows through a closed duct, the hydraulic diameter of the duct D_h and the average flow velocity at a section are invariably used for l and V respectively.
The hydraulic diameter D_h is defined as $D_h = 4A/P$ where A and P are the cross-sectional area and wetted perimeter respectively.

Dynamic Similarity of Flows with Gravity, Pressure and Inertia Forces

A flow of the type in which **significant forces** are **gravity force, pressure force and inertia force,** is found **when a free surface is present.**

Examples can be

the flow of a liquid in an open channel.
the wave motion caused by the passage of a ship through water.
the flows over weirs and spillways.
The condition for dynamic similarity of such flows requires

the equality of the Euler number Eu (the magnitude ratio of pressure to inertia force), and

the equality of the magnitude ratio of gravity to inertia force at corresponding points in the systems being compared.

Thus ,

$$\frac{\text{Gravity force}}{\text{Inertia Force}} = \frac{|\vec{F}_g|}{|\vec{F}_i|} \propto \frac{\rho l^3 g}{\rho V^2 l^2} = \frac{lg}{V^2}$$

In practice, it is often convenient to use the square root of this ratio so to deal with the first power of the velocity.

From a physical point of view, equality of $(lg)^{1/2}/V$ implies equality of lg/V^2 as regard to the concept of dynamic similarity.

The **reciprocal of the term** $(lg)^{1/2}/V$ **is known as Froude number** (after William Froude who first suggested the use of this number in the study of naval architecture.)

$$\frac{|\vec{F}_s|}{|\vec{F}_i|} \propto \frac{\sigma l}{\rho V^2 l^2} = \frac{\sigma}{\rho V^2 l} \qquad (18.2g)$$

Hence **Froude number,** $Fr = V/(lg)^{1/2}$ **.**

Therefore, **the primary requirement for dynamic similarity between the prototype and the model involving flow of fluid with gravity as the significant force, is the equality of Froude number, Fr,** i.e.,

$$\frac{(l_p g_p)^{1/2}}{V_p} = \frac{(l_m g_m)^{1/2}}{V_m}$$

Dynamic Similarity of Flows with Surface Tension as the Dominant Force

Surface tension forces are important in certain classes of practical problems such as ,

flows in which capillary waves appear
flows of small jets and thin sheets of liquid injected by a nozzle in air
flow of a thin sheet of liquid over a solid surface.
Here the significant parameter for dynamic similarity is the magnitude ratio of the surface tension force to the inertia force.

This can be written as

The term $\sigma/\rho V^2 l$ **is usually known as Weber number, Wb** (after the German naval architect Moritz Weber who first suggested the use of this term as a relevant parameter.)

Thus for dynamically similar flows $(Wb)_m = (Wb)_p$

$$\frac{\sigma_m}{\rho_m V_m^2 L_m} = \frac{\sigma_p}{\rho_p V_p^2 L_p^2}$$

Dynamic Similarity of Flows with Elastic Force

When the compressibility of fluid in the course of its flow becomes significant, the elastic force along with the pressure and inertia forces has to be considered.Therefore, the magnitude ratio of inertia to elastic force becomes a relevant parameter for dynamic similarity under this situation.

Thus we can write,

$$\boxed{\frac{\text{Inertia force}}{\text{Elastic Force}} = \frac{|\vec{F}_i|}{|\vec{F}_e|} \propto \frac{\rho V^2 l^2}{E l^2} = \frac{\rho V^2}{E}}$$

The parameter $\rho V^2/E$ **is known as Cauchy number** ,(after the French mathematician A.L. Cauchy)If we consider the **flow to be isentropic** , then it can be written

$$\frac{|\vec{F}_i|}{|\vec{F}_e|} \propto \frac{\rho V^2}{E_s}$$

(where E_S is the isentropic bulk modulus of elasticity)

Thus for dynamically similar flows $(\text{cauchy})_m = (\text{cauchy})_p$

$$\frac{\rho_m V_m^2}{(E_s)_m} = \frac{\rho_p V_p^2}{(E_s)_p}$$

The velocity with which a sound wave propagates through a fluid medium equals to $\sqrt{E_s/\rho}$ .Hence, the term $\rho V^2/E_s$ can be written as V^2/a^2 where **a is the acoustic velocity** in the fluid medium.

The ratio V/a is known as Mach number, Ma (after an Austrian physicist Earnst Mach)

It has been shown in Chapter 1 that the effects of compressibility become important when the Mach number exceeds 0.33.

The situation arises in the flow of air past high-speed aircraft, missiles, propellers and rotory compressors. In these cases equality of Mach number is a condition for dynamic similarity. Therefore,

$$(\text{Ma})_p = (\text{Ma})_m$$

i.e. $\boxed{V_p/a_p = V_m/a_m}$

Ratios of Forces for Different Situations of Flow

$\Delta p / \rho V^2$	$\dfrac{\text{Pressure force}}{\text{Inertia force}}$	Euler number Eu
$V / (lg)^{1/2}$	$\dfrac{\text{Inertia force}}{\text{Gravity force}}$	Froude number Fr
$\sigma / \rho V^2 l$	$\dfrac{\text{Surface Tesion force}}{\text{Inertia force}}$	Weber number Wb
$V / \sqrt{E_s / \rho}$	$\dfrac{\text{Inertia force}}{\text{Elastic force}}$	Mach number Ma

Buckingham's Pi Theorem

Assume, a physical phenomenon is described by **m number of independent variables like x_1 , x_2 , x_3 , ..., x_m**

The phenomenon may be expressed analytically by an implicit functional relationship of the controlling variables as

$$f(x_1, x_2, x_3, \ldots\ldots, x_m) = 0 \tag{19.2}$$

Now if **n be the number of fundamental dimensions like mass, length, time, temperature etc** ., involved in these m variables, then according to Buckingham's p theorem -

The phenomenon can be described in terms of (m - n) independent dimensionless groups like π_1 ,π_2 , ..., π_{m-n} , where p terms, represent the dimensionless parameters and consist of

different combinations of a number of dimensional variables out of the m independent variables defining the problem.

Therefore. the analytical version of the phenomenon given by Eq. (19.2) can be reduced to

$$\boxed{F(\pi_1, \pi_2, \ldots\ldots\ldots, \pi_{m-n}) = 0} \qquad (19.3)$$

according to Buckingham's pi theorem

This physically implies that the **phenomenon** which is basically described by m independent dimensional variables, **is ultimately controlled by (m-n) independent dimensionless parameters known as π terms.**

Alternative Mathematical Description of (π) Pi Theorem

A physical problem described by m number of variables involving n number of fundamental dimensions (n < m) leads to a system of n linear algebraic equations with m variables of the form

$$a_{11}x_1 + a_{12}x_2 + \ldots\ldots a_{1m}x_m = b_1$$
$$a_{21}x_1 + a_{22}x_2 + \ldots\ldots a_{2m}x_m = b_2$$
$$a_{n1}x_1 + a_{n2}x_2 + \ldots\ldots a_{nm}x_m = b_n$$

or in a matrix form,

$$\boxed{Ax = b} \qquad (19.5)$$

$$A = \begin{bmatrix} a_{11} & a_{12} & \cdots & a_{1m} \\ a_{21} & a_{22} & \cdots & a_{2m} \\ a_{n1} & a_{n2} & \cdots & a_{nm} \end{bmatrix}$$

Determination of π terms

A group of n (n = number of fundamental dimensions) variables out of m (m = total number of independent variables defining the problem) variables is first chosen to form a basis so that all n dimensions are represented . These n variables are referred to as repeating variables.

Then the p terms are formed by the product of these repeating variables raised to arbitrary unknown integer exponents and anyone of the excluded (m -n) variables.

*For **example*** , if x_1 x_2 ...x_n are taken as the repeating variables. Then

$$\pi_2 = x_1^{a_1} x_2^{a_2} \ldots\ldots x_n^{a_n} x_{n+2}$$

$$\pi_1 = x_1^{a_1} x_2^{a_2} \ldots\ldots x_n^{a_n} x_{n+1}$$

$$\pi_{m-n} = x_1^{a_1} x_2^{a_2} \ldots\ldots x_n^{a_n} x_m$$

The sets of integer exponents **a_1, a_2 . . . a_n are different for each p term.**

Since p terms are dimensionless, it requires that when all the variables in any p term are expressed in terms of their fundamental dimensions, the exponent of all the fundamental dimensions must be zero.

This leads to a system of n linear equations in a, a_2 . . . a_n which gives a unique solution for the exponents. This gives the values of a_1 a_2 . . . a_n for each p term and hence the p terms are uniquely defined.
In selecting the repeating variables, the following points have to be considered:

The repeating variables must include among them all the n fundamental dimensions, not necessarily in each one but collectively.

The dependent variable or the output parameter of the physical phenomenon should not be included in the repeating variables.
No physical phenomena is represented when -

- **m < n** because there is no solution **and**

m = n because there is a unique solution of the variables involved and hence all the parameters have fixed values.
. **Therefore all feasible phenomena are defined with m > n .**

When m = n + 1, then, according to the Pi theorem, the number of pi term is one and the phenomenon can be expressed as

$$f(\pi_1) = 0$$

where, the non-dimensional term π_1 is some specific combination of n + 1 variables involved in the problem.

When m > n+ 1 ,

the number of π terms are more than one.
A number of choices regarding the repeating variables arise in this case.
Again, it is true that if one of the repeating variables is changed, it results in a different set of π terms. Therefore the interesting question is **which set of repeating variables is to be chosen** , to arrive at the correct set of π terms to describe the problem. The **answer to this question lies in**

the fact that different sets of π terms resulting from the use of different sets of repeating variables are not independent. Thus, anyone of such interdependent sets is meaningful in describing the same physical phenomenon.

From any set of such π terms, one can obtain the other meaningful sets from some combination of the π terms of the existing set without altering their total numbers (m-n) as fixed by the Pi theorem.

Navier-Strokes Equation

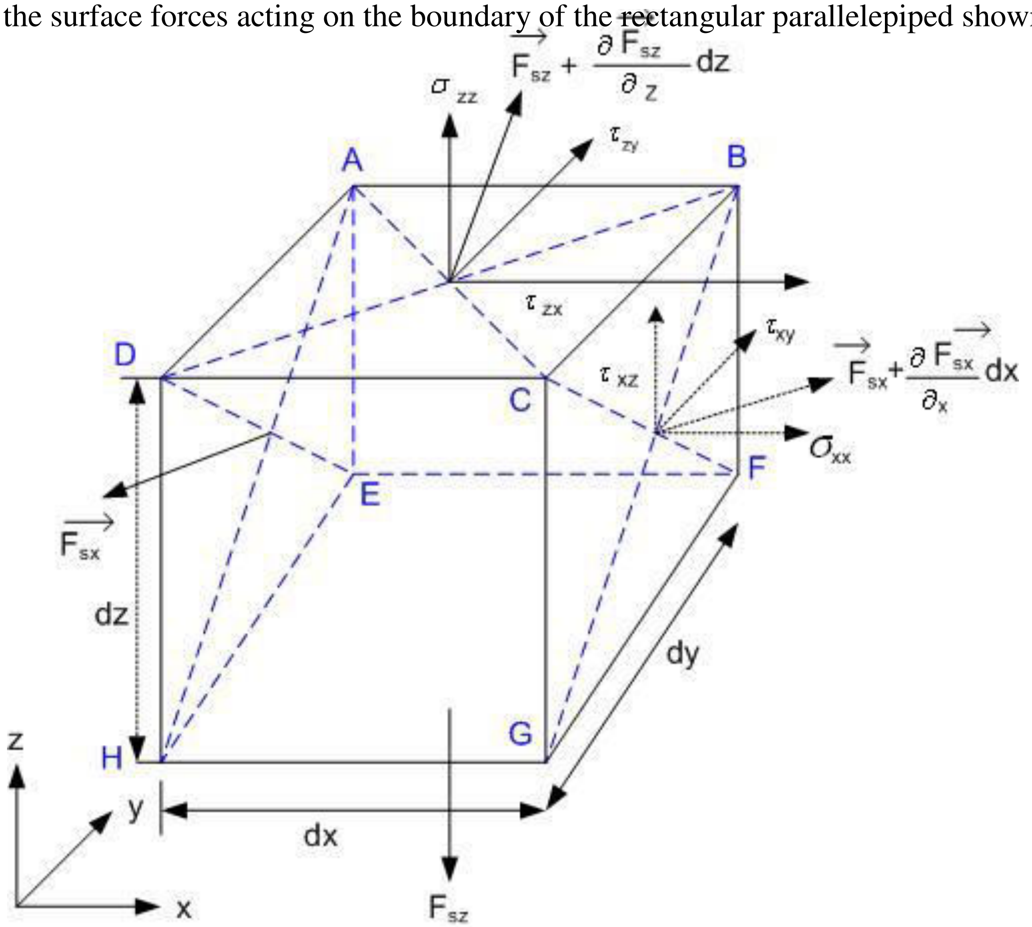
Generalized equations of motion of a real flow named after the inventors CLMH Navier and GG Stokes are derived from the **Newton's second law**

Newton's second law states that the **product of mass and acceleration is equal to sum of the external forces acting on a body.**

External forces are of two kinds-

- one acts throughout the mass of the body ----- **body force** (gravitational force, electromagnetic force)
- another acts on the boundary--------------------- **surface force** (pressure and frictional force).

