HYDRAULICS

A fluid is defined as a substance that deforms continuously under the action of a shear stress. However the magnitude of the shear stress is small.

By contrast a solid deforms when a constant shear stress is applied, but its deformation does not continue with increasing time. In short, a fluid continues in motion under the application of a shear stress and cannot sustain any shear stress when at rest.

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

PROPERTIES OF FLUIDS:-

Some of the basic properties of fluids are discussed below.

Density :- The density of a substance is its mass per unit volume. In fluid mechanic, it is expressed in three different ways:-

(i) **Mass density :-** It is the mass of the fluid per unit volume and is denoted by 'p'.

Unit- kg/m^3

Dimension:- $\left[ML^{-3}\right]$

Typical values :-

Water - 1000 kg/m^3

Air :- 1.23 kg/m^3 at (STP)

(ii) **Specific Weight(W):-** As we express a mass 'M' has a weight "W = Mg". The specific weight of the fluid can be defined similarly as its weight per unit volume.

$$W = Pg$$

Unit :-N/m^3

Dimension:-$\left[ML^{-2}T^{-2}\right]$

Typical values :- Water - 9.810 N/m^3

Air ;- 12.07 N/m^3(STP)

(iii) **Relative density (Specific gravity), S:-** Specific gravity is the ratio of fluid density(Specific weight) with a standard refrence bluid. For liquids water at 4°C is considered as standard fluid. (At 4°C water has maximum density).

$$S_{\lim it} = \frac{P_{\text{liquid}}}{P_{\text{water}} at 4^\circ C}$$

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

$$S_{gas} = \frac{P_{gas}}{P_{gas} \text{ at STP}}$$

Units:- Pure number having no units.

Dimension:- $M^\circ L^\circ T^\circ$

Typical value:- Mercury - 13.6

Water - 1

(iv) **Specific Volume (v):-** Specific volume of a fluid means volume per unit mass i.e, the reciprocal of mass density.

$$V_s = \frac{1}{P}$$

units:- m^3/kg

Dimension:- $\left[M^{-1}L^3\right]$

Typical value :- water :- $10^{-3} m^3/kg$

Air:- 1.23×10^{-3} m^3/kg.

(v) **Viscosity:-** Viscosity is a fluid property, which determines the relationship between the fluid strain rate and the applied shear stress. It can be noted that in fluid flows, shear strain rate is considered not shear strain as commonly used in solid mechanics. Viscosity can be inferred as a quantative measure of a fluid resistance to the flow. For example moving an object through air requires very less force compared to water. This means that air has low viscosity than water.

let us consider a fluid element placed between two infinite plates. The upper plate move at a constant velocity $S\mu$ under the action of constant shear force SF. The shear stress, 't' is expressed as:- T $\lim_{SA\ 0} \frac{SF}{SA}$ $\frac{dF}{dA}$

Where SA is the area of contact of the fluid element with the top plate. Under the action of shear force, the fluid element is deformed from position ABCD at time't' to position ABCD at time 't' to position AB'CD' at time T + St. The shear strain rate is given by:

Shear strain rate $= \lim_{St \to 0} \frac{Sa}{St} = \frac{da}{dt}$

Where 'Sa' is the angulars deformation.
In terms of geometry of the figure,

small Sa, tan Sa = $\frac{S\mu\, St}{Sy}$

Therefore, $\frac{Sa}{St} = \frac{S\mu}{Sy}$ ...(5)

The limit of both side of the equality gives

$$\frac{da}{dt} = \frac{d\mu}{dy}$$

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

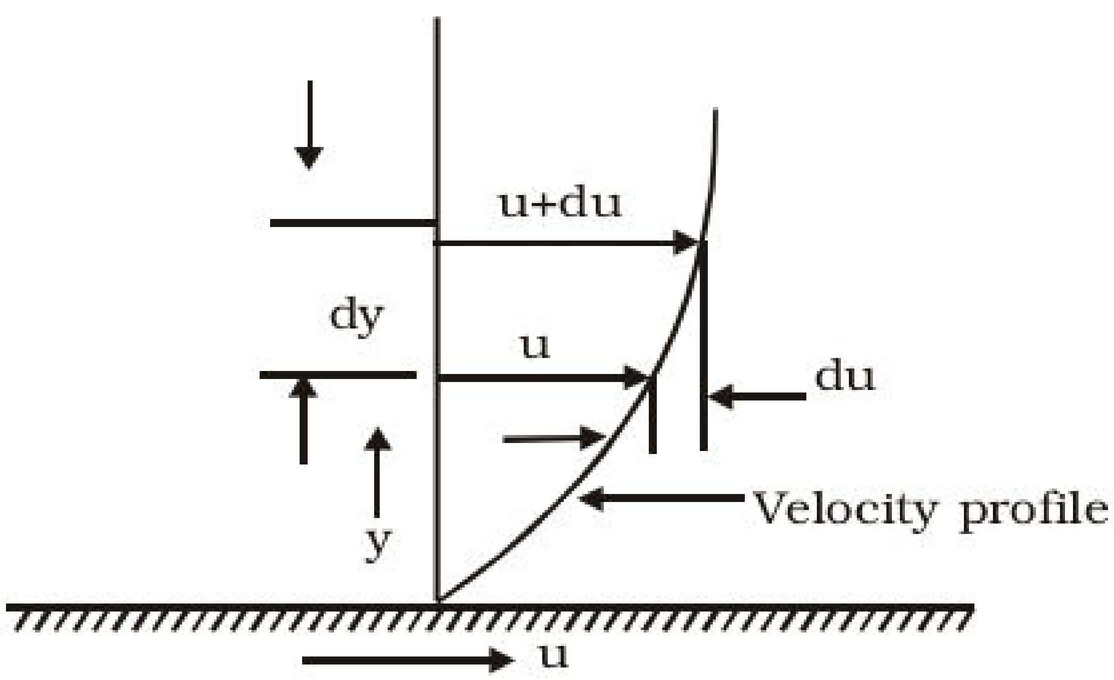


Fig:- Velocity variation near a solid boundary.

(Vi) **Newton's Viscosity Law**:-

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

$ta = \frac{da}{dt}$

Substituting the relation gives in equation (5)

$t_a = \frac{da}{dt}$ ...(6)

Introducing the constant of proportionality

$t_a = \frac{d\mu}{dy}$ ...(7)

Where 'μ' is called absolute or dynamic viscosity. Dimensions and units for "μ" are $[ML^{-1}T^{-1}]$ and N-s/m^2 respectively.[In the absolute metric system basic unit of Co-efficient of viscosity is called poise. 1 Poise =1N-S/m^3].

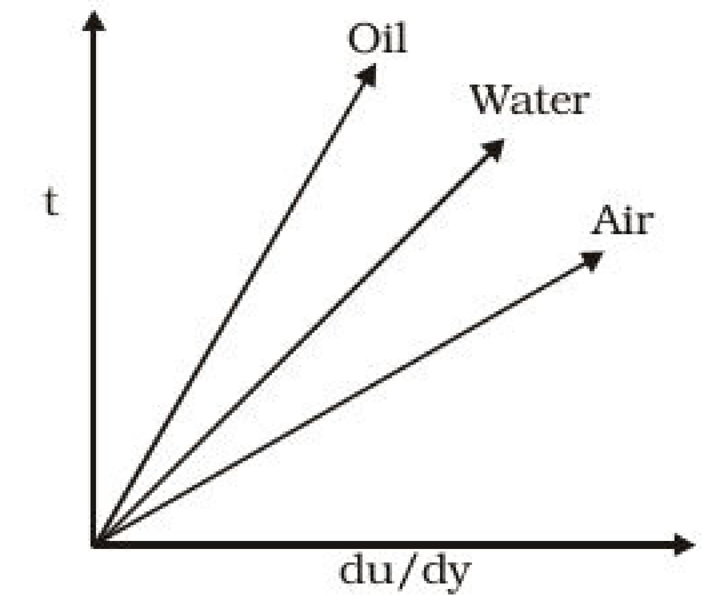


Fig:- Relationship between shear stress and velocity gradient of newtonian fluids

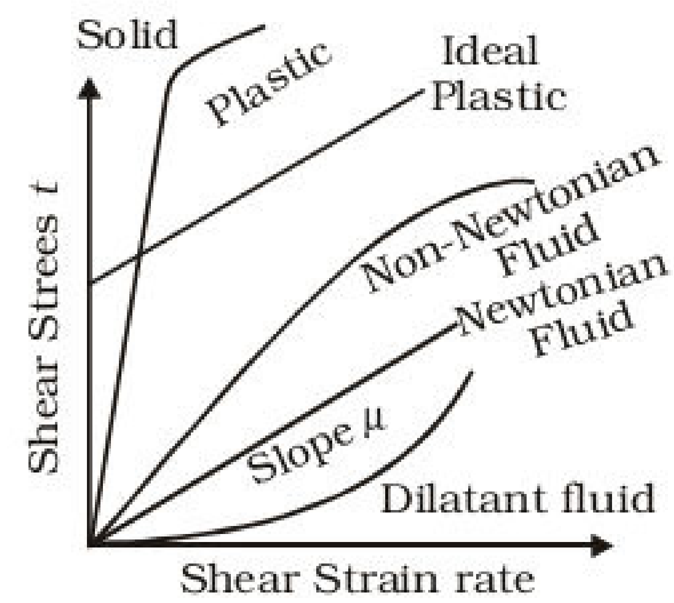
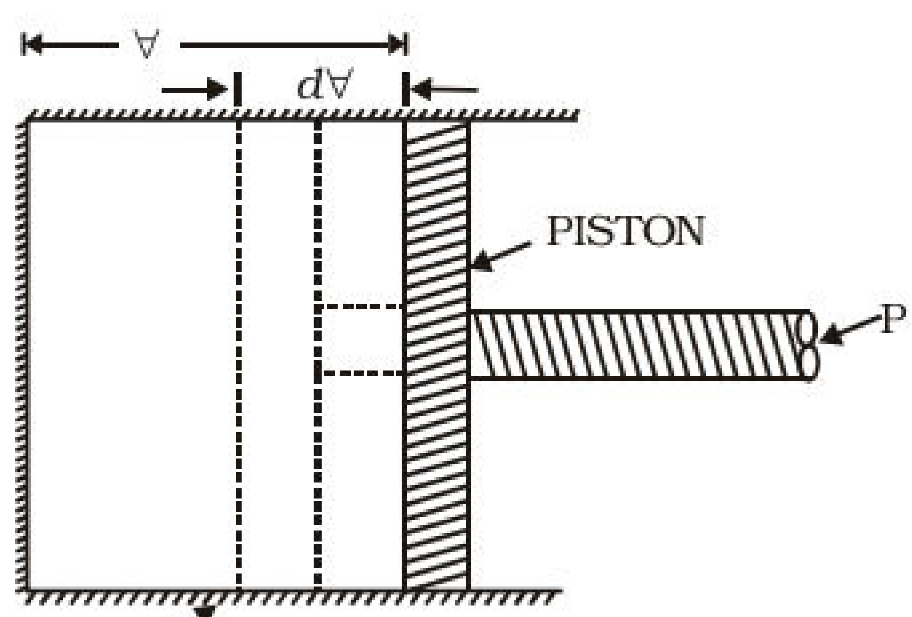


Fig:- Relationship between shear stress and shear strain rate of different fluid.

