SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

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

www.sathyabama.ac.in

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

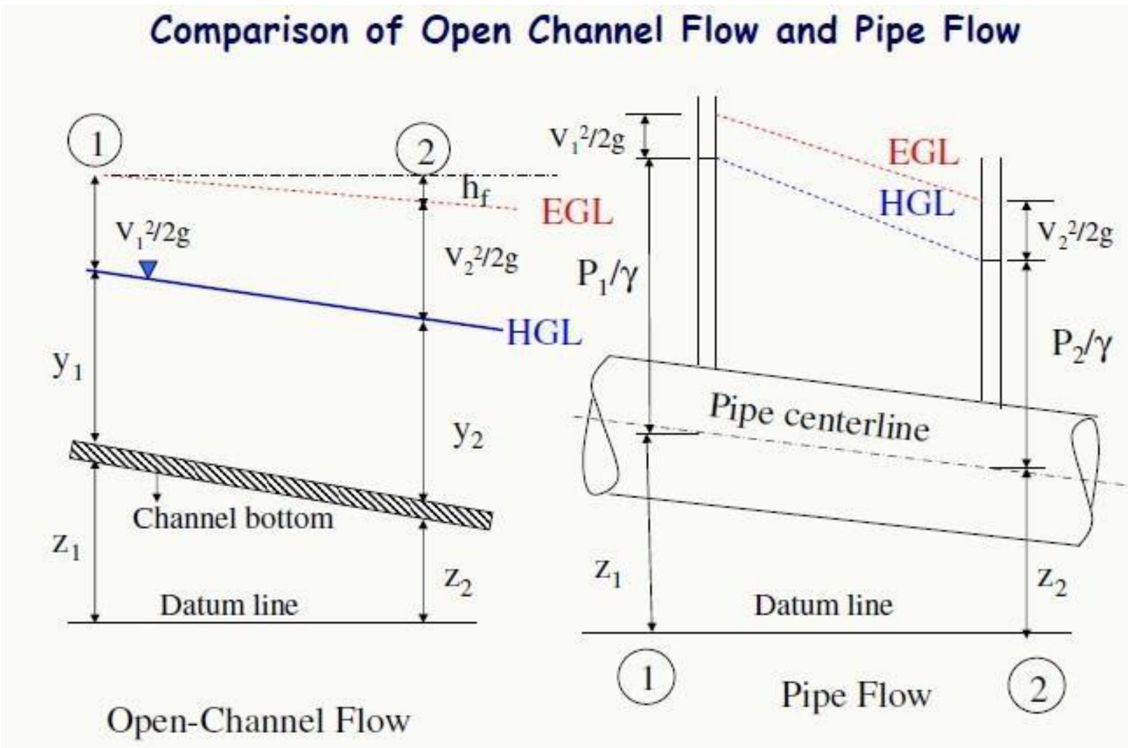
UNIT – I – APPLIED HYDRAULIC ENGINEERING – SCIA1402

1.0 Introduction

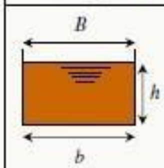
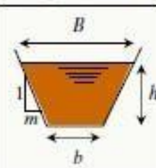
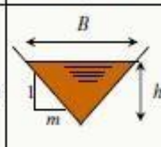
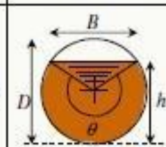
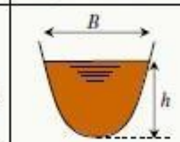
1.1 Rigid and Mobile Boundary Channels:

Rigid channels are those in which the boundary is not deformable. The shape and roughness magnitudes are not functions of flow parameters. For example, lined canals and non erodible unlined canals. In Rigid channels the flow velocity and shear stress distribution will be such that no major scouring, erosion or deposition will take place in the channel and the channel geometry and roughness are essentially constant with respect to time.

When the boundary of the channel is mobile and flow carries considerable amounts of sediment throughsuspension and is in contact with the bed. Such channels are classified as mobile channels. In the mobile channel, not only depth of flow but also bed width, longitudinal slope of channel may undergo changes with space and time depending on type of flow.
The resistance to flow, quantity of sediment transported and channel geometry all depends on interaction of flow with channel boundaries. A general mobile boundary channel can be considered to have four degree of freedom. In rigid channel we have one degrees of freedom.



Geometric elements for different channel cross sections

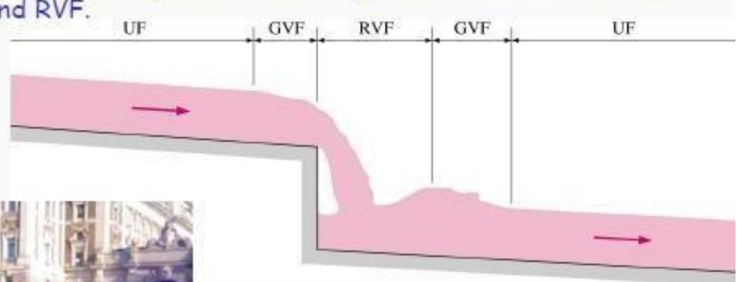
	rectangular	*trapezoidal*	*triangular*	*circular*	*parabolic*
					
flow area A	bh	$(b+mh)h$	mh^2	$\frac{1}{8}(\theta-\sin\theta)D^2$	$\frac{2}{3}Bh$
wetted perimeter P	$b+2h$	$b+2h\sqrt{1+m^2}$	$2h\sqrt{1+m^2}$	$\frac{1}{2}\theta D$	$B+\frac{8}{3}\frac{h^2}{B}$ *
hydraulic radius R_h	$\frac{bh}{b+2h}$	$\frac{(b+mh)h}{b+2h\sqrt{1+m^2}}$	$\frac{mh}{2\sqrt{1+m^2}}$	$\frac{1}{4}\left[1-\frac{\sin\theta}{\theta}\right]D$	$\frac{2B^2h}{3B^2+8h^2}$ *
top width B	b	$b+2mh$	$2mh$	$(\sin\theta/2)D$ or $2\sqrt{h(D-h)}$	$\frac{3}{2}\frac{A}{h}$
hydraulic depth D_h	h	$\frac{(b+mh)h}{b+2mh}$	$\frac{1}{2}h$	$\left[\frac{\theta-\sin\theta}{\sin\theta/2}\right]\frac{D}{8}$	$\frac{2}{3}h$

* *Valid for* $0<\xi\le 1$ *where* $\xi=4h/B$

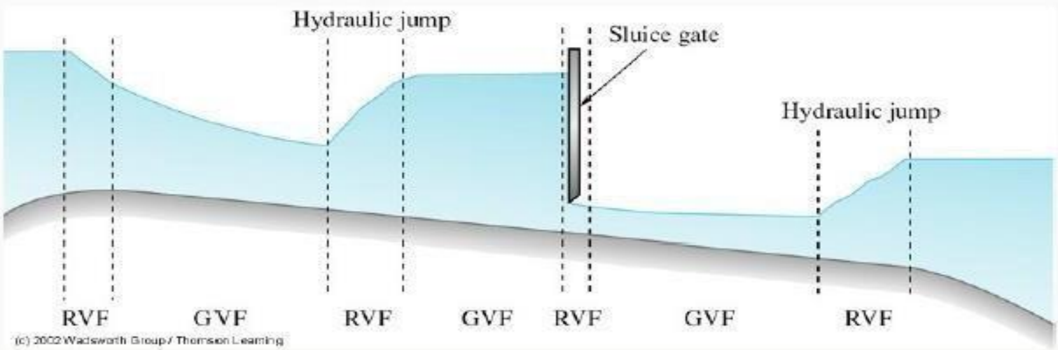
If $\xi>1$ *then* $P=(B/2)\left[\sqrt{1+\xi^2}+(1/\xi)\ln\left(\xi+\sqrt{1+\xi^2}\right)\right]$

Classification of Open-Channel Flows

- Obstructions cause the flow depth to vary.
- Rapidly varied flow (RVF) occurs over a short distance near the obstacle.
- Gradually varied flow (GVF) occurs over larger distances and usually connects UF and RVF.



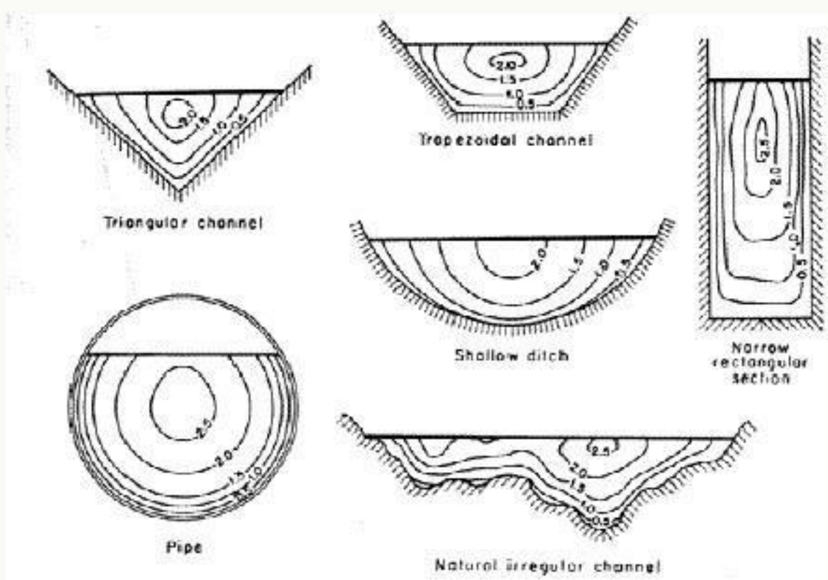
Steady non-uniform flow in a channel.



Velocity Distribution

The velocity distribution in an open-channel flow is quite nonuniform because of :

- Nonuniform shear stress along the wetted perimeter,
- Presence of free surface on which the shear stress is zero.



1.2 Specific Energy

Bernoulli's theorem expresses the energy content of unit volume of fluid as $U = \rho gz + p + \rho V^2/2$, and states that it is constant along a streamline in the absence of dissipation. It is usually more convenient to express the terms as lengths, or "heads," by dividing by $\gamma = \rho g$: $H = z + p/\gamma + V^2/2g$. The three energy components are elevation, pressure, and velocity. All play a role in open-channel flow. For any flow, there is an *energy grade line* that can be imagined above the flow, and its slope is S'. The water surface is the *hydraulic grade line* (HGL), which is below the energy grade line by the velocity head $V^2/2g$. Below this is the *bed grade line*, with slope S, and (usually) below that is the horizontal *datum*, the reference surface.

The streamlines of the flow are parallel. Along any streamline, $z + p/\gamma + V^2/2g$ is a constant. Let us now assume z is the elevation of the bottom streamline, so that if the flow depth is y, the elevation of the surface streamline is z + y. The gauge pressure here is zero, so Bernoulli's Equation for this streamline is $z + y + V^2/2g = C$. Now for the bottom streamline, the gauge pressure is γy, so that $p/\gamma = y$, and Bernoulli's equation is $z + y + V^2 = C$, where C has the same value as for the top streamline. At any intermediate height y', $z + y' + \gamma(y - y')/\gamma + V^2/2g = C$. Therefore, C, the energy per unit weight, has the same value at any depth. The part $y + V^2/2g$ is called the *specific energy* E, and is the energy per unit weight referred to the stream bed.

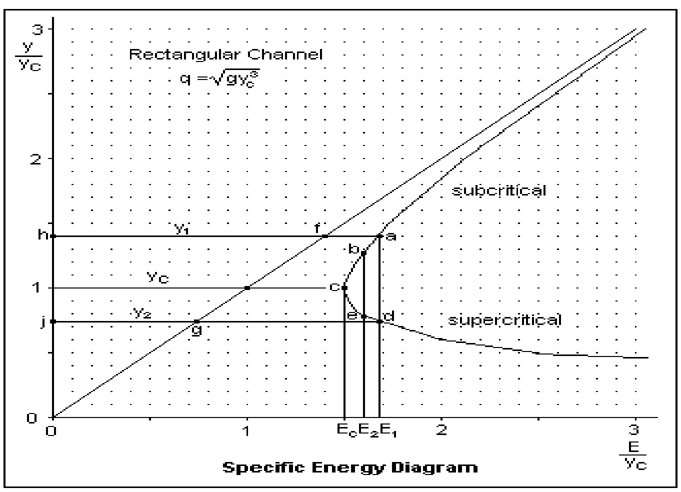
When a closed channel runs full, then the depth can no longer vary to accommodate thedischarge, and the pressure becomes different from the atmospheric pressure, and must be taken into account in using Bernoulli's theorem. This is the fundamental difference between open channel flow and pipe flow.

We can express E as a function of Q easily, using Q = AV: $E = y + Q^2/2gA^2$. For simplicity, consider a rectangular channel of width b, for which A = by. Then $E = y + q^2/2gy^2$, expressing E as a function of the discharge q and the depth y, or $q = y\sqrt{[2g(E - y)]}$, expressing q as a function of E and y.

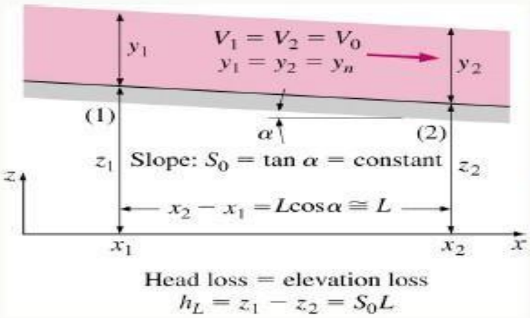
The curve of q as a function of y for a fixed E is plotted at the right. We notice that q is a double-valuedfunction of y, and has a maximum possible value q_m. The corresponding depth y can be found by differentiating q with respect to y and setting the derivative equal to zero. The result is $y_c = 2E/3$, called the *critical depth*. The corresponding value of q, $q_m = \sqrt{(gy_c^3)}$ is the critical flow, and $V_c = q_c/y_c$ is the critical velocity. For depths greater than the critical depth, the velocity is smaller than the critical velocity.Flow in this region is called *subcritical*. For depths smaller than the critical depth, the velocity is greater than the critical velocity. Flow in this region is called *supercritical*. Note that the sub- and super- refer to the velocity of flow. The same discharge q is possible with given E in either region. In the upper region, we have greater flow area, in the lower region greater flow velocity. Because the frictional resistance varies rapidly with velocity, subcritical uniform flow is associated with gentle slopes, supercritical uniform flow with steep slopes.

Note that the curve is plotted with respect to dimensionless variables, so the same curve can be used for any E or q_m. Consider flows described by points a and c. Since they are on the same vertical line, the discharge is the same for each. The distance from the y = 0 axis to point a corresponds to the static part of E, while the distance from a to the y = E line corresponds to the kinetic energy, which sum to E. The sameholds for point c, but here the static part is much smaller and the dynamic part larger. At the point of maximum discharge for this value of E, point e, the static energy is twice the dynamic energy.

Suppose that there is a lateral constriction in the channel, reducing its area so that q/q_m increases from 0.6 to 0.8. Assume there is no head loss, the specific energy does not change, so the flows in the constriction are represented by points b and d. We note that in subcritical flow, the depth of flow decreases, while in supercritical flow the depth increases.

