SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE

www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – I – MECHANICS OF FLUID – SCIA1301

FLUID PROPERTIES AND FLUID STATICS

I. Properties of Fluids

Fluid Mechanics is that section of applied mechanics, concerned with the statics and dynamics of liquids and gases. Knowledge of fluid mechanics is essential for the chemical engineer, because the majority of chemical processing operations are conducted either partially or totally in the fluid phase. The handling of liquids is much simpler, much cheaper, and much less troublesome than handling solids. Even in many operations a solid is handled in a finely divided state so that it stays in suspension in a fluid.

Fluid Statics: treats fluids in the equilibrium state of no shear stress

Fluid Mechanics: treats when portions of fluid are in motion relative to other parts.

A fluid is defined as a substance that deforms continuously under the action of a shear stress, however small magnitude present. It means that a fluid deforms under very small shear stress, but a solid may not deform under that magnitude of the shear stress. It is a substance, as a liquid or gas, that is capable of flowing and that changes its shape at a steady rate when acted upon by a force tending to change its shape. The differences between the behaviours of solids and fluids under an applied force are as follows:

For a solid, the strain is a function of the applied stress, providing that the elastic limit is not exceeded. For a fluid, the rate of strain is proportional to the applied stress. The strain in a solid is independent of the time over which the force is applied and, if the elastic limit is not exceeded, the deformation disappears when the force is removed. A fluid continues to flow as long as the force is applied and will not recover its original form when the force is removed.

Newtonian fluids:

Fluids which obey the Newton's law of viscosity are called as Newtonian fluids. Newton's law of viscosity is given by

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

where = shear stress

μ = viscosity of fluid

du/dy = shear rate, rate of strain or velocity gradient

All gases and most liquids which have simpler molecular formula and low molecular weight such as water, benzene, ethyl alcohol, CCl_4, hexane and most solutions of simple molecules are Newtonian fluids.

Non-Newtonian fluids:

Fluids which do not obey the Newton's law of viscosity are called as non-Newtonian fluids.

Generally non-Newtonian fluids are complex mixtures: slurries, pastes, gels, polymer solutions etc.

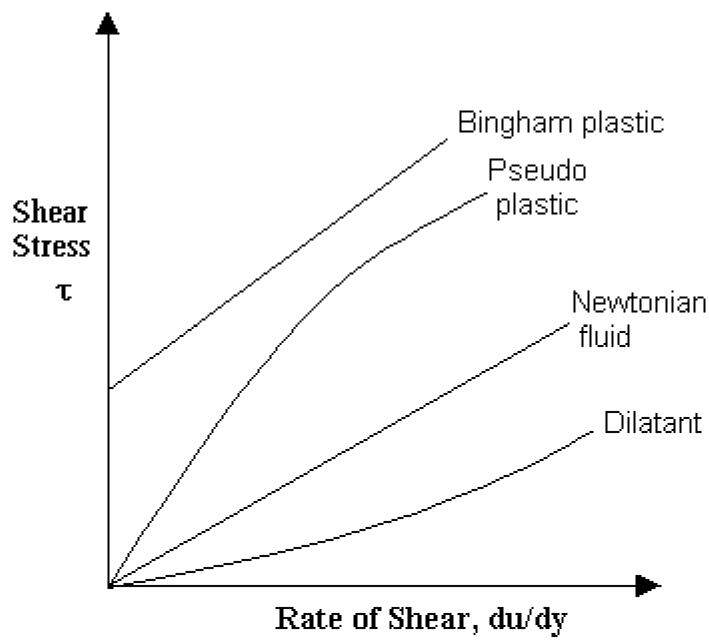


Fig.1: Newtonian Fluid

Density

The density of a substance is the quantity of matter contained in a unit volume of the substance. It can be expressed in three different ways.

Mass Density

Mass Density, ρ, is defined as the mass of substance per unit volume.

Units: Kilograms per cubic metre, kg/m^3 (or kgm^{-3})

Dimensions: ML^{-3}

Typical values:

Water = 1000 kgm^{-3}, Mercury = 13546 kgm^{-3} Air = 1.23 kgm^{-3}, Paraffin Oil = 800 kgm^{-3}.

(at pressure =1.013 $\times 10^{-5}$ Nm^{-2} and Temperature = 288.15 K.)

Specific Weight

Specific Weight ω, (sometimes, and sometimes known as *specific gravity*) is defined as the weight per unit volume.

or

The force exerted by gravity, g, upon a unit volume of the substance.

The Relationship between g and ω can be determined by Newton's 2nd Law, since

weight per unit volume = mass per unit volume g

$\omega = \rho g$

Units: Newton's per cubic metre, N/m^3 (or Nm^{-3})

Dimensions: $ML^{-2}T^{-2}$.

Typical values:

Water =9814 Nm^{-3}, Mercury = 132943 Nm^{-3}, Air =12.07 Nm^{-3}, Paraffin Oil =7851 Nm^{-3}

Relative Density

Relative Density, σ, is defined as the ratio of mass density of a substance to some standard mass density.

For solids and liquids this standard mass density is the maximum mass density for water (which occurs at 4°c) at atmospheric pressure.

$$\sigma = \frac{\rho_{substance}}{\rho_{H_2O(at 4^\circ c)}}$$

Units: None, since a ratio is a pure number.

Dimensions: 1.

Typical values: Water = 1, Mercury = 13.5, Paraffin Oil =0.8.

Viscosity

Viscosity, is the property of a fluid, due to cohesion and interaction between molecules, which offers resistance to sheer deformation. Different fluids deform at different rates under the same shear stress. Fluid with a high viscosity such as syrup, deforms more slowly than fluid with a low viscosity such as water.

Fig .2 Viscous fluid

All fluids are viscous, "Newtonian Fluids" obey the linear relationship

given by Newton's law of viscosity. $\tau = \mu \frac{du}{dy}$, which we saw earlier.

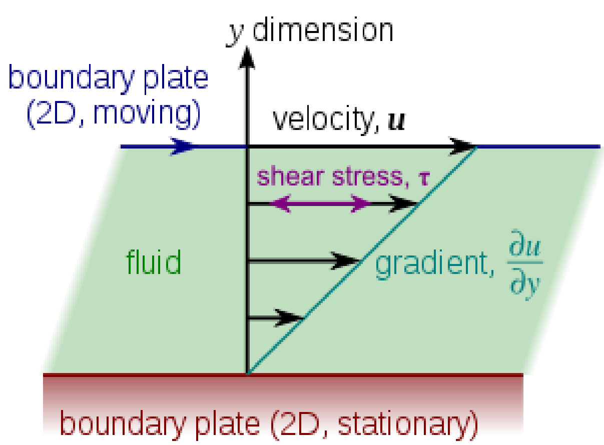


Fig .3 Variation in velocity

where τ is the shear stress,

Units $N m^{-2}$; $kg\ m^{-1} s^{-2}$

Dimensions $ML^{-1}T^{-2}$.

$\frac{du}{dy}$ is the velocity gradient or rate of shear strain, and has

Dimensions t^{-1}

- Resistance of liquid to flow. It is the measure of consistency of the fluid and inability to flow.
- Generally highly viscous fluid flows slowly and all the fluids are viscous in nature called real fluids.
- Ideal fluids have no viscosity

Coefficient of Dynamic Viscosity

The Coefficient of Dynamic Viscosity, , μ is defined as the shear force, per unit area, (or shear stress τ), required to drag one layer of fluid with unit velocity past another layer a unit distance away.

$$\mu = \tau \Big/ \frac{du}{dy} = \frac{\text{Force}}{\text{Area}} \Big/ \frac{\text{Velocity}}{\text{Distance}} = \frac{\text{Force} \times \text{Time}}{\text{Area}} = \frac{\text{Mass}}{\text{Length} \times \text{Area}}$$

Units: Newton seconds per square metre, $N s m^{-2}$ or Kilograms per meter per second, $kgm^{-1} s^{-1}$.

(Although note that μ is often expressed in Poise, P, where 10 P= 1 $kgm^{-1} s^{-1}$.)

Typical values:

Water =1.14 $\times 10^{-3}\ kgm^{-1} s^{-1}$, Air =1.78 $\times 10^{-5}\ kgm^{-1} s^{-1}$, Mercury =1.552 $kgm^{-1} s^{-1}$, Paraffin Oil =1.9 $kgm^{-1} s^{-1}$.

Kinematic Viscosity

Kinematic Viscosity, ν, is defined as the ratio of dynamic viscosity to mass density.

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

Units: square metres per second, m^2s^{-1}

(Although note that is often expressed in Stokes, St, where 10^4 St = 1 m^2s^{-1}.)

Dimensions: L^2T^{-1}.

Typical values:

Water =1.14 $\times 10^{-6}\ m^2s^{-1}$, Air =1.46 $\times 10^{-5}\ m^2s^{-1}$, Mercury =1.145 $\times 10^{-4}\ m^2s^{-1}$, Paraffin Oil =2.375 $\times 10^{-3}\ m^2s^{-1}$.

5.6m^3 of oil weighs 46 800 N. Find its mass density, ρ, and relative density, γ.

Weight 46 800 = mg

Mass m = 46 800 / 9.81 = 4770.6 kg

Mass density ρ = Mass / volume = 4770.6 / 5.6 = 852 kg/m^3

Relative density $\gamma = \dfrac{\rho}{\rho_{water}} = \dfrac{852}{1000} = 0.852$

The density of an oil is 850 kg/m^3. Find its relative density and Kinematic viscosity if the dynamic viscosity is 5 × 10^{-3} kg/ms.

ρ_{oil} = 850 kg/m^3 ρ_{water} = 1000 kg/m^3

γ_{oil} = 850 / 1000 = 0.85

Dynamic viscosity = μ = 5×10^{-3} kg/ms

Kinematic viscosity = ν = μ / ρ

$$\nu = \frac{\mu}{\rho} = \frac{5\times10^{-3}}{1000} = 5\times10^{-6}\ m^2s^{-1}$$

The velocity distribution of a viscous liquid (dynamic viscosity $\mu = 0.9$ Ns/m^2) flowing over a fixed plate is given by $u = 0.68y - y^2$ (u is velocity in m/s and y is the distance from the plate in m).
What are the shear stresses at the plate surface and at y=0.34m?

$$u = 0.68y - y^2$$
$$\frac{\partial u}{\partial y} = 0.68 - 2y$$

At the plate face y = 0m,

$$\frac{\partial u}{\partial y} = 0.68$$

Calculate the shear stress at the plate face

$$\tau = \mu \frac{\partial u}{\partial y} = 0.9 \times 0.68 = 0.612\ N/m^2$$

At y = 0.34m,

$$\frac{\partial u}{\partial y} = 0.68 - 2 \times 0.34 = 0.0$$

As the velocity gradient is zero at y=0.34 then the shear stress must also be zero.

Vapour Pressure

The pressure at which a liquid will boil is called its vapor pressure. This pressure is a function of temperature (vapor pressure increases with temperature). In this context we usually think about the temperature at which boiling occurs. For example, water boils at 100^{o}C at sea- level atmospheric pressure (1 atm abs). However, in terms of vapor pressure, we can say that by increasing the temperature of water at sea level to 100 ^{o}C, we increase the vapor pressure to the point at which it is equal to the atmospheric pressure (1 atm abs), so that boiling occurs.

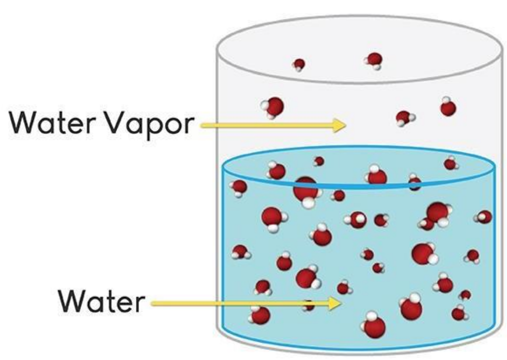


Fig .4: Vapour Pressure

It is easy to visualize that boiling can also occur in water at temperatures much below 100^{o}C if the pressure in the water is reduced to its vapor pressure. For example, the vapor pressure of water at 10^{o}C is 0.01 atm. Therefore, if the pressure within water at that temperature is reduced to that value, the water boils. Such boiling often occurs in flowing liquids, such as on the suction side of a pump. When such boiling does occur in the flowing liquids, vapor bubbles start growing in local regions of very low pressure and then collapse in regions of high downstream pressure. This phenomenon is called as ***cavitation***

Compressibility

All materials, whether solids, liquids or gases, are compressible, i.e. the volume V of a given mass will be reduced to V - dV when a force is exerted uniformly all over its surface. If the force per unit area of surface increases from p to p + dp, the relationship between change of pressure and change of volume depends on the bulk modulus of the material.

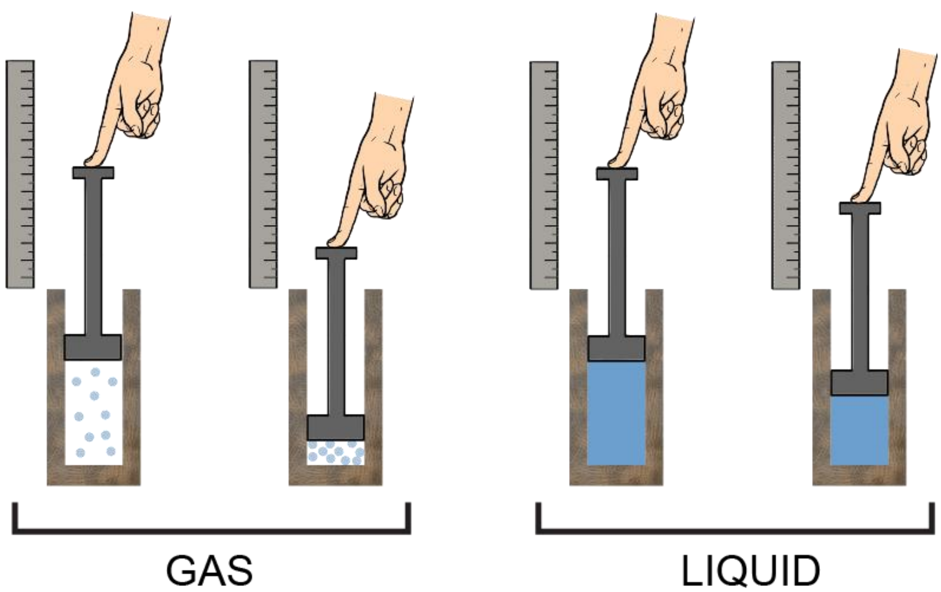


Fig .5: Compressibility

Bulk modulus (K) = (change in pressure) / (volumetric strain)

Volumetric strain is the change in volume divided by the original volume. Therefore,

(change in volume) / (original volume) = (change in pressure) / (bulk modulus)

i.e., $-dV/V = dp/K$

Negative sign for dV indicates the volume decreases as pressure increases. The concept of the bulk modulus is mainly applied to liquids, since for gases the compressibility is so great that the value of K is not a constant. The relationship between pressure and mass density is more conveniently found from the characteristic equation of gas.

For liquids, the changes in pressure occurring in many fluid mechanics problems are not sufficiently great to cause appreciable changes in density. It is therefore usual to ignore such changes and consider liquids as incompressible.

Gases may also be treated as incompressible if the pressure changes are very small, but usually compressibility cannot be ignored. In general, compressibility becomes important when the velocity of the fluid exceeds about one-fifth of the velocity of a pressure wave (velocity of sound) in the fluid.

Typical values of Bulk Modulus:

K = 2.05 x 10^9 N/m^2 for water

K = 1.62 x 10^9 N/m^2 for oil.

Surface Tension

It is a fluid property which occurs at the interfaces of a liquid and gas or at the interface of two immiscible liquids. As shown in Fig ,the liquid molecules- 'A' is under the action of molecular attraction between like molecules (cohesion). However the molecule ‘B' close to the interface is subject to molecular attractions between both like and unlike molecules (adhesion). As a result the cohesive forces cancel for liquid molecule 'A'. But at the interface of molecule 'B' the cohesive forces exceed the adhesive force of the gas. The corresponding net force acts on the interface; the interface is at a state of tension similar to a stretched elastic membrane.

As explained, the corresponding net force is referred to as surface tension, σ In short it is apparent tensile stresses which acts at the interface of two immiscible fluids.

Dimension: MT^{-2}

Unit: N/m

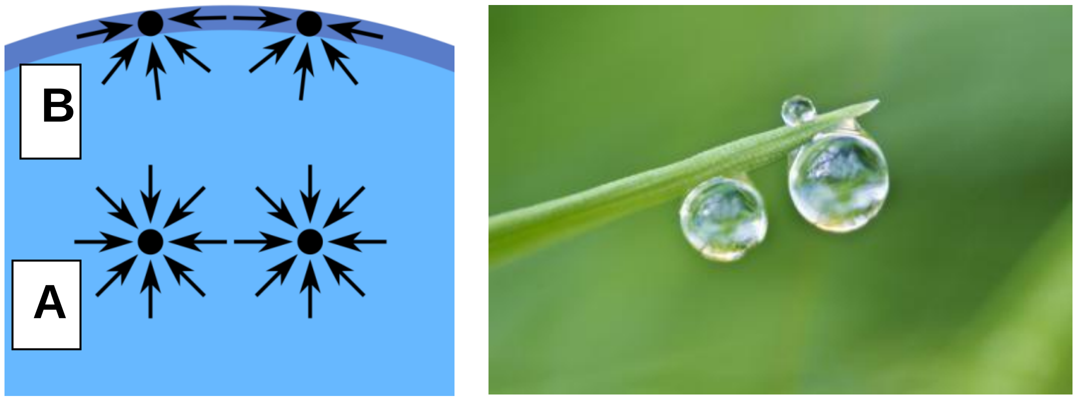


Fig .6: Surface Tension

Formation of water droplet- Phenomenon of surface tension

Capillarity

It is important (in fluid measurements) when using tubes smaller than about 10 mm in diameter. Capillary rise (or depression) in a tube can be calculated by making force balances. The forces acting are force due to surface tension and gravity.

The force due to surface tension,

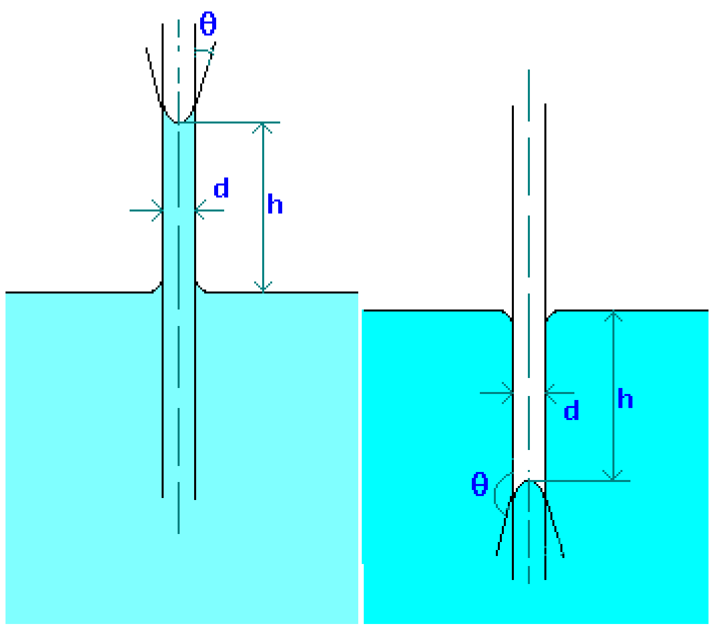


Fig .7: Capillarity

F_s = pdscos(q),

where q is the wetting angle or contact angle. If tube (made of glass) is clean q is zero for water and about 140^o for Mercury.

This is opposed by the gravity force on the column of fluid, which is equal to the height of the liquid which is above (or below) the free surface and which equals

$F_g = (p/4)d^2hgr$,

where r is the density of liquid.

Equating these forces and solving for Capillary rise (or depression) ‘h’

$h = 4s\cos(q)/(rgd)$

1. Water has a surface tension of 0.4 N/m. In a 3 mm diameter vertical tube if the liquid rises 6 mm above the liquid outside the tube, calculate the contact angle.

Data:

Surface tension (s) = 0.4 N/m

Dia of tube (d) = 3 mm = 0.003 m

Capillary rise (h) = 6 mm = 0.006 m

Formula:

Capillary rise due to surface tension is given by

$h = 4s\cos(q)/(rgd)$, where q is the contact angle.

Calculations:

$\cos(q) = hrgd/(4s) = 0.006 \times 1000 \times 9.812 \times 0.003 / (4 \times 0.4) = 0.11$

Therfore, contact angle q = $\mathbf{83.7^o}$

Pressure Measuring Instruments

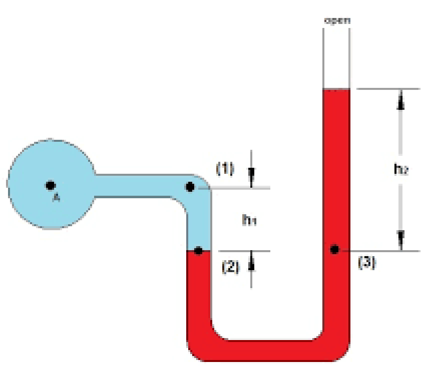
A somewhat more complicated device for measuring fluid pressure consists of a bent tube containing one or more liquid of different specific gravities. Such a device is known as *manometer*.

In using a manometer, generally a known pressure (which may be atmospheric) is applied to one end of the manometer tube and the unknown pressure to be determined is applied to the other end.

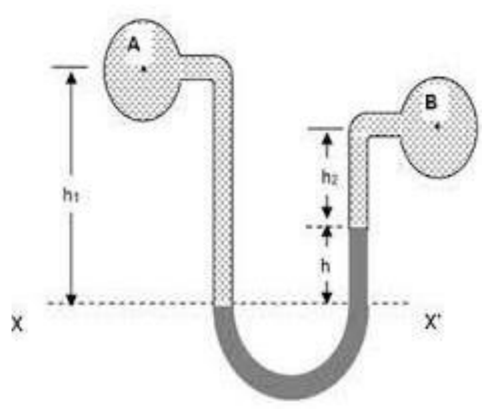
In some cases, however, the difference between pressure at ends of the manometer tube is desired rather than the actual pressure at the either end. A manometer to determine this differential pressure is known as *differential pressure manometer*.

Manometers - Various forms

1. Simple U - tube Manometer
2. Inverted U - tube Manometer
3. U - tube with one leg enlarged
4. Two fluid U - tube Manometer
5. Inclined U - tube Manometer

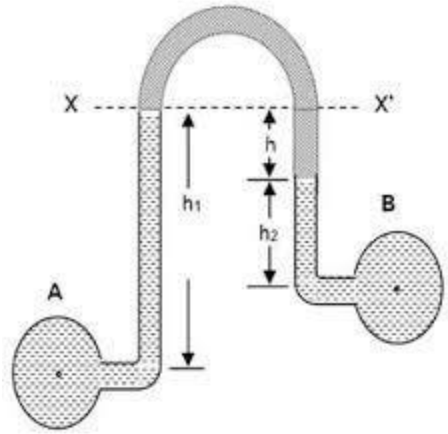


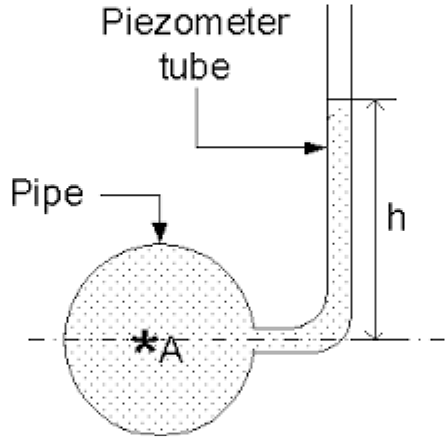
Simple U tube Manometer



Differential Manometer

Fig .8: Simple U tube Manometer





Inverted U tube Manometer

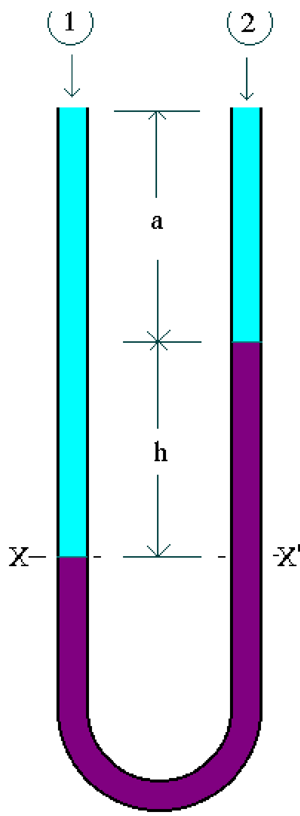


Fig .9: Simple U tube Manometer

For the left hand side:

$P_x = P_1 + \rho g(a+h)$

For the right hand side:

$P_{x'} = P_2 + \rho ga + \rho_m gh$

Since $P_x = P_{x'}$

$P_1 + \rho g(a+h) = P_2 + \rho ga + \rho_m gh$

$P_1 - P_2 = \rho_m gh - \rho gh$

i.e. $P_1 - P_2 = (\rho_m - \rho)\square\, gh.$

The maximum value of $P_1 - P_2$ is limited by the height of the manometer.