Objective - We shall consider a differential fluid element in the flow field (Fig. 24.1). Evaluate the surface forces acting on the boundary of the rectangular parallelepiped shown below.



$$\vec{F}_{sx} = \hat{i}\,\sigma_{xx} + \hat{j}\,\tau_{xy} + \hat{k}\,\tau_{xz}$$

Fig. 24.1 Definition of the components of stress and their locations in a differential fluid element

Let the body force per unit mass be

$$\vec{f}_b = \hat{i}\,f_x + \hat{j}\,f_y + \hat{k}\,f_z$$

and surface force per unit volume be

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

Consider surface force on the surface AEHD, per unit area,
[Here second subscript x denotes that the surface force is evaluated for the surface whose outward normal is the x axis]

Surface force on the surface *BFGC* per unit area is

$$\vec{F}_{sx} + \frac{\partial \vec{F}_{sx}}{\partial x} dx$$

Net force on the body due to imbalance of surface forces on the above two surfaces is

$$\frac{\partial \vec{F}_{sx}}{\partial x}\,dx\,dy\,dz \tag{24.8}$$

(since area of faces AEHD and BFGC is dydz)

Total force on the body due to net surface forces on all six surfaces is

$$\left(\frac{\partial \vec{F}_{sx}}{\partial x} + \frac{\partial \vec{F}_{sy}}{\partial y} + \frac{\partial \vec{F}_{sz}}{\partial z}\right) dx\,dy\,dz \tag{24.9}$$

And hence, the resultant surface force *dF*, per unit volume, is

$$d\vec{F} = \frac{\partial \vec{F}_{sx}}{\partial x} + \frac{\partial \vec{F}_{sy}}{\partial y} + \frac{\partial \vec{F}_{sz}}{\partial z}$$

24.10)

(since Volume= dx dy dz)

The quantities $\vec{F}_{sx}$, $\vec{F}_{sy}$ and $\vec{F}_{sz}$ are vectors which can be resolved into normal stresses denoted by σ and shearing stresses denoted by τ as

$$\vec{F}_{sx} = \hat{i}\,\sigma_{xx} + \hat{j}\,\tau_{xy} + \hat{k}\,\tau_{xz}$$
$$\vec{F}_{sy} = \hat{i}\,\tau_{yx} + \hat{j}\,\sigma_{yy} + \hat{k}\,\tau_{yz}$$

$$\vec{F}_{sz} = \hat{i}\,\tau_{zx} + \hat{j}\,\tau_{zy} + \hat{k}\,\sigma_{zz}$$

The **stress system has nine scalar quantities. These nine quantities form a stress tensor.**

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

Consider a general flow field as represented in Fig. 25.1.

Imagine a closed control volume, $\forall_0$ within the flow field. The control volume is fixed in space and the fluid is moving through it. The control volume occupies reasonably large finite region of the flow field.A control surface , A_0 is defined as the surface which bounds the volume $\forall_0$.

According to Reynolds transport theorem, "The rate of change of momentum for a system equals the sum of the rate of change of momentum inside thecontrol volume and the rate of efflux of momentum across the control surface".The rate of change of momentum for a system (in our case, the control volume boundary and the system boundary are same) is equal to the net external force acting on it.**Now**, we shall transform these statements into equation by accounting for each term,

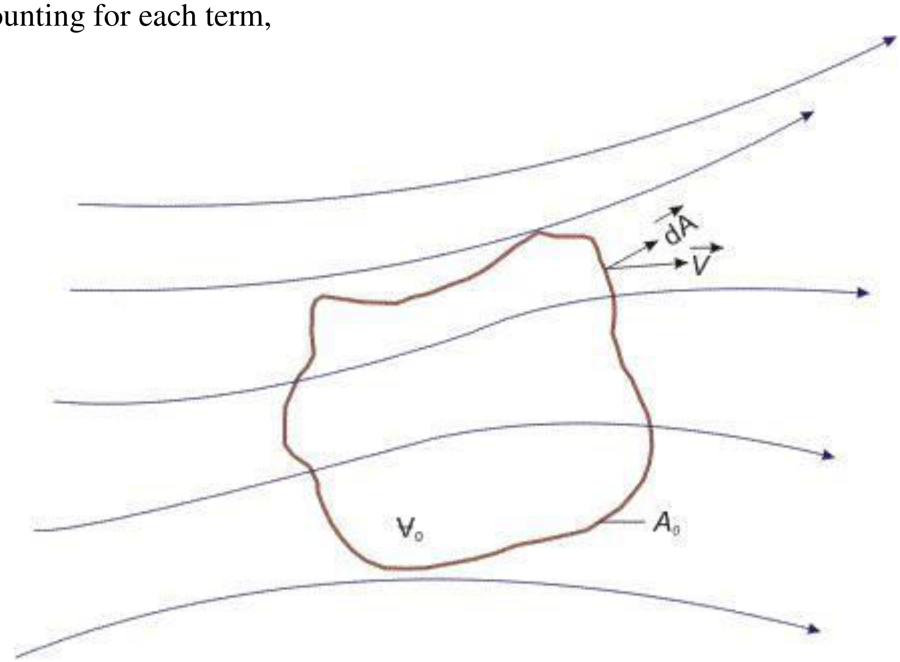


FIG 25.1 Finite control volume fixed in space with the fluid moving through it

Rate of change of momentum inside the control volume

$$= \frac{\partial}{\partial t}\int\int\int_{\forall_0} \rho\vec{V}d\forall$$

$$= \int\int\int_{\forall_0} \frac{\partial}{\partial t}(\rho\vec{V})d\forall$$

(since t is independent of space variable)

Rate of efflux of momentum through control surface

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

$$= \int\int\int_{\forall_0} \left(\vec{V}(\nabla.\rho\vec{V}) + \rho\vec{V}.\nabla\vec{V}\right)d\forall$$

Surface force acting on the control volume

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

(σ is symmetric stress tensor)

Body force acting on the control volume

$$\int\int\int(\nabla.\sigma)d\forall$$

$$\int\int\int_{\forall_0} \rho\vec{f}_b d\forall$$

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

Finally, we get,

$$\int\int\int_{\forall_0} \left(\frac{\partial}{\partial t}(\rho\vec{V}) + \left\{\vec{V}(\nabla.\rho\vec{V}) + \rho\vec{V}.\nabla\vec{V}\right\}\right)d\forall$$

$$= \int\int\int_{\forall_0} \left(\nabla.\sigma + \rho\vec{f}_b\right)d\forall$$

or

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

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

Invoking this relationship in Eq. (25.5), we obtain

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

$$\rho \frac{D\vec{V}}{Dt} = \nabla.\sigma + \rho \vec{f}_b$$

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

Equation (25.6) is referred to as **Cauchy's equation of motion** . In this equation, σ is the stress tensor,

After having substituted σ we get

$$\nabla.\sigma = -\nabla p + (\mu' + \mu)\nabla(\nabla.\vec{V}) + \mu \nabla^2 \vec{V} \quad (25.8)$$

$$\mu' + \frac{2}{3}\mu = 0$$

(25.9)

From **Stokes's hypothesis** we get,

Invoking above two relationships into Eq.(25.6) we get

$$\rho \frac{D\vec{V}}{Dt} = -\nabla p + \mu \nabla^2 \vec{V} + \frac{1}{3}\mu \nabla(\nabla.\vec{V}) + \rho \vec{f}_b \quad (25.10)$$

This is the most general form of Navier-Stokes equation.

Exact Solutions Of Navier-Stokes Equations

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

We **choose** x **to be the direction along which all fluid particles travel** ,

i.e. $u \neq 0, v = w = 0$. Invoking this in continuity equation, we get

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

$$\frac{\partial u}{\partial z} = 0$$

which means $u = u(y, z, t)$

Now. Navier-Stokes equations for incompressible flow become

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

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

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

So, we obtain

$$\frac{\partial p}{\partial t} = \frac{\partial p}{\partial z} = 0$$

which means $p = p(x)\ alone$

$$\frac{\partial u}{\partial t} = -\frac{1}{\rho}\cdot\frac{\partial p}{\partial x} + \nu\left[\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right] \qquad (25.11)$$

Introduction

The **boundary layer** of a flowing fluid is **the thin layer close to the wall**
In a flow field, **viscous stresses are very prominent within this layer**.
Although the layer is thin, it is very important to know the details of flow within it.

The **main-flow velocity** within this layer **tends to zero** while approaching the wall (**no-slip condition**).

Also the gradient of this velocity component in a direction normal to the surface is large as compared to the gradient in the streamwise direction.

Boundary Layer Equations

In 1904, **Ludwig Prandtl,** the well known German scientist, introduced the concept of boundary layer and **derived the equations for boundary layer flow** by correct reduction of Navier-Stokes equations.
He hypothesized that **for fluids having relatively small viscosity**, **the effect of internal friction in the fluid is significant only in a narrow region surrounding solid boundaries or bodies over which the fluid flows.**

Thus, close to the body is the boundary layer where **shear stresses exert an increasingly larger effect** on the fluid **as one moves from free stream towards the solid boundary.**
However, **outside the boundary layer where the effect of the shear stresses on the flow is small compared to values inside the boundary layer (since the velocity gradient** $\partial u / \partial y$ **is negligible**),---------

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

Hence, the ***surface*** **at the boundary layer interface** is a rather fictitious one,

that **divides rotational and irrotational flow.** Fig 28.1 shows Prandtl's model regarding boundary layer flow.

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

About the boundary layer

Boundary layer **thickness is** δ which is a **function of** the coordinate direction ***x*** .

The thickness is considered to be **very small compared to the characteristic length *L*** of the domain.

In the normal direction, **within this thin layer**, the gradient $\partial u / \partial y$ **is very large compared to** the gradient in the flow direction $\partial u / \partial x$ **.**

Now we take up the Navier-Stokes equations for : steady, two dimensional, laminar, incompressible flows.

Considering the Navier-Stokes equations together with the equation of continuity, the following dimensional form is obtained.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \tag{28.1}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \tag{28.2}$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{28.3}$$

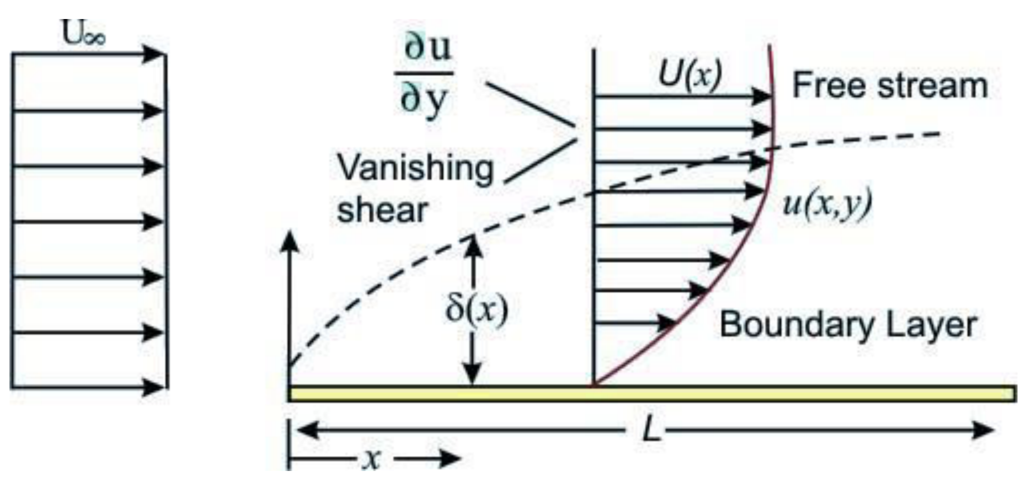


Fig 28.1 Boundary layer and Free Stream for Flow Over a flat plate

u - velocity component along x direction.
v - velocity component along y direction
p - static pressure
ρ - density.
μ - dynamic viscosity of the fluid

The equations are now non-dimensionalised. $x^* = \frac{x}{L}, y^* = \frac{y}{L}$

The l**ength and the velocity scales** are chosen as ***L* and U_∞ respectively.**

The non-dimensional variables are:

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

where U_∞ is the dimensional free stream velocity and the pressure is non-dimensionalised by twice the dynamic pressure $p_d = (1/2)\rho U_\infty^2$.

Using these non-dimensional variables, the Eqs (28.1) to (28.3) become

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

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

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

where the Reynolds number,

$$\mathrm{Re} = \frac{\rho U_\infty L}{\mu}$$

Order of Magnitude Analysis

Let us examine what happens to the *u* velocity as we go across the boundary layer. At the **wall** the *u* velocity is **zero** [with respect to the wall and absolute zero for a stationary wall (which is normally implied if not stated otherwise)].

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

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

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

Variable	*Dimensional scale*	*Non-dimensional scale*

y	δ	$\varepsilon = \delta / L$
[redacted]	[redacted]	[redacted]

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

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

One **general rule** of incompressible fluid mechanics is that **we are not allowed to drop any term from the continuity equation**.

- From the scales of boundary layer variables, the derivative is of the order 1.
- The second term in the continuity equation should also be of the order 1.The reason being has to be of the order because becomes at its maximum.

$\partial u^* / \partial x^*$

Eq 28.4 - x direction momentum equation ν ϵ

y^* $\varepsilon(=\delta / L)$

- Inertia terms are of the order 1.
- is of the order 1
- $\partial^2 u / \partial x^2$ is of the order .

$\partial^2 u / \partial y^2$ $(1/\varepsilon^2)$

However after multiplication with 1/Re, the sum of the two second order derivatives should produce at least one term which is of the same order of magnitude as the inertia terms. This is possible only if the Reynolds number (Re) is of the order of .

