CONTENTS

CHAPTER 1

FLOW MEASUREMENT

1.1 Venturimeter

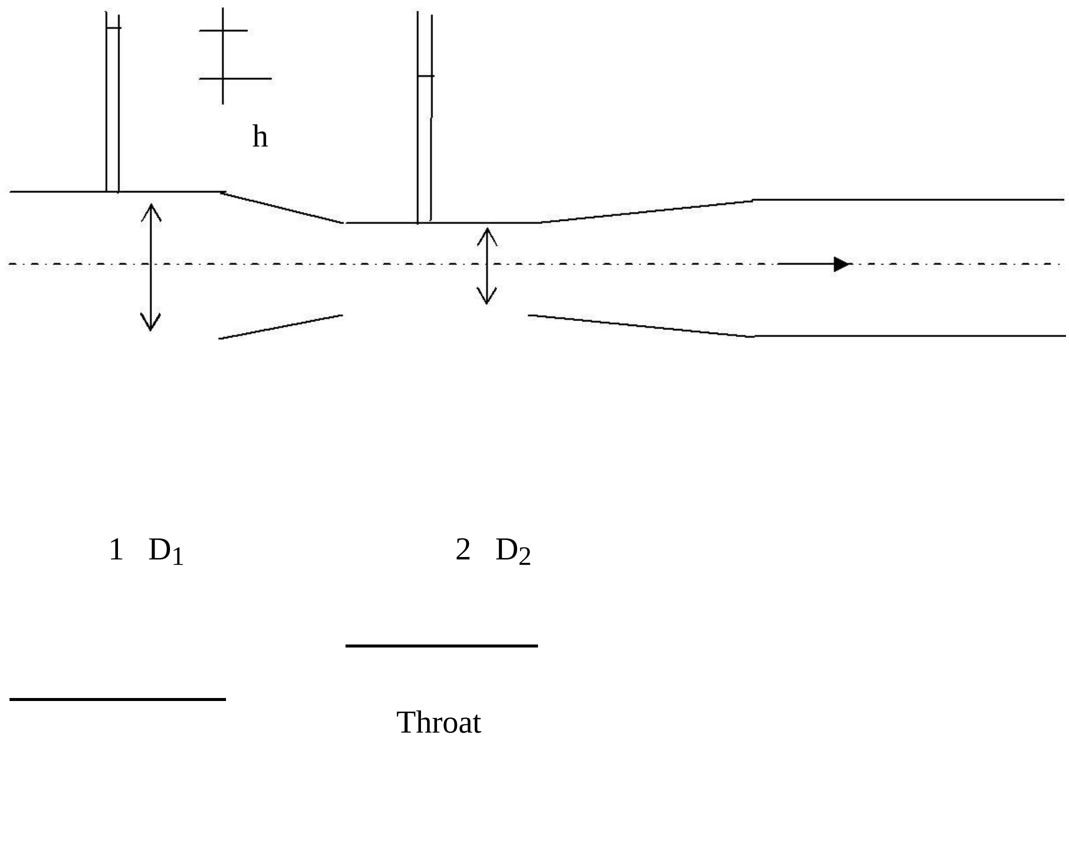


Fig.1.1 Venturimeter

A venturimeter is a device which is used for measuring the rate of flow through a pipe. As shown in Fig.1.1, a venturimeter consists of (1) inlet section followed by a convergent cone, (2) the throat, and (3) a gradually divergent cone. Since the cross sectional area of the throat section is smaller than the cross-sectional area of the inlet section, the velocity of flow at the throat section will become greater than that at the inlet section, according to the continuity equation.

The increase in the velocity of flow at the throat section results in the decrease in the pressure at this section. As such a pressure difference is developed between the inlet and the throat sections of the venturimeter. The pressure difference between these sections can be determined either by connecting a differential manometer between the pressure tappings provided at these sections or by connecting a separate pressure gauge at each of the pressure tappings.

Bernoulli's equation between the sections 1 and 2

$$\frac{P_1}{\omega} + \frac{v_1^2}{2g} + z_1 = \frac{P_2}{\omega} + \frac{v_2^2}{2g} + z_2 + losses$$

$$\frac{P_1}{\omega} - \frac{P_2}{\omega} = \frac{v_2^2}{2g} - \frac{v_1^2}{2g} = h \qquad \ldots\ldots\ldots\ldots\ldots \quad (1.1)$$

Continuity equation

$$Q = A_1V_1 = A_2V_2 \qquad \ldots\ldots\ldots\ldots\ldots \ (1.2)$$

From equation (1.1) and (1.2)

$$V_1 = A_2 \sqrt{\frac{2gh}{A_1^2 - A_2^2}}$$

$$\text{Theoretical } Q = A_1V_1 = \frac{A_1 A_2 \sqrt{2gh}}{\sqrt{A_1^2 - A_2^2}} \qquad \ldots\ldots\ldots\ldots\ldots \ (1.3)$$

Equation (1.3) is an ideal equation obtained by neglecting all losses.

The actual discharge may be obtained by multiplying the coefficient of discharge which is defined as the ratio between the actual discharge and the theoretical discharge of the venturimeter.

$$\text{Actual } Q = C_d \frac{A_1 A_2 \sqrt{2gh}}{\sqrt{A_1^2 - A_2^2}} \qquad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\text{(1.4)}$$

As shown in Fig.1.2 if a U tube manometer is used for measuring the difference between the pressure heads at sections 1 and 2, then for a difference between heads at section 1 and 2, then for a difference in the levels of manometric liquid in the two limbs equal to x, we have

$$\frac{p_1}{\omega} - \frac{p_2}{\omega} = h = x\left(\frac{\omega_m}{\omega} - 1\right)$$

where ω_m and ω are the specific weights of the manometric liquid and the liquid flowing in the venturimeter respectively. If S_m and S are respectively the specific gravities of the manometric liquid and the liquid flowing in the ventutimeter, then the expression for the venturi head becomes

$$\left(\frac{p_1}{\omega} - \frac{p_2}{\omega}\right) = h = x\left(\frac{S_m}{S} - 1\right) \qquad \text{...............(1.5)}$$

On the other hand if an inverted U tube manometer is used for measuring the difference between the pressure heads at sections 1 and 2, then since $S_m < S$, we have

$$\frac{p_1}{\omega} - \frac{p_2}{\omega} = h = x\left(1 - \frac{S_m}{S}\right) \qquad \text{.................(1.6)}$$

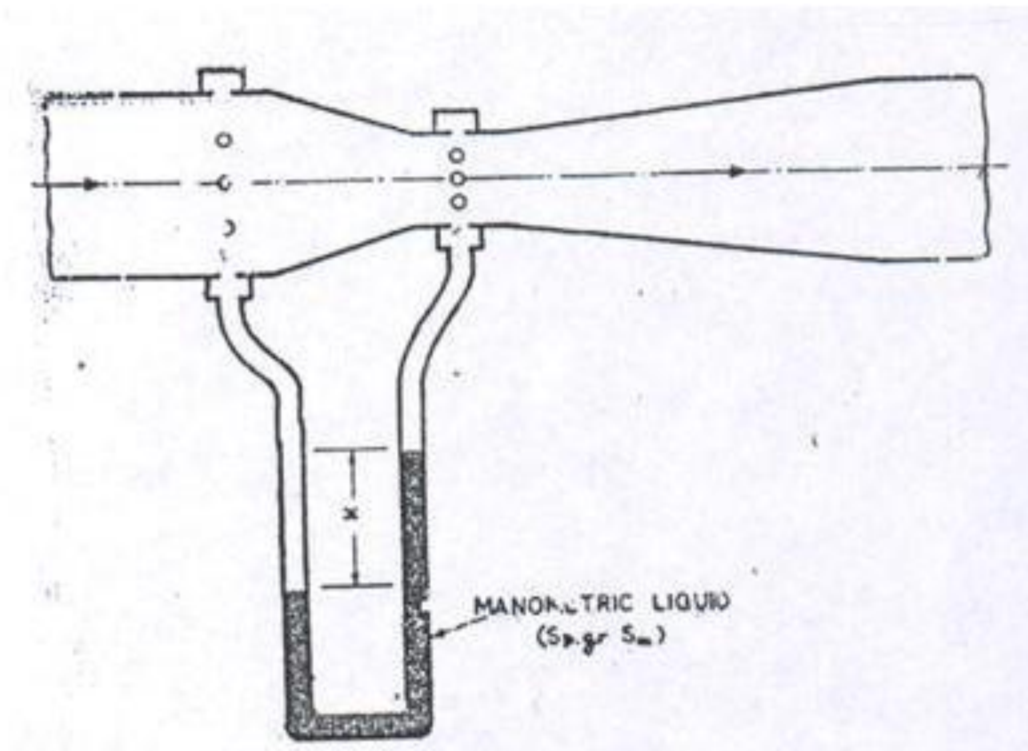
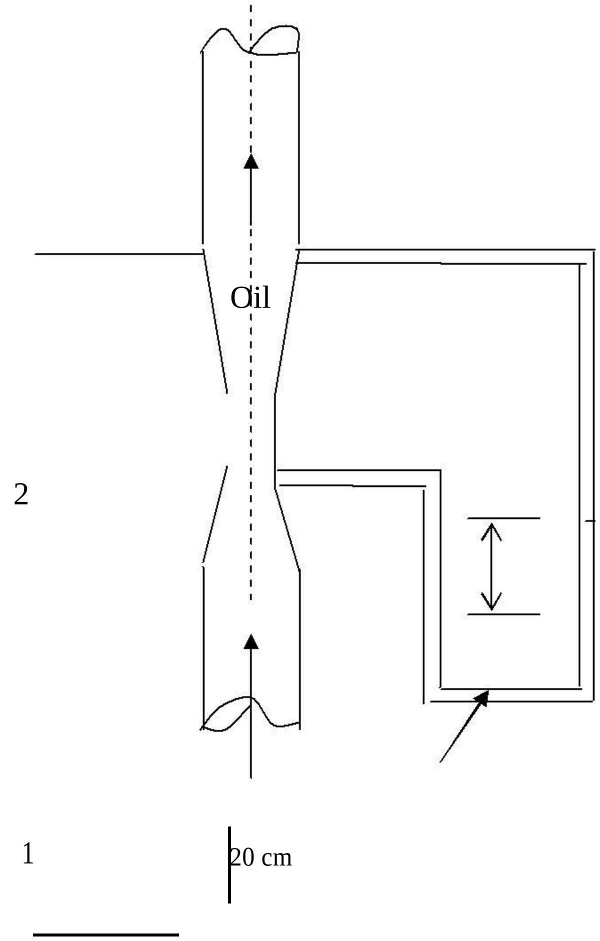


Fig.1.2 Horizontal venturimeter with U tube differential manometer

Example 1.1 A vertical venturimeter is provided in a vertical pipe to measure the flow of oil (Sp.gr 0.8). The difference in elevations of the throat section and the entrance section is 1 m. Determine the quantity of oil flowing in the pipe. Neglect losses.



$$h = 0.4\left(\frac{13.6}{0.8} - 1\right) = 6.4 \text{ m of oil}$$

C_d = 1.0 as the lossess are neglected

A_1 = 0.126 m^2, A_2 = 0.0314 m^2

$$Q = C_d \frac{A_1 A_2 \sqrt{2gh}}{\sqrt{A_1^2 - A_2^2}}$$

= 0.363 m^3/s = 363 l/s

1.2 Orifice Meter

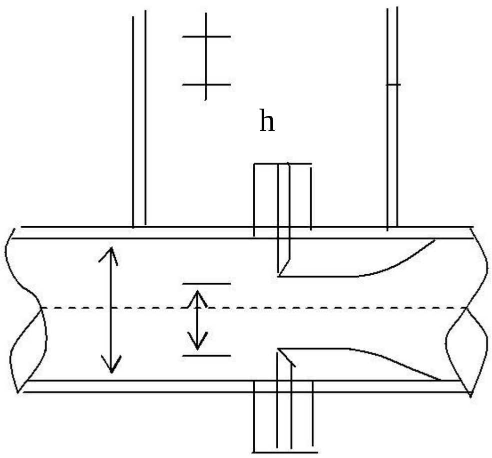


Fig.1.3 Orifice meter

An orifice meter is another simple device used for measuring the discharge through pipes. It also works on the same principle as that of venturimeter i.e, by reducing the cross-sectional area of the flow passage a pressure difference between the two sections is developed and the measurement of the pressure difference enables the determination of the discharge through the pipe.

An orifice meter consists of flat circular plate which has a circular sharp edged hole called orifice, which is concentric with the pipe. Two pressure tappings are provided one at section 1 which is located on upstream side and the other at section 2 which is located on the downstream side from the orifice plate. On the downstream side the pressure tapping is provided quite close to the orifice plate at the section where the converging jet of fluid has almost the smallest cross sectional area (which is known as *vena contracta*) resulting in almost the maximum velocity of flow and consequently the minimum pressure at this section.

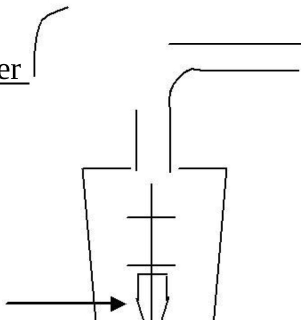
Actual $$Q = C_d \frac{A_1 A_2 \sqrt{2gh}}{\sqrt{A_1^2 - A_2^2}}$$

The coefficient of discharge for an orifice meter is much smaller than that for a venturimeter. This is because in the case of an orifice meter there are no gradual converging and diverging flow passages as in the case of a venturimeter, which results in a greater loss of energy and consequent reduction of the coefficient of discharge for an orifice meter.

1.3 Other Flow Measurement Devices

Besides the above described device there are some more devices for measuring discharge through pipes. These devices are rotameter, nozzle meter and elbow meter.

Variable-area Meter/ Rotameter



Float

Fig.1.4 Rotameter

It consists of a vertical transparent conical tube in which there is a rotor or float having a sharp circular upper edge. As the liquid begins to flow through the meter, it lifts the rotor until it reaches a steady level corresponding to the discharge. This rate of flow of liquid can then be read from graduations engraved on the tube by prior calibration, the sharp edge of the float serving as a pointer.

Example 1.2 An orifice meter consisting of 10 cm diameter orifice in a 25 diameter pipe has coefficient 0.63. The pipe delivers oil (Sp.gr 0.8). The pressure difference on the two sides of the orifice plate is measured by a mercury oil differential manometer. If the differential gauge reads 80 cm of mercury, calculate the rate of flow in litre per sec.

$A_1 = 490.87 \text{ cm}^2$

$A_2 = 78.54 \text{ cm}^2$

$h = 80$ cm of mercury

$$= 80 \times \frac{(13.6-0.8)}{0.8}$$

$= 1280$ cm of oil

$$Q = C_d \frac{A_1 A_2 \sqrt{2gh}}{\sqrt{A_1^2 - A_2^2}}$$

$= 81970\ cm^3/sec$

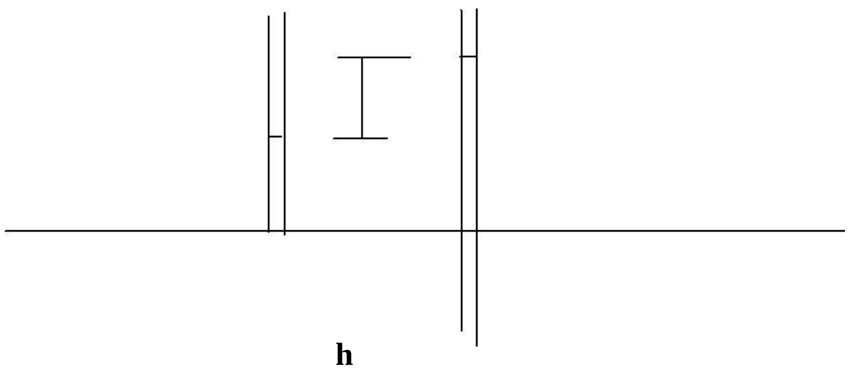
$= 81.97$ litres/sec

1.4 Pitot Tube

A pitot tube is a simple device used for measuring the velocity of flow. The basic principle used in this device is that if the velocity of flow at a particular point is reduced to zero, which is known as stagnation point, the pressure there is increased due to the conversion of the kinetic energy into pressure energy, and by measuring the increase in the pressure energy at this point the velocity of flow may be determined. In its simplest form a pitot tube consists of a glass tube, large enough for capillary effects to be negligible, and bent at right angles.