To measure larger pressure differences we can choose a manometer with higher density, and to measure smaller pressure differences with accuracy we can choose a manometer fluid which is having a density closer to the fluid density.

Inverted U-tube manometer

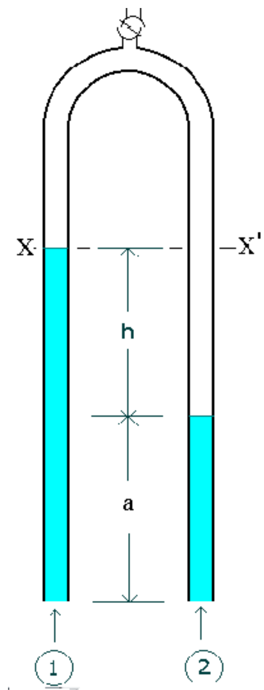


Fig .10: Inverted U tube Manometer

It is used for measuring pressure differences in liquids. The space above the liquid in the manometer is filled with air which can be admitted or expelled through the tap on the top, in order to adjust the level of the liquid in the manometer.

Equating the pressure at the level XX'(pressure at the same level in a continuous body of static fluid is equal), For the left hand side:

$P_x = P_1 - \rho g(h+a)$

For the right hand side:

$P_x' = P_2 - (\rho ga + \rho_m gh)$

Since $P_x = P_{x'}$

$P_1 - \rho g(h+a) = P_2 - (\rho ga + \rho_m gh)$

$P_1 - P_2 = (\rho - \rho_m)gh$

If the manometric fluid is choosen in such a way that $\rho_m << \rho$ then,

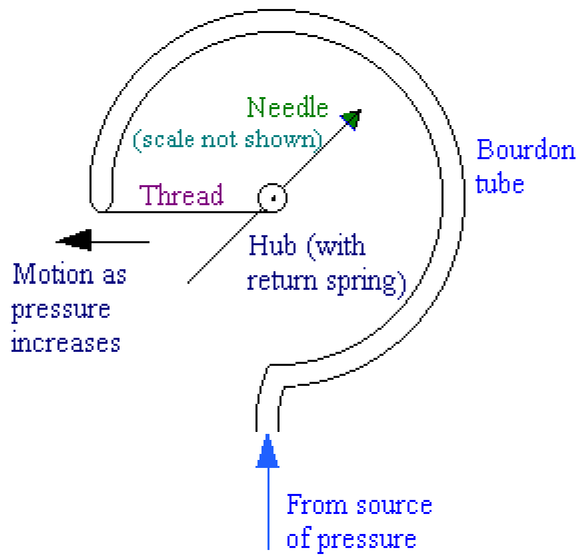
$P_1 - P_2 = \rho gh.$

For inverted U - tube manometer the manometric fluid is usually air.

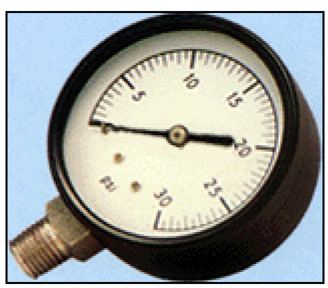
The manometer in its various forms is an extremely useful type of pressure measuring instrument, but suffers from a number of limitations.

- While it can be adapted to measure very small pressure differences, it can not be used conveniently for large pressure differences - although it is possible to connect a number of manometers in series and to use mercury as the manometric fluid to improve the range. (limitation)
- A manometer does not have to be calibrated against any standard; the pressure difference can be calculated from first principles. (Advantage)
- Some liquids are unsuitable for use because they do not form well-defined menisci. Surface tension can also cause errors due to capillary rise; this can be avoided if the diameters of the tubes are sufficiently large - preferably not less than 15 mm diameter. (limitation)
- A major disadvantage of the manometer is its slow response, which makes it unsuitable for measuring fluctuating pressures.(limitation)
- It is essential that the pipes connecting the manometer to the pipe or vessel containing the liquid under pressure should be filled with this liquid and there should be no air bubbles in the liquid.(important point to be kept in mind)

Bourdon Gauge:



The pressure to be measured is applied to a curved tube, oval in cross section. Pressure applied to the tube tends to cause the tube to straighten out, and the deflection of the end of the tube is communicated through a system of levers to a recording needle. This gauge is widely used for steam and compressed gases. The pressure indicated is the difference between that communicated by the system to the external (ambient) pressure, and is usually referred to as the gauge pressure.



The basic property of a static fluid is pressure. Pressure is familiar as a surface force exerted by a fluid against the walls of its container. Pressure also exists at every point within a volume of fluid. For a static fluid, as shown by the following analysis, pressure turns to be independent direction.

At what depth below' the surface of oil. rclatii c dcrisity‘ 0.8, will producc a F•«• urc of 120 IN/m ? fi'hat depth of water is this equivalent to?
[I 5.3m, 1 2.2m]

a)

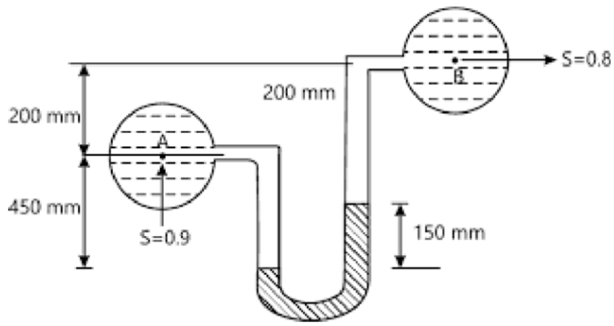
$$= 0.8 \times 1000\ kg/m^3$$

$$\frac{120\ IO'}{/\ g \quad 800 = 96\ I} = lSJ9m \text{ of oil}$$

b)

$$\rho = 1000\ kg/m^3$$

$$h = \frac{120 = 10^3}{1000 \times 9.81} = 12.23m \text{ of water}$$



Equating the pressure at the level XX' (pressure at the same level in a continuous body of fluid is equal),

For the left hand side:

$P_x = P_1 + \rho_1 g(0.45)$

For the right hand side:

$P_{x'} = P_2 + \rho 2\ g\ (0.5) + \rho_m g \times 0.15$

Since $P_x = P_{x'}$

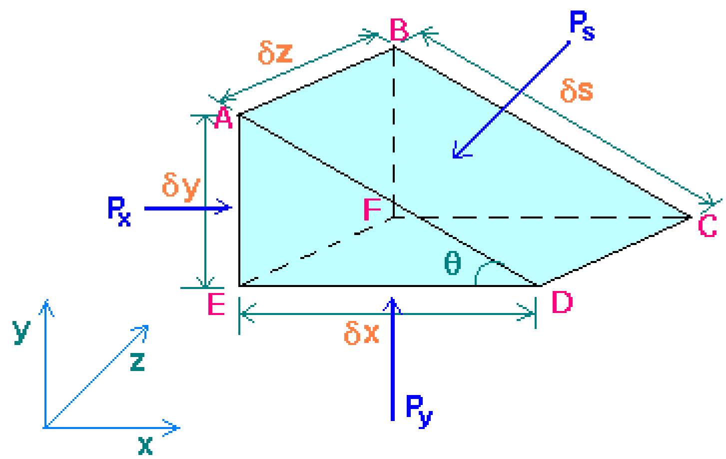
$P_1 + \rho_1 g(0.45) = P_2 + \rho 2\ g\ (0.5) + \rho_m g \times 0.15$

$P_1 - P_2 = \rho 2\ g\ (0.5) + \rho_m g \times 0.15 - \rho_1 g(0.45)$

$= (0.8 \times 1000 \times 9.81 \times 0.5) + (13.6 \times 1000 \times 9.81 \times 0.15) - (0.9 \times 1000 \times 9.81 \times 0.5)$

$= 19521\ N/m^2$

Pascal's Law



By considering the equilibrium of a small fluid element in the form of a triangular prism ABCDEF surrounding a point in the fluid, a relationship can be established between the pressures P_x in the x direction, P_y in the y direction, and Ps normal to any plane inclined at any angle q to the horizontal at this point.

P_x is acting at right angle to ABEF, and Py at right angle to CDEF, similarly P_s at right angle to ABCD.

Since there can be no shearing forces for a fluid at rest, and there will be no accelerating forces, the sum of the forces in any direction must therefore, be zero. The forces acting are due to the pressures on the surrounding and the gravity force.

Force due to $P_x = P_x$ x Area ABEF = $P_x dydz$

Horizontal component of force due to $P_s = -$ (P_s x Area ABCD) sin(q) = $- P_s dsdz\ dy/ds = - P_s dydz$

As P_y has no component in the x direction, the element will be in equilibrium, if

$P_x dydz + (-P_s dydz) = 0$

i.e. $P_x = P_s$

Similarly in the y direction, force due to $P_y = P_y dxdz$

Component of force due to $P_s = -$ (P_s x Area ABCD) cos(q) = $- P_s dsdz\ dx/ds = - P_s dxdz$

Force due to weight of element = - mg = - rVg = - r (dxdydz/2) g

Since dx, dy, and dz are very small quantities, dxdydz is negligible in comparison with other two vertical force terms, and the equation reduces to,

$P_y = P_s$

Therefore, $P_x = P_y = P_s$

i.e. pressure at a point is same in all directions. This is ***Pascal's law***. This applies to fluid at rest.

Fine powdery solids resemble fluids in many respects but differs considerably in others. For one thing, a static mass of particulate solids, can support shear stresses of considerable magnitude and the pressure is not the same in all directions.

In a stationary fluid the pressure is exerted equally in all directions and is referred to as the *static pressure*. In a moving fluid, the static pressure is exerted on any plane parallel to the direction of motion. The fluid pressure exerted on a plane right angles to the direction of flow is greater than the static pressure because the surface has, in addition, to exert sufficient force to bring the fluid to rest.

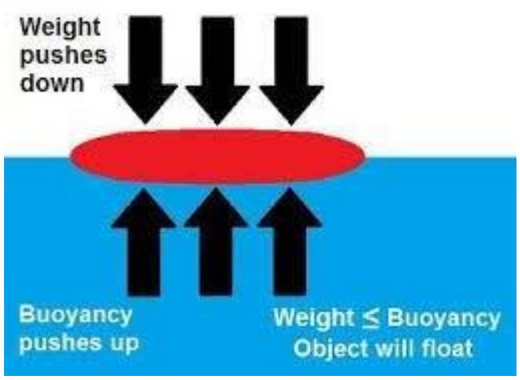
Buoyancy

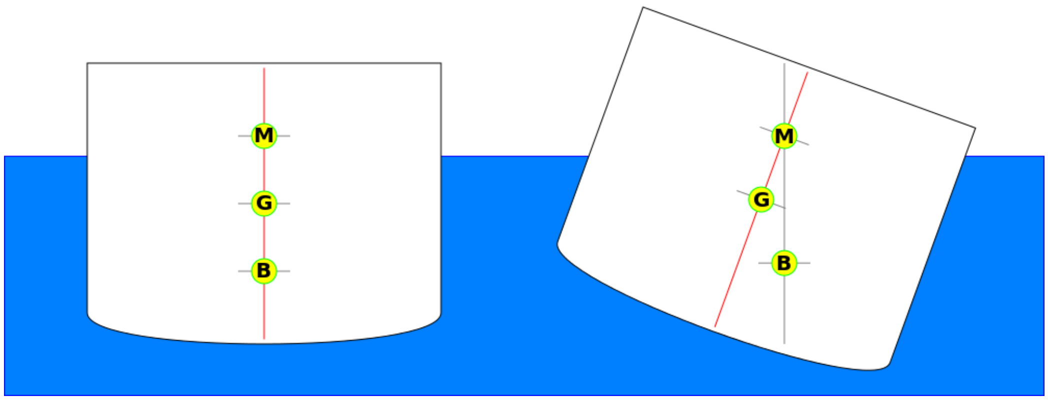
Upthrust on body = weight of fluid displaced by the body This is known as *Archimedes* principle.

Buoyancy we know that wooden objects float on water, but a small needle of iron sinks into water. This means that a fluid exerts an upward force on a body which is immersed fully or partially in it. The upward force that tends to lift the body is called the buoyant force, . The buoyant force acting on floating and submerged objects can be estimated by employing hydrostatic principle.

Center of Buoyancy

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



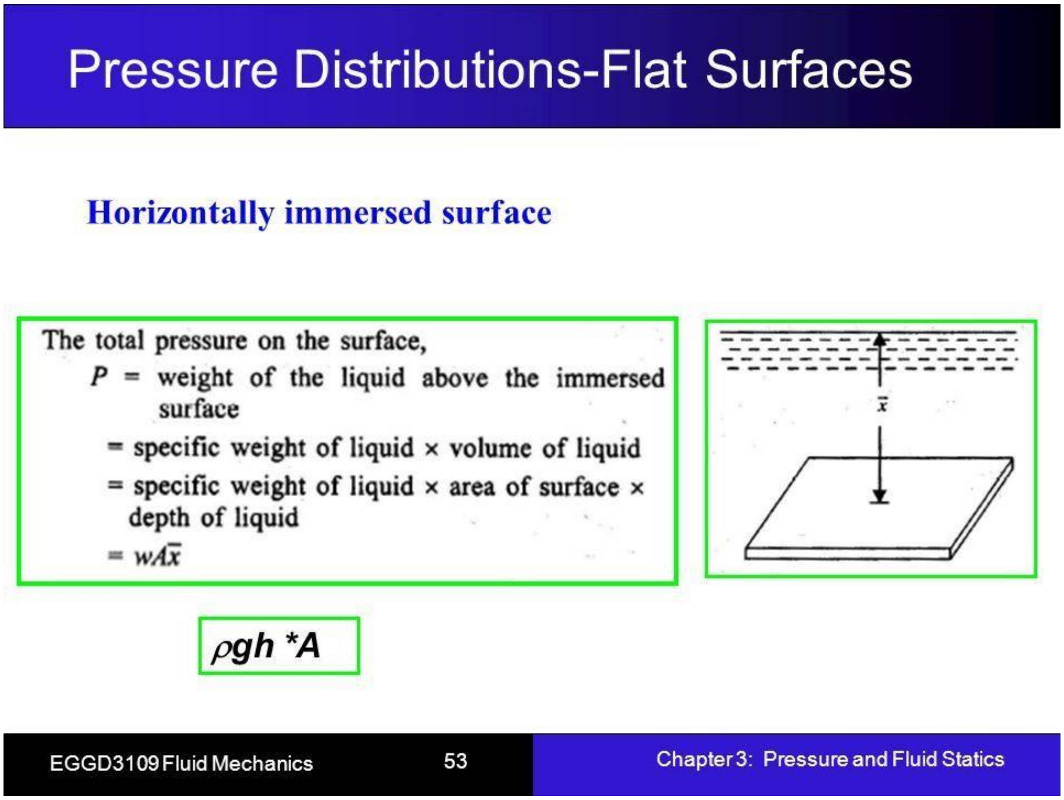


submerged and floating objects. This principle is known as the Archimedes principle which states A body immersed in a fluid experiences a vertical buoyant force which is equal to the weight of the fluid displaced by the body and the buoyant force acts upward through the centroid of the displaced volume"

Metacentric height (GM)

The **metacentric height** (GM) is a measurement of the initial static stability of a floating body. It is calculated as the distance between the centre of gravity of a ship and its metacentre. A larger metacentric height implies greater initial stability against overturning. The metacentric height also influences the natural period of rolling of a hull, with very large metacentric heights being associated with shorter periods of roll which are uncomfortable for passengers. Hence, a sufficiently high but not excessively high metacentric height is considered ideal for passenger ships.

The *centre of buoyancy* is at the centre of mass of the volume of water that the hull displaces. This point is referred to as B in naval architecture. The centre of gravity of the ship is commonly denoted as point G or VCG. When a ship is stable, the centre of buoyancy is vertically in line with the centre of gravity of the ship. It is defined as the point about which a body starts oscillating when the body is tilted by a small angle. The meta center may also defined as the point at which the line of action of force of buoyancy will meet the normal axis of the body is given a small angular displacement.



The **Total pressure** on an immersed surface, may be defined as the total pressure exerted by the liquid on it. Mathematically **total pressure**,

$$P = p_1 a_1 + p_2 a_2 + p_3 a_3 \ldots.$$

where,

- p_1, p_2, p_3 = Intensities of pressure on different strips of the surface, and
- a_1, a_2, a_3 = Areas of corresponding strips.

Total Pressure

When a static mass of the fluid comes in contact with a surface either plane or curved, a force exerted by the fluid on the surface. This force is known as the total pressure. Since the fluid at rest, no tangential force exists, hence the total pressure will act in the direction normal to the surface.

Centre of Pressure

The point of application of total pressure on the surface is known as centre of pressure.

The position of an immersed surface may be,

- Horizontal
- Vertical
- Inclined

Example - Total Pressure on a Horizontal Immersed Surface
A tank 3m $\times$ 4m contains 1.2m deep oil of specific gravity 0.8. Find

(a) intensity of the pressure at the base of the tank, and

(b) total pressure on the base of the tank.

Example

Given,

- Area of tank, A = 3m $\times$ 4m = 12m2
- Depth of oil, $\bar{x}$ = 1.2m
- Specific gravity of oil = 0.8

$\therefore$ Specific weight of oil, $w = 0.8 \times 9.24 = 7.85KN/m^3$

(a) Intensity of pressure at the base of the tank

$$= w\bar{x} = 7.85 \times 1.2 = 9.42KN/m^2 = 9.42KPa$$

(b) Total pressure on the base of the tank

$$P = wA\bar{x} = 9.42 \times 12 = 113.4KN$$

Intensity of pressure at the base of the tank = 9.42 KPa

Total pressure on the base of the tank = 113.4 KN

For horizontally submerged surfaces , depth of plane surface(h), distance of center of gravity from the top surface and depth of centre of pressure from top surface are same that is

h = Distance of C.G with top water surface = Distance of centre of pressure with top water surface

Here centre of pressure = 1.2m

Total Pressure On A Vertically Immersed Surface

Consider a plane vertical surface immersed in a liquid shown in figure

Let the whole immersed surface is divided into a number of small parallel stripes as shown in figure.

Here,

- ρg = Specific weight of the liquid

- A = Total area of the immersed surface

$\bar{x}$ or h = = Depth of the center of gravity of the immersed surface from the liquid surface
∴ Pressure on the strip = Intensity of pressure * Area = ρghdA

Now, Total pressure on the surface,

F = ρgh dA = ρg h dA

h dA = Area of surface x distance of C.G from free surface = A X h

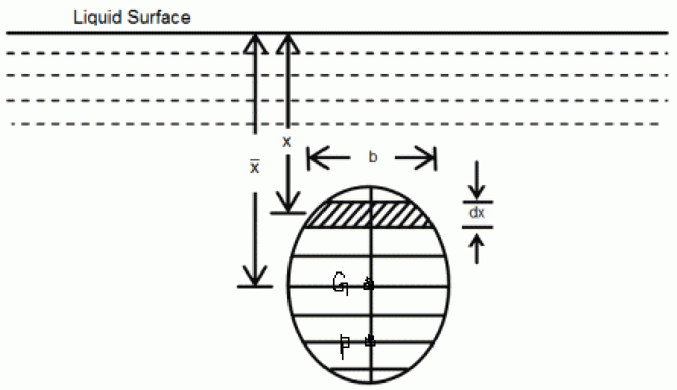


Fig 2 : Vertically immersed surface

F = ρgA h

Centre of pressure (h*)

According to the principle of moment theory, moment of the resultant force about an axis is equal to the sum of the moments of the components about the same axis. The resultant force F is acting at P, at a distance h* (center of pressure) from free surface of liquid.

Moment of force of F about the free surface of the liquid = Fx h*
Moment of force dF acting on the strip about free surface = Df xh = ώh.dAXh = ώh2dA

Sum of the moments of all such forces about the free surfaces

$$= \rho g h^2\, dA = \rho g\ h^2 dA$$

h^2dA = IO = Moment of Inertia of the surface about free surface of the liquid Sum

of the moments of all such forces about the free surfaces = ρg IO

Hence Fx h* = ρgIO

ρg A h X h* = ρg IO

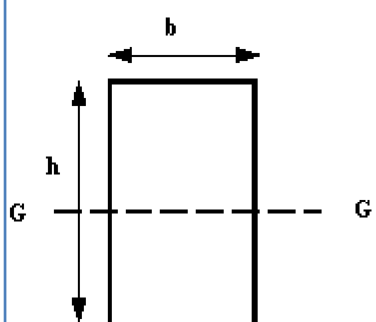
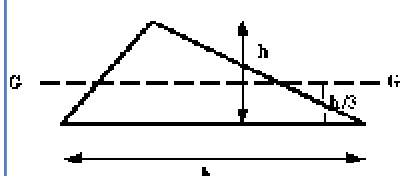
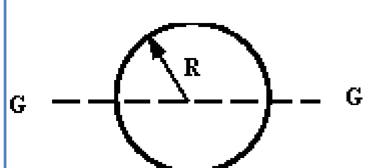
$$h^* = \frac{IO}{A\,h}$$

By the theorem of parallel axis,

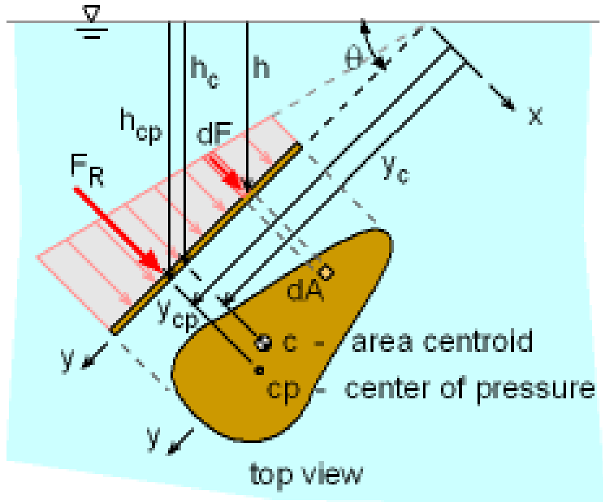
$$IO = IG + A\,h^2$$

$$h^* = \frac{IG + A\,h^2}{A\,h} = \frac{IG}{A\,h} + h$$

Table.1: Moment of Inertia

Shape	Area A	2nd moment of area, I_{GG}, about an axis through the centroid
Rectangle 	bd	$\frac{bd^3}{12}$
Triangle 	$\frac{bd}{2}$	$\frac{bd^3}{36}$
Circle 	πR^2	$\frac{\pi R^4}{4}$
Semicircle 	$\frac{\pi R^2}{2}$	$0.1102R^4$

Hydrostatic forces on inclined submerged surfaces



- ρg = Specific weight of the liquid
- A = Total area of the immersed surface

$\bar{x}$ or h = = Depth of the center of gravity of the immersed surface from the liquid surface

here $h = \frac{y}{\sin\theta}$

$$h = \frac{y}{\sin\theta}$$

$$h^* = \frac{y'}{\sin\theta}$$

Pressure force dF acting on the strip about AXIS O-O = dF = ρgy sin θ dA
Now, Total pressure on the surface,

F = ρgy sin θ dA = ρg sin θ y dA

y dA = Area of surface x distance of C.G from free surface = A X y

$$\sin\theta \;\; y = h$$

$$F = \rho g \sin\theta \; A \times y = \rho g \; A \times h$$

$$= \rho g A \, h$$

Centre of pressure

According to the principle of moment theory, moment of the resultant force about an axis is equal to the sum of the momentsof the componentsabout the same axis. The resultant force F is acting at P, at a distance h* (center of pressure) from free surface of liquid.

