COURSE FILE

HYDRAULIC MACHINES

(sub code:ME 402)

II Year B.Tech. (Mechanical Engineering)

Prepared by Ranjeet kumar

K.K COLLEGE OF ENGINEERING & TECHNOLOGY,

Govindpur ,Dhanbad

Course Description:

Fluid mechanics is the branch of physics that studies fluids (liquids, gases, and plasmas) and the forces on them.

Fluid mechanics can be divided into fluid statics, the study of fluids atrest; fluid kinematics, the study of fluids in motion; and fluid dynamics, the study of the effect of forces on fluid motion. It is a branch of continuum mechanics, a subject which models matter without using the information that it is made out of atoms, that is, it modelsmatter from a macroscopic viewpoint rather than from a microscopic viewpoint. Fluidmechanics, especially fluid dynamics, is an active field of research with many unsolved orpartly solved problems. Fluid mechanics can be mathematically complex. Sometimes it canbest be solved by numerical methods, typically using computers

Course Objectives of the HydraulicMachines

The objective of the course is to enable the student;

1. To introduce the concepts of momentum principles
2. To impart the knowledge on pumps and turbines
3. To impart the knowledge of impact of jets.
4. To introduce the concepts of the working and design aspects of hydraulic machines liketurbines and pumps and their applications.

FLUID MACHINE

COURSE CODE:ME 402

MODULE 1

Introduction:impulse of jet and impulse turbines. Classification of fluid machine &devices,application of momentum and momentum equation to flow through hydraulic machinery, Euler's fundamental equation. Introduction to hydrodynamic thrust of jet on a fixed and moving surface(flat & curve, Classification of turbines, impulse turbines, Constructional details, Velocity triangles ,Power and efficiency calculations, governing of Pelton wheel

MODULE II

Reaction turbines: Francis and Kaplan turbines, Constructional details, velocity triangles, power and efficiency calculations, degree of reaction, draft tube, Cavitationin turbines, principles of similarity, unit and specific speed,performance characteristics,selection of turbines

MODULE III

Centrifugal pumps: Classification of centrifugal pumps, vector diagram, work done by impellor , efficiencies of centrifugal pumps, specific speeds, Cavitation and separation, performance characteristics.

MODULE IV

Positive displacement and other pumps: Reciprocating pump theory, slip,indicator diagram, effect of acceleration,air vessel, comparison of centrifugal and reciprocating pumps, performance characteristics.

IMPORTANT QUESTIONS & NUMERICALS

1. (a) Define the term 'Governing of a turbine'. Describe with a neat sketch, the working of an oil pressure governor for a pelton wheel.

(b) Give the range of specific speed values of the Kaplan, Francis turbine and Pelton wheels. What factors decide whether Kaplan, Francis, or a Pelton type turbine

2. (a) Describe the theory of a draft tube with the help of a neat sketch.

3. A jet of water having a velocity of 60m/sec is deflected by a vane moving at 25m/sec in a direction at 300 to the direction of jet. The water leaves the vane normally to the motion of the vane. Draw the inlet and outlet velocity triangles and find out the vane angles for no shock at entry and exit. Take the relative velocity at the exit as 0.8 times the relative velocity at the entrance.

4. A jet of water having a velocity of 30 m/s strikes a series of radial curved vanes mounted on a wheel which is rotating at 200 rpm. The jet makes an angle of 200 with the tangent to the wheel at inlet and leaves the wheel with a velocity of 5 m/s at an angle of 1300 to the tangent to the wheel at outlet. Water is flowing from outward in a radial direction. The outer and inner radii of the wheel are 0.5 m and 0.25 m respectively. Determine; (i) Vane angles at inlet and outlet (ii) Work done per unit weight of water, and (iii) Efficiency of the wheel

.

5. (a) Draw a schematic diagram of a Francis turbine and explain its construction and working.

(b) The jet of water coming out of nozzle strikes the buckets of a Pelton wheel which when stationary would deflect the jet through 1650. The velocity of water at exit is 0.9 times at the inlet and the bucket speed is 0.45 times the jet speed. If the speed of the Pelton wheel is 300 rpm and the effective head is 150m, determine: (i) Hydraulic efficiency, and (ii) Diameter of the Pelton wheel. Take co-efficient of velocity cv = 0.98.

6. (a) Define specific speed of a turbine and derive an expression for the same. Show that Pelton turbine is a low specific speed turbine.

(b) What is specific speed? State its significance in the study of hydraulic machines.

7. (a) What is governing and how it is accomplished for different types of water turbines?

(b) A Kaplan turbine develops 1480 kW under a head of 7 m. The turbine is set 2.5 m above the tailrace level. A vacuum gauge inserted at the turbine outlet records a suction head of 3.1 m. If the hydraulic efficiency is 85%, what would be the efficiency of draft tube having inlet diameter of 3 m? What would be the reading of suction gauge if power developed in reduced to half (740 kW), the head and speed remaining constant.

9. (a) A jet of water is moving at 60 m/s and is deflected by a vane moving 25 m/s in a direction at 300 to the direction of the jet. The water leaves the blades with no velocity component in the direction of motion of vane. Determine the inlet and outlet angles of the vanes for no shock at entry or exit. Assume outlet velocity of water relative to the blades to be 0.85 of the relative velocity at entry.

(b) A 100 mm diameter jet discharging at 0.40 m3/sec impinges in a series of curved vanes moving at 18 m/s. the direction of the jet and the direction of motion of the vane are the same at inlet. Each vane is so shaped that if stationary it would deflect the jet at 1700. Calculate (i) The force exerted in the direction of motion of the vanes (ii) The power developed and (iii) The Hydraulic Efficiency.

10. (a) A jet of water having a velocity of 35 m/s impinges on a series of vanes moving with a velocity of 22 m/s. The jet makes an angle of 300 to the direction of motion of vanes when entering and leaving at and angle of 1200. Draw the velocity triangles at inlet and outlet and find: (i) the angles of vanes tip so that water enters and leaves without shock (ii) The work done per N of water entering the vanes and (iii) The efficiency (b) Prove that the force exerted by a jet of water on a fixed semi0circular plate in the direction of the jet when jet strikes at the center of the semi circular plate is two times the force exerted by the jet on a fixed vertical plate.

11. (a) An inward flow reaction turbine is required to produce a power of 280 kW at 220 rpm. The effective head on the turbine is 20 m. The inlet diameter is twice as the outlet diameter. Assume hydraulic efficiency as 83% and overall efficiency as 80%. The radial velocity is 3.75 m/s and is constant. The ratio of wheel diameter to breadth

is 0.1 and 6% of the flow area is blocked by vane thickness. Determine the inlet and outlet diameters, inlet and exit vane angle and guide blade angle at the inlet. Assume radial discharge.