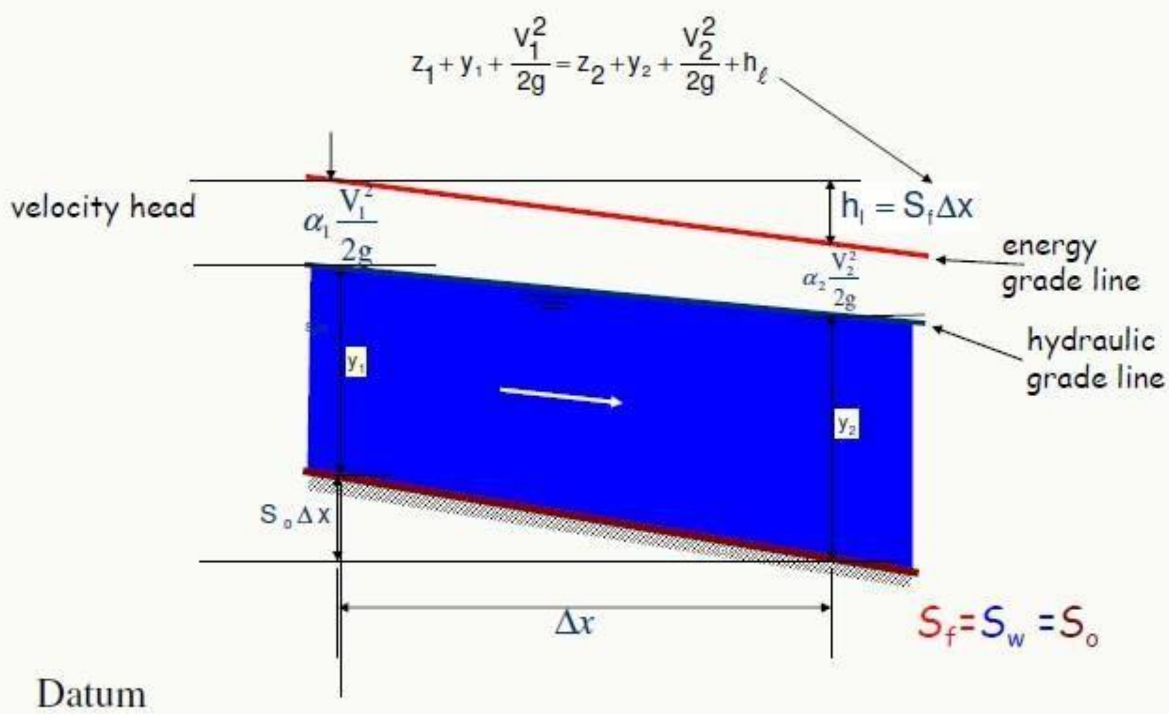


Uniform Flow in Channels



- Flow in open channels is classified as being ***uniform*** or ***nonuniform***, depending upon the depth *y*.
- Depth in Uniform Flow is called **normal depth y_n**
- ***Uniform depth*** occurs when the flow depth (and thus the average flow velocity) remains **constant**
- Common in long straight runs
- Average flow velocity is called **uniform-flow velocity V_o**
- *Uniform depth* **is maintained as long as the slope, cross-section, and surface roughness of the channel remain unchanged.**
- During uniform flow, the terminal velocity reached, and the head loss equals the elevation drop

Uniform Flow in Channels



Problem : *A trapezoidal channel has side slopes of 1 horizontal to 2 vertical and the slope of the bed is 1 in 1500. The area of the section is 40 m^2. Find the dimensions of the section if it is most economical. Determine the discharge of the most economical section if C = 50.*

Solution. Given :

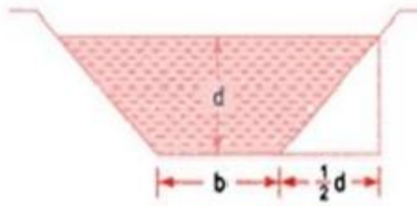


Fig. 16.12

Side slope, $n = \frac{\text{Horizontal}}{\text{Vertical}} = \frac{1}{2}$

Bed slope, $i = \frac{1}{1500}$

Area of section, $A = 40 \text{ m}^2$

Chezy's constant, $C = 50$

For the most economical section, using equation (16.11)

$$\frac{b + 2nd}{2} = d\sqrt{n^2 + 1} \quad \text{or} \quad \frac{b + 2 \times \frac{1}{2} \times d}{2} = d\sqrt{\left(\frac{1}{2}\right)^2 + 1}$$

or $$\frac{b + d}{2} = d\sqrt{\frac{1}{4} + 1} = 1.118\, d$$

or $$b = 2 \times 1.118d - d = 1.236\, d \qquad ...(i)$$

But area of trapezoidal section, $A = \frac{b + (b + 2nd)}{2} \times d = (b + nd)\ d$

$$= (1.236\, d + \tfrac{1}{2}\, d)\, d \qquad (\because \ b = 1.236\, d \text{ and } n = \tfrac{1}{2})$$

$$= 1.736\, d^2$$

But $A = 40 \text{ m}^2$ (given)

$\therefore$ $40 = 1.736\, d^2$

$\therefore$ $$d = \sqrt{\frac{40}{1.736}} = \textbf{4.80 m. Ans.}$$

Substituting the value of d in equation (i), we get

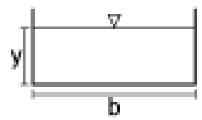
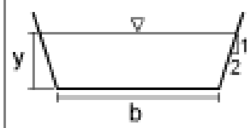
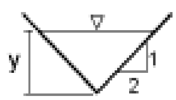
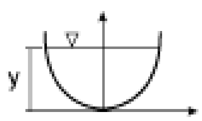
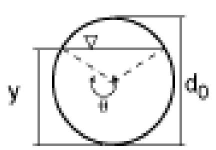
$$b = 1.236 \times 4.80 = \textbf{5.933 m. Ans.}$$

Discharge for most economical section. Hydraulic mean depth for most economical section is

$$m = \frac{d}{2} = \frac{4.80}{2} = 2.40 \text{ m}$$

$\therefore$ Discharge $$Q = AC\sqrt{mi} = 40 \times 50 \times \sqrt{2.40 \times \frac{1}{1500}}$$

$$= \textbf{80 m}^3\textbf{/s. Ans.}$$

Channel type	Area A	Wetted permiter P	Hydraulic radius R	Top width T	Hydraulic depth D
	by	$\frac{by}{b+2y}$	$\frac{by}{b+2y}$	b	y
	b+2y	$b+2y\sqrt{1+z^2}$	$\frac{(b+zy)y}{b+2y\sqrt{1+z^2}}$	b+2zy	$\frac{(b+zy)y}{b+2zy}$
	zy^2	$2y\sqrt{1+z^2}$	$\frac{zy}{2\sqrt{1+z^2}}$	2zy	$\frac{1}{2}y$
	$\frac{2}{3}Ty$	$T+\frac{8}{3}\frac{y^2}{T}$	$\frac{2T^2y}{3T^2+8y^2}$	$\frac{3}{2}\frac{A}{y}$	$\frac{2}{3}y$
	$\frac{1}{8}(\theta - \sin\theta)$	$\frac{1}{2}\theta d_0$	$\frac{1}{4}\left[1-\frac{\sin\theta}{\theta}\right]d_0$	$2\sqrt{y(d_0-y)}$	$\frac{1}{8}\left(\frac{\theta - \sin\theta}{\sin\frac{\theta}{2}}\right)d_0$

Problem 2 : Find the slope of the bed of a rectangular channel of width 5 m when depth of water is 2 m and rate of flow is given as 20 m^3/s. Take Chezy's constant, C=50.

$$\therefore \quad 20.0 = 10 \times 50 \times \sqrt{\frac{10}{9}\times i} \quad \text{or} \quad \sqrt{\frac{10}{9}}\ i = \frac{20.0}{500} = \frac{2}{50}$$

Squaring both sides, we have $\frac{10}{9}i = \frac{4}{2500}$

$$i = \frac{4}{2500} \times \frac{9}{10} = \frac{36}{2500} = \frac{1}{\frac{25000}{36}} = \frac{1}{694.44} \quad \text{Ans.}$$

∴ Bed slope is 1 in 694.44. Ans.

Problem 2 : Find the velocity of flow and rate of flow of water through a rectangular channel of 6 m wide and 3 m deep, when it is running full. The channel is having bed slope as 1 in 2000. Take Chezy's constant C = 55.

Width of rectangular channel, $b = 6$ m

Depth of channel, $d = 3$ m

∴ Area $A = 6 \times 3 = 18\ m^2$

$I = 1 \text{ in } 2000 = \frac{1}{2000}$

Chezy's constant, $C = 55$

Perimeter, $P = b + 2d = 6 + 2 \times 3 = 12$ m

∴ Hydraulic mean depth, $m = \frac{A}{P} = \frac{18}{12} = 1.5$ m

Velocity of flow is given by equation (4.4) as,

$$V = C\sqrt{mi} = 55\sqrt{1.5\ \frac{1}{2000}} = 1.506\ m/s. \text{ Ans.}$$

$$Q = V \times Area = V \times A = 1.506 \times 18 = 27.108\ m^3/s. \text{ Ans.}$$

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

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – II – APPLIED HYDRAULIC ENGINEERING – SCIA1402

UNIT II

GRADUALLY VARIED FLOW

2.1 NON UNIFORM FLOW

In a channel the depth of flow changes from section to section along the length of the channel.

Types

1. Gradually varied flow(G.V.F)
2. Rapidly Varied flow (R.V.F)

Gradually Varied Flow (G.V.F)

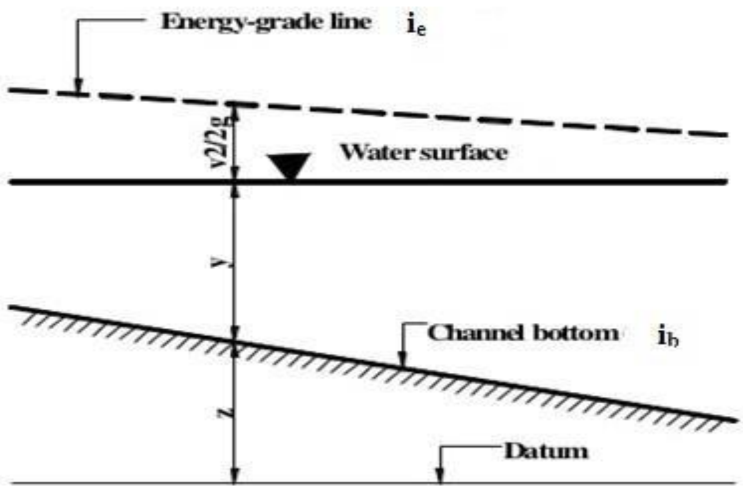
If the depth of flow in a channel changes gradually over a long length of the channel the flow is said tobe gradually varied FLOW.

Dynamic Equation Of Gradually Varied Flow (G.V.F)

Assumptions:

The bed slope of the channel is small
2.Steady flow (constant discharge)
3.Acceleration effect is negligible
4.Energy correction factor α is unity
5.Roughness co-efficient is constant
6.Chezy's and Manning's formulas are applicable
7.The Channel is Prismatic.

Consider a rectangular channel having G.V.F. The depth of flow is gradually decreasing in the directionof flow.



Let,
z- Height of bottom of channel above datum.h- depth of flow.
v- mean velocity of flow. i_b-slope of the

channel bedi_e–slope
of the energy line b-
width of the channel
Q-discharge through the channel

Energy equation at any section by Bernoullis equation,

$$E = Z+h+V^2/2g \qquad (1)$$

Differentiate equation (1) with respect to x, where x is measured along the bottom of the channel in thedirection of flow

$$\frac{dE}{dx} = \frac{dz}{dx} + \frac{dh}{dx} + \frac{d}{dx}\left(\frac{V^2}{2g}\right) (2)$$

$$\frac{d}{dx}\left(\frac{V^2}{2g}\right) = \frac{d}{dx}\left[\frac{Q^2}{A^2 X 2g}\right]$$

$\left(V = \frac{Q}{A}\ ;\ V^2 = \frac{Q^2}{A^2}\ ; V^2 = \frac{Q^2}{b^2 X\ h^2}\right)$

$$= \frac{d}{dx}\left(\frac{1}{h^2}\right)\left(\frac{Q^2}{b^2 X 2g}\right)$$

$$= \frac{Q^2}{b^2 X 2g}\frac{d}{dh}\left(\frac{1}{h^2}\right).\frac{dh}{dx}$$

$$= \frac{Q^2}{b^2 X 2g}\left(\frac{-2}{h^3}\right).\frac{dh}{dx}$$

$$= \frac{-Q^2}{b^2 X g h^2 X h}.\frac{dh}{dx}$$

$$\frac{d}{dx}\left(\frac{V^2}{2g}\right) = \frac{-V^2}{gh}.\frac{dh}{dx}$$

Substitute the value of $\frac{d}{dx}\left(\frac{V^2}{2g}\right)$ in equation (2)

$$\frac{dE}{dx} = \frac{dz}{dx} + \frac{dh}{dx} - \left(\frac{V^2}{gh}\right).\frac{dh}{dx}$$

$$\frac{dE}{dx} = \frac{dz}{dx} + \frac{dh}{dx}\left[1 - \frac{V^2}{gh}\right] \qquad (3)$$

Now, $\frac{dE}{dx} = -i_e$; $\frac{dz}{dx} = -i_b$

-ve sign of i_e,$i_b$indicates that with the increaseof x the value of

E and Z decreases. Substitute the value of $\frac{dE}{dx}$ and $\frac{dZ}{dx}$ in equation (3)

$$-i_e = -i_b + \frac{dh}{dx}[1 - \frac{V^2}{gh}]$$

$$i_b - i_e = \frac{dh}{dx}[1 - \frac{V^2}{gh}]$$

$$\frac{dh}{dx} = \frac{i_b - i_e}{[1 - \frac{V^2}{gh}]}$$

$$\frac{V^2}{\sqrt{gh}} = F_e$$

$$\frac{V^2}{gh} = F_e^2$$

$$\frac{dh}{dx} = \frac{i_b - i_e}{[1 - F_e^2]} \frac{dh}{dx}$$

When, $\frac{dh}{dx} = 0$ depth of water above the bottom of the channel is constant.

When, $\frac{dh}{dx} > 0$ depth of water increases in the direction of flow. The profile of the water so obtained iscalled back water curve.

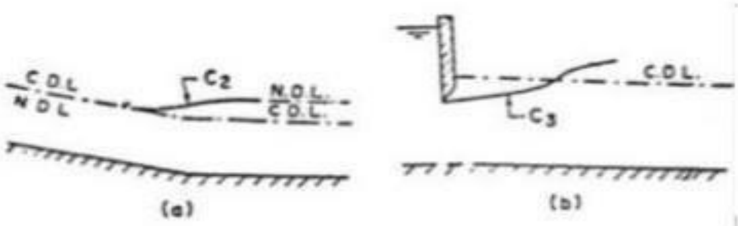
When, $\frac{dh}{dx} < 0$ depth of water decreases in the direction of flow. The profile of the water so obtained iscalled drop down curve.

CLASSIFICATION OF CHANNEL BOTTOM SLOPE AND SURFACE PROFILE

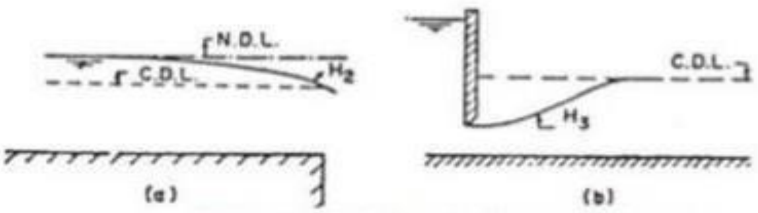
The surface curves of water are called flow profiles or (water surface profiles) Depending on the slope and profile of the bed the classification of water profile are,

1. Mild slope (M1,M2,M3)
2. Steep slope (S1,S2,S3)
3. Critical slope (C1,C2,C3)
4. Horizontal slope(H2,H3)
5. Adverse slope (A2,A3)

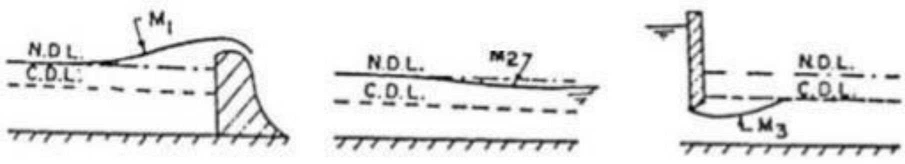
In all the curves letter indicate slope type and subscript indicate the zone.



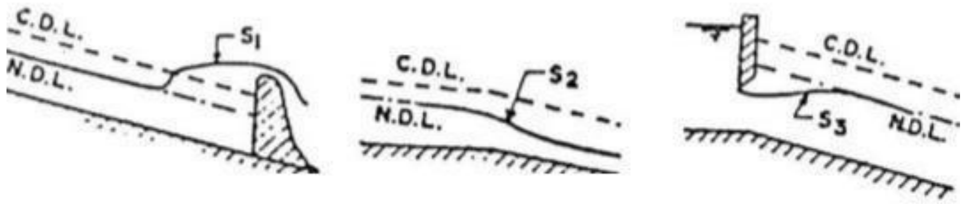
Flow Profiles in Critical slope



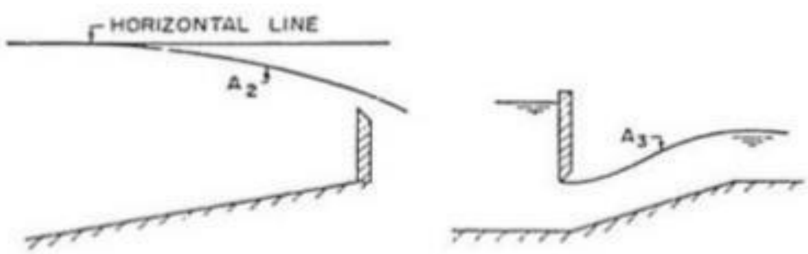
Flow Profiles in Horizontal slope



Flow Profiles in Mild slope



Flow Profiles in Steep slope



Flow Profiles in Adverse slope

Profile determination by Numerical method:

Length of surface profile is determined with the help of anyone of the following methods.1.Graphical Integration method
1.Direct step method 3.Standard step method

Graphical Integration method:

The differential equation of the gradually varied flow may be inverted and integrated to give the length of thesurface profile between Y1 and Y2 . Hence the equation for gradually varied flow is given by,

$$\frac{dy}{dx} = \frac{S - S_e}{1 - F^2}$$

$$\frac{dx}{dy} = \frac{1 - F^2}{S - S_e}$$

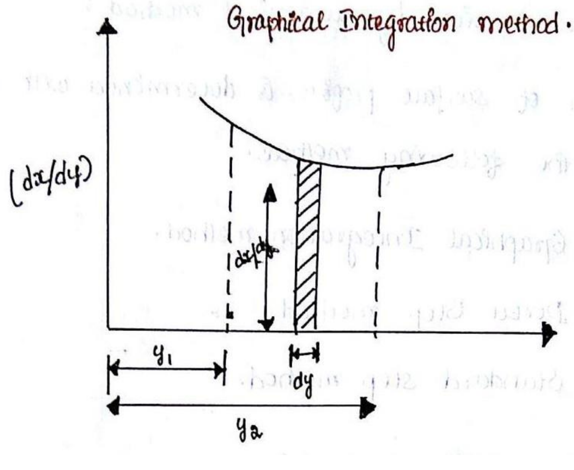
$$dx = \left[\frac{1 - F^2}{S - S_e}\right] dy$$

Integrating on both sides,

$$\int dx = \int \left[\frac{1 - F^2}{S - S_e}\right] . dy$$

It may be written as,

$$x = L = \int_{y_1}^{y_2} \left[\frac{1 - F^2}{S - S_o}\right] . dy \longrightarrow (2)$$



By Plotting dx/dy against Y and computing the area under the curve enclosed by the Y axis and the ordinatesof dx/dy corresponding to depths Y1 and Y2.