(vii) **Compressibility and Bulk modulus**:-

Compressibility is the reciprocal of the bulk modulus of elasticity, k which is defined as the ratio of compressive stress to volumetric strain.



Consider a cylinder fitted with a piston as shown in fig.

Let ∀ = Volume of a gas enclosed in the cylinder.

P = Pressure of gas when volume is ∀

Let the pressure is increased to (p + dp). The volume of gas decreases from ∀ to ∀ - d∀

Then, increase in pressure = dp kgf /m^2

Decrease in volume = $d\forall$

$\therefore$ Volumetric strain = $-d\forall / \forall$

Negative sign means the volume decreases with increase of pressure.

$\therefore$ Bulk modulus, $K = \dfrac{\text{Increase of pressure}}{\text{Volumetric Strain}}$

$$\frac{dp}{\frac{-d\forall}{\forall}} = \frac{-dp}{d\forall}\forall \quad ...(1)$$

Compressibility is given by = $\dfrac{1}{K}$ ...(2)

(viii)**Surface Tension and Capillarity**:-

Surface tension is defined as the tensile force acting on the surface of a liquid in contact with a gas or on the surface between two immiscible liquids such that the contact surface behaves like a membrane under tension. The magnitude of this force per unit length of the free surface will have the same value as the surface energy per unit area. It is denoted by Greek letter σ (called sigma). In MKS units it is expressed as $\dfrac{kg}{s^2}$ while in SI units as N/m.

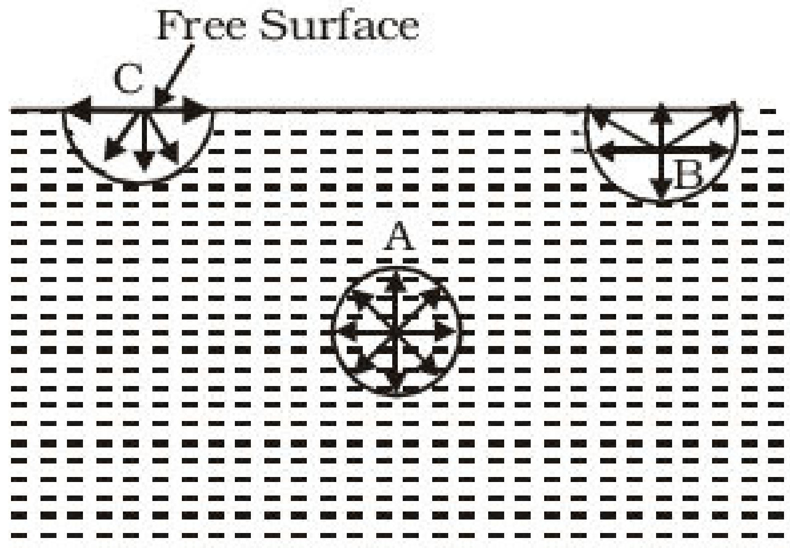


Fig:- Surface tension

Capillarity:- Capillarity is defined as a phenomenon of rise or fall of a liquid surface in a small tube relative to the adjacent general level of liquid when the tube is held vertically in the liquid. The rise of liquid surface is known as capillary rise while the fall of the liquid surface is known as capillary depression. It is expressed in terms of cm or mm of liquid. Its value depends upon the specific weight of the liquid, diameter of the capillary tube and surface tension of the liquid.

Expression for capillary Rise:-

Consider a glass tube of small diameter 'd' opened at both ends and is inserted in a liquid, say water. The liquid will rise in the tube above the level of the liquid.

Let h = height of liquid in tube. Under a state of equilibrium the weight of liquid of height 'h' is balanced by the force at the surface of the liquid in the tube. But the force at the surface of the liquid in the tube is due to surface tension.

Let σ = Surface tension of liquid

θ = angle of contact between liquid and glass tube.

The weight of liquid of height 'h' in the tube

= (Area of tube × h) × p × g

$$= \frac{\Pi}{4}d^2 \times h \times p \times g \qquad ...(1)$$

where p = Density of liquid.

Vertical component of the surface tensile force

= $(\sigma \times \text{Circumference}) \times \cos\theta$

$$= \sigma \times \Pi d \times \cos\theta \qquad ...(2)$$

For equilibrium, equating (1) & (2), we get

$$\frac{\Pi}{4}d^2 \times h \times p \times g = \sigma \times \Pi d \times \cos\theta$$

$$\text{or} \quad h = \frac{\sigma \times \Pi d \times \cos\theta}{\frac{\Pi}{4}d^2 \times p \times g} = \frac{4\sigma\cos\theta}{p \times g \times d} \qquad ...(3)$$

The value of "θ" between water and clean glass tube is approximately equal to zero hence, $\cos\theta$ is equal to unity. The rise of water is given by:-

$$h = \frac{4\sigma}{p \times g \times d} \qquad ...(4)$$

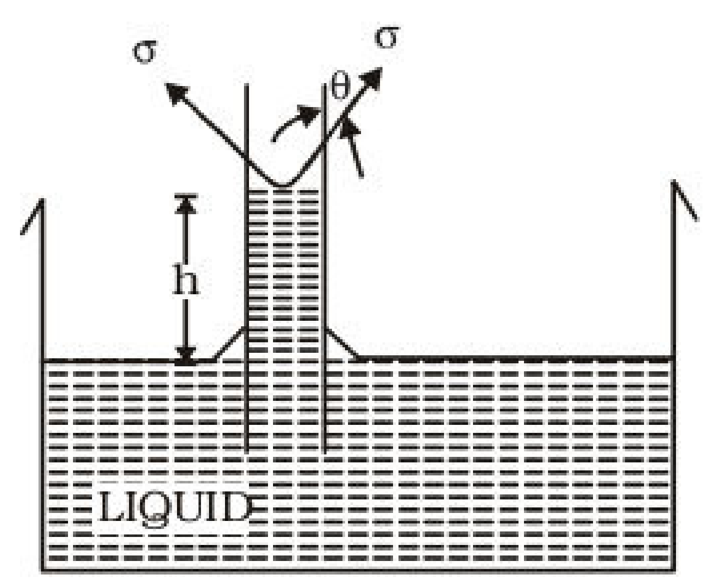


Fig:-Capillary rise.

(ix) **Vapour pressure and cavitation**:-

A change from the liquid state to the gaseous state is known as vaporization. The vaporization which depends upon the prevailing pressure and temperature condition Which occur because of continuous escaping of the molecules through the free liquid surface.

Consider a liquid (say water) which is confined in a closed vessel. Let the temperature of liquid is 20°C and pressure is in atm. This liquid will vaporise at 100°C. When Vaporization takes place, the molecules escapes from the free surface of liquid. These vapour molecules get accumulated in the space between the free liquid surface and top of vessel. These accumulated vapours exert a pressure on the liquid surface. The pressure exerted is known as vapour pressure of liquid or this is the pressure at which the liquid is converted into vapour.

Now Consider a flowing liquid in a system. If the pressure at any point in this flowing liquid becomes equal to or less than the vapour pressure, then the vaporization of the liquid starts. The bubble of these vapours are carried by the flowing liquid into the region of high pressure where they collapse, giving rise to high impact pressure. The pressure developed by the collapsing bubbles is so high that the material from the adjoining boundaries gets eroded forming cavities. This phenomenon is known as cavitation.

HYDROSTATICS:-

Static means something stationary or in rest. Therefore, The shear stress which is equal to $\mu \, \partial u / \partial y$ will be zero. Then the only forces acting on the fluid particle will be :-

(i) due to pressure of fluid normal to the surface,

(ii) due to gravity (or self-weight of fluid particles).

(1) **Total pressure and centre of Pressure:-**

Total pressure is defined as the force exerted by a static fluid on a surface either plane or curved when the fluid comes in contact with the surfaces. This force always acts normal to the surface.

Centre of pressure is defined as the point of application of the total pressure on the surface. There are four cases of submerged surfaces on which the total pressure force and centre of pressure is to be determined. The submerged surfaces may be:-

(i) Vertical plane surface.

(ii) Horizontal plane surface.

(iii) Inclined plane surface, and

(iv) Curved surface.

MEASUREMENTS OF FLOW:-

Accurate measurement of flow rate of liquids and gases is an essential requirement for maintaining the quality of industrial processes. In fact, most of the industrial control loops control the flow rates of incoming liquids or gases in order to achieve the control objective. As a result, accurate measurement of flow rate is very important.

There are different types of flowmeters as listed and destribed below:

(1) **Obstruction type flowmeter**

Obstruction or head type flowmeters are of two types: differential pressure type and variable area type. orifice meter, venturimeter, pitot tube fall under the first category while rotameter is of the second category. In all the cases, an obstruction is created in the flow passage and the pressure drop across the obstruction is related with the flow rate.

Basic principle:-

It is well known that flow can be of two types:- viscous and turbulent. whether a flow is viscous or turbulent can be decided by the Reynold's number R_D. If $R_D > 2000$, the flow is turbulent. In the present case, we assume that the flow is turbulent, that is the normal case for practical situations. We consider the fluid flow through a closed channel of various cross-section, as shown in fig.1. The channel is of varying cross-section and we consider two cross-section of the channel,1 and 2. Let the pressure, velocity, cross-sectional area and height above datum be expressed as P_1, V_1, A_1, and Z_1 for section 1 and the corresponding volumes for section 2 be P_2, V_2, A_2 and Z_2 respectively. We also assume that the fluid flowing is incompressible.

Now from Bernoulli's equation:-

$$\frac{P_1}{Y} + \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{Y} + \frac{V_2^2}{2g} + Z_2 \qquad \text{...(1)}$$

Where 'y' is the specific weight of fluid

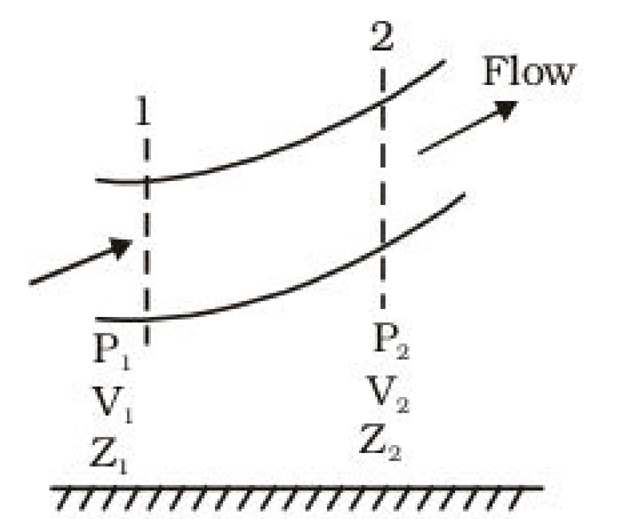


Fig:1:- Flow through a varying cross section.