Moment of force of F about the free surface of the liquid = Fx y*

Pressure force dF acting on the strip about AXIS O-O = dF $= \rho g y \sin\theta \, dA$

Moment of force dF acting on the strip about AXIS O-O = dF xy $= \rho g y \sin\theta \, dA X y$

Sum of the moments of all such forces about the free surfaces

$$= \rho g y^2 \sin\theta \, dA = \rho g \sin\theta \, y^2 dA$$

$y^2 dA$ = IO = Moment of Inertia of the surface about free surface of the liquid

Sum of the moments of all such forces about the free surfaces = ρg IO

$$\text{Hence } F x\, y^* = \rho g \sin\theta \; IO$$

$$\rho g \sin\theta \; A\, y\, X\, y^* = \rho g \sin\theta \; IO$$

$$y^* = \frac{IO}{A\, y}$$

By the theorem of parallel axis,

$$IO = IG + A h^2$$

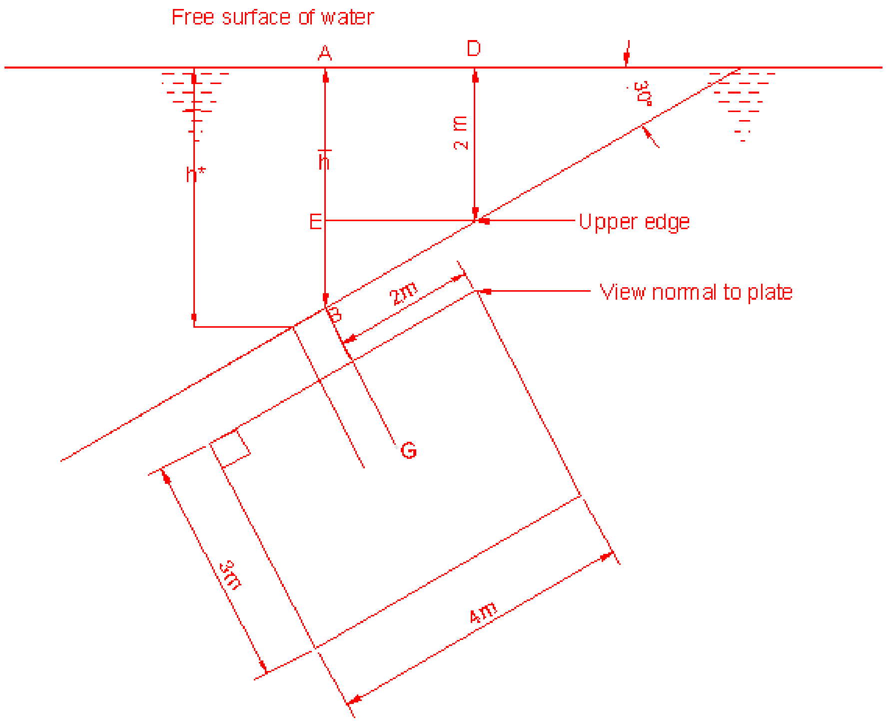
$$\frac{h*}{\sin\theta} = \frac{IG + A y^2}{A y} = \frac{IG}{A y} + y$$

$$\frac{h*}{\sin\theta} = \frac{IG}{A \frac{h}{\sin\theta}} + \frac{h}{\sin\theta}$$

$$\frac{h*}{\sin\theta} = \frac{IG \sin\theta}{A h} + \frac{h}{\sin\theta}$$

$$\text{Cp or } h^* = \frac{I_G \sin^2\theta}{A h} + h$$

A rectangular plane surface 3m wide and 4m deep lies in water in such a way that its plane makes an angle of 30 with the free surface of water. Determine the total pressure and position of center of pressure when the upper edge is 2m below the free water surface.



Total pressure F = F = ρgA h

= 1000x9.81x4x3x h h =

AE+EB

Centre of Pressure

EB = 2 sin30 h = 2+2 sin30

h = 3m

F = ρgA h = 1000x9.81x4x3x 3= 353.167 kN

$$h^* = \frac{I_G \sin^2\theta}{A\,h} + h$$

IG = bd3/12 = 3x43/12 = 16 mm4

$$h^* = \frac{IG\ \sin^2 30}{\ } + h = [(16X(1/4))/(12X3)] + 3 = 3.111m$$

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – II – MECHANICS OF FLUID – SCIA1301

UNIT II – FLUID KINEMATICS AND DYNAMIC

1.0 INTRODUCTION

- Kinematics means the study of motion. Fluid kinematics is the study of how fluids flow and how to describe fluid motion. Fluid kinematics deals with describing the motion of fluids without considering (or even understanding) the forces and moments that cause the motion. Branch of fluid mechanics which deals with response of fluids in motion without considering forces and energies in them. The study of kinematics is often referred to as the geometry of motion.
- Kinematics of fluid flow deals with the motion of fluid particles without considering the agency producing the motion. This deals with the geometry of motion of fluid particles. This also deals with the velocity and acceleration of fluid particles in motion. The motion of a fluid can be analysed on the same principles as those applied in the motion of a solid.
- There, however exists a basic difference between the motion of a solid and the motion of a fluid. A solid body is compact and moves as one mass. There is no relative motion between the particles of a solid body. Hence, we study the motion of the entire body and there is no necessity to study the motion of any particle of a solid body.

Methods of Describing Fluid Motion:

We know that each particle of a fluid in motion has at any instant a certain definite value of its properties like density, velocity, acceleration etc. As the fluid moves on, the values of these properties will change from one position to other positions, from time to time. Thus, it may be realized that two methods are possible to describe fluid motion. In the first method called the Lagrangian method, we study the velocity, acceleration etc. of an individual fluid particle at every instant of time as the particle moves to different positions.

This method of studying the properties of a single fluid particle is a very tedious process and therefore this method is not generally adopted. In the second method called the Eulerian method, we describe the flow by studying the velocity, acceleration, pressure, density etc. at a fixed point in space. Due to its easy application, this method is most commonly adopted.

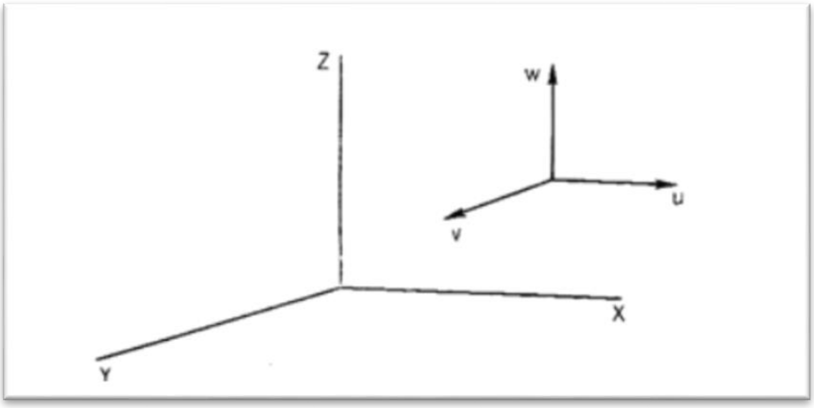


Fig 1: ordinates

Let x, y and z denote the space coordinates and t the time. Let V be the resultant velocity at any point in space in a fluid body. Let u, v and w be the components of the resultant velocity V at any point in the directions of the x, y and z axes. Fig. 6.1 illustrates the notations.

In the Eulerian method the velocity at a point (x,y,z) can be expressed as

$$u = f_1(x, y, z, t) \quad \text{...}(i)$$

$$v = f_2(x, y, z, t) \quad \text{...}(ii)$$

$$w = f_3(x, y, z, t) \quad \text{...}(iii)$$

$$V = f_4(x, y, z, t) \quad \text{...}(iv)$$

Relation (i) can be explained as follows. The velocity component u is some function of the space coordinates x,y,z and time t. Hence for certain values of x, y, z and t there is a corresponding value of u.

Stream Line:

A stream line is a continuous line in a fluid which shows the direction of the velocity of the fluid at each point along the line. The tangent to the stream line at any point on it is in the direction of the velocity at that point. Fluid particles lying on a stream line at an instant move along the stream line.

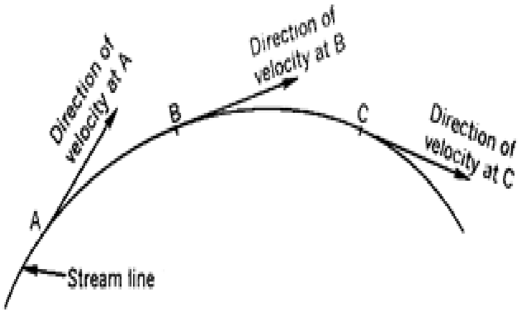


Fig. 6.2.

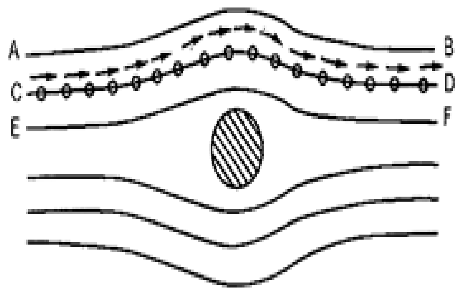


Fig. 6.3.

Fig. 6.2 shows a stream line *ABC*. The velocity at *A* is along the tangent to the stream line at *A*. The velocity at *B* is along the tangent to the stream line at *B*. Fig. 6.3 shows a number of stream lines *AB, CD, EF etc*. For the sake of clarity, the fluid particles on the stream line *CD* are also shown. These fluid particles move along the stream line *CD*.

Equation to a stream line

Suppose a particle moving along a stream line describes a small distance ds in a small interval of time dt. Let dx, dy, dz be the components of ds along three mutually perpendicular axes x, y and z. If V be the velocity of the particle, then the time taken by the particle to describe the distance ds is given by

$$dt = \frac{ds}{V}$$

If u, v and w are the components of the velocity V along the x, y and z axis, the above relation can also be given as

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

Thus, the differential equation of a stream line is given by.

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

When a fluid is in motion there are many stream lines and these stream lines indicate the flow pattern at that particular instant. For example, as a fluid flows round a cylindrical body, the stream line pattern will be as shown in Fig. 6.3. In steady flow the velocity at a point does not change in its magnitude and direction.

Hence, there is no change in the direction of the velocity vector at a point. In other words, the stream line is fixed in position. Conversely, if the stream line pattern remains constant the flow is steady. In the case of an unsteady flow, the direction of the velocity changes with time at every point. This means the position of a stream line is not constant. The position of a stream line changes from instant to instant.

Path Line

A path line means the path or a line actually described by a single fluid particle as it moves during a period of time. The path line indicates the direction of the velocity of the same fluid particle at successive instants of time.

In the case of a steady flow since there are no fluctuations of the velocity, the path line coincides with the stream line. In the case of an unsteady flow the stream lines change their positions at every instant and thus the path line may fluctuate between different stream lines during an interval of time.

Fig. 6.4. Path line.

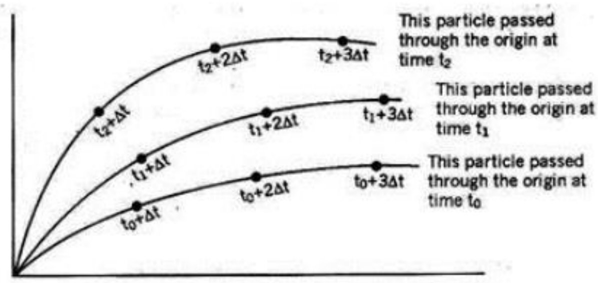


Fig 6.5.

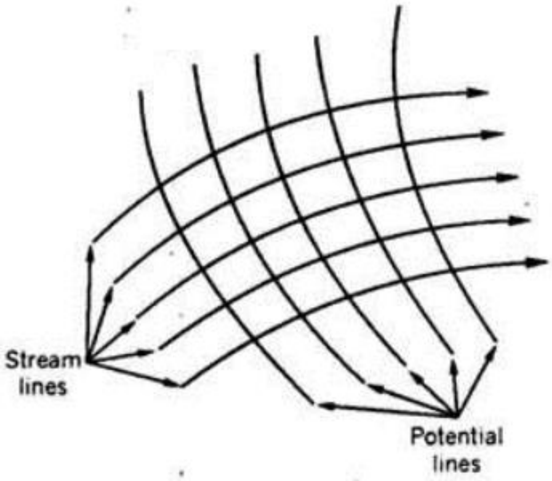
Fig. 6.4 shows the path line of a particular fluid particle. It is the locus of the positions of the same particle as it moves.Fig. 6.5 shows the path lines described by three particles which had passed through the origin at times t_0, t_1 and t_2. Consider the particle which passed through the origin at time to. Fig. 6.5 shows the positions taken by the particle at times $t_0 + \Delta t$, $t_0 + 2\Delta t$, $t_0 + 3\Delta t$ etc. as it has traced its path line.

Streak Line:

The streak line is the locus of the positions of fluid particles which have passed through a given point in succession. Suppose A, B, C, D… are fluid particles which passed through a reference point say the origin one after the other in succession. These particles have described their own path lines. Suppose at a time t, these particles A, B, C, D… are at P_a, P_b, P_c, P_d …. The line P_a, P_b, P_c, P_d …. is the streak line, at time t.

Potential Lines:

On a surface consisting of stream lines, we can imagine lines running orthogonally with the stream lines. Such line are called potential lines



A set of stream lines and potential lines constitutes a flow net.

Types of Flow:

We come across the following types of flow:

(i) Laminar flow and turbulent flow.

(ii) Steady flow and unsteady flow.

(iii) Uniform flow and non-uniform flow.

(iv) Rotational and irrotational flow.

(i) Laminar and Turbulent Flow:

Laminar Flow:

This is a type of flow in which the fluid particles move in layers, gilding smoothly over adjacent layers. There is no transportation of fluid particles from one layer to another. The fluid particles in any layer move along well defined stream lines.

The paths of the individual particles do not cross each other. This type of flow is also called stream line flow or viscous flow. This is a smooth flow of one layer of fluid over another. This type of flow occurs in viscous fluids where viscosity influences the flow.

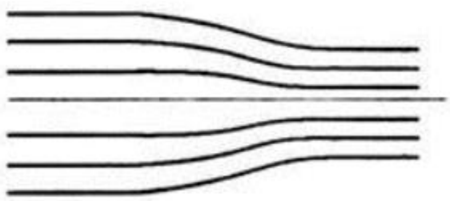


Fig. 6.9. Laminar flow.

Turbulent Flow:

This is the most common type of flow that occurs in nature. This flow is characterised by random, erratic, unpredictable motion of fluid particles which result in eddy currents. There is a general mixing up of fluid particles, in motion. The velocity changes in direction and magnitude from point to point.

There is a continuous collision between particles resulting in transference of momentum between them. The eddy currents cause a considerable loss of energy compared to the loss of energy in laminar flow. This greater loss of energy is due to the fact that turbulent shear stresses are very much greater than laminar shear stress given by Newton's law of viscosity.

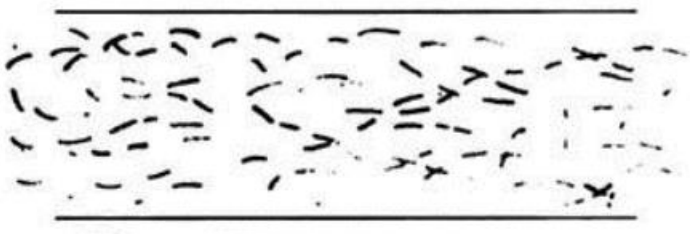


Fig. 6.10. Turbulent flow.

In a turbulent flow the distinguished characteristic of turbulence is its irregularity, indefinite frequency and no definite observable pattern. This type of flow cannot be truly mathematically analysed and any analysis is possible by statistical evaluation. Flow of water in rivers is generally turbulent. Flow of water in pipes at high velocity is turbulent. Flow of thick oil in narrow tubes, flow of ground water, flow of blood in blood vessels are laminar.

As the velocity of water in a pipe is gradually increased the flow will change from laminar to turbulent flow. The velocity at which the flow changes from laminar to turbulent flow in a pipe is called the critical velocity. The type of flow that exists in any case depends upon the value of a non-dimensional number dv/γ called the Reynolds's number, where d is the diameter of the pipe, v is the mean velocity of flow in the pipe and γ is the kinematic viscosity of the fluid. When the Reynolds number is less than 2000, the flow is generally laminar. When the Reynolds number is greater than 2800, the flow is generally turbulent. If the Reynolds number lies between the above limits the flow may be either laminar or turbulent. Thus the critical velocity has no fixed or definite value.

The velocity corresponding to Reynolds number equal to 2000, is called the lower critical velocity and the velocity corresponding to Reynolds number equal to 2800 is called the upper critical velocity.

(ii) Steady Flow and Unsteady Flow:

Steady Flow:

If the flow characteristics like, velocity, density, pressure etc. at a given point in a flowing mass of fluid does not change with the passage of time, the flow is said to be steady. On the contrary, if these flow characteristics at a given point change with respect to time, the flow is said to be unsteady flow.

Since velocity is a commonly adopted characteristic of flow it is quite sufficient to regard the flow to be steady if the velocity at a given point does not change with respect to time.

Suppose V is the velocity at a point (x_1, y_1, z_1). At this point if V remains constant at all times the flow is a steady flow. But, if at this point, V changes with time the flow is unsteady flow i.e.,

$$\left(\frac{\partial V}{\partial t}\right)_{(x_1, y_1, z_1)} = 0 \text{ for steady flow}$$

$$\left(\frac{\partial V}{\partial t}\right)_{(x_1, y_1, z_1)} \neq 0 \text{ for unsteady flow.}$$

In a steady flow the velocity at a point depends on the position of the point only, and is independent of the time *i.e.*

For steady flow, $V = f(x, y, z)$

and, $$\left(\frac{\partial v}{\partial t}\right)_{(x_1, y_1, z_1)} = 0$$

But, for unsteady flow $V = f(x, y, z, t)$

and $$\left(\frac{\partial v}{\partial t}\right)_{(x_1, y_1, z_1)} \neq 0$$

(iii) Uniform and Non-Uniform Flow:

If the flow characteristics like velocity, density, pressure etc. at a given instant remain the same at all points, the flow is uniform. If V is chosen as a flow characteristic, then, at a given instant V has the same value at all points and is independent of the space position. If the flow characteristics have different values at different points at a given instant of time, the flow is non-uniform flow.

For uniform flow, $$\left(\frac{\partial v}{\partial s}\right)_{t_1} = 0$$

For non-uniform flow, $$\left(\frac{\partial v}{\partial s}\right)_{t_1} \neq 0$$

***i.e.*, for uniform flow, at time $t = t_1$, V is independent of (x, y, z),**
but, for non-uniform flow, at time $t = t_1$, $V = f(x, y, z)$ if the flow is steady.

We use the terms uniform and non-uniform flow often in connection with open channels. In a channel where the section of the channel is uniform and the depth of flow is uniform the flow will be uniform as the velocity will be the same at all sections. But if the sectional dimensions of the channel are different at different sections, the depths of flow will be different at different sections.

Obviously, the velocity will be different at different sections and the flow will be non-uniform,

whether the flow is uniform or non-uniform if the rate of flow is constant the flow is steady

and if the rate of flow changes with time, the flow is unsteady. Thus, we may come across steady or unsteady or uniform or non-uniform flow. Any type of flow can exist independently of the other.

A combination of two types of flow is also possible.

Some combinations are:

a. Steady uniform flow.

b. Steady non-uniform flow.

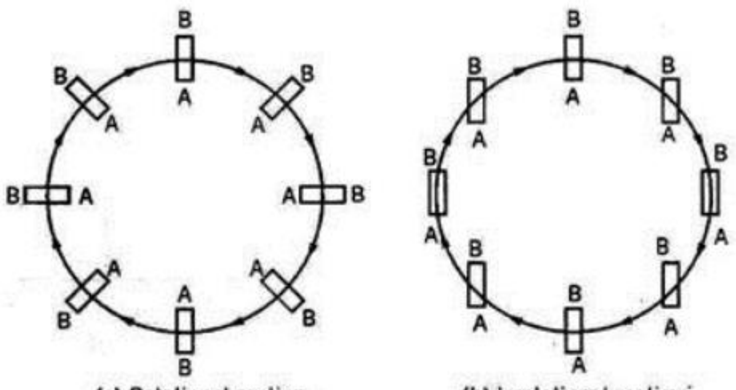
c. Unsteady uniform flow.

d. Unsteady non-uniform flow.

(iv) Rotational and Irrotational Flows:

As a fluid moves the fluid particles may be subjected to translatory or rotatory displacements. Suppose a particle which is moving along a stream line rotates about its own axis also then the particle is said to have a rotational motion. Whereas if the particle as it moves along the stream line does not rotate about its own axis the particle is said to have irrotational motion.

Fig. 6.11 (a) shows a rotational motion. Consider the fluid particle AB. As this particle moves along the stream line it rotates about its own axis also. Fig 6.11 (b) shows an irrotational motion. The fluid particle AB in this case, as it moves along the stream line, does not rotate about its own axis.



(a) Rotational motion (b) Irrotational motion

Fig. 6.11.

Various Types of Fluid Movements:

A fluid element may undergo four types of movements, namely,

(i) A pure translation,

(ii) A linear deformation,

(iii) A pure rotation,

(iv) An angular (shearing) deformation

One, Two and Three-Dimensional Flows:

This is another way of describing fluid motion. The velocity of a fluid element in the most general case is dependent upon its position. If any point in space be defined in terms of the space coordinates (x, y, z), then at any given instant the velocity at the point is given by $V = f(x, y, z)$. The flow in such a case is called a three-dimensional flow. Sometimes, the flow condition may be such that the velocity at any point depends only on two space coordinates say (x, y) at a given instant, i.e., in this case, at the given instant $V = f(x, y)$. In this case the flow conditions are potential in planes normal to the Z-axis. This type of flow is called a two-dimensional flow.

In a two-dimensional flow, the flow is identical in parallel planes. Fig. 6.16 shows a two-dimensional flow. In this figure is shown a channel whose walls are perpendicular to the plane of the diagram. Note the velocity vectors at sections 1-1 and 2-2. At section 1-1 the velocity varies across the channel. Similarly at section 2-2 the velocity varies across the channel. But the flow is identical in all planes parallel to the plane of the figure.

There is no component of the velocity perpendicular to the plane of the figure. In quite a number of cases it is usual to consider the motion as one-dimensional. This is no doubt a simplification over the two- dimensional and three-dimensional fluid motions. In this type of flow the velocity V at a given instant is a function of one space coordinate say x only i.e., at a given instant, $V = f(x)$

Fig. 6.17 shows a one-dimensional flow. At section 1-1 the velocity is constant over the entire section. Similarly at section 2-2 the velocity is constant over the entire section. A one-dimensional or two-dimensional or three-dimensional flow may be a steady flow or an unsteady flow.

	Steady	*Unsteady*
One-dimensional flow	$V=f(x)$	$V=f(x, t)$
Two-dimensional flow	$V=f(x, y)$	$V=f(x, y, t)$
Three-dimensional flow	$V=f(x, y, z)$	$V=f(x, y, z, t)$

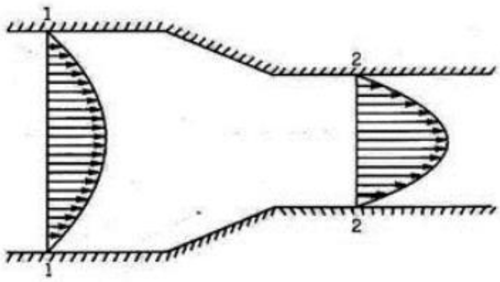


Fig. 6.16. Two-dimensional flow (vectors shown are velocity vectors.)

