UNIT-I

Learning Material

Objective: To familiarize with the design principles of channels

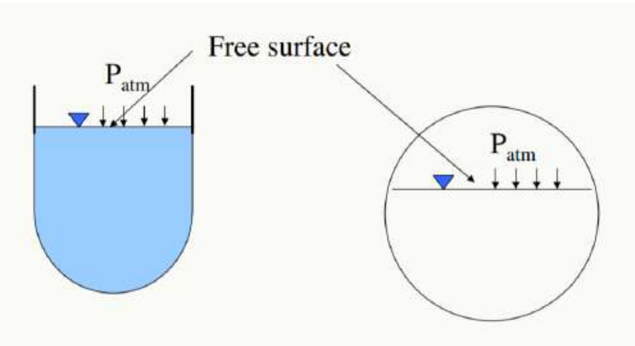
Outcome: Students will be able to design most economical section of open channel

Syllabus: Open Channel Flow - I

Type of channels – Velocity distribution – Energy and momentum correction factors – Chezy's, Manning's; and Bazin formulae for uniform flow – Most Economical sections. Critical flow: Specific energy-critical depth – computation of critical depth – critical sub-critical and super critical flows.

OPEN-CHANNEL FLOW

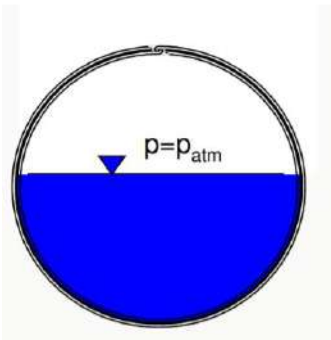
Open-channel flow is a flow of liquid (basically water) in a conduit with a free surface.That is a surface on which pressure is equal to local atmospheric pressure.



Classification of Open Channel Flows: Open-channel flows are characterized by the presence of a liquid-gas interface called the free surface.

Natural flows: rivers, creeks, floods, etc.

Human-made systems: fresh-water aqueducts, irrigation, sewers, drainage ditches, etc

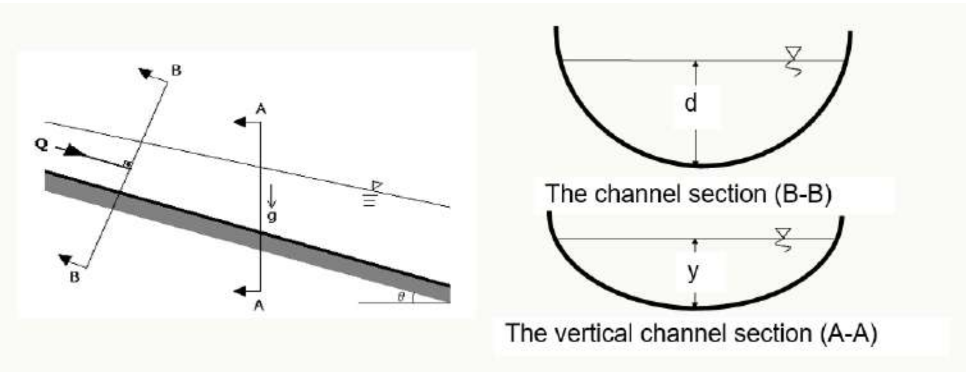


Types of Channels

- **Natural Channels:** It is one with irregular sections of varying shapes.developing in natural way..e.g., rivers, streams etc
- **Artificial Chlannel:** It is the one built artificially for carrying water for various purposes. e.g.. canals,
- **Open Channel:** A channel without any cover at the top. e.g., canals, rivers streams etc
- **Covered Channels:**A channel having cover at the top. e.g., partially filled conduits carrying water

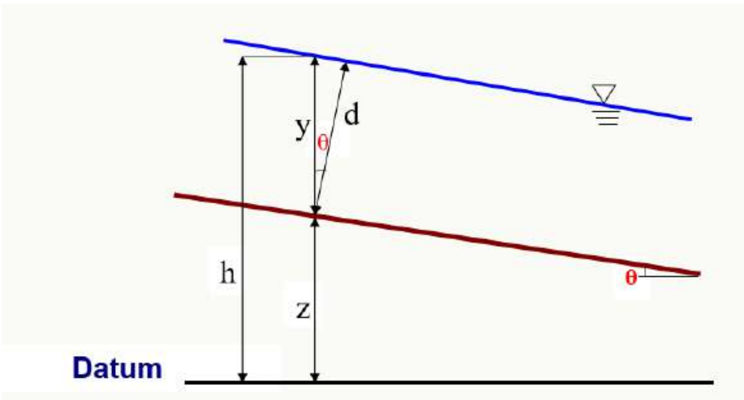
Channel Geometry:

- A channel built with constant cross section and constant bottom slope is Called a prismatic channel. Otherwise, the channel is non-prismatic.
- The channel section is the cross section of a channel taken normal to the direction of the flow.
- The vertical channel section is the vertical section passing through the lowest or bottom point of the channel section.



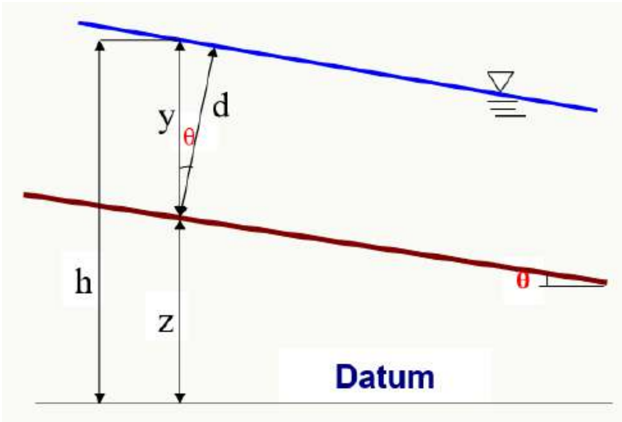
Geometric Elements of Channel Section:

THE DEPTH OF FLOW, y, is the vertical distance of the lowest point of a channel section from the free surface.



THE DEPTH OF FLOW SECTION, d, is the depth of flow normal to the direction of flow. $\theta$is the channel bottom slope

d = ycosθ. For mild-sloped channels y ≈ d.



Geometric Elements of Channel Section:

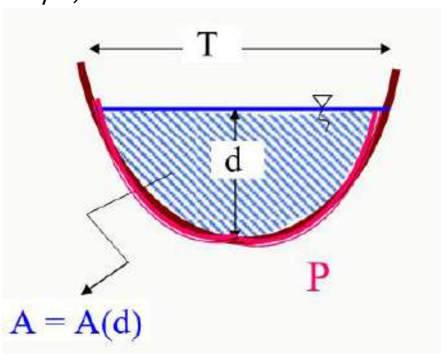
The top width, t,is the width of the channel section at the free surface.

The water area, a,is the cross-sectional area of the flow normal to the direction of flow.

The wetted perimeter, p,is the length of the line of intersection of the channel wetted surface with a cross- sectional plane normal to the direction of flow.

the hydraulic radius, r = a/p,is the ratio of the water area to its wetted perimeter.

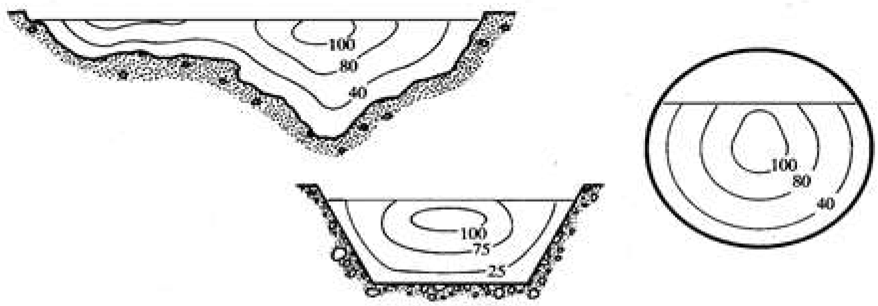
The hydraulic depth, d = a/t,is the ratio of the water area to the top width.



Velocity distribution in open channels

The measured velocity in an open channel will always vary across the channel section because of friction along the boundary. Neither is this velocity distribution usually axisymmetric (as it is in pipe flow) due to the existence of the free surface. It might be expected to find the maximum velocity at the free surface where the shear force is zero but this is not the case.

The figure below shows some typical velocity distributions across some channel cross sections. The number indicates percentage of maximum velocity.



Flow in open channels

Classification of flow in channels the flow in open channel is classified into the following types:

1. Steady flow and unsteady flow,
2. Uniform flow and non-uniform flow,
3. Laminar flow and turbulent flow, and
4. 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 open channel flow do not change with respect to time, the flow is said to be steady 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.

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.

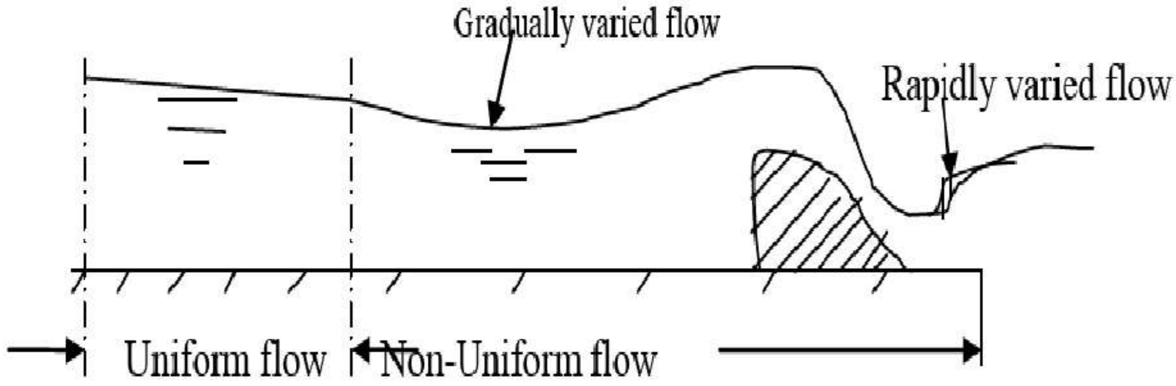
On the other hand, if for a given length of the channel, the velocity
of flow, depth of flow, etc. do not remain constant, the flow is said to be non-uniform flow

Non-uniform flow in open channels is also called varied flow, which is classified as:

Rapidly varied flow (R.V.F), and Gradually varied flow (G.V.F).

Rapidly Varied Flow (R.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 any obstruction in the path of flow of water, the level of water rises above the obstruction and then falls and again rises over a short length of the channel



Gradually Varied Flow (G.V.F) 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 Reynolds number (Re) is less than 500.

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

Sub-critical, Critical and Super Critical Flow:

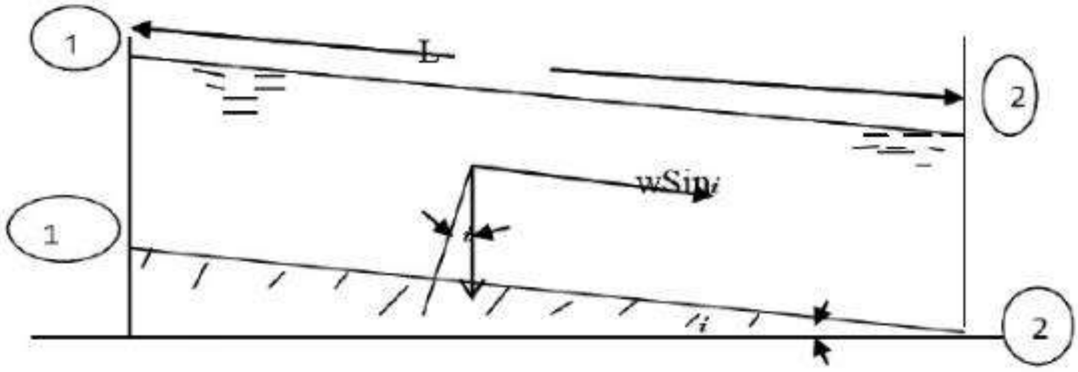
- The flow in open channel is said to be sub-critical if the Froude number (F_e) is less than 1.0 .Sub-critical flow is also called tranquil or streaming flow
- The flow is called critical if F_e = 1.0 , if Fe > 1.0 the flow is called super critical or shooting or rapid or torrential.

$$F_e = \frac{V}{\sqrt{gD}}$$

Discharge through open channel by Chezy's formula:

Consider flow of water in a channel as shown in Fig.

As the flow is uniform, it means the velocity, depth of flow and area of flow will be constant for a given length of channel.



Consider sections 1-1 and 2-2.
Let L = Length of channel,
A = Area of flow of water,
i = Slope of bed,
V = Mean velocity of flow of water,
P = Wetted perimeter of the cross-section,
f = Frictional resistance per unit velocity per unit area
The weight of water between sections 1-1 and 2-2
W = Specific weight of water x volume of water $= w \times A \times L$
Component of W along direction of flow=W x sin i = $wAL \sin i$
Frictional resistance against motion of water = f x surface area x $(\text{velocity})^n$
The value of n is found experimentally to be equal to 2 and the surface area =P x L
Frictional resistance against motion $= f \times P \times L \times V^2$
The forces acting on the water between sections 1-1 and 2-2 are
1. Component of weight of water along the direction of flow,
2. Frictional resistance against flow of water,
3. Pressure force at section 1-1, 4. Pressure force at section 2-2,
As the depths of water at the sections 1 -1 and 2-2 are the same, the pressure forces on these two sections are the same and acting in opposite directions. Hence they cancel each other

In the case of uniform flow, the velocity of flow is constant for a given length of the channel. Hence there is no acceleration acting on the water. Hence the resultant force acting in the direction of flow must be zero.
Resolving all forces in the direction of flow, we get

$$w AL \sin i - f \times P \times L \times V^2 = 0$$

$$V^2 = \frac{w\, A\, L \sin i}{f\, x\, P\, x\, L} = \frac{w}{f} \times \frac{A}{P} \times \sin i$$

$$\text{Or } V = \sqrt{\frac{w}{f}} \times \sqrt{\frac{A}{P}\sin i}$$

$$But\ \frac{A}{P} = m$$

$\sqrt{\frac{w}{f}}$= C= Chezy's constant

Substituting the values of $\frac{A}{P}$ *and* $\sqrt{\frac{w}{f}}$

$V = c\sqrt{msini}$

Discharge, Q = Area x velocity= A x C$\sqrt{mi}$

Table : Selected values of C.

Type of channel bed	Mean value of C
Smooth cement	90
Well-laid brickwork	70
Cement concrete	70
Natural channel (in good condition)	35
Natural channel (in bad condition	25

The value of C can also be estimated using the Ganguillet and Kutter formula, which has been developed based on measurements in open channels of various types.

Kutter's formula: C= $\dfrac{23+\frac{0.00155}{i}+\frac{1}{N}}{1+\left(23+\frac{0.00155}{i}\right)\frac{N}{\sqrt{m}}}$

Where N= Roughness coefficient

i = slope of the bed

The Manning equation

Many studies have been made on the evaluation of C for different natural and manmade channels. Today most practicing engineers use some form of these relationships to give C:

$$C = \frac{R^{1/6}}{n}$$

This is known as Manning's formula and the n as Manning's n.

Substituting Chezy's equation in to the above formula gives velocity of uniform flow: $V = \frac{R^{2/3} S_o}{n}$

Bazin's Formula:

A French hydraulic engineer H. Bazin (1897) proposed the following empirical formula for *C*

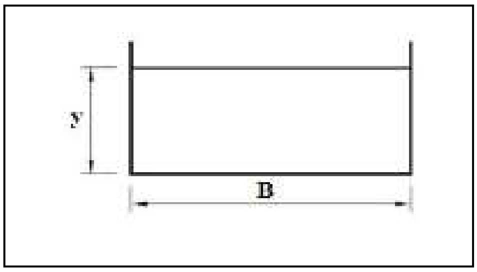
$$c = \frac{157.6}{1.81 + \frac{m}{\sqrt{R}}}$$

Most Economical Sections:

A channel section is considered to be most economical or most efficient when it passes a maximum discharge for given cross section area, resistance coefficient and bottom slope. From Chézy formula and Manning formula for a certain value of slope and surface roughness velocity is maximum, when the hydraulic radius is maximum and if we take area as constant hydraulic radius is maximum if the wetted perimeter is minimum.

Most Economical Rectangular Channel:

Consider a rectangular section of channel as shown.



Let B = width of channel, y = depth of flow.

Area of flow, A = B x y

Wetted perimeter, P = 2y + B

$$B = \frac{A}{y}$$

Substituting the value of B in equation we get

$$P = \frac{A}{y} + 2y$$

Assuming the area to be constant Eq. 1.3 can be differentiated with respect to y and equated to zero for obtaining the condition for minimum p

$$\frac{dp}{dy} = \frac{-A}{y^2} + 2 = 0$$

From this

$$A = 2y^2 = By$$

$$B = 2y \text{ or } y = B/2$$

So hydraulic Radius, $R = \frac{A}{P} = \frac{By}{B+2y}$

Substituting the value of B=2y

$$R = \frac{2y^2}{2y+2y} = \frac{y}{2}$$

Thus it can be seen that rectangular channel section will be most economical or efficient when either the depth of flow is equal to half of the bottom width, or hydraulic radius equal to half of the depth of flow.