Direct Method:

The channel is divides into short distances and computations are carried out between two ends. Two ends are mentioned as sections (1-1) and (2-2). Bernoullis equation is used to calculate the length of backwater. It is called as step by step method. It is applicable to prismatic channels only.

Standard Step method:

This method is used for non-prismatic/natural channels. In natural channels, the cross-section varies fromsection to section and also the cross section is known only at a few locations along the channel. The cross section information at two adjacent sections and the discharge and datum at one section are required to determine the datum at the other section.
The section of the above problem is obtained by a trail and error method from the basic energy equation

APPLICATIONS:

Direct step method, Standard step methOd and graphical methods are used for the determination of length of the surface profile for gradually varied flows(GVF). For GVF problems, the channel is divided into short distances and computations are carried out from one end ofthe reach to the other. In direct step method, Mannings formula is sufficient to accurately evaluate the slope of the total energy line. The standard step method is mainly applicable to non-prismatic channels, example: natural river.A trial and error method is employed in this method.

Problem.
A long rectangular channel of width 5 m has a slope of 1:5000 and a Manning’s roughness coefficient n of 0.02 $m^{-1/3}$ s. The total discharge is 10 m^3 s^{-1}. The channel narrows to a width of 1.2 m over a short length.

(a) Determine the normal depth at this flow rate in the 5 m-wide channel.
(b) Show that critical conditions occur at the narrow section.
(c) Determine the depth just upstream of the narrowed section, where the width is 5 m.
(d) Determine the distance upstream to where the depth is 5% greater than the normal depth, using two steps in the gradually-varied-flow equation.

(a)
Given:

$b = 5 \text{ m} \qquad (b_{min} = 1.2 \text{ m})$

$S_0 = 0.0002$

$n = 0.02 \text{ m}^{-1/3} \text{ s}$

$Q = 10 \text{ m}^3 \text{ s}^{-1}$

For the normal depth,

$$Q = VA, \qquad \text{where} \quad V = \frac{1}{n} R_h^{2/3} S^{1/2}, \quad R_h = \frac{bh}{b + 2h} = \frac{h}{1 + 2h/b}, \qquad A = bh$$

$$\Rightarrow \qquad Q = \frac{b\sqrt{S}}{n} \frac{h^{5/3}}{(1 + 2h/b)^{2/3}}$$

Rearranging as an iterative formula for h to find the normal depth at the *channel* slope S_0:

$$h = \left(\frac{nQ}{b\sqrt{S_0}}\right)^{3/5} (1 + 2h/b)^{2/5}$$

Here, with lengths in metres:

$$h = 1.866(1 + 0.4h)^{2/5}$$

Iteration (from, e.g., $h = 1.866$) gives normal depth:

$$h_n = 2.453 \text{ m}$$

Answer: 2.45 m.

(b) To determine whether critical conditions occur, compare the total head in the approach flow with that assuming critical conditions at the throat.

The total head, assuming normal flow and measuring heights from the bed of the channel is

$$H_a = E_a = h_n + \frac{V_n^2}{2g} = h_n + \frac{Q^2}{2gb^2h_n^2} = 2.487 \text{ m}$$

At the throat the discharge per unit width is

i	h_i	x_i	h_{mid}	$(dx/dh)_{mid}$	Δx
0	2.855	0			
			2.786	17120	–2388
1	2.716	–2388			
			2.646	26800	–3739
2	2.576	**–6127**			

$$q_m = \frac{Q}{b_{\min}} = 8.333\ \mathrm{m^2\ s^{-1}}$$

The critical depth and critical specific energy at the throat are

$$h_c = \left(\frac{q_m^2}{g}\right)^{1/3} = 1.920\ \mathrm{m}$$

$$E_c = \frac{3}{2} h_c = 2.880\ \mathrm{m}$$

Since the bed of the flume is flat ($z_b = 0$), the critical head $H_c = E_c$.

Since the approach-flow head H_a is less than the critical head H_c (the minimum head required to pass this flow rate through the venturi), the flow must back up and increase in depth just upstream to supply this minimum head. It will then undergo a subcritical to supercritical transition through the throat. The total head throughout the venturi is $H = H_c = 2.880$ m.

(c) In the vicinity of the venturi the total head is $H = 2.880$ m. Just upstream (where width $b = 5$ m), we seek the subcritical solution of

$$H = z_s + \frac{V^2}{2g} = h + \frac{Q^2}{2gb^2h^2}$$

Rearrange for the deeper solution:

$$h = H - \frac{Q^2}{2gb^2h^2}$$

Here, with lengths in metres:

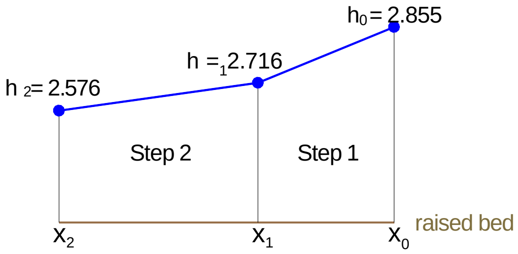
$$h = 2.880 - \frac{0.2039}{h^2}$$

Iterate (from, e.g., $h = 2.880$) to get the depth just upstream of the venturi:

$$h = 2.855\ \mathrm{m}$$

Answer: 2.85 m

(d) Do a GVF calculation (subcritical, so physically it should start at the fixed downstream control and work upstream, although mathematically it can be done the other way) from the pre-venturi depth ($h = 2.855$ m) to where $h = 2.576$ m (i.e. $1.05 \times h_n$). Using two steps the depth increment per step is



$$\Delta h = \frac{2.576 - 2.855}{2} = -0.1395\ \mathrm{m}$$

Both depth and specific-energy methods are shown on the following pages.

METHOD 1: using the depth form of the GVF equation

$$\frac{\mathrm{d}h}{\mathrm{d}x} = \frac{S_0 - S_f}{1 - \mathrm{Fr}^2}$$

$$\Rightarrow \quad \frac{\mathrm{d}x}{\mathrm{d}h} = \frac{1 - \mathrm{Fr}^2}{S_0 - S_f}$$

$$\frac{\Delta x}{\Delta h} \approx \left(\frac{\mathrm{d}x}{\mathrm{d}h}\right)_{mid}$$

('*mid*' means mid-point of interval, half way between h_i and h_{i+1}; sometimes written $h_{i+1/2}$)

For convenience, work out numerical expressions for Fr^2 and S_f in terms of h:

$$\mathrm{Fr}^2 = \frac{V^2}{gh} = \frac{Q^2/b^2}{gh^3} = \frac{0.4077}{h^3}$$

Manning's equation (see earlier) gave

$$Q = \frac{b\sqrt{S}}{n}\frac{h^{5/3}}{(1 + 2h/b)^{2/3}}$$

Assuming that the rate of loss of energy (S_f) at a general depth h is the same as the channel slope that would give normal flow at that depth, rearrangement for the slope gives

$$S_f = \left(\frac{nQ}{b}\right)^2 \frac{(1 + 2h/b)^{4/3}}{h^{10/3}} = 16 \times 10^{-4}\frac{(1 + 0.4h)^{4/3}}{h^{10/3}}$$

Hence,

$$\frac{\mathrm{d}x}{\mathrm{d}h} = \frac{1 - \mathrm{Fr}^2}{S_0 - S_f} = \frac{1 - \dfrac{0.4077}{h^3}}{\left(2 - 16 \times \dfrac{(1 + 0.4h)^{4/3}}{h^{10/3}}\right) \times 10^{-4}}$$

With

$$\Delta x = \left(\frac{dx}{dh}\right)_{mid} \Delta h \qquad \text{and} \qquad \Delta h = -0.1395 \text{ m}$$

working may then be set out in tabular form. (All depths assumed to be in metres.)

This gives a distance of about 6.1 km upstream.

Problem : Water flows at a depth of 10 cm with a velocity of 6 m/s in a rectangular channel. Is the flow subcritical or supercritical? What is the alternate depth?

Check Froude number

Fr = V/ √ gy = 6 m/s / 9.81 /s^2 x 0.1 m = 6.06 > 1

so the flow is supercritical.

E = y + V ^{2} / 2g = 0.1 m + (6 m/s)^2 2 x 9.81 m/s^2 = 1.935 m

Solving for the alternate depth for an E = 1.935 m yields y_{alt} = 1.93 m

Problem : Rain water flows on a concrete surface. For given values of flow depth and velocity, it is to be determined whether the flow is subcritical or supercritical.

Sol:

Assumptions 1 The flow is uniform. 2 The thickness of water layer is constant.

The Froude number is

$$\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{1.3 \text{ m/s}}{\sqrt{(9.81 \text{ m/s}^2)(0.02 \text{ m})}} = 2.93,$$

which is greater than 1. Therefore, the flow is supercritical

Problem : Water flows with a velocity of 2 m/s and at a depth of 3 m in a rectangular channel. What is the change in depth and in water surface elevation produced by a gradual upward change in bottom elevation (upstep) of 60 cm? What would be the depth and elevation changes if there were a gradual downstep of 15 cm? What is the maximum size of upstep that could exist before upstream depth changes would result? Neglect head losses.

$$E_1 = y_1 + \frac{V_1^2}{2g} = 3\text{ m} + \frac{(2\text{ m/s})^2}{2 \cdot 9.81\text{ m/s}^2} = 3.20\text{ m.}$$

$$E_2 = E_1 - \Delta z = 3.20\text{ m} - 0.60\text{ m} = 2.60\text{ m.}$$

Also

$$E_2 = y_2 + \frac{q^2}{2gy_2^2} = y_2 + \frac{(6\text{ m}^3\text{/s/m})^2}{2 \cdot 9.81\text{ m/s}^2 \cdot y_2^2} = 2.60\text{ m}$$

so $y_2 = 2.24$ m. $\Delta y = y_2 - y_1 = -0.76$ m so water surface drops 0.16 m.

For a downward step of 15 cm we have

$$E_2 = E_1 - \Delta z = 3.20\text{ m} - (-0.15\text{ m}) = 3.35\text{ m.}$$

giving $y_2 = 3.17$ m and $\Delta y = y_2 - y_1 = 0.17$ m so water surface rises 0.02 m.

The maximum upstep possible before affecting upstream water surface levels is for $y_2 = y_c$

$$y_c = \sqrt[3]{\frac{q^2}{g}} = \sqrt[3]{\frac{(6\text{ m}^3\text{/s/m})^2}{9.81\text{ m/s}^2}} = 1.54\text{ m.}$$

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

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – III– APPLIED HYDRAULIC ENGINEERING – SCIA1402

UNIT III

RAPIDLY VARIED FLOW

3.1 RAPIDLY-VARIED FLOW (RVF)

Rapidly-varied flow is a significant change in water depth over a short distance (a few times water depth). It occurs where there is a local disturbance to the balance between gravity and friction (e.g. at a weir, venturi, sluice, free overfall, sudden change in slope,) or a mismatch between the depths imposed by upstream and downstream conditions (hydraulic jump).

Often there is a flow transition between deep, slow flow (subcritical; $Fr < 1$) and shallow, fast flow (supercritical; $Fr > 1$).

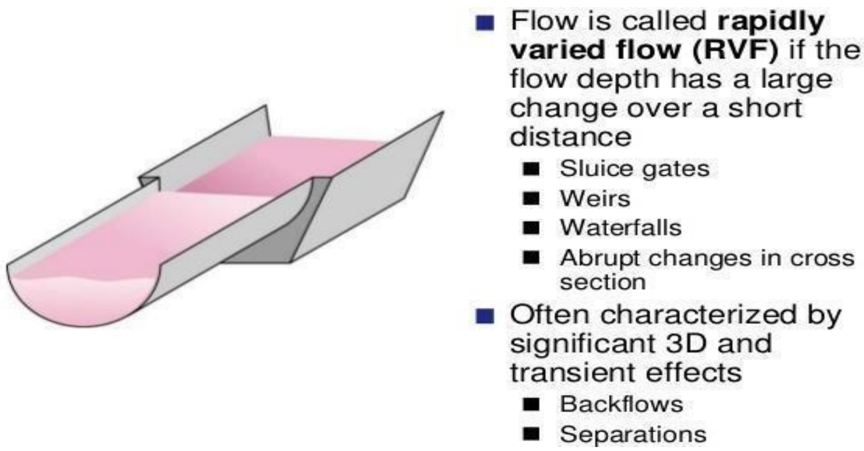
The assumption that the flow varies rapidly over a relatively short distance means that bed friction is unimportant. Thus, for a smooth transition (e.g. weir, venturi or sluice), the total head is usually assumed constant through this short region.

For an abrupt transition (hydraulic jump) there may be significant head loss, but it is associated with high levels of turbulence, not bed friction. Note: that the hydrostatic pressure assumption can only be applied where near-parallel flow has been established, either side of the rapidly-varying-flow region.

Hydraulic Jumps

Whenever the flow profile changes from supercritical to subcritical, hydraulic jumps will occur. A hydraulic jump represents a significant head loss that manifests in available energy for scour and creation of turbulence.

A hydraulic jump is an abrupt change from a shallow, high-speed flow to a deep, low-speed flow of lower energy. It occurs when a depth difference is imposed by upstream and downstream conditions. Rapid, shallow flow may be created by, for example, a steep spillway or sluice. A slower and deeper downstream flow may be controlled by a downstream weir or by a reduction in slope. The triggering of a hydraulic jump at the base of a spillway is desirable to remove surplus kinetic energy in order to reduce downstream erosion.



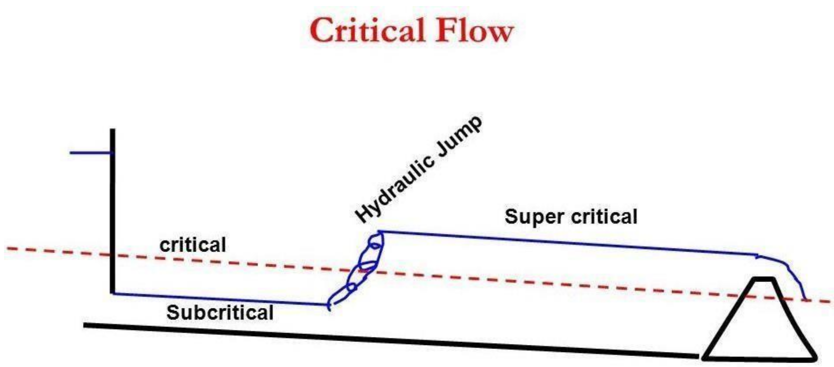
Across a hydraulic jump:

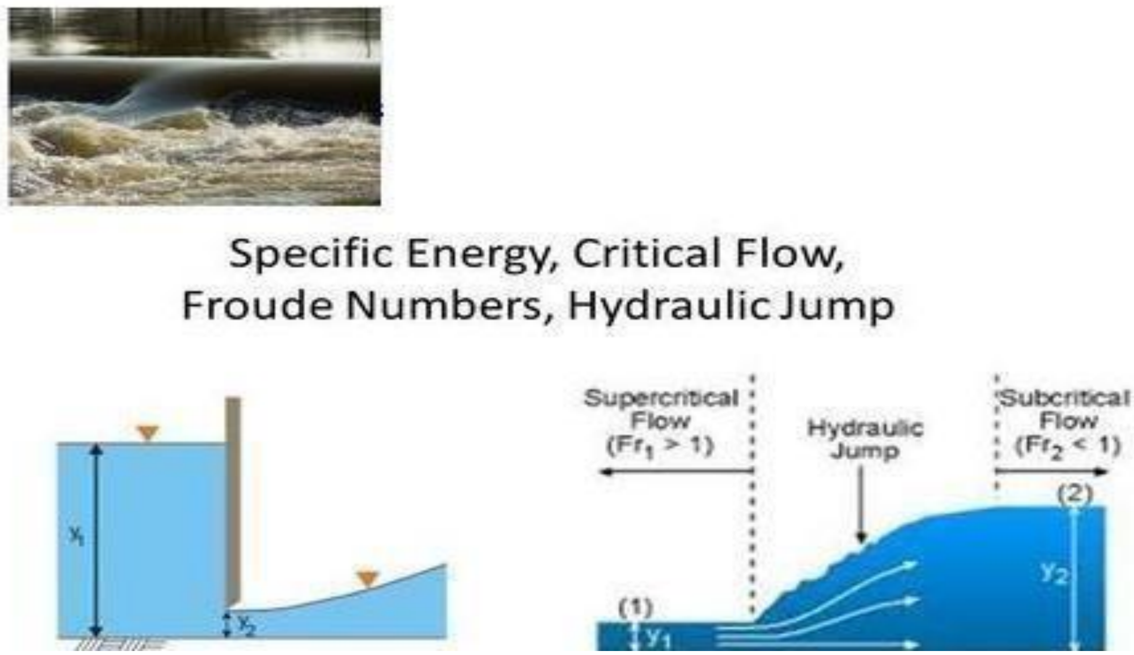
- mass is conserved;
- the momentum principle is satisfied;
- Mechanical energy is lost (mostly as heat).