If $Z_1 = Z_2$, then

$$\frac{P_1}{Y} + \frac{V_1^2}{2g} = \frac{P_2}{Y} + \frac{V_2^2}{2g} \qquad \text{...(2)}$$

If the fluid is incompressible, then $V_1A_1 = V_2A_2$

Therefore, $V_2^2 - V_1^2 = \frac{2g}{Y}(P_1 - P_2)$

or, $V_2^2\left(1 - \frac{A_2^2}{A_1^2}\right) = \frac{2g}{Y}(P_1 - P_2)$

Therefore, $V_2 = \frac{1}{\sqrt{\left(1 - \frac{A_2^2}{A_1^2}\right)}} \sqrt{\frac{2g}{y}(P_1 - P_2)}$

$$= \frac{1}{\sqrt{(1 - B^4)}} \sqrt{\frac{2g}{Y}(P_1 - P_2)}$$

Consider circular cross section, we define 'B' as the ratio of the two diameters, i.e $B = \frac{d_2}{d_1}$ and so,

$A_2 / A_1 = B^2$.

Therefore, the volumetric flow rate through the channel can be expressed as:

$$\theta = V_2A_2 = \frac{A_2}{\sqrt{1 - B^4}} \sqrt{\frac{2g}{Y}(P_1 - P_2)} \qquad \text{...(3)}$$

From the above expression, we can infer that if there is an obstruction in the flow path that causes the variation of the cross-sectional area inside the closed flow channel, there would be difference in static pressure at two points and by measuring the pressure difference, one can obtain the flow rate using eq^n (3). However this expression is valid for incompressible fluids (i.e liquid)only and the relationship between the volumetric flow rate and pressure difference is non-linear.

(2)**Orifice meter**:- It is a device used for measuring rate of flow of a fluid through a pipe. It is a cheaper device as compared to venturimeter. Depending on the type of obstruction, we can have different type of flow meters. Most common among them is the orifice type flowmeter, where an orifice plate is placed in the pipe line as shown in figure.

If d_1 and d_2 are the diameter of the pipe line and the orfice opening, Then the flow rate can be obtained using equation.

(3) by measuring the pressure difference (P_1-P_2).

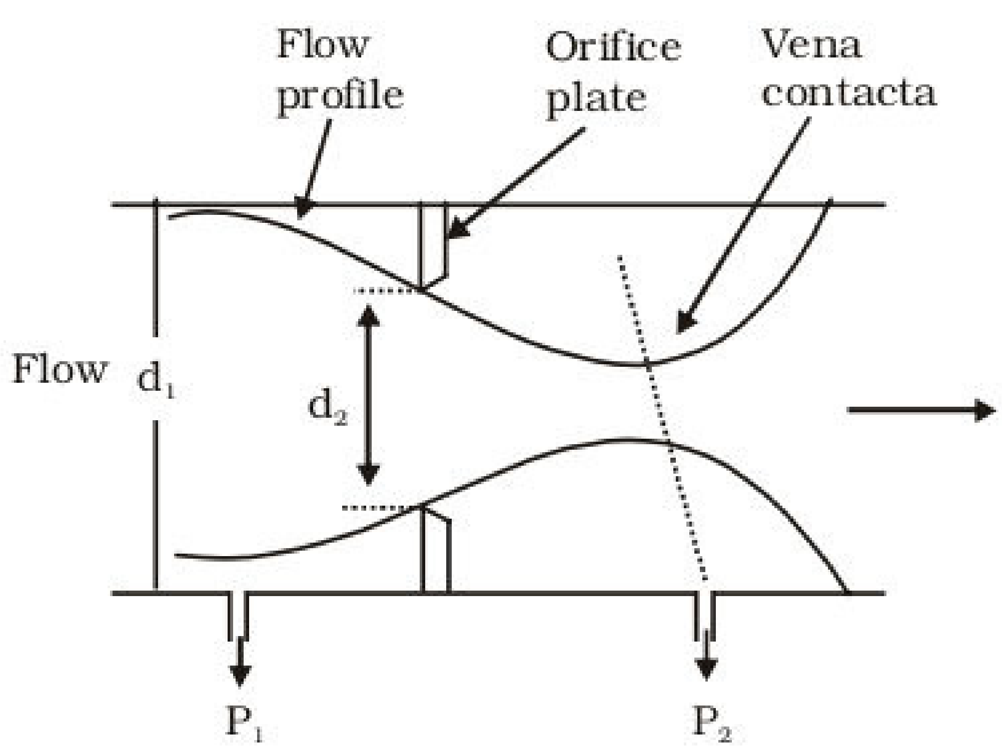


Fig 2:- Orifice type flow meter.

Corrections:-

The flow expression obtained from equation(3) is not an accurate expression in the actual case and some correction on factor named as discharge co-efficient (C_d) has to be incorporated,as :-

$$Q = V_2 A_2 = \frac{C_d A_2}{\sqrt{1-\beta^4}} \sqrt{\frac{2g}{Y}(P_1 - P_2)} \qquad \ldots(4)$$

"C_d" is defined as the ratio of the actual flow and is always less then one. There are infact two main reasons due to which the actual flow rate is less than the ideal one. The first is that the assumption of frictionless flow is not always valid. The amount of friction depends on the Reynold's number (R_D). The more important point is that the minimum flow area is not the orifice area A_2 but is somewhat less and it occurs at a distance from the orifice plate known as vena contracta and secondly we are taking pressure tapping around that point in order to obtain the maximum pressure drop. As a result the correction factor $C_d<1$ has to be incorporated.

In fact C_d depends on B,as well as on R_D. But it has been observed that for $R_D > 10^4$, The flow is totally turbulent and C_d is independent on R_D. In this range the typical value of C_d for orifice plate varies between 0.6 to 0.7.

(3)**Flow measurement of compressible fluids**:-

For compressible fluid i.e. gases the flow rates are normally expessed in terms of mass flow rates. The same obstruction type flow meter can be used but an additional correction factor needs to be introduced to take into account the compressibility of the gas used. The mass flow rate of gases can be expressed as.

$$W = Y\left[\frac{C_d A_2}{\sqrt{1-\beta^4}} \sqrt{\frac{2g(P_1 - P_2)}{V_1}}\right] \qquad \ldots(5)$$

Where,

V_1 = Specific volume of the gas in m^3/kgf

$$Y = 1 - (0.41 + 0.35\beta^4)\frac{P_1 - P_2}{P_1}\frac{1}{K}$$

K= Specific heat ratio.

(4) **Pitot Tube**:-

Pitot tube is widely used for velocity measurement in air craft. Its basic principle can be underestood from the fig below. If a blunt object is placed in the flow channel, the velocity of fluild at the point just before it will be zero. Then, considering the fluid to be incompressible, from eq (2),

We have:-

$$\frac{P_1}{Y} + \frac{V_1^2}{2g} = \frac{P_2}{Y} + \frac{V_2^2}{2g}$$

Now, V_2 = 0, Therefore, $\frac{V_1^2}{2g} = \frac{P_2 - P_1}{Y}$

or, $V_1 = \sqrt{\frac{2g}{Y}(P_2 - P_1)} \qquad \ldots(6)$

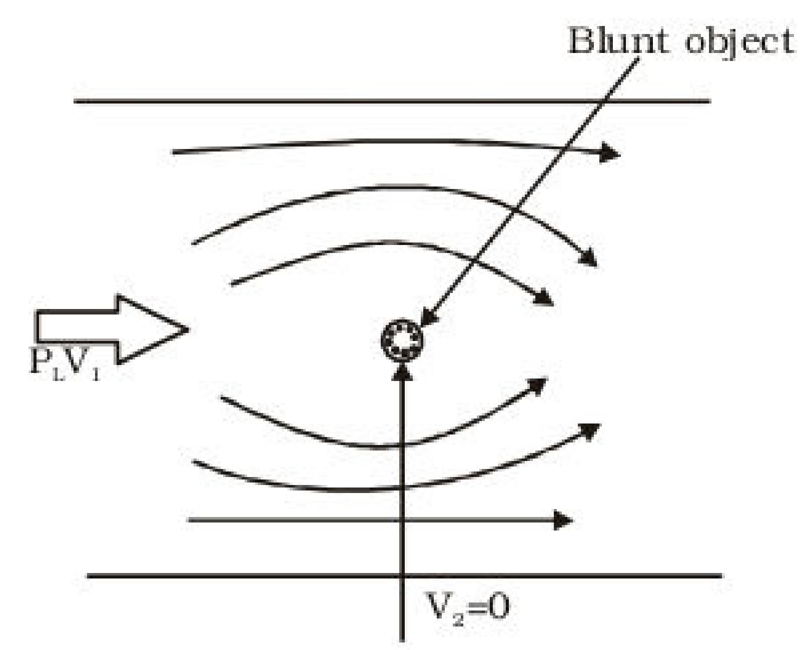


Fig:- Pitot tube :- Basic principle

BERNOULLI'S THEOREM:-

(1) **Euller's Equation of motion**:-

This is equation of motion in which the forces due to gravity and pressure are taken into consideration . This is derived by considering the motion of a fluid element along a stream-line.

Let's consider a stream-line in which flow is taking place in s-direction as shown in figure. Consider a cylinderical element of cross-section dA and length dS. The force acting on the cylindrical element are:-

(i) Pressure force pdA in the direction of flow.

(ii) Pressure force $\left(P+\frac{\partial P}{\partial S}dS\right)dA$ opposite to the direction of flow.

(iii)Weight of element PgdAds.

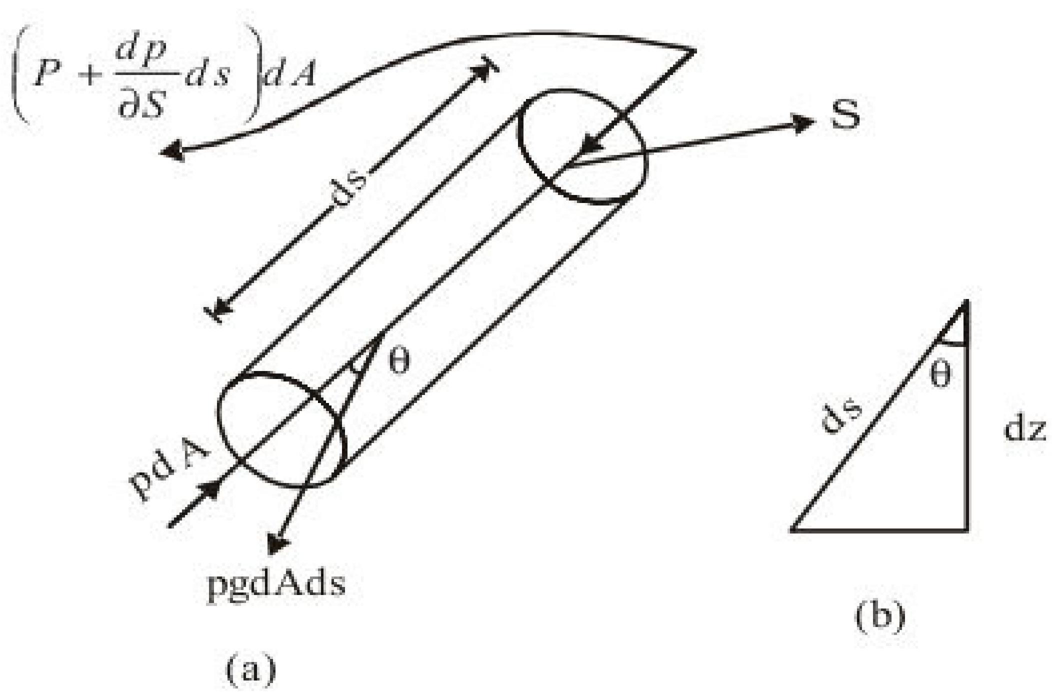


Fig:- Force on a fluid element.