Most Economical Trapezoidal channel:

The trapezoidal section of a channel will be most economical, when wetted perimeter is minimum. Consider a trapezoidal section of channel

(i) The side slope is given as 1 vertical to n horizontal.

Area of flow, A = $\frac{BC+AD}{2} X\, d$

by solving A = (b+nd) x d

$\frac{A}{d} = b + nd$

now wetted perimeter, p = AB+BC+CD = BC+2CD

$= b+2d\sqrt{n^2+1}$

substituting the value of b from equation (ii), we gt

$P = \frac{A}{d} - nb+2d\sqrt{n^2+1}$

For most economical section P should be minimum or $\frac{dp}{d(d)} = 0$

Differentiating equation (iii) with respective to 'd' and equating it equal to zero, we get

$$\frac{dp}{d(d)}[\frac{A}{d} - nb + 2d\sqrt{n^2+1}] = 0$$

$$\frac{A}{d^2} + nb = 2d\sqrt{n^2+1}$$

the value of A from equation (i) in the above equation

$$\frac{b+2nd}{2} = d\sqrt{n^2+1}$$

$$\frac{b+2nd}{2} = \text{half of top width}$$

$d\sqrt{n^2+1}$ = CD = one of the sloping edge

ii) Hydraulic mean depth:

Hydraulic mean depth, m = $\frac{A}{P}$

Value of A = $\frac{BC+AD}{2} X\, d$

Value of P = $b+2d\sqrt{n^2+1}$ = 2(b+nd)

By solving m = $\frac{d}{2}$

Hence for a trapezoidal section to be most economical hydraulic mean depth must be equal to half the depth of flow.

Hence the conditions for the most economical trapezoidal section are :

1. $\frac{b+2nd}{2} = \mathbf{d}\sqrt{n^2+1}$
2. **m** $= \frac{d}{2}$

Specific energy (E) and critical depth(y_c):

Specific energy of flow at any channel is defined as the energy per unit weight of the water measured with respective to channel bottom as the datum. Thus specific energy E at any section is the sum of depth of flow at that section and the velocity head.

$$E = \frac{V^2}{2g} + y = \frac{Q^2}{2gA^2}$$

So from the equation for E, it can be seen that for a given channel section and discharge Q, the specific energy is depth of flow only. So if we draw a graph for a channel as shown in fig 1.4, it can be observed that there is one point 'C', where specific energy is minimum. The depth of flow at which the specific energy is minimum is called as critical depth (y_c)

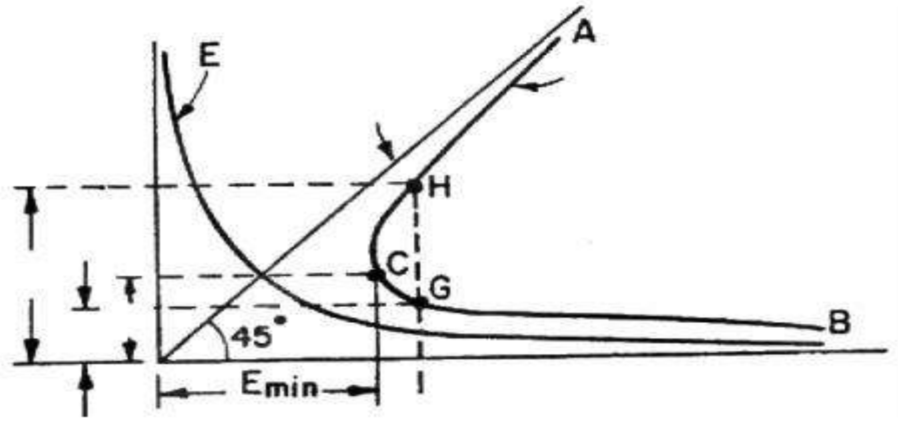


Fig 4: Specific Energy Curve

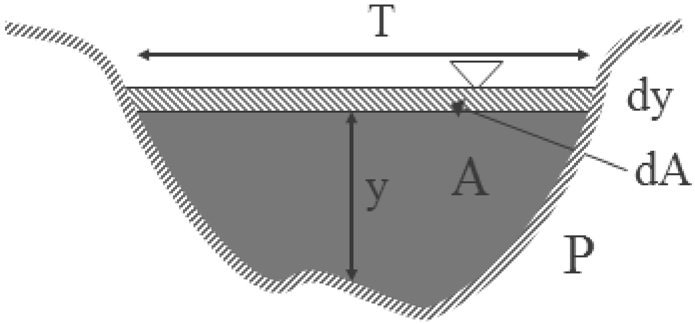
1. The depth of flow at point C is referred to as critical depth, y_c. It is defined as that depth of flow of liquid at which the specific energy is minimum. The flow that corresponds to this point is called critical flow (Fr = 1.0).

2. For values of E greater than E_{min} , there are two corresponding depths. One depth is greater than the critical depth and the other is smaller then the critical depth. These two depths for a given specific energy are called the alternate depths.

3. If the flow depth $y > y_c$, the flow is said to be sub-critical ($Fr < 1.0$).

4. If the flow depth $y < y_c$, the flow is said to be super-critical ($Fr > 1.0$). In this case Es increases as y increases.

Determination of Critical Depth:



$$E = y + \frac{Q^2}{2gA^2}$$

For a given discharge condition the minimum specific energy is obtained by differentiating 'E' with respective to 'y'.

$$\frac{dE}{dy} = 1 - \frac{Q^2}{gA^3}\frac{dA}{dy} = 0$$

For the given condition dA= T dy. so substituting that in the above equation gives

$$1 = \frac{Q^2 T_c}{gA_c^3}$$

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

Since V= Q/A and D= A/T, the above equation may be written as

i) For a rectangular channel:

$Q = qb,\ T = b\ and\ A = by$ this equation becomes

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} \quad \text{as } V_c y_c = q$$

$$V_c = \sqrt{g y_c}$$

Substituting this in to the specific energy equation

$$E_{sc} = y_c + \frac{V_c^2}{2g} = y_c + \frac{y_c}{2}$$

$$y_c = \frac{2}{3} E_{sc}$$

UNIT-II
Learning Material

Objective: To impart knowledge on Non-Uniform flow in open channels.

Outcome: determine the hydraulic jump for energy dissipation at the downstream of irrigation structures

Syllabus: UNIT - II: Open Channel Flow - II
Non uniform flow-Dynamic equation for G.V.F., Mild, Critical, Steep, horizontal and adverse slopes-surface profiles- Rapidly varied flow, hydraulic jump, energy dissipation, boundary layer introduction

Non uniform Flow in Channels:
The dynamic equation of gradually varied flow is an expression giving the relationship between the water surface slope and other characteristics of flow. The following assumptions are made in the derivation of the equation:
(1) The flow is steady: depth and other hydraulic characteristics at a particular section do not change with time.
(2) The streamlines are practically parallel: which is true when the variation in depth along the direction of flow is very gradual. Thus the hydrostatic distribution of pressure is assumed over the section.
(3) The loss of head at any section, due to friction, in gradually varied flow is equal to that in the corresponding uniform flow with the same depth and flow characteristics. According to this assumption, the uniform flow formulas, such as Manning's formula, may be used to calculate the slope of the energy line (S_f) in gradually varied flow as well.
(4) The slope of the channel is small: for small slopes, the depth of flow is approximately equal to the depth of flow section
(5) The channel is prismatic, i.e.; it has constant shape, size, slope and alignment.
(6) The velocity distribution across the section is fixed.
(7) The roughness coefficient (Manning's coefficient) is constant in the reach. Also, it does not depend upon the depth of flow.
A non-uniform flow is characterized by a varied depth and a varied mean flow velocity. If the bottom slope and the energy line slope are not equal, the flow depth will vary along the channel, either increasing or decreasing in the flow direction. Physically, the difference between the component of weight and the shear forces in the direction of flow produces a change in the fluid momentum which requires a change in velocity and, from continuity considerations, a change in depth. Whether the depth increases or decreases depends on various parameters of the flow, with many types of surface profile configurations possible
Equation of gradually-varied flow
In addition to the basic gradually-varied flow assumption, we further assume that the flow occurs in a prismatic channel, or one that is approximately so, and that the slope of the energy grade line can be evaluated from uniform flow formulas with uniform flow resistance coefficients, using the local depth as though the flow were locally uniform. The total energy head at any cross-section is

Differentiating each term with respect to x, where x is measured along channel bottom

In the above equation, dH/dx is the slope of energy gradient and hence

dz/dx is the slope of bottom bed and hence

dA/dy= T. Hence substitution of these terms in above differential equation helds

Classification of flow profiles

Flow profiles are classified by the slope of the channel (So), yn, and yc. There are five slope classifications designated by the letters C, M , S, A, and H (critical, mild, steep, adverse, and horizontal) respectively.

Mild (M) if $y_n > y_c$

Steep (S) if $y_n < y_c$

Critical (C) if $y_n = y_c$

Adverse (A) if $S_o < 0$ (if slope is positive in the downstream direction)

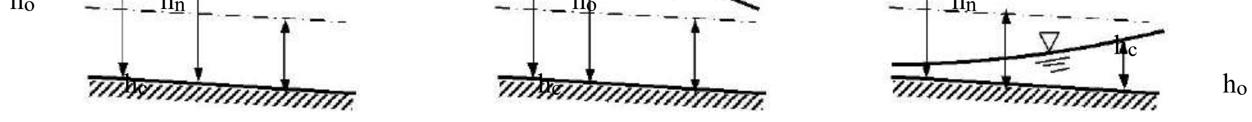
Horizontal (H) if $S_o = 0$

The conditions at which flow in an open channel can take place and the possible relationships between the *observed depth* y_o, the *normal depth* at which flow is uniform y_n, and the *critical depth* y_c are illustrated in Fig. 1.5. It is evident from this figure that there are three zones of channel depths at which flow can be observed:

Zone 1, with y_o greater than h_n and h_c (i.e. $y_o > y_n > y_c$)

Zone 2, with y_o between h_n and h_c (i.e. $y_n > y_o > y_c$)

Zone 3, with y_o less than y_n and y_c (i.e. $y_n > y_c > y_o$)



$y_o > y_n > y_c$	$y_n > y_o > y_c$	$y_n > y_c > y_o$

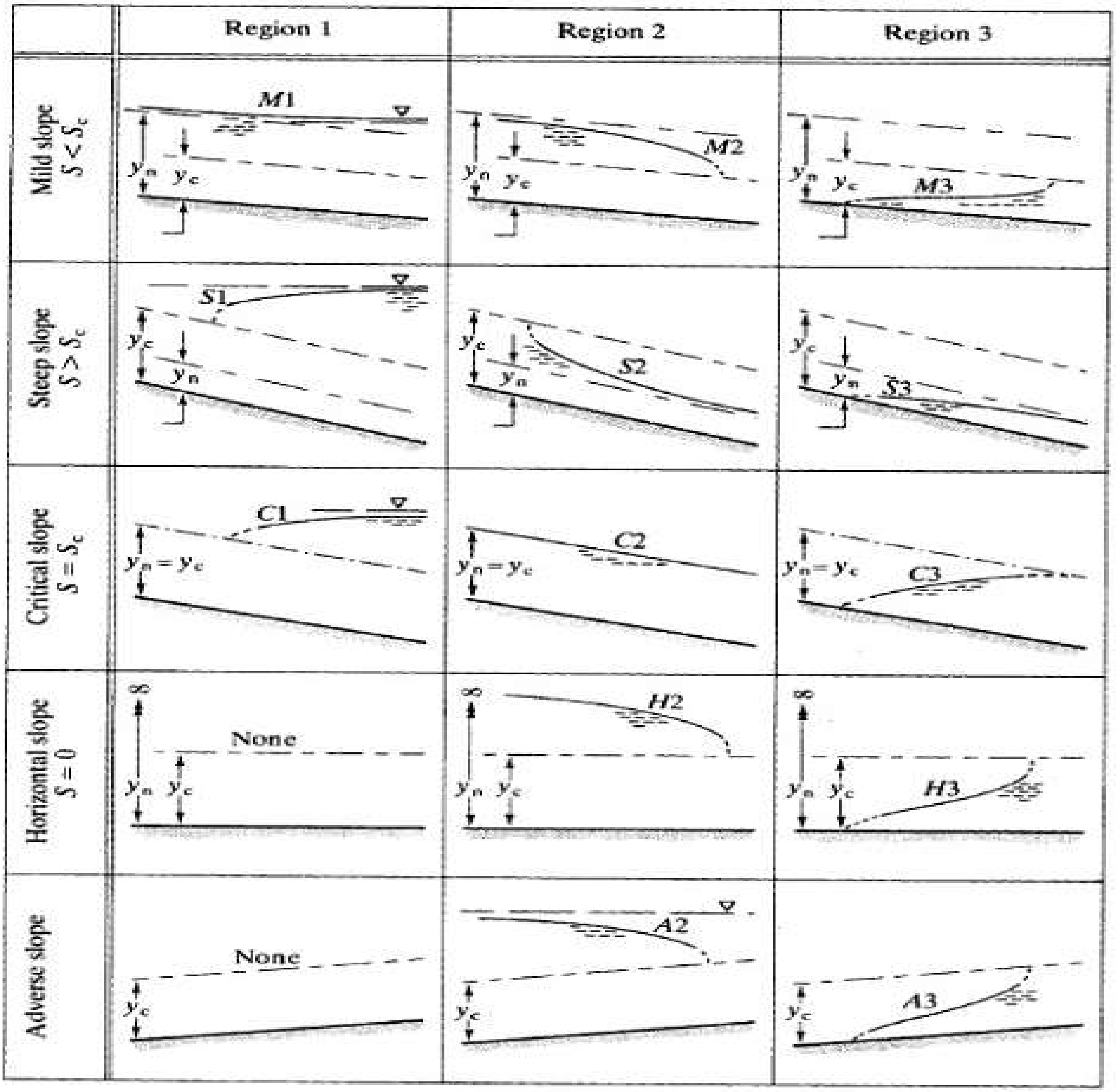
Three zones of channel depths

The relative bottom slope defines whether uniform flow is subcritical or supercritical. Determine the associated Froude-number Fr_e.

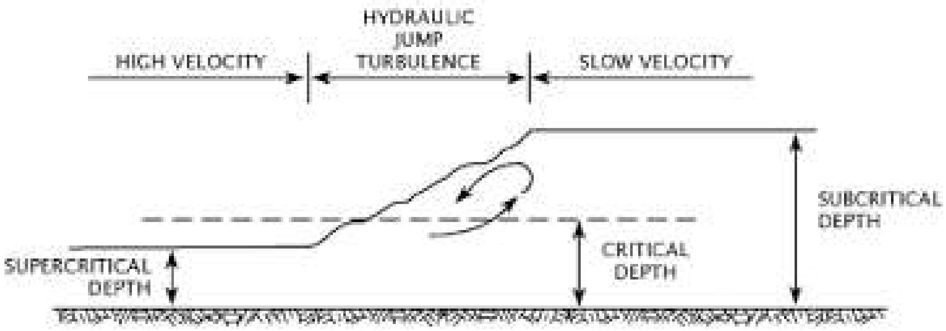
The Hydraulic jump

The hydraulic jump is an important feature in open channel flow and is an example of rapidly varied flow. A hydraulic jump occurs when a super-critical flow and a sub-critical flow meet. The jump is the mechanism for the two surfaces to join. They join in an extremely turbulent manner which causes large energy losses.

Because of the large energy losses the energy or specific energy equation cannot be use in analysis, the momentum equation is used instead.



Profile slopes



Hydraulic Jump

Resultant force in x- direction = F_1 - F_2

Momentum change = $M_2 - M_1$

$$F_1 - F_2 = M_2 - M_1$$

Or for a constant discharge

$$F_1 + M_1 = F_2 + M_2 = \text{constant}$$

For a rectangular channel this may be evaluated using

$$F_1 = \rho g \frac{y_1}{2} y_1 b \qquad F_2 = \rho g \frac{y_2}{2} y_2 b$$

$$M_1 = \rho Q V_1 \qquad M_2 = \rho Q V_2$$

$$= \rho Q \frac{Q}{y_1 b} \qquad = \rho Q \frac{Q}{y_2 b}$$

Substituting for these and rearranging gives

$$y_2 = \frac{y_1}{2}\left(\sqrt{1+8F_{N1}^2} - 1\right)$$

$$or$$

$$y_1 = \frac{y_2}{2}\left(\sqrt{1+8F_{N2}^2} - 1\right)$$

So knowing the discharge and either one of the depths on the upstream or downstream side of the jump the other - or *conjugate depth* - may be easily computed.

More manipulation with the above equation and the specific energy give the energy loss in the jump as

$$\Delta E = \frac{(y_2 - y_1)^3}{4 y_1 y_2}$$

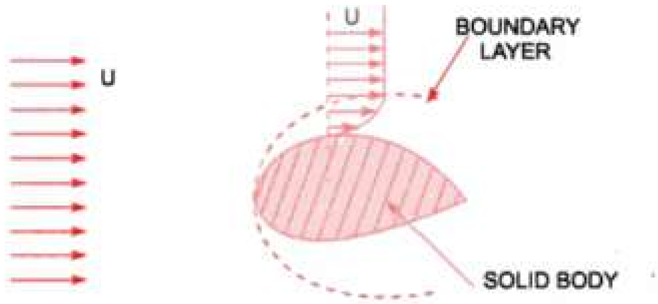
These are useful results and which can be used in gradually varied flow calculations to determine water surface profiles.