Assume, for simplicity:

- velocity uniform over upstream and downstream cross-sections;
- small slope (so that the down slope component of weight can be neglected);
- the length of the jump is short (so that bed friction can be neglected);
- Wide or rectangular cross-section (but see the Examples for an alternative).

Energy Principles in open channel flow





Alternate depth

In the open-channel flow of rectangular channels, the alternate depth equation relates the upstream (y1) and downstream (y2) steady-state flow depths of a flow that encounters a control device, such as a sluice gate, which conserves energy for a given discharge.

Conjugate Depth

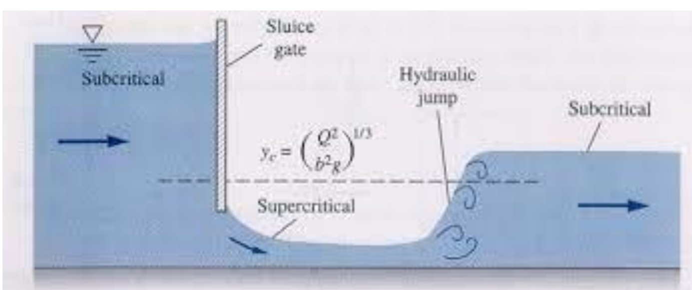
In fluid dynamics, the conjugate depths refer to the depth (y1) upstream and the depth (y2) downstream of the hydraulic jump whose momentum fluxes are equal for a given discharge (volume flux) q. The depth upstream of a hydraulic jump is always supercritical.

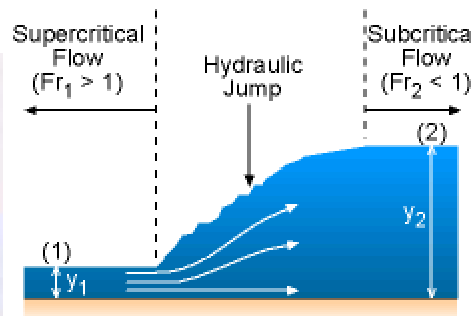
Conjugate depths can also be calculated using the Froude number and depth of either the supercritical or subcritical flow. The following equations can be used to determine the conjugate depth to a known depth in a rectangular channel:

Descriptive Hydraulic Jump Characteristics

Characteristic	Before the jump	After the jump
fluid speed	supercritical (faster than the wave speed) also known as shooting or supernal	subcritical also known as tranquil or subundal
fluid height	low	high
flow	typically smooth turbulent	typically turbulent flow (rough andchoppy)

If water depth or velocity change abruptly over a short distance and the pressure distribution is not hydrostatic, the water surface profile is characterized as Rapidly Varying Flow (RVF). The occurrence of RVF is usually a local phenomenon. RVF can often be observed near the inlet and outlet of culverts, and wherever hydraulic jumps occur.





Hydraulic Jump Characteristics

Amount upstream flow is supercritical (i.e., prejump Froude Number)	Ratio of height after to height before jump	Fraction of energy dissipated by jump
≤ 1.0	1.0	none
1.0–1.7	1.0–2.0	< 5%
1.7–2.5	2.0–3.1	5% – 15%
2.5–4.5	3.1–5.9	15% – 45%

What are the effects of hydraulic jump?

The use of hydraulic jump in hydraulic engineering is not uncommon and the creation of such jumps has several purposes:

(i) Its main aim is to perform as an energy-dissipating device to reduce the excess energy of

water flows.
(i) The jump generates significant disturbances in the form of eddies and reverse flow rollers to facilitate mixing of chemicals.

(i) During the jump formation, considerable amount of air is entrained so that it helps in the aeration of streams which is polluted by bio-degradable wastes.
(iv) It enables efficient operation of flow measuring device like flumes.

Uses of Hydraulic Jump:
The hydraulic jump is necessarily formed to reduce the energy of water while the discharge downfalls a spillway. It becomes necessary to reduce its energy and maintain stable velocities, that phenomenon is called energy dissipation in hydraulic structures.

Subcritical flow is deep, slow **flow** with a low energy state and has a Froude number less than one (F<1). Critical **flow** occurs when the Froude number equals one (F=1); there is a perfect balance between the gravitational and inertial forces.

Basic Characteristics of Hydraulic Jump:
1. The jump is unsteady, irregular2. Based on wind directions and heavy wind blow, it changes its property and can be choppy and undular sometimes.

Types of Hydraulic Jumps – Based on Froude's Number:
Basically a hydraulic jump occurs in many types depending on topographical features and bed surface roughness and many other natural interface relations. This hydraulic jump types can be probably expressed based on Froude's number:

1. Undular Hydraulic Jump – Froude Number (1 to 3):
Undular Jump is irregular, not properly formed and there are certain turbulences in water particles.

2. Weak Jump – Froude Number (3 to 6)
Weak jump takes place when the velocity in water is very less and the water particles cannot be stable and flows in various ways.

3. Oscillating Hydraulic Jump – Froude Number (6-20)

Oscillating jump forms when an oscillating jet enter into super critical state and there the numberof particles starts oscillating in clockwise or either anticlockwise direction, forming slighter tidesor waves to the top surface. Also the flow is dependent on heavy blow of air in one direction.

4. Steady Hydraulic Jump – Froude Number (20 to 80)

The rapidly-varied transient phenomenon is an open channel, commonly known under the general term surge, occurs wherever there is a sudden change in the discharge or depth or both. Such situations occur,for example, during the sudden closure of a gate.

Positive and Negative Surge

A surge producing an increase in depth is called positive surge and the one which causes a decrease indepth is known as negative surge.

Types of Positive Surge:

Type A- This may occur when a gate provided at the head of a channel is suddenly opend.It is having anadvancing wave front moving downstream.

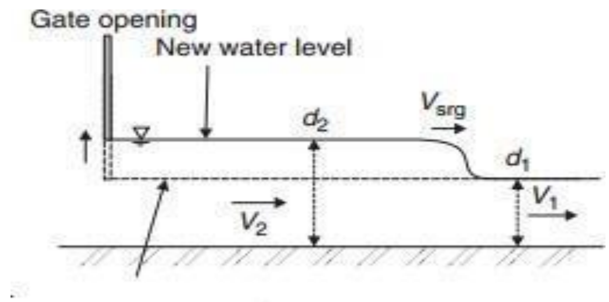
Type – B- This may occur when a gate provided at the tail end of a channel is suddenly closed.It ishaving an advancing wave front moving upstream.

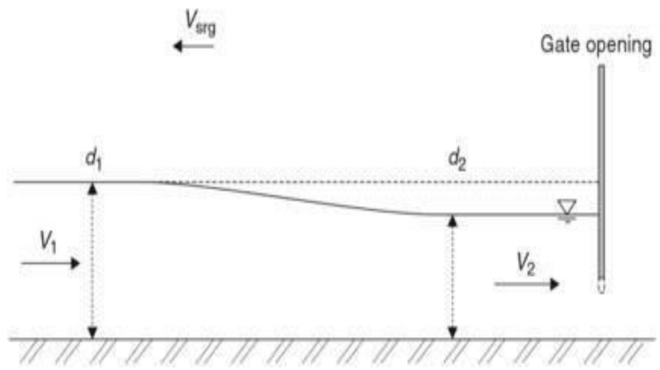
Types of Negative Surge:

Type- C- This may occur when a gate provided at the head of a channel is

suddenly closed. Type – B- This may occur when a gate provided at the tail

end of a channel is suddenly opened.

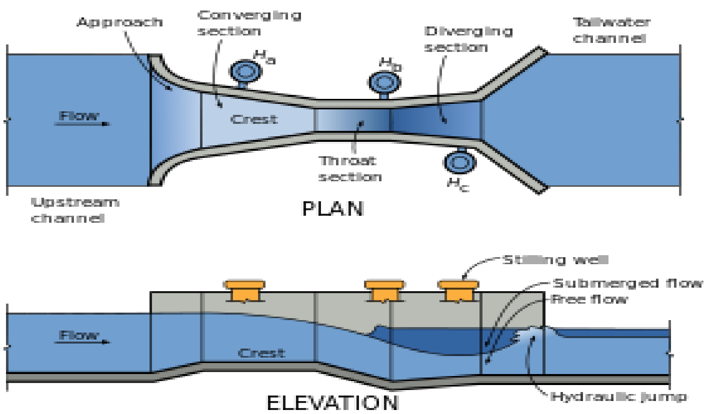




Flumes & Channel Transitions

I. General Characteristics of Flumes

- Flumes are often used: Along contours of steep slopes where minimal excavation isdesired
- On flat terrain where it is desired to minimize pumping, except perhaps at the source
- On flat terrain where it is desired to avoid pumping, except maybe at the water source
- Where cross-drainage is required over a depression



- The average flow velocity in a flume is higher than that for most canals
- But the flow regime in flumes is usually subcritical, as opposed to chutes, which usuallyoperate under supercritical flow conditions
- Flume cross-section shapes are typically rectangular, but may also be semi-circular orparabolic
- Several irrigation systems in Morocco have networks of elevated semi-circular flumes
- Flumes with non-rectangular sections are usually pre-cast concrete, or concrete mixedwith other materials
- Flumes may have under-drains, side inlet structures, and over-pass structures to handlecross flows, especially for cross flows going down a slope

Function of Channel Transitions

- Channel transitions occur at locations of cross-sectional change, usually over a shortdistance
- Transitions are also used at entrances and exits of pipes such as culverts and invertedsiphons

Below are some of the principal reasons for using transitions:

1. Provide a smooth change in channel cross section
2. Provide a smooth (possibly linear) change in water surface elevation
3. Gradually accelerate flow at pipe inlets, and gradually decelerate flow at pipe outlets
4. Avoid unnecessary head loss through the change in cross section
5. Prevent occurrence of cross-waves, standing waves, and surface turbulence in general
6. Protect the upstream and downstream channels by reducing soil erosion
7. To cause head loss for erosion protection downstream; in this case, it is an energydissipation & transition structure

Energy Dissipation

Outlet protection for culverts, storm drains, BMP outlets, and steep open channels is essential topreventing erosion from damaging downstream channels and drainage structures. Erosion problems at culverts or at the outlet from detention basins are a commonoccurrence. Determination of the flow conditions, scour potential, and channel erosion resistance shall be standard procedure for all designs.

- Outlet protection can be a channel lining, structure, or flow barrier designed to lowerexcessive flow velocities and prevent erosion and scour.
- Outlet protection shall be employed whenever the velocity of flow at a pipe or open
- Channel outlet exceeds the erosive velocity of the immediate downstream reach

Energy dissipation n may take the form of the following:

- Erosion control stone-lined channel
- Riprap outlet basins; or
- Concrete baffled outlets.


I. Locations of Excess Energy

What are the locations of excess energy in open channels?

- Channel constrictions (such as gates, weirs, others)
- Steep longitudinal bed slopes
- Drops in elevation
- Energy dissipation is almost always needed downstream of supercritical flow sections
- Energy dissipation may also be desired in lined channelsEnergy dissipation

structures are typically located at:

1. Sudden drops in bed elevation
2. Downstream ends of channel branches, flumes and chutes (especiallywheredischarging into earthen sections)

3. Outlets of culverts and inverted siphons
4. Structures causing supercritical flow (e.g. underflow gates)
5. Structures causing downstream turbulence and eddies

II. Energy Dissipation Structure Types

Most energy dissipation structures in open channels are based on:
1. The creation of a stable hydraulic jump
2. Head-on impaction a solid, immovable obstruction

Hydraulic Jumps for Energy Dissipation
- In open channels, a transition from subcritical to supercritical flow regimes results invery little localized hydraulic energy loss
- But, the opposite transition, from supercritical to subcritical, involves a hydraulic jumpand energy loss
  - The energy loss through a hydraulic jump can be significant, so jumps can be applied toenergy dissipation applications in open channels

Problem :

Water flows at a depth of 10 cm with a velocity of 6 m/s in a rectangular channel. Is the flow subcritical or supercritical? What is the alternate depth?

Check Froude number

$$\mathrm{Fr} = \frac{V}{\sqrt{gy}} = \frac{6\ \mathrm{m/s}}{\sqrt{9.81\ /\mathrm{s}^2 \cdot 0.1\ \mathrm{m}}} = 6.06 > 1$$

so the flow is supercritical.

$$E = y + \frac{V^2}{2g} = 0.1\ \mathrm{m} + \frac{(6\ \mathrm{m/s})^2}{2 \cdot 9.81\ \mathrm{m/s}^2} = 1.935\ \mathrm{m}$$

Solving for the alternate depth for an $E = 1.935$ m yields $y_{\mathrm{alt}} = 1.93$ m

Problem :

Water flows with a velocity of 2 m/s and at a depth of 3 m in a rectangular channel. What is the change in depth and in water surface elevation produced by a gradual upward change in bottom elevation (upstep) of 60 cm? What would be the depth and elevation changes if there were a gradual downstep of 15 cm? What is the maximum size of upstep that could exist before upstream depth changes would result? Neglect head losses.

$$E_1 = y_1 + \frac{V_1^2}{2g} = 3\text{ m} + \frac{(2\text{ m/s})^2}{2 \cdot 9.81\text{ m/s}^2} = 3.20\text{ m.}$$

$$E_2 = E_1 - \Delta z = 3.20\text{ m} - 0.60\text{ m} = 2.60\text{ m.}$$

Also

$$E_2 = y_2 + \frac{q^2}{2gy_2^2} = y_2 + \frac{(6\text{ m}^3\text{/s/m})^2}{2 \cdot 9.81\text{ m/s}^2 \cdot y_2^2} = 2.60\text{ m}$$

so $y_2 = 2.24$ m. $\Delta y = y_2 - y_1 = -0.76$ m so water surface drops 0.16 m.
For a downward step of 15 cm we have

$$E_2 = E_1 - \Delta z = 3.20\text{ m} - (-0.15\text{ m}) = 3.35\text{ m.}$$

giving $y_2 = 3.17$ m and $\Delta y = y_2 - y_1 = 0.17$ m so water surface rises 0.02 m.
The maximum upstep possible before affecting upstream water surface levels is for $y_2 = y_c$

$$y_c = \sqrt[3]{\frac{q^2}{g}} = \sqrt[3]{\frac{(6\text{ m}^3\text{/s/m})^2}{9.81\text{ m/s}^2}} = 1.54\text{ m.}$$

Problem : The spillway shown has a discharge of 1.2 m3 /s per meter of width occurring over it. What depth y2 will exist downstream of the hydraulic jump? Assume negligible energy loss over the spillway

$$y_0 + \frac{q^2}{2gy_0^2} = y_1 + \frac{q^2}{2gy_1^2}$$

$$5\text{ m} + \frac{(1.2\text{ m}^3\text{/s/m})^2}{2 \cdot 9.81\text{ m/s}^2 \cdot (5\text{ m})^2} = y_1 + \frac{(1.2\text{ m}^3\text{/s/m})^2}{2 \cdot 9.81\text{ m/s}^2 \cdot y_1^2}$$

solving for y_1 we get $y_1 = 0.123$ m.

$$\mathrm{Fr}_1 = \frac{q}{\sqrt{g y_1^3}} = \frac{(1.2\ \mathrm{m^3/s/m})^2}{\sqrt{9.81\ \mathrm{m/s^2} \cdot (0.123\ \mathrm{m})^3}} = 8.88.$$

$$y_2 = \frac{y_1}{2}\left(\sqrt{1 + 8\mathrm{Fr}_1^2} - 1\right) = \frac{0.123\ \mathrm{m}}{2}\left(\sqrt{1 + 8 \cdot 8.88^2} - 1\right) = 1.48\ \mathrm{m}.$$

Problem:
Water flows at a depth of 10 cm with a velocity of 6 m/s in a rectangular channel. Is the flow subcritical or supercritical? What is the alternate depth

Check Froude number

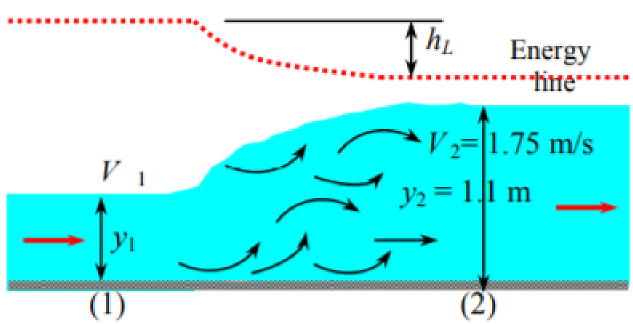
$$\mathrm{Fr} = \frac{V}{\sqrt{gy}} = \frac{6\ \mathrm{m/s}}{\sqrt{9.81\ /\mathrm{s^2} \cdot 0.1\ \mathrm{m}}} = 6.06 > 1$$

so the flow is supercritical.

$$E = y + \frac{V^2}{2g} = 0.1\ \mathrm{m} + \frac{(6\ \mathrm{m/s})^2}{2 \cdot 9.81\ \mathrm{m/s^2}} = 1.935\ \mathrm{m}$$

Solving for the alternate depth for an $E = 1.935$ m yields $y_{alt} = 1.93$ m

Problem : Determine the flow depth and velocity before the jump as well as the fraction of mechanical energy dissipated. The flow depth and average velocity of water after a hydraulic jump are measured and given in fig.