(b) In a Francis turbine, the blade angle is 150 and the flow enters in a radial direction. The flow velocity is constant and is equal to 8.25 m/s. The outlet diameter is 0.6 times the inlet diameter and the runner rotates at 400 rpm. The width of the wheel is 0.1 times the inlet diameter and 7% of the flow area is blocked by blade thickness. Assume radial flow at outlet. Calculate: (i) Diameters at outlet and inlet (ii) Blade angle at outlet (iii) The head and power developed.

12. (a) The following are the data of a Pelton wheel turbine; Head at nozzle is 600 m; shaft power is 70 MW; speed is 550 rpm; Discharge is 13 m3/sec; number of jects are 4; runner diameter is 2 m; Diameter of jets is 0.20 ml outlet vane angle is 160; mechanical efficiency is 98%. Determine the head lost in the nozzle, head lost in the buckets. Find also the power lost in the nozzle and the buckets.

(b) The runner of pelton wheel turbine has tangential velocity of 18 m/s and works under a head of 62 m. The jet is turned through 170. The discharge through the nozzle is 110 liters per second. Determine the power developed by the runner and the efficiency of Assume Cv=0.98.

13. (a) A free jet of water of velocity V strikes against a series of curved semi-circular vanes tangentially. The vanes are moving in the direction of the jet with velocity equal to 0.6V. Assuming the relative velocity of water is reduced by 10% by moving over the vanes, show that the vanes have an efficiency of 91.33%.

(b) A jet of water of diameter 40 mm and 22 m/s impinges on: (i) A normal flat vane moving in the direction of jet at 8 m/s and (ii) A series of normal flat vanes mounted on a wheel which has a tangential velocity of 7.5 m/s. Calculate force exerted, work done by water and efficiency of the system in both cases.

14. (a) What are the different types of efficiencies of turbine?

(b) Hydraulic tests were conducted on Francis turbine of 0.75 m diameter under a head of 10 m. The turbine developed 120 kW running at 240 rpm and consuming 1.25 m3/sec. If the same turbine is operated under a head of 15 m predict its new speed, discharge and power.

15.a) A 1250 m long pipeline, with frictional coefficient 0.005, supplies three single jet Pelton wheels the top water level of the reservoir being 350 m above the nozzles. The Cv for each nozzle is 0.98. The efficiency of each turbine based on the head at the nozzle is 85%. The head lost in the friction is 12.50 m. The specific speed of each wheel is 15, and the working speed is 550 rpm. Find the; (i) Total power developed (ii) Discharge (iii) Diameter of each nozzle (iv) Diameter of the pipe line.

(b) For maximum conversion of hydraulic power into mechanical power, what should be the shape of velocity diagram at the outlet in case of a reaction turbine.

16. (a) A jet of diameter 40 mm strikes horizontally on a plate held vertically. What force is required to hold plate for a flow of oil of specific gravity 0.8 with a velocity of 30 m/s.

(b) A 75 mm diameter jet having a velocity of 37 m/s strikes normally a flat plate, the normal at 450 to the axis of the jet. Find the normal pressure on the plate; (i) When the plate is stationary (ii) When the plate is moving with a velocity of 17 m/s in the direction of the away from the jet. Also determine the power and the efficiency of the jet when the plate is moving.

LECTURES NOTES

IMPACT OF JETS

The liquid comes out in the form of a jet from the outlet of a nozzle, which is fitted to a pipe through which the liquid if flowing under pressure. If some plate, which may be fixed or moving, is placed in the path of the jet, a force is exerted by the jet on the plate. This force is obtained from Newton's second law of motion or from impulse-moment equation. Thus impact of jet means the force exerted by the jet on a plate which may be stationary or moving. In this chapter, the following cases of the impact jet i.e., the force exerted by the jet on a plate, will be considered:

1. **Force exerted by a jet on a stationary plate when**
   a. Plate is vertical to the jet,
   b. Plate is inclined to the jet, and
   c. Plate is curved.

2. **Force is exerted by the jet on the moving plate, when**
   a. Plate is vertical to the jet,
   b. Plate is inclined to the jet, and
   c. Plate is curved

Force exerted by a jet on a stationary vertical plate :
Consider a jet of water coming out of the nozzle, strikes a flat vertical plate as shown in the Figure 1.

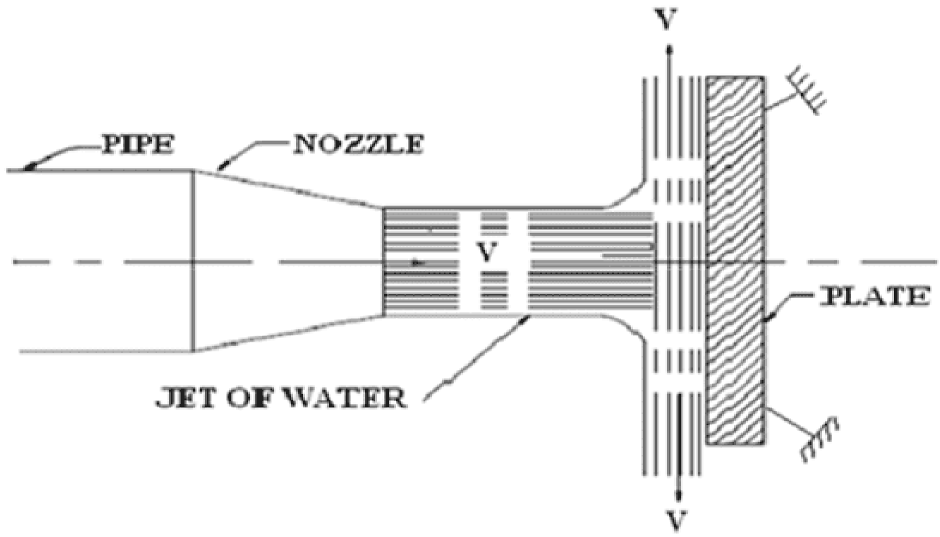


Figure 1

Let,

ρ = density of water

a = area of cross section of the jet= $\frac{\pi d^2}{4}$

v = velocity of the water jet

The jet after striking the plate will move along the plate. But the plate is right angles to the jet. Hence the jet after striking will get deflected by 90Â°. Hence the component of the velocity of the jet, in the direction of the jet, after striking will be zero. The force exerted by the jet on the plate in the direction of the jet.

= (initial momentum - final momentum)/ time

= (mass x v_1 - mass x v_2)/time

= mass rate of flow (v_1 - v_2)

$= \rho a v [v - 0]$

$= \rho a v^2$

For deriving the above equation, we have taken initial velocity minus final velocity and not final velocity minus initial velocity. If the force exerted on the jet is to be calculated then final minus the initial velocity is taken. But if the force exerted by the jet on the plate is to calculated, then initial velocity minus the final velocity is taken.

Force exerted by a jet on a stationary inclined flat plate :

Let a jet of water, coming out from the nozzle; strike an inclined flat plate as shown in the figure.2.