In summary, a hydraulic jump will only occur if the upstream flow is super-critical. The higher the upstream Froude number the higher the jump and the greater the loss of energy in the jump.

Boundary layer theory:

When a real fluid flows past 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 same as that of the boundary. If the boundary is stationary, the velocity of fluid at the boundary will be zero. Farther away from the boundary, the velocity will be higher and as a result of this variation of velocity, the velocity gradient du/dy will exist. The velocity of fluid increases from zero velocity on the stationary boundary to free-stream velocity (U) of the fluid in the direction normal to the boundary.

This variation of velocity from zero to free-stream velocity in the direction normal to the boundary takes place in a narrow region in the vicinity of solid boundary. This narrow region of the fluid is called boundary layer. The theory dealing with boundary layer flows is called boundary layer theory. According to boundary layer theory, the flow of fluid in the neighbor-hood of the solid boundary may be divided into two regions.



UNIT – III

Objective: To understand the working principles of hydraulic machines

Outcome: To learn types and working of hydraulic machines

Syllabus Basics of Turbo Machines:

Hydrodynamic force of jets on stationary and moving flat, inclined and curved vanes, jet striking centrally and at tip, velocity triangles at inlet and outlet

Impact of jets

Definition: The liquid comes out in the form of jet from the outlet of a nozzle, which is fitted to a pipe through which is flowing under pressure.

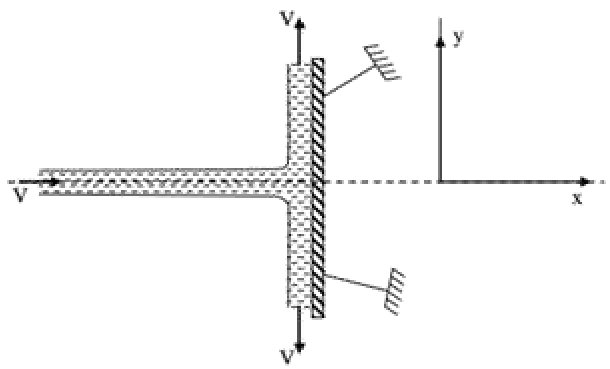
Jet: it is a stream of fluid issuing from a nozzle with high velocity and hence a high kinetic energy. When a jet impinges on a plate or vane, it exerts a force on it (due to change in momentum). This force can be evaluated by using Impulse momentum principle.

Here we study about the application of the impulse momentum equation for evaluating the hydrodynamic force on the stationary and moving vanes.

1. Force exerted by the jet on a stationary plate when
   (a) Plate is vertical to the jet
   (b) Plate is inclined to the jet
   (c) Plate is curved
2. Force exerted by the jet on a moving plate when
   (a) Plate is vertical to the jet
   (b) Plate is inclined to the jet
   (c) Plate is curved

Case 1(a): Force exerted by the jet on a stationary vertical plate:

Consider a jet of water coming out from the nozzle, strikes a flat vertical plate.
Let V = velocity of jet,   d = diameter of the jet,   a = area of cross-section of the jet.



The jet after striking the plate, will move along the plate. But the plate is at right angles to the jet. Hence the jet after striking, will get deflected through 90^0. Hence the component of the velocity of jet in the direction of jet, after striking will be zero.

The force exerted by the jet on the plate in the direction of jet,

F_x = *Rate of change of momentum in the direction of force*

$$=\left({}^{Mass}/_{sec}\right).\,(velocity\ of\ jet\ before\ striking - velocity\ of\ jet\ after\ striking)$$

$$= \rho aV[V-0]$$

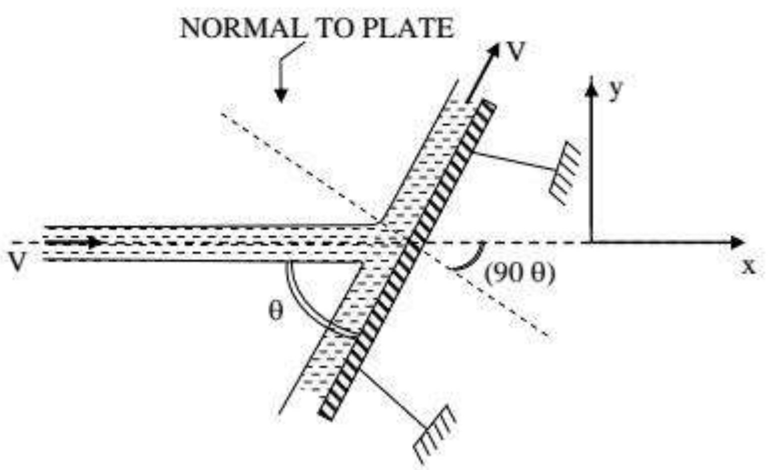
$$= \rho aV^2$$

Case 1(b): Force exerted by a jet on stationary inclined flat plate:

Let a jet of water, coming out from the nozzle, strikes an inclined flat plate

Let V = velocity of jet in the direction of x, Θ = angle between the jet and plate,

If the plate is smooth and if it is assumed that there is no loss of energy due to impact of the jet, then jet will move over the plate after striking with velocity equal to initial velocity.



Force exerted by the jet on the plate in the direction normal to the plate = F_n

F_n=

$$\left({}^{Mass}/_{sec}\right).(velocity\ of\ jet\ before\ striking\ in\ the\ normal\ direction - velocity\ of\ jet\ after\ striking\ in\ the\ normal\ direction)$$

$$= \rho aV[Vsin\theta\ - 0]$$

$$= \rho aV^2 sin\theta$$

This force can be resolved into two components, one in the direction of the jet and other perpendicular to the direction of flow.

F_x = component of F_n in the direction of flow

$= F_n \cos(90-\Theta)$

$= \rho a V^2 \sin^2\Theta$

F_y = component of F_n in perpendicular to flow

$= F_n \sin(90-\Theta)$

$= \rho a V^2 \sin\Theta.\cos\Theta$

Case 1(c): Force exerted by a jet on stationary curved plate:

Let a jet of water strikes a fixed curved plate at the centre, the jet after striking

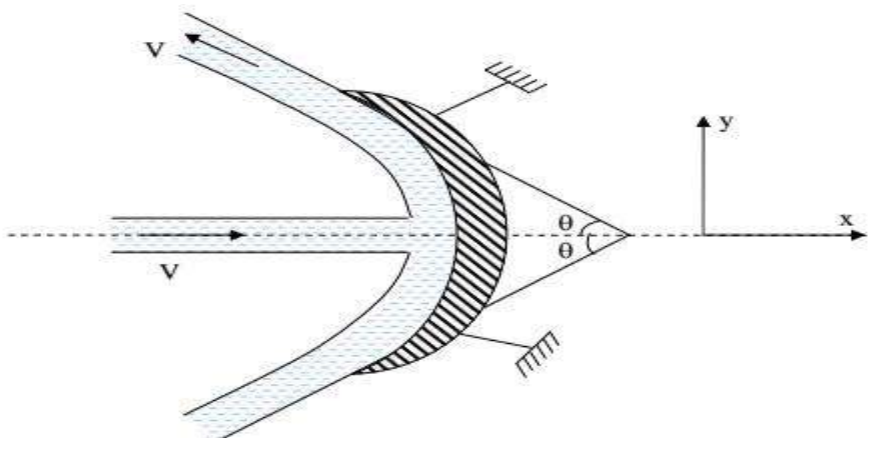
The plate comes out with the same velocity if the plate is smooth and there is no loss of energy due to impact of the jet, in the tangential direction of the curved plate. The velocity at outlet of the plate can be resolved into two components, one in the direction of jet and other perpendicular to the direction of the jet.

Component of velocity in the direction of jet = $-V\cos\Theta$.
Component of velocity perpendicular to the jet = $V\sin\Theta$.
Force exerted by the jet in the direction of jet,

$$F_x = \rho a V[V-(-V\cos\Theta)]$$
$$= \rho a V^2[1+\cos\Theta]$$

Force exerted by the jet in the direction perpendicular to jet,

$$F_y = \rho a V[0-(V\sin\Theta)]$$
$$= -\rho a V^2 \sin\Theta$$



Let the jet strikes the curved fixed plate at one end tangentially and curved plate is symmetrical about x-axis. Then the angle made by the tangents at the two ends of the plate will be same.

Θ = angle made by jet with x-axis at inlet tip of the curved plate.

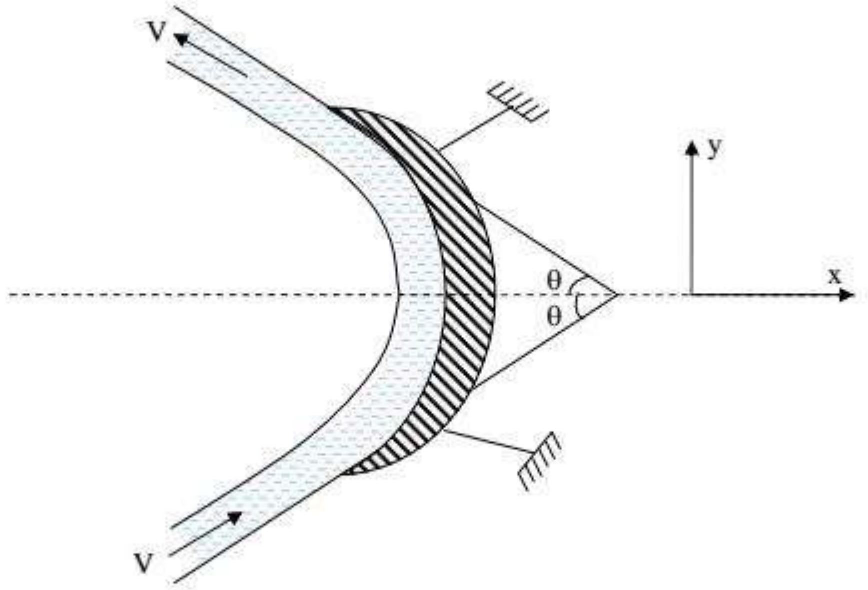
If the plate is smooth and loss of energy due to impact is zero, then the velocity of water at the outlet tip of the curved plate will be equal to V. the forces exerted by the jet of water in the direction of x and y are

$F_x = \rho aV[V\cos\Theta - (-V\cos\Theta)]$

$= 2\rho aV^2 \cos\Theta$

$F_y = \rho aV[V\sin\Theta - V\sin\Theta]$

$= 0$



Case 1 (d):Jet strikes the curved plate at one end tangentially when the plate is unsymmetrical:

When the curved plate is unsymmetrical about x-axis, then angle made by the tangents drawn at the inlet and outlet tips of the plate with x-axis will be different.

Let Θ =angle made by tangent at inlet tip with x-axis

Φ = angle made by tangent at outlet tip with x-axis

The two components of the velocity at inlet are

$V_{1x} = V\cos\Theta$, $V_{1y} = V\sin\Theta$

The two components of the velocity at outlet are

$V_{2x} = -V\cos\Phi$, $V_{2y} = V\sin\Phi$

The force exerted by the jet of water in the directions of x and y are

$F_x = \rho aV[V\cos\Theta-(-V\cos\Phi)]$

$= \rho aV^2[\cos\Theta+\cos\Phi]$

$F_y = \rho aV[V\sin\Theta- V\sin\Phi]$

$= \rho aV^2[\sin\Theta- \sin\Phi]$

Force exerted by a jet on moving plates:

The following cases of the moving plates will be considered:

1. Flat vertical plate moving in the direction of the jet and away from the jet
2. Inclined plate moving in the direction of the jet
3. Curved plate moving in the direction of the jet or in the horinzontal direction

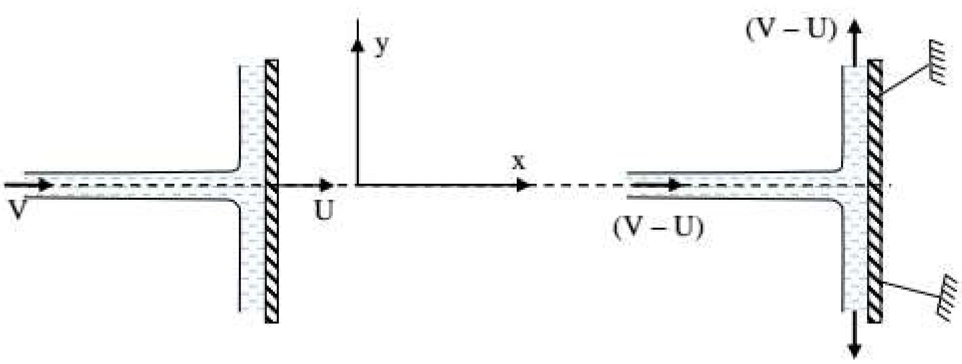
Case 2(a): Force on flat vertical plate moving in the direction of jet:

Let V = velocity of the jet

A = area of cross section of the jet

u= velocity of the flat plate

in this case, the jet does not strike the plate with a velocity, but it strikes with relative velocity, which is equal to the velocity of jet of water minus the velocity of the plate,



hence relative velocity of the jet with respect to plate = (V-u)

mass of water striking the plate per sec =ρa[V-u]

force exerted by the jet on the moving plate in the direction of the jet,

$$F_x = \rho a[V-u][(V-u)-0]$$

$$= \rho a[V-u]^2$$

Work done per sec by the jet on the plate

= force. (distance in the direction of force/time)

$= F_x.u$

$= \rho a[V-u]^2.u$

Case 2(b): Force on the inclined plate moving in the direction of the jet:

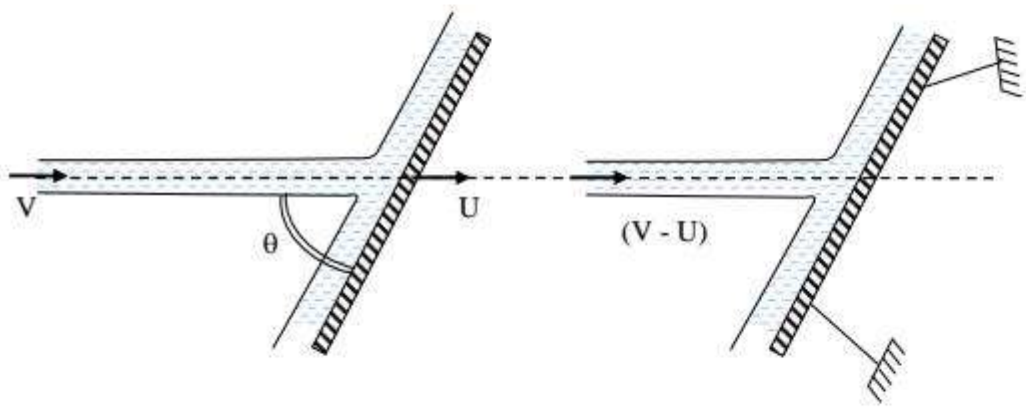
Let a jet of water strikes an inclined plate, which is moving with a uniform velocity in the direction of the jet.

V = velocity of the jet

a= area of cross section of the jet

u= velocity of the flat plate

Ɵ = angle between jet and plate



Relative velocity of the jet with respect to plate = (V-u)

Velocity with which jet strikes = (V-u)

Mass of water striking the plate per sec =ρa[V-u]

If the plate is smooth and loss of energy due to impact of the jet is assumed zero, the jet of water will leave the inclined plate with a velocity equal to (V-u)

Force exerted by the jet of water on the plate in the direction normal to the plate is given by

$F_n = \rho a[V-u][(V-u)\sin\theta - 0]$

$= \rho a[V-u]^2 \sin\theta$

This normal force is resolved into components namely F_x and F_y in the direction of jet and perpendicular to the direction of the jet respectively.

$F_x = F_n \sin\theta = \rho a[V-u]^2 \sin^2\theta$

$F_y = F_n \cos\theta = \rho a[V-u]^2 \sin\theta . \cos\theta$

Work done per sec by the jet on the plate

$= F_x . u$

$= \rho a[V-u]^2 \sin^2\theta . u$

Case 2(c): Force on the curved plate when the plate is moving in the direction of jet:

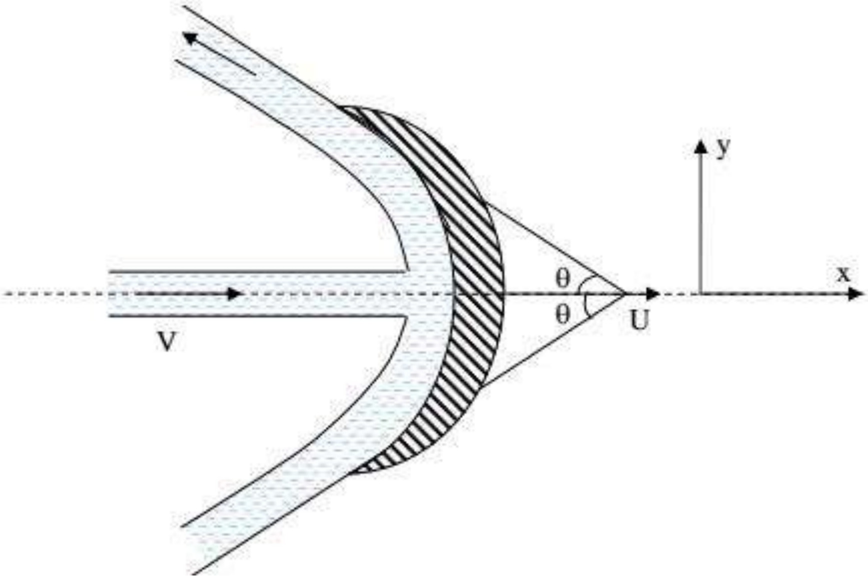
Let a jet of water strikes a curved plate, which is moving with a uniform velocity in the direction of the jet.