- It follows from that $-\partial p^* / \partial x^*$ will not exceed the order of 1 so as to be in balance with the remaining term.

- Finally, Eqs (28.4), (28.5) and (28.6) can be rewritten as

$$u^* \frac{\partial u^*}{\partial x^*} + v^* \frac{\partial u^*}{\partial y^*} = -\frac{\partial p^*}{\partial x^*} + \frac{1}{\mathrm{Re}}\left[\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}}\right] \tag{28.4}$$

$$(1)\frac{(1)}{(1)} \qquad (\varepsilon)\frac{(1)}{(\varepsilon)} = (1) \qquad (\varepsilon^2)\left[\frac{(1)}{(1)} + \frac{1}{(\varepsilon^2)}\right]$$

$$u^* \frac{\partial v^*}{\partial x^*} + v^* \frac{\partial v^*}{\partial y^*} = -\frac{\partial p^*}{\partial x^*} + \frac{1}{\mathrm{Re}}\left[\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{\partial^2 v^*}{\partial y^{*2}}\right] \tag{28.5}$$

$$(1)\frac{(\varepsilon)}{(1)} \qquad (\varepsilon)\frac{(\varepsilon)}{(\varepsilon)} = (?) \qquad (\varepsilon^2)\left[\frac{(\varepsilon)}{(1)} + \frac{\varepsilon}{(\varepsilon^2)}\right]$$

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

$$\frac{(1)}{(1)} \qquad \frac{(\varepsilon)}{(\varepsilon)}$$

$$p+\frac{1}{2}\rho U^2 =$$

As a consequence of the order of magnitude analysis, $\partial^2 u^* / \partial x^{*2}$ can be dropped from

the *x* direction momentum equation, because on multiplication with $1/Re$ it assumes the smallest order of magnitude.

Eq 28.5 - *y* direction momentum equation.

All the terms of this equation are of a smaller magnitude than those of Eq. (28.4).

This equation can only be balanced if $\partial p^* / \partial y^*$ is of the same order of magnitude as other terms.

Thus they momentum equation reduces to

$$\frac{\partial p^*}{\partial y^*} = O(\varepsilon) \qquad (28.7)$$

This means that the **pressure across the boundary layer does not change**.

The **pressure is impressed on the boundary layer**, and its value is determined by hydrodynamic considerations.

This also implies that the **pressure *p* is only a function of *x***. The pressure forces on a body are solely **determined by the inviscid flow outside the boundary layer.**

The application of Eq. (28.4) at the outer edge of boundary layer gives

$$u^*\frac{\partial u^*}{\partial x^*} = -\frac{\partial p^*}{\partial x^*} \qquad (28.8a)$$

In dimensional form, this can be written as

$$U\frac{dU}{dx} = -\frac{1}{\rho}\frac{\partial p}{\partial x} \qquad (28.8b)$$

On integrating Eq (28.8b) the well known Bernoulli's equation is obtained *a constant*

Finally, it can be said that by the order of magnitude analysis, the Navier-Stokes equations are simplified into equations given below.

$$u^*\frac{\partial u^*}{\partial x^*}+v^*\frac{\partial u^*}{\partial y^*}=-\frac{\partial p^*}{\partial x^*}+\frac{1}{\mathrm{Re}}\frac{\partial^2 u^*}{\partial y^{*2}} \tag{28.10}$$

- $$\frac{\partial p^*}{\partial y^*}=0$$

$$\frac{\partial u^*}{\partial x^*}+\frac{\partial v^*}{\partial y^*}=0 \tag{28.12}$$

These are known as Prandtl's boundary-layer equations.

The available boundary conditions are:

Solid surface

$at\ y^*=0, u^*=0=v^*$

$at\ y=0, u=0=v$

Outer edge of boundary-layer

or $at\ y=\delta, u=U(x)$ (28.14)

The unknown pressure *p* in the x-momentum equation can be determined from Bernoulli's Eq. (28.9), if the inviscid velocity distribution *U(x)* is also known.
We solve the Prandtl boundary layer equations for $u^*(x,y)$ and $v^*(x,y)$ with *U* obtained from the outer inviscid flow analysis. The equations are solved by commencing at the leading edge of the body and moving downstream to the desired locationit allows the no-slip boundary condition to be satisfied which constitutes a significant improvement over the potential flow analysis while solving real fluid flow problems.

The **Prandtl boundary layer equations** are thus a simplification of the Navier-Stokes equations.

Boundary Layer Coordinates

The boundary layer equations derived are in Cartesian coordinates.
The Velocity components u and v represent x and y direction velocities respectively.
For objects with small curvature, these equations can be used with -x coordinate : streamwise direction y coordinate : normal component

y coordinate : normal component

They are called **Boundary Layer Coordinates.**

Application of Boundary Layer Theory

The Boundary-Layer Theory is not valid beyond the point of separation.

At the point of separation, boundary layer thickness becomes quite large for the thin layer approximation to be valid.

It is important to note that boundary layer theory can be used to locate the point of seperation itself.

In applying the boundary layer theory although U is the free-stream velocity at the outer edge of the boundary layer, it is interpreted as the fluid velocity at the wall calculated from inviscid flow considerations (known as **Potential Wall Velocity**)

Mathematically, application of the boundary - layer theory converts the character of governing Navier-Stroke equations from elliptic to parabolic

This allows the marching in flow direction, as the solution at any location is independent of the conditions farther downstream

Blasius Flow Over A Flat Plate

The classical problem considered by H. Blasius was

Two-dimensional, steady, incompressible flow over a flat plate at zero angle of incidence with respect to the uniform stream of velocity U_∞ .The fluid extends to infinity in all directions from the plate.The physical problem is already illustrated in Fig. 28.1

Blasius wanted to determinethe velocity field solely within the boundary layer,the boundary layer thickness (δ) ,the shear stress distribution on the plate, andthe drag force on the plate.The Prandtl boundary layer equations in the case under consideration are

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\nu\frac{\partial^2 u}{\partial y^2}$$

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

$$\frac{\partial u}{\partial x}+\frac{\partial u}{\partial y}=0 \qquad (28.15)$$

The boundary conditions are

- Note that the substitution of the term $-\frac{1}{\rho}\frac{dp}{dx}$ in the original boundary layer momentum equation in terms of the free stream velocity produces $U_\infty \frac{dU_\infty}{dx}$ which is equal to zero.

- Hence the governing Eq. (28.15) does not contain any pressure-gradient term.

- However, the characteristic parameters of this problem arethat U_∞, ν, x, y is, $u = u(U_\infty, \nu, x, y)$

- This relation has five variables. U_∞, ν, x, y

- It involves two dimensions, length and time.

- Thus it can be reduced to a dimensionless relation in terms of (5-2) =3 quantities (**Buckingham Pi Theorem**)

• Thus a similarity variables can be used to find the solution •
Such flow fields are called **self-similar flow field .**

Law of Similarity for Boundary Layer Flows

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

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

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

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

Thus in terms of x we get

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

i.e.,

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

$$\frac{u}{U_\infty} = F\left(\frac{y}{\sqrt{\frac{\nu x}{U_\infty}}}\right) = F(\eta)$$

where $\eta \sim \frac{y}{\delta}$ *and* or more precisely,

$$\eta = \frac{y}{\sqrt{\nu x}}$$

$$y = \eta\sqrt{\frac{\nu x}{U_\infty}} \qquad (28.19)$$

$$dy = \sqrt{\frac{\nu x}{U_\infty}}\,d\eta$$

$$\psi = \int u\,dy = \int U_\infty F(\eta)\sqrt{\frac{\nu x}{U_\infty}}\,d\eta = \sqrt{U_\infty \nu x}\int F(\eta)\,d\eta$$

The stream function can now be obtained in terms of the velocity components as

or $\psi = \sqrt{U_\infty \nu x}\, f(\eta) + \mathrm{D}$ (28.20) where D is a constant. Also $\int F(\eta)d\eta = f(\eta)$ and the constant of integration is zero if the stream function at the solid surface is set equal to zero.

Now, the velocity components and their derivatives are:

$$u = \frac{\partial \psi}{\partial y} = \frac{\partial \psi}{\partial \eta}\cdot\frac{\partial \eta}{\partial y} = U_\infty f'(\eta)$$

$$v = -\frac{\partial \psi}{\partial x} = -\sqrt{U_\infty v}\left[\frac{1}{2}\cdot\frac{1}{\sqrt{x}} f(\eta) + \sqrt{x} f'(\eta)\left\{-\frac{1}{2}\frac{y}{\sqrt{vx/U_\infty}}\frac{1}{x}\right\}\right] \qquad (28.21a)$$

$$v = \frac{1}{2}\sqrt{\frac{vU_\infty}{x}}\left[\eta f'(\eta) - f(\eta)\right]$$

$$\frac{\partial u}{\partial y} = U_\infty f''(\eta)\frac{\partial \eta}{\partial y} = U_\infty f''(\eta)\left[\frac{1}{\sqrt{vx/U_\infty}}\right]$$

$$\frac{\partial u}{\partial y} = U_\infty\sqrt{\frac{U_\infty}{vx}} f''(\eta)$$

$$\frac{\partial^2 u}{\partial y^2} = U_\infty\sqrt{\frac{U_\infty}{vx}} f''(\eta)\left\{\frac{1}{\sqrt{vx/U_\infty}}\right\}$$

$$\frac{\partial^2 u}{\partial y^2} = \frac{U_\infty^2}{vx} f'''(\eta)$$

Substituting (28.2) into (28.15), we have

$$-\frac{U_\infty^2}{2}\frac{\eta}{x} f'(\eta) f''(\eta) + \frac{U_\infty^2}{2x}\left[\eta f'(\eta) - f(\eta)\right] f''(\eta) = \frac{U_\infty^2}{x} f'''(\eta)$$

$$-\frac{1}{2}\frac{U_\infty^2}{x} f(\eta) f''(\eta) = \frac{U_\infty^2}{x} f'''(\eta)$$

$$\eta = \frac{y}{\sqrt{\frac{\nu x}{U_\infty}}}$$

and

This is known as Blasius Equation .

Contd. from Previous Slide

The boundary conditions as in Eg. (28.16), in combination with Eg. (28.21a) and (28.21b) become

$$\eta = 0 : f(\eta) = 0, f'(\eta) = 0$$

at $y = 0, u = 0$, therefore

at $y = \infty, u = U_\infty$ therefore $\eta = \infty : F(\eta) = f'(\eta) = 1$ (28.23)

Equation (28.22) is a **third order nonlinear differential equation .**

Blasius obtained the solution of this equation in the form of series expansion through analytical techniques

We shall not discuss this technique. However, we shall discuss a numerical technique to solve the aforesaid equation which can be understood rather easily.

Note that the **equation for** f **does not contain** x .

Boundary conditions at $x = 0$ **and** $y = \infty$ **merge into the condition** $\eta \rightarrow \infty, u/U_\infty = f' = 1$ **. This is the key feature of similarity solution.**

We can rewrite Eq. (28.22) as three first order differential equations in the following way

$$f' = G \tag{28.24a}$$

$$G' = H \tag{28.24b}$$

(28.24c)

Let us next consider the boundary conditions.

The condition $f(0) = 0$ remains valid.
The condition $f'(0) = 0$ means that $G(0) = 0$.
The condition $f'(\infty) = 1$ gives us $G(\infty) = 1$.

Note that the equations for *f* and *G* have initial values. However, the value for H(0) is not known. Hence, we do not have a usual initial-value problem.

Shooting Technique

We handle this problem as an initial-value problem by choosing values of $H(0)$ and solving by numerical methods $f(\eta), G(\eta)$, and $H(\eta)$.

In general, the condition $G(\infty) = 1$ will not be satisfied for the function G arising from the numerical solution.

We then choose other initial values of H so that eventually we find an $H(0)$ which results in $G(\infty) = 1$.

This method is called the shooting technique.