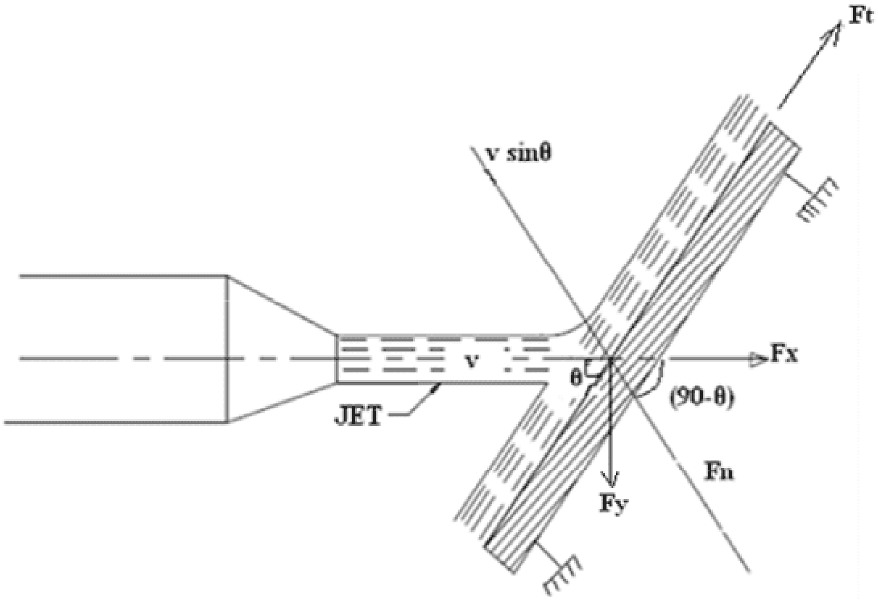


Figure.2.

Let

a= area of cross section of the jet = $\frac{\pi d^2}{4}$

v = velocity of the jet in the direction of X

θ = Angle between the jet and the plate

Then the mass of the water per sec stricking the plate = ρ x av

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

Let find the force exerted by the jet on the plate In the direction normal to the plate. Let this force is represented by F_n

then, F_n = Mass of the jet striking per second Ã— [initial velocity of the jet before striking in the direction of n - final velocity of the jet after striking in the direction of n

$$F_n = \rho av\left[v\sin\theta - 0\right] = \rho av^2 \sin\theta$$

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

$$F_x = \rho av^2 \sin^2\theta$$

(along the direction of the flow) and

$$F_y = \rho av^2 \sin\theta\cos\theta$$

Force exerted by a jet on a stationary cover plate

Jet strikes the curved plate at the centre. Let a jet of water strike a fixed curved plate at the centre as shown in figure.3. The jet after striking the plate comes out with the same velocity if the plate is smooth and there is no loss of energy due to impact of the jet, in the tangential direction of the curved plate. The velocity at the outlet of the plate can be resolved in to 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 = $-V\cos(\theta)$

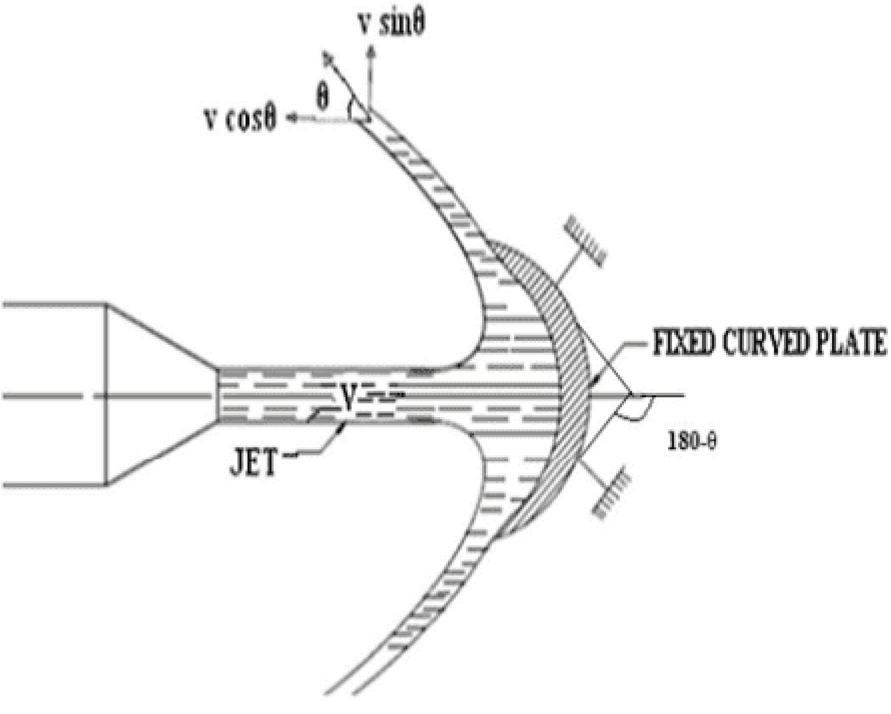


Figure. 3.

(-ve sign is taken as the velocity at the outlet is in the opposite direction of the jet of water coming out from nozzle).

Component of the velocity perpendicular to the jet = $V\sin(\theta)$

Force exerted by the jet In the direction of the jet,

F_x = Mass per sec x $[v_{1x} - v_{2x}]$

where v_{1x} = initial velocity in the diection of jet = v

v_2 = final velocity in the direction of the jet= v cos θ

$\therefore F_x = \rho av[\mathrm{v} - (-v\cos\theta)] = \rho av[v + v\cos\theta)]$

$= \rho av^2[1 + \cos\theta)]$

similarly F_y = mass per sec x $[v_{1y} - v_{2y}]$

where v_{1y} = initial velocity in the direction of y = 0

v_{2y} = final velocity in the direction of y = $V\sin\ \theta$

$F_y = \rho av\left[0 - v\sin\theta\right] = -\rho av^2 \sin\theta$

-ve sign means the force is acting in the downward direction. In this case the angle of deflection of the jet = $= 180^\circ - \theta$

Jet strikes the curved plate at on e end tangentially when the plate is symmetrical Let the jet strike the curved plate at one end tangentially as shown in the figure.4. Let the curved is symmetrical about x-axis. Then the angle made by the tangents at the two ends of the plate will b same.