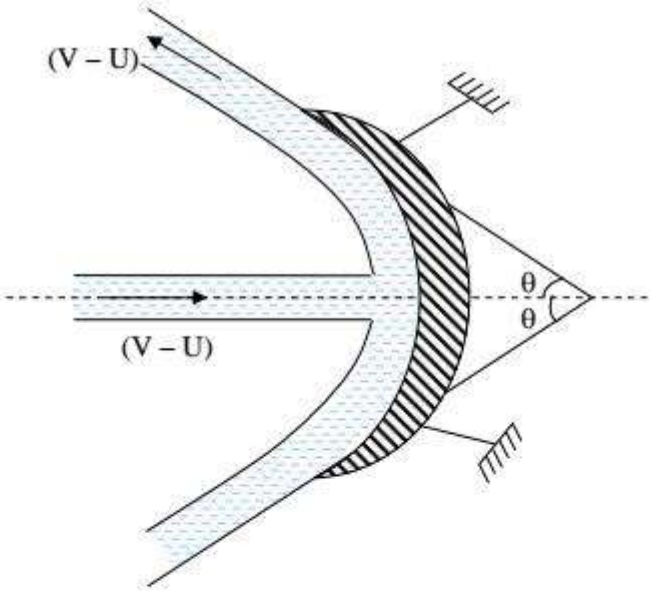
V = velocity of the jet

a= area of cross section of the jet

u= velocity of the flat plate

Ө = angle between jet and plate





Relative velocity of the jet with respect to plate = (V-u)

Velocity with which jet strikes = (V-u)

If the plate is smooth and loss of energy due to impact of the jet is assumed zero, the jet of water will leave the plate with a velocity equal to (V-u)

This velocity can be resolved into two components, one in the direction of the jet and other perpendicular to the direction of jet.

Component of the velocity in the direction of jet = -(V-u)cosθ

-ve sign is taken as the outlet, the component is in the opposite of the jet.

Component of the velocity in the direction perpendicular to the direction of the jet

= (V-u)sinθ

Mass of water striking the plate per sec =ρa[V-u]

Force exerted by the jet of water on the plate in the direction of the jet

$$= \rho a[V-u][\,(V-u)-(\,-(V-u)\cos\theta)]$$

$$= \rho a[V-u]^2[1+\cos\theta]$$

Work done per sec by the jet on the plate = $F_x.u$

$$= \rho a[V-u]^2 u[1+\cos\theta]$$

Case 2(d): Force exerted by a jet of water on an unsymmetrical moving curved plate when jet strikes tangentially at one of the tips:

A jet of water striking a moving curved plate tangentially, at one of its tips. As the jet strikes tangentially, the loss of energy due to impact of the jet will be zero. In this case as plate is moving, the velocity with which jet of water strikes is equal to the relative velocity of the jet with respect to the plate. Also as the plate is moving in different direction of the jet, the relative velocity at inlet will be equal to the vector difference of the velocity of jet and velocity of the plate an inlet.

Let

V_1 = velocity of the jet at inlet

U_1 = velocity of the plate at inlet

V_{f1} = relative velocity of jet and plate at inlet

$\alpha = angle\ between\ the\ direction\ of\ the\ jet\ and\ direction\ of$

$motion\ of\ the\ plate, also\ called\ guide\ balde\ angle$

ɵ = angle made by the relative velocity with the direction of motion at inlet also called vane angle at inlet

V_{w1} and V_{f1} = the components of the velocity of the jet V1, in the direction of motion and perpendicular to the direction of motion of the vane respectively.

V_{w1} = it is also known as velocity of whirl at inlet

V_{f1} = it is also known as velocity of flow at inlet

V_2 = velocity of the jet, leaving the vane or velocity of jet at outlet of the vane

U_2 = velocity of the vane at outlet

V_{r2} = relative velocity of the jet with respect to the vane at outlet

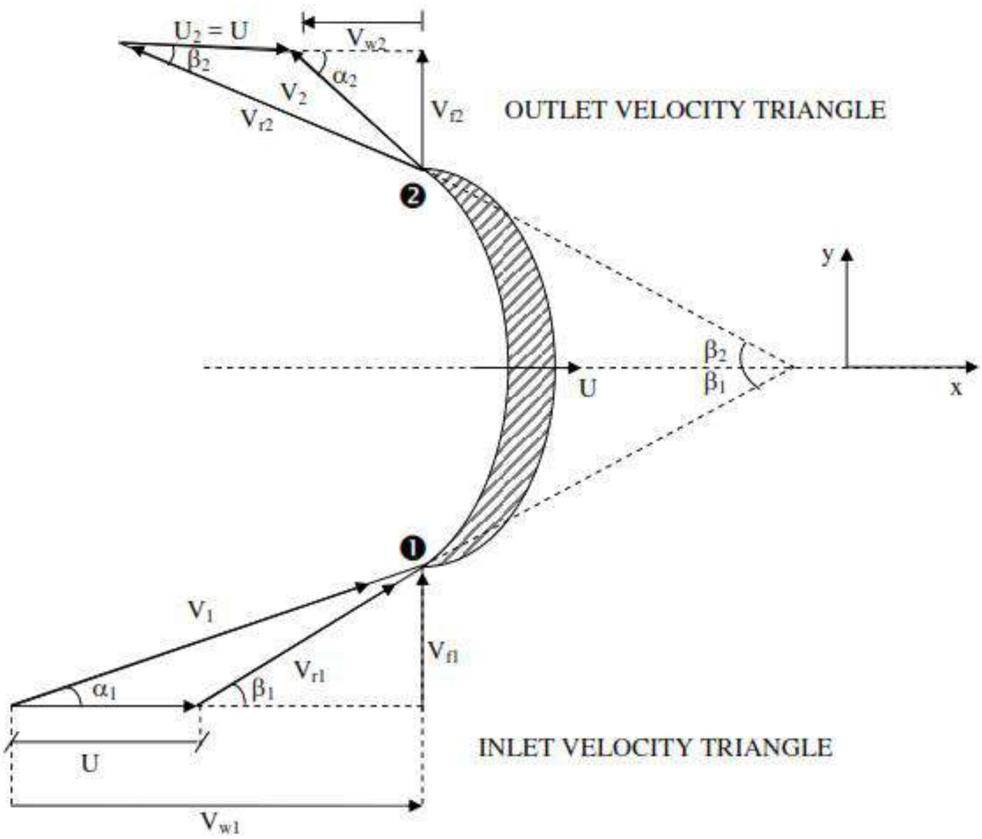
β= angle made by the velocity V_2 with the direction of motion of the vane at outlet

Φ= angle made by the relative velocity with the direction of motion of the vane at outlet and also called vane angle at outlet

V_{w2} and V_{f2} = components of the velocity V_2, in the direction of motion of vane and perpendicular to the direction of motion of vane at outlet.

V_{w2} = it is also called the velocity of whirl at outlet

V_{f2} = velocity of flow at outlet.



Mass of water striking vane per sec = $\rho a V_{r1}$

$$F_x = \rho a V_{r1}\,[(V_{w1} - u_1) - \{-(u_2 + V_{w2})\}]$$

$$= \rho a V_{r1}[V_{w1} + V_{w2}]$$

Its true only when $\beta = 90$, then $V_{w2} = 0$

$$F_x = \rho a V_{r1}[V_{w1}]$$

If β is obtuse angle,

$$F_x = \rho a V_{r1}[V_{w1} - V_{w2}]$$

Workdone per sec per unit weight of fluid striking per sec = $F_x\,.u$

$$= \rho a V_{r1}[V_{w1} \pm V_{w2}]u/(g\ \rho a V_{r1})$$

$$= \frac{1}{g}[V_{w1} \pm V_{w2}].u$$

Workdone per sec per unit mass of fluid striking per sec

$$=[V_{w1} \pm V_{w2}].u$$

HYDRAULICS AND HYDRAULIC MACHINES

Syllabus UNIT – IV: Hydraulic Turbines – I

Layout of a typical Hydropower installation, Heads and efficiencies, classification of turbines - Pelton wheel - Francis turbine - working, working proportions, velocity diagrams, work done and efficiency, draft tube – theory and efficiency.

4.1. Typical Layout of Hydro-electrical power plant:

It consists of the following:

1. A ***Dam*** constructed across a river or a channel to store water. The reservoir is also known as ***Head race.***

2. Pipes of large diameter called ***Penstocks*** which carry water under pressure from storage reservoir to the turbines. These pipes are usually made of steel or reinforced concrete.

3. ***Turbines*** having different types of vanes or buckets or blades mounted on a wheel called runner.

4. ***Tail race*** which is channel carrying water away from the turbine after the water has worked on the turbines. The water surface in the tailrace is also referred to as tailrace.

5. ***Surge tanks*** are usually provided in high or medium head power plants when considerably long penstock is required. A surge tank is a small reservoir or tank which is open at the top. It is fitted between the reservoir and the power house. The water level in the surge tank rises or falls to reduce the pressure swings in the penstock. When there is sudden reduction in load on the turbine, the governor closes the gates of the turbine to reduce the water flow. This causes pressure to increase abnormally in the penstock. This is prevented by using a surge tank, in which the water level rises to reduce the pressure. On the other hand, the **surge tank** provides excess water needed when the gates are suddenly opened to meet the increased load demand.

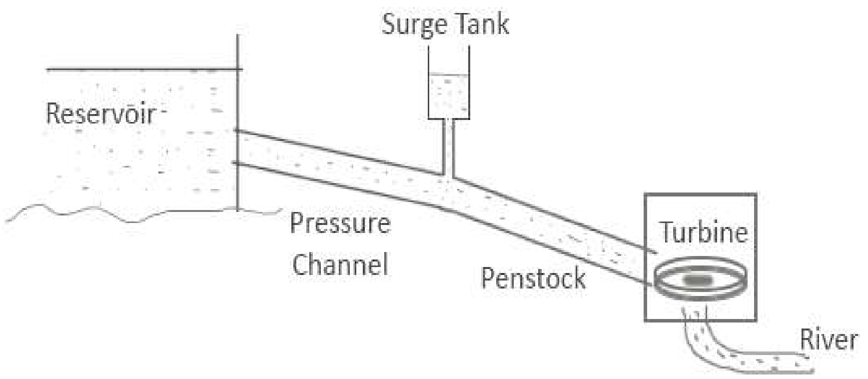


Fig 1: Typical layout of hydro-electric power plant

4.2. Heads of a turbine:

Gross head: It is the difference between the head race and tail race level when there is no flow. As such it is termed as static head and is denoted as H_s or H_g

Effective head: It is the head available at the inlet of the turbine. It is obtained by considering all losses. If h_f is the total loss then the effective head above the turbine is

$$H = H_g - H_f$$

Definition: The device which converts hydraulic energy into mechanical energy or vice versa is known as Hydraulic Machines.

The hydraulic machines which convert hydraulic energy into mechanical energy are known as **Turbines** and that convert mechanical energy into hydraulic energy is known as **Pumps**.

Hydro electricity is a reliable form of renewable energy. Water turbines are highly efficient and easily controlled to provide power as and when it is needed. A hydraulic turbine is a prime mover (a machine which uses the raw energy of a substance and converts into mechanical energy) that uses the energy of flowing water and converts it into the mechanical energy (in the form of rotating of the runner). This mechanical energy is used in running an electric generator which is directly coupled to the shaft of the hydraulic turbine; from this electric generator, we get electric power which can be transmitted over long distances by means of transmission lines and transmission towers. The hydraulic turbines are also known as "Water turbines" since the fluid medium used in them is water.

Hydro (Water) Power is a conventional renewable source of energy which is clean, free from pollution and generally has a good environmental effect.

The disadvantage of energy from water is that it is strictly limited, and widely distributed in small amounts that are difficult to exploit. Only where a lot of water is gathered in a large river, or where descent is rapid, is it possible to take economic advantage.

The following factors are major obstacles in the utilization of hydropower resources.

(i) Large investments.
(ii) Long gestation period, and
(iii) Increased cost of power transmission.

4.3. Types of efficiencies

The following are the important efficiencies of Turbine.

a) Hydraulic Efficiency, η_h

b) Mechanical Efficiency, η_m

c) Volumetric Efficiency, η_v

d) Overall Efficiency, η_o

a) Hydraulic Efficiency (η_h): it is defined as the ratio of power given by the water to the runner of a turbine (runner is a rotating part of a turbine and on the runner vanes are fixed) to the power supplied by the water at the inlet of the turbine. The power at the inlet of the turbine is more and this power goes on decreasing as the water flows over the vanes of the turbine due to hydraulic losses as the vanes are not smooth. Hence power delivered to the runner of the turbine will be less than the power available at the inlet of the turbine.

$$\eta_h = \frac{\text{Power delivered to the runner}}{\text{Power supplied at inlet}} = \frac{\text{R.P}}{\text{W.P}}$$

R.P = Power delivered to the runner $= \frac{W}{g}\frac{[V_{w_1}+V_{w_2}]\times u}{1000}$ kW ----------- for Pelton Turbine

$= \frac{W}{g}\frac{[V_{w_1}u_1+V_{w_2}u_2]\times u}{1000}$ kW ----- Radial flow Turbine.

W.P = power supplied at inlet of turbine = $\frac{W\times H}{1000}$ kW

Where W = weight of water striking the vanes of the turbine per second = $\rho\ g\ Q$

Q = Volume of water per second

V_{w_1} =Velocity of whirl at inlet. V_{w_2} = Velocity of whirl at outlet

u = Tangential velocity of vane

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

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

H = Net head on the Turbine.

Power supplied at the inlet of the turbine in S I Units is known as Water Power.

W.P = $\frac{\rho\times g\times Q\times H}{1000}$ K.W (For water ρ = 1000Kg/m^3)

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

b) Mechanical Efficiency (η_m): The power delivered by the water to the runner of a turbine is transmitted to the shaft of the turbine. Due to mechanical losses, the power available at the shaft of the turbine is less than the power delivered to the runner of the turbine. The ratio of

power available at the shaft of the turbine (Known as S.P or B.P) to the power delivered to
the runner is defined as Mechanical efficiency.

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

c) Volumetric Efficiency (η_v): The volume of the water striking the runner of the turbine is slightly less than the volume of water supplied to the turbine. Some of the volume of the water is discharged to the tailrace without striking the runner of the turbine. Thus the ratio of the volume of the water supplied to the turbine is defined as Volumetric Efficiency.

$$\eta_v = \frac{\text{Volume of water actually striking the Runner}}{\text{Volume of water supplied to the Turbine}}$$

d) Overall Efficiency (η_0): It is defined as the ratio of power available at the shaft of the turbine to the power supplied by the water at the inlet of the turbine.

$$\eta_0 = \frac{\text{Power available at the shaft of the turbine}}{\text{Power supplied at the inlet of the turbine}} = \frac{\text{Shaft power}}{\text{Water power}}$$

$$= \frac{S.P}{W.P} = \frac{S.P}{W.P} \times \frac{R.P}{R.P}$$

$$= \frac{S.P}{R.P} \times \frac{R.P}{W.P}$$

$$\boldsymbol{\eta_0 = \eta_m \times \eta_h}$$

If shaft power (S.P) is taken in kW, Then water power should also be taken in kW. Shaft power is represented by P.

Water power in $kW = \frac{\rho \times g \times Q \times H}{1000}$ Where ρ = 1000Kg/m^3

$$\eta_0 = \frac{\text{Shaft Power in kW}}{\text{Water Power in kW}} = \frac{P}{\frac{\rho \times g \times Q \times H}{1000}}$$ Where P = Shaft Power

CLASSIFICATION OF HYDRAULIC TURBINES:

4.4. Classification of Turbines:

The hydraulic turbines are classified as follows:

(a) According to the head and quantity of water available.
(b) According to the name of the originator
(c) According to the action of water on moving blades
(d) According to the direction of flow of water in the runner.
(e) According to the disposition of thee turbine shaft
(f) According to the specific speed N.

4.4.1. According to the head and quantity of water available:

(i) Impulse turbine ….. requires high head and small quantity of flow

(ii) Reaction, turbine … requires low head and high rate of flow

Actually there are two types of reaction turbines, one for medium head and medium flow and the other for low head and large flow.

Turbine		Type of Energy	Head	Discharge	Direction of flow	Specific Speed
Name	Type					
Pelton Wheel	Impulse	Kinetic	High Head > 250m to1000m	Low	Tangential to runner	Low 5 Single jet 35 Multiple jet
Francis Turbine	Reaction Turbine	Kinetic + Pressure	Medium 60 m to 150 m	Medium	Radial flow Mixed Flow	Medium 60 to 300
Kaplan Turbine			Low < 30 m	High	Axial Flow	High 300 to 1000

4.4.2. According to the Name of the originator:

(i) Pelton turbine - Named after Allen Pelton of California (USA). It is an impulse type of turbine and is used for high head and low discharge.

(ii) Francis turbine - named after James Bichens Francis. It is a reaction type of turbine from medium high to medium low heads and medium small to medium large quantities of water.

(iii) Kaplan turbine - named after Dr.Victor Kaplan. It is a reaction type of turbine for low heads and large quantities of flow.