In Eq. (28.24), the primes refer to differentiation wrt. the similarity variable η. The integration steps following Runge-Kutta method are given below.

$$m_1 = hF_3(f_n, G_n, H_n, \eta_n)$$

$$k_2 = hF_1\left\{\left((f_n + \frac{1}{2}k_1), (G_n + \frac{1}{2}l_1), (H_n + \frac{1}{2}m_1), (\eta_n + \frac{h}{2})\right)\right\}$$

$$l_2 = hF_2\left\{\left((f_n + \frac{1}{2}k_1), (G_n + \frac{1}{2}l_1), (H_n + \frac{1}{2}m_1), (\eta_n + \frac{h}{2})\right)\right\} \qquad (28.25a)$$

$$m_2 = hF_3\left\{\left((f_n + \frac{1}{2}k_1), (G_n + \frac{1}{2}l_1), (H_n + \frac{1}{2}m_1), (\eta_n + \frac{h}{2})\right)\right\}$$

(28.25b)

$$H_{n+1} = H_n + \frac{1}{6}(m_1 + 2m_2 + 2m_3 + m_4) \qquad (28.25c)$$

$$f(\Delta\eta) = f(0) + G(0)\Delta\eta$$

$$G(\Delta\eta) = G(0)$$

One moves from η_n to $\eta_{n+1} = \eta_n + h$. A fourth order accuracy is preserved if h is constant along the integration path, that is, $\eta_{n+1} - \eta_n = h$ for all values of n. The values of k, l and m are as follows.

$$H(\Delta\eta) = H(0) + H'(0)\Delta\eta$$

$$H'(\Delta\eta) = -\frac{1}{2}f(\Delta\eta)H(\Delta\eta)$$

For generality let the system of governing equations be

$$f''(0) = H(0) = \lambda$$

$$f'(\infty) = G(\infty) = 1$$

$$f' = F_1(f, G, H, \eta), G' = F_2(f, G, H, \eta)$$

$$k_1 = hF_1(f_n, G_n, H_n, \eta_n)$$

$$l_1 = hF_2(f_n, G_n, H_n, \eta_n)$$

η $\eta = 10$ λ

In a similar way K_3, l_3, m_3 and k_4, l_4, m_4 mare calculated following standard formulae for the Runge-Kutta integration. For example, K_3 is given by

G η

$$k_3 = hF_1\left\{\left((f_n + \frac{1}{2}k_2), \ldots, (H_n + \frac{1}{2}m_2), (\eta_n + \frac{h}{2})\right)\right\}$$

$G(\infty)$ $H(0)$ G

The functions F_1, F_2 and F_3 are G, H , $-fH/2$ respectively. Then at a distance $\Delta\eta$ from the wall, we have

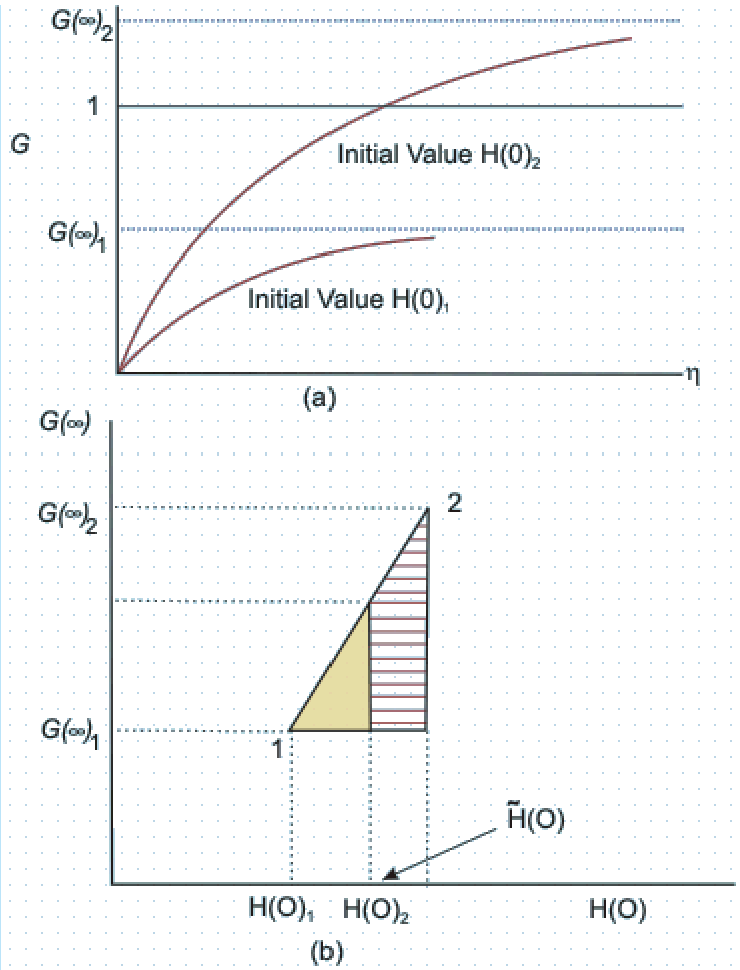
$H(0)$ $H(0)$ $G(\infty) = 1$

$$\frac{\tilde{H}(0) - H(0)_1}{1 - G(\infty)_1} = \frac{H(0)_2 - H(0)_1}{G(\infty)_2 - G(\infty)_1}$$

$\tilde{H}(0)$

- Next we repeat the same calculation as above by using $H(0)$ and the better of the two initial values of $\tilde{H}(0)$. Thus we get another improved value $\tilde{H}(0)$. This process may continue, that is, we use $\tilde{H}(0)$ and $\tilde{H}(0)$ as a pair of values to find more improved values for $H(0)$, and so forth. The better guess for H (0) can also be obtained by using the Newton Raphson Method. It should be always kept in mind that for each value of $H(0)$, the curve $G(\eta)$ versus η is to be examined to get the proper value of $G(\infty)$.
- The functions $f(\eta), f'(\eta) = G$ and $f''(\eta) = H$ are plotted in Fig. 28.3.The velocity components, *u* and *v* inside the boundary layer can be computed from Eqs (28.21a) and (28.21b) respectively.

- A sample computer program in FORTRAN follows in order to explain the solution procedure in greater detail. The program uses Runge Kutta integration together with the Newton Raphson method



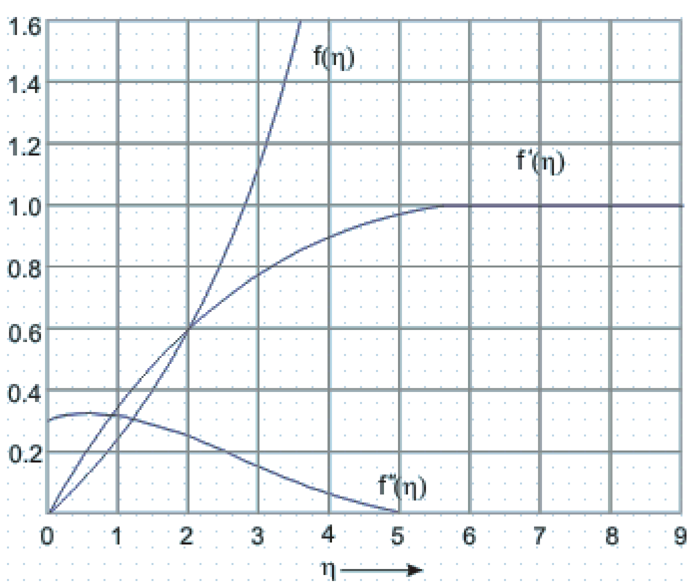


Fig 28.2 Correcting the initial guess for H(O)

Measurements to test the accuracy of theoretical results were carried out by many scientists. In his experiments, J. Nikuradse, found excellent agreement with the theoretical results with respect to velocity distribution (u/U_∞) within the boundary layer of a stream of air on a flat plate.

In the next slide we'll see some values of the velocity profile shape $f'(\eta) = u/U_\infty = G$ and $f''(\eta) = H$ in tabular format.

Values of the velocity profile shape $f'(\eta) = u/U_\infty = G$ and $f''(\eta) = H$

Table 28.1 The Blasius Velocity Profile $G = u/U_\infty$ f and H

0	0	0	0.33206
0.2	0.00664	0.006641	0.33199
0.4	0.02656	0.13277	0.33147
0.8	0.10611	0.26471	0.32739
1.2	0.23795	0.39378	0.31659
1.6	0.42032	0.51676	0.29667
2.0	0.65003	0.62977	0.26675
2.4	0.92230	0.72899	0.22809

2.8	1.23099	0.81152	0.18401
3.2	1.56911	0.87609	0.13913
3.6	1.92954	0.92333	0.09809
4.0	2.30576	0.95552	0.06424
4.4	2.69238	0.97587	0.03897
4.8	3.08534	0.98779	0.02187
5.0	3.28329	0.99155	0.01591
8.8	7.07923	1.00000	0.00000

Wall Shear Stress

With the profile known, wall shear can be evaluated as

$$\tau_w = \mu \left.\frac{\partial u}{\partial y}\right|_{y=0}$$

$$\frac{\partial u}{\partial y} = U_\infty f'(\eta)\frac{\partial \eta}{\partial y}$$

$$\tau_\omega = \mu U_\infty f''(\eta)\left.\frac{\partial \eta}{\partial y}\right|_{\eta=0}$$

Now, $$= \mu U_\infty H \left.\frac{\partial \eta}{\partial y}\right|_{\eta=0}$$

$$\tau_\omega = \mu U_\infty \times 0.3326 \times \frac{1}{\sqrt{(\nu x)/U_\infty}}$$

$$[f''(0) = 0.3326]$$

or

from Table 28.1

$$\tau_\omega = \frac{0.332\rho U_\infty^2}{\sqrt{\mathrm{Re}_x}}$$

(29.1a)

(Wall Shear Stress)

$$C_{fx} = \frac{\tau_\omega}{1/2\rho U_\infty^2}$$

and the local skin friction coefficient is

Substituting from (29.1a) we get

$$C_{fx} = \frac{0.664}{\sqrt{\mathrm{Re}_x}}$$

(Skin Friction Coefficient)

In 1951, Liepmann and Dhawan , measured the shearing stress on a flat plate directly. Their results showed a striking confirmation of Eq. (29.1).

Total frictional force per unit width for the plate of length L is

$$F = \int_0^L \tau_\omega dx$$

$$F = \int_0^L \frac{0.332 \rho U_\infty^2}{\sqrt{U_\infty / \nu}} \frac{dx}{\sqrt{x}}$$

or

$$F = \left[\frac{0.332 \rho U_\infty^2}{\sqrt{U_\infty / \nu}} \times \frac{x^{1/2}}{1/2} \right]_0^L$$

or

$$F = 0.664 \times \rho U_\infty^2 \sqrt{\frac{\nu L}{U_\infty}} \qquad (29.2)$$

and the average skin friction coefficient is

$$\overline{C_f} = \frac{F}{1/2\left(\rho U_\infty^2 L\right)} = \frac{1.328}{\sqrt{\mathrm{Re}_L}}$$

where, $\mathrm{Re}_L = U_\infty L/\nu$.

For a flat plate of length L in the streamwise direction and width w perpendicular to the flow, the Drag D would be

$$D = F(2wL) = 0.664(2wL)\rho U_\infty^2\left(\frac{\nu L}{U_\infty}\right)^{1/2} = 1.328wL\left(\frac{\rho\mu U_\infty^3}{L}\right)^{1/2}$$

Boundary Layer Thickness

- Since $u/U_\infty \to 0.99, as \;\; y \to \infty$, it is customary to select the boundary layer thickness δ as that point where u/U_∞ **approaches 0.99.**
- From Table 28.1, u/U_∞ **reaches 0.99 at η= 5.0** and we can write

$$\delta \Big/ \sqrt{\left(\frac{\nu x}{U_\infty}\right)} \approx 5.0$$

$$or \quad \delta \approx 5.0\sqrt{\left(\frac{\nu x}{U_\infty}\right)} = \frac{5.0x}{\sqrt{\mathrm{Re}_x}} \qquad (29.5)$$

However, the aforesaid definition of boundary layer thickness is somewhat arbitrary, a physically more meaningful measure of boundary layer estimation is expressed through **displacement** thickness .

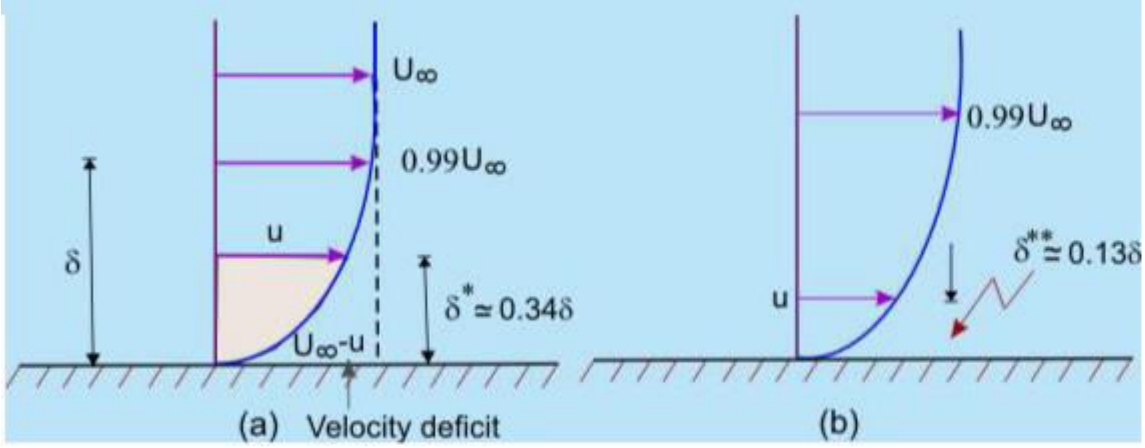


Fig. 29.1 (Displacement thickness) (b) Momentum thickness

Displacement thickness (δ^*) **:** It is defined as the distance by which the external potential flow is displaced outwards due to the decrease in velocity in the boundary layer.

$$U_\infty \delta^* = \int_0^\infty (U_\infty - u)dy$$

$$\delta^* = \int_0^\infty \left(1 - \frac{u}{U_\infty}\right)dy \qquad (29.6)$$

Therefore,

$$dy = \delta d\eta = \sqrt{\frac{\nu x}{U_\infty}} d\eta$$

$$\delta^* = \sqrt{\frac{\nu x}{U_\infty}} \int_0^\infty (1 - f')d\eta = \sqrt{\frac{\nu x}{U_\infty}} \lim_{\eta \to \infty} [\eta - f(\eta)]$$

$$\delta^* = 1.7208 \sqrt{\frac{\nu x}{U_\infty}} = \frac{1.7208x}{\sqrt{\mathrm{Re}_x}}$$

Substituting the values of u/U_∞ and η from Eqs (28.21a) and (28.19) into Eq.(29.6), we obtain

Following the analogy of the displacement thickness, a momentum thickness may be defined.