Let V = Velocity of the jet of water

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

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

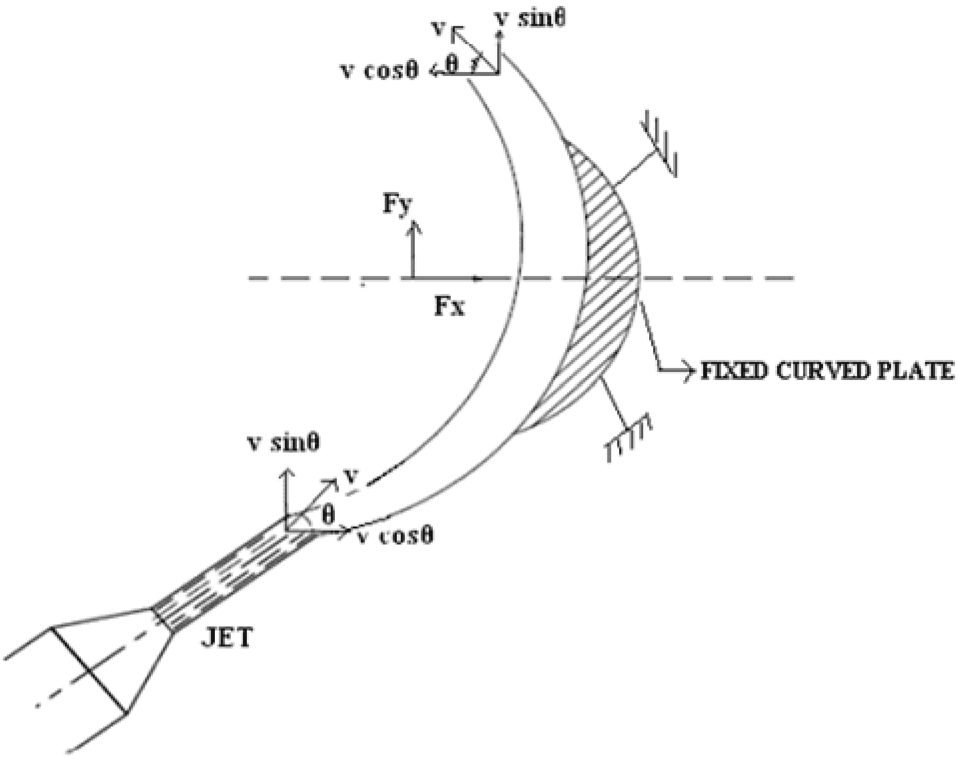
F_x = mass pe sec x $[v_{1x} - v_{2x}]$

$= \rho av[v\cos\theta - (-\mathrm{v}\cos\theta)]$

$= \rho av[v\cos\theta + \mathrm{v}\cos\theta]$

$F_y = \rho av[v_{1y} - v_{2y}]$

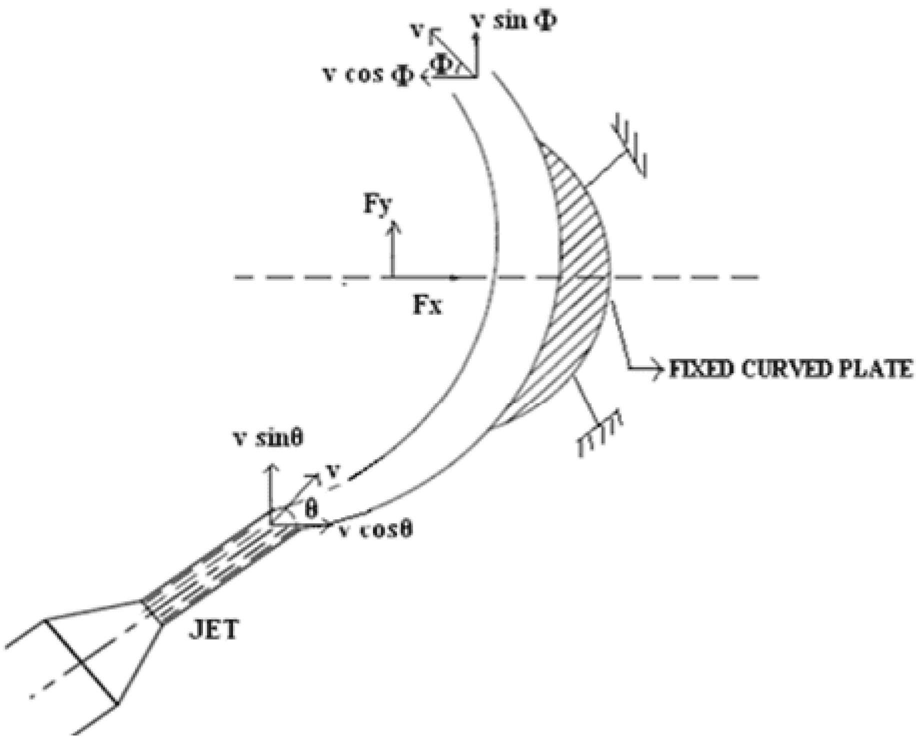
$= \rho av[v\sin\theta - \mathrm{v}\sin\theta] = 0$



Jet strikes the curved plate at one end tangentially when the plate is unsymmetrical

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

θ = angle made by the tangent at the tip with the x " axis,



φ = angle made by the tangent at the outlet tip with x-axis.

The two component of the velocity at inlet are

$v_{1x} = v\cos\theta$ and $v_{1y} = v\sin\theta$

The two component of the velocity at outlet are

$v_{2x} = -v\cos\phi$ and $v_{2y} = v\sin\phi$

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

$$F_x = \text{mass rate of flow } (v_{1x} - v_{2x})$$

$$= \rho a v[v\cos\theta - (-v\cos\phi)]$$

$$= \rho a v^2[\cos\theta + \cos\phi]$$

$$F_y = \text{mass rate of flow } (v_{1y} - v_{2y})$$

$$= \rho a v[v\sin\theta + v\sin\phi]$$

$$= \rho a v^2[\sin\theta + \sin\phi]$$

Force on a moving surface

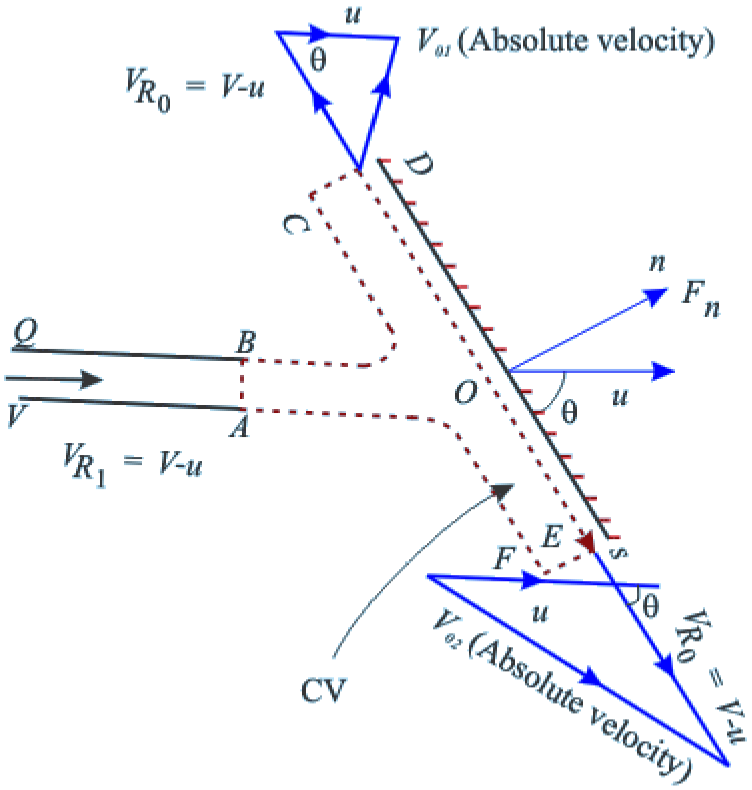


Fig 11.5 Impingement of liquid jet on a moving flat plate

If the plate in the above problem moves with a uniform velocity u in the direction of jet velocity V (Fig. 11.5). The volume of the liquid striking the plate per unit time will be