4.4.3 According to direction of flow of water in the runner

(i) Tangential flow turbines (Pelton turbine)

(ii) Radial flow turbine (no more used)

(iii) Axial flow turbine (Kaplan turbine)

(iv) Mixed (radial and axial) flow turbine (Francis turbine).

In tangential flow turbine of Pelton type the water strikes the runner tangential to the path of rotation.

In axial flow turbine water flows parallel to the axis of the turbine shaft. Kaplan turbine is an axial flow turbine. In Kaplan turbine the runner blades are adjustable and can be rotated about pivots fixed to the boss of runner. If the runner blades of the axial flow turbines are fixed, these are called “Propeller turbines”

In mixed flow turbines the water enters the blades radially and comes out axially, parallel to the turbine shaft. Modern Francis turbines have mixed flow runners.

4.4.4. According to the disposition of the turbine shaft:

Turbine shaft may be either vertical or horizontal. In modern practice, Pelton turbines usually have horizontal shafts whereas the rest, especially the large units, have vertical shafts.

4.4.5. According to specific speed:

The specific speed of a turbine is defined as the speed of a geometrically similar turbine that would develop 1 KW under 1 m head. All geometrically similar turbines (irrespective of the size) will have the same specific speeds when operating under the same head.

Specific speed, $N_s = N \frac{\sqrt{P}}{H^{5/4}}$

Where N = the normal working speed (rpm) P = power output (K_w) of the turbine, and

H = the net or effective head in meters.

Turbines with low specific speeds work under high head and low discharge conditions, while high specific speed turbines work under low head and high discharge conditions.

Sl. No.	Aspects	Impulse Turbine	Reaction turbine
1	Conversion of fluid energy	The available fluid energy is converted into K.E by a nozzle.	The energy of the fluid is partly transformed in to K.E before it (fluid) enters the runner of the turbine.
2	Changes in pressure and velocity	The pressure remains same (atmospheric) throughout the action of water on the runner	After entering the runner with an excess pressure, water undergoes changes both in velocity and pressure while passing through the runner.
3	Admittance of water over the wheel	Water may be allowed to enter a part or whole of the wheel circumference.	Water is admitted over the circumference of the wheel
4	Water-tight causing	Not necessary	Required

5	Extent to which the water fills the wheel/turbine	The wheel/turbine does not run full and air has a free access to the buckets.	Water completely fills all the passages between the blades and while flowing between inlet and outlet sections does work on the blades.
6	Installation of Unit	Always installed above the tail race. No draft tube is used.	Unit may be installed above or below the tail race-use of a draft tube is made
7	Relative velocity of water	Either remaining constant or reduces slightly due to friction.	Due to continuous drop in pressure during flow through the blade, the relative velocity increases.
8	Flow regulation	• By means of a needle valve fitted into the nozzle. • Impossible without less	• By means of a guide – vane assembly. • Always accompanied by loss

4.5. Pelton Wheel:

Pelton wheel, named after an eminent engineer, is an impulse turbine wherein the flow is tangential to the runner and the available energy at the entrance is completely kinetic energy. Further, it is preferred at a very high head and low discharges with low specific speeds. The pressure available at the inlet and the outlet is atmospheric.

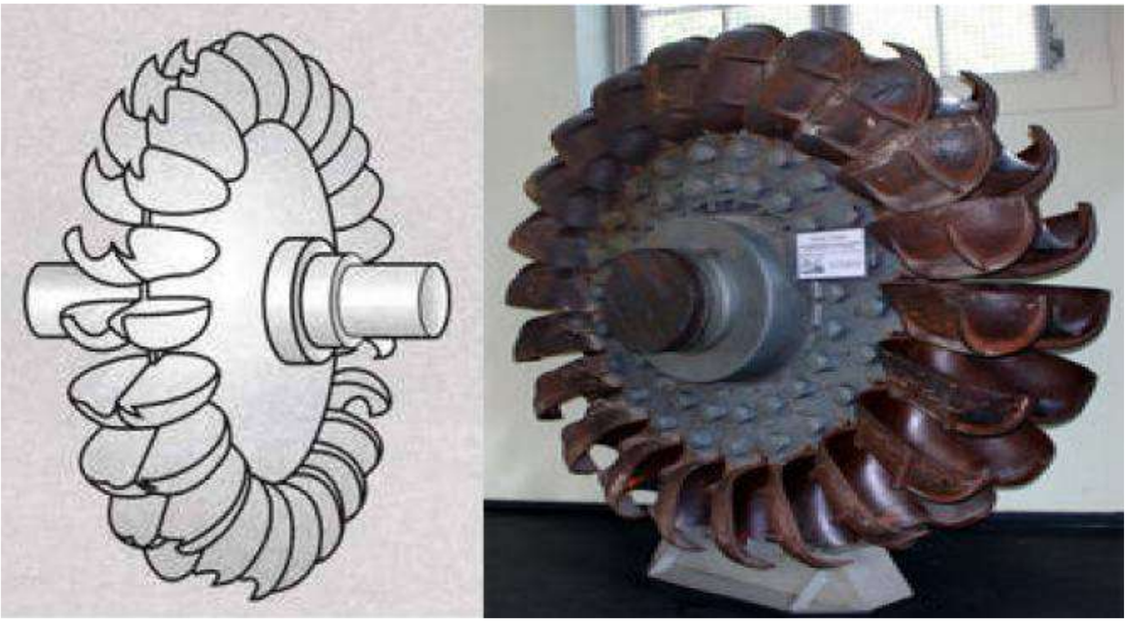
The main components of a Pelton turbine are:

(i) *Nozzle and flow regulating arrangement:*

Water is brought to the hydroelectric plant site through large penstocks at the end of which there will be a nozzle, which converts the pressure energy completely into kinetic energy. This will convert the liquid flow into a high-speed jet, which strikes the buckets or vanes mounted on the runner, which in-turn rotates the runner of the turbine. The amount of water striking the vanes is controlled by the forward and backward motion of the spear. As the water is flowing in the annular area between the annular area between the nozzle opening and the spear, the flow gets reduced as the spear moves forward and vice-versa.

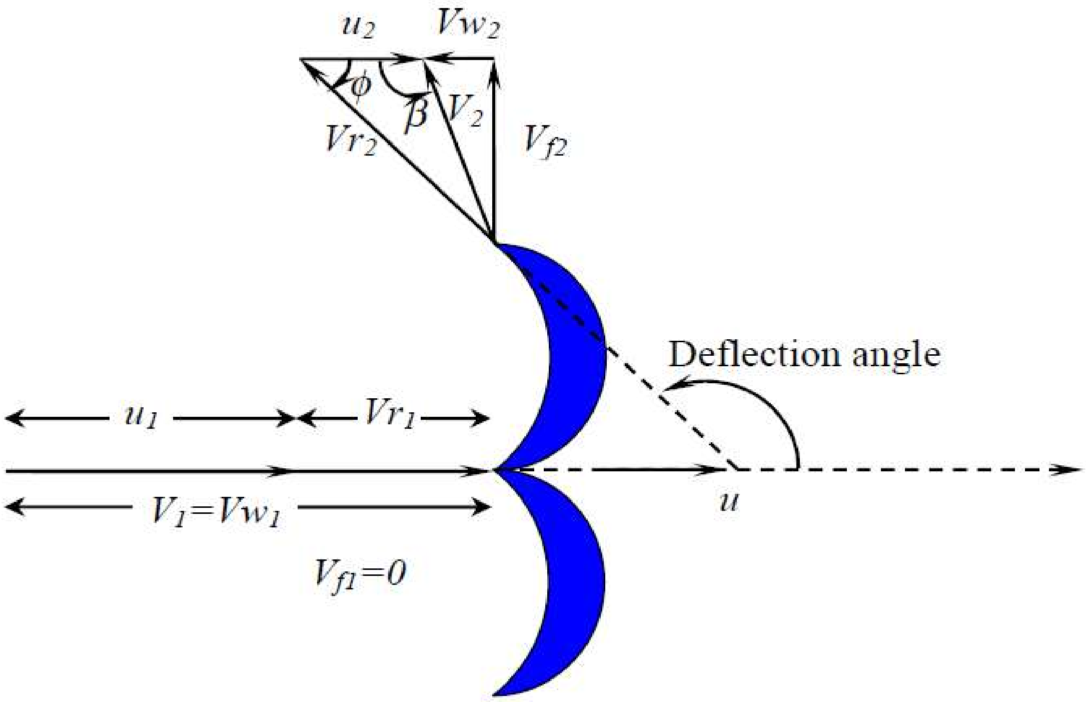
(ii) *Runner with buckets:*

Runner is a circular disk mounted on a shaft on the periphery of which a number of buckets are fixed equally spaced as shown in Fig. The buckets are made of cast -iron cast -steel, bronze or stainless steel depending upon the head at the inlet of the turbine. The water jet strikes the bucket on the splitter of the bucket and gets deflected through (□) 160-170^{0}.



iii) Casing:
It is made of cast -iron or fabricated steel plates. The main function of the casing is to prevent splashing of water and to discharge the water into tailrace.

(iv) *Breaking jet:*
Even after the amount of water striking the buckets is completely stopped, the runner goes on rotating for a very long time due to inertia. To stop the runner in a short time, a small nozzle is provided which directs the jet of water on the back of bucket with which the rotation of the runner is reversed. This jet is called as breaking jet.



Velocity triangles for the jet striking the bucket

From the impulse-momentum theorem, the force with which the jet strikes the bucket along the direction of vane is given by

F_x = rate of change of momentum of the jet along the direction of vane motion

F_x = (Mass of water / second) x change in velocity along the x direction

$$F_x = \rho a V_1 [V_{w1}-(-V_{w2})]$$

$$= \rho a V_1[V_{w1}+V_{w2}]$$

Work done per second by the jet on the vane is given by the product of Force exerted on the vane and the distance moved by the vane in one second

$$W.D./S = F_x \times u$$

$$W.D./S = \rho a V_1[V_{w1}+V_{w2}]\, u$$

Input to the jet per second = Kinetic energy of the jet per second= $\frac{1}{2}\rho a V_1^{\,3}$

Efficiency of the jet = $\frac{\text{Output to the jet per second}}{\text{Input to the jet per second}}$

$$\eta=\frac{\rho aV1\,[Vw1+Vw2]\,u}{\frac{1}{2}\rho a V_1^{\,3}}$$

$$\eta=\frac{\rho aV_1[Vw1+Vw2]\,u}{\frac{1}{2}\rho a V_1^{\,3}}$$

$$\eta=\frac{[Vw1+Vw2]\,u}{V_1^{\,2}}$$

From inlet velocity triangle,

$$V_{w1} = V_1$$

Assuming no shock and ignoring frictional losses through the vane, we have

$$V_{r1} = V_{r2} = (V_1 - u_1)$$

In case of Pelton wheel, the inlet and outlet are located at the same radial distance from the centre of runner and hence

$$u_1 = u_2 = u$$

From outlet velocity triangle, we have

$$V_{w2} = V_{r2}\,Cos\phi - u_2$$

$$= (V_1 - u)Cos\phi - u$$

$$F_x= \rho a V_1 [V_1+(V_1-u)\, Cos\phi - u]$$

$$F_x= \rho a V_1 [V_1-u]\, [1+Cos\phi]$$

Substituting these values in the above equation for efficiency, we have

$$\eta=\frac{2u[V_1+(V_1-u)Cos\emptyset-u]}{V_1^{\,2}}$$

$$\eta = \frac{2u[V_1 - u + (V_1 - u)\,Cos\emptyset]}{V_1{}^2}$$

The above equation gives the efficiency of the jet striking the vane in case of Pelton wheel. To obtain the maximum efficiency for a given jet velocity and vane angle, from maxima-minima, we have

$$\frac{d\eta}{du} = 0$$

$$\frac{d\eta}{du} = 0 = \frac{2}{V_1{}^2}\,[1+\cos\emptyset]\,\frac{d}{du}(uV_1 - u^2)$$

V_1-2u=0 or u= V_1/2

When the bucket speed is maintained at half the velocity of the jet, the efficiency of a Pelton wheel will be maximum. Substituting we get,

$$n_{max} = \frac{2u}{(2u)^2}\,(2u - u)[1 + cos\emptyset]$$

$$n_{max} = \frac{1}{2}\,[1 + cos\emptyset]$$

From the above it can be seen that more the value of cos □□ more will be the efficiency. Form maximum efficiency, the value of cos□ should be 1 and the value of □ should be 0^0 . This condition makes the jet to completely deviate by 180^0 and this, forces the jet striking the bucket to strike the successive bucket on the back of it acting like a breaking jet.

Hence to avoid this situation, at least a small angle of □ =5^0 should be provided.

Reaction Turbine:

Reaction turbines are those turbines which operate under hydraulic pressure energy and part of kinetic energy. In this case, the water reacts with the vanes as it moves through the vanes and transfers its pressure energy to the vanes so that the vanes move in turn rotating the runner on which they are mounted.
The main types of reaction turbines are

Radially outward flow reaction turbine:

This reaction turbine consist a cylindrical disc mounted on a shaft and provided with vanes around the perimeter. At inlet the water flows into the wheel at the centre and then glides through radially provided fixed guide vanes and then flows over the moving vanes. The function of the guide vanes is to direct or guide the water into the moving vanes in the correct direction and also regulate the amount of water striking the vanes. The water as it flows along the moving vanes will exert a thrust and hence a torque on the wheel thereby rotating the wheel. The water leaves the moving vanes at the outer edge. The wheel is enclosed by a water-tight casing. The water is then taken to draft tube.

Radially inward flow reaction turbine:

The constitutional details of this turbine are similar to the outward flow turbine but for the fact that the guide vanes surround the moving vanes. This is preferred to the outward flow turbine as this turbine does not develop racing. The centrifugal force on the inward moving body of water decreases the relative velocity and thus the speed of the turbine can be controlled easily.

The main component parts of a reaction turbine are:

(1) Casing, (2) Guide vanes (3) Runner with vanes (4) Draft tube

Casing:

This is a tube of decreasing cross -sectional area with the axis of the tube being of geometric shape of volute or a spiral. The water first fills the casing and then enters the guide vanes from all sides radially inwards. The decreasing cross -sectional area helps the velocity of the entering water from all sides being kept equal. The geometric shape helps the entering water avoiding or preventing the creation of eddies.

Guide vanes: Already mentioned in the above sections.

Runner with vanes: The runner is mounted on a shaft and the blades are fixed on the runner at equal distances. The vanes are so shaped that the water reacting with the m will pass through them thereby passing their pressure energy to make it rotate the runner.

Draft tube: This is a divergent tube fixed at the end of the outlet of the turbine and the other end is submerged under the water level in the tail race. The water after working on the turbine, transfers the pressure energy there by losing all its pressure and falling below atmospheric pressure. The draft tube accepts this water at the upper end and increases its pressure as the water flows through the tube and increases more than atmospheric pressure before it reaches the tailrace.

Reaction Turbine :- In reaction turbine runner utilizes both potential and kinetic energy. As a water flows through the stationary parts of the turbine, whole of its pressure energy is not transferred to kinetic energy.

When the water flows through the moving parts, there is a change both in pressure & in the direction & velocity of flow of water. As the water give up its energy to the runner, both its pressure & absolute velocity gets reduced.

Important reaction turbines are francis, kaplon and propeller

The water which acts as the runner blades is under a pressure above atmospheric and the runner passages are always completely filled with water

Francis Turbine:-

Penstock, spiral/scroll casing

Guide cranes

Governing mechanism, runner and runner blades

Draft tube

Inward flow reaction turbine	Outward flow reaction turbine
1.water enters at the outer periphery, flows inward and toward the centre of the turbine & discharges at the outer periphery	1.water enters at the inner periphery flows outward & discharges at the outer periphery
2.centrifugal head is negative	2. centrifugal head is positive
3. Discharge does not increase	3. Discharge increases
4.Speed control is easy and effective	4. Very difficult
5. The turbine adjust the speed by itself	5. The turbine can't adjust the speed itself
6. Suitable for medium high heads	6. Suitable for low or medium heads

Workdone and efficiency of francis turbine

$H = H_g - h_f$

Workdone (W.D) = $\rho\, Q\, (V_{w1}u_1 \pm V_{w2}u_2)$

$= wQ/g\, (V_{w1}u_1 \pm V_{w2}u_2)$

Q = discharge through the runner m^3/s

The maximum output under given condition is obtained when $V_{w2} = 0$

Maximum workdone = $wQ/g\, (V_{w1}u_1)$

H is net head

Input to the turbine = wQH

$\eta_h = wQ/g\, (V_{w1}u_1) / wQH$

$= V_{w1}u_1/gH$

If the velocity of whirl at exit not equal to 0.

$\eta_h = V_{w1}u_1 \pm V_{w2}u_2 / wQH$

η_h = 85 to 90%

η_m = P / Power developed

η_o = P / wQH

= 80 to 90 %

Working proportions of a francis turbine

1. **Ratio of width to diameter (B/D)**

The ration of width to the diameter of the wheel at the inlet is represented by n

$n = B_1 / D_1$ (0.1 to 0.45)

2. Flow ratio (K_f):-

   Flow ratio is the ration of the velocity of flow at inlet to the theoretical jet velocity

   $$K_f = V_{f1} / \sqrt{2gH}$$

   $$K_f = 0.15 \text{ to } 0.3$$

3. Speed ratio (K_u)

   $K_u = u\ 4/ \sqrt{2gH} = 0.6 \text{ to } 0.9$

Design of francis turbine runner:-

The runner of a francis turbine is required to the designed to develop a known power P, with N r.p.m , under H, design of the runner involves the determination of its size & the crane angles

1. Assume suitable values of η_0 , η_h, n,K_f & K_u
2. Determine the required discharge 'Q' from the relation
   $P = \eta_0\ wQH$
3. Obtain the velocity of flow from the discharge & flow area

   Let B_1 = width

   D_1 = diameter

   t_1 = thickness of runner vane at inlet.