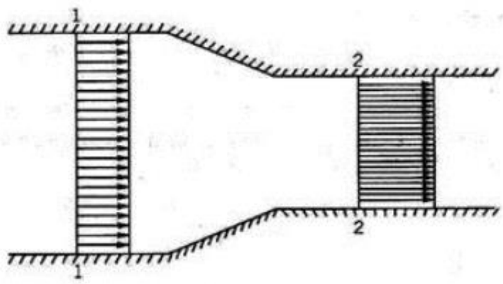


Fig. 6.17. One-dimensional flow (vectors shown are velocity vectors).

Control Volume:

This is a certain well defined extent of space. For the purpose of understanding the changes that take place in the fluid characteristics we may introduce a control volume so that we may compare the flow characteristics of a fluid just before it enters the control volume and just after it leaves the control volume.

Continuity Equation:

This is an equation based on the principle of conservation of mass. Suppose we consider a stream tube. Since the stream tube is always full of the fluid, the quantity of the fluid entering the stream tube at one end per unit of time should be equal to the quantity of the fluid leaving the stream tube at the other end per unit of time.

Let V be the average velocity at any section and A the area of the section. If w be the specific weight of the fluid, the quantity of the fluid flowing per second across the section

$$= wAV$$

If at sections 1-1 and 2-2 the specific weights of the fluid be w_1 and w_2 and if A_1 and A_2 be the sectional areas at the sections 1-1 and 2-2, and if V_1 and V_2 be the velocities at these sections, then

$$w_1 A_1 V_1 = w_2 A_2 V_2$$

If the fluid is incompressible $w_1 = w_2$ *and* the above relation reduces to

$$A_1 V_1 = A_2 V_2$$

i.e., volume of the fluid flowing across a section per unit of time is constant.

Continuity Equation in Three Dimensions:

Consider an infinitesimal parallelopiped of space in a fluid body. Let the sides of parallelopiped have length dx, dy and dz respectively. See Fig. 6.20. Let u, v and w be the inlet velocity components in the directions of the X, Y and Z axes. Mass of the fluid entering the left face = ρu dy dz. Mass of the fluid leaving the right face

Velocity Potential and Stream Functions:

Velocity potential function:

This is a function which is devised to expedite the analytical study of velocity fields. If ϕ is some function of the coordinates x and y in a two-dimensional flow, such that

$$\frac{\partial \phi}{\partial x} = -u$$

and $$\frac{\partial \phi}{\partial y} = -v$$

where u and v are the velocity components in the directions of the X and the Y axis, then the function ϕ is called a velocity potential function.

For example, suppose the velocity potential function for a flow is given by

$$\phi = 4x + 8y$$

Then $$\frac{\partial \phi}{\partial x} = -u = 4$$

∴ $$u = -4 \textit{ units/sec.}$$

and $$\frac{\partial \phi}{\partial y} = -v = 8$$

∴ $$v = -8 \textit{ units/sec}$$

We know, for a two-dimensional flow of an incompressible fluid, the condition of continuity of flow is

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

∴ $$-\frac{\partial^2 \phi}{\partial x^2} - \frac{\partial^2 \phi}{\partial y^2} = 0$$

∴ $$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0 \qquad \textit{(Laplace equation)}$$

Stream function. If ψ is some function of x and y such that

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

then the function ψ is called stream function.

Equipotential Line:

This is a line along which the velocity potential ϕ is constant *i.e.*, along this line $d\phi = 0$.

But $$d\phi = \frac{\partial \phi}{\partial x} dx + \frac{\partial \phi}{\partial y} dy = -[udx + vdy]$$

$\therefore$ For the equipotential line,

$$d\phi = 0$$

$\therefore$ $$u\,dx + v\,dy = 0$$

$\therefore$ $$\frac{dy}{dx} = -\frac{u}{v}$$ [*Slope of the equipotential line*]

PROBLEM

Solution We are to calculate the material acceleration for a given velocity field.

Analysis The velocity field is

$$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

The acceleration field components are obtained from its definition (the material acceleration) in Cartesian coordinates,

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 0 + (U_0 + bx)b + (-by)0 + 0$$
$$a_y = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 0 + (U_0 + bx)0 + (-by)(-b) + 0 \qquad (2)$$

where the unsteady terms are zero since this is a steady flow, and the terms with w are zero since the flow is two-dimensional. Eq. 2 simplifies to

Material acceleration components: $$\boxed{a_x = b(U_0 + bx) \qquad a_y = b^2 y} \qquad (3)$$

In terms of a vector,

Material acceleration vector: $$\boxed{\vec{a} = b(U_0 + bx)\vec{i} + b^2 y\vec{j}} \qquad (4)$$

Solution For a given velocity field we are to calculate the acceleration.

Assumptions **1** The flow is steady. **2** The flow is two-dimensional in the x-y plane.

Analysis The velocity components are

Velocity components: $u = 1.85 + 2.33x + 0.656y \qquad v = 0.754 - 2.18x - 2.33y$ (1)

The acceleration field components are obtained from its definition (the material acceleration) in Cartesian coordinates,

$$\begin{aligned} a_x &= \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 0 + \left(1.85 + 2.33x + 0.656y\right)\left(2.33\right) + \left(0.754 - 2.18x - 2.33y\right)\left(0.656\right) + 0 \\ a_y &= \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 0 + \left(1.85 + 2.33x + 0.656y\right)\left(-2.18\right) + \left(0.754 - 2.18x - 2.33y\right)\left(-2.33\right) + 0 \end{aligned} \qquad (2)$$

where the unsteady terms are zero since this is a steady flow, and the terms with w are zero since the flow is two-dimensional. Eq. 2 simplifies to

Acceleration components: $\boxed{a_x = 4.8051 + 3.9988x \qquad a_y = -5.7898 + 3.9988y}$ (3)

At the point $(x,y) = (-1,2)$, the acceleration components of Eq. 3 are

Acceleration components at (-1,2): $a_x = 0.80628 \cong \mathbf{0.806} \qquad a_y = 2.2078 \cong \mathbf{2.21}$

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – III – MECHANICS OF FLUID – SCIA1301

UNIT III -FLOW THROUGH PIPES

1.0 INTRODUCTION

Fluid flow is classified as external and internal, depending on whether the fluid is forced to flow over a surface or in a conduit. If the conduit is completely filled with the fluid, and flow isdriven primarily by a pressure difference whereas in open-channel flow where the conduit is partially filled by the fluid and thus the flow is partially bounded by solid surfaces, as in an irrigation ditch, and flow is driven by gravity alone.

PIPE FLOW VS OPEN CHANNEL FLOW

Pipe flow: Flows completely filling the pipe (a) .The pressure gradient along the pipe is main driving force. Open channel flow: Flows without completely filling the pipe (b). The gravity alone is the driving force.

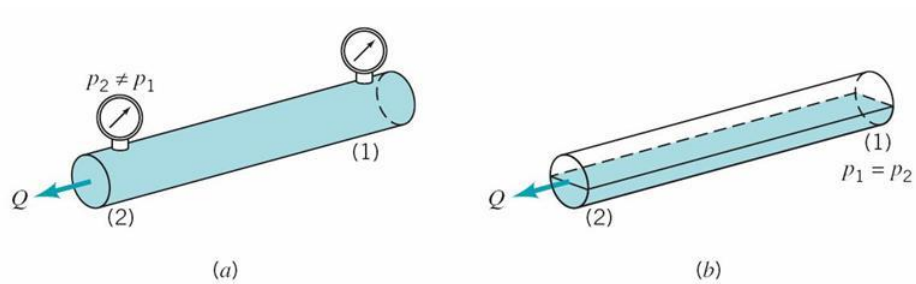


Fig 1. Pipe flow

PIPE SYSTEM

A pipe system include the pipes themselves (perhaps of more than one diameter), the various fittings, the flow rate control devices valves and the pumps or turbines.

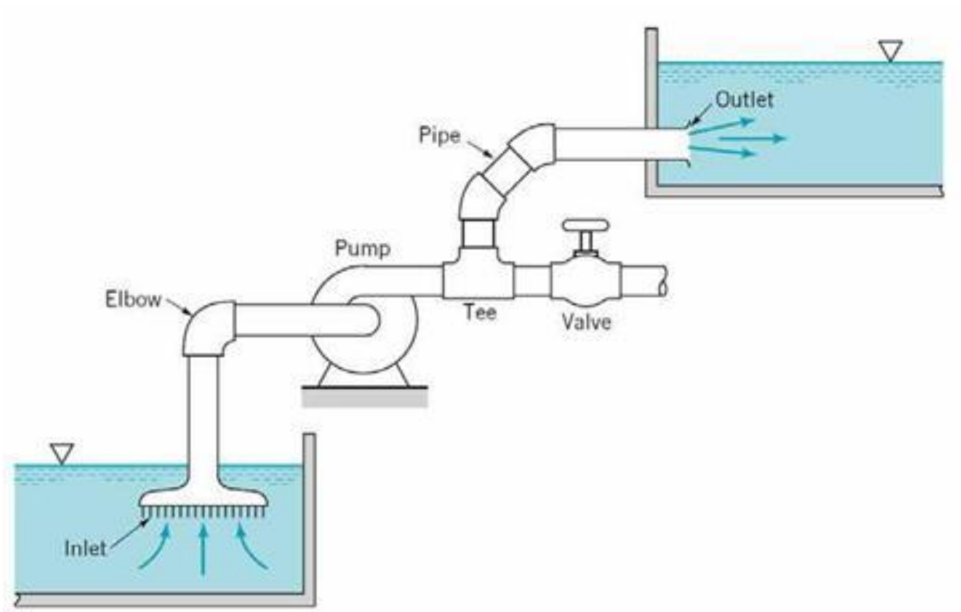


Fig 2. Pipe flow system

LAMINAR OR TURBULENT FLOW

Osborne Reynolds, a British scientist and mathematician,was the first to distinguish the difference between the classifications of flow by using a **simple apparatus** as shown in figure.

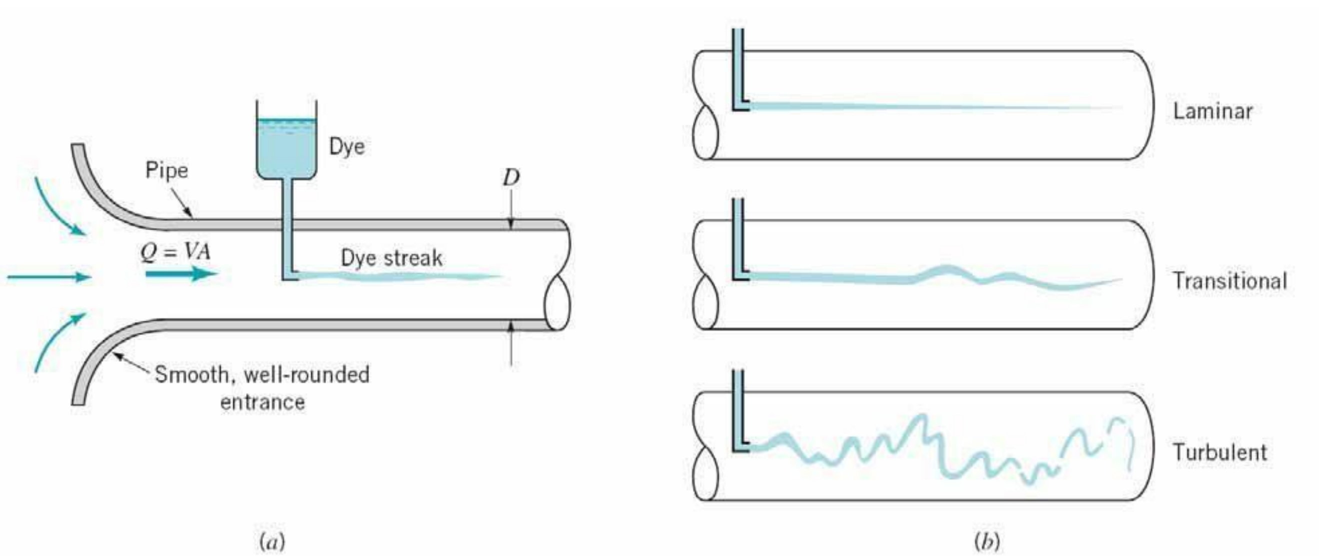


Fig 3. Pipe flow system

- For “small enough flow rate” the dye streak will remain as a well-defined line as it flows along, with only slight blurring due to molecular diffusion of the dye into the surrounding water.
- For a somewhat larger “intermediate flow rate”the dye fluctuates in time and space, and intermittent bursts of irregular behavior appear along the streak.
- For “large enough flow rate” the dye streakalmost immediately become blurred and spreads across the entire pipe in a random fashion.

LAMINAR FLOW

Fluid particles move along straight parallel path in layers. The path of individual fluid particles does not cross those of neighboring particles. If Reynolds Number is less than 2000, is termed as laminar flow.

TURBULENT FLOW:

The fluid particles move in random manner resulting in general mixing of the particles. If Reynolds Number is greater than 4000, is termed as turbulent flow. Reynolds number between 2000-4000 termed as transition flow.

Background to Pipe Flow Theory

To explain the various pipe flow theories we will follow the historical development of the subject:

Date	**Name**	**Contribution**
~1840	Hagen and Poiseuille	Laminar flow equation
1850	Darcy and Weisbach	Turbulent flow equation
1883	Reynolds	Distinction between laminar and turbulent flow
1913	Blasius	Friction factor equation or smooth pipes
1914	Stanton and Pannell	Experimental values of friction factor for smooth
1930	Nikuradse	Experimental values of friction factor for artificially rough pipes
1930s	Prandtl and von Karman	Equations for rough and smooth friction factors
1937	Colebrook and White	Experimental values of the friction factor for commercial pipes and the transition formula

1944	Moody	The Moody diagram for commercial pipes

LAMINAR FLOW

Steady Uniform Flow in a Pipe: Momentum Equation

The development that follows forms the basis of the flow theories applied to laminar flows. We remember from before that at the boundary of the pipe, the fluid velocity is zero, and the maximum velocity occurs at the centre of the pipe. This is because of the effect of viscosity. Therefore, at a given radius from the centre of the pipe the velocity is the same and so we consider an elemental annulus of fluid:

Consider a horizontal pipe of radius R.

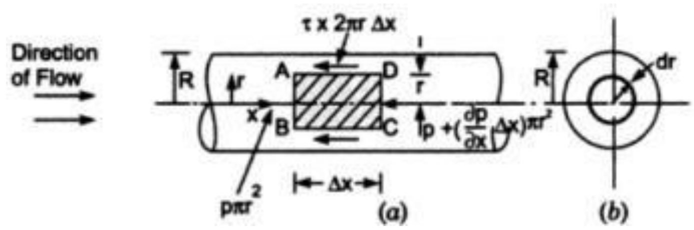


Fig 4. Pipe flow system

In the figure we have the following:

- Δr– thickness of the fluid element ;
- Δx– length of pipe considered;
- R – radius of pipe;

The forces acting on the elements are

- The pressure forces:

On face AB - $p\pi r^2$

On face CD – ($p+ \frac{\partial p}{\partial x} \Delta x$) x πr^2

- The shear force = τ x (2 πr x Δx)

The sum of the forces acting is equal to the change in momentum. However, the change in momentum is zero since the flow is steady and uniform. Thus:

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

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

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

$$\tau = -\frac{\partial p}{\partial x}\frac{r}{2} \qquad (1)$$

Thus the shear stress at any radius is known in terms of the piezometric pressure.

Velocity Distribution:

We can use the knowledge of the shear stress at any distance from the centre of the pipe in conjunction with our knowledge of viscosity as follows:

Shear stress, $\tau = \mu \frac{du}{dy}$

Y is measured from pipe wall

$$y = R - r$$

$$d_y = -\ d_r$$

$$\tau = -\ \mu \frac{du}{dr}$$

substitute the values in equation (1)

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

$$\frac{du}{dr} = \frac{1}{2\mu}\frac{\partial p}{\partial x} r$$

Integrating the above equation with r

$$u = \frac{1}{4\mu}\frac{\partial p}{\partial x} r^2 + c \qquad (2)$$

Value of C from boundary condition

when, r=R, u=0

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

Sub C in equation (2)

$$u = \frac{1}{4\mu}\frac{\partial p}{\partial x} r^2 - \frac{1}{4\mu}\frac{\partial p}{\partial x} R^2$$

$$u = -\frac{1}{4\mu}\frac{\partial p}{\partial x} [R^2 - r^2] \qquad (3)$$

Thus the velocity distribution is parabolic (i.e. a quadratic in *r*).

Ratio of Maximum velocity to average velocity:

From equation (3) the velocity is maximum when, r = 0. Sub r=0 in equation (3)

$$U_{max} = -\frac{1}{4\mu}\frac{\partial p}{\partial x} R^2 \qquad (4)$$

The average velocity ū, is obtained by dividing the discharge of fluid across the section by the area of the pipe (πr^2). The discharge Q across the secion is obtained by considering the flow through a circular ring element of radius r and thickness dr.

The fluid flowing per second through this elementary ring,

$$dQ = \text{velocity at radius r x Area of ring element}$$

$$dQ = \text{u x} 2\pi \text{r } dr \qquad (5)$$

Substitute value of u in equation (5)

$$dQ = -\frac{1}{4\mu}\frac{\partial p}{\partial x}[R^2 - r^2] \text{x } 2\pi\text{r } dr$$

$$Q = \quad dQ$$

$$= \quad -\frac{1}{4\mu}\frac{\partial p}{\partial x} \; [R^2 - r^2] \; \text{x } 2\pi\text{r } dr$$

$$= \quad -\frac{1}{4\mu}\frac{\partial p}{\partial x} \times 2\pi \quad [R^2 - r^2] r dr$$

$$= -\frac{1}{4\mu}\frac{\partial p}{\partial x} \times 2\pi \int_0^R (R^2 - r^3) \qquad dr$$

$$= -\frac{1}{4\mu}\frac{\partial p}{\partial x} \times 2\pi[\frac{R^2 r^2}{2} - \frac{r^4}{4}]$$

$$= -\frac{1}{4\mu}\frac{\partial p}{\partial x} \times 2\pi[\frac{2R^4 - R^4}{4}]$$

$$Q = \frac{\pi}{8\mu}(-\frac{\partial p}{\partial x})R^4 \qquad (6)$$

Average Velocity $\qquad \bar{u} = \frac{Q}{A}$

$$= \frac{\pi}{8\mu}(-\frac{\partial p}{\partial \text{x}})R^4 \quad \text{X} \; \frac{1}{\pi r^2}$$

$$\bar{u} = \frac{1}{8\mu}(-\frac{\partial p}{\partial x})\; R^2 \qquad (7)$$

Dividing equation (4) by equation (7)

$$Q = \int_{r=0}^{r=R} u(r) 2\pi r \mathrm{d}r,$$

$$= 2\pi V_c \int_0^R \left\{ 1 - \left(\frac{r}{R} \right)^2 \right\} r \mathrm{d}r,$$

$$= \frac{\pi R^2 V_c}{2}. \tag{16}$$

Substituting V_c in Eq. (16), we obtain

$$Q = \frac{\pi D^4 \Delta p}{128 \mu L}. \tag{17}$$

Average velocity of flow:

The average velocity of flow, V is given by

$$V = \frac{Q}{A}. \tag{18}$$

Substituting Q from Eq. (17) in Eq. (18) and cross-sectional area $A{=}\pi D^2$, we get

$$V = \frac{\Delta p D^2}{32 \mu L}. \tag{19}$$

Loss of pressure head (h_f):

$$h_f = \frac{\Delta p}{\rho g}. \tag{20}$$

Using Eq. (17) or Eq. (19) for Δp in Eq. (20), one obtains

$$h_f = \frac{32 \mu V L}{\rho g D^2} = \frac{128 \mu L Q}{\pi \rho g D^4}. \tag{21}$$

Eq. (21) is called the **Hagen Poiseuille** equation.

$$\boldsymbol{h_f} = \frac{\mathbf{32}\boldsymbol{\mu}\bar{u}\boldsymbol{L}}{\rho g \boldsymbol{D}^{\mathbf{2}}}$$ **(Hagen poiseuille equation)**

Example: Laminar Flow in Pipe

Problem:

A crude oil of viscosity 0.97 poise and density 900 kg/m^3 is flowing through a horizontal circular pipe of diameter 100 mm and of length 10 m. Calculate the difference of pressure at the two ends of pipe, if 100 kg of the oil is collected in a tank in 30 seconds.

Solution:

given,

μ = 0.97 poise = 0.097 Ns/m^2
density = 900 kg/m^3
D = 100 mm = 0.1 m L
= 10m
Mass M = 100 Kg
Time = 30 sec
Pressure drop

$$P_1 - P_2 = \frac{32\mu \bar{u} L}{D^2}$$

Average Velocity $\bar{u} = \frac{Q}{A}$

$$Q = \frac{Mass\ of\ oil\ collected\ per\ second}{density}$$

= (100 x 1)/ (30 x 900)

Q = 3.704 X10^{-3} m^3

$$\bar{u} = \frac{3.704 \times 10^{-3}}{\pi \times D^2} \times 4 = 0.472 \text{ m/s}$$

$$P_1 - P_2 = \frac{32\mu \bar{u} L}{D^2}$$

$$= \frac{32 \times 0.097 \times 0.472 \times 10}{0.1^2}$$

P_1-P_2 = 1465.1N/m^2

Turbulent Flow

Description

Since the shearing action in laminar flows is well understood, equations describing the flow were easily determined. In turbulent flows there is no simple description of the shear forces that act in the fluid. Therefore the solutions of problems involving turbulent flows usually involve experimental results.

In his work, Reynolds clarified two previous results found experimentally:

- Hagen and Poiseuille found that friction head loss is proportional to the mean velocity:

$$h_f \propto V$$

Reynolds found that this only applies to laminar flows, as we have seen.

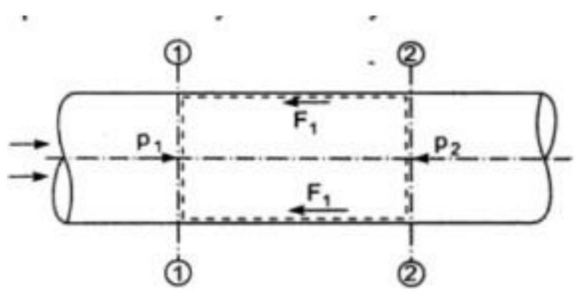
- Darcy and Weisbach found that friction head loss is proportional to the mean velocity squared:

$$h_f \propto V^2$$

Reynolds found that this applies to turbulent flows.

DARCY'S WEISBACH EQUATION

Consider a uniform horizontal pipe,having a steady flow as shown in fig.Let 1-1 and 2-2 are two setions of pipe.



Let,

P_1 and P_2= Pressure intensity at section 1-1 and 2-2, V_1 and

V_2=Velocity of flow at sec 1-1 and 2-2,

L = length of the pipe between sections 1-1 and 2-2 d = dia of the pipe,

f'' = frictional resistance per unit wetted area per unit velocity, h_f = loss of head due to friction

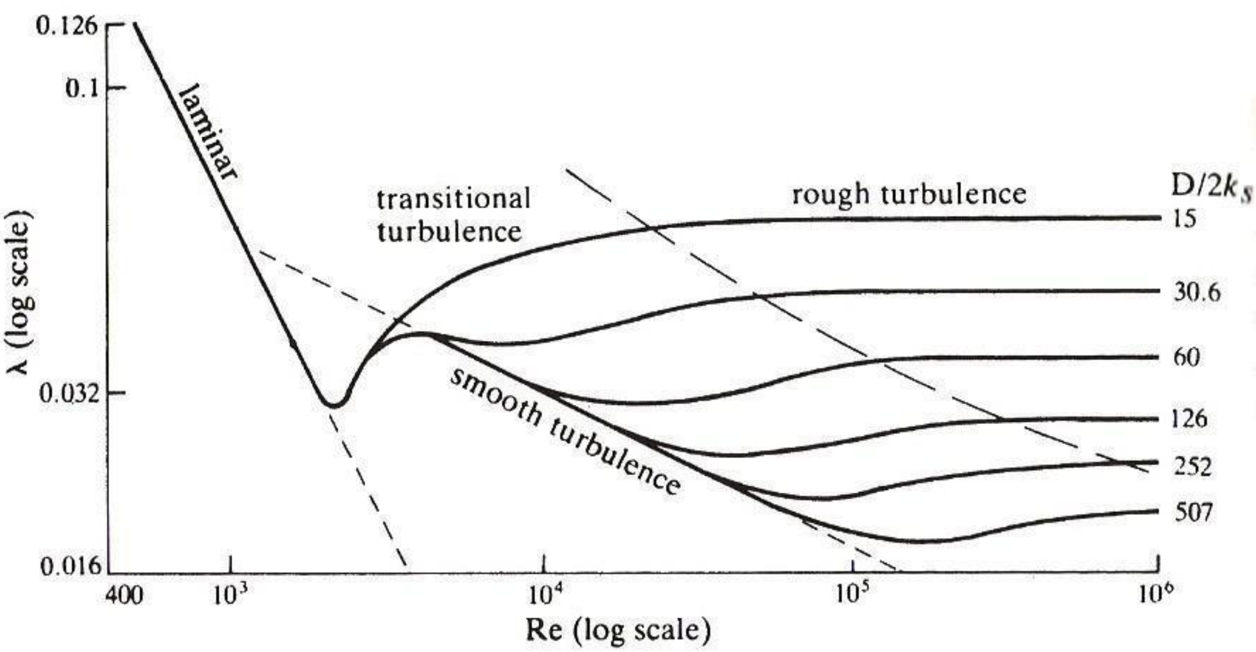
PIPE FRICTION FACTOR

Many experiments have been performed to determine the pipe friction factor for many different arrangements of pipes and flows.