$$Q = a(V - u) \quad (11.10)$$

Physically, when the plate recedes away from the jet it receives a less quantity of liquid per unit time than the actual mass flow rate of liquid delivered, say by any nozzle. When u = V, Q = 0 and when u > V, Q becomes negative. This implies physically that when the plate moves away from the jet with a velocity being equal to or greater than that of the jet, the jet can never strike the plate.

The control volume ABCDEFA in the case has to move with the velocity u of the plate. Therefore we have to apply Eq. (10.18d) to calculate the forces acting on the control volume. Hence the velocities relative to the control volume will come into picture. The velocity of jet relative to the control volume at its inlet becomes $V_{R1} = V - u$

Since the pressure remains same throughout, the magnitudes of the relative velocities of liquid at outlets become equal to that at inlet, provided the friction between the plate and the liquid is neglected. Moreover, for a smooth shockless flow, the liquid has to glide along the plate and hence the direction of V_{R0}, the relative velocity of the liquid at the outlets, will be along the plate. The absolute velocities of the liquid at the outlets can be found out by adding vectorially the plate velocity u and the relative velocity of the jet V - u with respect to the plate. This is shown by the velocity triangles at the outlets (Fig. 11.5). Coordinate axes fixed to the control volume ABCDEFA are chosen as ”0s” and ”0n” along and perpendicular to the plate respectively.

The force acting on the control volume along the direction ”*0s*” will be zero for a frictionless flow. The net force acting on the control volume will be along ”*0n*” only. To calculate this force F_n, the momentum theorem with respect to the control volume ABCDEFA can be written as

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

Substituting Q from Eq (11.10),

$$F_n = -\rho a(V-u)^2 \sin\theta$$

Hence the force acting on the plate becomes

$$F_p = -F_n = \rho a(V-u)^2 \sin\theta \qquad (11.11)$$

If the plate moves with a velocity u in a direction opposite to that of V (plate moving towards the jet), the volume of liquid striking the plate per unit time will be Q = a(V + u) and, finally, the force acting on the plate would be

$$F_p = -F_n = \rho a(V+u)^2 \sin\theta \qquad (11.12)$$

From the comparison of the Eq. (11.9) with Eqs (11.11) and (11.12), conclusion can be drawn that for a given value of jet velocity V, the force exerted on a moving plate by the jet is either greater or lower than that exerted on a stationary plate depending upon whether the plate moves towards the jet or-away from it respectively.

The power developed due to the motion of the plate can be written (in case of the plate moving in the same direction as that of the jet) as

$$P = F_p \,.\, U$$

$$P = F_p \sin\theta |u| = \rho a(V-u)^2 u \sin^2\theta \qquad (11.13)$$

Dynamic Forces on Curve Surfaces due to the Impingement of Liquid Jets

The principle of fluid machines is based on the utilization of useful work due to the force exerted by a fluid jet striking and moving over a series of curved vanes in the periphery of a wheel rotating about its axis. The force analysis on a moving curved vane is understood clearly from the study of the inlet and outlet velocity triangles as shown in Fig. 11.6.

The fluid jet with an absolute velocity V_1 strikes the blade at the inlet. The relative velocity of the jet V_{r1} at the inlet is obtained by subtracting vectorially the velocity u of the vane from V_1. The jet strikes the blade without shock if β_1 (Fig. 11.6) coincides with the inlet angle at the tip of the blade. If friction is neglected and pressure remains constant, then the relative velocity at the outlet is equal to that at the inlet ($V_{r2} = V_{r1}$).

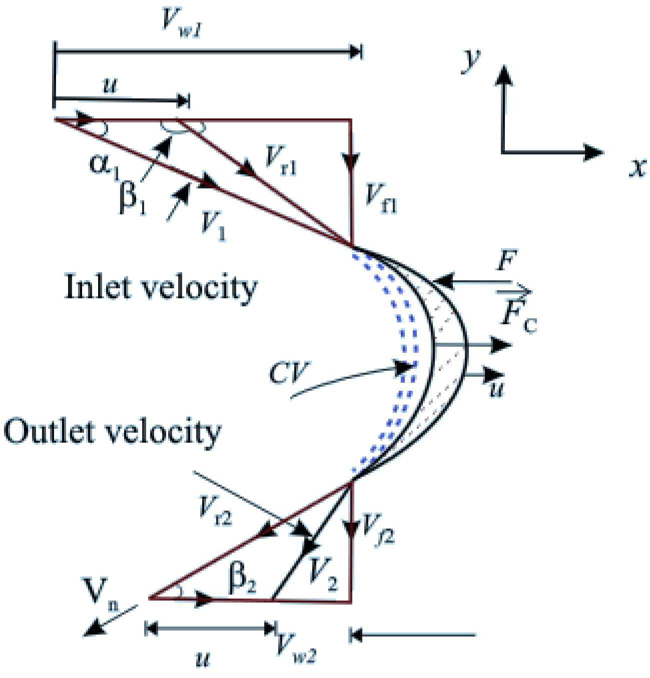


Fig 11.6 Flow of Fluid along a Moving Curved Plane

The control volume as shown in Fig. 11.6 is moving with a uniform velocity u of the vane.Therefore we have to use Eq.(10.18d) as the momentum theorem of the control volume at its steady state. Let F_c be the force applied on the control volume by the vane.Therefore we can write

$$F_c = \dot{m}V_{r2}\cos\beta_2 - \dot{m}V_{r1}\cos(180^0 - \beta_1)$$
$$= \dot{m}\,V_{w2} + \dot{m}V_{w1}$$

$$= \dot{m}(V_{w1} + V_{w2})$$

To keep the vane translating at uniform velocity, u in the direction as shown.the force F has to act opposite to F_c Therefore,

$$F = -F_c = -\dot{m}(V_{w1} + V_{w2}) \qquad (11.14)$$

From the outlet velocity triangle, it can be written

$$(V_{w2} + u)^2 = V_{r2}^2 - V_{f2}^2$$

or, $$V_{w2}^2 + u^2 + 2V_{w2}u = V_{r2}^2 - V_{f2}^2$$

or, $$V_2^2 - V_{f2}^2 + u^2 + 2V_{w2}u = V_{r2}^2 - V_{f2}^2$$

or, $$V_{w2}u = \frac{1}{2}[V_{r2}^2 - V_2^2 - u^2] \qquad (11.15a)$$

Similarly from the inlet velocity triangle.it is possible to write

$$V_{w1}u = \frac{1}{2}[-V_{r1}^2 + V_1^2 + u^2] \qquad (11.15b)$$

Addition of Eqs (11.15a) and (11.15b) gives

$$(V_{w1} + V_{w2})u = \frac{1}{2}(V_1^2 - V_2^2)$$

Power developed is given by

$$P = \dot{m}(V_{w1} + V_{w2})u = \frac{\dot{m}}{2}(V_1^2 - V_2^2) \qquad (11.16)$$