   Total area at the outer periphery

   $$A = (\pi\ D_1 - Z\ t_1)\ B_1$$

   $$= K_{f1}\ \pi\ D_1\ B_1$$

   K_{f1} = vane thickness factor / coefficient

   Q = Area of flow x Velocity of flow

   $= K_{f1}\ \pi\ D_1\ B_1 \times V_{f1}$

   $V_{f1} = Q / K_{f1}\ \pi\ D_1\ B_1 = Q / K_{f1}\ \pi n D_1^2$ $(B_1 = n\ D_1)$

   $D_1 = (Q/ K_f\ 2gH\ K_f\ \pi n)^{(1/2)}$

4. Find the rim velocity u,

   $u_1 = \pi\ n\ D_1/60$

5. Find $V_{w1}u_1$

   $$\eta_h = V_{w1}u_1/ gH$$

$V_{w1} = \eta_h gH/ u_1$

6. Obtain the guide vane angle α
   Runner vane angle θ

   $\tan \alpha = V_{f1}/ V_{w1}$

   $\tan \theta = V_{f1}/ V_{w1} - u_1$
7. Assume runner dia D_2 at the outlet to be approximately one-half the diameter of inlet

$$D_2 = D_1/2 \;\&\; u_2 = u_1/2$$

8. The velocity of flow at exit V_{f2}

   $Q = K_{f1} \pi D_1 B_1 V_{f1}$

   $= K_{f2} \pi D_2 B_2 V_{f2}$

   $V_{f1}/ V_{f2} = K_{f1} \pi D_1 B_1 V_{f1}/ K_{f2} \pi D_2 B_2 V_{f2}$

Usually $V_{f1} = V_{f2}$

$K_{f1} = K_{f2}$

& gives $B_2 = 2B_1$

9. Runner vane angke at exit ϕ

& assumed

$\tan \phi = V_{f2} / u_2$

10. Number of vanes are 16 to 24

To avoid the periodic impulse

Guide vanes = vanes

DRAFT TUBE:

The draft tube is a pipe of gradually increasing area, which connects the outlet of the runner to the tail race. It is used for discharging water from the exit of the turbine to the tail race. This pipe of gradually increasing area is called a draft tube. One end of the draft tube is connected to the outlet of the runner and the other end is submerged below the level of water in the tail race. The draft tube in addition to save a passage for water discharge has the following two purposes also:

1. It permits a negative head to be established at the outlet of the runner and their by increase the net head on the turbine. The turbine may be placed above the tail race without any loss of net head and hence turbine may be inspected properly.

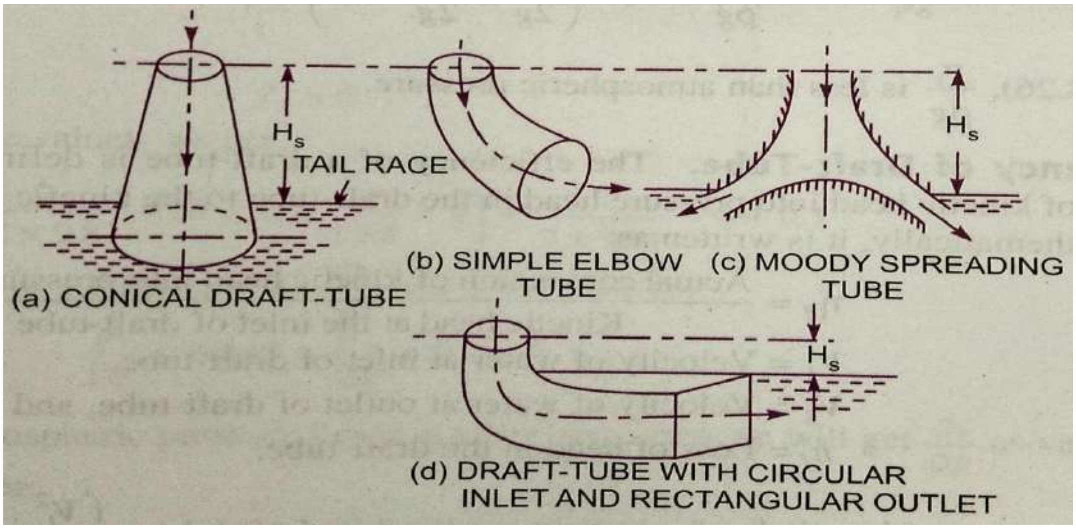
2. It converts a large portion of the kinetic energy $\left(\frac{V_2{}^2}{2g}\right)$ rejected at the outlet of the turbine into useful energy. Without the draft tube the kinetic energy rejected at the turbine will go waste to the tail race.

Hence by using the draft tube, the net head on turbine increases. The turbine develops more power and also the efficiency of the turbine increase.

If a reaction turbine is not fitted with a draft tube, the pressure at the outlet of the runner will be equal to atmospheric pressure. The water from the outlet of the runner will discharge freely into the tail race. The net head on the turbine will be less than that of
a reaction turbine fitted with a draft tube. Also without draft tube the kinetic energy $\left(\frac{V_2{}^2}{2g}\right)$ rejected at the outlet of the will go water to the tail race.

Types of Draft Tube:

1. Conical Draft Tube
2. Simple Elbow Tubes
3. Moody Spreading tubes
4. Elbow Draft Tubes with Circular inlet and rectangular outletThe conical draft



tubes

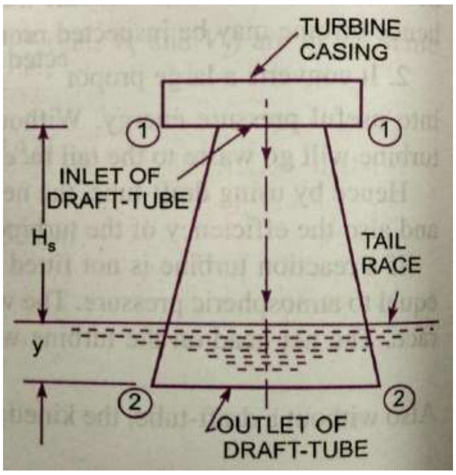
and moody spreading draft tubes are most efficient while simple elbow draft tube and elbow draft tubes with circular inlet and rectangular outlet require less space as compared to other draft tubes.

Draft tube theory: Consider a conical draft tube H_s = Vertical height of draft tube above tail race Y= Distance of bottom of draft tube from tail race.

Applying Bernoulli‟s equation to inlet section 1-1 and outlet section 2-2of the

draft
tube and taking section 2-2 a datum, we get

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+(H_s+y)=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+0+h_f \qquad \text{\_\_\_\_\_\_\_\_\_\_\_\_(1)}$$



Wher h_f = loss of energy between section 1-1 and 2-

But $\frac{p_2}{\rho g}$ = Atmospheric Pressure + y = $\frac{p_a}{\rho g}+y$

Substituting this value of $\frac{p_2}{\rho g}$ in equation (1) we

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+(H_s+y)=\frac{p_a}{\rho g}+y+\frac{V_2^2}{2g}+h_f$$

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+H_s=\frac{p_a}{\rho g}+\frac{V_2^2}{2g}+h_f$$

$$\frac{p_1}{\rho g}=\frac{p_a}{\rho g}+\frac{V_2^2}{2g}+h_f-\frac{V_1^2}{2g}-H_s \qquad (2)$$

In equation (2) is less than atmospheric pressure.

$$\frac{p_1}{\rho g}=\frac{p_a}{\rho g}-H_s-\left[\frac{V_1^2}{2g}-\frac{V_2^2}{2g}-h_f\right]$$

Efficiency of Draft Tube: the efficiency of a draft tube is defined as the ratio of actual conversion of kinetic head in to pressure in the draft tube to the kinetic head at the inlet of the draft tube.

$$\eta_d=\frac{Actual\ conversion\ of\ Kinetic\ head\ in\ to\ Pressure\ head}{Kintic\ head\ at\ the\ inlet\ of\ draft\ tube}$$

Let V_1 = Velocity of water at inlet of
draft tube V_2 = Velocity of water
at outlet of draft tube h_f = Loss
of head in the draft tube

Theoretical conversion of Kinetic head into Pressure head in

$$\text{Draft tube}=\left[\frac{V_1^2}{2g}-\frac{V_2^2}{2g}\right]$$

Actual conversion of Kinetic head into pressure head

$=\left[\frac{V_1^2}{2g}-\frac{V_2^2}{2g}\right]-h_f$ Now Efficiency of draft tube

$$\eta_d=\frac{\left[\frac{V_1^2}{2g}-\frac{V_2^2}{2g}\right]-h_f}{\frac{V_1^2}{2g}}$$

HYDRAULICS AND HYDRAULIC MACHINES
Unit-V

Syllabus UNIT – V: Hydraulic Turbines – II

Surge tanks, unit quantities -unit speed, unit discharge, unit power; specific speed, characteristic curves, geometric similarity, cavitation.

Objective:
To understand the working principles of hydraulic machines

Outcome:
Choose appropriate hydraulic machine for specific use

Surge Tank:

Surge tank is a water storage device used as pressure neutralizer in hydropower water conveyance system to resists excess pressure rise and pressure drop conditions.

Functions of Surge Tanks

The important functions of surge tank are as follows

- It should Protects the conduit system from high internal pressures.
- It should help the hydraulic turbine regarding its regulation characteristics.
- It should store the water to raise the pressure in pressure drop conditions.

Location of Surge Tanks

The location of surge tank is also important to produce better results. It should be located in such a way that

- Surge tanks are located near to the power house to reduce length of penstocks.
- No limitations regarding surge tank height.
- Location at which flat sloped conduit and steep sloped penstock meets.

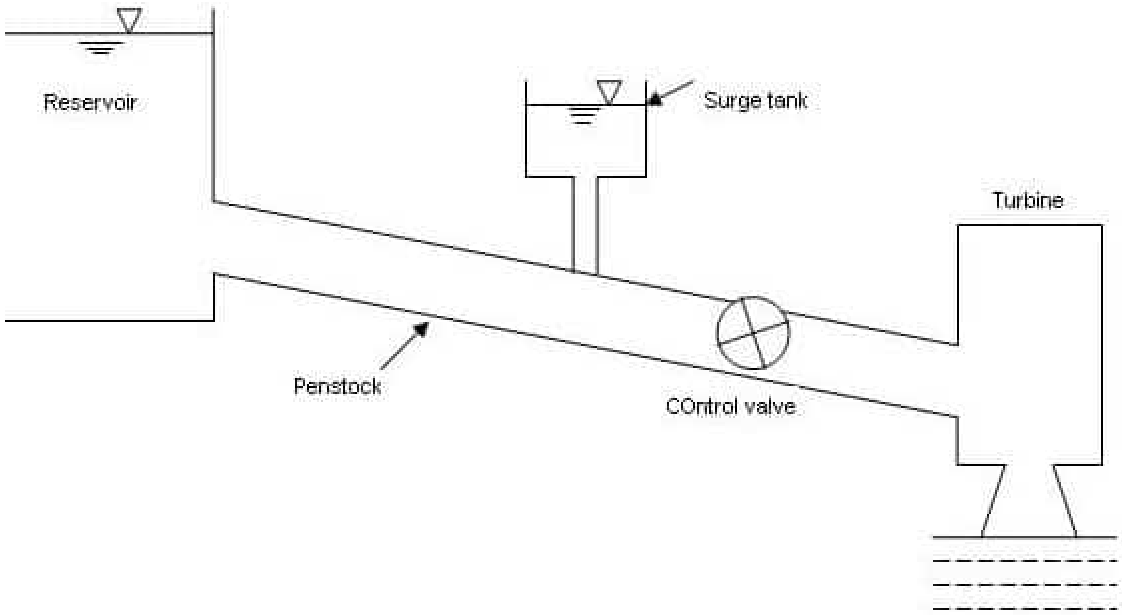
Types of Surge Tanks

Various types of surge tanks used in the hydropower water conveyance system are as follows.

- Simple surge tank
- Gallery type surge tank
- Inclined surge tank
- Restricted orifice surge tank
- Differential surge tank

Simple Surge Tank

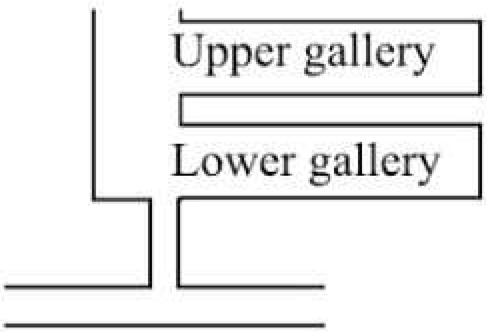
A simple surge tank is like vertical pipe which is connected in between penstock and turbine generator. These are constructed with greater height and supports are also provided to hold the tank. Whenever the water flow suddenly increased the water is collected in the surge tank and neutralize the pressure. Top of the surge tank is opened to atmosphere If surge tank is filled completely then it overflows to maintain the pressure



neutralization.

Gallery Type Surge Tank

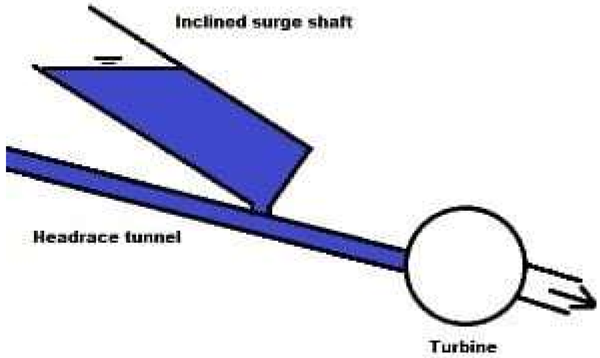
Gallery type surge tank consists extra storage galleries in it. These storage galleries are also called as expansion chambers. So, gallery type surge tank can also be called as expansion chamber type surge tanks. These expansion chambers are generally provided at below and above the surge levels. Below surge level chambers are used to storage excess water in it and released when it is required or there is a brief drop in pressure. Upper surge level chambers are used to absorb the excess



pressure.

Inclined Surge Tank

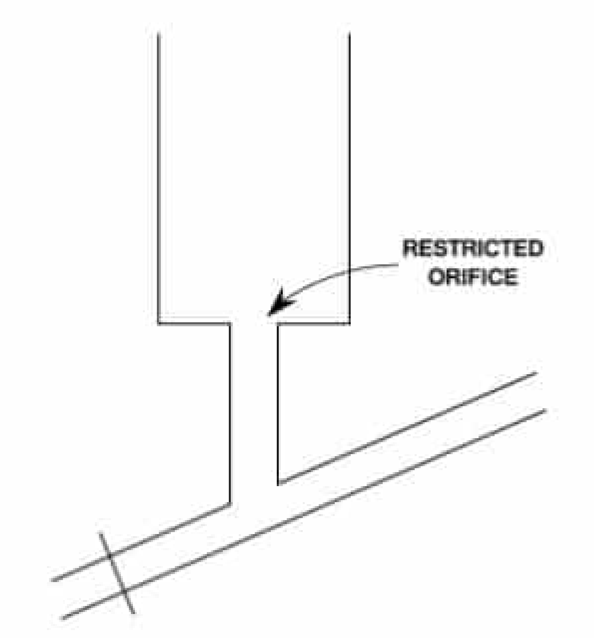
In case of inclined surge tank, the surge tank is provide with some inclination. It is provided when there is a limit in height of tank. By providing inclined surge tank the overflowed water under excess pressure is



entered into inclined tank and pressure destroyed.

Restricted Orifice Surge Tank

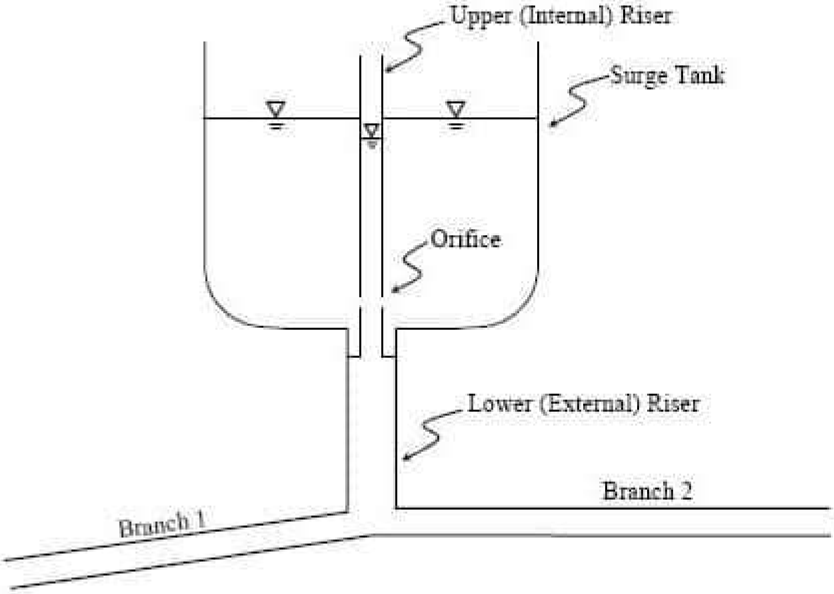
Restricted orifice consists an orifice between pipeline and surge tank. This orifice is also called as throttle so, it is also called as throttled surge tank. This throttle or orifice have very small diameter. If the water overflows it should enter into the surge tank through this orifice. Because of small diameter frictional losses will developed and excess pressure in main pipe line is destroyed. This will creates quickly a retarding or accelerating head in the conduit. To reduce the water hammer effect, diameter of orifice should be well



designed for full rejection of load by the turbine.