Laminar Flow

We can just equate the Hagen-Poiseuille and the Darcy-Weisbach Equations:

$$\frac{\mathbf{32}\boldsymbol{\mu}\bar{u}\boldsymbol{L}}{\rho g\boldsymbol{D}^{\mathbf{2}}} = \frac{4f\,L\,v^2}{2gd}$$



Hence, for laminar flow we have:

$$f = \frac{64}{Re}$$

Smooth Pipes – Blasius Equation

Blasius determined the following equation from experiments on „smooth" pipes:

$$f = \frac{0.316}{Re^{0.25}}$$

Stanton and Pannell confirmed that this equation is valid for Re $<10^5$. Hence it is for „smooth" pipes.

Nikuradse's Experiments

Nikuradse carried out many experiments up to Re = 3×10^6 . In the experiments, he artificially roughened pipes by sticking uniform sand grains to smooth pipes. He defined the relative roughness (k_S/D) as the ration of the sand grain size to the pipe *diameter. He plotted his results as log f against log Re for each* k_s/D *,* shown below.

There are 5 regions of flow in the diagram:

1. Laminar Flow – as before;
2. Transitional flow – as before, but no clear f;
3. Smooth turbulence – a limiting line of turbulence as Re decreases for all k_S/D ;
4. Transitional turbulence – f varies both with Re and k_s/D , most pipe flows are in this region;
5. Rough turbulence - f is constant for a given k_s/D and is independent of Re.

The von Karman and Prandlt Laws

von Karman and Prandlt used Nikuradse‟s experimental results to supplement their own theoretical results which were not yet accurate. They found semi-empirical laws:

- Smooth pipes:

$$\frac{1}{\sqrt{f}} = 2\log\frac{Re\sqrt{f}}{2.51}$$

- Rough pipes:

$$\frac{1}{\sqrt{f}} = 2\log\frac{3.7}{K_S D}$$

The Von Karman and Prandlt Law for smooth pipes better fits the experimental data than the Blasius Equation.

The Colebrook-White Transition Formula

The friction factors thus far are the result of experiments on artificially roughened pipes. Commercial pipes have roughnesses that are uneven in both size and spacing. Colebrook and White did two things:

1. They carried out experiments and matched commercial pipes up to Nikuradse‟s results by finding an „effective roughness‟ for the commercial pipes:

Pipe/Material	k_s(mm)
Brass, copper, glass, Perspex	0.003
Wrought iron	0.06
Galvanized iron	0.15
Plastic	0.03
Concrete	6.0

this equation is known as the **Colebrook-White transition formula** and it gives results very close to experimental values for transitional behaviour when using effective roughnesses for commercial pipes.

$$\frac{1}{f} = -2\log \frac{K_S}{3.7\,D} + \frac{2.51}{R_e\, f}$$

The transition formula must be solved by trial and error and is not expressed in terms of the preferred variables of diameter, discharge and hydraulic gradient. Hence it was not used much initially.

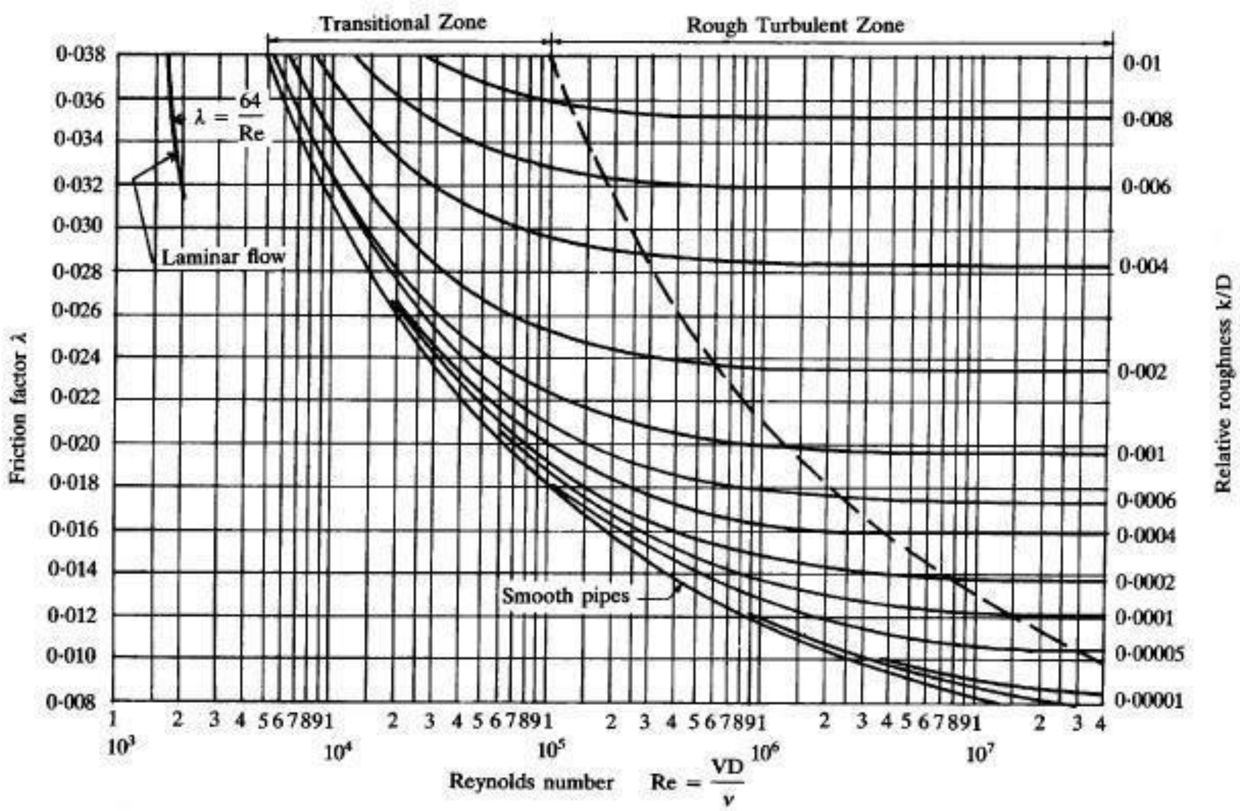
Moody

Moody recognized the problems with the Colebrok-White transition formula and did two things to remove objections to its use:

1. He presented an approximation to the Colebrook-White formula:

$$f=\left(\frac{200K_s}{D}+\frac{10^6}{R_e}\right)^{\frac{1}{3}}$$

2. He plotted *f (or)λ* against log Re for commercial pipes, this is now known as the Moody diagram:



LOSS OF ENERGY IN PIPES:

When a fluid is flowing through a pipe, the fluid experiences some resistance due to which some of the energy of fluid is lost. The energy loss is classified as

Major Energy Losses - due to friction

Minor Energy Losses

- Sudden expansion of pipes
- Sudden contraction of pipes
- Loss of head at the entrance of the pipe
- Loss of head at the exit of the pipe
- Bend in pipe
- Pipe fittings
- An obstruction in pipe

Problem for Major Loss:

Find the head lost due to friction in a pipe of diameter 300 mm and length 50 m, through which water is flowing at velocity of 3 m/s using (i) Darcy formula, (2) Chezy‟s formula for which C= 60. Take Kinematic viscosity ϑ = 0.01 stoke.

Solution:

Given:

d= 300 mm=0.3 m

L= 50 m

V= 3 m/s

C= 60

ϑ = 0.01 stoke = 0.01 x 10^{-4} m^2/s

(1) Darcy formula

$$\square\, f = \frac{4f\,l\,v^2}{2gd}$$

$$R_e = \frac{vd}{\vartheta}$$

$= (3 \text{ x } .3)/(0.01 \text{ x } 10^{-4})$

$R_e = 9 \text{ x } 10^5$

for R_e 4000 to 10^6

$$f = \frac{0.079}{R_e^{1/4}}$$

$= (0.079)/(9 \text{ x } 10^5)^{1/4}$ f = 0.00256

$h_f = (4 \text{ x } 0.00256 \text{ x } 50 \text{ x } 3^2)/(0.3\text{x } 2\text{x}9.81)$ h_f = 0.7828 m.

(2) Chezy‟s Formula:

$$R_e = \frac{vd}{\vartheta}$$

$= (3 \text{ x } .3)/(0.01 \text{ x } 10^{-4})$

$R_e = 9 \text{ x } 10^5$

for R_e 4000 to 10^6

$$f = \frac{0.079}{R_e^{1/4}}$$

$= (0.079)/(9 \text{ x } 10^5)^{1/4}$

f = 0.00256

$h_f = (4 \text{ x } 0.00256 \text{ x } 50 \text{ x } 3^2)/(0.3\text{x } 2\text{x}9.81)$

h_f = 0.7828 m.

(2) Chezy's Formula:

$$V = C\sqrt{mi}$$

m = d/4=0.3/4=0.075m

$3 = 60\sqrt{0.075 \ x \ i}$

i = 0.0333

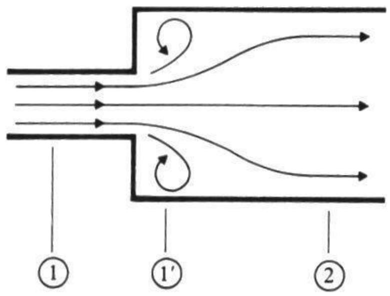
$$i = \frac{h_f}{L} = \frac{h_f}{50}$$

h_f = 50 x 0.0333 = 1.665m.

MINOR LOSSES

Losses at Sudden Enlargement:

Consider the flow in the sudden enlargement, shown in figure , fluid flows from section 1 to section 2. The velocity must reduce and so the pressure increases (this follows from Bernoulli). At position 1' turbulent eddies occur which give rise to the local head loss.



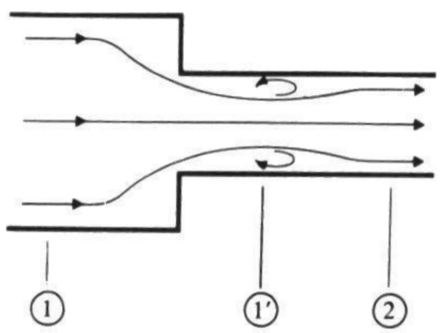
$$h_e = \frac{(v_1 - v_2)^2}{2g}$$

h_e = loss of head due to sudden enlargement

v_1 and v_2 velocity of flow at section 1-1 and 2-2.

Losses at Sudden Contraction

In a sudden contraction, flow contracts from point 1 to point 1', forming a vena contraction



$$h_c = 0.5\frac{v_2^2}{2g}$$

h_c - Loss of head due to sudden contraction.

Loss of head at the entrance of the pipe (h_i):

It occurs when a liquid enters a pipe which is connected to large tank or reservoir.

$$h_i = 0.5 \frac{v^2}{2g}$$

v- velocity of liquid in a pipe.

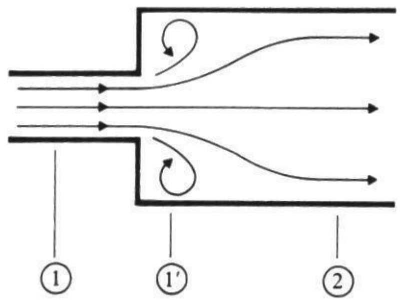
Loss of head at the exit of the pipe (h_o) :

This will occur due to the velocity of liquid at outlet of the pipe which is dissipated in the form of free jet or it lost in tank or reservoir.

$$h_o = \frac{v^2}{2g}$$

v- velocity of liquid in a pipe.

Bend in pipe (h_b):



Due to bend in pipe the velocity of flow changes hence separation of the flow from boundary and eddies will occur.

$$h_b = K\frac{v^2}{2g}$$

v- velocity of flow

k- co- efficient of bend k

value depends on

- Angle of bend
- Radius of curvature
- Diameter of pipe

Pipe fittings

The loss of head in the various pipe fittings such as valves, couplings = $K\frac{v^2}{2g}$

v- velocity of flow

k- co- efficient of pipe fitting.

An obstruction in pipe

The loss of energy takes place due to reduction of the area of the cross section of the pipe at the place where obstruction is found.

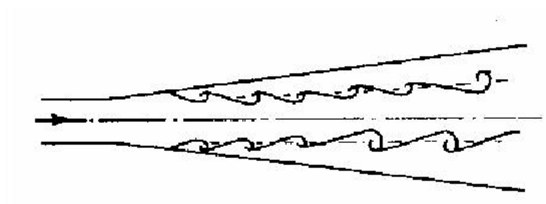
Head loss due to obstruction = $\frac{v^2}{2g}\left(\frac{A}{C_c(A-a)} - 1\right)^2$

v- velocity of flow

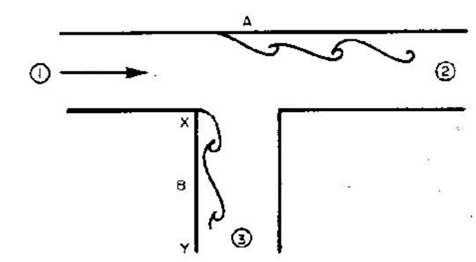
C_C – Coefficient of contraction A-

Area of pipe
a- Maximum area of obstruction.

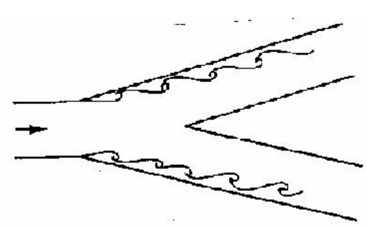
Some common situation where significant head losses occur in pipe are shown in figure



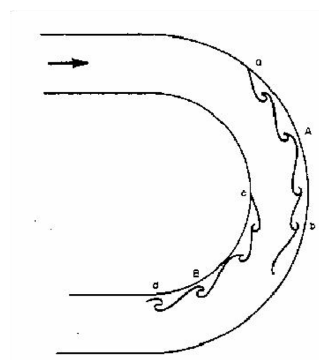
A divergent duct or diffuser



Tee-Junctions



Y-Junctions



Bends

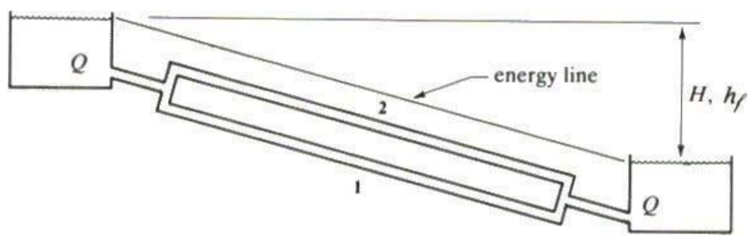
Pipes in series

When pipes of different diameters are connected end to end to form a pipe line, they are said to be in series. The total loss of energy (or head) will be the sum of the losses in each pipe plus local losses at connections.

Pipes in parallel

When two or more pipes in parallel connect two reservoirs, as shown in Figure 17, for example, then the fluid may flow down any of the available pipes at possible different rates. But the head difference over each pipe will always be the same.
The total volume flow rate will be the sum of the flow in each pipe.

The analysis can be carried out by simply treating each pipe individually and summing flow rates at the end.



Energy Grade Line (EGL) and Hydraulic Grade Line (HGL)

Graphical interpretations of the energy along a pipeline may be

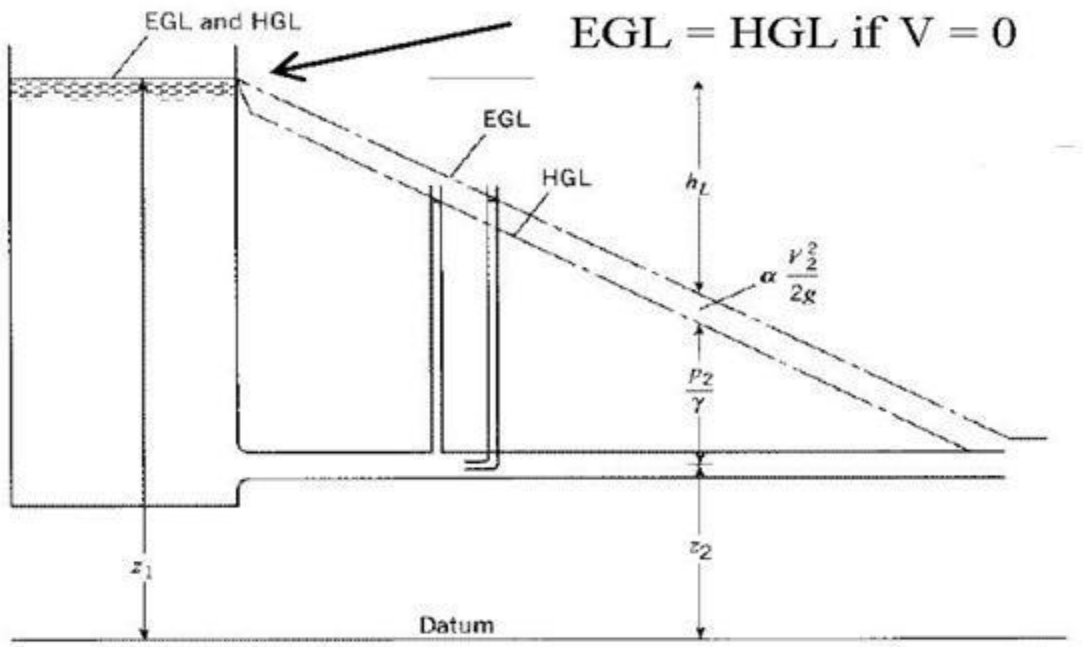
obtained through the EGL and HGL:

$$EGL = \frac{p}{\rho g} + \frac{v^2}{2g} + z$$

$$HGL = \frac{p}{\rho g} + \frac{v^2}{2g}$$

EGL and HGL may be obtained via a pitot tube and a piezometer tube,

Respectively.



$h_f = h_L$ loss of head due to friction.

Hints:

1. EGL = HGL + $V^2/2g$, EGL = HGL for V=0

2. If p=0, then HGL=z

3. A change in pipe diameter leads to a change in V ($V^2/2g$) due to continuity and thus a change in distance between HGL and EGL

4. A change in head loss (h_L) leads to a change in slope of EGL and HGL.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – IV – MECHANICS OF FLUID – SCIA1301

UNIT IV

BOUNDARY LAYER

1.0 BOUNDARY LAYER:

- When a real fluid flows over a solid body or a solid wall, the fluid particles adhere to the boundary and condition of no slip occurs. This means that the velocity of fluid close to the boundary will be as that of the boundary.
- If the boundary is stationary, velocity of fluid at the boundary will be zero. Farther away from the boundary, velocity will be higher and as a result of this variation of velocity, velocity gradient du/dy will exist.
- The velocity of fluid increases from zero velocity on the stationary boundary to free stream velocity (U) of fluid in the direction normal to the boundary. This variation of velocity takes place in a narrow region in the vicinity of solid boundary. This narrow region of the fluid is called boundary layer.

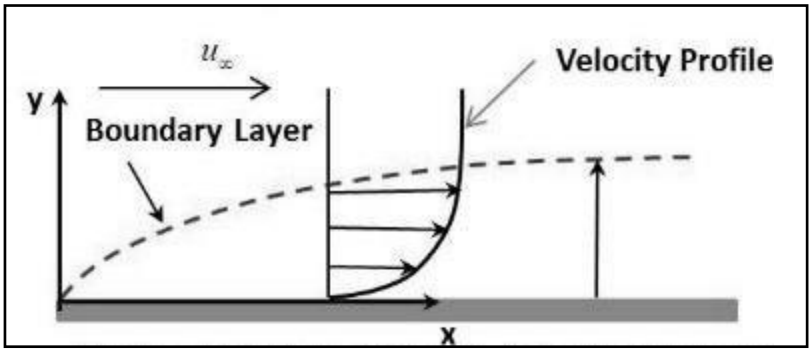
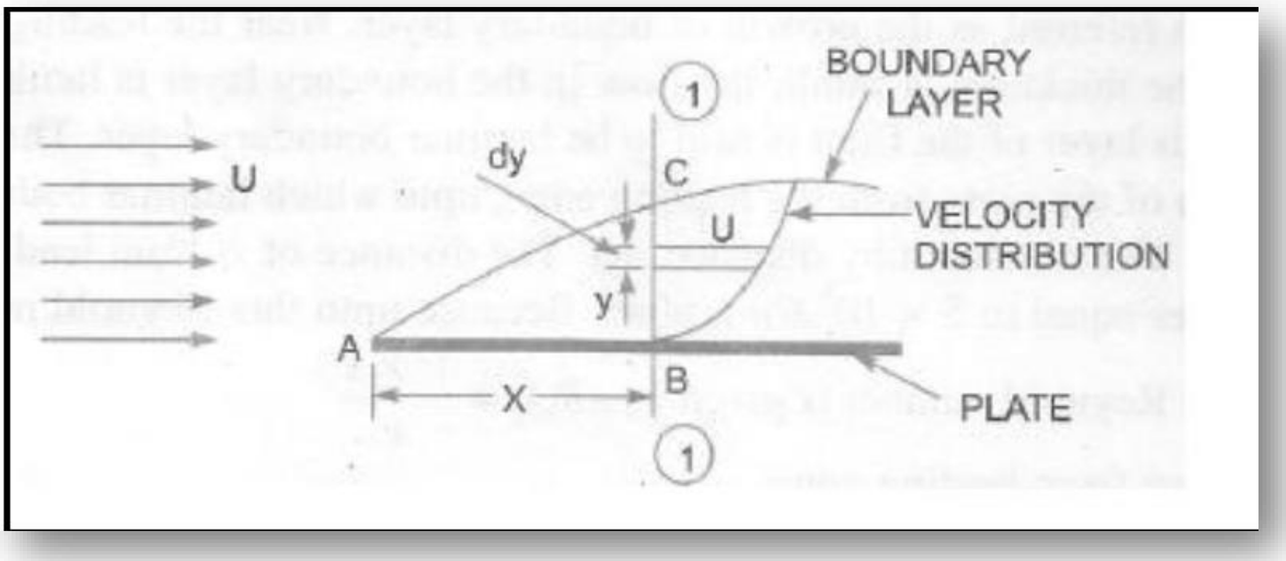


Fig.1: Velocity Profile

BOUNDARY LAYER THEORY:

- The theory dealing with boundary layer flows is called boundary layer theory. According to this theory, the flow of fluid of solid boundary is divided into two regions.
- A very thin layer of fluid called boundary layer in the immediate neighborhood of solid boundary, where the variation of velocity from zero to free stream velocity. In this region, velocity gradient du/dy exists and hence the fluid exerts a shear stress on the wall in the direction of motion. The value of shear stress , $\tau = \mu$ du/dy.

- The remaining fluid which is outside the boundary layer. The velocity outside the boundary layer is constant and equal to free stream velocity. As there is no variation of velocity in this region, velocity gradient du/dy becomes zero. As a result of this, shear stress is zero.