$V = \sqrt{2gh}$ ....(1.7) When a pitot tube is used for measuring the velocity of flow in a pipe or other closed

conduit then the pitot tube may be inserted in the pipe as shown in Fig.1.5.



Stagnation point

Fig.1.5 Pitot tube used for measuring velocity in pipes

CHAPTER 2

FLOW THROUGH ORIFICES AND MOUTHPIECES

2.1 Definitions

An orifice is an opening having a closed perimeter, made in the walls or the bottom of a tank or a vessel containing fluid through which the fluid may be discharged. The orifice may be classified on the basis of their size, shape, shape of the upstream edges and the discharge conditions.

A mouthpiece is a short tube of length not more than two to three times its diameter, which is fitted to a circular opening or orifice of the same diameter, provided in a tank or vessel containing fluid, such that it is an extension of the orifice and through which also the fluid may be discharged. Both orifices and mouthpieces are usually used for measuring the rate of flow of fluid.

2.2 Classification of Orifices and Mouthpieces

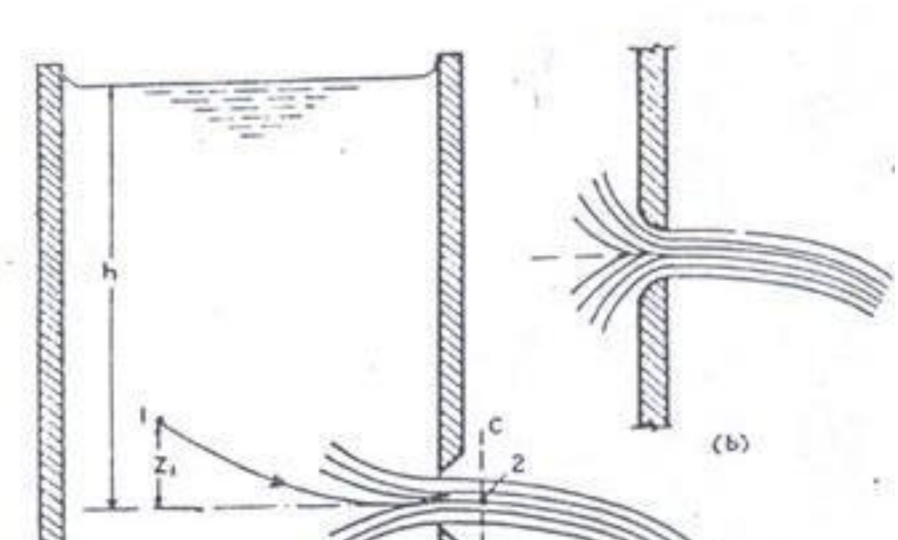
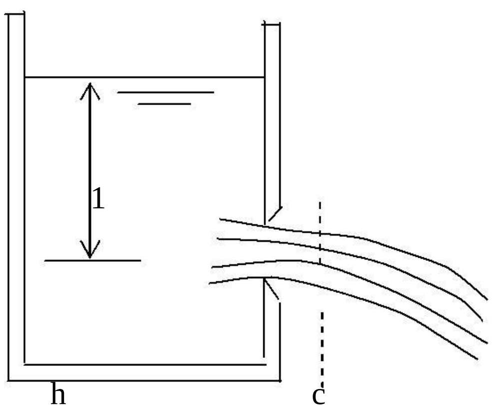


Fig.2.1 (a) Sharp-edged orifice discharging free (b) Bell-mouth orifice

The orifices may be classified on the basis of their size, shape, shape of the upstream edges and the discharge conditions. According to the size, the orifices may be classified as *small and large orifices*. According to the shape, the orifices may be classified as *circular, rectangular, square and triangular*. According to the shape of the upstream edge the orifices may be classified as *sharp edged orifices and bell mouthed orifices or orifices with the round corner*. As shown in Fig.2.1 a sharp edged orifice has the bevelled side facing the downstream so that there is a minimum contact with the fluid flowing through the orifices and consequently the minimum frictional effects. According to discharge condition, the orifices may be classified as orifices *discharging free and drowned or submerged orifices.*

2.3 Flow Through Small Orifice



2

c

Fig2.2 Small Orifice

Fig.2.2 shows a sharp edged small orifice in one side of a reservoir containing liquid. The liquid will emerge from the orifice as a free jet, that is, a jet discharged in the atmosphere and will therefore be under the influence of gravity only. By Bernoulli's equation between the points 1 and 2,

$$\frac{P_1}{\omega} + \frac{v_1^2}{2g} + z_1 = \frac{P_2}{\omega} + \frac{v_2^2}{2g} + z_2 + losses$$

$P_1 = P_2$ = atmospheric pressure

Neglecting losses, velocity through orifice

$$V_2 = \sqrt{2gh} \qquad \text{……… (2.1)}$$

Equation (2.1) is known as Torricelli's theorem and represents theoretical velocity of the jet.

Actual velocity $V = C_v \sqrt{2gh}$ ……………(2.2)

where C_v =coefficient of velocity = $\dfrac{\text{actual velocity of jet at vena contracta}}{\text{theoretical velocity of the jet}}$ …… (2.3)

The jet area is much less than the area of the orifice due to contraction and the corresponding coefficient of contraction is defined as

$C_c = \dfrac{\text{area of jet at vena contracta}}{\text{area of orifice}}$ …………….(2.4)

At the section very close to the orifice, known as vena contracta, the velocity is normal to the cross section of the jet and hence the discharge is Actual Q = Area of jet x velocity of jet at vena contracta

$= C_c\, a \times C_v \sqrt{2gh}$

$= C_cC_v\, a \sqrt{2gh}$

$Q = C_d\, a \sqrt{2gh}$ …………….(2.5)

where C_d = coefficient of discharge = $\dfrac{\text{actual discharge}}{\text{theoretical discharge}}$ …………… (2.6)

2.4 Flow Through Large Rectangular Orifice

A vertical orifice provided in the side of the tank has its vertical dimension large enough as compared with head then the velocity of the liquid flowing through such an orifice varies over the entire cross section of the jet because of the considerable variation in head at different points in the vertical section of the orifice. Therefore for a large vertical orifice, since the velocity of the flowing liquid be considered as constant for the same equation as that for a small orifice, but it has to be determined by integrating the discharges through small elements of the area.

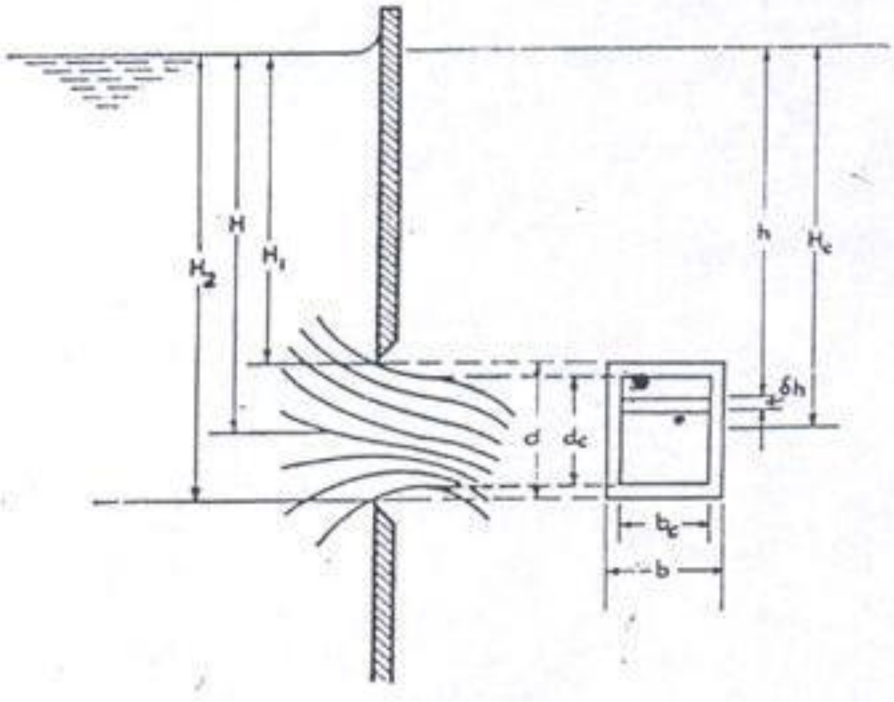


Fig.2.3 Large rectangular orifice

If we consider a small area, bdh, at a depth h, velocity through small area = √2gh

Actual discharge through the strip area dq = C_d bdh √ 2gh

Total discharge through the entire orifice $Q = \int dq = C_d\, b\sqrt{2g}\int_{H1}^{H2} h^{1/2}\, dh$

$$\text{Actual } Q = \frac{2}{3} C_d \sqrt{2g}\, b\left(H_2^{3/2} - H_1^{3/2}\right) \qquad \ldots\ldots\ldots\ldots(2.7)$$

If approach velocity is considered,

$$\text{Actual } Q = \frac{2}{3} C_d \sqrt{2g}\, b\left[\left(H_2 + \frac{V_a^2}{2g}\right)^{3/2} - \left(H_1 + \frac{V_a^2}{2g}\right)^{3/2}\right] \qquad \ldots\ldots\ldots\ldots(2.8)$$

Example 2.1 A reservoir discharges through a sluice 0.915m wide by 1.22 m deep. The top of the opening is 0.61m below the water level in the reservoir and the downstream water level is below the bottom of the opening. Calculate (a) the discharge through the opening if C_d = 0.6 and (b) the percentage error if the opening is treated as a small orifice.

(a) H_1 =0.61m; H_2 = 1.83 m

$$Q = \frac{2}{3} C_d \sqrt{2g}\, b\left(H_2^{3/2} - H_1^{3/2}\right)$$

$$= \frac{2}{3} \times 0.6 \times \sqrt{2 \times 9.81} \times 0.915 \left(1.83^{3/2} - 0.61^{3/2}\right)$$

$= 3.243\ m^3/sec$

(b) For a small orifice

$$Q = C_d A\sqrt{2gh}$$

$A = 1.116\ m^2$

h =1 .22m

Q = 0.6 x 1.116 x$\sqrt{2x9.81x1.22}$

= $3.276\ m^3/s$

Error = $\frac{3.276-3.243}{3.243}$ = 0.0102 (or) 1.02

2.5 Flow Through Submerged Orifice

(a) Totally Submerged Orifice

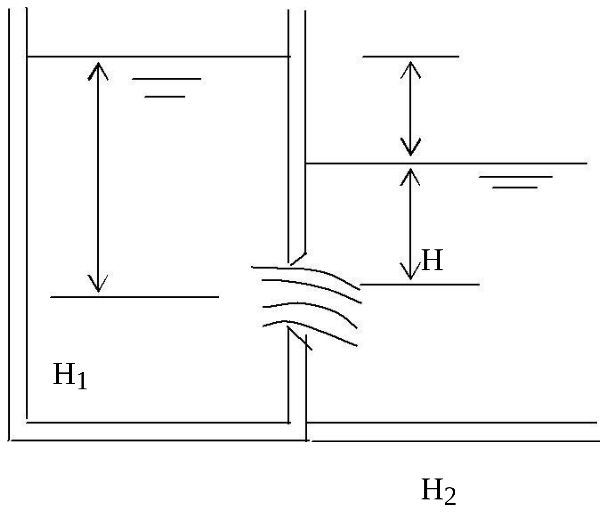


Fig.2.4 Totally submerged orifice

If an orifice has its whole of the outlet side submerged under liquid so that it discharges a jet of liquid into the liquid of the same kind then it is known as totally submerged or totally drowned orifice. The actual discharge through a totally submerged orifice is given by

$$Q = C_d A \sqrt{2gH} \quad \ldots\ldots\ldots\ldots\ldots(2.9)$$

(b)Partially Submerged Orifice

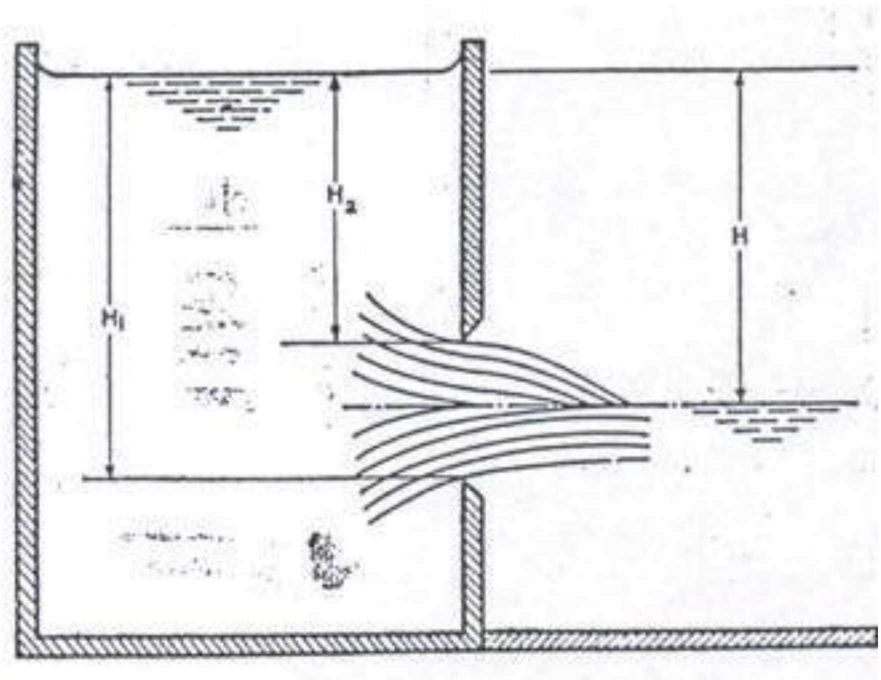


Fig.2.5 Partially submerged orifice

If the outlet side of an orifice is only partly submerged under liquid then it is known as partially drowned orifice. As such in a partially submerged orifice its upper portion behaves as an orifice discharging free, while the lower portion behaves as a submerged orifice. The discharge through a partially submerged orifice may be determined by computing separately the discharges through the free and the submerged portions and then adding

together the two discharges thus computed.

$$Q_1 = \frac{2}{3} C_{d1} b \sqrt{2g} \left\{ H^{3/2} - H_2^{3/2} \right\} \qquad \text{..........(2.10)}$$

$$Q_2 = C_{d2} b (H_1 - H) \sqrt{2gH} \qquad \text{................(2.11)}$$

$$Q = Q_1 + Q_2 \qquad \text{………..(2.12)}$$

Example 2.2 An orifice, in one side of a large tank is rectangular in shape 2 meters broad and 1 meter deep. The water level on one side, of the orifice, is 4 meters above its top edge. The water level on the other side, of the orifice, is 0.5 meter below its top edge. Calculate the discharge through the orifice per second, if $C_d = 0.625$.