1 The flow is steady or quasi-steady. 2 The channel is sufficiently wide so that the end effects are negligible. 3 The channel is horizontal.

The Froude number after the hydraulic jump is

$$\mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{1.75\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(1.1\ \mathrm{m})}} = 0.5327$$

It can be shown that the subscripts in the relation $y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right)$ are interchangeable. Thus,

$$y_1 = 0.5y_2\left(-1+\sqrt{1+8\mathrm{Fr}_2^2}\right) = 0.5(1.1\ \mathrm{m})\left(-1+\sqrt{1+8\times 0.5327^2}\right) = 0.4446\ \mathrm{m}$$

$$V_1 = \frac{y_2}{y_1}V_2 = \frac{1.1\ \mathrm{m}}{0.4446\ \mathrm{m}}(1.75\ \mathrm{m/s}) = 4.329\ \mathrm{m/s}$$

The Froude number before the jump is

$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{4.329\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(0.4446\ \mathrm{m})}} = 2.073$$

which is greater than 1. Therefore, the flow is indeed supercritical before the jump. The head loss is determined from the energy equation to be

$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (0.4446\ \mathrm{m}) - (1.1\ \mathrm{m}) + \frac{(4.329\ \mathrm{m/s})^2 - (1.75\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 0.1437\ \mathrm{m}$$

The specific energy of water before the jump and the dissipation ratio is

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (0.4446\ \mathrm{m}) + \frac{(4.329\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 1.400\ \mathrm{m}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s1}} = \frac{0.1437\ \mathrm{m}}{1.400\ \mathrm{m}} = \mathbf{0.103}$$

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

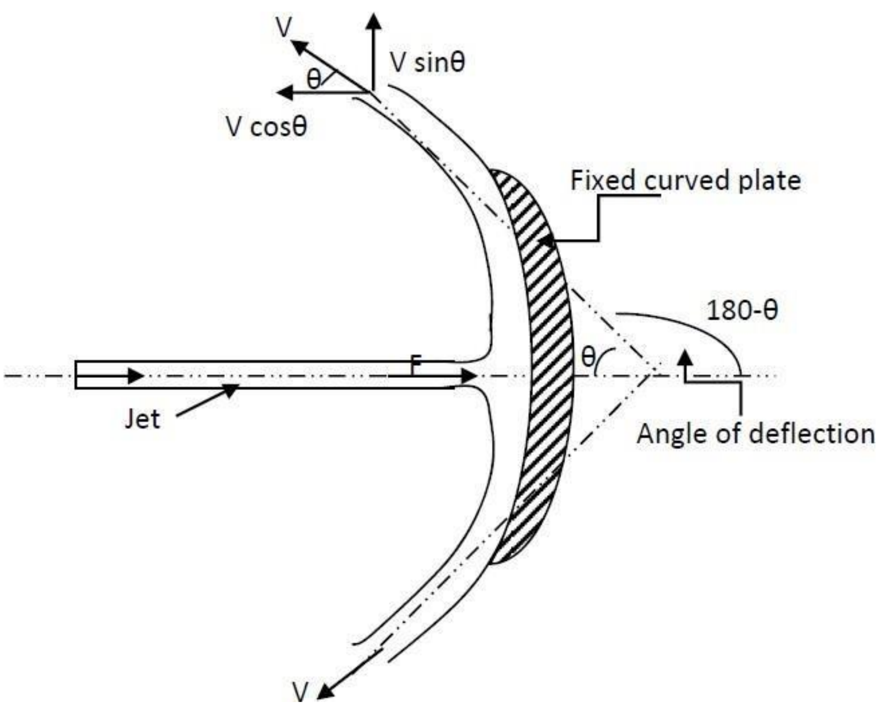
UNIT – IV – APPLIED HYDRAULIC ENGINEERING – SCIA1402

UNIT IV

HYDRAULIC MACHINES-TURBINES

Impact of jets on curved plates

The impact of jet on the curved surface is of much practical significance. The jet of water can be introduced tangentially, whereas in other case it can strikes at some angle to the entrance portion of the surface. Here, both such cases deserve individual consideration.
Consider the case when the jet strikes horizontally at the centre of the vane on the concave side as shown in figure below.



After it strikes, it gets divided, glides over the surface and leaves the vane tangentially with same velocity 'v'.

The velocity at outlet of the plate can be resolved into two Components, one in the direction of the jet and the other perpendicular to the direction of the jet.

Component of velocity in the direction of the jet is given by, - $V \cos\theta$ here -ive sign has been considered because; outlet velocity is in the opposite direction of the jet of water coming out of from nozzle.

Component of velocity perpendicular to the jet is given by, $\boldsymbol{V}$ **sinθ**
Force exerted by the jet in the direction of jet is given

$$Fx= \rho av\{ v- v\cos\theta\}$$

$$= \rho av^2 \{1+ \cos\theta\}$$

Similarly, the force exerted by the jet in the direction normal to jet direction is given by,

$$Fy= \rho av\{ 0- v\sin\theta\}$$

$$=[-\rho av^2 \text{sins}\ \theta]$$

If θ is made 90^0 (cosθ=0) then the jet gets deflected through 90^0 as in the case of a flat plate.
When θ= 0 (cosθ=1) then the vane becomes semicircular and the jets coming off the outlet tips are parallel and reverse in direction to the striking jet.

$$F= 2\rho av^2 \quad = 2*\text{force on flat plate}$$

Thus, it is observed that, as the angle of deflection increases, the value of F increases.

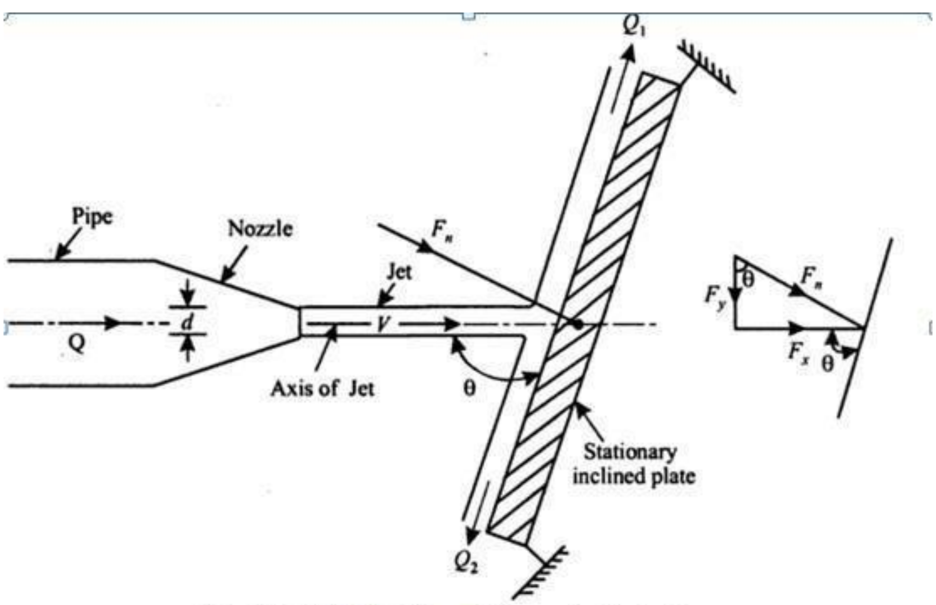


Fig. 17.2. Fluid jet striking a stationary inclined plate.

Turbines

Definition: A **turbine** is a turbo machine with at least one moving part called a rotor assembly, which is a shaft or drum with blades attached. Moving fluid acts on the blades so that they move and impart rotational energy to the rotor.

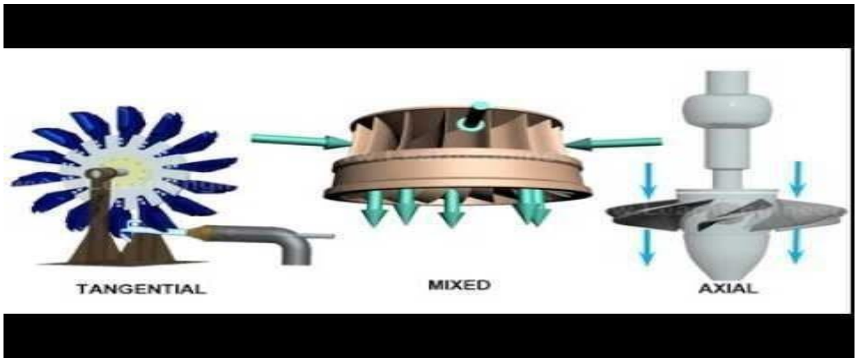
Classification of Hydraulic Turbines: Based on flow path

Water can pass through the Hydraulic Turbines in different flow paths. Based on the flow path of the liquid Hydraulic Turbines can be categorized into three types.

1. **Axial Flow Hydraulic Turbines:** This category of Hydraulic Turbines has the flow path of the liquid mainly parallel to the axis of rotation. Kaplan Turbines has liquid flow mainly in axial direction.
2. **Radial Flow Hydraulic Turbines:** Such Hydraulic Turbines has the liquid flowing mainlyin a plane perpendicular to the axis of rotation.
3. **Mixed Flow Hydraulic Turbines:** For most of the Hydraulic Turbines used there is a significant component of both axial and radial flows. Such types of Hydraulic Turbines are called as Mixed Flow Turbines. Francis Turbine is an example of mixed flow type, in Francis Turbine water enters in radial direction and exits in axial direction.

None of the Hydraulic Turbines are purely axial flow or purely radial flow. There is always a component of radial flow in axial flow turbines and of axial flow in radial flow turbines.

4. **Tangential flow turbin**e: if the water flows along the tangent of the runner it is known as tangential flow turbine.



Classification of Hydraulic turbines:

1) Based on type of energy at inlet to the turbine:

- **Impulse Turbine**: The energy is in the form of kinetic form. e.g: Pelton wheel, Turbo wheel.
- **Reaction Turbine**: The energy is in both Kinetic and Pressure form. e.g: Tubular, Bulb, Propeller, Francis turbine.

2) Based on direction of flow of water through the runner:

- **Tangential flow:** water flows in a direction tangential to path of rotational, i.e. Perpendicular to both axial and radial directions.
- **Radial outward flow** e.g : Forneyron turbine.
- **Axial flow:** Water flows parallel to the axis of the turbine. e.g: Girard, Jonval, Kalpan turbine.
- **Mixed flow:** Water enters radially at outer periphery and leaves axially. e.g : Modern Francis turbine.

3) Based on the head under which turbine works:

- High head, impulse turbine. e.g : Pelton turbine.
- Medium head, reaction turbine. e.g : Francis turbine.
- Low head, reaction turbine. e.g : Kaplan turbine, propeller turbine.

4) Based on the specific speed of the turbine:

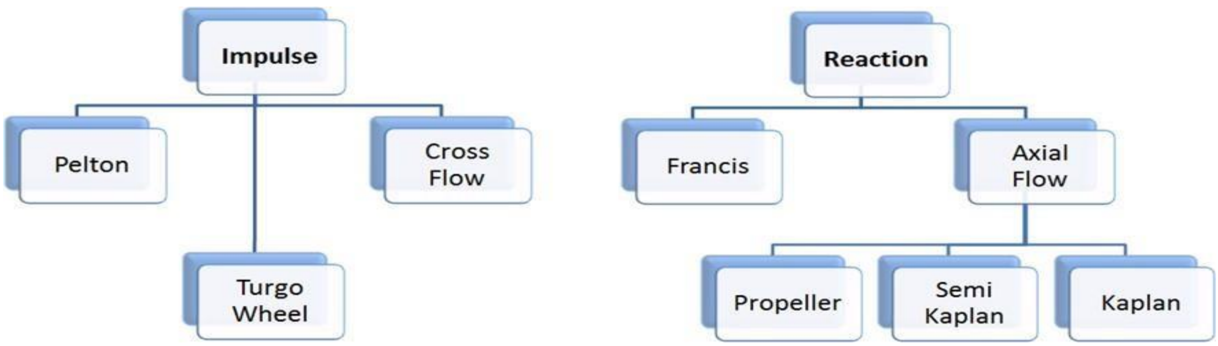
- Low specific speed, impulse turbine. e.g: Pelton wheel.
- Medium specific speed, reaction turbine. e.g : Francis wheel.
- High specific speed, reaction turbine. e.g : Kaplan and Propeller turbine.

5) Based on the name of the originator:

- Impulse turbine – Pelton wheel, Girard, Banki turbine.
- Reaction turbine – Forneyron, Jonval, Francis, Dubs, Deriaze, Thomson kalpan, Barker, Moody, Nagler, Bell.

Classification of Hydraulic Turbines: Based on pressure change (or type of energy)

One more important criterion for classification of Hydraulic Turbines is whether the pressure of liquid changes or not while it flows through the rotor of the Hydraulic Turbines. Based on the pressure change Hydraulic Turbines can be classified as of two types.



1. **Impulse Turbine:** The pressure of liquid does not change while flowing through the rotor of the machine. In Impulse Turbines pressure change occur only in the nozzles of the machine. One such example of impulse turbine is Pelton wheel.

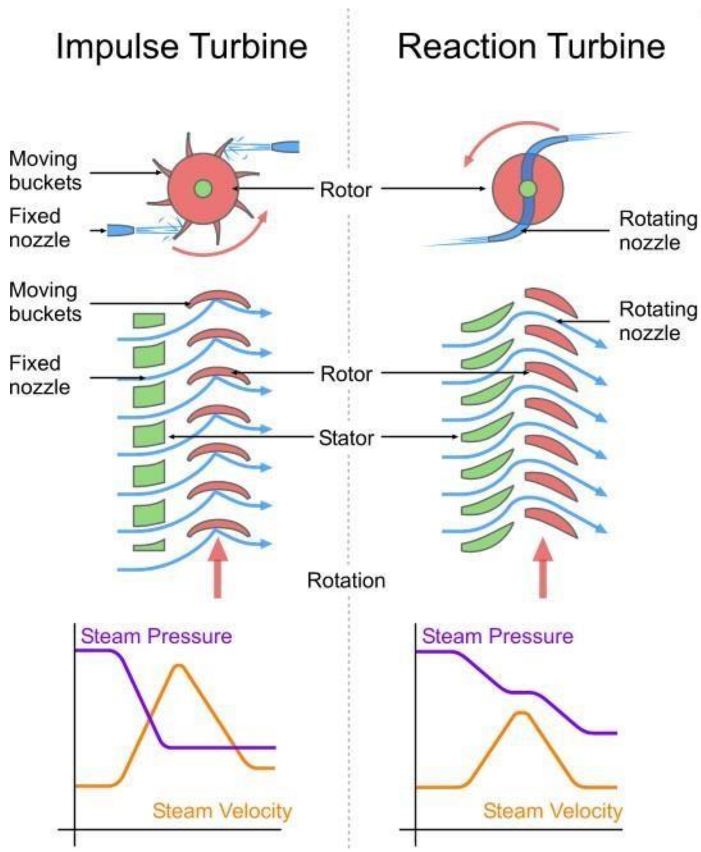
2. **Reaction Turbine:** The pressure of liquid changes while it flows through the rotor of the machine. The change in fluid velocity and reduction in its pressure causes a reaction on the turbine blades; this is where from the name Reaction Turbine may have been derived. Francis and Kaplan Turbines fall in the category of Reaction Turbines.