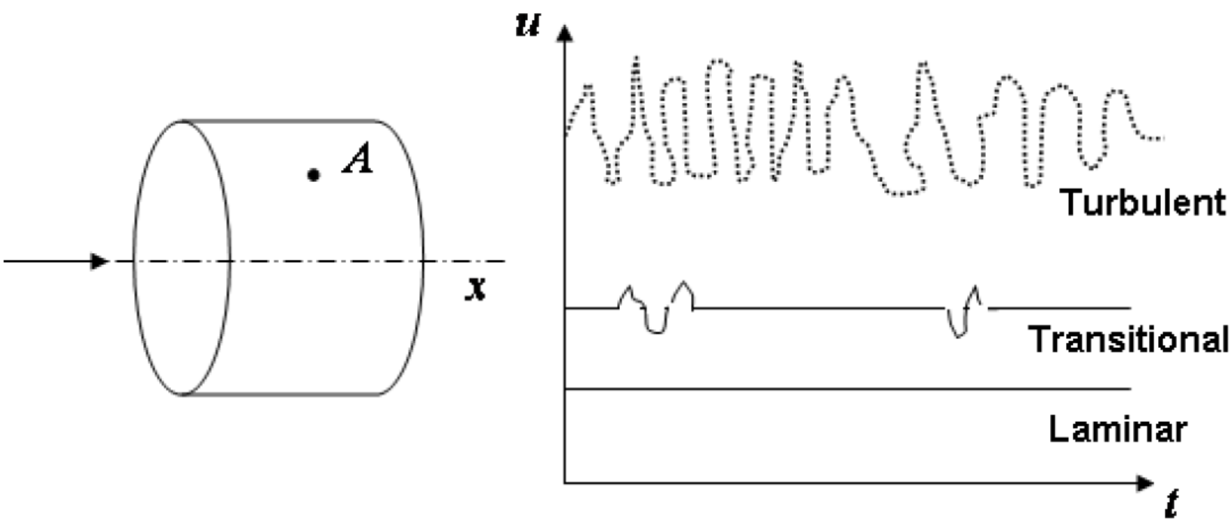


Fig. 5.1.1: Time dependent fluid velocity at a point.

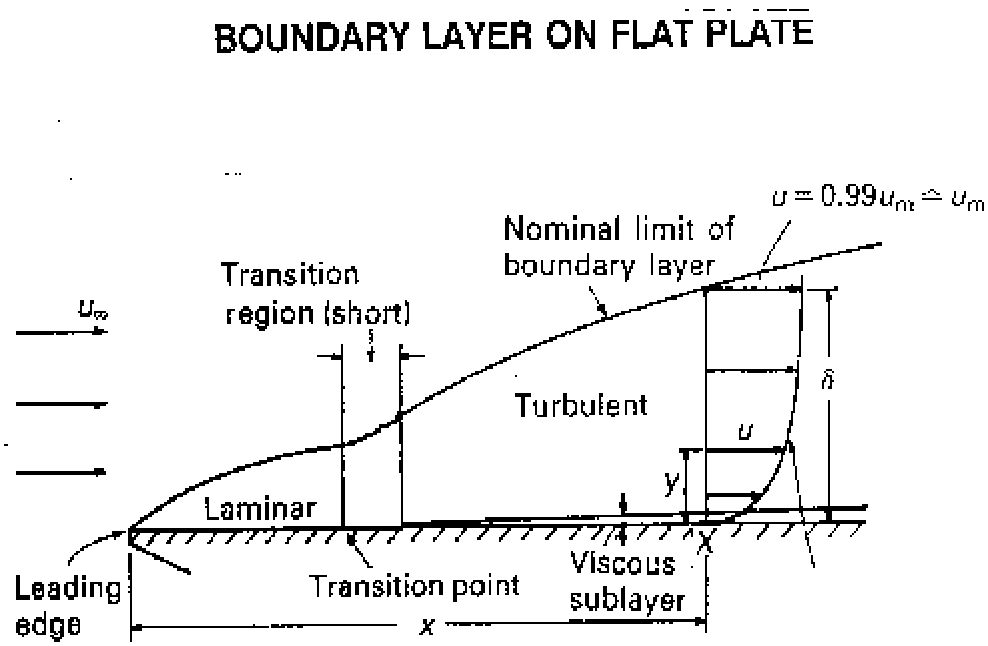


Fig.2: Boundary layer on flat plate

LAMINAR BOUNDARY LAYER:

- A laminar boundary layer is one where the flow takes place in layers, i.e., each layer slides past the adjacent layers.
- Laminar boundary layers are found only when the Reynolds numbers are small.
- A thin layer over the surface of a body immersed in a fluid, in which the fluidvelocity rel ative to the surface increases rapidly with distance from the surface and the flow is laminar.

TURBULENT BOUNDARY LAYER:

- If the length of the plate is more, the thickness of boundary layer will go on increasing in the downstream direction. Then the laminar boundary layer becomes unstable and motion of fluid within it, is disturbed and irregular which leads to a transition from laminar to turbulent layer.
- This short length over which the boundary layer flow changes from laminar to turbulent is called transition zone. Further downstream the transition zone, the boundary layer is turbulent and continues to grow in thickness. This layer of boundary is called turbulent boundary layer.

LAMINAR SUB LAYER:

- This is the region in the turbulent boundary layer zone, adjacent to the solid surface of the plate. In this zone, the velocity variation is influenced only by viscous effects.
- The shear stress in the laminar sub layer would be constant and equal to the boundary shear stress τ_0. Thus the shear stress in the sub layer is

$$\tau_0 = \mu \left[\frac{du}{dy}\right]_{y=0}$$

BOUNDARY LAYER THICKNESS (δ):

- It is defined as the distance from the boundary of the solid body measured in the y – direction to the point, where the velocity of the fluid is approximately equal to 0.99 times the free stream velocity(U) of the fluid.

DISPLACEMENT THICKNESS(δ *):

- It is defined as the distance measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation.

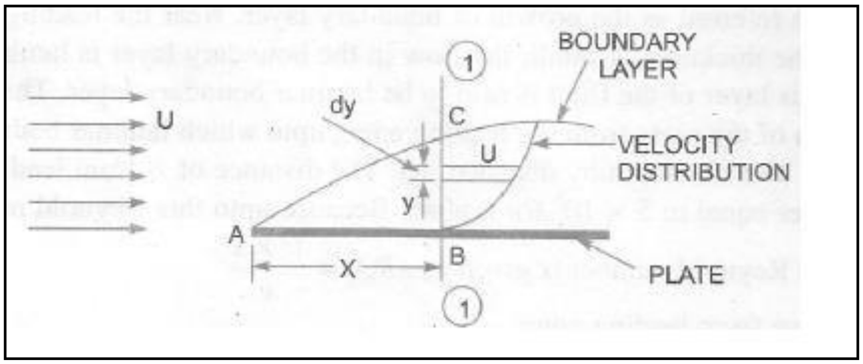


Fig.3 : Boundary layer on flat plate

Consider the flow of a fluid having free stream velocity equal to U over a thin smooth plate. At a distance x from the leading edge consider a section 1-1. The velocity of fluid at B is zero and at C, which lies on the boundary layer is U. thus velocity varies from zero at B to U at C, where BC is equal to the thickness of boundary layer.

Distance BC = δ

At the section 1-1, consider an elemental strip.

Let y = distance of elemental strip from the plate.

dy = thickness of the elemental strip

u = velocity of fluid at the elemental strip

b = width of plate.

Then area of elemental strip, dA = b x dy

Mass of fluid per second flowing through elemental strip = ρ X velocity X Area of elemental Strip.

$$= \rho u \times dA = \rho u \times b \times dy \quad \text{——— (i)}$$

If there had been no plate, then the fluid would have been flowing with a constant velocity equal to free stream velocity (U) at the section 1-1. Then mass of fluid per second flowing through elemental strip would have been = ρ x velocity x Area = ρ x U x b x dy —— (ii)

As U is more than u, hence due to the presence of the plate and consequently due to the formation of the boundary layer , there will be a reduction in mass flowing perr second through the elemental strip.

This reduction in mass/sec flowing through elemental strip

= mass/sec given by equation (ii) - mass/sec given by equation (i)

= ρUbdy – ρubdy = ρb (U–u) dy.

Therefore total reduction in mass of fluids flowing through BC due to plate

$$= \int_0^{\delta} \rho b (U - u)\, dy = \rho b \int_0^{\delta} (U - u)\, dy \qquad \text{(iii)}$$

Let the plate is displaced by a distance δ^* and velocity of flow for the distance δ^* is equal to the free stream velocity (U). loss of the mass of the fluid/sec flowing through the distance δ^*

= ρ x velocity x Area

= ρ x U x δ^* x b (iv)

Equating (iii) and (iv) , we get

$$\rho b \int_0^{\delta} (U - u)\, dy = \rho \times U \times \delta^* \times b$$

Cancelling ρb on both sides, we have

$$\int_0^{\delta} (U - u)\, dy = U \times \delta^*$$

$$\delta^* = \frac{1}{U}\int_0^{\delta} (U - u)\, dy = \int_0^{\delta} \frac{(U-u)\, dy}{U}$$

$$\delta^* = \int_0^{\delta} \left(1 - \frac{u}{U}\right) dy.$$

MOMENTUM THICKNESS (θ):

- It is defined as the distance measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in momentum of the flowing fluid on account of boundary layer formation.
- Consider the flow over a plate. Consider the section 1-1 at a distance x from leading edge. Take an elemental strip at a distance y from the plate having thickness dy. The mass of fluid flowing per second through this elemental strip is given by equation (i) and is equal to ρubdy.

  Momentum of this fluid = Mass X velocity = (ρubdy) u

  Momentum of this fluid in the absence of boundary layer = Mass X velocity = (ρubdy) U

  Loss of momentum through elemental strip = (ρubdy) U - (ρubdy) u = ρbu(U - u)dy

  Total loss of momentum/sec through BC = $\int_0^\delta \rho b u (U - u)\, dy$ ———— (v)

  Let θ = distance by which plate is displaced when the fluid is flowing with a constant velocity U.

  Loss of momentum/sec of fluid flowing through distance θ with a velocity U

  = Mass of fluid through X velocity

  =(ρ X Area Xvelocity) Xvelocity

  =(ρ X θ X b X U) X U

  = $\rho\theta b U^2$. ———— (vi)

  Equating equations (v) and (vi), we have

  $$\rho\theta b U^2 = \int_0^\delta \rho b u (U - u)\, dy = \rho b \int_0^\delta u (U - u)\, dy$$

  $$\theta U^2 = \int_0^\delta u (U - u)\, dy$$

  $$\theta = \frac{1}{U^2}\int_0^\delta u (U - u)\, dy$$

  $$\boldsymbol{\theta} = \int_0^\delta \frac{u}{U}\left(1 - \frac{u}{U}\right) dy$$

The displacement/momentum thickness has the following physical implications;

- The displacement thickness represents the amount of distance that thickness of the body must be increased so that the fictitious uniform non viscous flow has the same mass flow rate properties as the actual flow.
- It indicates the outward displacement of the streamlines caused by the viscous effects on the plate.
- The flow conditions in the boundary layer can be simulated by adding the displacement thickness to the actual wall thickness and thus treating the flow over a thickened body as in the case of non viscous flow.
- Both δ^* and θ^* are the integral thicknesses and the integrant vanishes in the free stream. So, it is relatively easier to evaluate δ^* and θ^* as compared to δ?

ENERGY THICKNESS (δ^{**}):

- It is defined as the distance measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in kinetic energy of the flowing fluid on account of boundary layer formation.
- Consider the flow over the plate having section 1-1 at a distance x from leading edge. The mas of fluid flowing per second through the elemental strip of thickness 'dy' at a distance y from the plate is given by = ρubdy.

  Kinetic energy of this fluid = $\frac{1}{2}$m X velocity^2

  $= \frac{1}{2}(\rho u b dy) \times u^2$

  Kinetic energy of this fluid in the absence of boundary layer = $\frac{1}{2}(\rho u b dy) \times U^2$

  Loss of kinetic energy through elemental strip $= \frac{1}{2}(\rho u b dy)U^2 - \frac{1}{2}(\rho u b dy)u^2$

  $= \frac{1}{2}\rho u b[U^2 - u^2]dy$

  Total loss of K.E of fluid passing through BC $= \int_0^{\delta} \frac{1}{2}\rho u b[U^2 - u^2]dy$

  $= \frac{1}{2}\rho b \int_0^{\delta} u(U^2 - u^2)dy$

  Let δ^{**} = distance by which the plate is displaced to compensate for the reduction in K.E.

  Loss of kinetic energy through δ^{**} of fluid flowing with velocity U

  $= \frac{1}{2}$(mass) velocity^2 $= \frac{1}{2}(\rho \times \text{area} \times \text{velocity})$ velocity^2

  $= \frac{1}{2}(\rho \times U \times \delta^{**} \times 1)U^2$

  $= \frac{1}{2}\rho b\,\delta^{**}U^3$

  Equating the two losses of K.E, we get

$$\frac{1}{2}\rho b\,\delta^{**}U^3 = \frac{1}{2}\rho b\int_0^{\delta} u(U^2 - u^2)dy$$

$$\delta^{**} = \frac{1}{U^3}\int_0^{\delta} u(U^2 - u^2)dy$$

$$\delta^{**} = \int_0^{\delta} \frac{u}{U}\left[1 - \frac{u^2}{U^2}\right]dy.$$

DRAG FORCE ON A FLAT PLATE DUE TO BOUNDARY LAYER:

Consider the flow of a fluid having free stream velocityequal to U, over a thin plate. The drag force on the plate can be determined if the velocity profile near the plate is known. Consider a small length Δx of the plate at a distance of x from the leading edge.

The shear stress $\tau_o$is given by $\tau_o = \mu\left(\frac{du}{dy}\right)_{y=0}$,where $\left(\frac{du}{dy}\right)_{y=0}$ is the velocity distribution near the plate at y = 0.

Then drag force or shear force on a small distance Δx is given by

ΔF_D = shear stress X area

$= \tau_o \times \Delta x \times b. \longrightarrow$ (i)

Where ΔF_D = drag force on distance Δx.

The drag force ΔF_D must also be equal to the rate of change of momentum over the distance . consider the flow over the small distance Δx.Let ABCD is the control volume of the fluid over the distance . the edge DC represents the outer edge of the boundary layer.

Let u – velocity at any point within the boundary layer.

b – width of plate.

Then mass rate of flow entering through the side AD = $\int_0^{□}$□ x velocity x area of strip of thickness dy

$= \int_0^{□}$□ x □ x □ x □□

$= \int_0^{□}$□ □□□□

Mass rate of flow leaving the side BC = mass through AD + $\frac{\partial}{\partial x}$ mass through AD x Δx.

$= \int_0^{□}$□ □□□□ + $\frac{\partial}{\partial x} \int_0^{□}$□ □□□□ x Δx.

From continuity equation for a steady incompressible fluid flow , we have

Mass rate of flow entering AD + Mass rate of flow entering DC = Mass rate of flow leaving BC

Mass rate of flow entering DC = Mass rate of flow leaving BC - Mass rate of flow entering AD

$= \int_0^{□}$□□□□□ + $\frac{\partial}{\partial x} \int_0^{□}$□ □□□□ x Δx - $\int_0^{□}$□□□□□

$= \frac{\partial}{\partial x} \int_0^{□}$□ □□□□ x Δx

The fluid is entering through side DC with a uniform velocity U.now let us calculate momentum flux through control volume.

Momentum flux entering through AD = $\int_0^{□}$□□□□□□□□ □□□□ □□□□□□□□ □□□□□ □□ □□□□□□□□□ □□

$= \int_0^{□}$□□□□ □□□□□□□□ □□□□□ x □□□□□□□□□

$= \int_0^{□}$□ □□□□ x u = $\int_0^{□}$□ u2 □□□

Momentum flux leaving the side BC = $\int_0^{□}$□ u2 □□□ + $\frac{\partial}{\partial x} \int_0^{□}$□ u2 □□□ x Δx

Momentum flux entering the side DC = mass rate through DC x velocity

$= \frac{\partial}{\partial x} \int_0^{□}$□ □□□□ x Δx x U

$$= \frac{\partial}{\partial x} \int_0^{\square} \square\ \square\ \square\square\square\square \text{ x } \Delta x$$

As U is constant and so it can be takeninside the differential and integral.

Rate of change of momentum of the control volume = Momentum flux through BC - Momentum flux through AD - Momentum flux through DC.

$$= \int_0^{\square} \square \text{ u2 } \square\square\square + \frac{\partial}{\partial x} \int_0^{\square} \square \text{ u2 } \square\square\square \text{ x } \Delta x - \int_0^{\square} \square \text{ u2 } \square\square\square - \frac{\partial}{\partial x} \int_0^{\square} \square\ \square\ \square\square\square\square \text{ x } \Delta x$$

$$= \frac{\partial}{\partial x} \square\square \int_0^{\square} (\square 2 - \square\ \square)\square\square \text{ x } \Delta x$$

$$= \square\square \frac{\partial}{\partial x} \left[\int_0^{\square} (\square^2 - \square\ \square)\square\square \right] \text{ x } \Delta x \longrightarrow \quad \text{(ii)}$$

Now the rate of change of momentum on the control volume ABCD must be equal to the total force on thecontrol volume in the same direction according to the momentum principle. But for a flat plate $\frac{\partial}{\partial x}$ = 0, which means there is no external pressure force on the control volume. Also the force on the side DC is negligible as the velocity is constant and velocity gradient is zero. The only external force acting on the control volume is the shear force acting on the side AB in the direction from B to A. The value of this force is given by,

$\Delta F_D = \tau_o \text{ x } \Delta x \text{ x b}$.

Total external force in the direction of rate of change of momentum

$$= -\tau_o \text{ x } \Delta x \text{ x b}. \longrightarrow \quad \text{(iii)}$$

According to momentum principle, the two values given by (ii) and (iii) should be same.

$$-\tau_o \text{ x } \Delta x \text{ x b} = \square\square \frac{\partial}{\partial x} \left[\int_0^{\square} (\square 2 - \square\ \square)\square\square \right] \text{ x } \Delta x$$

Cancelling Δx x b on both sides, we have

$$-\tau_o = \square \frac{\partial}{\partial x} \left[\int_0^{\square} (\square 2 - \square\ \square)\square\square \right]$$

$$\tau_o = \square \frac{\partial}{\partial x} \left[\int_0^{\square} (\square\ \square - \square 2)\square\square \right]$$

$$= \frac{\partial}{\partial x} \int_{\square}^{\square\square} \frac{-}{\square} \left(\square - \frac{\square}{\square}\right) \square\square$$

$$\boldsymbol{\frac{\tau_0}{\rho U^2} = \frac{\partial \Theta}{\partial x}}$$

The above equation is known as Von karman momentum integral equation for boundary layer flows.

This is applied to

1. Laminar boundary layers
2. Transition boundary layers and
3. Turbulent boundary layers flows.

PROBLEMS:

For the velocity profile $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$, find the thickness of boundary layer at the end of the plate and the drag force on one side of a plate 1 m long and 0.8 m wide when placed in water flowing with a velocity of 150 mm per second. Calculate the value of co efficient of drag also. Take μ for water = 0.01 poise.

Solution:

Reynold number at the end of the plate is given by,

$R_{eL} = \frac{\rho UL}{\mu}$

$= \frac{1000 \times 0.15 \times 1}{0.001}$

$= 150000.$

(i) As laminar boundary layer exists upto Reynold number = 2 X 10^5. Hence this is the case of laminar boundary layer.

Thickness of boundary layer at x = 1 m is given by,

$\delta = 5.48 \frac{x}{\sqrt{Rex}} = \frac{5.48 \times 10}{\sqrt{150000}}$

$= 0.01415$ m or 14.15 mm.

(ii) Drag force on one side of the plate is given by,

$F_D = 0.73\, b\, \mu U \sqrt{\rho UL/\mu}$

$= 0.73 \times 0.8 \times 0.001 \times 0.15 \times 150000$

$= 0.0338$ N.

(iii) Co - efficient of drag is given by,

$C_D = \frac{1.46}{\sqrt{ReL}}$

$= \frac{1.46}{\sqrt{150000}} = 0.00376.$

LOCAL CO - EFFICIENTOF DRAG (C_D^*) :

- It is defined as the ratioof the shear stress to the quantity½ρU^2.

$$C_D = \frac{\tau_o}{½\rho U^2}$$

AVERAGE CO - EFFICIENT OF DRAG (C_D) :

- It is defined as the ratio of the total drag force to the quantity ½ρAU^2.
- It is also called coefficient of drag.

$$C_D = \frac{F_D}{½\rho AU^2}$$

BOUNDARY CONDITIONS FOR THE VELOCITY PROFILES:

- The following are the boundary conditions which must be satisfied by any velocity profile whether it is laminar or turbulent boundary layer zone.

1) At y = 0, u = 0 and du/dy.
2) At y = δ , u = U.
3) At y = δ , du/dy = 0.

BOUNDARY LAYER SEPARATION:

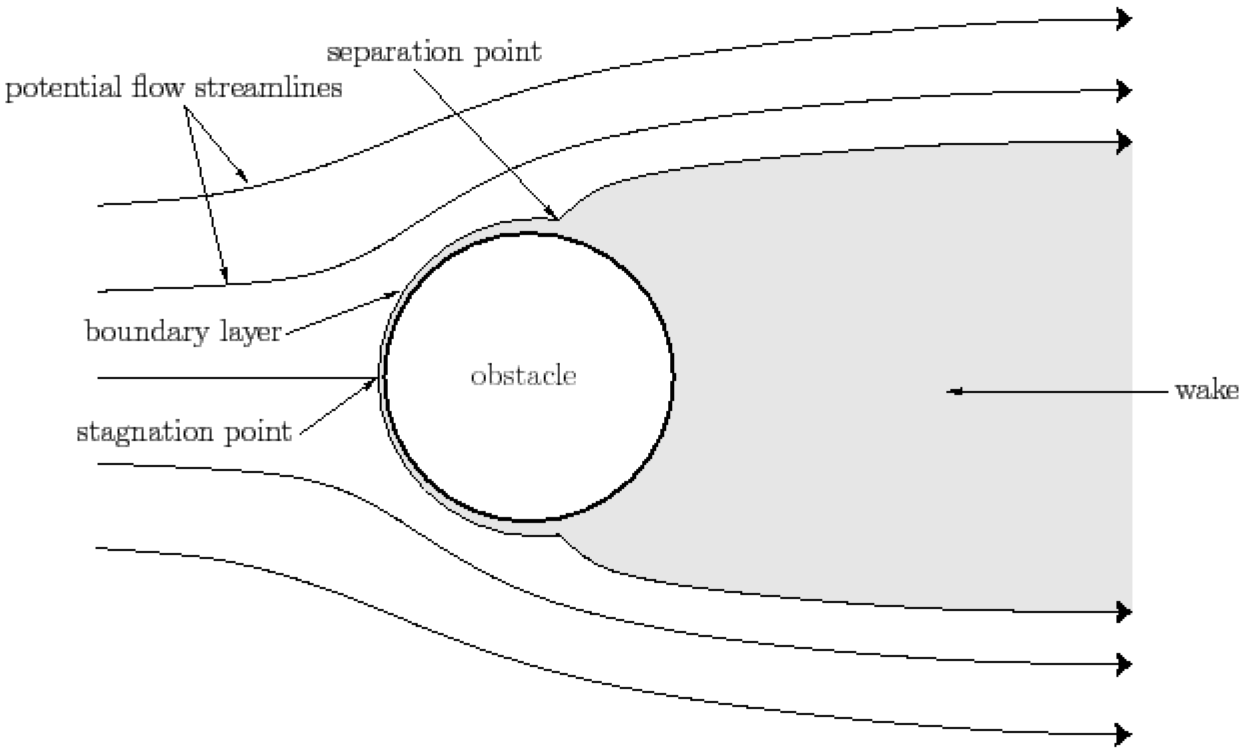
- In a flowing fluid when a solid body is immersed, a thin layer of fluid called the boundary layer is formed adjacent to solid body.