$H_2 = 4$ m ; $H_1 = 4+1 = 5$ m; $H = 4+0.5 = 4.5$ m

The discharge through the free portion

$$Q_1 = \frac{2}{3} C_{d1} b \sqrt{2g} \left\{ H^{3/2} - H_2^{3/2} \right\}$$

$$Q_1 = \frac{2}{3} x 0.625 x 2 x \sqrt{2x9.81} \left\{ 4.5^{3/2} - 4^{3/2} \right\}$$

$=5.707$ m^3/s

Discharge through the submerged portion

$$Q_2 = C_{d\,2}\, b \left(H_1 - H\right)\sqrt{2gH}$$

$$Q_2 = 0.625x2 \left(5 - 4.5\right)\sqrt{2x9.81x4.5}$$

$= 5.873$ m^3/s

$$Q = Q_1 + Q_2$$

$$= 11.58\ m^3/s$$

CHAPTER 3

FLOW OVER NOTCHES AND WEIRS

3.1 Definition

A *notch* may be defined as an opening provided in the side of a tank (or vessel) such that the liquid surface in the tank is below the top edge of the opening. A *weir* is a concrete or masonry built across a river (or stream) in order to raise the level of water on upstream side and to allow the excess water to flow over its entire length to the downstream side.

The sheet of water flowing through a notch or over a weir is known as the *nappe* or *vein*. The top of a weir over which the water flows is known as the *sill* or *crest*.

3.2 Classification of Notches and Weirs

The notches are usually classified according to the shape of the opening as

rectangular notch, triangular notch, trapezoidal notch and stepped notch. Notches may also be classified according to the effect of the sides on the nappe emerging from a notch, as notch with end contraction and notch without end contraction or suppressed notch.

Weirs may be classified according to the shape of the opening, the shape of the crest, the effect of the sides on the issuing nappe and the discharge conditions. According to the shape of the opening, the weirs may be classified as rectangular, triangular and trapezoidal weirs. According to the shape of the crest, the weirs may be classified as sharp crested weir, narrow crested weir, broad crested weir and Ogee shaped weir. Weirs may also be classified according to the effect of the sides on the issuing nappe as weir with end contraction and weir without end contraction. According to the discharge conditions weir may be classified as freely discharging weir and submerged weir.

3.3 Flow Over a Rectangular Sharp Crested Weir (or) Notch

Consider a rectangular sharp crested weir or notch provided in a channel carrying water as shown in Fig3.1. Let L be the length of the crest of the weir or notch and H be the height of the water surface above its crest, which is known as the head causing the flow over the weir or notch. Consider an elementary horizontal strip of water of thickness dh and length L at a depth h below the water surface as shown in Fig.3.1.

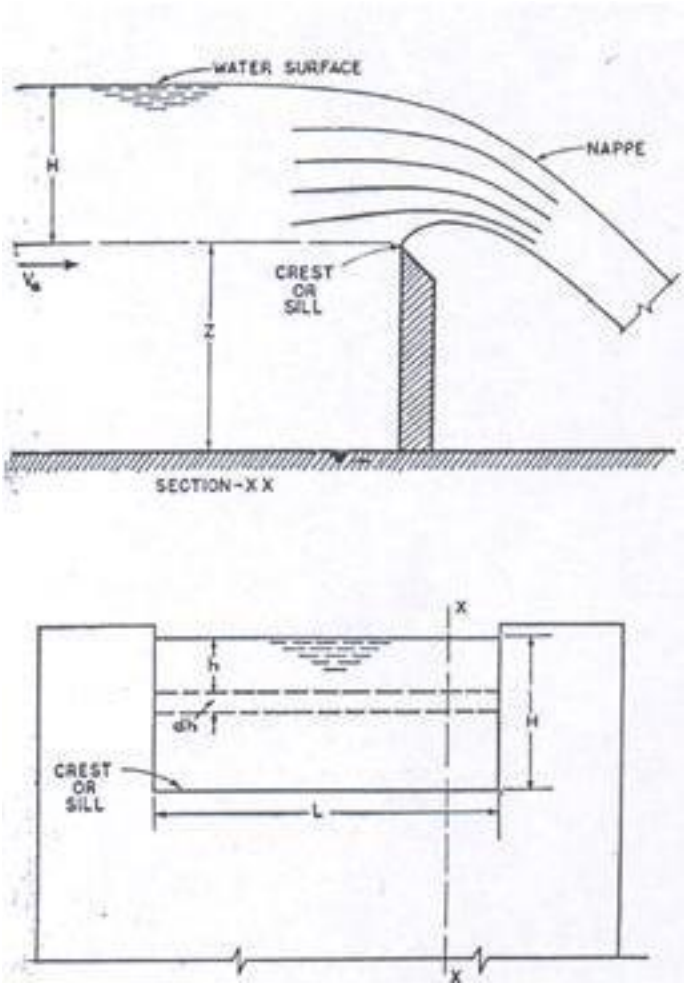


Fig.3.1 Flow over rectangular sharp-crested weir or notch

Velocity through the strip = $\sqrt{2g(h + \frac{V_a^2}{2g})}$

Discharge dq = $C_d \ L \ dh \sqrt{2g(h + \frac{V_a^2}{2g})}$

$$Q = \int dq = C_d \sqrt{2g}\, L \int_0^H \left(h + \frac{V_a^2}{2g}\right)^{1/2} dh$$

$$Q = \frac{2}{3} C_d \sqrt{2g}\, L\left[\left(H + \frac{V_a^2}{2g}\right)^{3/2} - \left(\frac{V_a^2}{2g}\right)^{3/2}\right] \qquad \text{..........(3.1)}$$

If the approach velocity V_a is not considered,

$$Q = \frac{2}{3} C_d \sqrt{2g}\, L H^{3/2} \qquad \text{...........(3.2)}$$

where H = head above the sill

3.4 Empirical Formula for Discharge Over Rectangular Weir

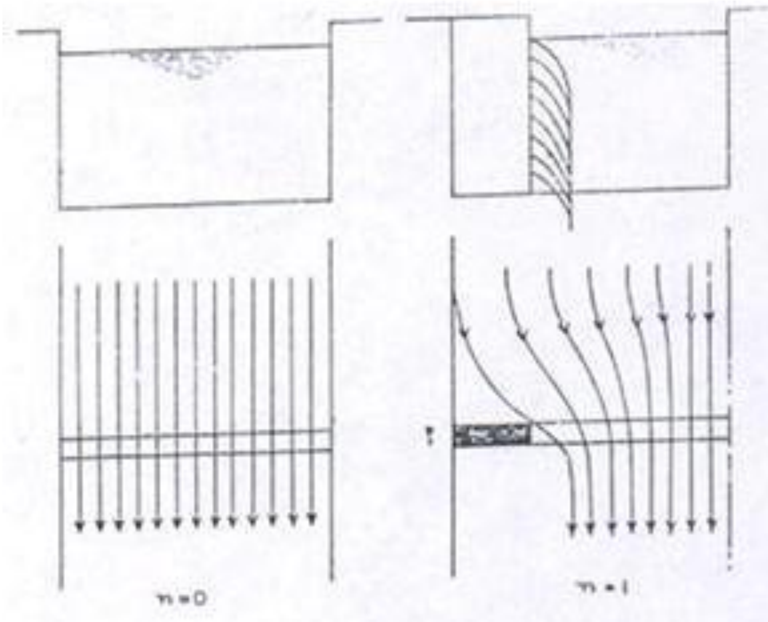
A rectangular weir is frequently used for measuring the rate of flow of water in channels. As such various investigators have conducted experiment with rectangular weirs and on the basis of the experimental results they have proposed a number of empirical formulae for computing the discharge over rectangular weirs. (i) Francis Formula

It is one of the most commonly used formula for computing the discharge over sharp or narrow crested weirs with or without end contraction. With the velocity approach taken into account

$$Q = 1.84\left[L - 0.1n\left(H + \frac{V_a^2}{2g}\right)\right]\left[\left(H + \frac{V_a^2}{2g}\right)^{3/2} - \left(\frac{V_a^2}{2g}\right)^{3/2}\right] \qquad \text{.........(3.3)}$$

$2g \qquad 2g \qquad 2g$

Fig.3.2 shows the typical cases of flow over rectangular weirs for which the different values of n as indicated thereon may be adopted.



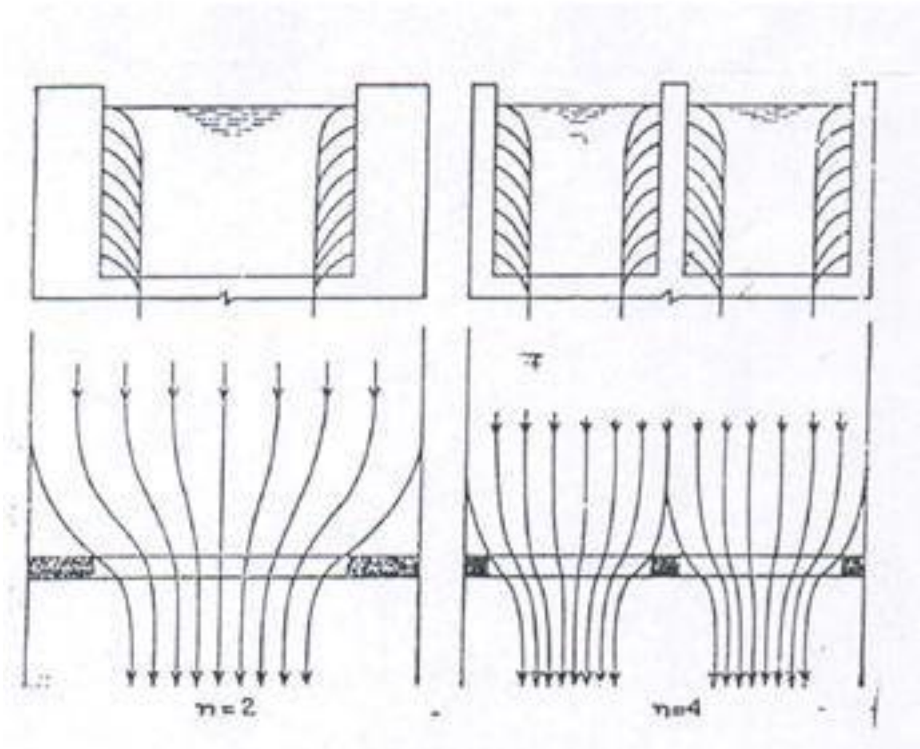


Fig.3.2 End contractions for rectangular weirs

(ii) Bazin's Formula

With the velocity of approach taken into account

$$Q = (0.405 + \frac{0.003}{H_1})\sqrt{2g}\,LH_1^{3/2} \qquad \text{...........(3.4)}$$

where $H_1 = H + 1.6(\frac{a}{2g})$

Example 3.1 A weir is 6m long and is situated centrally in a channel 9m wide. If the head above the sill is 25 cm, calculate the discharge. Use Francis formula. Neglecting the velocity of approach,

$$Q = 1.84\,(L - 0.1n\,H)\,H^{3/2}$$

$$Q = 1.84\,(6\text{-}0.1 \times 2 \times 0.25)\,0.25^{3/2}$$

$$= 1.365\ \mathrm{m^3/s}$$

Example 3.2 A stream approaching a water fall having a fall of 36m is gauged by a weir. The measured head over the weir is 27.5 cm and the length of the weir is 3m. The velocity of approach is 1.2 m/s. Determine the discharge. Use Bazin formula.

$$H_1 = H + 1.6\left(\frac{V_a^2}{2g}\right)$$

$$H_1 = 0.275 + 1.6\left(\frac{1.2^2}{2 \times 9.81}\right)$$

$$= 0.392 \text{ m}$$

$$Q = \left(0.405 + \frac{0.003}{H_1}\right)\sqrt{2g}\,LH_1^{3/2}$$

$$Q = \left(0.405 + \frac{0.003}{0.392}\right)\sqrt{2x9.81}\; x3x(0.392)^{3/2}$$

$$= 1.346 \text{ m}^3/\text{s}$$

3.5 Flow over a Triangular Weir or Triangular Notch

A triangular weir is an ordinary weir which is having a triangular or V-shaped opening or notch provided in its body, so that water is discharged through this opening only.

Generally a triangular weir or notch is preferred to a rectangular weir or notch for measuring the low discharges.

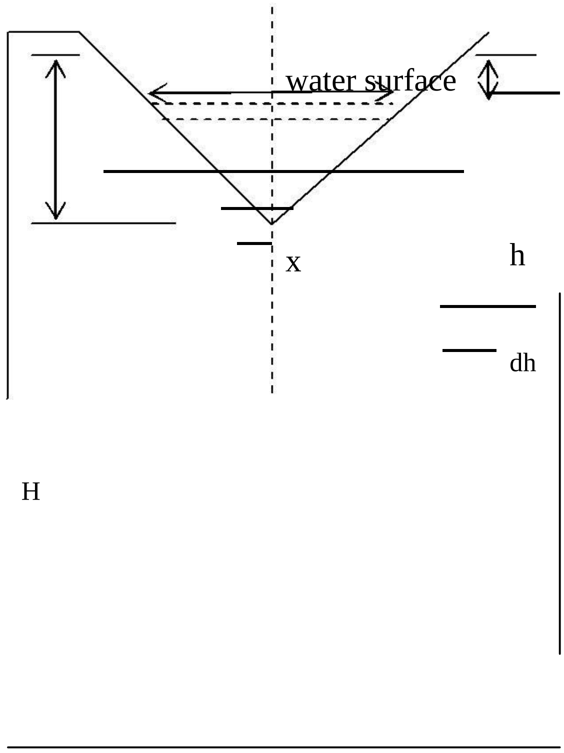


Fig.3.3   Flow over triangular weir or notch

If the approach velocity is neglected,

$$\text{Actual } Q = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \qquad \text{……….(3.5)}$$

If the vertex angle equals 90° then for a right- angled triangular weir or notch

Actual $Q = \frac{8}{15} C_d \sqrt{2g} H^{5/2}$ ……….(3.6)

3.6 Flow over a Trapezoidal Weir or Notch

As shown in Fig.3.4 a trapezoidal weir or notch is a combination of a rectangular and a triangular weir or notch. As such the discharge over such a weir or notch may be determined by adding the discharge over the two different types.

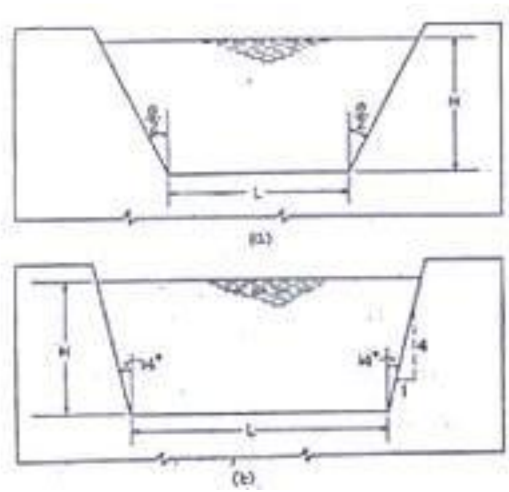


Fig.3.4 (a) Trapezoidal Weir (b) Cipolletti weir

$$Q = \frac{2}{3} C_d \sqrt{2g} L H^{3/2} + \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \qquad \text{…………(3.7)}$$

A Cipolletti weir is a particular type of trapezoidal weir, the sloping sides of which have an inclination of 1 horizontal to 4 vertical.

$$Q = \frac{2}{3} C_d \sqrt{2g} L H^{3/2} \qquad \text{………..(3.8)}$$

3.7 Broad Crested Weir

A weir having a wide crest is known as a broad crested weir. Such a weir differs from a narrow or sharp crested weir in this respect that in the case of a narrow or sharp crested weir as water flows over it the jet of water touches only the upstream edge and it flows clear of the downstream edge. On the other hand in the case of a broad crested weir as water flows over it the jet of water after touching the upstream edge flows over the surface of the crest as shown in Fig.3.5.