The efficiency of the vane in developing power is given by

$$\eta = \frac{\dot{m}(V_{w1} + V_{w2})u}{\frac{\dot{m}}{2}V_1^2} = 1 - \frac{V_2^2}{V_1^2} \qquad (11.17)$$

Hydraulic machines

A fluid machine is a device which converts the energy stored by a fluid into mechanical energy or vice versa . The energy stored by a fluid mass appears in the form of potential, kinetic and intermolecular energy. The mechanical energy, on the other hand, is usually transmitted by a rotating shaft. Machines using liquid (mainly water, for almost all practical purposes) are termed as hydraulic machines. In this chapter we shall discuss, in general, the basic fluid mechanical principle governing the energy transfer in a fluid machine and also a brief description of different kinds of hydraulic machines along with their performances. Discussion on machines using air or other gases is beyond the scope of the chapter.

CLASSIFICAITONS OF FLUID MACHINES

The fluid machines may be classified under different categories as follows:

Classification Based on Direction of Energy Conversion.

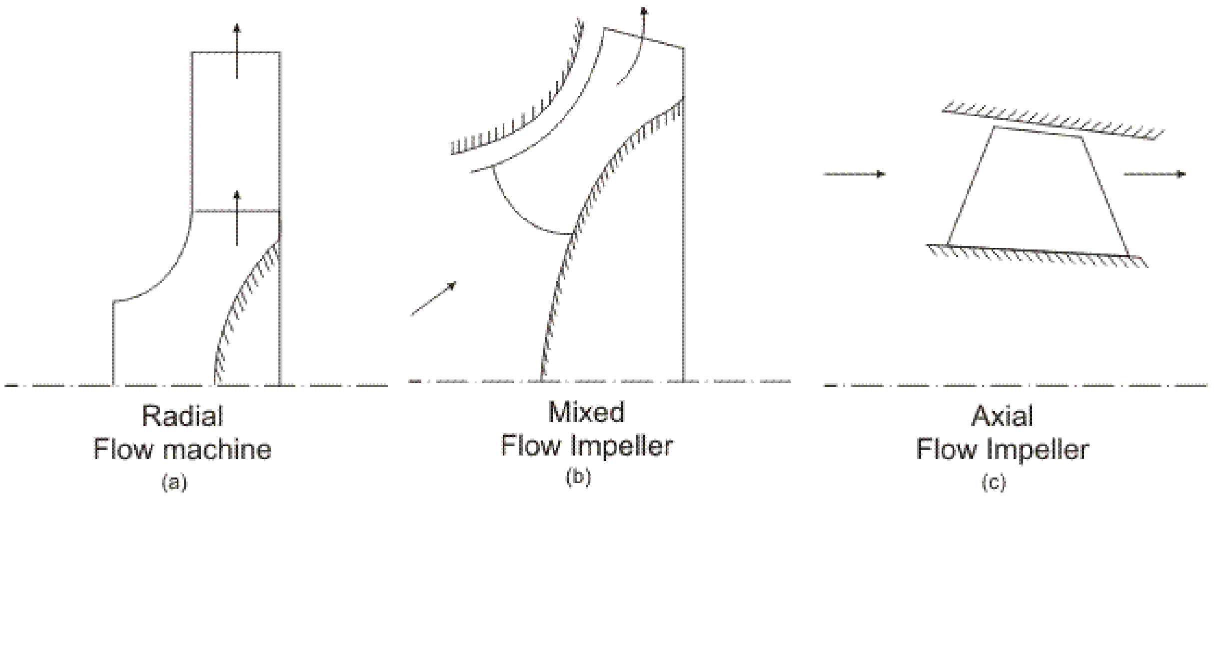
The device in which the kinetic, potential or intermolecular energy held by the fluid is converted in the form of mechanical energy of a rotating member is known as a turbine . The machines, on the other hand, where the mechanical energy from moving parts is transferred to a fluid to increase its stored energy by increasing either its pressure or velocity are known as pumps, compressors, fans or blowers .

Classification Based on Principle of Operation

The machines whose functioning depend essentially on the change of volume of a certain amount of fluid within the machine are known as positive displacement machines . The word positive displacement comes from the fact that there is a physical displacement of the boundary of a certain fluid mass as a closed system. This principle is utilized in practice by the reciprocating motion of a piston within a cylinder while entrapping a certain amount of fluid in it. Therefore, the word reciprocating is commonly used with the name of the machines of this kind. The machine producing mechanical energy is known as reciprocating engine while the machine developing energy of the fluid from the mechanical energy is known as reciprocating pump or reciprocating compressor.

The machines, functioning of which depend basically on the principle of fluid dynamics, are known as rotodynamicmachines . They are distinguished from positive displacement machines in requiring relative motion between the fluid and the moving part of the machine. The rotating element of the machine usually consisting of a number of vanes or blades, is known as rotor or impeller while the fixed part is known as stator. Impeller is the heart of rotodynamic machines, within which a change of angular momentum of fluid occurs imparting torque to the rotating member.

For turbines, the work is done by the fluid on the rotor, while, in case of pump, compressor, fan or blower, the work is done by the rotor on the fluid element. Depending upon the main direction of fluid path in the rotor, the machine is termed as radial flow or axial flow machine . In radial flow machine, the main direction of flow in the rotor is radial while in axial flow machine, it is axial. For radial flow turbines, the flow is towards the centre of the rotor, while, for pumps and compressors, the flow is away from the centre. Therefore, radial flow turbines are sometimes referred to as radially inward flow machines and radial flow pumps as radially outward flow machines. Examples of such machines are the Francis turbines and the centrifugal pumps or compressors. The examples of axial flow machines are Kaplan turbines and axial flow compressors. If the flow is partly radial and partly axial, the term mixed-flow machine is used. Figure 1.1 (a) (b) and (c) are the schematic diagrams of various types of impellers based on the flow direction.