The forces acting on the fluid in the boundary layer are

1. Inertia force
2. Viscous force
3. Pressure force

- When the pressure gradient in the direction of flow is negative (dp/dx <0) that is when the pressure decreases in the direction of flow, the flow is accelerated.
- In this case, pressure and inertia force add together and jointly tend to reduce the effect of viscous forces in boundary layer.

- This results in a decreases in the thickness of boundary layer in the direction of flow as a consequence of which there are low losses and high efficiencies in accelerating flows.
- When the pressure increases in the direction of flow(dp/dx > 0), pressure forces acts opposite to the direction of flow and further increases the retarding effect of viscous forces. Subsequently the thickness of boundary layer increases rapidly in the direction of flow.
- If these forces act over a long stretch, the boundary layer gets separated from the surface and moves into the main stream. This phenomenon is called separation. The point of the body at which the boundary layer is on the verge of the separation from the surface is called point of separation.



- Boundary layer separation is the detachment of a boundary layer from the surface into a broader wake.
- Boundary layer separation occurs when the portion of the boundary layer closest to the wall or leading edge reverses in flow direction.
- The separation point is defined as the point between the forward and backward flow, where the shear stress is zero.
- The overall boundary layer initially thickens suddenly at the separation point and is then forced off the surface by the reversed flow at its bottom.
- The flow separation depends upon factors such as

Curvature of the surface

Reynolds number of the flow

Roughness of the surface

- The velocity gradient for a given velocity profile exhibits the following characteristics for the flow to remain attached, get detached or be the verge of separation.

1) (δu/ δy) @ y = 0 is positive $\longrightarrow$ Attached flow
2) (δu/ δy) @ y = 0 is zero $\longrightarrow$ Flow is on the verge of separation
3) (δu/ δy) @ y = 0 is negative $\longrightarrow$ Separated flow

- Separation occurs in the following cases,
    1) Diffusers
    2) Open channel transitions
    3) Pumps
    4) Fans
    5) Aerofoils
    6) Turbine blades etc.

METHODS OF PREVENTING THE SEPARATION OF BOUNDARY LAYER:

- Following are some of the methods generally adopted to retard the flow separation.

1) Streamlining the body shape.
2) Tripping the boundary layer from laminar to turbulent by provision of surface roughness.
3) Sucking the retarded flow.
4) Injecting high velocity fluid in the boundary layer.
5) Providing slots near the leading edge.
6) Guidance of flow in a confined passage.
7) Providing a rotating cylinder near the leading edge.
8) Energizing the flow by introducing optimum amount of swirl in the incoming flow.

Problem:

Find the displacement thickness, the momentum thickness and energy thickness for the velocity distribution in the boundary layer given by $\frac{u}{U}=\frac{y}{\delta}$, where u is the velocity at a distance y from the plate and u=U at y=δ, where δ= boundary layer thickness. Also calculate the value of $\frac{\delta^*}{\theta}$.

Solution:

Velocity distribution $\frac{u}{U} = \frac{y}{\delta}$

- Displacement Thickness (δ^*):

$$\delta^* = \int_0^{\delta} \left(1 - \frac{u}{U}\right) dy$$

$$\delta^* = \int_0^{\delta} \left(1 - \frac{y}{\delta}\right) dy$$

$$= \left[y - \frac{y^2}{2\delta} \right]_0^{\delta}$$

$$= \delta - \frac{\delta^2}{2\delta}$$

$$\delta^* = \frac{\delta}{2}$$

- Momentum Thickness (θ):

$$\delta^{**} = \frac{\delta^2}{4}$$

S.NO	Velocity Distribution	δ	τ_0	C_D
1	$\frac{u}{U} = 2\frac{y}{\delta} - \left(\frac{y}{\delta}\right)^2$	$\frac{5.48x}{\sqrt{Re_x}}$	$0.365\frac{\rho U^2}{\sqrt{Re_x}}$	$\frac{1.46}{\sqrt{Re_L}}$
2	$\frac{u}{U} = \frac{3}{2}\frac{y}{\delta} - \frac{1}{2}\left(\frac{y}{\delta}\right)^3$	$\frac{4.64x}{\sqrt{Re_x}}$	$0.323\frac{\rho U^2}{\sqrt{Re_x}}$	$\frac{1.292}{\sqrt{Re_L}}$
3	$\frac{u}{U} = 2\frac{y}{\delta} - 2\left(\frac{y}{\delta}\right)^3 + \left(\frac{y}{\delta}\right)^4$	$\frac{5.84x}{\sqrt{Re_x}}$	$0.34\frac{\rho U^2}{\sqrt{Re_x}}$	$\frac{1.36}{\sqrt{Re_L}}$
4	$\frac{u}{U} = \sin\left(\frac{\pi y}{2\delta}\right)$	$\frac{4.79x}{\sqrt{Re_x}}$	$0.327\frac{\rho U^2}{\sqrt{Re_x}}$	$\frac{1.31}{\sqrt{Re_L}}$
5	Blasius ‘s Solution	$\frac{4.91x}{\sqrt{Re_x}}$	$0.332\frac{\rho U^2}{\sqrt{Re_x}}$	$\frac{1.328}{\sqrt{Re_L}}$

For the velocity profile $\frac{u}{U} = 2\left(\frac{y}{\delta}\right) - \left(\frac{y}{\delta}\right)^2$, find the thickness of boundary layer at the end of the plate and the drag force on one side of a plate 1 m long and 0.8 m wide when placed in water flowing with a velocity of 150 mm per second. Calculate the value of co efficient of drag also. Take μ for water = 0.01 poise.

Solution:

Reynold number at the end of the plate is given by,

$$R_{eL} = \frac{\rho UL}{\mu}$$

$$= \frac{1000 \text{ X } 0.15 \text{ X } 1}{0.001}$$

$= \mathbf{150000.}$

(i) As laminar boundary layer exists upto Reynold number = 2 X 10^5. Hence this is the case of laminar boundary layer.

Thickness of boundary layer at x = 1 m is given by,

$$\delta = 5.48 \frac{x}{\sqrt{Rex}} = \frac{5.48 \text{ X } 10}{\sqrt{150000}}$$

= 0.01415 m or 14.15 mm.

(ii)Drag force on one side of the plate is given by,

$$F_D = 0.73 \text{ b } \mu U \sqrt{\rho UL/?}$$

= 0.73 X 0.8 X 0.001 x 0.15 X 150000

= 0.0338 N.

(iii)Co - efficient of drag is given by,

$$C_D = \frac{1.46}{\sqrt{ReL}}$$

$$= \frac{1.46}{\sqrt{150000}} = \mathbf{0.00376.}$$

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – V – MECHANICS OF FLUID – SCIA1301

UNIT-V

Dimensional Analysis and Model Studies

1. Introduction

Many practical flow problems of different nature can be solved by using equations and analytical procedures, as discussed in the previous modules. However, solutions of some real flow problems depend heavily on experimental data and the refinements in the analysis are made, based on the measurements. Sometimes, the experimental work in the laboratory is not only time -consuming, but also expensive. So, the dimensional analysis is an important tool that helps in correlating analytical results with experimental data for such unknown flow problems. Also, some dimensionless parameters and scaling laws can be framed in order to predict the prototype behavior from the measurements on the model. The important terms used in this module may be defined as below;

Dimensional Analysis: The systematic procedure of identifying the variables in a physical phenomena and correlating them to form a set of dimensionless group is known as dimensional analysis.

Dimensional Homogeneity: If an equation truly expresses a proper relationship among variables in a physical process, then it will be dimensionally homogeneous. The equations are correct for any system of units and consequently each group of terms in the equation must have the same dimensional representation. This is also known as the law of dimensional homogeneity

Dimensional analysis is a mathematical technique used to predict physical parameters that influence the flow in fluid mechanics, heat transfer in thermodynamics, and so forth. The analysis involves the fundamental units of dimensions MLT: mass, length, and time. It is helpful in experimental work because it provides a guide to factors that significantly affect the studied phenomena. It is commonly used to determine the relationships between several variables, i.e. to find the force as a function of other variables when an exact functional relationship is unknown. Based on understanding of the problem, we assume a certain functional form.

Dimensional analysisis a means of simplifying a physical problem by appealing to dimensional homogeneity to reducethe number of relevant variables.

It is particularly useful for:

- presenting and interpreting experimental data;
- attacking problems not amenable to a direct theoretical solution;
- checking equations;
- establishing the relative importance of particular physical phenomena;
- Physical modeling.

Dimensional variables: These are the quantities, which actually vary during a given case and can be plotted against each other.

Dimensional constants: These are normally held constant during a given run. But, they may vary from case to case.

Pure constants: They have no dimensions, but, while performing the mathematical manipulation, they can arise.

- Primary Dimensions
    - Length (L)
    - Time (T)
    - Mass (M)
    - Temperature (q)
- For any relationship A=B
    - Units (A)=Units (B) called Dimensional Homogeneity

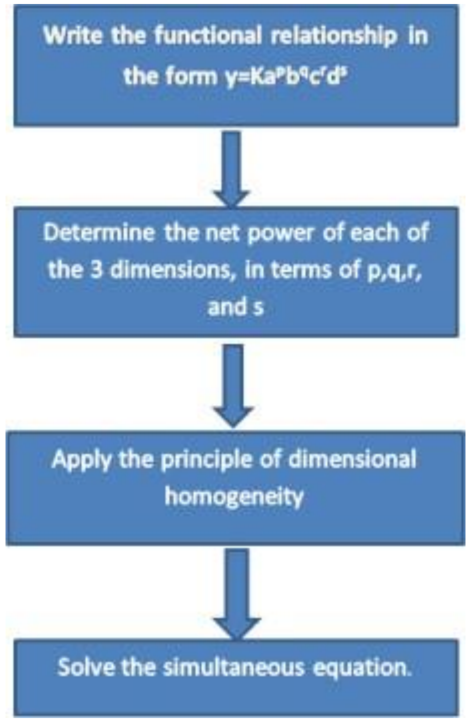
Dimensions of Derived Quantities

Table 1. Dimensions of common derived mechanical quantities are given in the following table.

	Quantity	Common Symbol(s)	Dimensions
Geometry	Area	A	L^2
	Volume	V	L^3
	Second moment of area	I	L^4
Kinematics	Velocity	U	LT^{-1}
	Acceleration	a	LT^{-2}
	Angle	θ	1 (i.e. dimensionless)
	Angular velocity	ω	T^{-1}
	Quantity of flow	Q	L^3T^{-1}
	Mass flow rate	$\dot{m}$	MT^{-1}
Dynamics	Force	F	MLT^{-2}
	Moment, torque	T	ML^2T^{-2}
	Energy, work, heat	E, W	ML^2T^{-2}
	Power	P	ML^2T^{-3}
	Pressure, stress	p, τ	$ML^{-1}T^{-2}$
Fluid properties	Density	ρ	ML^{-3}
	Viscosity	μ	$ML^{-1}T^{-1}$
	Kinematic viscosity	ν	L^2T^{-1}
	Surface tension	σ	MT^{-2}
	Thermal conductivity	k	$MLT^{-3}\Theta^{-1}$
	Specific heat	c_p, c_v	$L^2T^{-2}\Theta^{-1}$
	Bulk modulus	K	$ML^{-1}T^{-2}$

Rayleigh Method

A basic method to dimensional analysis method and can be simplified to yield dimensionless groups controlling the phenomenon. Flow chart below shows the procedures.



$$X = CX_1^aX_2^bX_3^c\cdots X_n^m$$

where C = dimensionless constant

$a, b, c, \ldots, m$ are arbitrary exponents.

X_n = independent variables

1.Example:

The velocity of propagation of a pressure wave through a liquid can be expected to depend on the elasticity of the liquid represented by the bulk modulus K, and its mass density ρ. Establish by D. A. the form of the possible relationship.

$$U = \text{velocity} = \frac{L}{T},\ \rho = \frac{M}{L^3},\ K = \frac{M}{LT^2}$$

$$U = c\rho^a K^b$$

$$\frac{L}{T} = \left(\frac{M}{L^3}\right)^a \left(\frac{M}{LT^2}\right)^b$$

For Mass, 0 = a + b
For Length, 1 = -3a – b
For Time, -1 = -2b

a = -b
1= 3b-b= 2b
b=1/2
so a = -1/2

$$U=C\sqrt{\frac{K}{\rho}}$$

Buckingham pi Theorem

The dimensional analysis for the experimental data of unknown flow problems leads to some non- dimensional parameters. These dimensionless products are frequently referred as pi terms. Based on the concept of dimensional homogeneity, these dimensionless parameters may be grouped and expressed in functional forms. This idea was explored by the famous scientist Edgar Buckingham (1867-1940) and the theorem is named accordingly.

Buckingham pi theorem, states that if an equation involving k variables is dimensionally homogeneous, then it can be reduced to a relationship among (k−r) independent dimensionless products, where r is the minimum number of reference dimensions required to describe the variable. For a physical system, involving k variables, the functional relation of variables can be written mathematically as,

$$y = f(x_1, x_2, x_3, \ldots\ldots x_k)$$

In the Equation., it should be ensured that the dimensions of the variables on the left side of the equation are equal to the dimensions of any term on the right side of equation. Now, it is possible to rearrange the above equation into a set of dimensionless products (pi terms), so that

$$\Pi_1 = \varphi(\Pi_2, \Pi_3 \ldots\ldots\ldots, \Pi_{k-r})$$

Here, $\varphi(\Pi_2, \Pi_3 \ldots\ldots\ldots, \Pi_{k-r})$ is a function of Π_2 through Π_{k-r}. The required number of *pi terms* is less than the number of original reference variables by r. These reference dimensions are usually the basic dimensions M, L and T (Mass, Length and Time).

Determination of pi Terms

Several methods can be used to form dimensionless products or pi terms that arise in dimensional analysis. But, there is a systematic procedure called method of repeating variables that allows in deciding the dimensionless and independent pi terms. For a given problem, following distinct steps are followed.

Step I: List out all the variables that are involved in the problem. The 'variable' is any quantity including dimensional and non-dimensional constants in a physical situation under investigation. Typically, these variables are those that are necessary to describe the "geometry" of the system (diameter, length etc.), to define fluid properties (density, viscosity etc.) and to indicate the external effects influencing the system (force, pressure etc.). All the variables must be independent in nature so as to minimize the number of variables required to describe the complete system.

Step II: Express each variable in terms of basic dimensions. Typically, for fluid mechanics problems, the basic dimensions will be either M, L and T or F, L and T. Dimensionally, these two sets are related through Newton's second law $(F = m.a)$ so that $F = MLT^{-2}$ e.g. $\rho = ML^{-3}$ or $\rho = FL^{-4}T^{2}$. It should be noted that these basic dimensions should not be mixed.

Step III: Decide the required number of *pi terms*. It can be determined by using *Buckingham pi theorem* which indicates that the number of *pi terms* is equal to $(k - r)$, where k is the number of variables in the problem (determined from Step I) and r is the number of reference dimensions required to describe these variables (determined from Step II).

Step IV: Amongst the original list of variables, select those variables that can be combined to form *pi terms*. These are called as *repeating variables*. The required number of *repeating variables is equal to the number of reference dimensions*. Each *repeating variable* must be dimensionally independent of the others, i.e. they cannot be combined themselves to form any dimensionless product. Since there is a possibility of repeating variables to appear in more than one *pi term*, so dependent variables should not be chosen as one of the repeating variable.

Step V: Essentially, the *pi terms* are formed by multiplying one of the non-repeating variables by the product of the repeating variables each raised to an exponent that will make the combination dimensionless. It usually takes the form of $x_i \, x_1^a \, x_2^b \, x_3^c$ where the exponents a, b and c are determined so that the combination is dimensionless.

Step VI: Repeat the 'Step V' for each of the remaining non-repeating variables. The resulting set of *pi terms* will correspond to the required number obtained from Step III.

Step VII: After obtaining the required number of *pi terms*, make sure that all the *pi terms* are dimensionless. It can be checked by simply substituting the basic dimension $(M, L \text{ and } T)$ of the variables into the *pi terms*.

Step VIII: Typically, the final form of relationship among the *pi terms* can be written in the form of Eq. (6.1.2) where, Π_1 would contain the dependent variable in the numerator. The actual functional relationship among *pi terms* is determined from experiment.

Illustration of Pi Theorem

Let us consider the following example to illustrate the procedure of determining the various steps in the pi theorem.

Example (Pressure drop in a pipe flow)

Consider a steady flow of an incompressible Newtonian fluid through a long, smooth walled, horizontal circular pipe. It is required to measure the pressure drop per unit length of the pipe and find the number of non-dimensional parameters involved in the problem. Also, it is desired to know the functional relation among these dimensionless parameters.

Step I: Let us express all the pertinent variables involved in the experimentation of pressure drop per unit length (Δp_l) of the pipe, in the following form;

$$\Delta p_l = f(D, \rho, \mu, V)$$

where, D is the pipe diameter, ρ is the fluid density, μ is the viscosity of the fluid and V is the mean velocity at which the fluid is flowing through the pipe.

Step II: Next step is to express all the variables in terms of basic dimensions i.e. M, L and T. It then follows that

$$\Delta p_l = ML^{-2}T^{-2};\; D = L;\; \rho = ML^{-3};\; \mu = ML^{-1}T^{-1};\; V = LT^{-1}$$

Step III: Apply *Buckingham theorem* to decide the number of *pi terms* required. There are five variables (including the dependent variable Δp_l) and three reference dimensions. Since, $k = 5$ and $r = 3$, only *two pi terms* are required for this problem.

Step IV: The repeating variables to form pi terms, need to be selected from the list D, ρ, μ and V. It is to be noted that the dependent variable should not be used as one of the repeating variable. Since, there are three reference dimensions involved, so we need to select three repeating variable. These repeating variables should be dimensionally independent, i.e. dimensionless product cannot be formed from this set. In this case, D, ρ and V may be chosen as the repeating variables.

Step V: Now, first *pi term* is formed between the dependent variable and the repeating variables. It is written as,

$$\Pi_1 = \Delta p_l D^a V^b \rho^c$$

Since, this combination need to be dimensionless, it follows that

$$\left(ML^{-2}T^{-2}\right)(L)^a\left(LT^{-1}\right)^b\left(ML^{-3}\right)^c = M^0L^0T^0$$

The exponents a, b and c must be determined by equating the exponents for each of the terms M, L and T i.e.

$$\begin{aligned}
&\text{For } M: \ 1+c=0\\
&\text{For } L: -2+a+b-3c=0\\
&\text{For } T: -2-b=0
\end{aligned}$$

The solution of this algebraic equations gives $a=1; b=-2; c=-1$. Therefore,

$$\Pi_1 = \frac{\Delta p_l\, D}{\rho V^2}$$

The process is repeated for remaining non-repeating variables with other additional variable (μ) so that,

$$\Pi_2 = \mu . D^d . V^e . \rho^f$$

Since, this combination need to be dimensionless, it follows that

$$\left(ML^{-1}T^{-1}\right)(L)^d\left(LT^{-1}\right)^e\left(ML^{-3}\right)^f = M^0L^0T^0$$

Equating the exponents,

$$\begin{aligned}
&\text{For } M: \ 1+f=0\\
&\text{For } L: -1+d+e-3f=0\\
&\text{For } T: -1-e=0
\end{aligned}$$

The solution of this algebraic equation gives $d=-1; e=-1; f=-1$. Therefore,

$$\Pi_2 = \frac{\mu}{\rho V D}$$

Step VI: Now, the correct numbers of *pi terms* are formed as determined in "Step III". In order to make sure about the dimensionality of *pi terms*, they are written as,

$$\Pi_1 = \frac{\Delta p_l\, D}{\rho V^2} = \frac{\left(ML^{-2}T^{-2}\right)(L)}{\left(ML^{-3}\right)\left(LT^{-1}\right)^2} = M^0L^0T^0$$

$$\Pi_2 = \frac{\mu}{\rho V D} = \frac{\left(ML^{-1}T^{-1}\right)(L)}{\left(ML^{-3}\right)\left(LT^{-1}\right)(L)} = M^0L^0T^0$$

Step VII: Finally, the result of dimensional analysis is expressed among the *pi terms* as,

$$\frac{D\Delta p_l}{\rho V^2} = \phi\left(\frac{\mu}{\rho V D}\right) = \phi\left(\frac{1}{\mathrm{Re}}\right)$$

It may be noted here that Re is the Reynolds number.

Remarks

- If the difference in the number of variables for a given problem and number of reference dimensions is equal to unity, then only *one Pi* term is required to describe the phenomena. Here, the functional relationship for the *one Pi* term is a constant quantity and it is determined from the experiment.

$$\Pi_1 = \text{Constant}$$

- The problems involving two *Pi* terms can be described such that

$$\Pi_1 = \phi(\Pi_2)$$

Here, the functional relationship among the variables can then be determined by varying Π_2 and measuring the corresponding values of Π_1.

Non Dimensional numbers in Fluid Dynamics

Forces encountered in flowing fluids include those due to inertia, viscosity, pressure, gravity, surface tension and compressibility. These forces can be written as follows;

$$\text{Inertia force}: m.a = \rho V \frac{dV}{dt} \propto \rho V^2 L^2$$

$$\text{Viscous force}: \tau A = \mu A \frac{du}{dy} \propto \mu V L$$

$$\text{Pressure force}: (\Delta p) A \propto (\Delta p) L^2$$

$$\text{Gravity force}: m g \propto g \rho L^3$$

$$\text{Surface tension force}: \sigma L$$

$$\text{Compressibility force}: E_v A \propto E_v L^2$$

The notations used in the Eq. are given in subsequent paragraph of this section. It may be noted that the ratio of any two forces will be dimensionless. Since, inertia forces are very important in fluid mechanics problems, the ratio of the inertia force to each of the other forces listed above leads to fundamental dimensionless groups.

Some of them are defined as given below;

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

Mathematically,

$$\mathrm{Re} = \frac{\rho V L}{\mu} = \frac{VL}{\nu}$$

where Vis the velocity of the flow, L is the characteristics length, and ρ,μ, ν are the density, dynamic viscosity and kinematic viscosity of the fluid respectively. If Re is very small, there is an indication that the viscous forces are dominant compared to inertia forces. Such types of flows are commonly referred to as "creeping/viscous flows". Conversely, for large Re , viscous forces are small compared to inertial effects and such flow problems are characterized as inviscid analysis. This number is also used to study the transition between the laminar and turbulent flow regimes.

Euler number (Eu) : In most of the aerodynamic model testing, the pressure data are usually

$$E_u = \frac{\Delta p}{\frac{1}{2}\rho V^2}$$

expressed mathematically as,

where Δ p is the difference in local pressure and free stream pressure,

V is the velocity of the flow, ρ is the density of the fluid. The denominator in Eq. is called "dynamic pressure". Eu is the ratio of pressure force to inertia force and many a times the pressure coefficient (c p) is a also common name which is defined by same manner. In the study of cavitations phenomena, similar expressions are used where, Δ p is the difference in liquid stream pressure and liquid- vapour pressure. This dimensional parameter is then called as **"cavitation number".**

Froude number (Fr) : It is interpreted as the ratio of inertia force to gravity force. Mathematically, it is written as,

$$F_r = \frac{V}{\sqrt{g.L}}$$

where Vis the velocity of the flow, Lis the characteristics length descriptive of the flow field and g is the acceleration due to gravity. This number is very much significant for flows with free surface effects such as in case of open-channel flow. In such types of flows, the characteristics length is the depth of water. rF less than unity indicates sub-critical flow and values greater than unity indicate super -critical flow. It is also used to study the flow of water around ships with resulting wave motion.

Weber number(We): It is defined as the ratio of the inertia force to surface tension force. Mathematically,

$$W_e = \frac{\rho V^2 L}{\sigma}$$

where Vis the velocity of the flow, L is the characteristics length descriptive of the flow field, ρ is the density of the fluid and σ is the surface tension force. This number is taken as an index of droplet formation and flow of thin film liquids in which there is an interface between two fluids.