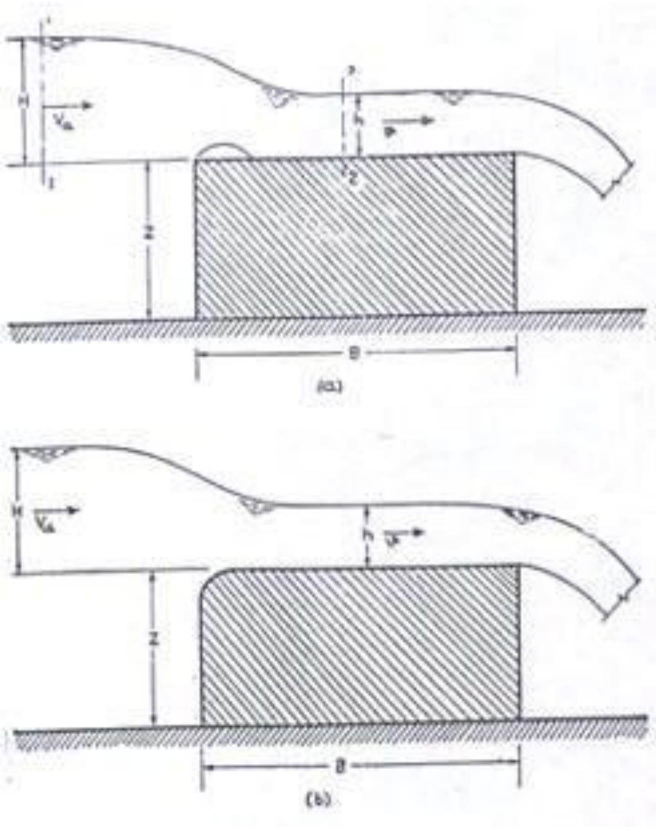


Fig.3.5 Broad crested weir (a) with sharp corner at the upstream end; (b) with round corner at the upstream end

Applying Bernoulli's equation section 1-1 just upstream of the weir and section 2-2 at the middle of the weir crest

$$H = h + V^2/2g$$

$$V = \sqrt{2g(H-h)}$$

Actual Q = C_d Lh $\sqrt{2g(H-h)}$ …………(3.9)

B < 0.5 H narrow crested weir

B > 0.5 H broad crested weir

3.8 Submerged Weir

When the water level on the downstream of the weir is above the crest of the weir then the weir is said to be a submerged weir. During floods often the weirs constructed across rivers become submerged. Submerged weirs have larger discharging capacity as compared with freely discharging weirs, thereby indicating that during floods when river carries huge quantity of water, the flow adjusts itself by setting the weir in the state of submergence, so that the discharge over the weir is increased and the flood water is quickly discharged to the downstream side.

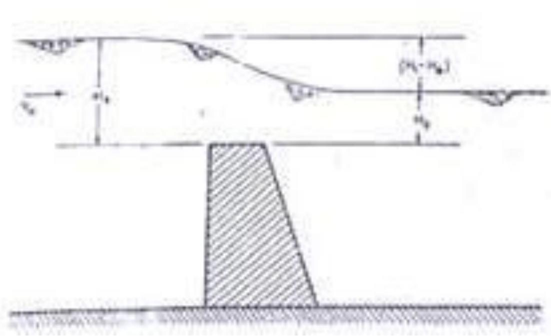


Fig.3.6 Submerged weir

Discharge through the free portion $Q_1 = \frac{2}{3} C_d L \sqrt{2g} (H_1 - H_2)^{3/2}$ ……….(3.10)

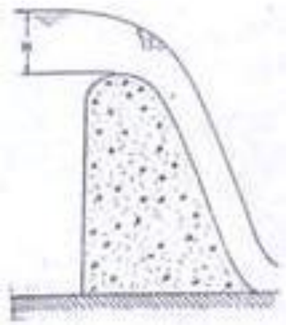
Discharge through drowned portion $Q_2 = C_d L H_2 \sqrt{2g (H_1 - H_2)}$ ………. (3.11)

Total discharge over a submerged weir $Q = Q_1 +$
Q_2 C_d is assumed to be constant throughout

3.9 Ogee Weir/Ogee Spillway

A spillway is a portion of a dam over which the excess water which cannot be stored in the reservoir formed on the upstream of dam, flows to the downstream side. The spillway is formed by filling the space between the sharp crested weir and the lower nappe with concrete or masonry as shown in Fig.3.7. Such a spillway is known as Ogee spillway. The main advantage of providing such a shape for the spillway is that the flowing sheet of water remains in contact with the surface of the spillway, thereby preventing the negative pressure being developed on the downstream side.

Fig.3.7



Ogee Spillway

$$Q = C\,L\,H^{3/2} \qquad \text{…………(3.12)}$$

where C is the coefficient of the spillway, L is the length of the spillway and H is the head above the crest of the spillway.

Example 3.3 A submerged weir 0.81m high stands clear across a channel having vertical sides and a width of 3.15m. The depth of water in channel of approach is 1.26 m and downstream side from the weir the depth of water is 0.93 m. Determine the discharge. Assume C_d =0.7

$L = 3.15$ m; $H_1 = 0.45$ m ; $H_2 = 0.12$ m

$$Q_1 = \frac{2}{3} C_d L \sqrt{2g} (H_1 - H_2)^{3/2}$$

$= 1.23$ m^3/s

$$Q_2 = C_d L H_2 \sqrt{2g(H_1 - H_2)}$$

$= 0.67$ m^3/s

$Q = Q_1 + Q_2$

$= 1.90$ m^3/s

CHAPTER 4

UNIFORM FLOW THROUGH OPEN CHANNELS

4.1 Introduction

An open channel is a passage, through which the water flows under the force of gravity i.e. under atmospheric pressure. Or in other words, when the free surface of the flowing water is in contact with the atmosphere, as in the case of a canal, a sewer of an aqueduct the flow is said to be through an open channel. A channel may be covered of open at the top.

The flow of water, in an open channel, is not due to any pressure as in the case of pipe flow. But it is due to the slope of the bed of the channel. Thus during the construction of a channel, a uniform slope in its bed is provided to maintain the flow of water.

4.2 Chezy's Formula for Discharge through an Open Channel

Consider an open channel of uniform cross section and bed slope.

Discharge, $Q = A \text{ x } v = AC\sqrt{\ } \text{mi}$ ……..(5.1)

where

C= Chezy's constant

V = velocity

A = area of flow

P = wetted perimeter of the cross section

i = uniform slope in bed

m = hydraulic mean depth = A/P

Example 4.1 A rectangular channel is 4 metres deep and 6 metres wide. Find the discharge through channel, when it runs full. Take slope of the bed as 1 in 1000 and Chezy's constant as 50.

Area $A = 4 \times 6 = 24\ m^2$

Perimeter $P = b+2d = 6+2 \times 4 = 14$ m

Hydraulic mean depth m = A/P

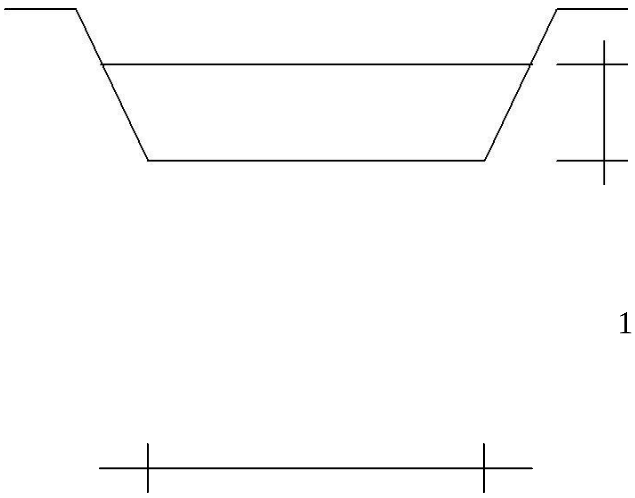
= 24/14

= 1.71 m

= 24 x 50 1.71 x0.001 = 49.62 m^3/sec

= 49620 liters/sec

Example 4.2 Water is flowing at the rate of 8.5 cubic meters per second in an earthen trapezoidal channel with bed width 9 meters, water depth 1.2 meter and side slope 1:2. Calculate the bed slope, if the value of C in the Chezy's formula be 49.5.



1.2 m

9 m

$A = 1/2(9+10.2) \times 1.2 = 11.52\ m^2$

$P = 9 + 2\sqrt{1.2^2 + 0.6^2}$

$= 11.68\ m$

$m = A/P$

$= 11.52/11.68$

$= 0.986$

$Q = AC\sqrt{mi}$

$8.5 = 11.52 \times 49.5 \times \sqrt{0.986 \times i}$

$i = 1/4440$

4.3 Discharge through a Circular Channel

Consider a circular channel, through which the water is flowing as shown in Fig.4.1.

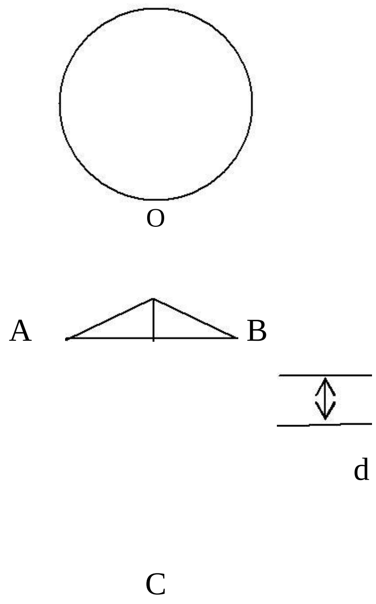


Fig.4.1 Circular Channel

Let r =radius of the circular channel

d = Depth of water in the channel

P = wetted perimeter ACB of the channel = $2r\theta$

A = Area of the section through which the water is flowing

$$A = r^2\theta - \frac{r^2 Sin 2\theta}{2} = r^2\left(\theta - \frac{Sin 2\theta}{2}\right) \qquad \text{…….(4.2)}$$

Example 4.3 A circular pipe of 1 metre radius is laid at an inclination of 5° with the horizontal. Calculate the discharge through the pipe, if the depth of water in the pipe is 0.75 meter. Take C = 65.

Inclination, $i = \tan 5° = 0.0875$

$\cos\theta = OD/OB = (1-0.75)/1 = 0.25$

$\theta = 75° 30' = 75.5 \times \pi/180 = 1.318$ rad

$P = 2\,r\,\theta = 2 \times 1 \times 1.318 = 2.636$ m

$$A = r^2\left(\theta - \frac{Sin\,2\theta}{2}\right)$$

$$= 1^2\left(1.318 - \frac{Sin\,2x75^o30'}{2}\right)$$

$= 1.076\ m^2$

Hydraulic mean depth m= A/P = 1.076/2.636 = 0.408 m

$$Q = AC\sqrt{mi}$$

$$= 1.076 \times 65 \sqrt{0.408 \times 0.0875}\ \text{m}^3/\text{sec}$$

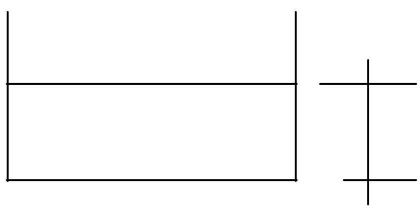
$$= 13.24\ \text{m}^3/\text{sec}$$

4.4 Channels of Most Economical Cross-sections

A channel which gives maximum discharge for a given cross-sectional area and bed slope is called a *channel of most economical cross-section*. Or in other words, it is a channel, which involves lesser excavation for a designed amount of discharge. A channel of most economical cross-section is, sometimes, also defined as a channel, which has a minimum wetted perimeter; so that there is a minimum resistance to flow and thus resulting in a maximum discharge. From the above definitions, it is obvious that while deriving the condition for a channel of most economical cross-section the cross-sectional area is assumed to be constant, and a relation between depth and breath of the section is found out, to gives maximum discharge.

4.5 Condition for Maximum Discharge through a Channel of Rectangular Section

Consider a channel of rectangular section as shown in Fig.4.2.



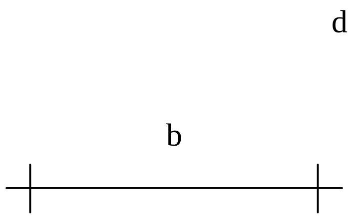


Fig.4.2 A rectangular channel

For the maximum discharge or maximum velocity, the following two conditions are satisfied.

$b = 2\,d$

$m = d/2$

Example 4.4 A rectangular channel has a cross-section of 50 square metres. Determine the discharge through the most economical section, if bed slope is 1 in 1000 . Take C = 52.5

Bed slope $i = 1/1000 = 0.001$

For the most economical rectangular section, $B = 2d$

$\therefore$ Area , $A = b \times d = 2\,d \times d = 2\,d^2$

or $50 = 2\,d^2$

$\therefore$ $d = 5$ m

and $b = 2d = 2 \times 5 = 10$ m

For the most economical rectangular section, hydraulic mean depth $m = d/2 = 5/2 = 2.5$ m

$$Q = AC\sqrt{mi}$$

$$= 50 \times 52.5\sqrt{2.5 \times 0.001}\ \text{m}^3/\text{sec}$$

$$= 131.25\ \text{m}^3/\text{sec}$$

CHAPTER 5

TURBINES

5.1 Introduction

Hydraulic turbines are the machines which use the energy of water and convert it to mechanical energy. The mechanical energy developed by a turbine is used in running an electric generator which is directly coupled to the shaft of the turbine. The electric generator thus develops electric power, which is known as hydro-electric power.

5.2 Elements of Hydraulic Power Plants

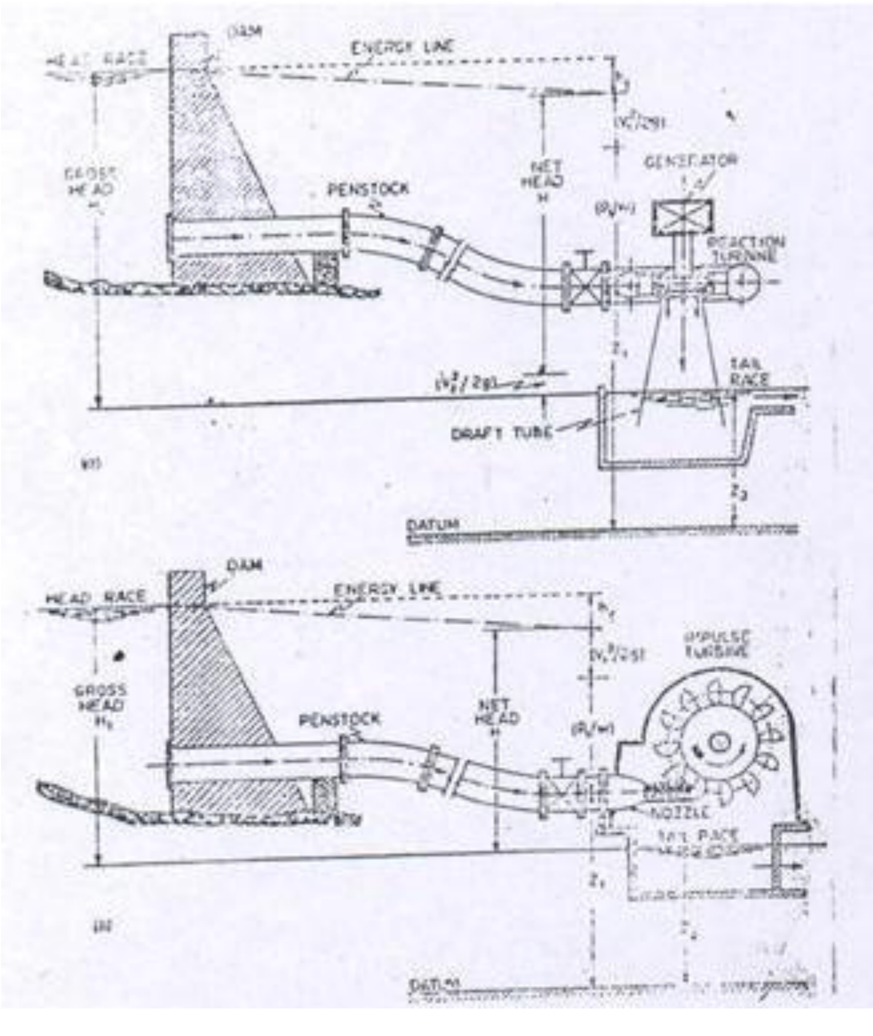


Fig.5.1   General Layout of a Hydraulic Power Plant

Fig.5.1 shows a general layout of hydraulic power plant, in which an artificial storage reservoir formed by constructing a dam has been shown.