Momentum thickness (δ^{**}): It is defined as the loss of momentum in the boundary layer as compared with that of potential flow. Thus

$$\rho U_\infty^2 \delta^{**} = \int_0^\infty \rho u (U_\infty - u) dy$$

$$\delta^{**} = \int_0^\infty \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right) dy$$

With the substitution of (u/U_∞) and η from Eg. (28.21a) and (28.19), we can evaluate numerically the value of δ^{**} for a flat plate as

$$\delta^{**} = \sqrt{\frac{\nu x}{U_\infty}} \int_0^\infty f'(1 - f') d\eta$$

$$or \quad \delta^{**} = 0.664\sqrt{\frac{\nu x}{U_\infty}} = \frac{0.664x}{\sqrt{\mathrm{Re}_x}}$$

The relationships between $\delta, \delta^{*} and\, \delta^{**}$ have been shown in Fig. 29.1.

Momentum-Integral Equations For The Boundary Layer

To employ boundary layer concepts in real engineering designs, we need approximate methods that would quickly lead to an answer even if the accuracy is somewhat less.

Karman and Pohlhausen devised a simplified method by **satisfying only the boundary conditions of the boundary layer flow** rather than satisfying Prandtl's differential equations for each and every particle within the boundary layer. We shall discuss this method herein.

Consider the case of steady, two-dimensional and incompressible flow, i.e. we shall refer to Eqs (28.10) to (28.14). Upon integrating the dimensional form of Eq. (28.10) with respect to *y = 0* (wall) to *y* = δ (where δ signifies the interface of the free stream and the boundary layer), we obtain

$$\int_0^\delta \left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} \right) dy = \int_0^\delta \left(-\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu \frac{\partial^2 u}{\partial y^2} \right) dy$$

$$\int_0^\delta u\frac{\partial u}{\partial x} dy + \int_0^\delta v\frac{\partial u}{\partial y} dy = \int_0^\delta -\frac{1}{\rho}\frac{\partial p}{\partial x} dy + \int_0^\delta \nu \frac{\partial^2 u}{\partial y^2} dy$$

$$\int_0^\delta 2u\frac{\partial u}{\partial x}dy - U_\infty\int_0^\delta \frac{\partial u}{\partial x}dy - \int_0^\delta U_\infty\frac{dU_\infty}{dx}dy = -\left(\frac{\mu\left.\frac{\partial u}{\partial y}\right|_{y=0}}{\rho}\right)$$

The second term of the left hand side can be expanded as

$$\int_0^\delta\left(2u\frac{\partial u}{\partial x} - U_\infty\frac{\partial u}{\partial x} - U_\infty\frac{dU_\infty}{dx}\right)dy = -\frac{\tau_w}{\rho}$$

$$\int_0^\delta v\frac{\partial u}{\partial y}dy = [vu]_0^\delta - \int_0^\delta u\frac{\partial v}{\partial y}dy$$

or, $\int_0^\delta v\frac{\partial u}{\partial y}dy = U_\infty v_\delta + \int_0^\delta u\frac{\partial u}{\partial x}dy\left(\sin ce\,\frac{\partial u}{\partial x} = -\frac{\partial v}{\partial y}\right)$ by continuity equation

$$\int_0^\delta v\frac{\partial u}{\partial y}dy = -U_\infty\int_0^\delta \frac{\partial u}{\partial x}dy + \int_0^\delta u\frac{\partial u}{\partial x}dy$$

29.11

Substituting Eq. (29.11) in Eq. (29.10) we obtain

$$\int_0^\delta 2u\frac{\partial u}{\partial x}dy - U_\infty\int_0^\delta \frac{\partial u}{\partial x}dy = -\int_0^\delta \frac{1}{\rho}\frac{\partial p}{\partial x}dy - v\left.\frac{\partial u}{\partial y}\right|_{y=0}$$

(29.12)

Substituting the relation between $\frac{\partial p}{\partial x}$ and the free stream velocity U_∞ for the inviscid zone in Eq. (29.12) we get

which is reduced to

$$\int_0^\delta \frac{\partial}{\partial x}[u(U_\infty - u)]dy + \frac{dU_\infty}{dx}\int_0^\delta (U_\infty - u)dy = \frac{\tau_w}{\rho}$$

Since the integrals vanish outside the boundary layer, we are allowed to increase the integration limit to infinity (i.e $\delta = \infty$.)

$$\int_0^\delta \frac{\partial}{\partial x}[u(U_\infty - u)]dy + \frac{dU_\infty}{dx}\int_0^\delta (U_\infty - u)dy = \frac{\tau_w}{\rho}$$

$$\frac{d}{dx}\int_0^\delta [u(U_\infty - u)]dy + \frac{dU_\infty}{dx}\int_0^\delta (U_\infty - u)dy = \frac{\tau_w}{\rho}$$

Substituting Eq. (29.6) and (29.7) in Eq. (29.13) we obtain

$$\frac{d}{dx}\left[U_\infty^2 \delta^{**}\right] + \delta^{*} U_\infty \frac{dU_\infty}{dx} = \frac{\tau_w}{\rho}$$

where $\delta^{*} = \int_0^{\delta} \left(1 - \frac{u}{U_\infty}\right) dy$ is the displacement thickness

$\delta^{**} = \int_0^{\delta} \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right) dy$ is momentum thickness

Equation (29.14) is known as momentum integral equation for two dimensional incompressible laminar boundary layer. The same remains valid for turbulent boundary layers as well.

Needless to say, **the wall shear stress** (τ_w) **will be different for laminar and turbulent flows.**

The term $U_\infty \frac{dU_\infty}{dx}$ signifies space-wise acceleration of the free stream. Existence of this term means that free stream pressure gradient is present in the flow direction.

For example, we get finite value of $U_\infty \frac{dU_\infty}{dx}$ outside the boundary layer in the entrance region of a pipe or a channel. For external flows, the existence of $U_\infty \frac{dU_\infty}{dx}$ depends on the shape of the body.

During the flow over a flat plate, $U_\infty \frac{dU_\infty}{dx} = 0$ and the momentum integral equation is reduced to

$$\frac{d}{dx}\left[U_\infty^2 \delta^{**}\right] = \frac{\tau_w}{\rho}$$

Seperation of Boundary Layer

It has been observed that the **flow is reversed at the vicinity of the wall** under certain conditions
The phenomenon is termed as **separation of boundary layer**.
Separation takes place **due to excessive momentum loss near the wall in a boundary layer trying to move downstream against increasing pressure, i.e.,** $\frac{dp}{dx} > 0$ **, which is called *adverse pressure gradient*.**

Figure 29.2 shows the flow past a circular cylinder, in an infinite medium.

Up to $\theta = 90^\circ$, the flow area is like a constricted passage and the flow behaviour is like that of a nozzle.

Beyond $\theta = 90^\circ$ the flow area is diverged, therefore, the flow behaviour is much similar to a diffuser
This dictates the inviscid pressure distribution on the cylinder which is shown by a firm line in Fig. 29.2.

Here

p_∞ **: pressure in the free stream**

U_∞ **: velocity in the free stream and**

p ***:* is the local pressure on the cylinder.**

Consider the forces in the flow field. In the
inviscid region,
Until $\theta = 90^\circ$ the pressure force and the force due to streamwise acceleration i.e. inertia forces are acting in the same direction (**pressure gradient** being **negative/favourable**)

Beyond $\theta = 90^\circ$, the **pressure gradient is positive or adverse**. Due to the adverse pressure gradient the pressure force and the force due to acceleration will be opposing each other in the in viscid zone of this part.
So long as no viscous effect is considered, the situation does not cause any sensation. In the **viscid region** (near the solid boundary),

Up to $\theta = 90^\circ$ **, th**e viscous force opposes the combined pressure force and the force due to acceleration. Fluid particles overcome this viscous resistance **due to continuous conversion of pressure force into kinetic energy.**
Beyond $\theta = 90^\circ$, within the viscous zone, the flow structure becomes different. It is seen that the force due to acceleration is opposed by both the viscous force and pressure force.
Depending upon the magnitude of adverse pressure gradient, **somewhere around** $\theta = 90^\circ$ **, the fluid particles, in the boundary layer are separated from the wal**l and driven in the upstream direction. However, the far field external stream pushes back these separated layers together with it and develops **a broad pulsating wake behind the cylinder**.

The mathematical explanation of flow-separation : The point of separation may be defined as the limit between forward and reverse flow in the layer very close to the wall, i.e., at the point of separation

$$\left(\frac{\partial u}{\partial y}\right)_{y=0} = 0 \tag{29.16}$$

This means that the shear stress at the wall, $\tau_w = 0$. But at this point, the adverse pressure continues to exist and at the downstream of this point the flow acts in a reverse direction resulting in a back flow.

We can also explain flow separation using the argument about the second derivative of velocity u at the wall. From the dimensional form of the momentum at the wall, where $u = v = 0$, we can write

$$\left(\frac{\partial^2 u}{\partial y^2}\right)_{y=0} = \frac{1}{\mu}\frac{dp}{dx}$$

Consider the situation due to a **favourable pressure gradient** where $\frac{dp}{dx} < 0$ we have,

1. $\left(\partial^2 u / \partial y^2\right)_{wall} < 0$. (From Eq. (29.17))

As we proceed towards the free stream, the velocity *u* approaches U_∞ **asymptotically,** so $\partial u / \partial y$ decreases at a continuously lesser rate in *y*direction.This means that $\partial^2 u / \partial y^2$ remains less than zero near the edge of the boundary layer.The curvature of a velocity profile $\partial^2 u / \partial y^2$ is always negative as shown in (Fig. 29.3a)Consider the case of **adverse pressure gradient**, $\partial p / \partial x > 0$ At the boundary, the curvature of the profile must be positive (since $\partial p / \partial x > 0$).Near the interface of boundary layer and free stream the previous argumentregarding and $\partial^2 u / \partial y^2$ still holds good and the curvature is negative.Thus we observe that for an adverse pressure gradient, there must exist a pointfor which $\partial^2 u / \partial y^2 = 0$. This point is known as *point of inflection* of the velocity profile in the boundary layer as shown in Fig. 29.3bHowever, point of separation means $\partial u / \partial y = 0$ at the wall. $\partial^2 u / \partial y^2 > 0$ at the wall since separation can only occur due to adverse pressure gradient. But we have already seen that at the edge of the boundary

layer, $\partial^2 u / \partial y^2 < 0$. It is therefore, clear that **if there is a point of separation, there must exist a point of inflection in the velocity profile.**

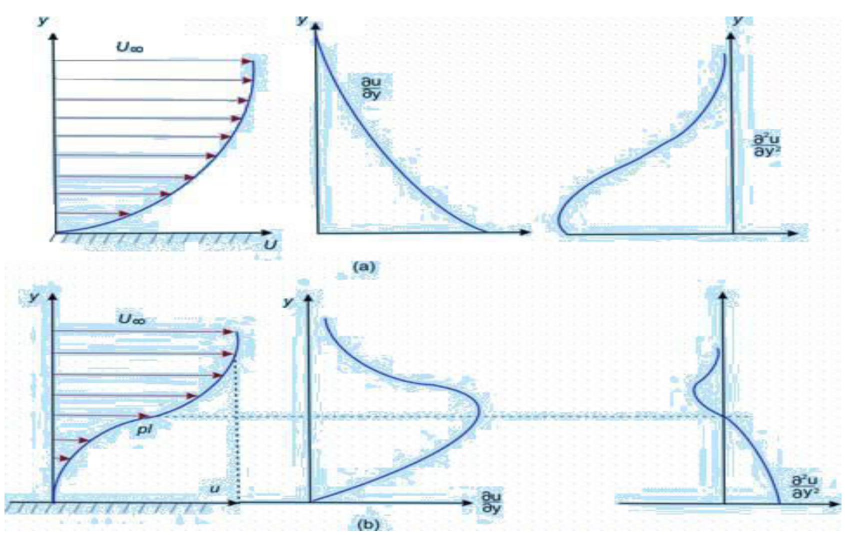


Fig. 29.3 Velocity distribution within a boundary layer

Favourable pressure gradient, adverse pressure gradient,

Let us reconsider the flow past a circular cylinder and continue our **discussion on the wake behind a cylinder.** The pressure distribution which was shown by the firm line in Fig. 21.5 is obtained from the potential flow theory. However. somewhere

near $\theta = 90^\circ$ **(in experiments it has been observed to be at** $\theta = 81^\circ$**) . the boundary layer detaches itself from the wall.**

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

Since the **wake zone pressure is less than that of the forward stagnation**

point (pressure at point A in Fig. 29.3), the cylinder experiences a drag force which is basically attributed to the pressure difference.

The drag force, brought about by the pressure difference is known as *form*

***drag* whereas the shear stress at the wall gives rise to *skin friction drag*.**Generally, these two drag forces together are responsible for resultant drag on a body

Karman-Pohlhausen Approximate Method For Solution Of Momentum Integral Equation Over A Flat Plate

The basic equation for this method is obtained by integrating the *x* direction momentum equation (boundary layer momentum equation) with respect

to *y* from the wall (at y = 0) to a distance $\delta(x)$ which is assumed to be outside the boundary layer. Using this notation, we can rewrite the Karman momentum integral equation as

$$U_\infty^2 \frac{d\delta^{**}}{dx} + \left(2\delta^{**} + \delta^*\right)U_\infty \frac{dU_\infty}{dx} = \frac{\tau_w}{\rho}$$

The effect of pressure gradient is described by the second term on the left hand side. For pressure gradient surfaces in external flow or for the developing sections in internal flow, this term contributes to the pressure gradient.