Mach number(M): It is the key parameter that characterizes the compressibility effects in a fluid flow and is defined as the ratio of inertia force to compressibility force. Mathematically,

$$M = \frac{V}{c} = \frac{V}{\sqrt{\frac{dp}{d\rho}}} = \frac{V}{\sqrt{\frac{E_v}{\rho}}}$$

where V is the velocity of the flow, c is the local sonic speed, ρ is the density of the fluid and v E is the bulk modulus. Sometimes, **the square of the Mach number is called “Cauchy number” (C a) i.e.**

$$C_a = M^2 = \frac{\rho V^2}{E_v}$$

Both the numbers are predominantly used in problems in which fluid compressibility is important.

Modeling and Similitude

A "model" is a representation of a physical system which is used to predict the behavior of the system in some desired respect. The physical system for which the predictions are to be made is called "prototype". Usually, a model is smaller than the prototype so that laboratory experiments/studies can be conducted. It is less expensive to construct and operate. However, in certain situations, models are larger than the prototype e.g. study of the motion of blood cells whose sizes are of the order of micrometers. "Similitude" is the indication of a known relationship between a model and prototype. In other words, the model tests must yield data that can be scaled to obtain the similar parameters for the prototype.

Flow Similarity

In order to achieve similarity between model and prototype behavior, all the corresponding pi terms must be equated to satisfy the following conditions.

Geometric similarity: A model and prototype are geometric similar if and only if all body dimensions in all three coordinates have the same linear

-scale ratio. In order to have geometric similarity between the model and prototype, the model and the prototype should be of the same shape, all the linear dimensions of the model can be related to corresponding dimensions of the prototype by a constant scale factor. Usually, one or more of these pi terms will involve ratios of important lengths, which are purely geometrical in nature.

Kinematic similarity: The motions of two systems are kinematically similar if homogeneous particles lie at same points at same times. In a specific sense, the velocities at corresponding points are in the same direction (i.e. same streamline patterns) and are related in magnitude by a constant scale factor.

Dynamic similarity: When two flows have force distributions such that identical types of forces are parallel and are related in magnitude by a constant scale factor at all corresponding points, then the flows are dynamic similar. For a model and prototype, the dynamic similarity exists, when both of them have same length -scale ratio, time - scale ratio and force- scale (or mass - scale ratio). In order to have complete similarity between the model and prototype, all the similarity flow conditions must be maintained. This will automatically follow if all the important variables are included in the dimensional analysis and if all the similarity requirements based on the resulting pi terms are satisfied. For example, in compressible flows, the model and prototype should have same Reynolds number,Mach number and specific heat ratio etc. If the flow is incompressible (without free surface), then same Reynolds numbers for model and prototype can satisfy the complete similarity.

Model scales

In a given problem, if there are two length variables l_1 and l_2, the resulting requirement based on the pi terms obtained from these variables is,

$$\frac{l_{1m}}{l_1} = \frac{l_{2m}}{l_2} = \lambda_l$$

This ratio is defined as the "length scale". For true models, there will be only one length scale and all lengths are fixed in accordance with this scale. There are other 'model scales' such as velocity scale $\left(\frac{V_m}{V} = \lambda_v\right)$, density scale $\left(\frac{\rho_m}{\rho} = \lambda_\rho\right)$, viscosity scale $\left(\frac{\mu_m}{\mu} = \lambda_\mu\right)$ etc. Each of these scales needs to be defined for a given problem.

Model laws or similarity laws

- For the dynamic similarity between the model and the prototype , the ratio of the corresponding forces acting at the corresponding points in the prototype and model should be same. The ratio of the forces are dimensionless numbers.
- For the dynamic similarity, the dimensionless numbers should be same for the model and proto type.
- But it is quite difficult to satisfy the condition that all the dimensionless numbers to be same between model and prototype
- Hence the models are designed based on the ratio of the forces, which are dominating in the phenomenon.
- The laws on which the models are designed for dynamic similarity are known as model laws or laws of similarity.

The following are the laws

1. **Reynolds model law**

   Pipe flow

   Resistance experienced by sub-marines, airplanes, fully immersed bodies etc.

2. **Froudes model law**

   Free surface flows such as flow over spillways, weirs, sluices, channels etc……

   Flow of jet from an orifice or nozzle,

   Where waves are likely to be formed on surface.

   Where fluids of different densities flow over one another.

3. **Eulers model law**

   Euler’s model law is applied for fluid flow problems where flow is taking place in a closed pipe in which case turbulence is fully developed so that viscous forces are negligible and gravity force and surface tension force is absent.

4. **Weber model law**

   Weber model law is applied in following cases:

   Capillary rise in narrow passages

Capillary movement of water in soil

Capillary waves in channels,

Flow over weirs for small heads.

5. **Mach model law**

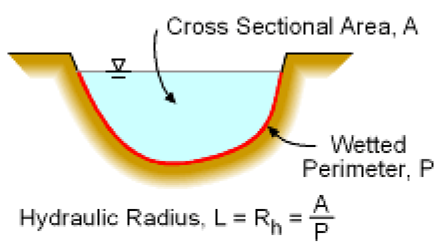
The Mach number is often used to classify the top speed of a fighter or passenger jet.

REYNOLD'S NUMBER

The Reynolds number perhaps is the most common dimensionless parameter used in fluid mechanics. It is defined as

- $Re = \rho VL/\mu$

where ρ is the density, V is the velocity, L is the characteristic length, and μ is the viscosity. The L term is different for each flow type. For example, for a pipe, L is the diameter of the pipe. For open channel flow, the hydraulic radius, R_h (see diagram) is commonly used.



Hydraulic Radius (Used with Open Channel Flow)

Reynolds number is the ratio of inertia force and viscous force, and hence fluid flow problems where viscous forces alone are predominant. A small Reynolds number implies that the viscous effects are important, while the inertial effects are dominant when the Reynolds number is large. The models are designed for dynamic similarity on Reynolds law, which states that the Reynolds number for the model must be equal to the Reynolds number for the prototype. According to Reynolds model law, models are based on Reynolds number. Models based on Reynolds number includes:

Let

V_m= Velocity of fluid in model,

ρ_m = Density of fluid in model,

L_m = Length or linear dimension of the model,

μ_m = Viscosity of fluid in model,

andV_p , ρ_p , L_p , and μ_p are the corresponding values of velocity, density, linear dimension and viscosity of fluid in prototype. Then according to Reynolds model law,

$$[Re]_m = [Re]_p \text{ or } \frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

$$\text{or } \frac{\rho_p . V_p . L_p}{\rho_m . V_m . L_m} \times \frac{1}{\frac{\mu_p}{\mu_m}} = 1 \text{ or } \frac{\rho_r . V_r . L_r}{\mu_r} = 1$$

$$\text{where } \rho_r = \frac{\rho p}{\rho m}, \ V_r = \frac{Vp}{Vm}, \ L_r = \frac{Lp}{Lm}, \frac{\mu p}{\mu m} = \mu_r$$

And also ρ_r,V_r, $L_r$and = μ_r are called the scale ratios for density, velocity, linear dimension and viscosity. The scale rations for time, acceleration, force and discharge for Reynolds model law are obtained as

$$t_r = \text{time scale ratio} = \frac{Lr}{Vr} \qquad \{V = \frac{L}{t}\}$$

$$a_r = \text{Acceleration scale ratio} = \frac{Vr}{tr}$$

$$F_r = \text{Force scale ratio} = (\text{Mass} \times \text{Acceleration})_r$$

$$= m_r \times a_r = \rho_r A_r V_r \times a_r \qquad \{A_r = \text{Area ratio}\}$$

$$= \rho_r L_r^2 V_r \times a_r$$

$$Q_r = \text{Discharge scale ratio} = (\rho AV)_r$$

$$= \rho_r A_r V_r = \rho_r : L_r^2 : V_r$$

Problem 1:

A pipe of diameter 1.5 m is required to transport an oil of sp.gr. 0.90 and viscosity 3×10^{-2} poise at 3000litre/s. Tests were conducted on a 15cm diameter pipe using water at 20. Find the velocity and rate of flow in the model. Viscosity of water at 20 = 0.01poise.

Solution.

Given:

Diameter of prototype, $D_p = 1.5$ m

Viscosity of fluid, $\mu_p = 3\times10^{-2}$ poise

Q for prototype, $Q_p = 3000$lit/s $= 3.0\ m^3/s$

S.p.g of oil, $S_p = 0.9 \Rightarrow$ Density of oil, $\rho_p = S_p \times 1000 = 900 kg/m^3$

Dia. of the model, $D_m = 15cm = 0.15m$

Viscosity of water at 20, $\mu_m = 0.01$poise $= 1\times 10^{-2}$ poise

Density of water, $\rho_m = 1000 kg/m^3$

For pipe flow, the dynamic similarity will be obtained if the Reynolds number in the model and prototype are equal

Hence using equation, $\frac{\rho m Vm Dm}{\mu m} = \frac{\rho p\ Vp Dp}{\mu p}$ {For pipe, linear dimension is D}

$$\frac{Vm}{Vp} = \frac{\rho p}{\rho m} \cdot \frac{Dp}{Dm} \cdot \frac{\mu m}{\mu p}$$

$$\therefore \quad = \frac{900}{1000} \times \frac{1.5}{0.15} \times \frac{1\times10^{-2}}{3\times10^{-2}} = \frac{900}{1000} \times 10 \times \frac{1}{3} = 3.0$$

But $$V_p = \frac{Rate\ of\ flow\ in\ prototype}{Area\ of\ prototype} = \frac{3.0}{\frac{\pi}{4}(Dp)^2} = \frac{3.0}{\frac{\pi}{4}(1.5)^2} = \frac{3.0\times4}{\pi\times2.25} = \frac{1.697m}{s}$$

$$\therefore \quad V_m = 3.0 \times V_p = 3.0 \times 1.697 = \boldsymbol{5.091\ m/s.\ Ans.}$$

Rate of flow through model, $Q_m = A_m \times V_m = \frac{\pi}{4}(Dm)^2 \times V_m = \frac{\pi}{4}(0.15)^2 \times 5.091 m^3/s$

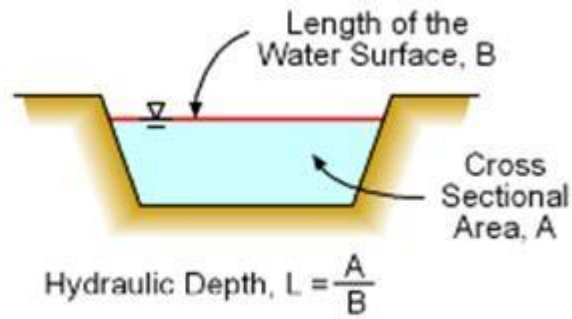
$$= 0.0899 m^3/s = 0.0899\times1000 lit/s = \boldsymbol{89.9 lit/s. Ans.}$$

Froude number

The Froude number is an important dimensionless parameter in the study of open-channel flow, and it is given by

$Fr = V / (gL)^{0.5}$

where V is the average velocity, L is the characteristic length associated with the depth (hydraulic depth for open channel flow), and g is the gravitational acceleration. For rectangular cross sections, the hydraulic depth is the water depth.



(Hydraulic depth)

Physically, the Froude number represents the ratio of inertial forces to gravitational forces.


(Open channel flow)

As discussed in the open-channel sections, open-channel flow can be classified according to the Froude number in the following manner:
(a) Fr< 1: subcritical (tranquil) flow
(b) Fr = 1: critical flow

(c) Fr> 1: supercritical (rapid) flow

It is also common to write Fr as V/c, where c is the wave celerity, c (speed of a wave in the fluid). This form is similar to the Mach Number in air. For subcritical flow ($V < c$), the waves created by any surface disturbances (e.g., throwing a stone in the water) at the downstream can travel upstream. On the other hand, for supercritical flow ($V > c$), all surface disturbances will be swept downstream. The wave will remain stationary for critical flow ($V = c$).

According to the Froude Model Law, the Froude numbers of the model and prototype should be equal to each other. Froude model law is the law in which the models are based on Froude number which means for dynamic similarity between the model and prototype, the Froude numbers for both of them should be equal. Froude model law is applicable when the gravity force is only predominant force which controls the flow in addition to the force of inertia. Froude model law is applied in the following fluid flow problems:

Let

V_m= velocity of fluid in model,

L_m = Linear dimension or length of model,

g_m= Acceleration due to gravity at a place where model is tested.

andV_p , $L_p$and$g_p$are the corresponding values of the velocity, length and acceleration due to gravity for the prototype. Then according to Froude model law,

$$(F_r)_{model} = (F_r)_{prototype} \text{ or } \frac{Vm}{\sqrt{gmLm}} = \frac{Vp}{\sqrt{gp\ Lp}} \qquad \cdots\cdots(1)$$

If the tests on the model are performed on the same place where prototype is to operate, then

$g_m = g_p$ and equation (1) becomes as

$$\frac{Vm}{\sqrt{Lm}} = \frac{Vp}{\sqrt{Lp}} \qquad \cdots\cdots (2)$$

$$\frac{Vm}{Vp} \times \frac{1}{\sqrt{\frac{Lm}{Lp}}} = 1$$

$$\frac{Vp}{Vm} = \sqrt{\frac{Lp}{Lm}} = \sqrt{Lr} \qquad \{\therefore \frac{Lp}{Lm} = Lr\}$$

where Lr = scale ratio for length

$\frac{Vp}{Vm} = V_r$ = Scale ratio for velocity.

$$\therefore \frac{Vp}{Vm} = V_r = \sqrt{Lr} \qquad \cdots\cdots (3)$$

Scale ratios for various physical quantities based on Froude model law are:

(a) **Scale ratio for time**

As time $= \frac{Length}{Velocity}$,

then ratio of time for prototype and model is

$$T_r = \frac{Tp}{Tm} = \frac{(\frac{L}{V})_p}{(\frac{L}{V})_m} = \frac{\frac{L_p}{V_p}}{\frac{L_m}{V_m}} = \frac{L_p}{L_m} \times \frac{V_m}{V_p} = L_r \times \frac{1}{\sqrt{L_r}} \qquad \{\therefore \frac{Vp}{Vm} = \sqrt{L_r}\ \}$$

$$= \sqrt{L_r} \qquad \cdots\cdots (4)$$

(b) Scale ratio for acceleration

Acceleration $= \frac{V}{T}$

$$\therefore \qquad a_r = \frac{a_p}{a_m} = \frac{(\frac{V}{T})_p}{(\frac{V}{T})_m} = \frac{V_p}{Tp} \times \frac{Tm}{V_m} = \frac{V_p}{V_m} \times \frac{Tm}{Tp}$$

$$= \sqrt{L_r} \times \frac{1}{\sqrt{L_r}} \qquad \{\frac{Vp}{Vm} = \sqrt{L_r}, \frac{Tp}{Tm} = \sqrt{L_r}\}$$

$$= 1 \qquad \cdots\cdots (5)$$

(c)Scale ratio for discharge

$$Q = A \times V = L^2 \times \frac{L}{T} = \frac{L^3}{T}$$

$$\therefore \qquad Q_r = \frac{Q_p}{Q_m} = \frac{(\frac{L^3}{T})_p}{(\frac{L^3}{T})_m} = (\frac{L_p}{L_m})^3 \times \left(\frac{T_m}{T_p}\right) = L_r{}^3 \times \frac{1}{\sqrt{L_r}} = L_r{}^{2.5} \quad \cdots\cdots (6)$$

(d)Scale ratio for force

$$\text{As force} = \text{Mass} \times \text{Acceleration} = \rho L^3 \times \frac{V}{T} = \rho L^2 . \frac{L}{T} . V = \rho L^2 V^2$$

$$\therefore \text{ Ratio for force, } F_r = \frac{F_p}{F_m} = \frac{\rho_p L_p{}^2 V_p{}^2}{\rho_m L_m{}^2 V_m{}^2} = \frac{\rho_p}{\rho_m} \times (\frac{L_p}{L_m})^2 \times (\frac{V_p}{V_m})^2$$

If the fluid used in model and prototype is same, then

$$\frac{\rho_p}{\rho_m} = 1 \quad \text{or} \quad \rho_p = \rho_m$$

$$\text{And hence} \qquad F_r = (\frac{L_p}{L_m})^2 \times (\frac{V_p}{V_m})^2 = L_r{}^2 \times (\sqrt{L_r})^2 = L_r{}^2 . L_r = L_r{}^3 \cdots\cdots (7)$$

(e)Scale ratio for pressure intensity

$$\text{As} \qquad p = \frac{Force}{Area} = \frac{\rho L^2 V^2}{L^2} = \rho V^2$$

$$\therefore \text{ Pressure ratio, } p_r = \frac{p_p}{p_m} = \frac{\rho_p V_p{}^2}{\rho_m V_m{}^2}$$

If fluid is the same, then $p_p = p_m$

$$\therefore \qquad p_r = \frac{V_p{}^2}{V_m{}^2} = \left(\frac{V_p}{V_m}\right)^2 = L_r \qquad \cdots\cdots (8)$$

(f)Scale ratio for work, energy, torque, moment etc.

$$\text{Torque} = \text{Force} \times \text{Distance} = F \times L$$

$$\therefore \quad \text{Torque ratio,} \quad T_r^{*} = \frac{T_p^{*}}{T_m^{*}} = \frac{(F\times L)_p}{(F\times L)_m} = F_r \times L_r = L_r^{3} \times L_r = L_r^{4} \quad \cdots\cdots (9)$$

(g)Scale ratio for power

As $\qquad \text{Power} = \text{Work per unit time}$

$$= \frac{F\times L}{T}$$

$$\therefore \quad \text{Power ratio,} \; P_r = \frac{P_p}{P_m} = \frac{\frac{F_p\times L_p}{T_p}}{\frac{F_m\times L_m}{T_m}} = \frac{F_p}{F_m} \times \frac{L_p}{L_m} \times \frac{1}{\frac{T_p}{T_m}}$$

$$= F_r . L_r . \frac{1}{T_r} = L_r^{3} . L_r . \frac{1}{\sqrt{L_r}} = L_r^{3.5} \quad \cdots\cdots (10)$$

Problem 2

A ship model of scale is towed through sea water at a speed of 1m/s. A force of 2N is required to tow the model. Determine the speed of ship and the propulsive force on the ship, if prototype is subjected to wave resistance only.

Solution:

Given: -Scale ratio of length, $L_r = 50$

-Speed of model, $\quad V_m = \frac{1m}{s}$

-Force required for model, $F_m = 2N$

-Let the speed of ship $\quad = V_p$

As prototype is subjected to wave resistance only for dynamic similarity, the Froude number should be same for model and prototype. Hence for velocity ratio, for Froude model law using equation(3), we have

$$\frac{Vp}{Vm} = \sqrt{Lr} = \sqrt{50}$$

$$V_p = \sqrt{50} \times V_m$$

$$= \sqrt{50} \times 1$$

$$= 7.071 \text{ m/s. Ans.}$$

Force scale ratio is given by equation (3),

$$F_p = \frac{F_p}{F_m} = {L_r}^3$$

$$F_p = F_m \times {L_r}^3 = 2 \times (50)^3 = 250000\text{N. Ans.}$$

Mach's Number (M)

Mach's number is defined as the square root of the ratio of the inertia force of a flowing fluid to the elastic force. Mathematically, it is defined as

$$M = \sqrt{\frac{Inertia\ force}{Elastic\ force}} = \sqrt{\frac{F_i}{F_e}}$$

Where $F_i = \rho A V^2$

And F_e = Elastic force = Elastic stress × Area

$= K \times A = K \times L^2$ $\{K = \text{elastic stress}\}$

$$M = \sqrt{\frac{\rho A V^2}{K \times L^2}} = \sqrt{\frac{\rho \times L^2 \times V^2}{K \times L^2}} = \sqrt{\frac{V^2}{K/\rho}} = \frac{V}{\sqrt{K/\rho}}$$

But $\sqrt{\frac{K}{\rho}} = c = velocity\ of\ sound\ in\ the\ fluid$

$$M = \frac{V}{c}$$

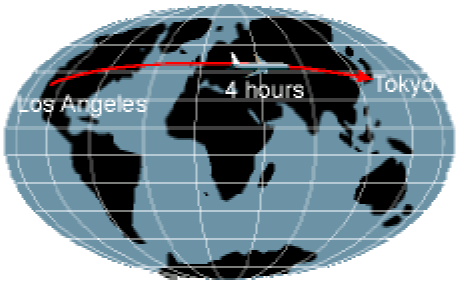
Mach Number (Ma): For high speed flows in some fluids, density is highly dependent on the pressure, and the compressibility effects become important. The Mach number is used to indicate if a flow is incompressible or compressible, and it is given by

Ma = V/c

where c is the speed of sound (343 m/s at 20°C) and V is the fluid velocity. The Mach number represents the ratio of inertia forces to compressibility forces. Flow can be characterized using the Mach number as folllows:
(a) Ma ≤ 0.3: incompressible
(b) 0.3 < Ma < 1.0: compressible subsonic flow
(c) Ma ≥ 1.0: compresible supersonic flow

The Mach number is often used to classify the top speed of a fighter or passenger jet. For example, the B-2 bomber shown in the picture is capable of reaching high subsonic speed. The Concorde of British Airways is a supersonic passenger jet, which cruises at Mach 2, and it takes only approximately 4 hours from Los Angeles to Tokyo.

B-2 Spirit Bomber: High Subsonic Speed



Concorde from British Airways:
Supersonic Speed (Mach 2)

Mach model law is the law in which models are designed on Mach number, which is the ratio of the square root of inertia force to elastic force of a fluid. Hence where the forces due to elastic compression prominent in addition to inertia force, the dynamic similarity between the model and its prototype is obtained by equating the Mach number of the model and its prototype.

Euler's Number (Eu)

$$E_u = \sqrt{\frac{F_i}{F_p}} = \sqrt{\frac{\rho A V^2}{p \times A}} = \sqrt{\frac{V^2}{p/\rho}} = \frac{V}{\sqrt{p/\rho}}$$

Where F_p = Intensity of pressure × Area = p×A

And $F_i = \rho A V^2$

According to the Euler's Model Law, the models are designed on Euler's number which means for dynamic similarity between the model and prototype, the Euler number for model and prototype should be equal. Euler's model law is applicable when the pressure forces are alone predominant in addition to the inertia force. According to this law:

$$(E_u)_{\text{model}} = (E_u)_{\text{prototype}}$$

V_m = velocity of fluid in model,

p_m = pressure of fluid in model,

ρ_m = Density of fluid in model,

V_p p_p ρ_p = Corresponding values in prototype, then

$$(E_u)_{\text{model}} = (E_u)_{\text{prototype}}$$

$$\frac{V_m}{\sqrt{p_m/\rho_m}} = \frac{V_p}{\sqrt{p_p/\rho_p}}$$

If fluid is same in model and prototype, then $\rho_m = \rho_p$

$$\frac{V_m}{\sqrt{p_m}} = \frac{V_p}{\sqrt{p_p}}$$

Euler's model law is applied for fluid flow problems where flow is taking place in a closed pipe in which case turbulence is fully developed so that viscous forces are negligible and gravity force and surface tension force is absent.

Weber Number

Weber Number (We): The dimensionless parameter associated with surface tension effects is the Weber number, and it is defined as

$We = \rho V^2 L/\sigma$

where σ is the surface tension. The Weber number denotes the ratio of the inertial forces to surface tension forces. The Weber number becomes an important parameter when dealing with applications involve two fluid interfaces such as the flow of thin films of liquid and bubble formation. Weber model law is the law in which models are based on Weber's number, which is the ratio of the square root of inertia force to surface tension force. Hence where surface tension effects predominate in addition to inertia force, the dynamic similarity between the model and prototype is obtained by equating the Weber number of the model and its prototype. Hence according to this law:

$$(W_e)_{\text{model}} = (W_e)_{\text{prototype}}$$

If V_m = Velocity of fluid in model

σ_m = Surface tension force in model

ρ_m = Density of fluid in model

L_m = Length of surface in model,

And $V_p, \sigma_p, \rho_p, L_p$ = Corresponding values of fluid in prototype.

Then according to Weber law, we have

$$\frac{V_m}{\sqrt{\sigma_m / \rho_m L_m}} = \frac{V_p}{\sqrt{\sigma_p / \rho_p L_p}}$$

Weber model law is applied in following cases:

1. Capillary rise in narrow passages
2. Capillary movement of water in soil
3. Capillary waves in channels,
4. Flow over weirs for small heads.