<u>Differences between Impulse and Reaction Turbines:</u>

S.No	Impulse Turbine	Reaction Turbine
1	In Impulse Turbine all hydraulic energy is converted into kinetic energy by a nozzle and it is is the jet so produced which strikes the runner blades.	In Reaction Turbine only some amount of the available energy is converted into kinetic energy before the fluid enters the runner.

2	The velocity of jet which changes, the pressure throughout remaining atmosphere.	Both pressure and velocity changes as fluid passes through a runner. Pressure at inlet is much higher than at outlet.
3	Water-tight casing is not necessary. Casing has no hydraulic function to perform. It only serves to prevent splashing and guide water to the tail race.	The runner must be enclosed within a watertight casing.
4	Water is admitted only in the form of jets. There may be one or more jets striking equal number of buckets simultaneously.	Water is admitted over the entire circumference of the runner.
5	The turbine doesn’t run full and air has a free access to the bucket.	Water completely fills at the passages between the blades and while flowing between inlet and outlet sections does work on the blades.
6	The turbine is always installed above the tail race and there is no draft tube used.	Reaction turbine are generally connected to the tail race through a draft tube which is a gradually expanding passage. It may be installed below or above the tail race.

7	Flow regulation is done by means of a needle valve fitted into the nozzle.	The flow regulation in reaction turbine is carried out by means of a guide-vane assembly. Other component parts are scroll casing, stay ring runner and the draft tub.
8	Example of Impulse turbine is Pelton wheel.	Examples of Reaction Turbine are Francis turbine, Kaplan and
		Propeller Turbine, Deriaz Turbine, Tubuler Turbine, etc.
9	Impulse Turbine have more hydraulic efficiency.	Reaction Turbine have relatively less efficiency.
10	Impulse Turbine operates at high water heads.	Reaction turbine operate at low and medium heads.
11	Water flow is tangential direction to the turbine wheel.	Water flows in radial and axial direction to turbine wheel.
12	Needs low discharge of water.	Needs medium and high discharge of water.
13	Degree of reaction is zero.	Degree of reaction is more than zero and less than or equal to one.
14	Impulse turbine involves less maintenance work.	Reaction turbine involves more maintenance work.



Classification according to the head at the inlet of turbine.

1. High head
2. Medium head
3. Low head.

Classification according to specific speed

1. Low
2. Medium
3. High

Efficiencies of turbine

1. **Hydraulicefficiency (η_h)** = Power delivered to the runner / Power supplied at inlet

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

= R.P/ W.P

2. **Mechanical efficiency**((η_m) = power at the shaft 0f the turbine/power delivered by the water to the runner

= S.P/R.P.

3. **Volumetric efficiency** (η_v) = volume of the water actually striking the runner/ Volume of water supplied to the turbine

4. **Overall efficiency** ((η_h) = shaft power/water power.

= $\eta_m \times \eta_h$

Pelton wheel turbine

The Pelton wheel is an impulse type water turbine. It was invented by Lester Allan Pelton in the 1870s. The Pelton wheel extracts energy from the impulse of moving water, as opposed to water's dead weight like the traditional overshot water wheel.

The Basic Working Principle

Working principle of Pelton turbine is simple. When a high speed water jet injected through a nozzle hits buckets of Pelton wheel; it induces an impulsive force. This force makes the turbine rotate. The rotating shaft runs a generator and produces electricity.

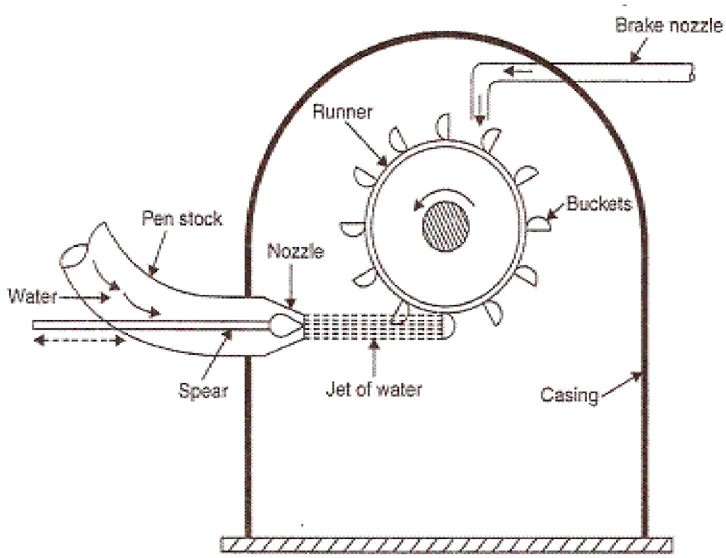
Pelton – An Impulse Turbine

Since the water jet is always open to atmosphere, inlet and exit pressure of water jet will be same and will be same as atmospheric pressure. However absolute velocity of fluid will have huge drop from inlet to exit of bucket. This kinetic energy drop is the maximum energy the bucket canabsorb.

So it is clear that Pelton turbine gains mechanical energy purely due to change in kinetic energy of jet, not due to pressure energy change. Which means Pelton turbine is a pure impulse machine.

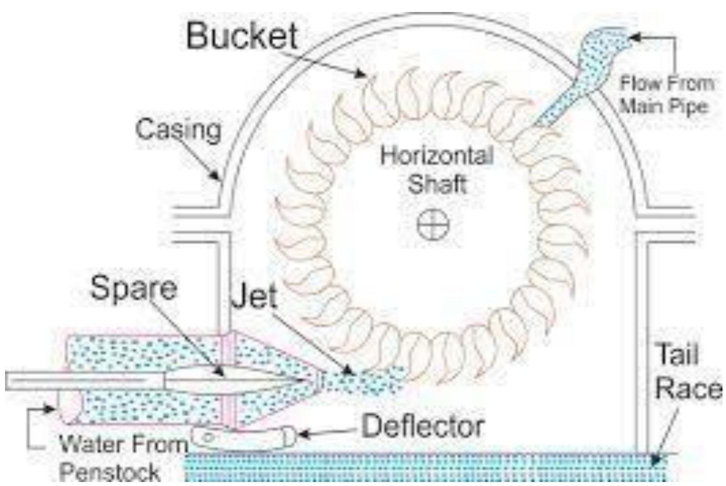
Impulse force produced by water jet is high when jet is having high velocity. Water stored at high altitude can easily produce high jet velocity. This is the reason why Pelton turbine is most suitable for operation, when water is stored at high altitude.

You can easily understand why there is a nozzle fitted at water jet injection portion. Nozzle will increase velocity of jet further, thus will aid in effective production of impulse force.



The only hydraulic turbine of the impulse type in common use, is named after an American engineer Laster A Pelton, who contributed much to its development around the year 1880. Therefore this machine is known as Pelton turbine or Pelton wheel. It is an efficient machine particularly suited to high heads. The rotor consists of a large circular disc or wheel on which a number (seldom less than 15) of spoon shaped buckets are spaced uniformly round is periphery

as shown in Figure .The wheel is driven by jets of water being discharged at atmospheric pressure from pressure nozzles. The nozzles are mounted so that each directs a jet along a tangent to the circle through the centers of the buckets. Down the center of each bucket, there is asplitter ridge which divides the jet into two equal streams which flow round the smooth inner surface of the bucket and leaves the bucket with a relative velocity almost opposite in direction to the original jet.



For maximum change in momentum of the fluid and hence for the maximum driving force on the wheel, the deflection of the water jet should be 180^0. In practice, however, the deflection is limited to about 165^0 so that the water leaving a bucket may not hit the back of the following bucket. Therefore, the camber angle of the buckets is made as $165^{0.}$

The number of jets is not more than two for horizontal shaft turbines and is limited to six for vertical shaft turbines. The flow partly fills the buckets and the fluid remains in contact with the atmosphere. Therefore, once the jet is produced by the nozzle, the static pressure of the fluid remains atmospheric throughout the machine. Because of the symmetry of the buckets, the side thrusts produced by the fluid in each half should balance each other.

Velocity triangles

What does velocity triangle indicates?

In turbo machinery, a **velocity triangle** or a **velocity** diagram is a **triangle** representing the various components of **velocities** of the working fluid in a turbo machine. **Velocity triangles** may be drawn for both the inlet and outlet sections of any turbo machine.

Terms used in velocity triangles.

1. $V1 = Jet\ Velocity\ at\ inlet$
2. $Vf1 = flow\ velocity$
3. $V2 = Jet\ Velocity\ at\ out\ let$
4. $vw1 = whirl\ velocity\ at\ inlet$
5. $vw2 = whirl\ velocity\ outlet,$

6. $Vr1 = relative\ velocity\ at\ inlet$

7. $vr2 \quad = \quad relative\ velocityat\ outlet.$

8. $Vf2 = velocity\ of\ flow\ at\ outlet$

9. $\alpha = guide\ bladeangle\ inlet$

10. $\theta = vane\ angle\ at\ inlet$

11. $\emptyset = vaneangle\ outlet$

12. $\beta = guideblade\ angle\ at\ outlet$

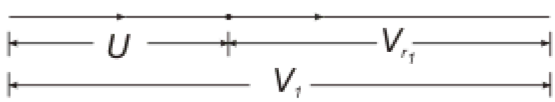
13. $u = vane\ velocity$

14. $u1 = vane\ velocity\ at\ inlet = \pi DN/60$

15. $u2 = vane\ velocity\ outlet = \pi DN/60$

16. $u = velocity\ of\ vane$

Inlet triangle: for pelton wheel.



(b)

The diagram of velocity vector at inlet becomes simply a straight line and the relative velocity isgiven by $vr1 = V1 - u1$

$$= V1 - u$$

Because u=u1= u2= πDN / 60

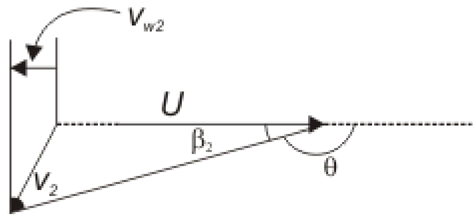
Also $w1 \ = V1$

$\alpha=0^0$ $\theta= 0^0$

$$V1 = velocity\ of\ jet\ at\ inlet = \sqrt{2}gH$$

$$Vw1 = whirl\ velocity\ at\ inlet$$

Velocity triangle at outlet



(c)

$$Vr2 = Vr1$$

$$Vw2 = Vr2\ cos\emptyset - u2$$

Work done by the jet on the runner per second=$Fx * u = \rho aV1(Vw1 + Vw2) * U\ Nm/s$

Power given to the runner by the jet

$$\frac{\rho aV1(Vw1 + Vw2) * U}{1000}\ kW$$

Work done /s per unit weight of water striking/s

$$= \rho aV1(Vw1 + Vw2) * U \Big/ weight\ of\ water\ striking$$

Problem:

At a project site, the head available is 160 m of water at a flow rate of 0.005 m3 /s. Select and design a suitable turbine to generate power, assuming the required coefficients with justification and stating all the relevant parameters.

Solution: A suitable turbine is to be designed for a site where the available head is 160 m and flow rate is 0.005 m^3/s.

1. This is a case of design of Pelton turbine.
2. The loss in the pipeline is taken as 10%. This makes the net head available at the nozzle as 160 – 16 = 144 m.
3. The overall efficiency of energy conversion is assumed as 95%.
4. The power, that can be developed, is therefore

$$P = \frac{\omega Q H \eta}{1000} = \frac{9810 \times 0.005 \times 144 \times 0.95}{1000}$$
$$= 6.71\,\text{kW}$$

For the hydropower development, this power is too small. However, the procedure of the design can be continued.

5. The jet velocity is calculated as

$$V_1 = c_v \sqrt{2gH}, \text{where } c_v = 0.98$$
$$= 0.98\sqrt{2 \times 9.81 \times 144}$$
$$= 52.09\,\text{m/s}$$

6. To calculate the jet diameter, d, we have

$$\frac{\pi d^2}{4} V_1 = Q$$

Therefore,

$$d = \left[\frac{Q \times 4}{\pi \times V_1}\right]^{0.5} = \left[\frac{0.005 \times 4}{\pi \times 52.09}\right]^{0.5}$$
$$= 0.01097\,\text{m}$$
$$= 1.1\,\text{cm}$$

7. The speed ratio is taken as 0.46.
8. The peripheral velocity of blades (Pelton buckets) is

$$U = 0.46 V_1 = 0.46 \times 52.09$$
$$= 23.96\,\text{m/s or } 24\,\text{m/s}$$

9. Now, the possible speeds of the rotor are synchronous speeds, that is, 3000, 1500, 1000, 750, 600, etc. We can choose each of the speed, to find the diameter, so that the peripheral velocity is U.

$$D = \frac{U \times 60}{\pi N}, U = 24 \text{ m/s}$$

N	3000	1500	1000	750	600
D	0.1528	0.3056	0.4584	0.611	0.764

10. At this juncture, the procedure is with respect to two cases of the speed, N = 3000 rpm and N = 1500 rpm.

11. (a) When N = 3000 rpm, the specific speed is

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{3000 \times \sqrt{6.71}}{144^{1.25}} = 15.6$$

(b) When N = 1500 rpm, the specific speed is

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{1500 \times \sqrt{6.71}}{144^{1.25}} = 7.789$$

It is possible to have both these values as alternate designs.

12. The jet diameter continues as 1.1 cm in either case.

The jet ratios are

(a) $\frac{D}{d} = \frac{15.28}{1.1} = 14.18$

(b) $\frac{D}{d} = \frac{30.56}{1.1} = 27.78$

13. The number of Pelton buckets are

(a) $Z = 7 + 15 = 22$

(b) $Z = 14 + 15 = 29$

14. Because jet diameter is 1.1 cm in both cases, the cup dimensions are same in the two cases.

Length	$L = 2.3\ d = 2.53$ cm
Breadth	$B = 2.8\ d = 3.08$ cm
Depth	$D = 0.6\ d = 0.66$ cm

15. The blade angles are $\beta_1 = 0°$ and $\beta_2 = 165°$.

This completes the designs.

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

SCHOOL OF BUILDING AND ENVIRONMENT

DEPARTMENT OF CIVIL ENGINEERING

UNIT – V – APPLIED HYDRAULIC ENGINEERING – SCIA1402

PUMPS

Pumps are used to transfer and distribute liquids in various industries. Pumps convert mechanical energy into hydraulic energy. Electrical energy is generally used to operate the various types of pumps.

Pumps have two main purposes.

Transfer of liquid from one place to another place (e.g. water from an underground into a water storage tank).

Circulate liquid around a system (e.g. cooling water or lubricants through machines and equipment).

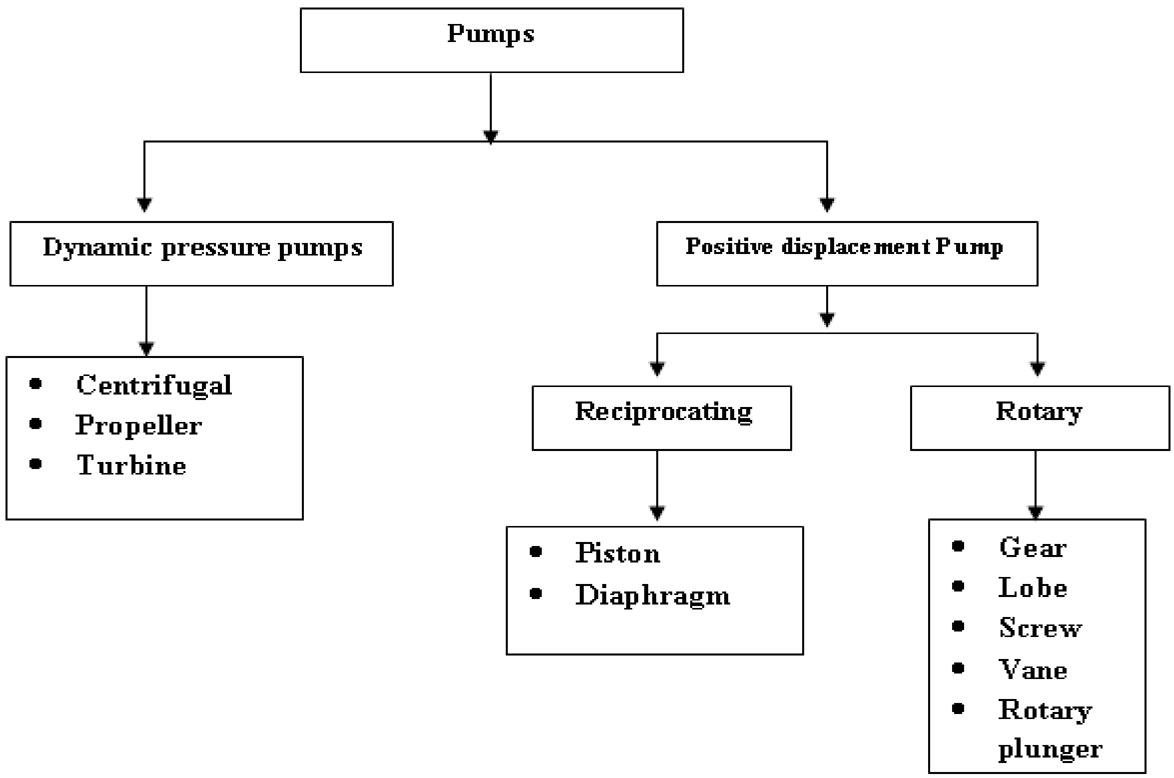
Components of a Pumping System

- Pump casing and impellers
- Prime movers: electric motors, diesel engines or air system
- Piping used to carry the fluid
- Valves, used to control the flow in the system
- other fittings, controls and instrumentation
- End-use equipment, which have different requirements (e.g. pressure, flow) and therefore determine the pumping system components and configuration. Examples include heat exchangers, tanks and hydraulic machines.