- We assume **a velocity profile which is a polynomial of** $\eta = y/\delta$. η being a

form of **similarity variable** , implies that **with the growth of boundary layer as distance *x* varies from the leading edge, the velocity**

profile (u/U_∞) **remains geometrically similar**.We choose a velocity profile in the form

$$\frac{u}{U_\infty} = a_0 + a_1\eta + a_2\eta^2 + a_3\eta^3 \tag{30.2}$$

In order to determine the constants $a_0, a_1, a_2 \ and \ a_3$ we shall prescribe the following boundary conditions

$$at \ \ y = 0, u = 0 \quad or \quad at \ \ \eta = 0, \frac{u}{U_\infty} = 0 \tag{30.3a}$$

$$at \ \ y = 0, \frac{\partial^2 u}{\partial y^2} = 0 \quad or \quad at \ \ \eta = 0, \frac{\partial^2}{\partial \eta^2}(u/U_\infty) = 0 \tag{30.3b}$$

$$at \ \ y = \delta, u = U_\infty \quad or \quad at \ \ \eta = 1, \frac{u}{U_\infty} = 1$$

at

$$at \ \ y = \delta, \frac{\partial u}{\partial y} = 0 \quad or \quad at \ \ \eta = 1, \frac{\partial(u/U_\infty)}{\partial \eta} = 0 \quad \text{30.3c)}$$

$$a_0 = 0, a_2 = 0, a_1 + a_3 = 1 \ \ and \ \ a_1 + 3a_3 = 0$$

Finally, we obtain the following values for the coefficients in Eq. (30.2),

$a_0 = 0, a_1 = 3/2, a_2 = 0 \ \ and \ \ +a_3 = -1/2$ and the velocity profile becomes

$$\frac{u}{U_\infty} = \frac{3}{2}\eta - \frac{1}{2}\eta^3 \qquad (30.4)$$

For flow over a flat plate, $\frac{dp}{dx} = 0, \quad hence \ U_\infty \frac{dU_\infty}{dx} = 0$ and the governing Eq. (30.1) reduces to

$$\frac{d\delta^{**}}{dx} = \frac{\tau_w}{\rho U_\infty^2} \qquad (30.5)$$

Again from Eq. (29.8), the momentum thickness is

$$\delta^{**} = \int_0^\infty \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right)dy \qquad or \ \ \delta^{**} = \int_0^\delta \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right)dy$$

$$or \ \ \delta^{**} = \delta\int_0^1 \left(\frac{3}{2}\eta - \frac{1}{2}\eta^3\right)\left(1 - \frac{3}{2}\eta + \frac{1}{2}\eta^3\right)d\eta$$

The wall shear stress is given by

$$\tau_w = \mu \left.\frac{\partial u}{\partial y}\right|_{y=0}$$

$$or \ \ \tau_w = \mu\left[\frac{\partial}{\delta\partial\eta}\left\{U_\infty\left(\frac{3}{2}\eta - \frac{1}{2}\eta^3\right)\right\}\right]_{\eta=0}$$

$$or \ \ \tau_w = \frac{3\mu U_\infty}{2\delta}$$

Substituting the values of δ^{**} and τ_w in Eq. (30.5) we get,

$$\frac{39}{280}\frac{d\delta}{dx} = \frac{3\mu U_\infty}{2\delta\rho U_\infty^2}$$

$$or \ \ \int \delta d\delta = \int \frac{140}{13}\frac{\mu}{\rho U_\infty}dx + C_1$$

0.6)

where C_1 is any arbitrary unknown constant.

$$or\ \frac{\delta^2}{2} = \frac{140}{13}\frac{\nu x}{U_\infty} + C_1$$

The condition at the leading edge ($at\ \ x = 0, \delta = 0$) yields $C_1 = 0$ Finally we obtain,

$$\delta^2 = \frac{280}{13}\frac{\nu x}{U_\infty}$$

$$\delta = \frac{5.83x}{\sqrt{\mathrm{Re}_x}} \tag{30.7}$$

$$or\ \delta = 4.64\sqrt{\frac{\nu x}{U_\infty}}$$

$$or\ \delta = \frac{4.64x}{\sqrt{\mathrm{Re}_x}}$$

This is the value of boundary layer thickness on a flat plate. Although, the method is an approximate one, the result is found to be reasonably accurate. The value is slightly lower than the exact solution of laminar flow over a flat plate . As such, **the accuracy depends on the order of the velocity profile.** We could have have used a fourth order polynomial instead --

$$\frac{u}{U_\infty} = a_0 + a_1\eta + a_2\eta^2 + a_3\eta^3 + a_4\eta^4 \tag{30.9}$$

In addition to the boundary conditions in Eq. (30.3), we shall require another boundary condition at

$$y = \delta, \frac{\partial^2 u}{\partial y^2} = 0\ \ or\ at\ \ \eta = 1, \frac{\partial^2 (u / U_\infty)}{\partial \eta^2} = 0$$

This yields the constants as $a_0 = 0, a_1 = 2, a_3 = -2\ \ and\ \ a_4 = 1$. Finally the velocity profile will be

Subsequently, for a fourth order profile the growth of boundary layer is given by

$$a = 2 + \frac{\lambda}{6},\ b = -\frac{\lambda}{2},\ c = -2 + \frac{\lambda}{2},\ d = 1 - \frac{\lambda}{6}$$

Integral Method For Non-Zero Pressure Gradient Flows

A wide variety of "integral methods" in this category have been discussed by Rosenhead . The Thwaites method is found to be a very elegant method, which is an extension of the method due to Holstein and Bohlen . We shall discuss the **Holstein-Bohlen method** in this section**.**

This is an **approximate method for solving boundary layer equations for two-dimensional generalized flow**. The integrated Eq. (29.14) for laminar flow with pressure gradient can be written as

$$\frac{d}{dx}\left[U^2 \delta^{**}\right] + \delta^* U \frac{dU}{dx} = \frac{\tau_\omega}{\rho}$$

(30.

$$U^2 \frac{d\delta^{**}}{dx} + \left(2\delta^{**} + \delta^*\right) U \frac{dU}{dx} = \frac{\tau_\omega}{\rho}$$

The **velocity profile** at the boundary layer is considered to be a **fourth-order polynomial** in terms of the dimensionless distance $\eta = y/\delta$, and is expressed as

$$u/U = a\eta + b\eta^2 + c\eta^3 + d\eta^4$$

The **boundary conditions are**

$$\eta = 0 : u = 0, v = 0 \qquad \frac{v}{\delta^2}\frac{\partial^2 u}{\partial \eta^2} = \frac{1}{\rho}\frac{dp}{dx} = -U\frac{dU}{dx}$$

$$\eta = 1 : u = U \qquad \frac{\partial u}{\partial \eta} = 0,\ \frac{\partial^2 u}{\partial \eta^2} = 0$$

A dimensionless quantity, known as **shape factor** is introduced as

$$\lambda = \frac{\delta^2}{v}\frac{dU}{dx}$$ (30.12)

The following relations are obtained

Now, the **velocity profile can be expressed as**

$$u/U = F(\eta) + \lambda G(\eta),$$

wher

$$F(\eta) = 2\eta + 2\eta^3 + \eta^4 \, , \; G(\eta) = \frac{1}{6}\eta(1-\eta)^3$$

The shear stress $\tau_{\omega} = \mu(\partial u / \partial y)_{y=0}$ is given by

$$\frac{\tau_{\omega}\delta}{\mu U} = 2 + \frac{\lambda}{6} \qquad (30.14)$$

We use the following dimensionless parameters,

$$L = \frac{\tau_{\omega}\delta^{**}}{\mu U} = \frac{\delta^{**}}{\delta}\left(2 + \frac{\lambda}{6}\right)$$

$$K = \frac{\left(\delta^{**}\right)^2}{\nu}\frac{dU}{dx} = \left(\frac{\delta^{**}}{\delta}\right)^2 \lambda \qquad (30.15)$$

$$H = \delta^{*} / \delta^{**}$$

The integrated momentum Eq. (30.10) reduces to

$$U\frac{d\delta^{**}}{dx} + \delta^{**}(2+H)\frac{dU}{dx} = \frac{\nu L}{\delta^{**}}$$

$$U\frac{d}{dx}\left[\frac{\left(\delta^{**}\right)^2}{\nu}\right] = 2\left[L - K(H+2)\right]$$

The parameter ***L* is related to the skin friction**
The parameter ***K* is linked to the pressure gradient.**

If we take ***K* as** the **independent variable** . ***L* and *H*** can be shown to be the **functions of *K*** since

$$\frac{\delta^{*}}{\delta} = \int_0^1 \left[1 - F(\eta) - \lambda G(\eta)\right] d\eta = \frac{3}{10} - \frac{\lambda}{120}$$

$$\frac{\delta^{*}}{\delta} = \int_0^1 \left(F(\eta) + \lambda G(\eta)\right)\left(1 - F(\eta) - \lambda G(\eta)\right) d\eta$$

$$K = \frac{[\delta^{**}]^2}{\delta^2}\lambda = \lambda\left(\frac{37}{315} - \frac{\lambda}{945} - \frac{\lambda^2}{9072}\right)^2$$

$$= \frac{37}{315} - \frac{\lambda}{945} - \frac{\lambda^2}{9072} \qquad (30.20)$$

Therefore,

$$L = \left(2 + \frac{\lambda}{6}\right)\frac{\delta^{**}}{\delta} = \left(2 + \frac{\lambda}{6}\right)\left(\frac{37}{315} - \frac{\lambda}{945} - \frac{\lambda^2}{9072}\right) = f_1(k)$$

$$H = \frac{\delta^*}{\delta^{**}} = \frac{(3/10) - (\lambda/120)}{(37/315) - (\lambda/945) - (\lambda^2/9072)} = f_2(k)$$

The right-hand side of Eq. (30.18) is thus a function of *K* alone. Walz pointed out that this function can be approximated with a good degree of accuracy by a linear function of *K* so that

$$2[L - K(H-2)] = a - bK \quad [Walz\text{'s approximation}]$$

[Walz's approximation]

Equation (30.18) can now be written as

$$\frac{d}{dx}\left(\frac{U[\delta^{**}]^2}{\nu}\right) = a - (b-1)\frac{U[\delta^{**}]^2}{\nu}\frac{1}{U}\frac{dU}{dx}$$

Solution of this differential equation for the dependent variable $U[\delta^{**}]^2/\nu$ subject to the boundary condition $U = 0$ when $x = 0$, gives

$$\frac{U[\delta^{**}]^2}{\nu} = \frac{a}{U^{b-1}}\int_0^x U^{b-1}dx$$

With $a = 0.47$ and $b = 6$. the approximation is particularly close between the stagnation point and the point of maximum velocity.

Finally the value of the dependent variable is

$$\left[\delta^{**}\right]^2 = \frac{0.47\nu}{U^6}\int_0^x U^5\,dx \qquad (30.22)$$

By taking the limit of Eq. (30.22), according to L'Hopital's rule, it can be shown that

$$[\delta^{**}]^2\,|_{x=0} = 0.47\,\nu/6U'(0)$$

This corresponds to **K = 0.0783.**

Note that $|\delta^{**}|$ is not equal to zero at the stagnation point. If $\left([\delta^{**}]^2/\nu\right)$ is determined from Eq. (30.22), *K(x)* can be obtained from Eq. (30.16).

Table 30.1 gives the necessary parameters for obtaining results, such as velocity profile and shear stress τ_ω The approximate method can be applied successfully to a wide range of problems.

Table 30.1 Auxiliary functions after Holstein and Bohlen

	K		
12	0.0948	2.250	0.356
10	0.0919	2.260	0.351
8	0.0831	2.289	0.340
7.6	0.0807	2.297	0.337
7.2	0.0781	2.305	0.333
7.0	0.0767	2.309	0.331
6.6	0.0737	2.318	0.328
6.2	0.0706	2.328	0.324
5.0	0.0599	2.361	0.310
3.0	0.0385	2.427	0.283
1.0	0.0135	2.508	0.252
0	0	2.554	0.235

-1	-0.0140	2.604	0.217
-3	-0.0429	2.716	0.179
-5	-0.0720	2.847	0.140

$$\Rightarrow \frac{\mu U}{\delta}\left(2+\frac{\lambda}{6}\right)$$

$$2+\frac{\lambda}{6}=0$$

As mentioned earlier, *K* and λ are related to the pressure gradient and the shape factor.

Introduction of *K* and λ in the integral analysis enables extension of Karman-Pohlhausen method for solving flows over curved geometry. However, the**analysis is not valid for the geometries, where** $\lambda < -12$ **and** $\lambda > +12$

Point of Seperation

$$\tau_{\omega} = 0$$

For point of seperation

$$\lambda = -12$$

TURBULENT FLOW

Introduction

The turbulent motion is an **irregular** motion.
Turbulent fluid motion can be considered as an irregular condition of flow in which various quantities (such as velocity components and pressure) show a **random variationwith time and space** in such a way that the statistical average of those quantities can be quantitatively expressed.
It is postulated that the fluctuations inherently come from **disturbances** (such as roughness of a solid surface) and they may be either dampened out due to viscous damping or may grow by drawing energy from the free stream.
At a **Reynolds number less than the critical**, the kinetic energy of flow is not enough to sustain the random fluctuations against the viscous damping and in such cases**laminar flow** continues to exist.
At somewhat **higher Reynolds number** than the critical Reynolds number, the kinetic energy of flow supports the growth of fluctuations and **transition to turbulence**takes place.