5.3 General Classification of Turbines

Turbines are hydraulic machines that convert energy into rotating mechanical energy which in turn generators to produce electrical energy. Originally developed from the water wheels, hydraulic turbines are the prime mortars of importance in modern water power development. According to their hydraulic action, turbines are broadly divided into two classes.

(1) **Impulse Turbine**: Impulse turbines are more efficient for high heads. At the inlet to the turbine runner, pressure head can be completely converted into kinetic head in the form of a jet of water issuing from one or more nozzles. The free jet will be at atmospheric pressure before as well as after striking the vanes. The turbines are regulated by nozzles which may be a simple straight flow type or a deflector type. The impulse turbines are commonly represented by *Pelton Wheels*. *Turgo* turbine is also an impulse turbine but with different buckets, when compared with pelton. *Turgo* and *cross flow* turbines are relatively new developments in this class.

The main advantages of these turbines are:

- They can be easily adopted to power variation with almost constant efficiency.
- The penstock overpressure and the runner overspeed control are easier.
- The turbine enables an easier maintenance.
- Due to the jet the manufacturer of these turbines impose a better solid particle control, conducting, consequently, to a lower abrasion effect.

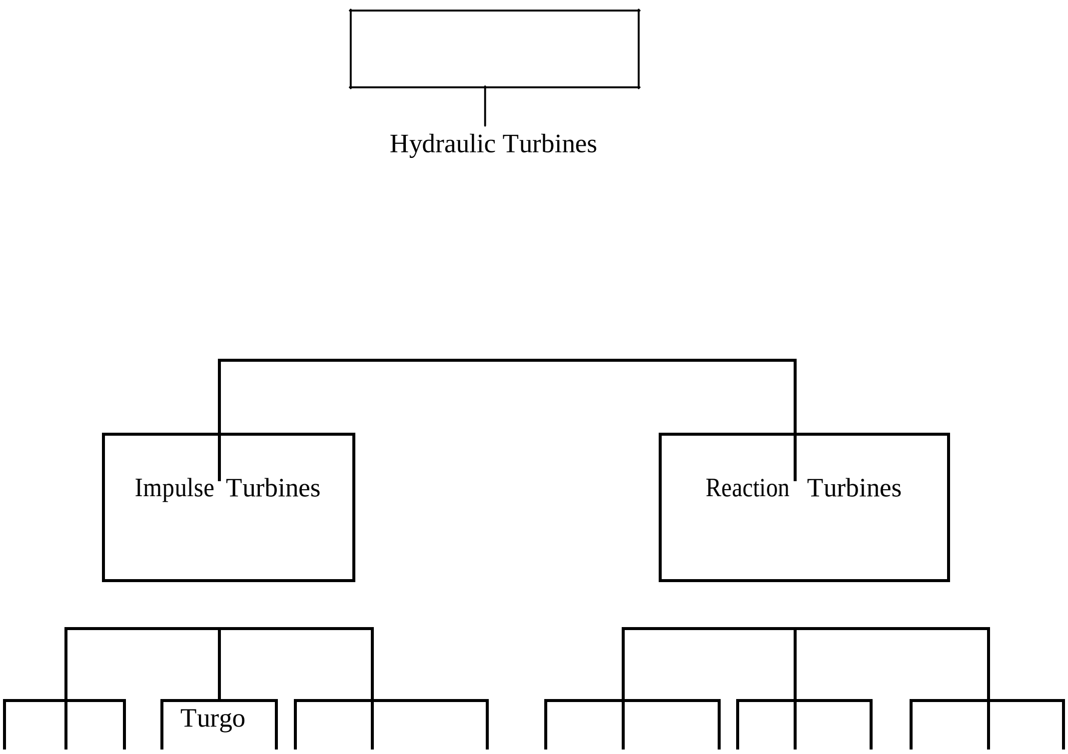
(2) **Reaction Turbine**: A turbine can be made to rotate under the action of water flowing under pressure through the runner. In such turbines the penstocks, the inlet passage to the runner, passage between the runner vanes, all form a continuous passage for the flow under a pressure which continuously decreases from inlet to outlet. The turbine runner directly converts both kinetic energy as well as the pressure energy into mechanical energy. Reaction turbines are represented in modern practice by two principal types: the Francis turbine where the flow is directed radial to the runner axis and the Propeller type

where the flow is axial to the runner axis. Propeller turbines may be fixed blade or adjustable blade types. Kaplan turbine has adjustable blades.

The main advantages of these turbines are:

- It needs lesser installation space.
- It provides a greater net head and a better protection against downstream high flood levels.
- It can have greater runner speed.
- It can attain greater efficiencies for high power values.

In order to distinguish different turbines, the hydraulically salient features like pressure, head, flow direction and magnitude, speed and power etc. The general classification of hydraulic turbines is illustrated in Fig.5.2.



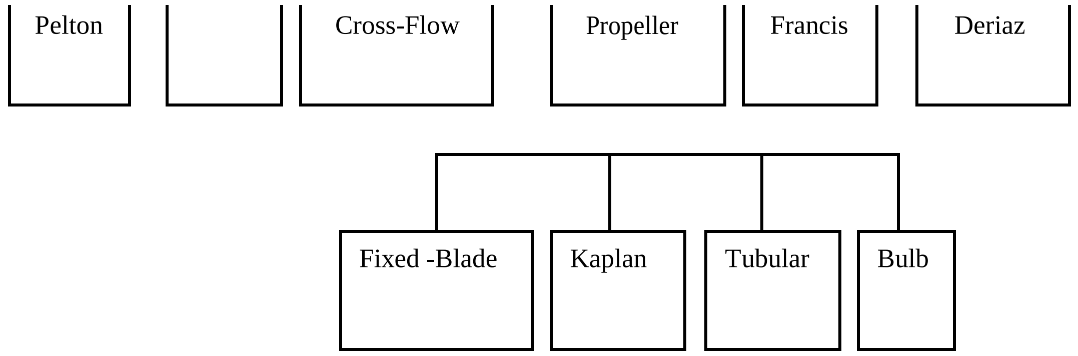


Fig.5.2 General Classification of Turbines

5.4 Number of Units

It is normally cost effective to have a minimum number of units at a given small hydropower installation. Multiple units may, however, be necessary from the operational point of view so that even one unit breaks down or is in the routine maintenance, the power generation can be achieved to a certain extent. The efficiency curves of turbines show that the efficiency of power generation from hydraulic turbines considerably decreases at low flow ratios or power ratios. In multiple units, it is possible to maintain the higher efficiency even in low flows and the low loads by running a certain number of the units at a time depending upon the available discharge and the load demand. Multiple units thus, make the most effective use of water where the flow as well as the load variations are significant.

5.5 Limits of Use of Turbine Types

The selection of best turbines for any particular small hydropower site depends on the site characteristics, the dominant beings the head and the available flow. There are some limits on the range of these parameters in the selection of turbines. Each turbine type is best suited to a certain range of pressure head and the flow rate. For instance, Pelton wheels operate with low flows discharged under great pressures where as Propeller turbines are effective in high flows under low heads. Francis turbines fall in the medium category covering a wide range of different heads and discharges.

The common practice of SHP systems is to develop standard unit sizes of equipment that will operate over a range of heads and flows. Either charts or nomographs are used to select appropriate units for site specific application. One such chart showing the head-flow range of normal SHP schemes applicable to each type of turbine is given in Fig5.3. The graph also indicates the approximate power generation for each combination of the head and the discharge applicable to SHP schemes.

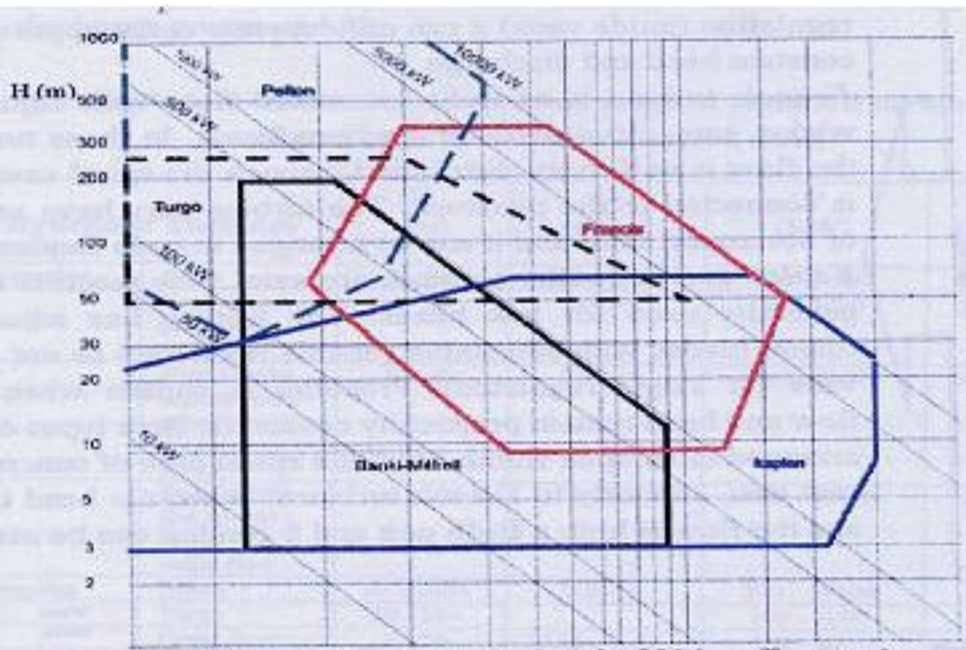


Fig.5.3　Head-Flow Ranges for Different Turbines

5.6 Pelton Wheel

Pelton wheel is well suited for operating under high heads. A pelton turbine has one or more nozzles discharging jets of water which strike a series of buckets mounted on the periphery of a circular disc. The *runner* consists of a circular disc with a number of buckets evenly spaced round its periphery. The *buckets* have a shape of a double semi-ellipsoidal cups. The pelton bucket is designed to deflect the jet back through 165° which is the maximum angle possible without the return jet interfering with the next bucket.

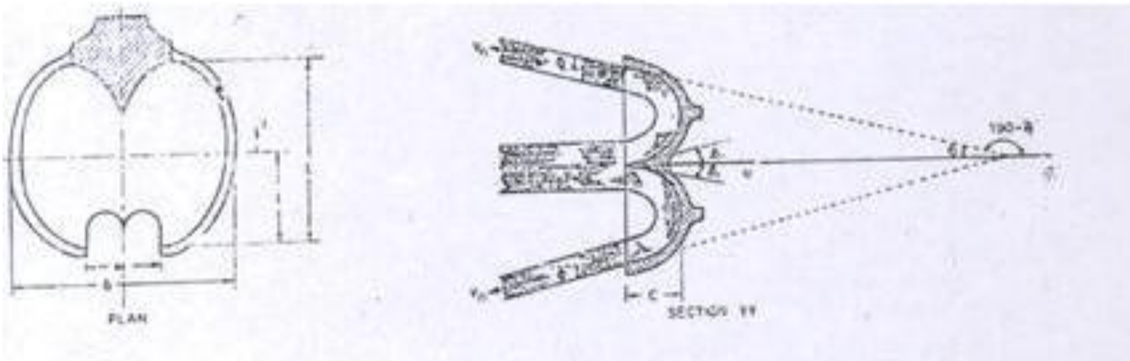


Fig.5.4 Pelton bucket

General arrangement of a pelton wheel is shown in the Fig.5.5. For SHP schemes, Pelton wheels are easier to fabricate and are relatively cheaper. The turbines are in general, not subjected to the cavitation effect. The turbines have access to working parts so that the maintenance or repairs can be effected in a shorter time.

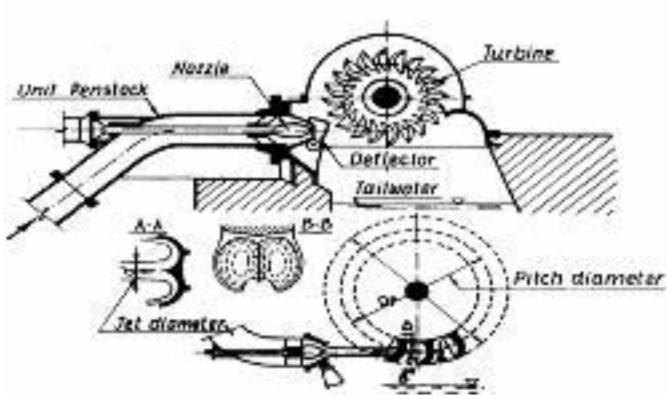


Fig.5.5 General Arrangement of a Pelton Wheel

Traditionally, micro hydro pelton wheels were always single jet because of the complexity and the cost of flow control governing of more than one jet. Advantages of multi-jet:

-Higher rotational speed

-Smaller runner

-Less chance of blockage

Disadvantages of multi-jet:

-Possibility of jet interference on incorrectly designed systems -Complexity of manifolds

5.7 Francis Turbine

Francis turbine is a mixed flow type, in which water enters the runner radially at its outer periphery and leaves axially at its center. Fig.5.6 illustrates the Francis turbine. The runner blades are profiled in a complex manner and the casing is scrolled to distribute water around the entire perimeter of the runner. The water from the penstock enters a *scroll case* which completely surrounds the runner. The purpose of the scroll case is to provide an even distribution of water around the circumference of the turbine runner, maintaining an approximately constant velocity for the water so distributed. The function of *guide vane* is to regulate the quantity of water supplied to the runner and to direct water on to the runner at an angle appropriate design. A *draft tube* is a pipe or passage of gradually increasing cross sectional area which connects the runner exit to the tail race.

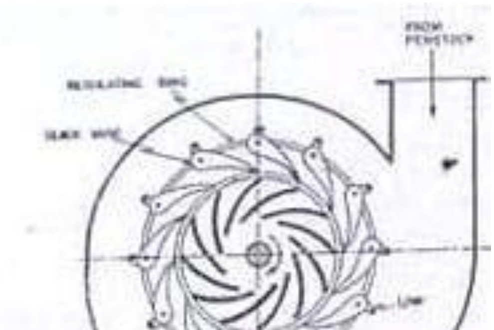


Fig.5.6 Francis Turbine

5.8 Kaplan Turbine

It is an axial flow turbine which is suitable for relatively low heads. From Fig.5.7, it will be seen that the main components of Kaplan turbine such as scroll casing, guide vanes, and the draft tube are similar to those of a Francis turbine.

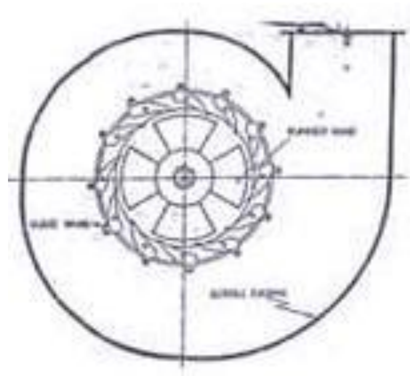


Fig.5.7 Kaplan Turbine

5.9 Specific Speed

The specific speed of any turbine is the speed in r.p.m of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 metric horsepower when working under unit head.

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} \qquad \text{..............(5.1)}$$

where Ns = specific speed

P = power in HP

5.10 Characteristic Curves

The turbines are generally designed to work at particular values of H,Q,P,N and η_o which are known as the designed conditions. It is essential to determine exact behaviour of the turbines under the varying conditions by carrying out tests either on the actual turbines or on their small scale models. The results of these tests are usually graphically represented and the resulting curves are known as characteristic curves.