Classification

There exist a wide variety of pumps that are designed for various specific applications. However, most of them can be broadly classified into two categories as mentioned below.

i. positive displacement
ii. Dynamic pressure pumps



Positive Displacement Pumps

The term positive displacement pump is quite descriptive, because such pumps are designed to displace a more or less fixed volume of fluid during each cycle of operation. The volumetric flow rate is determined by the displacement per cycle of the moving member (either rotating or reciprocating) times the cycle rate (e.g. rpm). The flow capacity is thus fixed by the design, size, and operating speed of the pump. The pressure (or head) that the pump develops depends upon the flow resistance of the system in which the pump is installed and is limited only by the size of the driving motor and the strength of the parts. Consequently, the discharge line from the pump should never be closed off without allowing for recycle around the pump or damage to the pump could result. They can be further classified as:

Types of Positive Displacement Pumps Reciprocating pumpsPumping takes place by to and fro motion of the piston or diaphragm in the cylinder. It is often used where relatively small quantity of liquid is to be handled and where delivery pressure is quite large.

Piston pump: A piston pump is a type of positive displacement pump where the high- pressure seal reciprocates with the piston. The pump has a piston cylinder arrangement. As the piston, goes away after the delivery stoke, low pressure is created in the cylinder which opens the suction valve. On forward stoke, the fluid filled inside the cylinder is compressed which intern opens the delivery valve for the delivery of liquid.

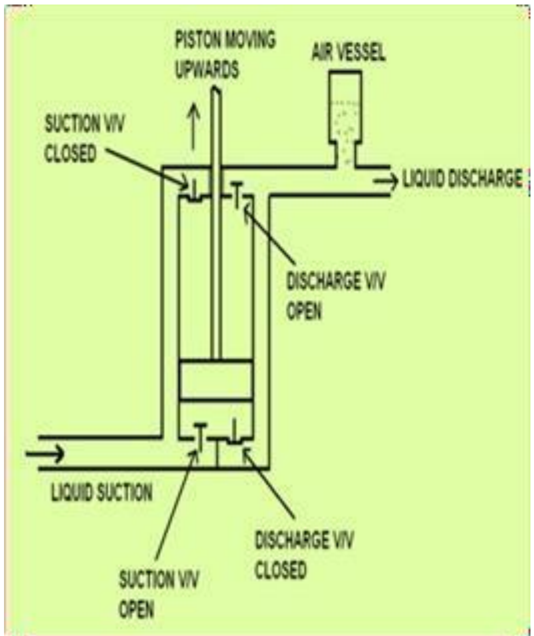


Fig. Piston pump

Diaphragm pump: uses a combination of the reciprocating action of a rubber, thermoplastic or Teflon diaphragm and suitable non-return check valves to pump a fluid. Sometimes this type of pump is also called a membrane pump.

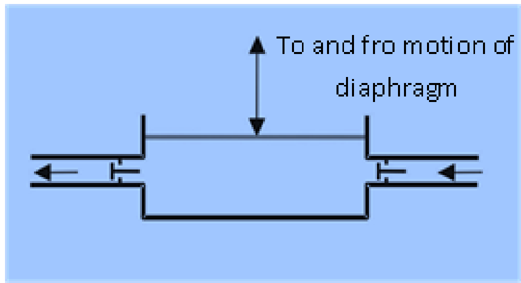


Fig. Diaphragm pump

Rotary pumps

In rotary pumps, relative movement between rotating elements and the stationary element of the pump cause the pumping action. The operation is different from reciprocating pumps, where valves and a piston are integral to the pump. They also differ from centrifugal pumps, where highvelocity is turned into pressure. Rotary pumps are designed so that a continuous seal is maintained between inlet and outlet ports by the action and position of the pumping elements andclose running clearances of the pump. Therefore, rotary pumps do not require valve arrangements similar to reciprocating pumps.

Gear pumps: uses the meshing of gears to pump fluid by displacement. They are one of the most common types of pumps for hydraulic fluid power applications. The rigid design of the gears and houses allow for very high pressures and the ability to pump highly viscous fluids.

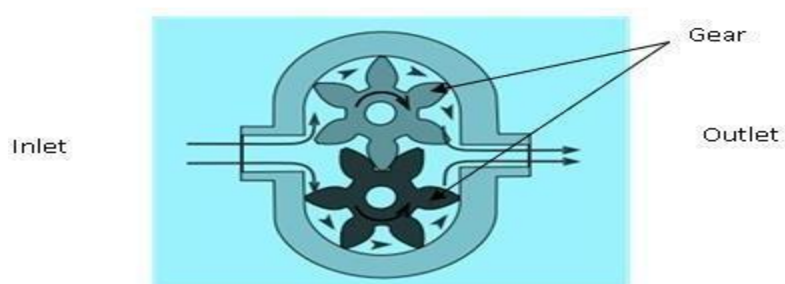


Fig. Gear pump

Screw Pump: These pumps are rotary, positive displacement pumps that can have one or more screws to transfer high or low viscosity fluids along an axis. Although progressive cavity pumps can be referred to as a single screw pumps, typically screw pumps have two or more intermeshing screws rotating axially clockwise or counter clockwise. Each screw thread is matched to carry a specific volume of fluid. Screw pumps provide a specific volume with each cycle and can be dependable in metering

applications.

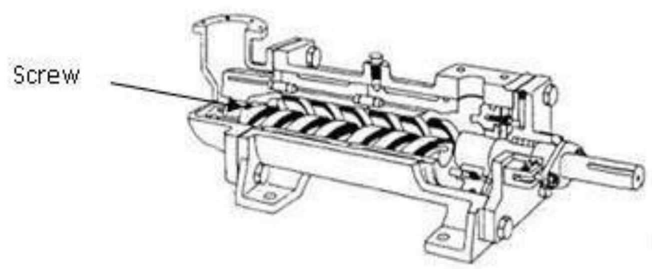


Fig. Screw pump

Vane pump: A rotary vane pump is a positive-displacement pump that consists of vanes mounted to a rotor that rotates inside of a cavity. In some cases, these vanes can be variable length and/or tensioned to maintain contact with the walls as the pump rotates.

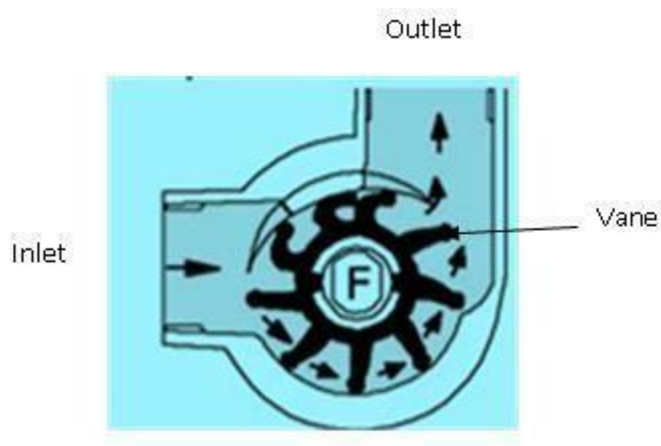


Fig. Vane pump

Dynamic Pressure Pumps

In dynamic pressure pump, during pumping action, tangential force is imparted which accelerates the fluid normally by rotation of impeller. Some systems which contain dynamic pump may require positive displacement pump for priming. They are normally used for moderateto high discharge rate. The pressure differential range for this type of pumps is in a range of low to moderate. They are popularly used in a system where low viscosity fluids are used.

Centrifugal pumps

They use a rotating impeller to increase the pressure of a fluid. Centrifugal pumps are commonly used to move liquids through a piping system. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits into the downstream piping system. Centrifugal pumps are used for large discharge through smaller heads. These types of pumps are used for supply of water and handling of milk in dairy plants.

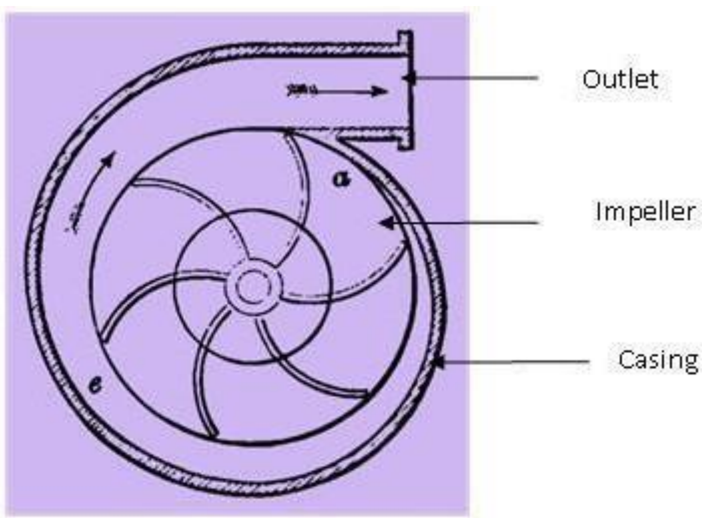


Fig. Centrifugal pump

Propeller pump

A propeller pump is a high flow, low lift impeller type device featuring a linear flow path. The propeller pump may be installed in a vertical, horizontal, or angled orientation and typically has its motor situated above the water level with the impeller below water. These pumps function by drawing water up an outer casing and out of a discharge outlet via a propeller bladed impeller head.

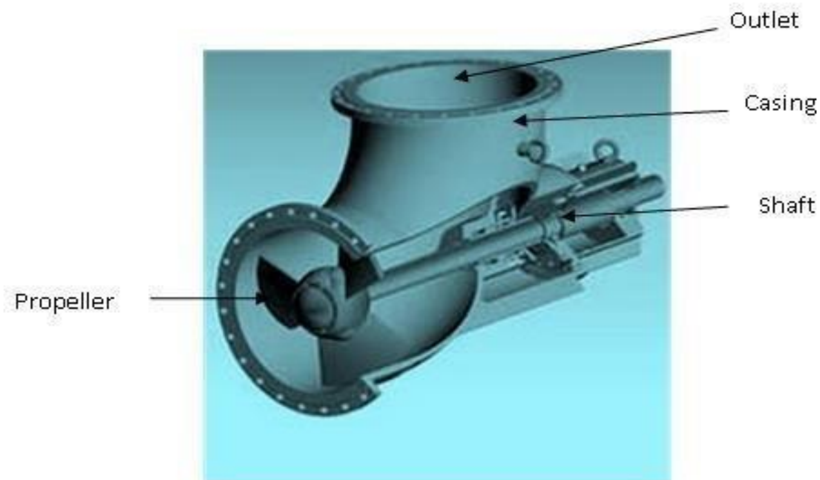


Fig. Propeller pump

Turbine pump

Turbine pumps are centrifugal pumps that use pressure and flow in combination with a rotary mechanism to transfer fluid. They typically employ blade geometry, which causes fluid circulation around the vanes to add pressure from inlet to outlet. Turbine pumps operate using kinetic energy to move fluid utilizing an impeller. The centrifugal force drives the liquid to the housing wall in close proximity to the vanes of the impeller or propeller. The cyclical movement of the impeller produces pressure in the pumping bowl. The shape of turbine pumps also contributes to suction and discharge rates.

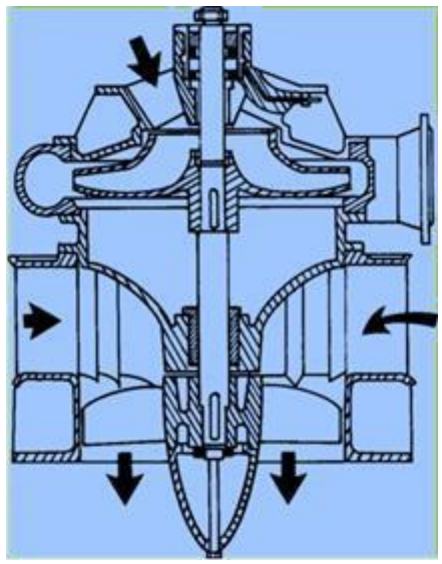


Fig. Turbine pump

Centrifugal Pump

Main parts of centrifugal pump:

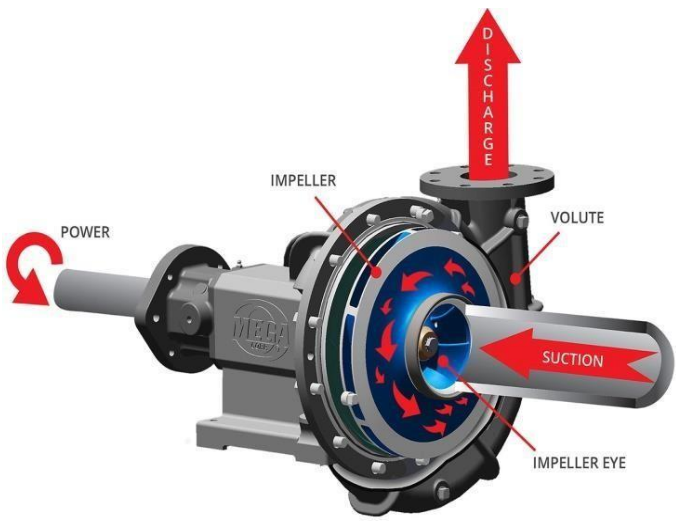
1. Impeller
2. Casing
3. Suction pipe with a float value and a strainer
4. Delivery pipe.

Working Principle of a Centrifugal Pump

1. A centrifugal pump can be defined as a special form of rot dynamic pump that uses a rotating impeller for raising the rate of the fluid.
2. The main function of centrifugal pumps is to convert the electric energy of a motor into usable kinetic energy that causes pressure and forces the liquid to come out.
3. The pump is comprised of two main components, i.e. the impeller and the volute in which the change of energy takes place. Kinetic energy is directly converted into pressure in the volute, which is an inactive part of the pumping device.
4. The impeller, on the other hand, is the active revolving part that directly converts the driver energy into kinetic energy.
5. The centrifugal force is developed as the liquid travels into the pumping section and also the eye part of the impeller. As the impeller starts to revolve, the liquid is instantly turned

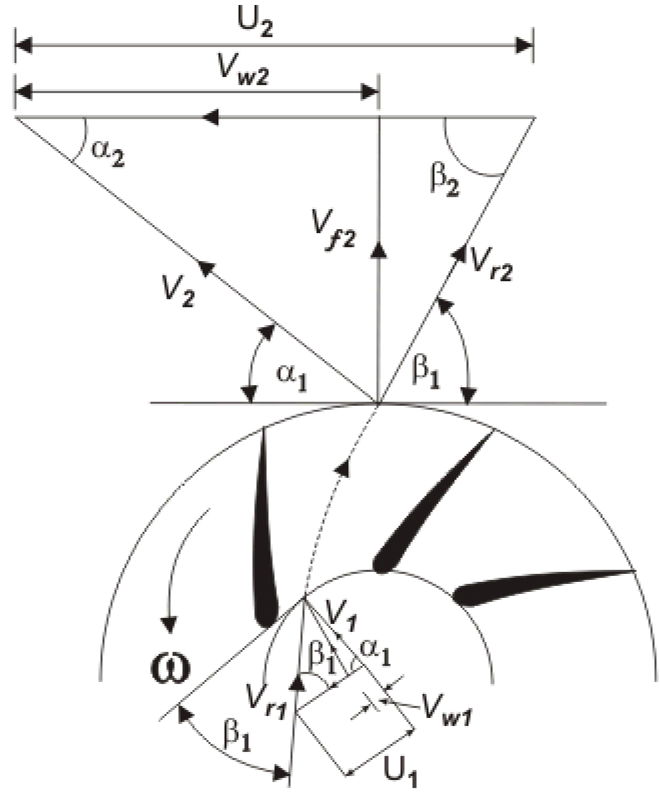
into centrifugal force that vanes outward. As this liquid moves out of the impeller, it causes a low pressure area to develop that allows more liquid to come out easily.