Characteristics Of Turbulent Flow

The most important characteristic of turbulent motion is the fact that **velocity and pressure** at a point **fluctuate with time** in a random manner.

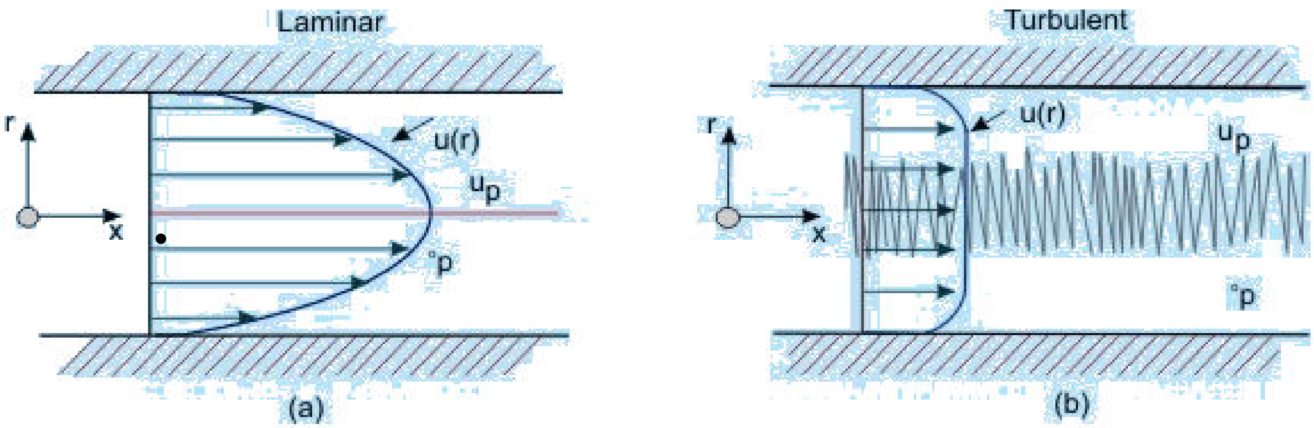


Fig. 32.1 Variation of horizontal components of velocity for laminar and turbulent flows at a point P

The mixing in turbulent flow is more due to these fluctuations. As a result we can see more uniform velocity distributions in turbulent pipe flows as compared to the laminar flows .

Fig. 32.2 Comparison of velocity profiles in a pipe for (a) laminar and (b) turbulent flows

Turbulence can be generated by -

frictional forces at the confining solid walls

the flow of layers of fluids with different velocities over one another

The turbulence generated in these two ways are considered to be different.

Turbulence generated and continuously affected by fixed walls is designated as **wall turbulence** , and turbulence generated by two adjacent layers of fluid in absence of walls is termed as **free turbulence** .

One of the effects of viscosity on turbulence is to make the flow more homogeneous and less dependent on direction.

Turbulence can be categorised as below -

Homogeneous Turbulence:

Turbulence has the same structure quantitatively in all parts of the flow field.

Isotropic Turbulence:

The statistical features have no directional preference and perfect disorder persists.

Anisotropic Turbulence:

The statistical features have directional preference and the mean velocity has a gradient.

Homogeneous Turbulence : The term homogeneous turbulence implies that the velocity fluctuations in the system are random but the average turbulent characteristics are independent of the position in the fluid, i.e., invariant to axis translation.
Consider the root mean square velocity fluctuations

$$u' = \sqrt{u^2}\ ,\quad v' = \sqrt{v^2}\ ,\quad w' = \sqrt{w^2}$$

In homogeneous turbulence, the rms values of *u', v'* and *w'* can all be different, but each value must be constant over the entire turbulent field. Note that even if the rms fluctuation of any component, say *u'* s are constant over the entire field the instantaneous values of u necessarily differ from point to point at any instant.

Isotropic Turbulence: The velocity fluctuations are independent of the axis of reference, i.e. invariant to axis rotation and reflection. Isotropic turbulence is by its definition always homogeneous . In such a situation, the gradient of the mean velocity does not exist, the mean velocity is either zero or constant throughout.
In isotropic turbulence fluctuations are independent of the direction of reference and

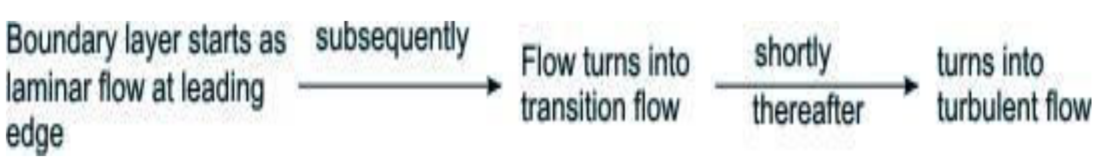
$$\sqrt{u^2} = \sqrt{v^2} = \sqrt{w^2} \quad \text{or} \quad u' = v' = w'$$

It is re-emphasised that even if the rms fluctuations at any point are same, their instantaneous values necessarily differ from each other at any instant.

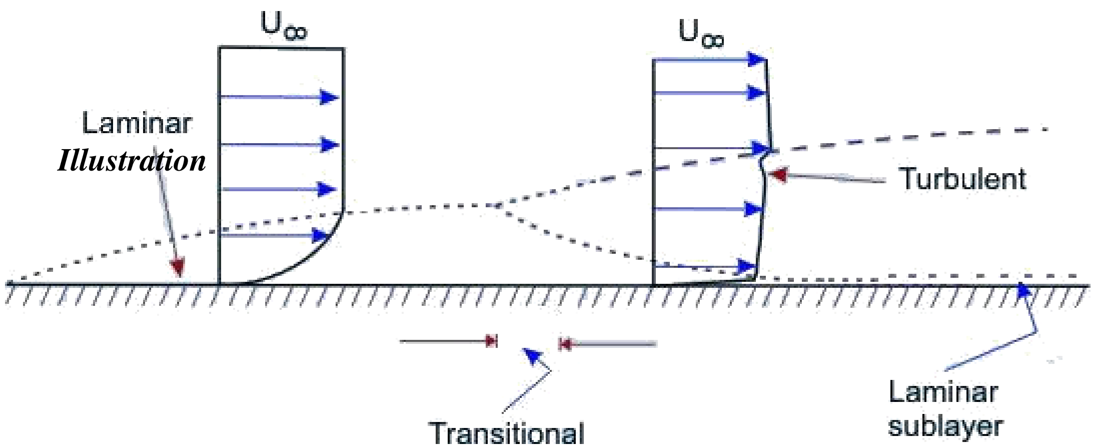
Turbulent flow is diffusive and dissipative . In general, turbulence brings about better mixing of a fluid and produces an additional diffusive effect. Such a diffusion is termed as "Eddy-diffusion ".(Note that this is different from molecular diffusion) At a large Reynolds number there exists a continuous transport of energy from the free stream to the large eddies. Then, from the large eddies smaller eddies are continuously formed. Near the wall smallest eddies destroy themselves in dissipating energy, i.e., converting kinetic energy of the eddies into intermolecular energy.

Laminar-Turbulent Transition

For a turbulent flow over a flat plate



The turbulent boundary layer continues to grow in thickness, with a small region below it called a **viscous sublayer**. In this sub layer, the flow is well behaved,just as the laminar boundary layer (Fig. 32.3 **Fig. 32.3 Laminar - turbulent transition**



(Fig. 32.3 **Fig. 32.3 Laminar - turbulent transition**

Observe that at a certain axial location, the laminar boundary layer tends to become unstable. Physically this means that the disturbances in the flow grow in amplitude at this location.Free stream turbulence, wall roughness and acoustic signals may be among the sources of such disturbances.

Transition to turbulent flow is thus initiated with the instability in laminar

flowThe possibility of instability in boundary layer was felt by **Prandtl** as early as 1912.The theoretical analysis of **Tollmien and Schlichting** showed that unstable waves could exist if the **Reynolds number was 575**.

The Reynolds number was defined as

$$\mathrm{Re} = U_\infty \delta^* / \nu$$

where U_∞ is the free stream velocity , δ^* is the displacement thickness and ν is the kinematic viscosity .

Taylor developed an alternate theory, which assumed that the transition is caused by a momentary separation at the boundary layer associated with the free stream turbulence. In a pipe flow the initiation of turbulence is usually observed at **Reynolds numbers (**

$U_\infty D / \nu$ **)in the range of 2000 to 2700**.

The development starts with a laminar profile, undergoes a transition, changes over to turbulent profile and then stays turbulent thereafter (Fig. 32.4). The length of development is of the order of 25 to 40 diameters of the pipe.

Fig. 32.4 Development of turbulent flow in a circular duct

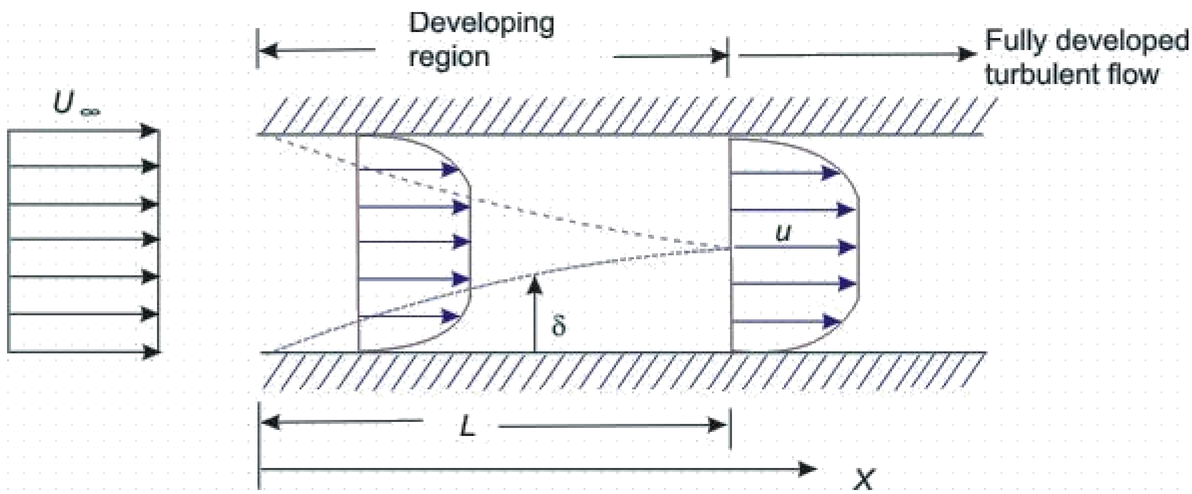
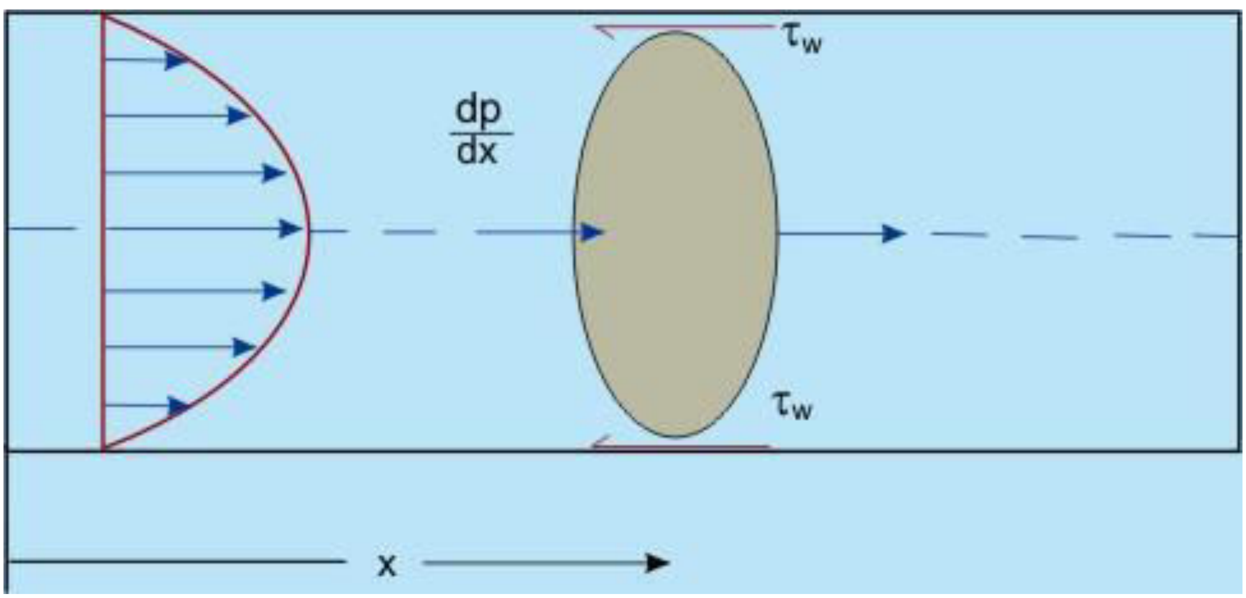


Fig. 32.4 Development of turbulent flow in a circular duct

Fully Developed Turbulent Flow In A Pipe For Moderate Reynolds Numbers



The entry length of a turbulent flow is much shorter than that of a laminar flow, J. Nikuradse determined that a fully developed profile for turbulent flow can be observed after an entry length of 25 to 40 diameters. We shall focus to fully developed turbulent flow in this section.