Differential Surge Tank

In case of differential surge tank, an internal riser is fixed in the tank. This riser have very small diameter through which water enters into the riser when it overflows. The riser also contains annular ports at its



lower end. These ports help the flow into or out of the tank. So, the excess pressure is destroyed by internal riser of surge tank and storage of water is done by outer tank. So, it is called as differential surge tank.

Unit Quantities:-

In order to predict the behaviour of a turbine working under varying conditions of head, speed, output and gate opening, the results are expressed in terms of quantities which may be obtained when the head on the turbine is reduced to unity. The conditions of the turbine under unit head are such that the efficiency of the turbine remains unaffected. The following are the three important unit quantities which must be studied under unit head.

1. Unit speed 2. Unit discharge, and 3. Unit power

Unit Speed:-It is defined as the speed of turbine working under a unit head (i.e., under a head of 1m). It is denoted by 'N_H'. The expression for unit speed is obtained as:

Let N = Speed of a turbine under a head H,

H = head under which a turbine is working,

u = Tangential velocity.

The tangential velocity, absolute velocity of water and head on the turbine are related as $u \propto V$, Where $V \propto H^{(1/2)}$

$\alpha\ H^{(1/2)}$…………(1)

Also tangential velocity (u) is given by

u = ,

Where D = Diameter of turbine.

For a given turbine, the diameter (D) is constant.

$u \propto N$ or $N \propto u$ or $N \propto H^{(1/2)}$

$N = K_1 H^{(1/2)}$ …………(2)

Where K_1 is constant of proportionality

If head on the turbine becomes unity, the speed becomes unit speed or

When $H = 1$, $N = N_u$

Substituting these values in equation (2) , we get

$N_u = K_1 1^{(1/2)} = k_1$

Substituting the value of K_1 in equation (2),

$N = N_u H^{(1/2)}$ or

$N_u =$

Unit Discharge:- It is defined as the discharge passing through a turbine, which is working under a unit head (i.e., 1m). It is denoted by the symbol 'Q_u'. The expression for unit discharge is given as:

Let H = head of water on turbine,

Q = Discharge passing through turbine when head is H on the turbine,

a = Area of flow of water.

The discharge passing through a given turbine under a head 'H' is given by,

Q = Area of flow * Velocity

But for a turbine, area of flow is constant and velocity is proportional to $H^{(1/2)}$.

$Q \alpha$ Velocity $\alpha H^{(1/2)}$

Or $Q = K_2 H^{(1/2)}$ ………..(3)

Where K_2 is constant of proportionality.

If $H = 1$, $Q = Q_u$.

Substituting these values in equation (3), we get

$Q_u = K\ 1^{(1/2)} = K_2$.

Substituting the values of K_2 in equation (3), we get

$Q = Q_u H^{(1/2)}$

$Q_u =$

Unit Power:- It is defined as the power developed by a turbine, working under s unit head (i.e., under a head or 1m). It is denoted by the symbol 'P_u'. The expression for unit power is obtained as:

Let H = Head of water on the turbine,

P = Power developed by the turbine under a head of H,

Q = Discharge through turbine under a head H.

The overall efficiency (η_0) is given as

$\eta_0 = \ =$

$P = \eta_0$ *

α Q*H

α $H^{(1/2)}$* H

α $H^{(3/2)}$ ………(4)

$P = K_3H^{(3/2)}$

Where K_3 is a constant of proportionality.

When H = 1m, $P = P_u$

$P_u = K\ (1)^{(3/2)} = K_3$.

Substituting the value of K_3 in equation (4), we get

$P = P_uH^{(3/2)}$

$P_u=$.

Specific Speed:-

It is defined as the speed of a turbine which is identical in shape, geometrical dimensions, blade angles, gate opening etc., with the actual turbine but of such a size that it will develop unit power when working under unit head. It is denoted by the symbol N_s. The specific speed is used in comparing the different types of turbines as every type of turbine has different specific speed.

In M.K.S. units, unit power is taken as one horse power and unit head as one metre. But in S.I. units, unit power is taken as one kilowatt and unit head as one metre.

Derivation of the Specific Speed:- The overall efficiency (η_0) of any turbine is given by,

$$\eta_0 = \ = \ = \ \ldots\ldots..(1)$$

Where H = Head under which the turbine is working,

Q = Discharge through turbine,

P = Power developed or shaft power.

From equation (1), $P = \eta_0$ *

α Q*H (as η_0 and ρ are constant) ……..(2)

Now let D = Diameter of actual turbine,

N = Speed of actual turbine,

u = Tangential velocity of the turbine,

N_S = Specific speed of the turbine,

V = Absolute velocity of water.

The absolute velocity, tangential velocity and head on the turbine are related as,

$u \; \alpha \; V$, where $V \; \alpha \; H^{(1/2)}$

$\alpha \; H^{(1/2)}$ ……..(3)

But the tangential velocity u is given by

$u =$

α DN ……..(4)

From equations (3) and (4) , we have

$H^{(1/2)} \; \alpha$ DN or D =

The discharge through turbine is given by

Q = Area*Velocity

But Area α B*D

$\alpha \; D^2$

And Velocity $\alpha \; H^{(1/2)}$

$Q \; \alpha \; D^2 * H^{(1/2)}$

$\alpha \; ()^2 * H^{(1/2)}$

$\alpha * H^{(1/2)} \; \alpha$ ……….(5)

Substituting the value of Q in equation (2), we get

P α * H α

P = K , where K = constant of proportionality.

If P = 1, H = 1, the speed N = Specific speed N_S, Substituting these values in the above equation, we get

1 = K or N_S^2 = K

P = N_S^2 or N_S^2 =

$N_S = ()^2$ = ………….(6)

In equation (6), if P is taken in metric horse power the specific speed is obtained in M.K.S. units. But if P is taken in kilowatts, the specific speed is obtained in S.I. units

Significance of Specific Speed:- Specific speed plays an important role for selecting the type of the turbine. Also the performance of a turbine can be predicted by knowing the specific speed of the turbine. The type of turbine for different specific speed is given in Table.

S.No.	Specific speed (M.K.S.)	Specific speed (S.I.)	Type of turbine
1.	10 to 35	8.5 to 30	Pelton wheel with single jet
2.	35 to 60	30 to 51	Pelton wheel with two or more jets
3.	60 to 300	51 to 225	Francis turbine
4.	300 to 1000	225 to 860	Kaplan or propeller turbine

Geometric similarity

The geometric similarity must exist between the model and its proto type. the ratio of all corresponding linear dimensions in the model and its proto type are equal.

Let

Lm =Length of model

Bm = Breadth of model

Dm = Diameter of model

Am = Area of model

Vm = Volume of model

And $L_P, b_P, D_P, A_P, \forall_P =$ Corresponding values of the prototype.

For geometrical similarity between model and prototype, we must have the relation,

$$\frac{L_P}{L_m} = \frac{b_P}{b_m} = \frac{D_P}{D_m} = L_r$$

Where L_r is called scale ratio.

For area"s ratio and volume"s ratio the relation should be,

$$\frac{A_P}{A_m} = \frac{L_P \times b_P}{L_m \times b_m} = L_r \times L_r = L_r^{\ 2}$$

$$\frac{\forall_P}{\forall_m} = \left(\frac{L_P}{L_m}\right)^3 = \left(\frac{b_P}{b_m}\right)^3 = \left(\frac{D_P}{D_m}\right)^3 = L_r^{\ 3}$$

Cavitation:-

Cavitation is defined as the phenomenon of formation of vapour bubbles of a flowing liquid in a region where the pressure of the liquid falls below its vapour pressure and the collapsing of these vapour bubbles in a region of higher pressure. When the vapour bubbles collapse, a very high pressure is created. The metallic surfaces, above which these vapour bubbles collapse, is subjected to these high pressures, which cause pitting action on the surface. Thus cavities are formed on the metallic surface and also considerable noise and vibrations are produced.

Precaution Against Cavitation:-The following precautions should be taken against cavitation:

1. The pressure of the flowing liquid in any part of the hydraulic system should not be allowed to fall below its vapour pressure. If the flowing liquid is water, then the absolute pressure head should not be below 2.5m of water.
2. The special materials or coatings such as aluminium-bronze and stainless steel, which are cavitation resistant materials, should be used.

Effects of Cavitation:- The following are the effects of cavitation:

1. The metallic surfaces are damaged and cavities are formed on the surfaces.
2. Due to sudden collapse of vapour bubble, considerable noise and vibrations are produced.
3. The efficiency of a turbine decreases due to cavitation. Due to pitting action, the surface of the turbine blades becomes rough and the force exerted by water on the turbine blades decreases. Hence, the work done by water or output horse power becomes less and thus efficiency decreases.

Hydraulic Machines Subjected to Cavitation:-The hydraulic machines subjected to cavitation are reaction turbines and centrifugal pumps.

Cavitation in Turbines:- In turbines, only reaction turbines are subjected to cavitation. In reaction turbines the cavitation may occur at the outlet of the runner or at the inlet of the draft tube where the pressure is considerably reduced (i.e., which may be below the vapour pressure of the liquid flowing through the turbine). Due to cavitation, the metal of the runner vanes and draft-tube is gradually eaten away, which results in lowering the efficiency of the turbine. Hence, the cavitation in a reaction turbine can be noted by a sudden drop in efficiency. In order to determine whether cavitation will occur in any portion of a reaction turbine, the critical value of Thoma's cavitation factor (σ, sigma) is calculated.

Thoma's Cavitation Factor for Reaction Turbines:-Prof.D. Thoma suggested a dimensionless number, called after his name Thoma's cavitation factor σ (sigma), which

can be used for determining the region where cavitation takes place in reaction turbines. The mathematical expression for the Thoma's cavitation factor is given by

$$\sigma = \quad = \quad \ldots\ldots\ldots\ldots..(1)$$

Where H_b = Barometric pressure head in m of water,

H_{atm} = Atmospheric pressure head in m of water,

H_v = Vapour pressure head in m of water,

H_S = Suction pressure at the outlet of reaction turbine in m of water or height of turbine runner above the tail water surface,

H = Net head on the turbine in m.

The following empirical relationships are used for obtaining the value of σ_C for different turbines:

For Francis turbines, $\sigma_C = 0.625\ ()^2$

$= 431*10^{-8} Ns^2$

For Propeller turbines, $\sigma_C = 0.28 + [\ ()^2]$

In the above expressions N_S is in (r.p.m., kW , m) units. If N_S is in (r.p.m., h.p.m.,) units, the empirical relationships would be as follows:

For Francis turbines, $\sigma_C = 0.625\ ()^2$

$= 371*10^{-8} Ns^2$

For Propeller turbines, $\sigma_C = 0.28 + [\ ()^2]$

PERFORMANCE CHARACTERISTIC CURVES OFTURBINES

Characteristic curves of a hydraulic turbine are the curves, with the help of which the exact behaviour and performance of the turbine under different working conditions can be known. These curves are plotted from the results of the tests performed on the turbine under different working conditions.

The important parameters which are varied during a test on turbine are:

1) Speed (N) 2) Head (H) 3) Discharge (Q)

4) Power (P) 5) Overall Efficiency (η_0) and 6) Gate opening.

Out of the above six parameters, three parameters namely speed (N), Head (H) and discharge (Q) are independent parameters.

Out of the three independent parameters, (N, H, Q) one of the parameter is kept constant (say H) and the variation of other two parameters with respect to any one of the remaining two independent variables (say N and Q) are plotted and various curves are obtained. These curves are called characteristic curves. The following are the important characteristic curves of a turbine.

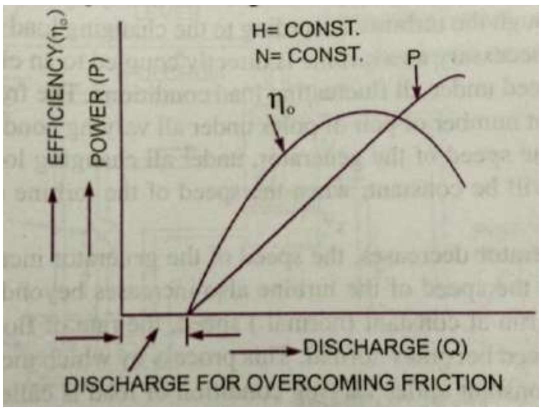
1. Main Characteristic Curves or Constant Head Curves.
2. Operating Characteristic Curves or Constant Speed Curves.
3. Muschel Curves or Constant Efficiency Curves.

1. Main Characteristic Curves or Constant Head Curves:

These curves are obtained by maintaining a constant head and a constant gate opening (G.O.) on the turbine. The speed of the turbine is varied by changing the load on the turbine. For each value of the speed, the corresponding values of the power (P) and discharge (Q) are obtained. Then the overall efficiency (η_0) for each value of the speed is calculated. From these readings the values of unit speed (N_u), unit power (P_u) and unit discharge (Q_u) are determined. Taking as N_u abscissa, the values of Q_u P_u P and η_0 are plotted. By changing the gate opening, the values of Q_u and and N_u are determined and taking N_u as abscissa, the values of Q_u P_u and η_0 are plotted.

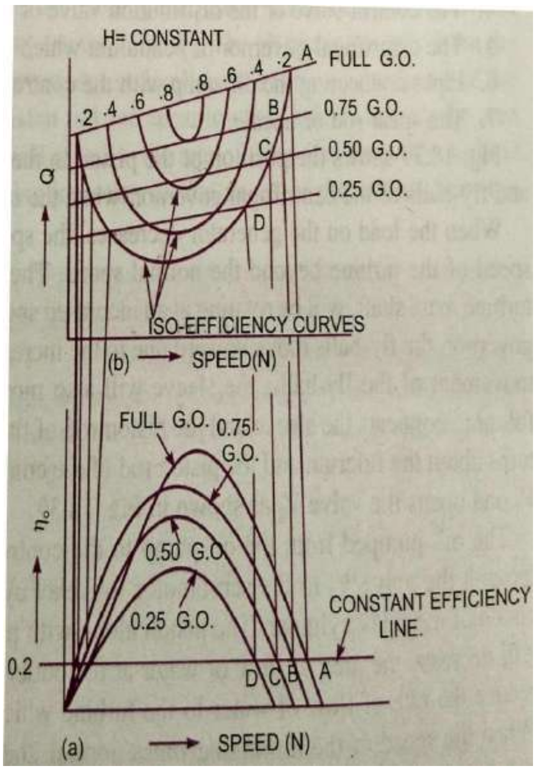
2. Operating Characteristic Curves or Constant Speed Curves:

These curves are plotted when the speed on the turbine is constant. In case turbines, the head is generally constant. As already discussed there are three independent parameters namely *N, H* and *Q*. For operating characteristics *N* and *H* are constant and hence the variation of power and the efficiency with respect to discharge *Q* are plotted. The power curve for the turbine shall not pass through the origin, because certain amount of discharge is needed to produce power to overcome initial friction. Hence the power and efficiency curves will be slightly away from the origin on the x-axis, as to overcome initial friction certain amount of discharge will be required.



3. Constant Efficiency Curves or Muschel Curves or Iso - Efficiency Curves:

These curves are obtained from the speed *vs.* efficiency and speed *vs.* discharge curves for different gate openings. For a given efficiency η_0 from the N_u *vs.* curves there are two speeds. From the N_u *vs.* Q_u curves, corresponding to two values of speeds there are two values of discharge. Hence for a given efficiency there are two values of discharge for a particular gate opening. This means for a given efficiency there are two values of speeds and two values of discharge for a given gate opening. If the efficiency is maximum there is only one value. These two values of speed and two values of discharge. Corresponding to a particular gate opening are plotted. The procedure is repeated for different gate openings and the curves *Q vs. N* are plotted. The points having the same efficiencies are joined. The curves having the same efficiency are called Iso-efficiency curves. There curves are helpful for determining the zone of constant efficiency and for predicating the performance of the turbine at various efficiencies.



For plotting the Iso-efficiency curves, horizontal lines representing the same efficiency are drawn on the $\eta_0 \sim$ speed curves. The points at which these lines cut the efficiency curves at various gate opening are transferred to the corresponding $Q \sim$ speed curves. The points having the same efficiency are then joined by smooth curves. These smooth curves represent the Iso-efficiency curves.

UNIT-VI
Centrifugal Pumps

Objective: To understand the working principles of hydraulic machines

Outcome: To explain the components, function, and use of different types of pumps.

Syllabus:

UNIT – VI: Centrifugal Pumps: Introduction to pumps, Centrifugal pump- Components, Working, Types, Work done, Heads, Losses and efficiencies, specific speed; Multi stage pumps- series, parallel; characteristic curves, NPSH, Cavitation.