Let "θ" is the angle between the direction of flow and the line of action of the weight of element.

The resultant force on the fluid element in the direction of 'S' must be equal to the mass of fluid element multiplied by acceleration in the direction 's'.

$$PdA-\left(P+\frac{\partial P}{\partial S}dS\right)dA-PgdA\ ds\cos\theta$$

$$= PdAds\times a_s \qquad \ldots(i)$$

Where a_s is the acceleration in the direction of S.

Now, $a_S=\frac{dv}{dt}$, where V is a function of 'S' and 't'.

$$=\frac{\partial v}{\partial s}\frac{ds}{dt}+\frac{\partial v}{\partial t}=\frac{v\partial v}{\partial s}+\frac{\partial v}{\partial t}\left[\because \frac{ds}{dt}=V\right]$$

If the flow is steady, $\frac{\partial v}{\partial t}=0$

$$\therefore a_S=\frac{V\partial V}{\partial S}$$

Substituting the value of a_S in eq (i) and simplying the equation, we get:-

$$\frac{\partial P}{\partial S}dsdA-PgdAds\cos\theta=PdAds\times\frac{V\partial V}{\partial s}$$

Dividing by $PdsdA,-\frac{\partial P}{P\partial s}-g\cos\theta=\frac{V\partial V}{\partial s}$

or $\frac{\partial P}{P\partial S}+g\cos\theta+V\frac{V\partial V}{\partial S}=0$

But from figure we have $\cos\theta = dz/ds$

$$\therefore \frac{1}{P}\frac{\partial P}{\partial S}+g\frac{dz}{ds}+\frac{v\partial v}{\partial s}=0$$

or $\frac{\partial p}{p}+gdz+vdv=0 \qquad \ldots(ii)$

Equation (ii) is known as Euler's equation of motion.

(2) **Bernoulli's equation from Euller's Equation**:-

Bernoulli's equation is obtained by integrating the Eulers equation of motion as:-

$$\int\frac{dp}{p}+\int gdz+\int vdv=\text{constant}$$

If flow is incompressible, P is constant and

$$\therefore \frac{p}{P}+gz+\frac{v^2}{2}=\text{constant}$$

or $\frac{p}{pg}+z+\frac{V^2}{2g}=\text{constant}$

or $\frac{p}{pg}+\frac{V^2}{2g}+Z=\text{constant} \qquad \ldots(iii)$

Equation (iii) is a Bernoulli's equation in which,

$\frac{p}{pg}$ = pressure energy per unit weight of fluid or pressure head.

$V^2/2g$ = kinetic energy per unit weight or kinetic head.

Z = potential energy per unit weight or potential head.

(3) **Assumptions**:-

The following are the asumptions made in the derivation of Bernoulli's equation.

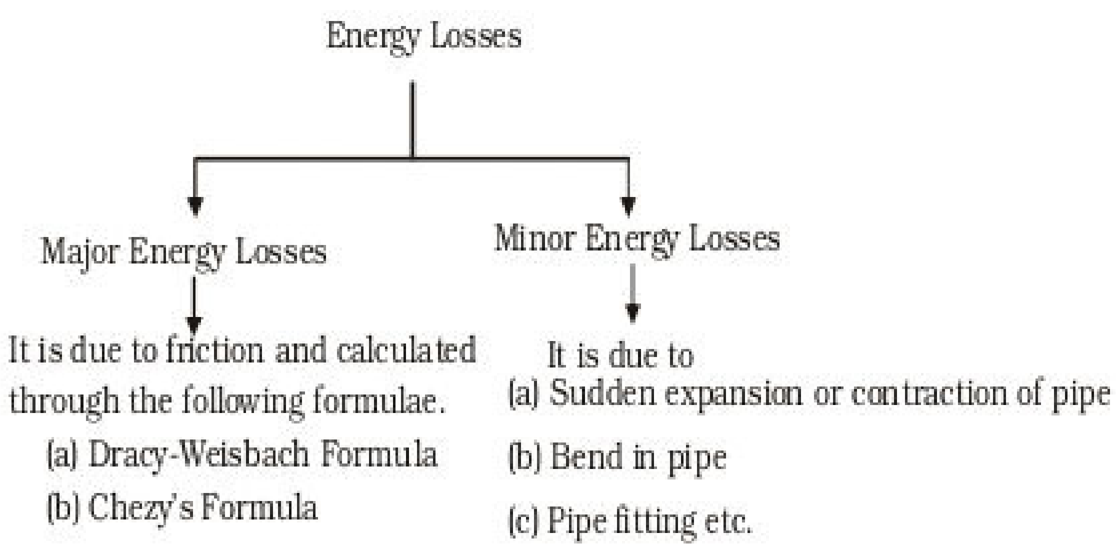
(i) The fluid is ideal, i.e, Viscosity is zero.
(ii) The flow is steady
(iii) The flow is imcompressible.
(iv) The flow is irrotational.

FLOW THROUGH PIPES

When the Reynold number is less than 2000 for pipe flow, then the pipe flow is known as laminar flow whereas when the Reynold number is more than 4000, the flow is known as turbulent flow. In this topic the turbulent flow of fluids through pipes running full will be considered. If the pipes are partially full as in the case of sewer lines the pressure inside the pipe is same and equal to atmospheric pressure. Such a flow of fluid in the pipe is not under pressure. But considering flow of fluids through pipes under pressure only.

(1) **Loss of energy in pipes :-**

When a fluid flows through a pipe, the fluid experience some resistance due to which some of the enegy of fluid is lost. This loss of energy is classified as :-



(2)**Flow through pipes in series or flow through compund pipes**

Pipes in series or compound pipes are defined as the pipes of different length and different diameters connected end to end(in series) to form a pipe line as shown in fig.

Let, L_1, L_2, L_3 = length of pipes 1,2 and 3 respectively.

d_1, d_2, d_3 = diameter of pipes 1,2,3 respectively.

$V_1\ V_2, V_3$ = velocity of flow through pipes 1,2,3.

f_1, f_2, f_3 = co-efficient of friction for pipes 1,2,3

H = difference of water level in two tanks.

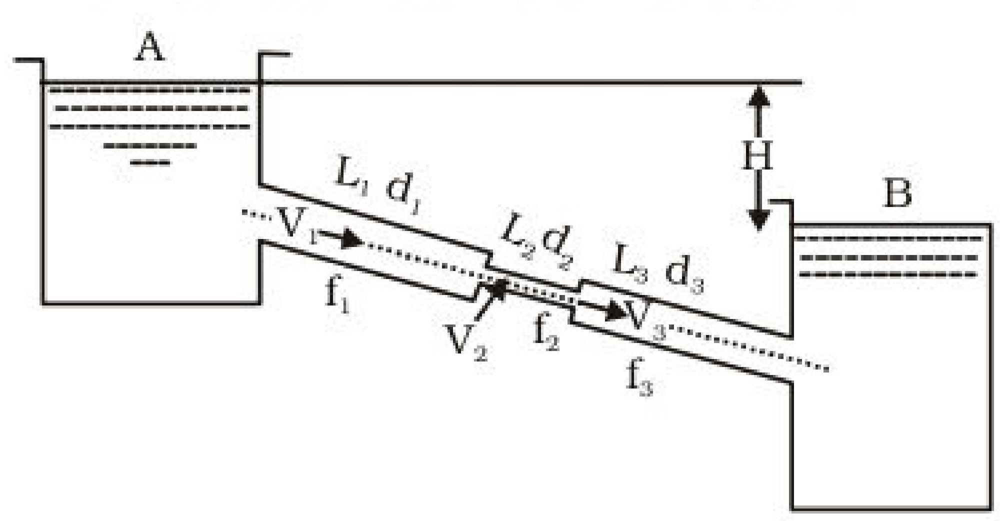


Fig:- Pipes connected in series

The discharge passing through each pipe is same.

$$\therefore \quad Q = A_1V_1 = A_2V_2 = A_3V_3$$

The difference in liquid surface levels is equal to the sum of the total head loss in the pipes.

$$\therefore\ H = \frac{0.5V_1^2}{2g} + \frac{4f_1.L_1V_1^2}{d_1 \times 2g} + \frac{0.5V_2^2}{2g} + \frac{4f_2L_2V_2^2}{d_2 \times 2g}$$

$$+ \frac{(V_2 - V_3)^2}{2g} + \frac{4f_3L_3V_3^2}{d_3 \times 2g} + \frac{V_3^2}{2g}$$

If minor losses are neglected, then above equa tion becomes as:-

$$H = \frac{4f_1L_1V_1^2}{d_1 \times 2g} + \frac{4f_2L_2V_2^2}{d_2 \times 2g} + \frac{4f_3L_3V_3^2}{d_3 \times 2g}$$

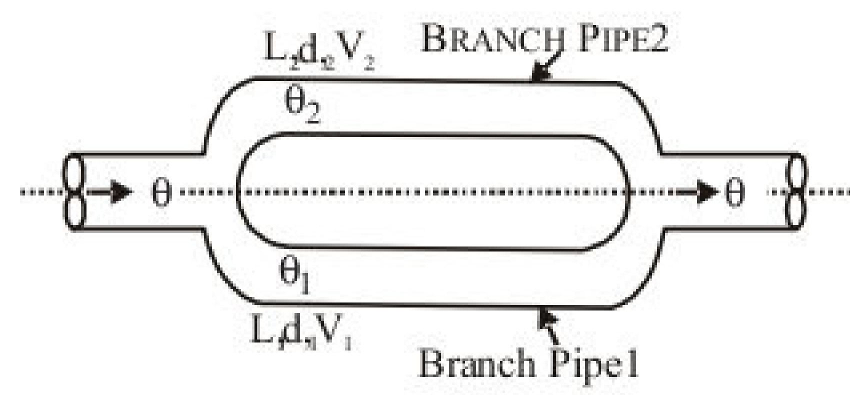
If the co-efficient of friction is same for all pipes i.e, $f_1 = f_2 = f_3 = f_1$, then equation becomes as,

$$H = \frac{4fL_1V_1^2}{d_1 \times 2g} + \frac{4fL_2V_2^2}{d_2 \times 2g} + \frac{4fL_3V_3^2}{d_3 \times 2g}$$

$$= \frac{4f}{2g}\left[\frac{L_1V_1^2}{d_1} + \frac{L_2V_2^2}{d_2} + \frac{L_3V_3^2}{d_3}\right]$$

(3) **Flow through parallel pipes gets**:-

Consider a main pipe which divided into two or more branches as shown in fig and again join together downstream to form a single pipe, then the branch pipes are said to be connected in parallel.The discharge through the main is increased by connecting pipes.



Fig;-

The rate of flow in the main pipe is equal to the sum of rate of flow through branch pipes. Hence, from above fig, We have, $\theta = \theta_1 + \theta_2$

In this arrangement, the loss of head for each branch pipe is same.