Considering a fully developed turbulent pipe flow (Fig. 34.3) we can write

$$2\pi R\tau_w = -\left(\frac{dp}{dx}\right)\pi R^2$$ (34.18)

$$\left(-\frac{dp}{dx} = \frac{2\tau_w}{R}\right)$$

Fig. 34.3 Fully developed turbulent pipe flow

It can be said that in a fully developed flow, the pressure gradient balances the wall shear stress only and has a constant value at any x. However, the friction factor (Darcy friction factor) is defined in a fully developed flow as

$$-\left(\frac{dp}{dx}\right) = \frac{\rho f U_{av}^2}{2D}$$

Comparing Eq.(34.19) with Eq.(34.20), we can write

$$\tau_w = \frac{f}{8}\rho U_{av}^2$$

H. Blasius conducted a critical survey of available experimental results and established the empirical correlation for the above equation as

$$f = 0.3164\,\mathrm{Re}^{-0.25} \text{ where } \mathrm{Re} = \rho U_{av} D/\mu$$ (34.22)

It is found that the Blasius's formula is valid in the range of Reynolds number of Re $\leq 10^5$. At the time when Blasius compiled the experimental data, results for higher Reynolds numbers were not available. However, later on, J. Nikuradse carried out experiments with the laws of friction in a very wide range of Reynoldsnumbers, $4 \times 10^3 \leq \mathrm{Re} \leq 3.2 \times 10^6$. The velocity profile in this range follows:

$$\frac{u}{\bar{u}} = \left[\frac{y}{R}\right]^{1/n}$$ (34.23)

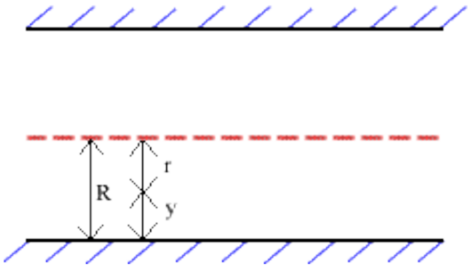
where $\bar{u}$ is the time mean velocity at the pipe centre and y is the distance from the wall .
The exponent n varies slightly with Reynolds number. In the range of Re ~ 10^5, n is 7.

Fully Developed Turbulent Flow In A Pipe For Moderate Reynolds Numbers

The ratio of $\bar{u}$ and U_{av} for the aforesaid profile is found out by considering the volume flow rate Q as

$$Q = \pi R^2 U_{av} = \int_0^R 2\pi r u \, dr$$

$$r = R - y$$



$$\pi R^2 U_{av} = 2\pi \bar{u} \int_0^R (R - y)(y / R)^{1/n} (-dy)$$

From equation (34.23)

$$\pi R^2 U_{av} = 2\pi \bar{u} \left[\frac{n}{n+1} \left(R^{\frac{n-1}{n}} y^{\frac{n+1}{n}} \right) - \frac{n}{2n+1} \left(y^{\frac{2n+1}{n}} R^{-\frac{1}{n}} \right) \right]_0^R$$

$$\pi R^2 U_{av} = 2\pi \bar{u} \left[R^2 \frac{n}{n+1} - \frac{n}{2n+1} R^2 \right]$$

$$\pi R^2 U_{av} = 2\pi R^2 \bar{u}\left[\frac{n^2}{(n+1)(2n+1)}\right]$$

$$\frac{U_{av}}{\bar{u}} = \frac{2n^2}{(n+1)(2n+1)}$$

Now, for different values of *n* (for different Reynolds numbers) we shall obtain

different values of $U_{av}/\bar{u}$ from Eq.(34.24a). On substitution of Blasius resistance formula (34.22) in Eq.(34.21), the following expression for the shear stress at the wall can be obtained.

$$\tau_w = \frac{0.3164}{8}\mathrm{Re}^{-0.25}\rho U_{av}^2$$

putting $\mathrm{Re} = \rho U_{av} 2R/\mu$

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

and where

$$\tau_w = 0.03955\rho U_{av}^2\left(\frac{\nu}{2RU_{av}}\right)^{1/4}$$

$$\tau = 0.03325\rho U^{7/4}\left(\frac{\nu}{}\right)^{1/4}$$

$$\tau_w = 0.03325\rho\left(\frac{U_{av}}{\bar{u}}\right)^{7/4}(\bar{u})^{7/4}\left(\frac{\nu}{R}\right)^{1/4}$$

For n=7, $U_{av}/\bar{u}$ becomes equal to 0.8. substituting $U_{av}/\bar{u} = 0.8$ in the above equation, we get

$$\tau_w = 0.03325\rho(0.8)^{1/4}(\bar{u})^{7/4}(\nu/R)^{1/4}$$

Finally it produces

$$u_\tau^2 \rho = 0.0225 \rho (\bar{u})^{7/4} \left(\frac{\nu}{R}\right)^{1/4}$$

$$\tau_w = 0.0225 \rho (\bar{u})^{7/4} (\nu / R)^{1/4}$$

where u_τ is friction velocity. However, u_τ^2 may be spitted into $u_\tau^{1/4}$ and $u_\tau^{7/4}$ and we obtain

$$\left(\frac{\bar{u}}{u_\tau}\right)^{7/4} = 44.44 \left(\frac{u_\tau R}{\nu}\right)^{1/4}$$

$$\frac{\bar{u}}{u_\tau} = 8.74 \left(\frac{u_\tau R}{\nu}\right)^{1/7}$$

Now **we can assume that the above equation is not only valid at the pipe axis (y = R) but also at any distance** from the wall y and a **general form is proposed as**

$$\frac{\bar{u}}{u_\tau} = 8.74 \left(\frac{y u_\tau}{\nu}\right)^{1/7}$$

Concluding Remarks :

It can be said that (1/7)th power velocity distribution law (24.38b) can be derived from Blasius's resistance formula (34.22) .

Equation (34.24b) gives the shear stress relationship in pipe flow at a moderate

Reynolds number, i.e $\mathrm{Re} \leq 10^5$. Unlike very high Reynolds number flow, here laminar effect cannot be neglected and the laminar sub layer brings about remarkable influence on the outer zones.The friction factor for pipe flows, f , defined by Eq. (34.22) is valid for a specific range of Reynolds number and for a particular surface condition.

Concept of Friction Factor in a pipe flow:

The friction factor in the case of a pipe flow was already mentioned in lecture 26.
We will elaborate further on friction factor or friction coefficient in this section.
Skin friction coefficient for a fully developed flow through a closed duct is defined as

$$C_f = \frac{\tau_w}{(1/2)\rho V^2}$$

where, V is the average velocity of flow given by $V = Q/A$, Q and A are the volume flow rate through the duct and the cross-sectional area of the duct respectively.From a force balance of a typical fluid element (Fig. 35.1) in course of its flow through a duct of constant cross-sectional area, we can write

$$\tau_w = \frac{\Delta p^* A}{SL}$$

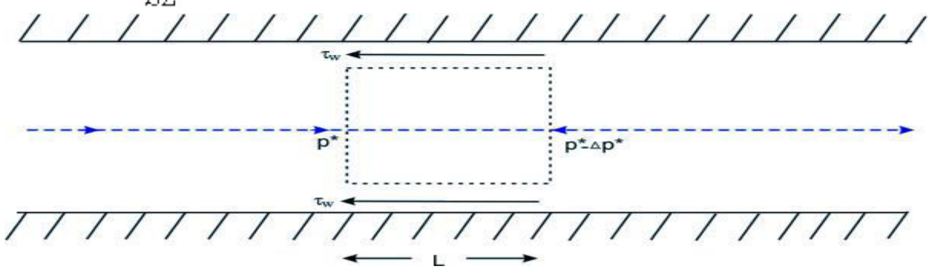


FIG 35.1 Force Balance of a fluid element in the course of flow through a duct

where, τ_w is the shear stress at the wall and Δp^* is the piezometric pressure drop over a length of **L** . **A** and **S** are respectively the cross-sectional area and wetted perimeter of the duct. Substituting the expression (35.2) in Eq. (35.1), we have,

$$C_f = \frac{\Delta p^* A}{SL(1/2)\rho V^2} = \frac{1}{4}\frac{D_h}{L}\frac{\Delta p^*}{(1/2)\rho V^2} \qquad (35.3)$$

where, $D_h = 4A/S$ and is known as the **hydraulic diameter** .

In case of a circular pipe, D_h=D, the diameter of the pipe. The coefficient C_f defined by Eqs (35.1) or (35.3) is known as **Fanning's friction factor** .

To do away with the factor 1/4 in the Eq. (35.3), Darcy defined a friction factor **f (Darcy's friction factor)** as

$$f = \frac{D_h}{L}\frac{\Delta p^*}{(1/2)\rho V^2}$$

$$f = \frac{64}{\mathrm{Re}}$$

(35.4)

Comparison of Eqs (35.3) and (35.4) gives $f = 4C_f$. Equation (35.4) can be written for a pipe flow as

$$f = \frac{D_h}{L} \frac{\Delta p^*}{(1/2)\rho V^2}$$

Equation (35.5) is written in a different fashion for its use in the solution of pipe flow problems in practice as

$$\Delta p^* = f \cdot \frac{L}{D_h} \cdot \frac{\rho}{2} V^2$$

or in terms of head loss (energy loss per unit weight)

$$h_f = \frac{\Delta p^*}{\rho g} = \frac{fLV^2}{2gD_h} \qquad (35.6b)$$

where, h_f represents the loss of head due to friction over the length L of the pipe.Equation (35.6b) is frequently used in practice to determine h_f

In order to evaluate h_f, we require to know the value of f. The value of f can be determined from Moody's Chart.

Variation of Friction Factor

In case of a laminar fully developed flow through pipes, the friction factor, *f* is found from the exact solution of the Navier-Stokes equation as discussed in lecture 26. It is given by f=64/R

In the case of a turbulent flow, friction factor depends on both the Reynolds number and the roughness of pipe surface.

Sir Thomas E. Stanton (1865-1931) first started conducting experiments on a number of pipes of various diameters and materials and with various fluids.Afterwards, a German engineer Nikuradse carried out experiments on flows through pipes in a very wide range of Reynolds number.

A comprehensive documentation of the experimental and theoretical investigations on the laws of friction in pipe flows has been presented in the form of a diagram, as shown in Fig. 35.2, by **L.F. Moody** to show the variation of friction factor, *f* with the pertinent governing parameters, namely, the Reynolds number of flow and the relativeroughness ε/D of the pipe. This diagram is known as **Moody's Chart** which is employed till today as the best means for predicting the values of *f* .

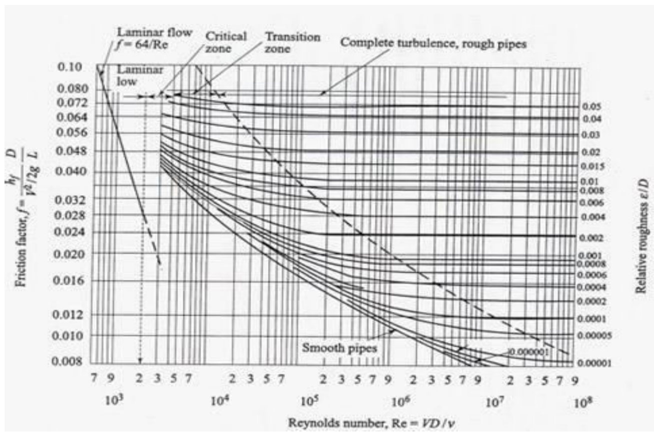


Fig. 35.2 Friction Factors for pipes (adapted from Trans. ASME, 66,672, 1944)

The friction factor *f* at a given Reynolds number, in the turbulent region, depends on the relative roughness, defined as the ratio of average roughness to the diameter of the pipe, rather than the absolute roughness.

For moderate degree of roughness, a pipe acts as a smooth pipe up to a value of Re where the curve of *f* vs *Re* for the pipe coincides with that of a smooth pipe. This zone is known as the **smooth zone of flow** .

The region where *f* vs Re curves (Fig. 35.2) become horizontal showing that **f** is independent of *Re*, is known as the **rough zone** and the intermediate region between the smooth and rough zone is known as the **transition zone**.

The position and extent of all these zones depend on the relative roughness of the pipe. In the smooth zone of flow, the laminar sublayer becomes thick, and hence, it covers appreciably the irregular surface protrusions. Therefore all the curves for smooth flow coincide.

With increasing Reynolds number, the thickness of sublayer decreases and hence the surface bumps protrude through it. The higher is the roughness of the pipe, the lower is the value of *Re* at which the curve of *f* vs *Re* branches off from smooth pipe curve (Fig. 35.2).

In the rough zone of flow, the flow resistance is mainly due to the form drag of those protrusions. The pressure drop in this region is approximately proportional to the square of the average velocity of flow. Thus *f* becomes independent of *Re* in this region.
In practice, there are three distinct classes of problems relating to flow through a single pipe line as follows:

The flow rate and pipe diameter are given. One has to determine the loss of head over a given length of pipe and the corresponding power required to maintain the flow over that length.

The loss of head over a given length of a pipe of known diameter is given. One has to find out the flow rate and the transmission of power accordingly.

The flow rate through a pipe and the corresponding loss of head over a part of its length are given. One has to find out the diameter of the pipe.
In the first category of problems, the friction factor f is found out explicitly from the given values of flow rate and pipe diameter. Therefore, the loss of head h_f and the power required, P can be calculated by the straightforward application of Eq.(35.6b).