Introduction

A pump is a hydraulic machine which converts mechanical energy into hydraulic energy or pressure energy. A centrifugal pump is also known as a Rotodynamic pump or dynamic pressure pump. It works on the principle of centrifugal force. In this type of pump the liquid is subjected to whirling motion by the rotating impeller which is made of a number of backward curved vanes. The liquid enters this impeller at its center or the eye and gets discharged into the casing enclosing the outer edge of the impeller. The rise in the pressure head at any point/outlet of the impeller is Proportional to the square of the tangential velocity of the liquid at that point *(i.e,* $\alpha u^2/2g$ *)* . Hence at the outlet of the impeller where the radius is more the rise In pressure head will be more and 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 higher level. Generally centrifugal pumps are made of the radial flow type only. But there are also axial flow or propeller pumps which are particularly adopted for low heads.

Advantages of centrifugal pumps:-

1. Its initial cost is low
2. Efficiency is high.
3. Discharge is uniform and continuous
4. Installation and maintenance is easy.
5. It can run at high speeds, without the risk of separation of flow

Comparisons of Centrifugal Pump over Reciprocating Pump

Sl No.	Centrifugal Pump	Reciprocating Pump
1	It is one of the rotary pumps which used kinetic energy of impeller.	It is a positive displacement type pump which is forced by piston.
2	It continuously discharges the fluid.	It does not discharge the fluid continuously.
3	In centrifugal pump the flow rate decreases which increasing the pressure.	The pressure does not affect flow rate in reciprocating pumps

4	It is used for pumping high viscous fluid.	It is used for pump low viscous fluid
5	In this pumps discharge is inversely promotional to the viscosity of fluid.	In reciprocating pump viscosity of fluid does not affect the discharge rate.
6	Centrifugal pump have problem of priming.	It does not have any problem of priming.
7	Efficiency of these pumps are low compare to reciprocating pump.	Efficiency is high
8	It uses impellers to transfer energy to fluid.	It uses piston cylinder device to transfer energy to fluid.
9	It gives higher discharge at low heads.	These gives higher heads at low discharge.
10	It is less costly	These are costly
11	These pumps required less maintenance.	These required higher maintenance.
12	Centrifugal pumps are easy to install. These required less floor space.	These pumps are difficult to install. These required more floor area.
13	It is mostly used for domestic purpose and where higher discharge at low head required.	These are mostly used in industries and high viscous fluid pumped at a high head

Components of a centrifugal pump
The main components of a centrifugal pump are:

i) Impeller ii) Casing iii) Suction pipe iv) Foot valve with strainer, v) Delivery pipe vi) Delivery valve.

Impeller is the rotating component of the pump. It is made up of a series of curved vanes. The impeller is mounted on the shaft connecting an electric motor.

Casing is an air tight chamber surrounding the impeller. The shape of the casing is designed in such a way that the kinetic energy of the impeller is gradually changed to potential energy. This is achieved by gradually increasing the area of cross section in the direction of flow.

Suction pipe It is the pipe connecting the pump to the sump, from where the liquid has to be lifted up.

Foot valve with strainer the foot valve is a non-return valve which permits the flow of the liquid from the sump towards the pump. In other words the foot valve opens only in the upward direction. The strainer is a mesh surrounding the valve; it prevents the entry of debris and silt into the pump.

Delivery pipe is a pipe connected to the pump to the overhead tank.

Delivery valve is a valve which can regulate the flow of liquid from the pump.

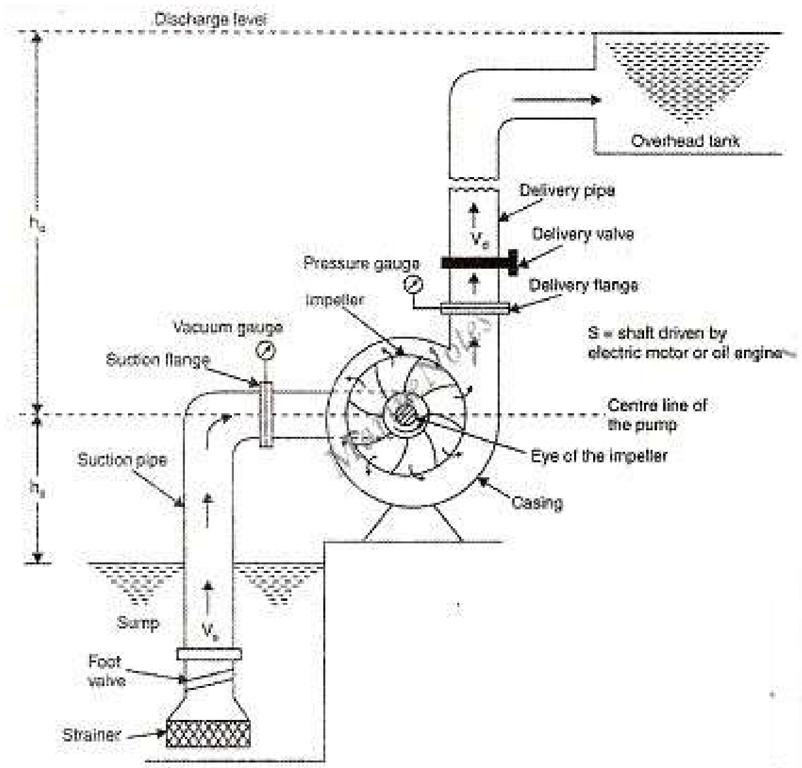


Fig 1. Centrifugal Pump Components

Priming of a centrifugal pump

Priming is the process of filling the suction pipe, casing of the pump and the delivery pipe upto the delivery valve with the liquid to be pumped. If priming is not done the pump cannot deliver the liquid due to the fact that the head generated by the Impeller will be in terms of meters of air which will be very small (because specific weight of air is very much smaller than that of water). Priming of a centrifugal pump can be done by any one of the following methods:

i) Priming with suction/vacuum pump.
ii) Priming with a jet pump.
iii) Priming with separator.
iv) Automatic or self priming.

Classification of centrifugal pumps:

Centrifugal pumps may be classified Into the following types

1.According to casing design
   a) Volute pump b) diffuser or turbine pump
2. According to number of impellers
   a) Single stage pump b) multistage or multi impeller pump

3. According to number of entrances to the impeller:
   a) Single suction pump
   b) Double suction pump

4. According to disposition of shaft
   a) Vertical shaft pump
   b) Horizontal shaft pump

5. According to liquid handled
   a) Semi open impeller
   b) Open impeller pump

6. According to specific speed
   a) Low specific speed or radial flow impeller pump
   b) Shrouded impeller
   c) Medium specific speed or mixed flow impeller pump
   d) High specific speed or axial flow type or propeller pump.

7. According to head (H)
   - Low head if $H<15m$
   - Medium head if $15<H<40m$
   - High head if $H>40m$

In the case of a <u>volute pump</u> a spiral casing is provided around the impeller. The water which leaves the vanes is directed to flow in the volute chamber circumferentially. The area of the volute chamber gradually increases in the direction flow. Thereby the velocity reduces and hence the pressure increases. As the water reaches the delivery pipe a considerable part of kinetic energy is converted into pressure energy. However, the eddies are not completely avoided , therefore some loss of energy takes place due to the continually increasing quantity of water through the volute chamber.

In the case of a diffuser pump the guide wheel containing a series of guide vanes or diffuser is the additional component. The diffuser blades which provides gradually enlarging passages surround the impeller periphery. They serve to augment the process of pressure built up that is normally achieved in the volute casing. Diffuser pumps are also called turbine pumps in view of their resemblance to a reaction turbine.
Multistage pumps and vertical shaft deep-well pumps fall under this category.
Centrifugal pumps can normally develop pressures upto 1000kpa (100m). If higher pressures are required there are three options.
a) Increase of impeller diameter. b) Increase of Rpm. c)Use of two or more impellers in series.

A radial flow impeller has small specific speeds (300 to 1000) & is suitable for discharging relatively small quantities of flow against high heads. The direction of flow at exit of the impeller is radial. The mixed flow type of impellers has a high specific speed (2500 to 5000), has large inlet diameter D and impeller width B to handle relatively large discharges against medium heads. The axial flow type or propeller impellers have the highest speed range (5000 to 10,000). They are capable of pumping large discharges against small heads. The specific speed of radial pump will be $10<Ns<80$, Axial pump $100<Ns<450$, Mixed flow pump $80<Ns<160$.

Working of a centrifugal Pump:

A centrifugal pump works on the principal that when a certain mass of fluid is rotated by an external source, it is thrown away from the central axis of rotation and a centrifugal head is impressed which enables it to rise to a higher level. Working operation of a centrifugal pump is explained in the following steps.

1) Close the delivery valve and prime the pump.
2) Start the motor connected to the pump shaft, this causes an increase in the impeller pressure.
3) Open the delivery valve gradually, so that the liquid starts flowing into the deliver pipe.
4) A partial vacuum is created at the eye of the centrifugal action, the liquid rushed from the sump to the pump due to pressure difference at the two ends of the suction pipe.
5) As the impeller continues to run, move & more liquid is made available to the pump at its eye. Therefore impeller increases the energy of the liquid and delivers it to the reservoir.
6) While stopping the pump, the delivery valve should be closed first, otherwise there may be back flow from the reservoir. It may be noted that a uniform velocity of flow is maintained in the delivery pipe. This is due to the special design of the casing. As the flow proceeds from the tongue of the casing to the delivery pipe, the area of the casing increases. There is a corresponding change in the quantity of the liquid from the impeller. Thus a uniform flow occurs in the delivery pipe.

Heads on a centrifugal pump:

Suction head (h_s): it is the vertical distance between the liquid level in the sump and the center line of the pump. It is expressed as meters.

Delivery head (h_d): It is the vertical distance between the centre line of the pump and the liquid level in the overhead tank or the supply point. It is expressed in meters.

Static head (H_s): It is the vertical difference between the liquid levels In the overhead tank and the sump, when the pump is not working. It is expressed as meters.

Therefore, $\mathbf{H_S = (hs + h_d)}$

Friction head (hf): It is the sum of the head loss due to the friction in the suction and delivery pipes. The friction loss in both the pipes is calculated using the Darcy"s equation,

$$h_f=(fLV^2/2gD).$$

Total head (H): It is the sum of the static head Hs, friction head (h_f) and the velocity head in the delivery pipe ($V_d{}^2/2g$). Where, V_d=velocity in the delivery pipe.

$$H = hs+hf+hd+(V_d{}^2/2g).$$

Manometric head (Hm): It is the total head developed by the pump. This head is slightly less than the head generated by the impeller due to some losses in the pump.

$$H_m= H+(V_s{}^2/2g).- (V_d{}^2/2g).$$

Efficiencies of centrifugal pump

Manometric efficiency (η): it is the ratio of the manometric head to the head actually generated by the impeller

$$\eta_{mano} = \left\{\frac{Hm}{Vw_2u2/g}\right\} = \left\{\frac{gHm}{Vw_2u_2}\right\}$$

Mechanical efficiency (η_{mech}): It is the ratio of the impeller power to the power of the motor or the prime mover.

η_{mech} =

Overall efficiency (η_o): It is the ratio of the work done by the pump in lifting water against gravity and friction in the pipes to the energy supplied by the motor.

Work done by the impeller of a centrifugal pump:

Figure shows the velocity triangles at the inlet and outlet tips of a vane fixed to the impeller.

Let N=speed of the impeller in RPM

U_1 = Tangential velocity of the impeller at inlet $\pi D1N/60$

U_2= tangential velocity of the impeller at outlet $\pi D2N/60$

V_1=absolute velocity of the liquid at inlet

V_2= absolute velocity of the liquid at outlet.

V_{f1} & V_{f2} are the velocities of flow at inlet and outlet.

Vr1 & Vr2 Relative velocities at inlet and outlet

Vw2 whirl velocity at outlet

α angle made by V_1 with respect to the motion of the vane

θ blade angle at inlet

ϕ = blade angle at outlet

For a series of curved vanes the force exerted can be determined using the impulse momentum equation Work=force x distance.

Similarly the work done/sec/unit weight of the liquid striking the vane=

But for a centrifugal pump $V_{\omega 1} = 0$

Work done/sec/unit weight =

And the work done/sec =

Where Q=volume of liquid flowing per second = Area x velocity of flow

$Q= \pi D_2 B_2 V_{f2}$

<u>Minimum speed for starting a centrifugal pump</u>

When a centrifugal pump is started, Will start delivering liquid only if the pressure rise in the impeller is more than or equal to the manometric head (H_{mano}). In other words, there will be no flow of liquid until the speed of the pump is such that the required centrifugal head caused by the centrifugal force or rotating water when the impeller is rotating, but there is no flow i.e flow will commence only if

For minimum starting speed, we must have

We know η_{mano} =

U_1= , U_2= ,

By substituting them

= η_{mano} x

Dividing both sides by □and simplifying N= N_{min}=

Multistage centrifugal pumps:

When a centrifugal pump consist of two or more impellers the pump is know as a multistage centrifugal pump. The important functions of a multistage centrifugal pump are;

(i) To produce high head (pumps in series)

(ii) To deliver or discharge large quantities of a liquid (pumps in parallel)

Pumps in parallel: it is an arrangement made by mounting a number of impellers on the shaft of a motor as shown. Such an arrangement is useful when the liquid has to be pumped to large heights keeping the discharge constant. If, H_m is the head developed by one impeller n= number of impellers.

Then, nxH_m= total head developed by the pump

Q=discharge through the pump.

Pumps in parallel: it is an arrangement made by connecting a number of pumps in parallel as shown. Such an arrangement is useful when a large quantity of liquid is to be pumped to a particular height. If Q=discharge from one pump N=identical number pumps.

Then, nxQ= total discharge delivered by the pump

H_m is the head developed by the pump

Specific speed (Ns):

The specific speed of a centrifugal pump is defined as the speed of a geometrically similar pump which would deliver unit quantity ($1m^3/s$) against a unit head (1m). It is denoted by Ns. Specific speed is a characterstic of pumps which can be used as a basis for comparing the performance of different pumps. Expression for specific speed(N_s).

Expression for specific speed(Ns)

$$N_s = \left(\frac{N\sqrt{Q}}{Hm^{\frac{3}{4}}} \right)$$

Note : Please refer derivation from text book most important

Performance of centrifugal pumps:

Generally a centrifugal pump is worked under its maximum efficiency conditions, however when the pump is run at conditions other than this it performs differently. In order to predict the behaviour of the pump under varying conditions of speed, discharge and head, full scale tests are usually performed. The results of these tests are plotted in the form of characteristic curves. These curves are very useful for predicting the performance of pumps under different conditions of speed, discharge and head. Following four types of characteristic curves are usually prepared for a centrifugal pump.

a. Main characteristic.

b. Operating characteristics

c. Constant efficiency or Muschel characteristic.

d. Constant head and constant discharge curves.

a. Main Characteristic: the pump is operated a particular constant speed, discharge is varied by adjusting the delivery valve. Manometric head H_m and the shaft power P are measured for each discharge. The overall efficiency is then calculated. The curves are plotted between H_m & Q, P & Q, & Q. A set of similar curves are plotted by running the pump at different speeds. They will be as shown.

b. Operating characteristic: The curves are obtained by running the pump at the design speed, which is also the driving speed of the motor. The design discharge and head are obtained from the corresponding Curves, where the efficiency is maximum as shown.

c. Constant efficiency curves: The constant efficiency curves are obtained from the main characteristic curves. The line of maximum efficiency is obtained by joining the points of the maximum curvature of the constant efficiency lines. These curves are useful in determining the range of operation of a pump.

d. Constant head and constant discharge curves: If the pump has a variable speed, the plots between Q and N and that between H_m and N may be obtained by varying the speed. In the first case H_m is kept constant & in the second Q is kept constant.

Cavitation in Centrifugal Pumps:- In centrifugal pumps the cavitation may occur at the inlet of the impeller of the pump, or at the suction side of the pumps, where the pressure is considerably reduced. Hence if the pressure at the suction side of the pump drops below the vapour pressure of the liquid then the cavitation may occur. The cavitation in a pump can be noted by a sudden drop in efficiency and head. In order to determine whether cavitation will occur in any portion of the suction side of the pump, the critical value of Thoma's cavitation factor (σ) is calculated.

Thoma's Cavitation Factor for Centrifugal Pumps:-The mathematical expression for Thoma's cavitation factor for centrifugal pump is given by

$$\sigma = \quad = \quad \ldots\ldots\ldots(2)$$

Where H_{atm} = Atmospheric pressure head in m of water or absolute pressure head at the liquid surface in pump,

H_v = Vapour pressure head in m of water,

H_S = Suction pressure head in m of water,

H_{LS} = Head lost due to friction in suction pipe, and

H = Head developed by the pump.

The value of Thoma's cavitation factor (σ) for a particular type of turbine or pump is calculated from equations 1 or 2. This value of Thoma's cavitation factor (σ) is compared with critical cavitation factor (σ_C) for that type of turbine pump. If the value of σ is greater than σ_C, the cavitation will not occur in that turbine or pump. The critical cavitation factor (σ_C) may be obtained from tables or empirical relationships.

The harmful effects of cavtation are:

a) Pitting and erosion of surfaceSudden drop in head, efficiency and power delivered to the fluid.
b) Noise and vibration produced by the collapse of bubbles.

The factors which facilitate outlet of Cavtation are as follows:

a) Restricted section

b) High runner speed

c) Too high specific speed for optimum design parameters

d) Too high temperature of the flowing liquid.