$\therefore$ Loss of head for branch pipe1= Loss of head for branch pipe 2.

$$\text{or.}\ \frac{4f_1L_1V_1^2}{d_1 \times 2g} = \frac{4f_2L_2V_2^2}{d_2 \times 2g}$$

$$\text{If, } f_1 = f_2, \text{ then } \frac{L_1V_1^2}{d_1 \times 2g} = \frac{L_2V_2^2}{d_2 \times 2g}$$

(4) **Flow through Branched Pipes:-**

When three or more reservoirs are connected by means of pipes having one or more junctions, the system is called branching pipe system. Fig shows three reservoirs at different levels connected to a single junction, by means of pipes which are called branched pipes. the lengths, diameters and co-efficient of friction of each pipe is given. It is required to find the discharge and direction of flow in each pipe. The basic equations used for solving such equation/problems are:-

(i) **Continuity equation:-** It means the inflow of fluid at the junction should be equal to the outflow of fluid.

(ii) Bernoullis equation and

(iii) Daray-weisbach equation.

Also it is assumed that reservoir are very large and the water surface levels in the reservoirs are constant so that steady conditions exists in the pipes. Also minor losses assumed are very small. The flow from reservoir 'A' takes place to junction D. The flow from junction 'D' is towards reservoir 'C'. Now the flow from junction 'D' towards reservoir 'B' will take place only when piezometrichead at 'D' (which is equal to $\frac{P_D}{Pg}+Z_D$) is more than piezometric head at B(i.e,Z_B).

Let us consider that flow is from D to resesvoir B.

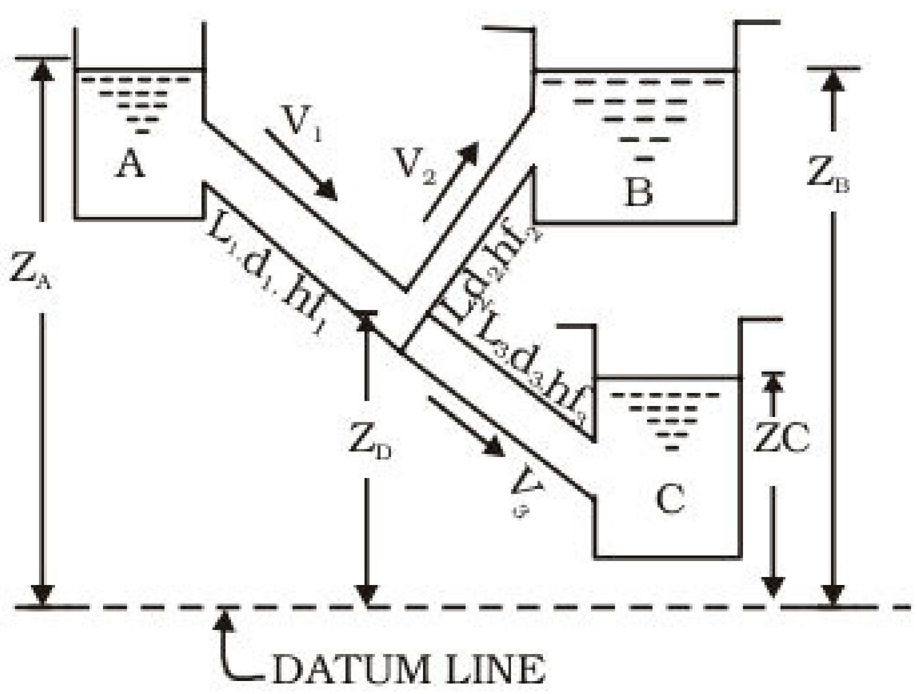


fig:-

For flow from 'A' to 'D' from Bernoulli's equation

$$Z_A = Z_D + \frac{P_D}{Pg} + hf_1$$

For flow from 'D' to 'B' from Bernoulli's equation

$$Z_D + \frac{P_D}{Pg} = Z_B + hf_2$$

For flow from 'D' to 'C' from Bernoulli's equation

$$Z_D + \frac{P_D}{Pg} = Z_C + hf_3$$

From continuity equation,

Discharge through AD = Discharge through (DB + DC).

$$\therefore \frac{\pi}{4} d_1^2 V_1 = \frac{\pi}{4} d_2^2 \times V_2 + \frac{\pi}{4} d_3^2 V_3$$

or $d_1^2 V_1 = d_2^2 V_2 + d_3^2 V_3$

There are four unknown,i.e, V_1, V_2,V_3 and $\frac{P_D}{Pg}$ and there are four equations (i),(ii),(iii) and (iv). Hence, unknowns can be calculated.

FLOW IN OPEN CHANNELS:-

Flow in open channels is defined as the flow of a liquid with a free surface. A free surface is a surface having constant pressure such as atmospheric pressure. Thus, a liquid flowing at atmospheric pressure through a passage is known as if flowing in open channels. In most of the cases, the liquid is taken as water. Hence, flow of water through pipes at atmospheric pressure or when the level of water in the pipe is below the top of the pipe is also classified as open channel flow.

In case of open channel flow, as the pressure is atmospheric, the flow takes place under the force of gravity which further means that the flow takes place due to the slope of the bed of the channel only. The hydraulic gradient line coincides with the free surface of water.

(1) **Classification of flow in channels**:-

The flow in open channel is classified into the follwoing types:-

(i) Steady flow and unsteady flow,

(ii) Uniform flow and non-uniform flow,

(iii) Laminar flow and turbulent flow, and

(iv) Sub-critical, critical and super critical flow.

Steady flow and unsteady flow:- If the flow characteristics such as depth of flow, velocity of flow, rate of flow at any point in an open channel flow does not change with respect to time, the flow is said to be steady flow. Mathematically,steady flow is expressed as,

$$\frac{\partial V}{\partial t} = 0, \frac{\partial \theta}{\partial t} = 0 \quad or \frac{\partial y}{\partial t} = 0$$

Where, V = velocity, θ = rate of flow and y = depth of flow.

If at any point in open channel flow, the velocity of flow, depth of flow or rate of flow changes with respect to time, the flow is said to be unsteady flow. Mathematically, unsteady flow means:-

$$\frac{\partial V}{\partial t} \neq o \quad or \frac{\partial y}{\partial t} \neq 0 \quad or \frac{\partial \theta}{\partial t} \neq 0.$$

Uniform flow and non-uniform flow:-

If for a given length of the channel, the velocity of flow, depth of flow, slope of the channel and cross-section remain constant, the flow is said to be uniform. If for a given length of the channel the velocity of flow, depth of flow etc. do not remain constant, the flow is said to be non-uniform flow.

Mathematically, uniform and non-uniform flow are written as :-

$$\frac{\partial y}{\partial s} = 0, \frac{\partial V}{\partial s} = 0 \quad \text{for uniform flow.}$$

and $\frac{\partial y}{\partial s} \neq 0, \frac{\partial V}{\partial S} \neq 0$ for non-uniform flow.

Non-uniform flow in open channel is also called varied flow which is classified in the flollowing two types as

(i) Rapidly varied flow(R.V.F)

(ii) Gradually varied flow (G. V.F).

Rapidly varied flow is defined as that flow in which depth of flow changes abruptly over a small length of the channel as shown in fig. When there is an obstruction in the path of flow of water, the level of water rises above the obstruction and then falls and again rises over a small length of channel.Thus, the depth of flow changes rapidly over a short length of the channel. For this short length of the channel, the flow is called rapidly varied flow(R.V.F)

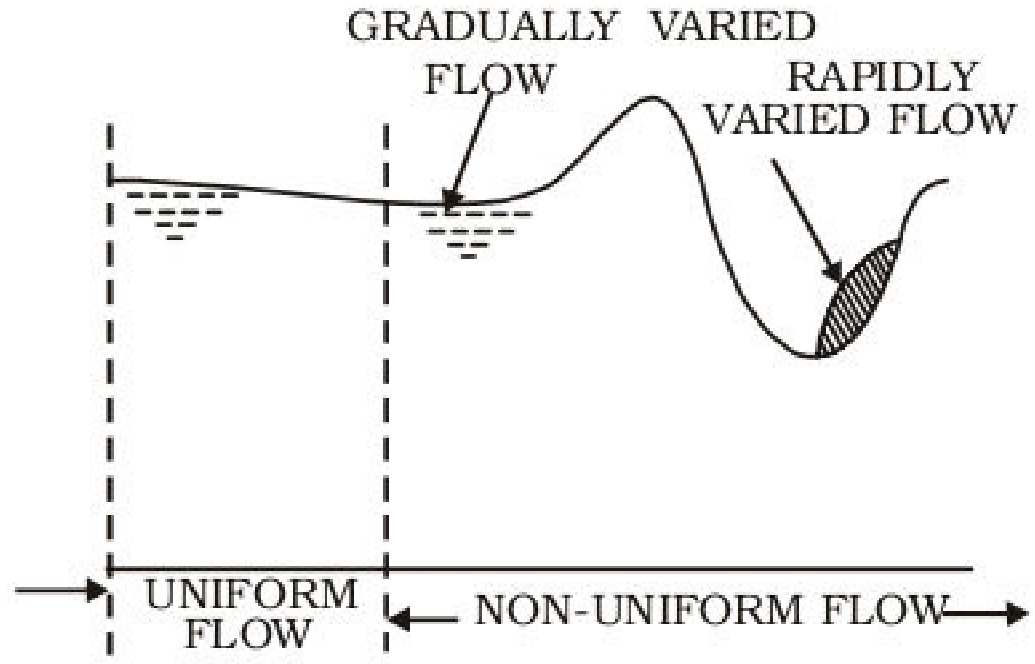


Fig:- Uniform and non-uniform flow.

If the depth of flow in a channel changes gradually over a long length of the channel, the flow is said to be gradually varied flow and is denoted by G.V.F.

Laminar flow and Turbulent Flow:-

The flow in open channel is said to be laminar if the Reynold number (Re) is less then 500 or 600. Reynold number in case of open channels is defined as:-

$$Re = \frac{PVR}{\mu}$$

Where, V= mean velocity of flow of water

R = Hydraulic radius or Hydraulic mean depth.

P and μ = Density and viscosity of water.

If the Reynold number is more than 2000, the flow is said to be turbulent in open channel flow. If Re lies between 500 to 2000, the flow is considered to be in transition state.

Sub-Critical, Critical and Super Critical flow:-

The flow in open channel is said to be sub-critical if the froude number (Fe) is less than 1.0. The fraude number is defined as :- $Fe = V / \sqrt{gD}$

Where V = Mean velocity of flow

D = Hydraulic depth of channel and is equal to the ratio of wetted area to the top width of channel = $\frac{A}{T}$, where T = TOP width of channel.

Sub-critical flow is also called tranquil or streaming flow. For sub-critical flow, Fe<1.0.

The flow is called critical if Fe = 1.0 and if Fe > 1.0 the flow is called super critical or shooting or rapid or torential.

WEIRS:-

A weir is a concrete or masonary structure, placed in an open channel over which the flow occurs. It is generally in the form of vertical wall, with a sharp edge at the top, running all the way across the open channel. The notch is of small size while the weir is of a bigger size.

Nappe or Vein:- The sheet of water flowing through a notch or over a weir is called Nappe or vein.

Crest or Sill :- The bottom edge of a notch or a top of a weir over which the water flows, is known as the sill or crest.

(1) **Classification of weirs**:-

Weirs are classified according to the shape of the opening, the shape of crest, the effect of the sides on the nappe and nature of discharge. The following are important classification.