-constant head characteristic curves

-constant speed characteristic curves

-constant efficiency characteristic curves

In order to obtain constant head characteristics curves the tests are performed on the turbine by maintaining a constant head and a constant gate opening and the speed is varied by changing the load on the turbine. A series of values of N are thus obtained and corresponding to each value of N, discharge Q and the output power P are measured. A series of such tests are performed by varying the gate opening, the head being maintained constant at the previous value. From the data of the tests the values of Q_u, P_u, n_u and η_o are computed for each gate opening. Then with N_u as abscissa the values of Q_u, P_u and η_o for each gate opening are plotted. The curves thus obtained for pelton wheel and the reaction turbines for four different gate openings are shown in Fig.5.8.

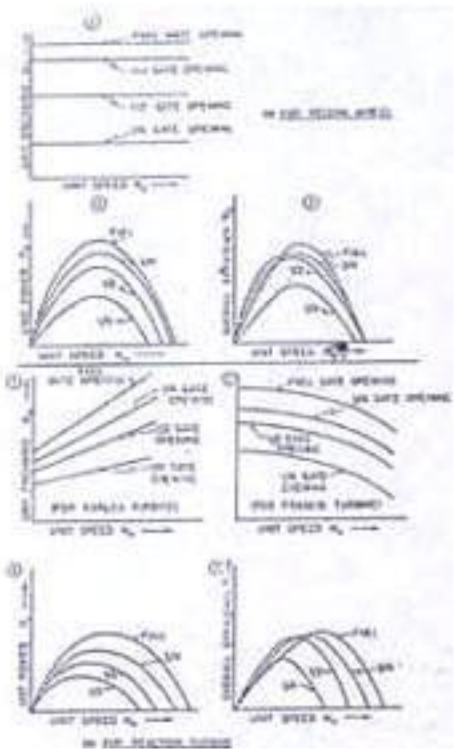


Fig.5.8 Constant head characteristics for Pelton wheel and reaction turbines

5.11 Cavitation in turbines

When the pressure in any part of the turbine reaches the vapour pressure of the flowing water, it boils and small bubbles of vapour form in large numbers. These bubbles are carried along by the flow, and on reaching the high pressure zones these bubbles suddenly collapse as the vapour condenses to liquid again. The alternate formation and collapse of vapour bubbles may cause severe damage to the surface which ultimately fails to fatigue and the surface becomes badly scored and pitted. This phenomenon is known as cavitation.

In order to determine whether cavitation will occur in any portion of the turbine, D.Thomas has developed a dimensionless parameter called Thomas'cavitation factor σ which is expressed as

$$\sigma = \frac{H_a - H_v - H_s}{H}$$

For Propeller turbines:

$$\sigma_c = 0.28 + [1 (N_s)^3]$$

where

H_a = atmospheric pressure head

H_v = vapour pressure head

H_s = suction pressure head

For Francis turbines:

Critical cavitation factor $\sigma_c = 0.625\,(N_s/444)^2$

$\overline{7.5}\ \overline{444}$

……………(5.2)

………..(5.3)

………..(5.4)

Example 5.1 Estimate the maximum height of straight conical draft tube of a 18000 h.p. Francis turbine running at 150 r.p.m under a net head of 27 m. The turbine is installed at a station where the effective atmospheric pressure is 10.6 m of water. The draft tube must sink at least 0.77 m below the tail race.

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

$= 327$

$\sigma_c = 0.625\,(N_s/444)^2$

$= 0.339$

Cavitation factor $\sigma = \dfrac{H_a - H_v - H_s}{H}$

Ha -Hv = 10.6 m, H =27 m

$$0.339 = \frac{10.6 - H_s}{27}$$

$$H_s = 1.45 \text{ m}$$

Max length of the draft tube = 1.45 + 0.77 = 2.22 m

5.12 Governing of Turbines

All the modern turbines are directly coupled to the electric generators. The generators are always required to run at constant speed irrespective of the variations in the load. This constant speed N of the generator is given by the expression

$$N = \frac{60f}{p} \qquad \text{…………..(5.5)}$$

where f = frequency (usually 50)

p = numbers of pairs of poles

5.13 Water Hammer

A gate or valve at the end of the penstock pipe controls the discharge to the turbine. As soon as this governor regulated gate opening is altered, the pipe flow has to be adjusted to the new magnitude of flow. In doing so, there are rapid pressure oscillations in the pipe, often accompanied by a hammering like sound. Hence this phenomenon is called as water hammer.

5.14 Jet Speed

The velocity of flow of the jet depends upon the total net head H at the base of the nozzle and is given by the nozzle equation:

$$v = C_V \cdot \sqrt{2gH} \qquad \text{…………(5.6)}$$

where the discharge coefficient velocity of the nozzle is taken as 0.95.

5.15 Bucket speed

$$V = \frac{\pi D N}{60} \qquad \ldots\ldots\ldots\ldots(5.7)$$

The bucket speed should be half of the jet speed. In practice, losses in the turbine cause the maximum efficiency to occur at slightly less than a half, typically 0.46.

V =0.46 v

5.16 Design of Pelton Wheel

Runner diameter:

Runner diameter can be found out from the rpm equation.

$$D = \frac{38 \cdot \sqrt{H}}{N} \qquad \ldots\ldots\ldots(5.8)$$

where N = runner speed(rpm)

H = net head

Nozzle diameter:

The nozzle diameter is given by the nozzle equation:

$$d = 0.54 \cdot \frac{Q^{0.5}}{H^{0.25}} \cdot \frac{1}{\sqrt{n_{jet}}} \quad \text{……….(5.9)}$$

Jet ratio:

Jet ratio D/d is a size parameter for the turbine. It has a value in a range of 10 to 24. For the high efficiency Pelton wheel design, the ratio of the runner diameter to the nozzle should be more than 9.

Number of buckets:

The number of buckets required for the efficient operation of the Pelton turbine is calculated as:

$$N_{buc} = 0.5 \cdot \frac{D}{d} + 15 \quad \text{…………..(5.10)}$$

In practice, the selection and the detail design of the turbine units are carried out by the manufactures based on the model performances.

Example 5.2 Powerhouse is equipped with a vertical shaft pelton turbine. The generator is provided with 6 pairs of poles. Design discharge is 1.4 m^3/s and net head is 425 m. The turbine will provide 6500 hp. Take coefficient of nozzle as 0.95. Determine

(a) the specific speed

(b) velocity of jet

(c) jet diameter

(d) pitch circle diameter of the wheel

(e) number of buckets

(a) $N = \frac{60 f}{p}$

= 60 x 50/6

= 500 rpm

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

$$= \frac{500\sqrt{6500}}{425^{5/4}}$$

= 20.9

Use single jet pelton turbine

(b) velocity of jet

$$v = C_v \cdot \sqrt{2gH}$$

$$= 0.95 \cdot \sqrt{2x9.81x425}$$

$$= 86.75 \text{ m/s}$$

(c) jet diameter

$$d = 0.54 \cdot \frac{Q^{0.5}}{H^{0.25}} \cdot \frac{1}{\sqrt{n_{jet}}}$$

$$= 0.54 \cdot \frac{1.4^{0.5}}{425^{0.25}} \cdot \frac{1}{\sqrt{1}}$$

$$= 0.14 \text{ m} = 14 \text{ cm}$$

(d) pitch circle diameter

$$D = \frac{38 \cdot \sqrt{H}}{N}$$

$= 1.57$ m

(e) Number of buckets

$$N = 0.5 \cdot \frac{D}{} + 15$$

$$N_{buc} = 0.5 \cdot \frac{1.57}{} + 15\ 0.14$$

$= 20.6$

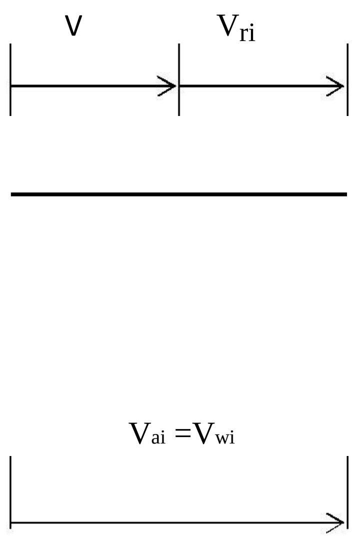
$= 21$

5.17 Work done of Pelton Wheel

In turbines, the water flows on to the runner, which itself is rotating with a certain speed. The water flows over the runner and leaves the runner at its outlet point. We can speak of absolute velocity of water before it flows in the runner, the relative velocity of water w.r.t the runner and again the absolute velocity of water after it has left the runner. In order to ascertain the relationship between these velocities, the velocity vector diagram prove to be very useful.

Fig.5.9 shows the velocity triangles at the tips of the bucket of a pelton wheel. At the outlet tip velocity triangles are different depending upon the magnitude of u corresponding to which it is slow, medium or fast runner.

Inlet velocity diagram



Outlet velocity diagram

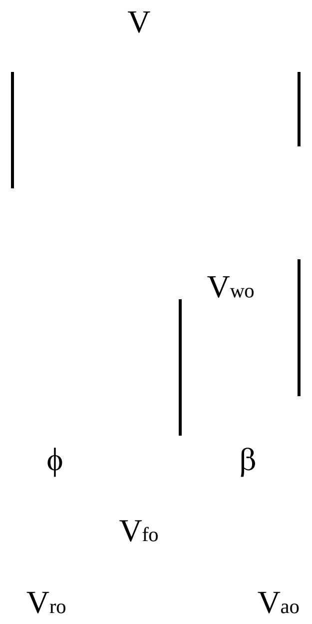


Fig.5.9 Velocity triangles

V = bucket velocity

V_{ai} = absolute velocity of jet at inlet tip $V_{ai} = C_v \sqrt{2gH_1}$

V_{ao} = absolute velocity of jet at outlet tip

V_{ri} = relative velocity of jet at inlet = V_{ai}-V

V_{ro} = relative velocity of jet at outlet = k.V_{ri}

V_{wi} = velocity of whirl at inlet =V_{ai}

V_{wo} = velocity of whirl at outlet = V-V_{ro}Cos ϕ

V_{fo} = velocity of flow at outlet

Mass/sec

$$m = \rho\, Q = \rho\, a\, V_{ai} = \rho\, \pi/4\, d^2\, V_{ai} \qquad \text{………(5.11)}$$

Workdone on the bucket/sec (power developed by turbine)

$$P = m\,(V_{wi} - V_{wo})\, V \qquad \text{………(5.12)}$$

Maximum hydraulic efficiency

$$\eta_{h\,max} = \frac{1}{2}(1 + k\,Cos\,\phi) \qquad \text{………(5.13)}$$

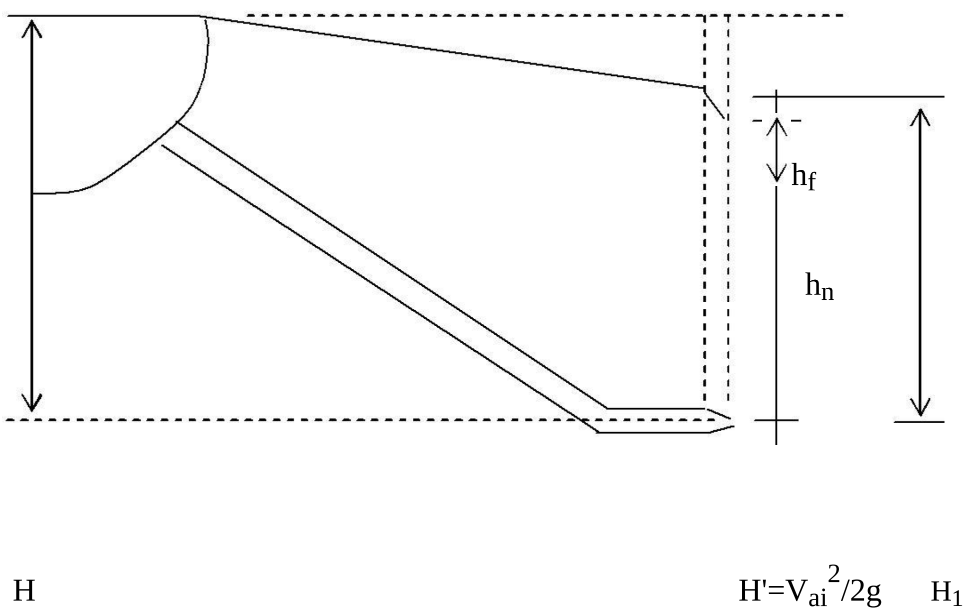
The hydraulic efficiency is maximum when the bucket speed is equal to half of the velocity of jet.

Example 5.3 The head available at entrance to the nozzle supplying a pelton wheel is 300 m and the coefficient of velocity for the nozzle is 0.98. The wheel diameter is 1.8 m and the nozzle diameter is 125 mm. The buckets deflect the jet through 165°. Assuming the relative velocity of the jet is reduced by 16%, calculate the theoretical speed in rev per min for the maximum hydraulic efficiency. What is the hydraulic efficiency when running at this speed, and what is the power developed?

Deflection angle =165° = (180- ϕ)

ϕ = 15°

k = 0.84



For max hydraulic efficiency $V/V_{ai} = 0.5$

$V_{ai} = C_v\sqrt{2gH_1}$

$= 75$ m/s

$V = V_{ai}/2 = 37.5$ m/s

$V = \pi DN/60$

$N = 60V/\pi D = 398$ rpm

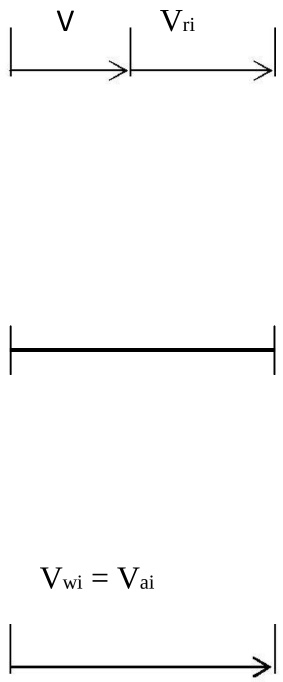
$\eta_{hmax} = 1/2\ (1 + k \cos \phi)$

$= 90.55\ \%$

mass/sec = $m = \rho Q = \rho\ \pi/4\ d^2\ V_{ai}$

$= 920$ kg/sec

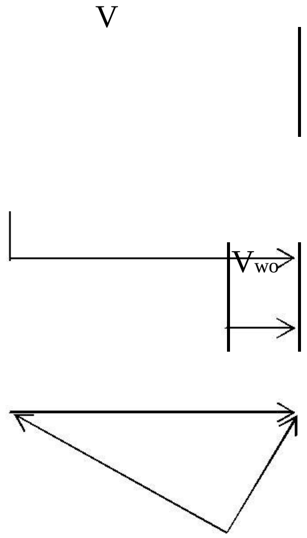
Inlet diagram:



From velocity diagram $V_{wi} = V_{ai}$ =75 m/s

$V_{ri} = V_{ai} - V$ =37.5 m/s

Outlet diagram:



$V_{wo} = V - V_{ro} \cos \phi$

$= V - k\, V_{ri} \cos \phi$

= 7 m/s

V_{ro} V_{ao}

Power = m V (V_{wi} -V_{wo})

= 234600 Watts

CHAPTER 6

CENTRIFUGAL PUMP

6.1 Introduction

Centrifugal pumps are classified as rotodynamic type of pumps in which dynamic pressure is developed which enables the lifting of liquids from a lower to a higher level. The basic principle on which a centrifugal works is that when a certain mass of liquid is made to rotate by an external force, it is thrown away from the central axis of rotation and a centrifugal head is impressed which enable it to rise to a higher level. Now, if more liquid is constantly made available at the centre of rotation, a continuous supply of liquid at a higher level may be ensured. Since in these pumps the lifting of the liquid is due to centrifugal action, these pumps are called 'centrifugal pumps'.