There are numerous different types of centrifugal pumps currently available in the market. These include single stage centrifugal pumps and multistage centrifugal pumps. They can also be classified based on their flow types, i.e. radial flow; axial flow and mixed flow. The pumps can also differ based on their individual capacities and sizes.



Efficiencies of a centrifugal pump:

1. Manometric efficiency
2. Overall efficiency
3. Mechanical efficiency



Velocity triangle

Manometric efficiency:

$$\eta_{man} = \frac{H_m g}{Vw_2 u_2}$$

Overall efficiency:

$$\eta_o = \frac{\frac{(wh_m)}{1000}}{sp}$$

Mechanical efficiency:

$$\eta_m = \frac{\frac{w}{g}}{sp}\left(\frac{Vw_2 u_2}{1000}\right)$$

Centrifugal pump is a reverse of a radially inward flow reaction on turbine:

1. Work done by the impeller on water per
   Second per unit weight of water = Work done increase of turbine
   Striking per second.

$$= \left[\frac{1}{g}(VW_1U_1 - VW_2U_2)\right] = \frac{1}{g}\left[Vw_2U_2 - Vw_1U_1\right]$$

$$= 1/g\ Vw_2u_2$$

2. Work done per sec = $\frac{w}{g} Vw_2U_2$
3. W= Weight of the water w= $\rho g Q$
   Q = volume of water
4. Q = A x VQ = πD_1B_1 x V_{f1}
   = $\pi D_2B_2V_{f2}$

Suction head (h_s):

It is the vertical height of the centre line of the centrifugal pump above his water surface in the tank or pump from which water is to be lifted. The height is also called as suction lift and denoted by 'h_s'.

Delivery head (h_d):

The vertical distance between the centre line of the pump and the water surface in the tank to where water is delivered is known as delivery head.

Static head (h_s):

The sum of suction head and delivery head is known as static head.

$H_s = h_s + h_d$

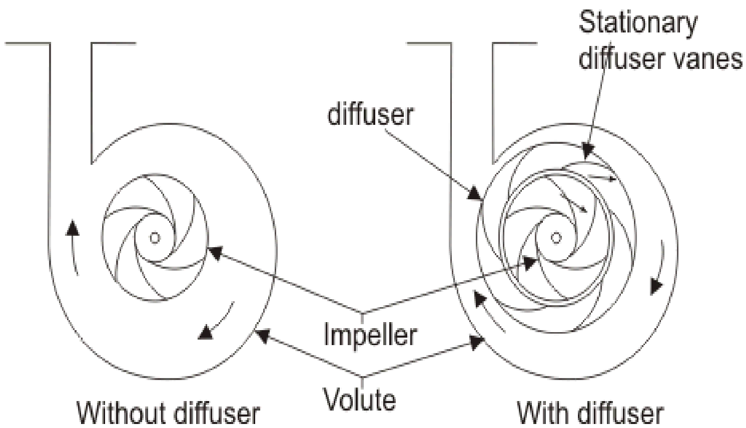
Manometric head (h_m):

The manometric head is defined as the head against which a centrifugal pump has to work.

(i) H_m = Head imported by the impeller to water – loss of head on the pump.

(ii) H_m= Vw_2u_2/g – loss of head in impeller and casing.

(iii) H_m= Vw_2u_2 /g -- if loss of pump is zero.



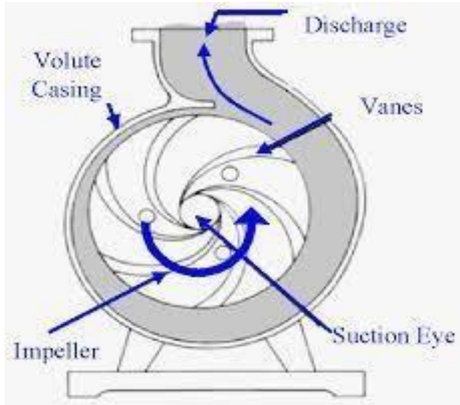
Working Mechanism of a Centrifugal Pump

A centrifugal pump is one of the simplest pieces of equipment in any process plant. Its purpose is to convert energy of a prime mover (a electric motor or turbine) first into velocity or kinetic energy and then into pressure energy of a fluid that is being pumped. The energy changes occur by virtue of two main parts of the pump, the impeller and the volute or diffuser. The impeller is the rotating part that converts driver energy into the kinetic energy. The volute or diffuser is the stationary part that converts the kinetic energy into pressure energy.

Generation of Centrifugal Force

The process liquid enters the suction nozzle and then into eye (center) of a revolving device known as an impeller. When the impeller rotates, it spins the liquid sitting in the cavities between the vanes outward and provides centrifugal acceleration. As liquid leaves the eye of the impeller a low-pressure area is created causing more liquid to flow toward the inlet. Because the impeller blades are curved, the fluid is pushed in a tangential and radial direction by the centrifugal force. This force acting inside the pump is the same one that keeps water inside a bucket that is rotating at the end of a string.

Figure below depicts a side cross-section of a centrifugal pump indicating the movement of the



liquid.
General Components of Centrifugal Pumps

A centrifugal pump has two main components:

I. A rotating component comprised of an impeller and a shaft
II. A stationary component comprised of a casing, casing cover, and bearings.

The general components, both stationary and rotary, are depicted in Figure 1. The main components are discussed in brief below. Figure 2 shows these parts on a photograph of a pump in the field.

The impellers are fitted inside the casings.

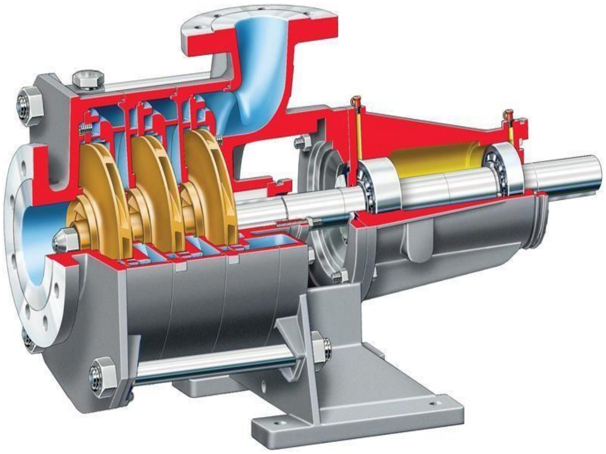
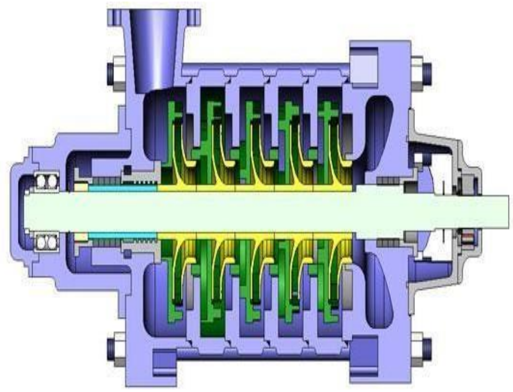
1. Volute casings build a higher head; circular casings are used for low head and high capacity.

- A volute is a curved funnel increasing in area to the discharge port as shown in Figure
  B.03. As the area of the cross-section increases, the volute reduces the speed of the liquidand increases the pressure of the liquid.
- One of the main purposes of a volute casing is to help balance the hydraulic pressure on the shaft of the pump. However, this occurs best at the manufacturer's recommended capacity. Running volute-style pumps at a lower capacity than the manufacturer recommends can put lateral stress on the shaft of the pump, increasing wear-and-tear on the seals and bearings, and on the shaft itself. Double-volute casings are used when the radial thrusts become significant at reduced capacities.

What is a multistage pump?

A centrifugal pump containing two or more impellers is called a multistage centrifugal pump. The impellers may be mounted on the same shaft or on different shafts. For higher pressures at

the outlet, impellers can be connected in series. For higher flow output, impellers can beconnected parallel.

1. single stage pump refers to only one impeller pump, highest head is only 125 meters;
2. the multistage pump means to have two or more impeller pump, highest head can more than 125 meters; Multistage pumps in single stage pump head must match two levels motor cases, by adding more impeller number to equip with level 4 motor, which can improve pump using life and reduces the noise, but multistage pumps maintenance is relatively more difficult than single stage pump.
3. pump actual required head is less than 125 m, can according to pump room area, pump prices (multistage pump price usually higher than single stage pump) factors comprehensive consideration to choose single stage pump or multistage pumps. With the development of technology, single stage impeller pump through improving the speed of the pump to improve pump head, can replace multistage pumps, just the price is more expensive.

Reciprocating Pumps

Reciprocating pumps are very important part of the ships machinery and any other industry which is present in the world. High pressure is the main characteristic of this pump and this high pressure output are being used in places like starting of the engine or you can say the building of pressure in the fluids. But there are used in limited application because they require lot of maintenance. These pumps are positive displacements pumps and that is the reason they do not

require any type of priming for their functioning in the starting period of the pump.

Reciprocating Pump is a plunger Pump that is used for positive displacement. Pump that is Reciprocating is used for handling a relatively low or small quantity of liquid. The delivery pressure of a Pump that is Reciprocating is considerably large.

A reciprocating pump is a class of positive-displacement pumps which includes the piston
pump, plunger pump and diaphragm pump In reciprocating pumps, the chamber inwhich the liquid is trapped, is a stationary cylinder that contains the piston or plunger.

Description

The main parts of a single acting reciprocating pump are discussed below.

1. Cylinder, Piston, Piston Rod, Connecting Rod and Crank

A single action reciprocating pump consists of a piston, which moves forwards and backwards inside a close fitting cylinder. The movement of the piston is obtained by connecting the piston rod to the crank by means of a connecting rod. The crank is rotated by an electric
motor.

2. Suction Pipe and Suction Valve

Suction pipe is connected to the cylinder. Suction valve is a one way valve, i.e., non-return valve. It allows the liquid to flow in one direction only. That is, it permits the liquid from the suction pipe to the cylinder.

3. Delivery Pipe and Delivery Valve

Delivery pipe is connected to the cylinder. Delivery valve is also one non-return valve. It permits the liquid to flow in one direction only. That is, it allows the liquid from the cylinder to the delivery pipe.

Working principal of the reciprocating pump

The working of the reciprocating pump is very simple and just like an I.C engine. First of all the piston has the function of providing the suction force, so that the liquid can be lift up or can be sucked in with great force. After that comes the compression part which will impart the required pressure energy to the fluids. In this part of the phase the piston have to do a great work so that the liquid can be compressed properly and its pressure can increased to the desired level. The inlet and the outlet valve open at a certain pressure which is set by the manufacturer.

If the piston is of single acting type which means it can suck from one side and transmit to the same side only. But we can have the double reciprocating pump too which have the function of the giving suction and discharge simultaneously in each stroke. This pump can be used as the compressor also but for that we have to have a good valve arrangement which can operate with good frequency.

Note: It is to be noted that the reciprocating pump is a positive displacements pump which meansthat the fluid can only move in one direction and can never reverse back. So due to this the pumpis always started with outlet valve open otherwise the pressure will keep on building and this willlead to rupturing of the pipeline or even the pump itself. But if relief valve is fitted then this pressure will come down

Uses of reciprocating pump

There are various uses of the reciprocating pump and they are as following:
1) The lubricating pump is a reciprocating pump and it supplies the lubrication oil to the mainengine.
2) Main bilge suction pump is also a reciprocating pump.
3) For ballast they are sometimes used.

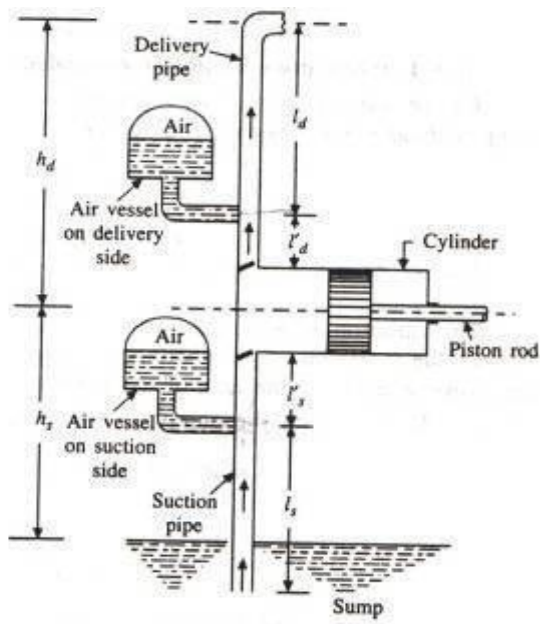
Advantages of Reciprocating pump

Advantages of the reciprocating pump are as given below:
1) Gives high pressure at outlet.
2) Gives high suction lift.
3) Priming is not required in this pump.
4) They are used for air also.

Disadvantages of Reciprocating pump

Disadvantages of the reciprocating pump are given below:
1) High wear and tear, so requires a lot maintenance.
2) The flow is not uniform, so we have to fit a bottle at both ends.
3) The flow is very less and cannot be used for high flow operations.
4) More heavy and bulky in shape.
5) Initial cost is much more in this pump.

The **air vessel**, in a **reciprocating pump**, is a cast iron closed chamber having an opening at itsbase. These are fitted to the suction pipe and delivery pipe close to the cylinder of the **pump**.
The **vessels** are used for the following purposes: (a) To get continuous supply of liquid at auniform rate.



Reciprocating Pump Fitted With Air Vessel At Both Suction AndDelivery Side

Air Vessel
An air vessel usually fitted in the discharge pipe work to dampen out the pressure variations during discharge. As the discharge pressure rises the air is compressed in the vessel, and as the pressure falls the air expands. The peak pressure energy is thus stored in the air and returned tothe system when pressure falls. Air vessels are not fitted on the reciprocating boiler feed pumpssince they may introduce air into the de-aerated water.

Purposes of Air vessel:

1) To obtain liquid at uniform discharge.

2) Due to air vessel frictional head and acceleration head decreases and the work overcomingfriction resistance in suction and delivery pipe considerably decreases which results in good amount of work.
3) Reciprocating pump can run at high speed without flow separation

Working:

The top half contains compressed air and lower half contains fluid being pumped. Air and water are separated by a flexible diaphragm which is movable as per difference of pressure betweentwo fluids. Air vessel is connected very near to the pump at nearly pump level. Without air vesselfrictional head increases and reaches a maximum value at mid stroke and decreases to zero. With air vessel frictional head is constant throughout the stroke.

Problem : Calculate the discharge of reciprocating pump (single cutting) if area of cylinder is $0.25\ m^2$, length of stroke is 0.15 m, number of cylinder =1 and speed of pump is 50 rpm.

$$Discharge\ Q = A \times L \times \frac{N}{60} \times n \times numberof\ cutting$$

$A = 0.25\ m^2$

L = 0.15 m

N = 50 rpm

n = 1

number of acting =1

$$Q = 0.25 \times 0.15 \times \frac{50}{60} \times 1 \times 1$$

$$= 0.03125\ m^3 /s$$

Problem :

A centrifugal pump having outer diameter equal to two times the inner diameter and running at 1200rpm works against a total head of 32m. The velocity of flow through the impeller is constant and equal to 3m/s. The vanes are set back at an angle of 30° at the outlet. If the outer diameter of the impeller is 600mm and width at outlet is 50mm, determine (a) vane angle at inlet, (b) work done per second by impeller, and (c) manometric efficiency.

Given data:

$D_2 = 2D_1$

Speed, $N = 1200 rpm$

Head, $H_m = 32m$

Velocity of flow, $V_{f_1} = V_{f_2} = 3m/s$

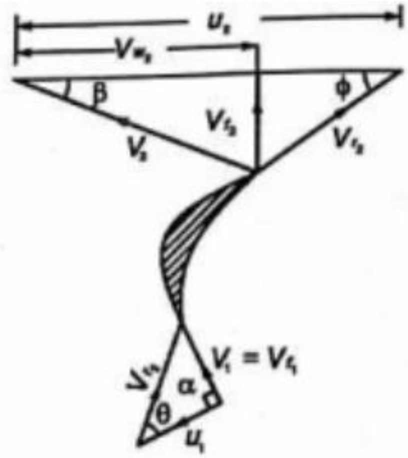


Figure 4.24

Work done per second by the impeller,

$$= \frac{W}{g} V_{w_2} u_2 = \frac{\rho g Q}{g} \times V_{w_2} u_2$$

$$= \frac{1000 \times 9.81 \times 0.2827}{9.81} \times 13.65 \times 37.7 = \mathbf{145478.8} \boldsymbol{m/s} \textbf{ Ans.}$$

Manometric efficiency,

$$\eta_{mano} = \frac{gH_m}{V_{w_2} u_2} = \frac{9.81 \times 32}{13.65 \times 37.7} = 0.61 = \mathbf{61\%} \qquad \textbf{Ans.}$$