Classification Based on Fluid Used

The fluid machines use either liquid or gas as the working fluid depending upon the purpose. The machine transferring mechanical energy of rotor to the energy of fluid is termed as a pump when it uses liquid, and is termed as a compressor or a fan or a blower, when it uses gas. The compressor is a machine where the main objective is to increase the static pressure of a gas. Therefore, the mechanical energy held by the fluid is mainly in the form of pressure energy. Fans or blowers, on the other hand, mainly cause a high flow of gas, and hence utilize the mechanical energy of the rotor to increase mostly the kinetic energy of the fluid. In these machines, the change in static pressure is quite small.

For all practical purposes, liquid used by the turbines producing power is water, and therefore, they are termed as water turbines or hydraulic turbines . Turbines handling gases in practical fields are usually referred to as steam turbine, gas turbine, and air turbine depending upon whether they use steam, gas (the mixture of air and products of burnt fuel in air) or air.

ROTODYNAMIC MACHINES

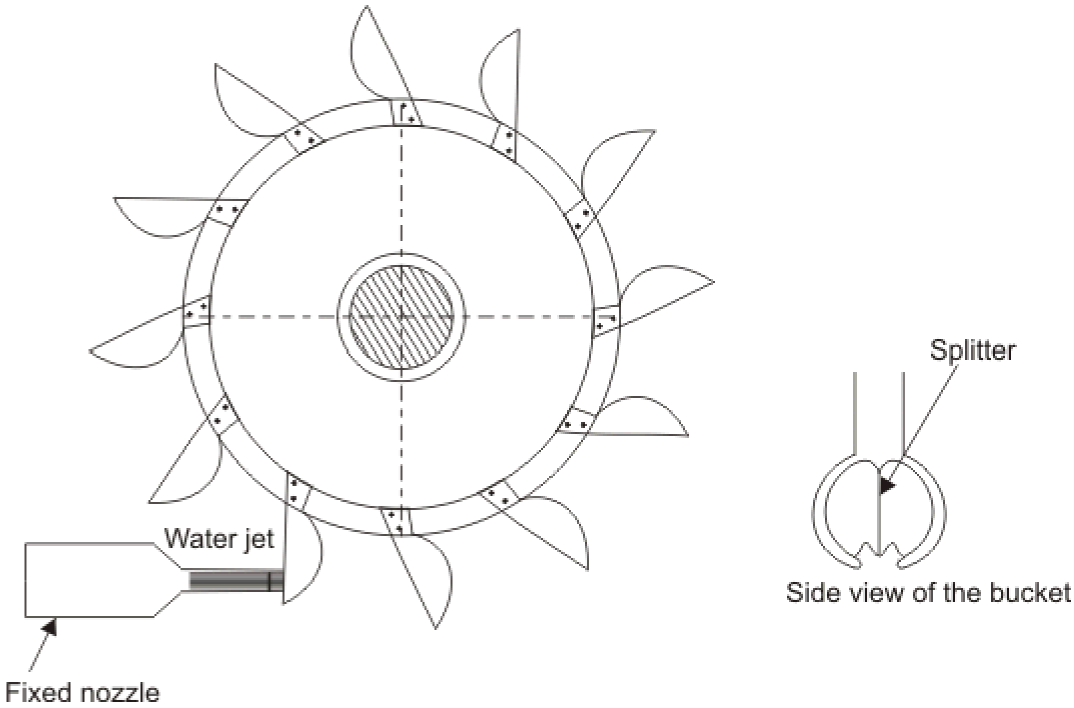
In this section, we shall discuss the basic principle of rotodynamic machines and the performance of different kinds of those machines. The important element of a rotodynamic machine, in general, is a rotor consisting of a number of vanes or blades. There always exists a relative motion between the rotor vanes and the fluid. The fluid has a component of velocity and hence of momentum in a direction tangential to the rotor. While flowing through the rotor, tangential velocity and hence the momentum changes.

The rate at which this tangential momentum changes corresponds to a tangential force on the rotor. In a turbine, the tangential momentum of the fluid is reduced and therefore work is done by the fluid to the moving rotor. But in case of pumps and compressors there is an increase in the tangential momentum of the fluid and therefore work is absorb

Hydropower is the longest established source for the generation of electric power. In this module we shall discuss the governing principles of various types of hydraulic turbines used in hydro-electric power stations.

Impulse Hydraulic Turbine : The Pelton Wheel

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 . 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 centres of the buckets. Down the centre of each bucket, there is a splitter 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.

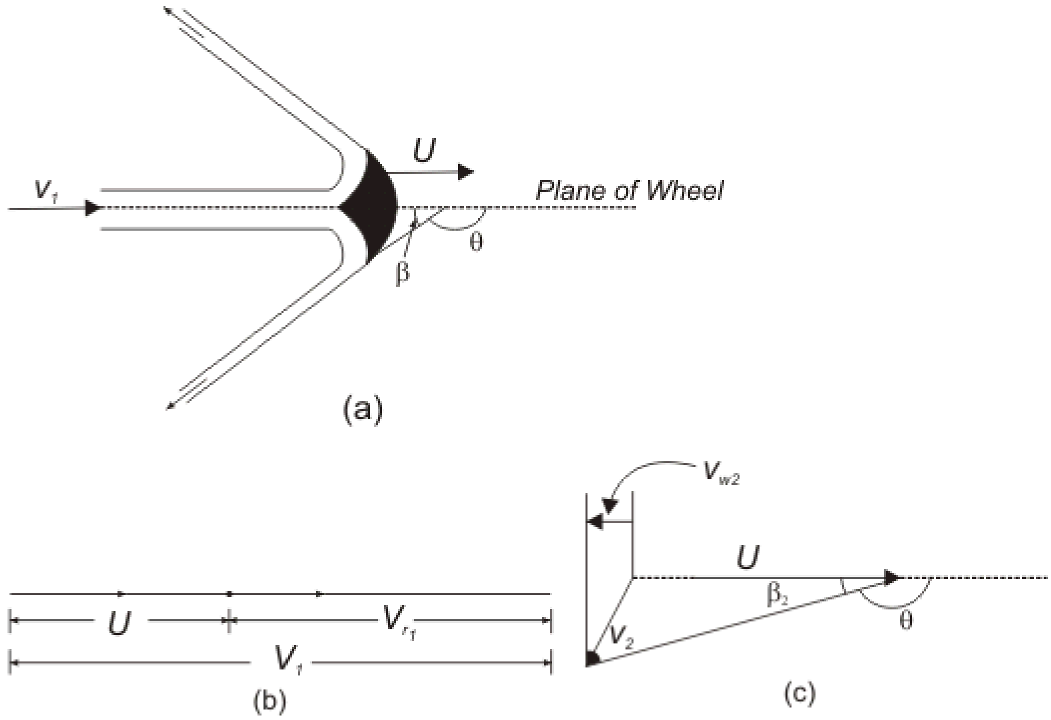


for maximum change in momentum of the fluid and hence for the maximum driving force on the wheel, the deflection of the water jet should be . In practice, however, the deflection is limited to about so that the water leaving a bucket may not hit the back of the following bucket. Therefore, the camber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

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.