6.2 Advantages of centrifugal pumps over reciprocating pumps

The main advantage of a centrifugal pump is that its discharging capacity is very much greater than a reciprocating pump which can handle relatively small quantity of liquid only. A centrifugal pump can be operated at very high speeds without any danger of separation and cavitation . The maintenance cost of a centrifugal pump is low and only periodical check up is sufficient . But for a reciprocating pump the maintenance cost is high because the parts such as valves etc., may need frequent replacement.

6.3 Component Parts of a Centrifugal Pump

The main component parts of a centrifugal pump are:

-impeller

-casing

-suction pipe

-delivery pipe

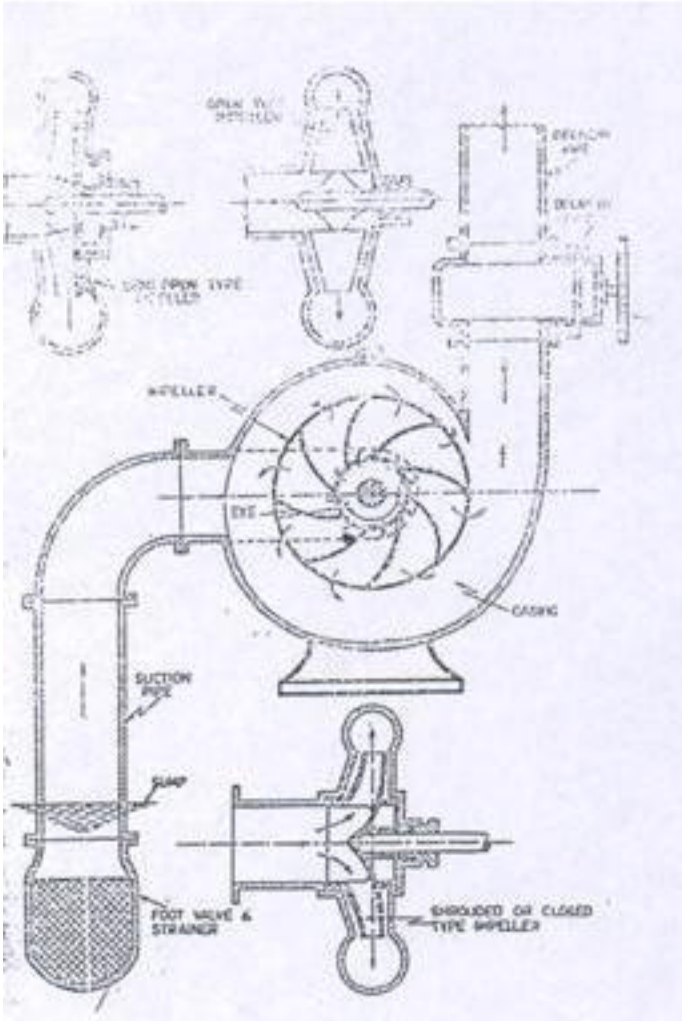

IMPELLER
EYE
CASING
SUCTION PIPE
SUMP
FOOT VALVE & STRAINER
SHROUDED OR CLOSED TYPE IMPELLER

Fig.6.1 Component part of a centrifugal pump

6.4 Workdone by the Impeller

The expression of the workdone by the impeller of a centrifugal pump on the liquid flowing through it may be derived in the same way as for a turbine. The liquid enters the impeller at its centre and leaves at its periphery. Fig.6.2 shows a portion of the impeller of a centrifugal pump with one vane and the velocity triangles at the inlet and outlet tips of the vane.

V is absolute velocity of liquid, u is tangential velocity of the impeller, V_r is relative velocity of liquid, V_f is velocity of flow of liquid, and V_w is velocity of whirl of the liquid at the entrance to the impeller. Similarly V_1,u_1,V_{r1},V_{f1} and V_{w1} represent their counterparts at the exit point of the impeller.

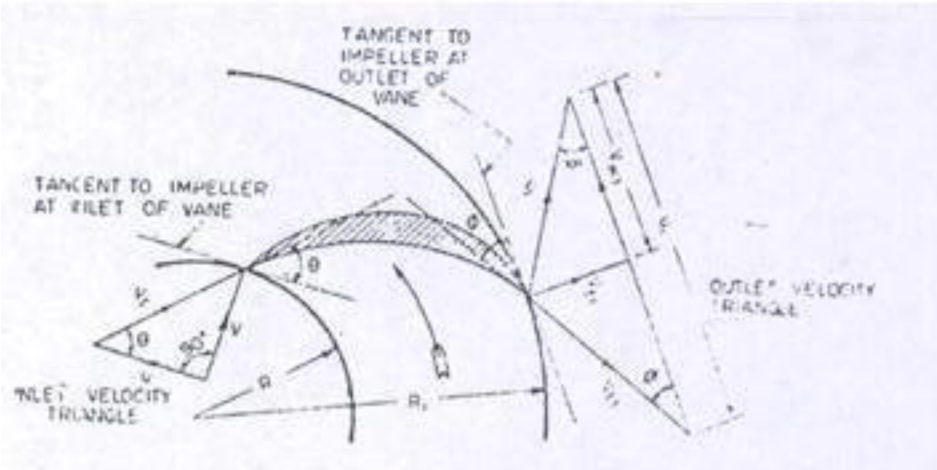


Fig.6.2 Velocity triangles for an impeller vane

θ = the impeller vane angle at the entrance

ϕ = the impeller vane angle at the outlet

α = the angle between the directions of the absolute velocity of entering liquid and the peripheral velocity of the impeller at the entrance

β = the angle between the absolute velocity of leaving liquid and the peripheral velocity of the impeller at the exit point

Work done per second by the impeller on the liquid may be written as

$$\text{Work done} = \frac{W}{g}\left(V_{w1}\, u_1 - V_w\, u\right) \qquad \text{-------------(6.1)}$$

where W kg of liquid per second passes through the impeller. Since the liquid enters the impeller radially α = 90 and hence V_w = 0. Thus equation (6.1) becomes

$$\text{Work done} = \frac{W}{g}\left(V_{w1} u_1\right) \qquad \text{-------------(6.2)}$$

6.5 Head of a Pump

The head of a centrifugal pump may be expressed in the following two ways:

(a) Static head

(b) Manometric head (or total head or gross head)

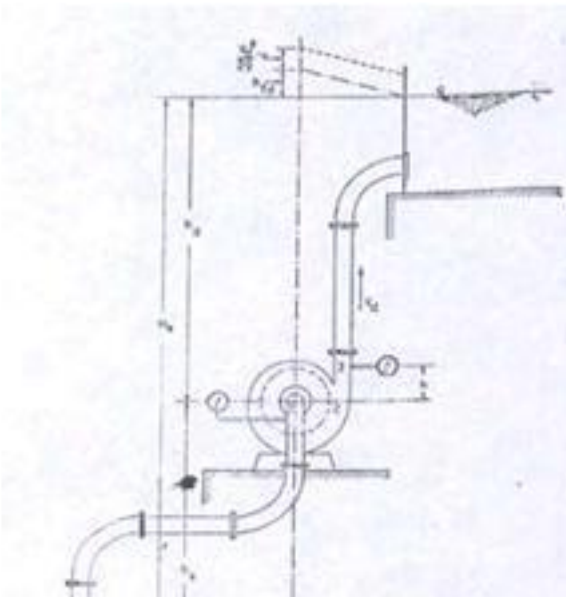


Fig.6.3 Head on a centrifugal pump

(a) Static Head

Static head is the vertical distance between the liquid surfaces in the pump and the tank to which the liquid is delivered by the pump.

Static head (or lift) $H_s = h_s + h_d$ ………..(6.3)

where h_s = static suction lift

h_d = static delivery lift

(b) Manometric Head

Manometric head is the total head that must be produced by the pump to satisfy external requirements. If there are no energy losses in the impeller and the casing of the pump, then the manometric head H_m will be equal to the energy given to the liquid by the impeller, i.e $\frac{V_{\omega 1} u_1}{g}$. But if losses occur in the pump then

$$H_m = \frac{V_{\omega 1} u_1}{g} - \textit{losses of head in the pump} \qquad \text{……….(6.4)}$$

Applying Bernoulli’s equation between the points, *O* at the liquid surface in the pump and 1 in the suction pipe just at the inlet to the pump (i.e., at the centre line of the pump), the following expression is obtained if the liquid surface in the sump is taken as datum.

$$0 = \frac{p_s}{\omega} + \frac{V_s^2}{2g} + h_s + h_{fs}$$

$$\frac{p_s}{\omega} = -\left[\frac{V_s^2}{2g} + h_s + h_{fs}\right] \qquad \text{……….(6.5)}$$

where p_s is the pressure at point l ; V_s is the velocity of flow in the suction pipe ; h_s is the suction lift and h_{fs} is the head loss in the suction pipe which includes the head loss due to friction and the other minor losses. It may however be pointed out that if the pump is situated below the level of the liquid surface in the sump, h_s will be negative. Equation (6.5) indicates that at the inlet to the pump there is always a suction or vacuum pressure developed which will be recorded by the vacuum gauge provided at this point as shown in Fig.6.3. The head expressed by equation (6.5) is called the *suction head* of the pump.

Also, applying Bernoulli's equation between points 1 and 2, which is just at the outlet of the impeller and is assumed to be at the same level as point 1, then since the impeller imparts a head equal to $(V_{w1}u_1/g)$ to the liquid the following expression is obtained:

$$\frac{p_s}{\omega} + \frac{V_s^2}{2g} + \frac{V_{\omega 1} u_1}{g} = \frac{p_2}{\omega} + \frac{V_1^2}{2g} + h_{Li} \qquad \text{……….(6.6)}$$

where p_2 is the pressure and V_1 is the absolute velocity of the liquid leaving the impeller and h_{Li} is the loss of head in the impeller.

6.6 Specific Speed of Centrifugal Pumps

In order to compare the performance of different pumps, it is necessary to have some term which will be common to all centrifugal pumps. The term used for this purpose is the specific speed. The specific speed of a centrifugal pump is the speed at which the specific pump must run to deliver unit quantity against unit head, the efficiency being the same as the actual pump.

$$N_S = \frac{N\sqrt{Q}}{H^{3/4}} \qquad \text{………..(6.7)}$$

where N_S= specific speed

N = rotational speed(rpm)

H = total head

6.7 Performance of Pumps- Characteristic Curves

A pump is usually designed for one speed, flow rate and head in actual practice, the operation may be at some other condition of head on flow rate, and for the changed conditions, the behaviour of the pump may be quite different. Therefore, in order to predict the behaviour and performance of a pump under varying conditions, tests are performed and the results of the tests are plotted. The curves thus obtained are known as the characteristic curves of the pump. The following three types of characteristic curves are usually prepared for the centrifugal pumps :

(a) Main and operating characteristics.

(b) Constant efficiency or Muschel curves .

(c) Constant head and constant discharge curves.

Main and Operating Characteristics

In order to obtain the main characteristic curves of a pump it is operated at different speeds. For each speed the rate of flow Q is varied by means of a delivery valve and for the different values of Q the corresponding values of manometric head H_m, shaft H.P., P , and overall efficiency η are measured or calculated. The same operation is repeated for different speeds of the pump. Then Q v/s H_m ; Q v/s P and Q v/s η curves for different speeds are plotted, so that three sets of curves, as shown in Fig.6.4 are obtained, which represent the *main characteristics* of a pump. The main characteristics are useful in indicating the performance of a pump at different speeds.

During operation a pump is normally required to run at a constant speed, which is its designed speed, (same as the speed of the driving motor). As such that particular set of main characteristics which corresponds to the designed speed is mostly used in the operations of a pump and is, therefore, known as the *operating characteristics*. A typical set of such characteristics of a pump is shown in Fig.6.5

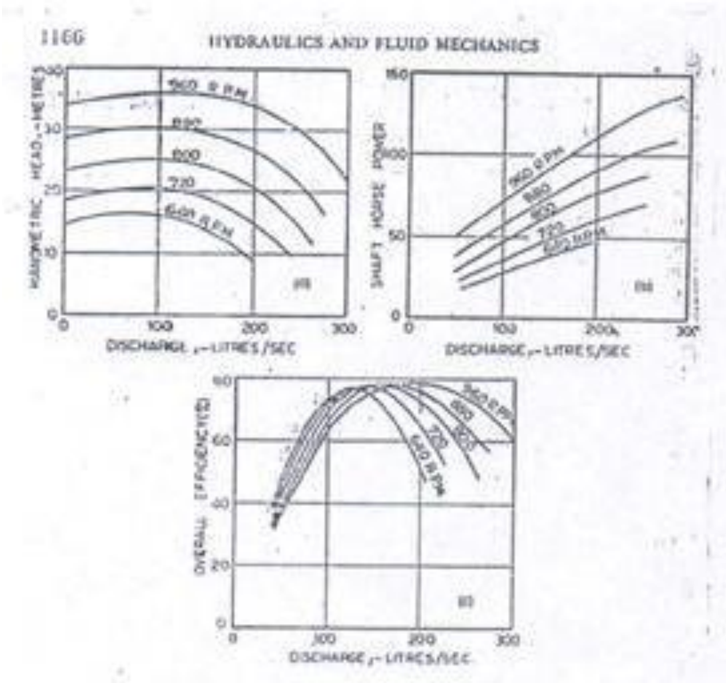


Fig.6.4 Main characteristics of a centrifugal pump

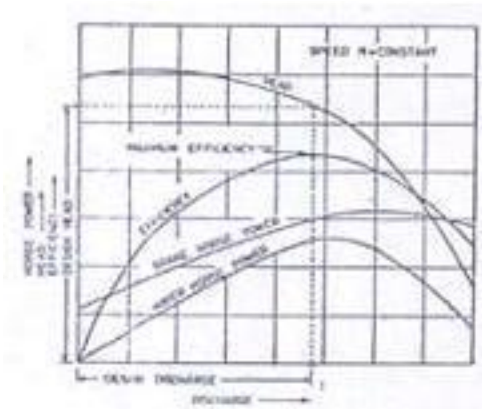


Fig. 6.5 Operating characteristic curves of a centrifugal pump

6.8 Parallel or Series Operation of Pumps

Pumps in series

Centrifugal pumps generate a relatively low head delivering a fairly high rate of discharge. Normally a pump with a single impeller can be used to deliver the required discharge against a maximum head of about 100 m. But if the liquid is required to be delivered against a still larger head then it can be done by using two or more *pumps in series.*

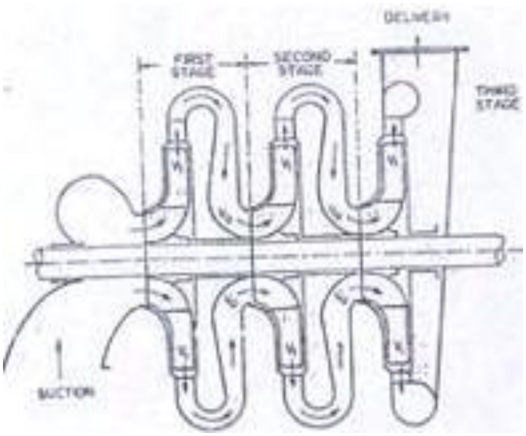


Fig.6.6 Three stage centrifugal pump

If the required head is more than that can be provided by one pump, the pumps are connected in series. The same discharge passes through both pumps but the head developed by one pump add the other. The total head developed is obtained by adding together the value of the head of each pump corresponding to the relevant discharge.