(i) **According to the shape of the opening:-**

(a) Rectangular weir

(b) Triangular weir, and

(c) Trapezoidal weir(Cippoletti weir)

(ii) **According to the shape of the crest:-**

(a) Sharp-crested weir

(b) Broad-crested weir,

(c) Narrow-crested weir, and

(d) Ogee-shaped weir.

(iii) **According to the effect of sides on the emerging nappe:-**

(a) Weir with end contraction, and

(b) Weir without end contraction

Discharge over Rectangular Weir:-

$$\theta = \frac{2}{3} C_d \times L \times \sqrt{2g}[H]^{3/2}.$$

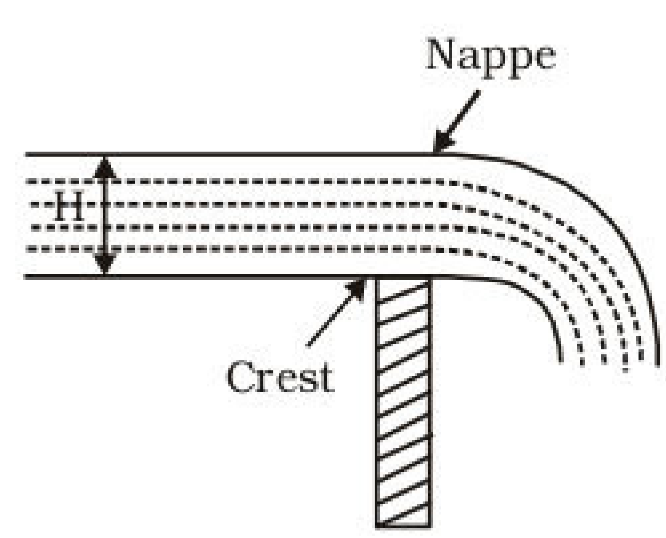


Fig:- Rectangular Weir

Where, θ = rate of discharge

C_d= Co-efficent of discharge

H = Head of water over the crest.

L = Length of the notch or weir.

Discharge over Triangular weir:-

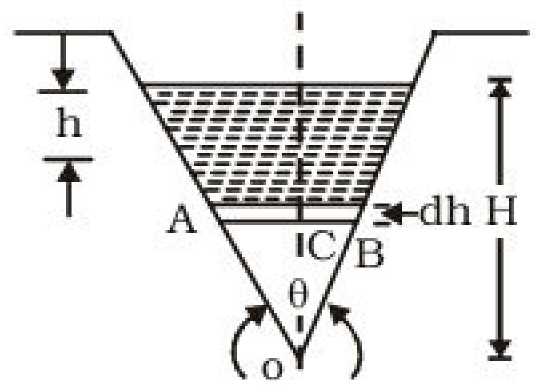


Fig:- Triangular weir

$\theta = 1.417\,H^{5/2}$

Where, H= head of water above V-weir

θ = angle of notch.

Advantage of Triangular WEIR OVER RECTAN GULAR WEIR:-

(i) The expression for discharge for a right angled weir is very simple.

(ii) For measuring low discharge, triangular weir gives more accurate results than a rectangular weir.

(iii) In case of triangular weir, only one reading, i.e,(H) is required for the computation of discharge.

(iv) Ventilation of a triangular weir is not necessary.

Discharge over a Trapezoidal weir:-

Let, H = Height of water over the notch

L = Length of the crest of the notch.

Cd_1= Co-efficient or discharge for rectangular portion ABCD.

Cd_2 = Co-efficent of discharge for triangular portion[FAD and BCE].

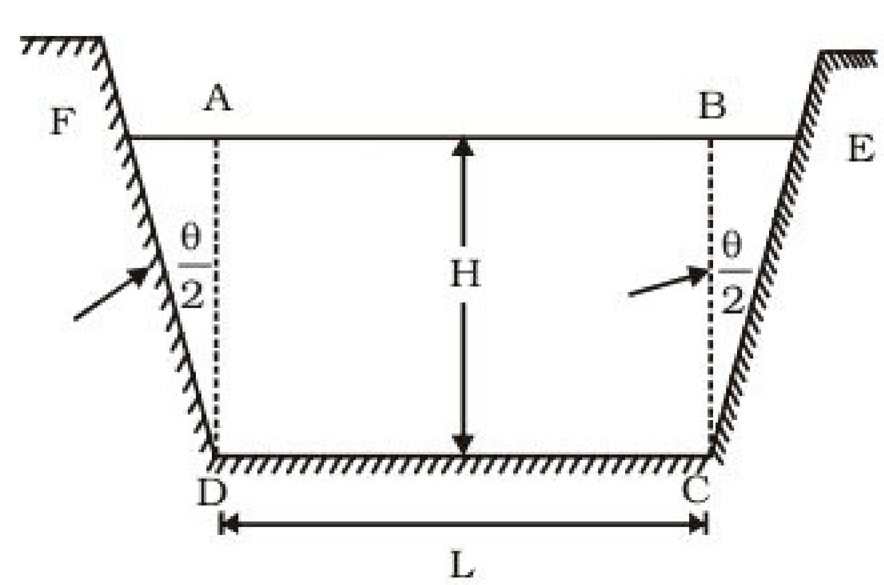


Fig:- The trapezoidal notch

$\therefore$ Discharge through trapezoidal notch or weir FDCEF= $\theta_1 + \theta_2$

$$= \left(\frac{2}{3} C_{d_1} L\sqrt{2g} \times H^{3/2}\right) + \left(\frac{8}{15} C_{d_2} \times \tan\frac{\theta}{2} \times \sqrt{2g} \times H^{5/2}\right)$$

FLUMES :

A flume is a human-made channel for water in the form of an open declined gravity chute whose walls are raised above the surrounding terrain, in contrast to a trench or ditch, Flumes are not to be confused with aqueducts, which are built to transport water, rather than transporting materials using flowing water. Flumes route water from a diversion dam or weir to a desired material collection location.

Flumes can accelerate slow, sub.critical (fr <1) flow to a supercritical state (Fr >1) by:-

(i) Change in elevation

(ii) Contraction of the sidewalls, or

(iii) Combination of the above two.

Accelerating slow flow to a supercritical state creates upstream conditions where under free flow conditions, the flow rate can be determined by measuring the water level at a single defined point in the flume (Ha).

The relationship between the water level at the point of measurement (Ha) and the flow rate can be obtained by test data (short-throated flumes) or derived formula (long-throated flumes).

Flume advantages :

(i) The ability to measure higher flow rates than a com parably sized weir.

(ii) Less head loss (generally $\frac{1}{4}$th that of a weir).

(iii) The ability to pass debris more rapidly.

(iv) Wide range of styles and sizes.

(v) Off the shelf availability.

(vi) Smaller installation footprint.

(vii) Less rigorous maintenance requirements.

SPILLWAYS :

Passages constructed either within a dam or in the periphery of the reservoir to safely pass the excess of the river water during flood are called spillways.

Ordinarily, the excess water is drawn from the top of the reservior created by the dam and conveyed through an artifically created waterway back to the river. In some cases the water may be diverted to an adjacent river valley. In addition to providing sufficient capacity, the spillway must be hydraulically adequate and structurally safe and must be located in such a way that the out-falling discharges back into the river do not erode or undermine the downstream toe of the dam. The surface of spillway should also be such that it is able to withstand erosions or scouring due to the very high water velocities generated during the passage of a flood through the spillway.

Usually, spillways are provided with gates, which provides a better control on the discharge passing through. However in remote areas, where access to the gates by people may not be possible during all times as during the rainy season or in the night ungated spillways may have to be provided.

The capacity of spillway is usually worked out on the basic of a flood routing study. As such that the capacity of a spillway is seen to depend upon the following major factors:

(i) The inflow flood
(ii) The volume of storage provided by the reservoir.
(iii) Crest height of the spillway.
(iv) Gated or ungated.

According to the Bureau of Indian standards guidelines IS: 11223 -1985 "Guidelines for fixing spillway capacity", the following values of inflow design floods (IDF) should be looked into for the design of spillway.

(i) For large dams (defined as those with gross storage capacity greater than 60 million m^3 or hydraulic head greater than 60 million m^3 or hydraulic head between (2m and 30m),IDF should be based on the Standard Projects flood (SPF).
(ii) For intermediate dams those with gross storage between 10 and 60 million m^3 or hydraulic head between (2m and 30m), IDF should be based on the standard project flood(SPF).
(iii) For small dams (gross storage between 0.5 to 10 million m^3 or hydraulic head between 7.5m to 12m), IDM may be taken as the 100 year return period flood.

Spillways are ordinarily classified according to their most prominent feature, either as it pertains to the control, to the discharge channel, or to some other component. The common types of spillway in use are the following:-

(i) Free overfall (straight Drop) Spillway
(ii) Overflow (ogee) Spillway
(iii) Chute (Open Channel/Trough) Spillway.
(iv) Side channel spillway.
(v) Soft (Drop Inlet/Morning Glory) Spillway
(vi) Tunnel (Conduit) Spillway
(vii) Siphon Spillway.

Free Overfall (Straight Drop) Spillway:-

This is the simplest type of spillway and may be constructed on small bunds or on thin arch dams etc. It is a low weir and simple vertical type fall structure as shown in fig.

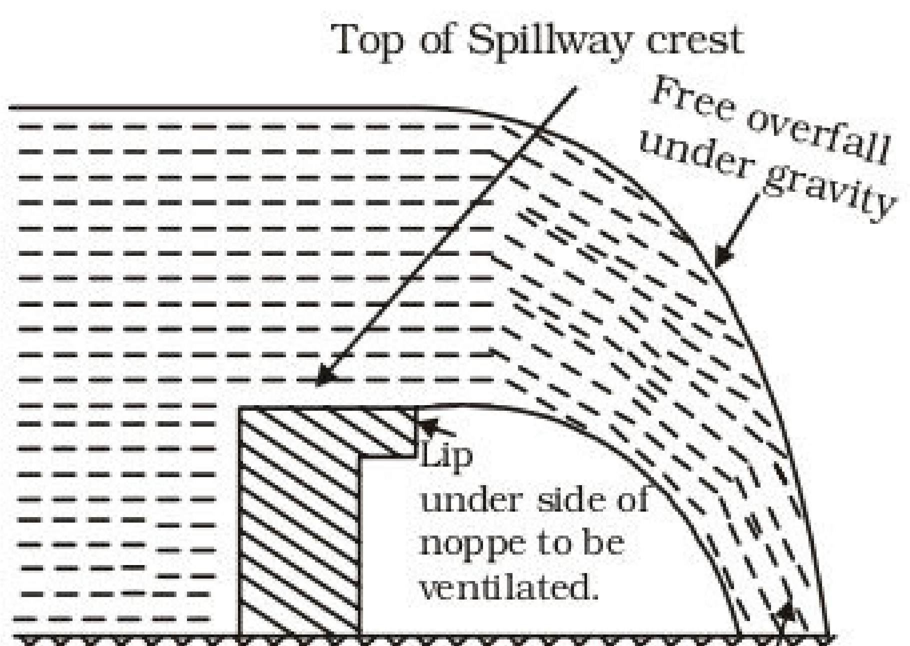
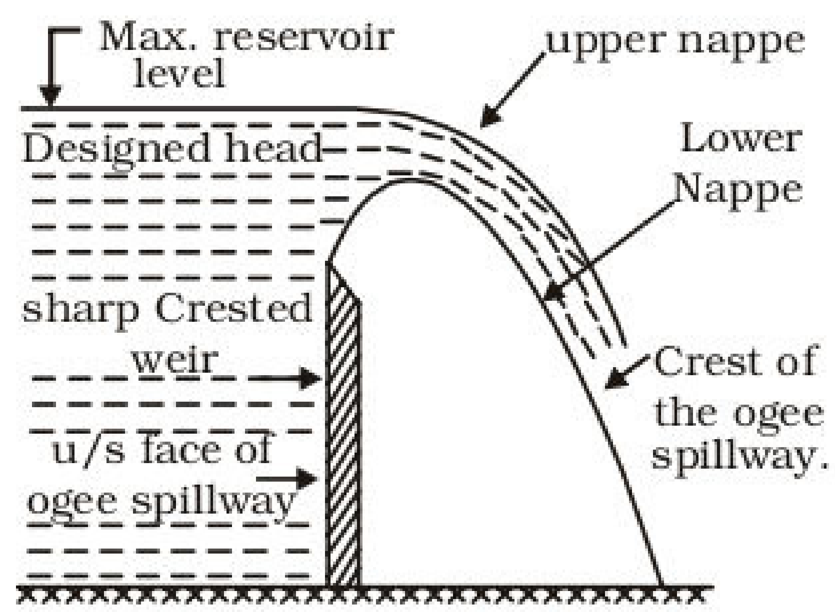