Pumps in Parallel

The multi-stage pumps or the pumps in series as described earlier are employed for delivering a relatively small quantity of liquid against very high heads. However, when a large quantity of liquid is required to be pumped against a relatively small head, then it may not be possible for a single pump to deliver the required discharge. In such cases two or more pumps are used which are so arranged that each of these pumps working separately lift the liquid from a common sump and deliver it to a common collecting pipe through which it is carried to the required height Fig.6.7. Since in this case each of the pumps deliver the liquid against the same head, the arrangement is known as *pumps in parallel*. If $Q_1, Q_2, Q_3 \ldots\ldots, Q_n$

are the discharging capacities of n pumps arranged in parallel then the total discharge delivered by these pumps will be

$$Q_t = (Q1+Q2+Q3+\ldots\ldots+Qn) \qquad \ldots\ldots\ldots\ldots(6.8)$$

If the discharging capacity of all the n pumps is same, equal to Q , then the total discharge delivered by these pumps will be

$$Q_t = nQ$$

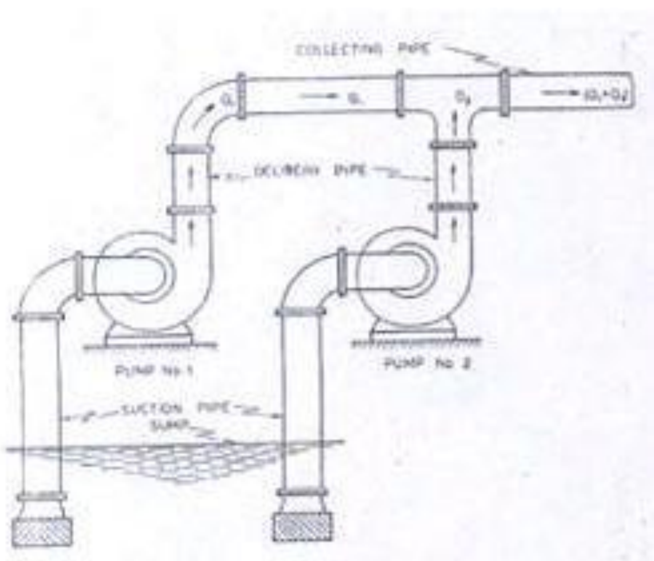


Fig.6.7 Two centrifugal pumps arranged in parallel

Example 6.1 A centrifugal pump, having four stages in parallel, delivers 11m^3/min of liquid against a head of 24.7m, the diameter of the impeller being 225mm and the speed 1700 rpm. A pump is to be made up with a number of identical stages in series, of similar constriction to those in the first pump, to run at 1250 rpm, and to deliver 14.5 m^3/min, against a head of 248m. Find the number of stages required for the second pump.

1st Pump	2nd Pump
Q = 11m^3/min	Q = 14.5m^3/min
H = 24.7 m	H = 248 m
N = 1700 rpm	N =1250 rpm
D = 225 mm	

Specific speed $N_S = \dfrac{N\sqrt{Q}}{H^{3/4}}$

Q for one pump = 11/4 = 2.75 m^3/min

$$N_S = \frac{1700\sqrt{2.75}}{24.7^{3/4}} = 254$$

For 2nd pump, with identical stages in series i.e multi-stage pump, if each stage is similar to those of each stage is similar to those of the first pump.

The specific speed of each stage N_S = 254

$$N_S = \frac{N\sqrt{Q}}{H^{3/4}}$$

$$254 = \frac{1250\sqrt{14.5}}{H^{3/4}}$$

H = 49.64 m

Total head required = 248 m

No of stages required = 248/49.64 = 5 stages

CHAPTER 7

DIMENSIONAL ANALYSIS, HYDRAULIC SIMILITUDE AND MODEL INVESTIGATION

7.1 Dimensional Analysis

Dimensional analysis is a mathematical method of obtaining the equations, changing units, determining a convenient arrangement of variable of a physical relation. In an equation expressing a physical relationship between quantities, absolute numerical and dimensional equality must exit. In general, all such physical relationships can be reduced to the fundamental quantities of mass M, length L and time T. It is a based on the assumption that the phenomenon can be expressed by a dimensionally homogeneous equation, with certain variable. The dimensional analysis is widely used in research work for developing design criteria and also for conducting model tests.

7.2 Dimensions and Units

All physical quantities are measured by comparison. This comparison is always made with respect to some arbitrarily fixed value for each independent quantity, called dimension(e.g., length, mass, time, etc.). Since there is no direct relationship between these dimensions, they are called fundamental dimensions. Some other quantities such as area, volume, velocity, force etc. can not be expressed in terms of fundamental dimensions and thus may be called derived dimensions.

There are two systems for fundamental dimensions namely FLT (i.e force, length, time) and MLT (i.e., mass, length, time). One common system employed in dimensional analysis is the M,L,T system. Table is a listing of some of the quantities used in fluid flow, together with their symbols and dimensions.

Quantity	Symbol	Dimensional Form
Length	l	L
Time	t	T
Mass	m	M
Velocity	v	$L\,T^{-1}$
Acceleration	a	$L\,T^{-2}$
Force	F	$M\,L\,T^{-2}$
Pressure	P	$M\,L^{-1}\,T^{-2}$
Discharge	Q	$L^{3}\,T^{-1}$
Power	P	$M\,L^{2}\,T^{-3}$
Work,energy	W,E	$M\,L^{2}\,T^{-2}$
Density	ρ	$M\,L^{-3}$
Dynamic viscosity	μ	$M\,L^{-1}\,T^{-1}$
Kinematic viscosity	ν	$L^{2}\,T^{-1}$
Surface tension	σ	$M\,T^{-2}$

7.3 Methods of Dimensional Analysis

The methods of dimensional analysis are:

-Buckingham's π theorem

-Ralyeigh's method

Buckingham's π Theorem

If there are 'n' variables in a dimensionally homogeneous equation, and if these variables contain 'm' fundamental dimensions such as (M,L,T) , they may be grouped into *(n-m) non-dimensional independent π terms*.

Mathematically, if a variable x_1 depends upon independent variables x_2, x_3,x_4, ….,x_n,

the functional equation may be written as

$$x_1 = f(x_2, x_3, x_4, \ldots, x_n)$$

The equation may be written in its general form as

f_1 (x_1, x_2, x_3,……., x_n) = C

In this equation there are *'n' variables*. If there are *'m' fundimental dimensions*, the according to π theorem

f_2 (π_1,π_2,π_3,….., π_{n-m}) = C_1

e.g Q = f (d,H,μ,ρ,g)

f_1 (Q, d, H, μ, ρ , g) = C

n = 6; m = 3; (n-m) = 3

f_2 (π_1, π_2, π_3) = C_1

Procedure

1. First of all, write the functional relationship with the given data.
2. Then write the equation in its general form.
3. Choose *'m' repeating variables* and write separate expressions for each term. Every π term will contain the repeating variables and one of the remaining variables. The repeating variables are written in exponential form.
4. With the help of the principle of dimensional homogeneity, find out the values of the exponents by obtaining simultaneous equations.
5. Substitute the value of these exponents in the π term.

6. After the π terms are obtained, write the functional relation in the required form.

e.g $Q = f(d, H, \mu, \rho, g)$

$f_1(Q, d, H, \mu, \rho, g) = C$

$n = 6$, $m = 3$, $(n-m) = 3$

$f_2(\pi_1, \pi_2, \pi_3) = C_1$

Choose ρ, g, d as repeating variable,

$\pi_1 = \rho^{a1} g^{b1} d^{c1} Q$

$\pi_2 = \rho^{a2} g^{b2} d^{c2} H$

$\pi_3 = \rho^{a3} g^{b3} d^{c3} \mu$

Selection of Repeating Variable

1. The variables should be such that none of them is dimensionless.
2. No two variables should have the same dimensions.
3. Independent variables should be as far as possible, be selected as repeating variable.

$\rho \rightarrow$ fluid property

$\nu \rightarrow$ flow characteristics

$l \rightarrow$ geometric characteristics

Example 7.1 A V-notch weir is a vertical plate with a notch angle ϕ cut into the top of it and placed across an open channel. The liquid in the channel is backed up and forced to flow through the notch. The discharge Q is some function of the elevation H of upstream liquid surface above the bottom of the notch. In addition it depends upon gravity and upon the velocity of approach V_o to the weir. Determine the form of discharge equation:

$$Q = \sqrt{g}\,H^{5/2} f\left(\frac{V_o}{\sqrt{gH}}, \phi \right)$$

$Q = f(H, g, V_o, \phi)$

$f_1(Q, H, g, V_o, \phi) = C$

Choose g and H as repeating variables

$n = 5$; $n-m = 3$; $m=2$

$\pi_1 = H^{a1}\, g^{b1}\, Q = (L)^{a1}\ (LT^{-2})^{b1}\, L^3 T^{-1}$

$\pi_2 = H^{a2}\, g^{b2}\, V_o = (L)^{a2}\ (LT^{-2})^{b2}\ LT^{-1}$

$\pi_3 = \phi$

$(M)^o\, (L)^o\, (T)^o = (L)^{a1}\, (LT^{-2})^{b1}\, L^3 T^{-1}$

$a_1+b_1+3 = 0 \Rightarrow a_1 = -5/2$

$-2b_1-1 = 0 \Rightarrow b_1 = -1/2$

$$\pi_1 = H^{-5/2}\, g^{-1/2}\, Q = \frac{Q}{\sqrt{g}\, H^{5/2}}$$

$(M)^o\, (L)^o\, (T)^o = (L)^{a2}\ (LT^{-2})^{b2}\ L\, T^{-1}$

$a_2 = -1/2$

$b_2 = -1/2$

$$\pi_2 = H^{-1/2} g^{-1/2} V_0 = \frac{V_0}{\sqrt{gH}}$$

$$f_2\left(\frac{Q}{\sqrt{g}\,H^{5/2}}, \frac{V_o}{\sqrt{gH}}, \phi\right) = C_1$$

$$\frac{Q}{\sqrt{g}\,H^{5/2}} = f\left(\frac{V_o}{\sqrt{gH}}, \phi\right)$$

$$Q = \sqrt{g}\,H^{5/2} f\left(\frac{V_o}{\sqrt{gH}}, \phi\right)$$

Example 7.2 Prove that the discharge over a spillway is given by the relation

$$Q = VD^2 f\left[\frac{\sqrt{gD}}{VD}, \frac{H}{D}\right]$$

where V= velocity of flow

D = depth of throat

H = Head of water

g = Acceleration due to gravity

$Q = f(V,D,H,G)$

$f_1(Q,V,D,H,G) = C$

Choose V and D as repeating variables

$n = 5$, $m = 2$, $n-m = 3$

$\pi_1 = V^{a1} D^{b1} Q = (LT^{-1})^{a1} (L)^{b1} (L^3T^{-1})$

$\pi_2 = V^{a2} D^{b2} H = (LT^{-1})^{a2} (L)^{b2} (L)$

$\pi_3 = V^{a3} D^{b3} g = (LT^{-1})^{a3} (L)^{b3} (LT^{-2})$

$M^0L^0T^0 = (LT^{-1})^{a1} (L)^{b1} (L^3T^{-1})$

$0 = -a_1-1$; $a_1 = -1$

$0 = a_1+b_1+3$; $b_1 = -2$

$\pi_1 = V^{-1} D^{-2} Q$

$$\pi_1 = \frac{Q}{VD^2}$$

$$M^0L^0T^0 = (LT^{-1})^{a2}(L)^{b2}(L)$$

$0 = -a_2$

$a_2+b_2+1 = 0$; $b_2 = -1$

$\pi_2 = V^0 D^{-1} H$

$= H/D$

$$M^0L^0T^0 = (LT^{-1})^{a3}(L)^{b3}(LT^{-2})$$

$0 = -a_3-2$; $a_3 = -2$

$0 = a_3+b_3+1$; $b_3 = 1$

$\pi_3 = V^{-2} D g$

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

$$f_2\left(\frac{Q}{VD^2}, \frac{H}{D}, \frac{\sqrt{gD}}{V}\right) = 0$$

$$\frac{Q}{VD^2} = f\left(\frac{\sqrt{gD}}{V}, \frac{H}{D}\right)$$

$$Q = VD^2\, f\left(\frac{\sqrt{gD}}{V}, \frac{H}{D}\right)$$

7.4 HYDRAULIC MODELS

Hydraulic models, in general, may be either true models or distorted models. True models have all the significant characteristics of the prototype reproduced to scale (geometrically similar) and satisfy design restrictions (kinematic and dynamic similitude). Model-prototype comparisons have clearly shown that the correspondence of behaviour is often well beyond expected limitations, as has been attested by the successful operation of many structures designed from model tests.

7.5 Hydraulic Similitude

To know the complete working and behaviour of the prototype, from its model, there should be a complete similarity between the prototype and its scale model. This similarity is known as hydraulic similitude. From the subject point of view, the following three types of hydraulic similitude are important.

(1) Geometric similitude

(2) Kinematic similitude

(3) Dynamic similitude

GEOMETRIC SIMILITUDE

The model and the prototype are identical in shape, but differ only in size.(The ratios of all the corresponding linear dimensions are equal) .

Let L = Length of the prototype B
= Breadth of the prototype D
= Depth of the prototype

l,b,d = corresponding values of the model

Linear ratio $L_r = \frac{L}{l} = \frac{B}{b} = \frac{D}{d}$

Area ratio $A_r = (\frac{L}{l})^2 = (\frac{B}{b})^2 = (\frac{D}{d})^2$

Volume ratio $V_r = (\frac{L}{l})^3 = (\frac{B}{b})^3 = (\frac{D}{d})^3$

KINEMATIC SIMILITUDE

The model and the prototype have identical motions. (The ratios of the velocities at corresponding points are equal)

Let V_1 = velocity of liquid in prototype at point
1 V_2 = velocity of liquid in prototype at point 2

v_1, v_2 = corresponding values of the model

Velocity ratio $V_r = \frac{V_1}{v_1} = \frac{V_2}{v_2} = \ldots..$

DYNAMIC SIMILITUDE

The model and prototype have identical forces.(The ratios of the corresponding forces acting at corresponding points are equal).

Force ratio $F_r = \frac{F_1}{f_1} = \frac{F_2}{f_2} = \ldots..$

7.6 CLASSIFICATION OF MODELS

(1) Undistorted model

(2) Distorted model

Undistorted model

A model which is geometrically similar to the prototype is known as undistorted model.

Distorted model

Model does not have complete geometric similarity with the prototype, is known as distorted model.

7.7 Comparison of an Undistorted Model and the Prototype

If the model is to be overall similar to the prototype, then all the three similarities (i.e, geometric, kinematic, dynamic) should exist. But this is not possible in actual practice, as it is difficult to exist two types of similarities simultaneously. In general, and undistorted model of a prototype is made keeping in view the geometric similarity only and the remaining similarities are then compared for the scale ratio.

7.8 Velocity of Water in Prototype for the Given Velocity of an Undistorted Model

Consider an undistorted model geometrically similar to a proposed prototype like a weir, dam, spillway etc.

Let h = head of water over the model

v = velocity of water at a point in the model

H,V = corresponding values for the prototype

1/s = scale ratio of the model to the prototype

Velocity of water in the model $v = C_v \sqrt{2gh}$

Velocity of water on the corresponding point in the prototype $V = C_v \sqrt{2gH}$

$$\frac{v}{V} = \frac{C_v \sqrt{2gh}}{C_v \sqrt{2gH}} = \sqrt{\frac{h}{H}} = \sqrt{\frac{1}{s}}$$

$V = v \sqrt{s}$

Example 7.3 The velocity at a point on a spillway model of a dam is 1.3m/sec for a

prototype of model ratio 1:10. What is the velocity at the corresponding point in the prototype?

Velocity in the model $v = 1.3$ m/s

Model ratio, $1/s = 1/10$

$s = 10$

Velocity in the prototype $V = v\sqrt{s}$

$V = 1.3 \times \sqrt{10}$

$= 4.11$ m/sec

References:

1.Hydraulics and Fluid Mechanics P.N.Modi and S.M.Seth

2. Fluid Mechanics and HydraulicMachines R.S.Khurmi

3. Fluid Mechanics Victor L.Streeter
4. Fluid Mechanics and Hydraulic Machines R. K BANSAL