Fig:- Straight Drop Spillway Without d/s protection. **Serious erosion is caused here if no apron etc. is provided**

Ogee Spillway or overflow spillway:-

Ogee spillway is an improvement upon the free overfall spillway. It is widely used with concrete masonry, arch and buttress dams. such a spillway can be easily used on valleys where the width of the river is sufficient to provide the required crest length and the river bed below can be protected from scour at moderate costs.



Chute Spillway:-

An ogee spillway is mostly suitable for concrete gravity dams especially when the spillway is located within the dams body in the same valley. But for earthen and rockfill dams, a seperate spillway is generally constructed in a blank or a saddle away from the main valley. The trough spillway or chute spillway is the simplest type of spillway which can be easily provided independently and at low costs.

Side channel spillway:-

A side channel spillway is one in which the control weir is placed approximately parallel to the upper portion of the discharge channel as may be seen from Fig. The flow over the crest falls into a narrow trough opposite to the weir turns an approximate right angle and then continues into the main discharge channel.

Discharge characteristics of a side channel spillway are similar to those of an ordinary overflow spillway and are dependent on the selected profile of the weir crest. Although the side channel is not hydraulically efficient, nor inexpensive, it has advantages which make it adoptable to spillway where a long overflow crest is required in order to limit the effect (surcharge held to cause flow) and the abutments are steep and precipitous.

Siphon Spillway:-

A siphon spillway is a closed conduit system formed in the shape of an inverted U, positioned so that the inside of the bend of the upper passageway is at normal reservoir storage level. This type of siphon is also called a saddle siphon spillway. The initial discharges of the spillway as the reservoir level rises above normal, are similar to flow over weir, Siphonic action takes place after the air in the bend over the crest has been exhausted. Continuous flow is maintainted by the suction effect due to the gravity pull of the water in the lower leg of the siphon.

Siphone spillway comprise usually of five components, which include an inlet, an upper leg, a throat or control section lower leg and an outlet. A siphon breaker air vent is also provided to control the siphonic action of the spillway so that it will cease operation when the reservoir water surface is drawn to the normal level. Otherwise the siphon would continue to operate until air entered the inlet. The inlet is generelly placed well below the full Reservoir level to prevent entrance of drifting materials and to avoid the formation of vortices and draw downs which might break siphonic action.

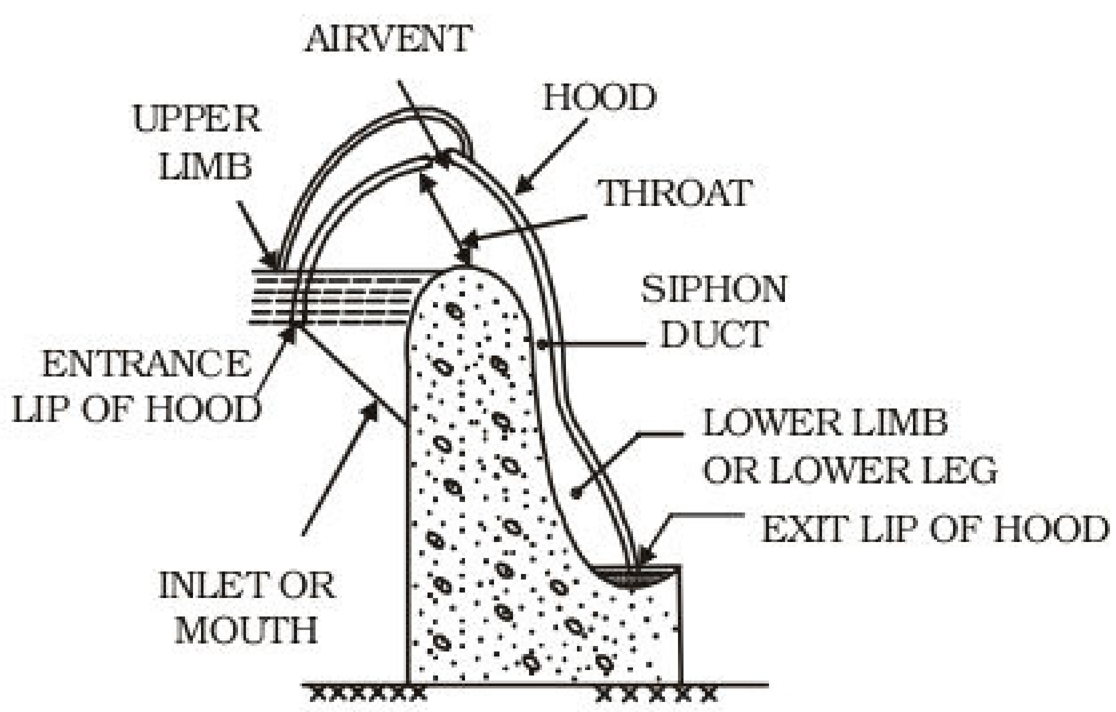


Fig:- SIPHON SPILLWAY

PUMPS :

The hydraulic machines which converts the mechanical energy into hydraulic energy are called pumps. The hydraulic energy is in the form of pressure energy. If the mechanical energy is converted into pressure energy by means of centrifugal force acting on the fluid, the hydraulic machine is called centrifugal pump.

(1) CENTRIFUGAL PUMPS

The centrifugal pumps acts as a reversed of an inward radial flow reaction turbine. This means that the flow in centrifugal pumps is in the radial outward reaction. The centrifugal pump works on the principie of forced vortex flow. According to this principle when a certain mass of liquid is rotated by an external torque, the rise in pressure head at any point of the rotating liquid is proportional to the square of tangential velocity of the liquid at that point (i.e, rise in pressure head,

$= \frac{V^2}{2g}$ or $\frac{w^2r^2}{2g}$. Thus at the outlet of the impeller, where radius is more than the rise in pressure head, the liquid will be discharged at the outlet with a high pressure head. Due to this high pressure head, the liquid can be lifted to a high level.

Main Parts of A Centrifugal Pump:-

The following are the main parts of a centrifugal pump:

(i) **Impeller**:- The rotating parts of a centrifugal pump is called impeller. It consists of a series of backward curved vanes. The impeller is mounted on a shaft that that is connected to an electric motor.

(ii) **Casing**:- The casing of a centrifugal pump is similar to the casing of a reaction turbine. It is an air-tight passage surrounding the impeller designed in such a way that the kinetic energy of the water discharge at the outlet of the impeller is converted into pressure energy before the water leaves the casing and enters the delivery pipe.

(iii) **Suction pipe**:-A pipe whose one end is connected to the inlet of the pump and other end dips into in a Pump is known as suction pipe.

(iv) **Delivery pipe**:- A pipe whose one end is connected to the outlet of the pump and other end delivers the water at a required height is known as delivery pipe.

DEFINITIONS OF HEADS AND EFFICIENCIES OF A CENTRIFUGAL PUMP :-

(i) **Suction Head(h_s):-** It is the vertical height of the centre line of the centrifugal pump above the water surface in the tank or pump from which water is to be lifted. The height is also called suction lift and is denoted by "h_s".

(ii) **Delivery Head (h_d) :-** The vertical distance between the centre line of the pump and the water surface in the tank to which water is delivered is known as delivery head . This is denoted by hd.

(iii) **Static Head (H_s) :-** $H_s = h_s + h_d$

Where, h_s = Suction Head, and

h_d = Static Head.

(iv) **Manometric Efficiency (n_{man}) :-**

$$n_{man} = \frac{\text{Manometric head}}{\text{Head imparted by Impeller to water}}$$

$$= \frac{H_m}{\left(\frac{V_{w_2}u_2}{g}\right)} = \frac{gH_m}{V_{w_2}u_2}.$$

(v) **Mechanical Efficiency (n_{mec})**

$$n_{mec} = \frac{\text{Power at impeller}}{\text{Power at the short}} = \frac{\frac{w}{g}\left(\frac{V_{w_2}\mu_2}{1000}\right)}{S.P}$$

where, S.P = Short Power.

(v) **Overall Efficiency (n_0)**:-

$$n_0 = n_{mec} \times n_m$$

(2) **RECIPROCATING PUMPS**:-

If the mechanical energy is converted into hydraulic energy (or pressure energy) by sucking the liquid into a cylinder in which a piston is reciprocating (moving backwards and forwards), which exerts the thrust on the liquid and increases its hydraulic energy(pressure energy), the pump is said to be as reciprocating pump.

The follwoing are the main parts of a reciprocating pump:-

(i) A cylinder with a piston, piston rod, connecting rod and a crank.
(ii) Suction pipe
(iii) Delivery pipe.
(iv) Suction value, and
(v) Delivery value.

DISCHARGE THROUGH A RECIPROCATING PUMP:-

Consider a single acting reciprocating pump

Let D = diameter of the cylinder.

A = Cross-sectional area of the piston or cylinder $\frac{\pi}{4}D^2$

r = radius of crank.

N = r.p.m of the crank.

L = Length of the stroke = 2×r.

h_s = Height of the axis of the cylinder from water surface in pump.

h_d = Height of delivery outlet above the cylinder axis (also called delivery head). volume of water delivered in one revolution or discharge of water in one revolution

= Area × Length of stroke = A × L

Number of revolution per second = $\frac{N}{60}$

∴ Discharge of the pump per second,

$$\theta = A \times L \times \frac{N}{60} = \frac{ALN}{60}$$

Weight of water delivered per second

$$W = p \times g \times \theta = \frac{PgALN}{60}$$

WORK DONE BY RECIPROCATING PUMP :-

Work done by the reciprocating pump per second is given by the reaction as :

work done per second = weight of water lifted per second × total height through which water is lifted

$= W \times (h_s + h_d)$ ...(i)

Where $(h_s + h_d)$ = Total height through which water is lifted from eq^n (20.2), Weight 'W' is given by :-

$$W = \frac{pg \times ALN}{60}$$

Substituting the value of 'W' in equation (i), work done per second $= \frac{pg \times ALN}{60} \times (h_s + h_d)$

∴ Power required to drive the pump in kW

$$P = \frac{\text{work done per second}}{1000} = \frac{Pg \times ALN \times (hs + hd)}{60 \times 1000}$$

$$= \frac{pg \times ALN \times (hs + hd)}{60{,}000} \text{kW.}$$

SLIP OF RECIPROCATING PUMP:-

Slip of a pump is defined as the difference between the theoretical discharge and actual discharge of the pump.

Slip = Q_{th} - Q_{act}

But slip mostly expressed as percentage slip that is given by:-

$$\text{Percentage slip} = \frac{Q_{th} - Q_{act}}{Q_{th}} \times 100 = \left(1 - \frac{Q_{act}}{Q_{th}}\right) \times 100$$

$$= (1 - C_d) \times 100.$$

Where C_d = Coefficient of discharge.

TURBINES:-

Turbines are defined as the hydraulic machine which convert hydraulic energy into mechanical energy. This mechanical energy is used in running an electric generator which is directly coupled to the short of the turbine. Thus, the mechanical energy is converted into electrical energy. This electrical power which is obtained from the hydraulic energy (energy of water) is known as Hydro-electric power. At present the generation of hydro electric power is the cheapest as compared with the the power generated by other sources such as oil, coal etc.

DEFINITIONS OF HEADS AND EFFICIENCIES OF A TURBINE:-

(i) **Gross Head:-** The difference between the head race level and tail race level when no water is flowing is known as gross head. It is denoted by 'Hg'.

(ii) **Net Head :-** It is also called effective head and defined as the head available at the inlet of the turbine. When water is flowing from head race to the turbine, a loss of head due to friction between the water and penstocks occurs.

H = Hg- hf

Where, Hg, = Gross head, $h_f = \frac{4 \times f \times L \times V^2}{D \times 2g}$

in which,V = Velocity of flow in penstock.

L = Length of penstock

D = Diameter of penstock.

(iii) **Efficiencies of a Turbine**:-

The follwoing are the important efficiencies of a turbine.

(a) Hydaulic Efficiency, N_h

(b) Mechanical Efficiency N_m

(c) Volumetric Efficiency, N_v and

(d) Overall Efficiency, N_o

(a) **Hydraulic Efficiency(N_h)** :-

$$N_h = \frac{\text{Power delivered to runner}}{\text{Power supplied at Inlet}} = \frac{R.P}{W.P}$$

Where, R.P = Runner Power

$$= \frac{W}{g}\frac{\left[V_{w_1} m_1 \pm V_{w_2} m_2\right]}{1000} kW.$$

$$\text{and , W. P} = \frac{W \times H}{1000} kW$$

where, W= weight of water striking the vanes of the turbine per second = Pg × Q

V_{w1} = Velocity of wheel at inlet

V_{w2} = Velocity of wheel at outlet.

u = Tangential velocity of vane

u_1 = Tangential velocity of vane at inlet for radial vane.

u_2 = Tangential velocity of vane at outlet for radial vane.

H = Net head on the turbine.

Power supplied at the inlet of turbine in SI units is known as water power. It is given by

$$W.P = \frac{P \times g \times Q \times H}{1000} kW$$

for water P = 1000 kg / m^3

$\therefore$W.P = g × Q × H kW.

(b) **Mechanical Efficiency(N_m):-**

$$n_m = \frac{\textit{Power at the shaft of turbine}}{\textit{Power delivered by water to the runner}} = \frac{S.P}{R.P}$$

(c) **Volumetric Efficiency (N_v)**

$$n_v = \frac{\text{Volume of water actually striking the runner}}{\text{Volume of water supplied to the turbine}}$$

(d) **Overall Efficiency(N_o):-**

$$N_o = \frac{\textit{Shaft power}}{\textit{water power}} = N_m \times N_h$$

If shaft power (S.P) is taken in kW then water power should also be taken in kW. Shaft power is commonly represented by P. But from Equation:-

$$\text{Water power in kW} = \frac{P \times g \times Q \times H}{1000}, \text{ where P = 1000 kg/ m}^3$$

$$\therefore \quad N_o = \frac{\text{Shaft power in kw}}{\text{Water power in kw}} = \frac{P}{\left(\frac{P \times g \times Q \times H}{1000}\right)}$$

Where P = shaft power.

CLASSIFICATION OF HYDRAULIC TURBINES:-

The hydraulic turbines are calssified according to the type of energy available at the inlet of the turbine, direction of flow through the vanes, head at the inlet of the turbine and specific speed of the turbine. Thus, the followings are the important classification of the turbines.

(1) **According to the type of energy at inlet:-**

(a) Impluse turbine and (b) Reaction turbine

(2) **According to the direction of flow through runner:-**

(a) Tangential flow turbine

(b) Radial flow turbine.

(c) Axial flow turbine, and

(d) Mixed flow turbine.

(3) **According to the head at the inlet of turbine:-**

(a) High head turbine

(b) Medium head turbine, and

(c) Low head turbine.

(4) **According to the specific speed of the turbine:-**

(a) Low Specific speed turbine

(b) Medium Specific speed turbine, and

(c) High Specific Speed turbine.

If at the inlet of turbine the energy available is only kinetic energy, the trubine is know as **impulse turbine**. As the water flows over the vanes, the pressure is atmospheric from inlet to outlet of the turbine. If at the inlet of the turbine the water possesses kinetic energy as well as pressure energy, the turbine is known as **reaction turbine.** As the water flows through the runners, the water is under pressure and the pressure keeps on changing into kinetic energy. The runner is completely enclosed in an air-tight casing and the runner and casing is completely full of water.

If the water flows along the tangent of the runner the turbine is known as **tangential flow turbine**. If the water flows in the **radial direction** through runner, the turbine is called **radial flow turbine**, If the water flows from outwards to inwards radially, the turbine is known as **inward radial flow turbine** on the other hand if water flows radially from inward to outwards, the turbine is known as **outward radial flow turbine**. If the water flows through the runner along the direction parallel to the axis of rotation of the runner, the turbine is called **axial flow** turbine. If the water flows through the runner in the radial direction but leaves in the direction parallel to axis of rotation of the runner, the turbine is called **mixed flow turbine**.

PELTON WHEEL (OR TURBINE):-

The pelton wheel or pelton turbine is a tangential flow impulse turbine. The water strikes the bucket along the tangent of the runner. The energy available at the inlet of the turbine is only kinetic energy. The pressure at the inlet and outlet of the turbine is atmospheric pressure. This turbine is used for high heads. The main parts of the peltton turbine are :-

(a) Nozzle and flow regulating arrangement(spear)

(b) Runner and buckets

(c) Casing,

(d) Breaking jet.

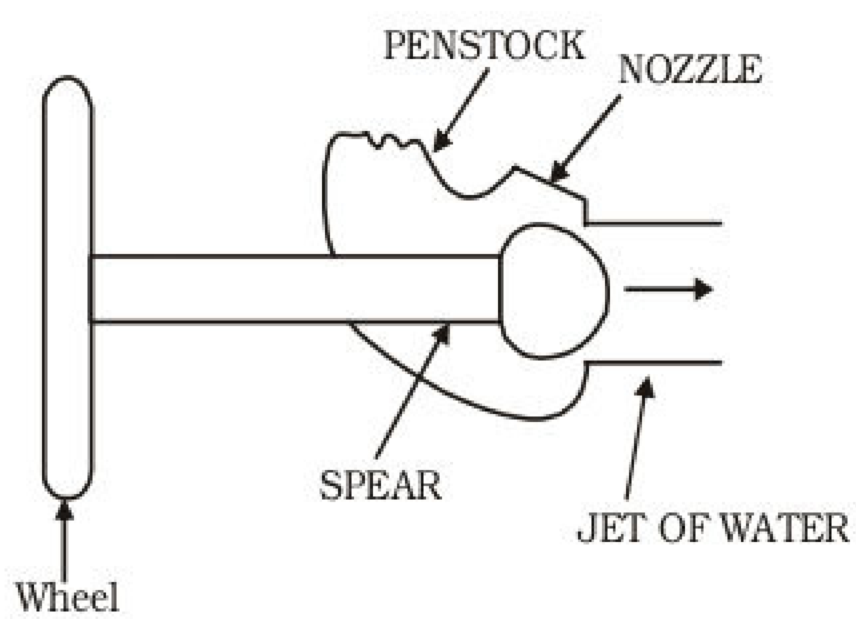


Fig:- **Nozzle with a spear to regulate flow:-**

Points to be remembered for pelton wheel:-

(i) The Velocity of the jet at inlet is given by

$$V_1 = C_r\sqrt{2}gH$$

Where C_v = Co-efficienct of velocity = 0.98 or 0.99

H = Net head on turbine.

(ii) The velocity of wheel (μ) is given by $u = \sqrt[\theta]{2gH}$

Where θ = Speed ratio. The value of speed ratio varies from 0.43 to 0.48.

(iii) The angle of deflection of the jet through buckets is taken as 165°, if no angle of deflection is given.

(iv) The mean diameter or the pitch diameter 'D' of the pelton wheel is given by:-

$$\mu = \frac{\pi DN}{60} \; or \; D = \frac{60u}{\pi N}.$$

(v) **Jet Ratio :-** It is defined as the ratio of the pitch diameter (D) of the pelton wheel to the diameter of the jet (d). It is denoted by 'm' and is given as :-

$$m = \frac{D}{d} (= 12 \text{ for Most cases})$$

(vi) **Number of bucket on a runner is given by**

$$Z = 15 + \frac{D}{2d} = 15 + 0.5m$$

Where m = Jet ratio.

(vii) **Number of jets:-** It is obtained by dividing the total rate of flow through the turbine by the rate of flow of water through a single jet.

RADIAL FLOW REACTION TURBINE:-

Radial flow turbine are those turbines in which the water flows in radial direction. Reaction turbines means that the water at the inlet of the turbine possesses kinetic energy as well as pressure energy.

FRANCIS TURBINE:-

The inward flow reaction turbine having radial discharge at outlet is known as Francis turbine.

Important Relations for Francis Turbine:-

(i) The ratio of width of the wheel to its diameter is given as n= B_1/D_1. The value of 'n' varies from 0.10 to 0.40.

(ii) The flow ratio is given as $V_{f1} / \sqrt{2gH}$ and varies from 0.15 to 0.30.

(iii) The speed ratio $= \mu_1 / \sqrt{2gH}$ varies from 0.6 to 0.9.

AXIAL FLOW REACTION TURBINE:-

If the water flows parallel to the axis of the rotation of the shaft, the turbine is known as axial flow turbine. And if the head at the inlet of the turbine is the sum of pressure energy and kinetic energy and during the flow of water through runner, a part of pressure energy is converted into kinetic energy, the turbine is known as reaction turbine.

SOME IMPORTANT POINTS FOR KAPLAN TURBINE:-

(i) The peripheral velocity at inlet and outlet are equal.

$\therefore \; \mu_1 = u_2 = \frac{\pi D_o N}{60}$, where D_o= outlet diameter of runner

(ii) Velocity of flow at inlet and outlet are equal

$$\therefore V_{f1} = V_{f2}$$

(iii) Area of flow at inlet = Area of flow at outlet

$$= \frac{\pi}{4}\left(D_a^2 - D_b^2\right)$$

❑❑❑