Analysis of force on the bucket and power generation.shows a section through a bucket which is being acted on by a jet. The plane of section is parallel to the axis of the wheel and contains the axis of the jet. The absolute velocity of the jet with which it strikes the bucket is given by

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



(a)Flow along the bucket of a pelton wheel

(b) Inlet velocity triangle

(c)Outlet velocity triangle

where, C_v is the coefficient of velocity which takes care of the friction in the nozzle. H is the head at the entrance to the nozzle which is equal to the total or gross head of water stored at high altitudes minus the head lost due to friction in the long pipeline leading to the nozzle. Let the velocity of the bucket (due to the rotation of the wheel) at its centre where the jet strikes be U. Since the jet velocity V_1 is tangential, i.e. V_1 and U are collinear, the diagram of velocity vector at inlet (Fig

26.3.b) becomes simply a straight line and the relative velocity is given by

$$V_{r_1} = V_1 - U$$

It is assumed that the flow of fluid is uniform and it glides the blade all along including the entrance and exit sections to avoid the unnecessary losses due to shock. Therefore the direction of relative velocity at entrance and exit should match the inlet and outlet angles of the buckets respectively. The velocity triangle at the outlet is shown in Figure 26.3c. The bucket velocity *U* remains the same both at the inlet and outlet. With the direction of *U* being taken as positive, we can write. The tangential component of inlet velocity (Figure 26.3b)

$$V_{w_1} = V_1 = V_{r_1} + U$$

$$V_{w_2} = -(V_{r_2} \cos \beta_2 - U)$$

where V_{r_1} and V_{r_2} are the velocities of the jet relative to the bucket at its inlet and outlet and β_2 is the outlet angle of the bucket.

From the Eq. (1.2) (the Euler's equation for hydraulic machines), the energy delivered by the fluid per unit mass to the rotor can be written as

$$\begin{aligned} E/m &= [V_{w_1} - V_{w_2}]\,U \\ &= [V_{r_1} + V_{r_2} \cos \beta_2]\,U \end{aligned} \qquad (26.1)$$

(since, in the present situation, $U_1 = U_2 = U$)

The relative velocity V_{r_2} becomes slightly less than V_{r_1} mainly because of the friction in the bucket. Some additional loss is also inevitable as the fluid strikes the splitter ridge, because the ridge cannot have zero thickness. These losses are however kept to a minimum by making the inner surface of the bucket polished and reducing the thickness of the splitter ridge. The relative velocity at outlet V_{r_2} is usually expressed as $V_{r_2} = KV_{r_1}$ where, K is a factor with a value less than 1. However in an ideal case (in absence of friction between the fluid and blade surface) K=1. Therefore, we can write Eq.(26.1)

$$E/m = V_{r_1}[1 + K\cos\beta_2]U \qquad (26.2)$$

If Q is the volume flow rate of the jet, then the power transmitted by the fluid to the wheel can be written as

$$P = \rho Q V_{r_1}[1 + K\cos\beta_2]U$$
$$= \rho Q[1 + K\cos\beta_2](V_1 - U)U \qquad (26.3)$$

The power input to the wheel is found from the kinetic energy of the jet arriving at the wheel and is given by $\frac{1}{2}\rho Q V_1^2$. Therefore the wheel efficiency of a pelton turbine can be written as

$$\eta_w = \frac{2\rho Q[1 + K\cos\beta_2](V_1 - U)U}{\rho Q V_1^2}$$
$$= 2\,[1 + K\cos\beta_2]\left[1 - \frac{U}{V_1}\right]\frac{U}{V_1} \qquad (26.4)$$

It is found that the efficiency η_w depends on K, β_2 and U/V_1. For a given design of the bucket, i.e. for constant values of β_2 and K, the efficiency η_w becomes a function of U/V_1 only, and we can determine the condition given by U/V_1 at which η_w becomes maximum.

For η_w to be maximum,

$$\frac{d\eta_w}{d(U/V_1)} = 2[1 + K\cos\beta_2][1 - 2\frac{U}{V_1}] = 0$$

or, $$U/V_1 = \frac{1}{2} \qquad (26.5)$$

$d^2\eta_w / d(U/V_1)^2$ is always negative.

Therefore, the maximum wheel efficiency can be written after substituting the relation given by eqn.(26.5) in eqn.(26.4) as

$$\eta_{w\,max} = 2(1 - K\cos\beta_2)/2 \qquad (26.6)$$

The condition given by Eq. (26.5) states that the efficiency of the wheel in converting the kinetic energy of the jet into mechanical energy of rotation becomes maximum when the wheel speed at the centre of the bucket becomes one half of the incoming velocity of the jet. The overall efficiency η_o will be less than η_w because of friction in bearing and windage, i.e. friction between the wheel and the atmosphere in which it rotates. Moreover, as the losses due to bearing friction and windage increase rapidly with speed, the overall efficiency reaches it peak when the ratio U/V_1 is slightly less than the theoretical value of 0.5. The value usually obtained in practice is about 0.46. The Figure 27.1 shows the variation of wheel efficiency η_w with blade to jet speed ratio U/V_1 for assumed values at k=1 and 0.8, and $\beta_2 = 165^0$. An overall efficiency of 85-90 percent may usually be obtained in large machines. To obtain high values of wheel efficiency, the buckets should have smooth surface and be properly designed. The length, width, and depth of the buckets are chosen about 2.5.4 and 0.8 times the jet diameter. The buckets are notched for smooth entry of the jet.

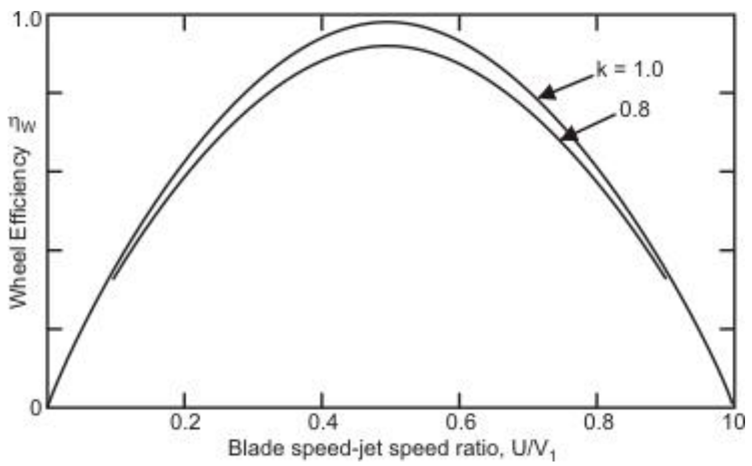


Figure 27.1 Theoretical variation of wheel efficiency for a Pelton turbine with blade speed to jet speed ratio for different values of k

Specific speed and wheel geometry . The specific speed of a pelton wheel depends on the ratio of jet diameter *d* and the wheel pitch diameter. *D* (the diameter at the centre of the bucket). If the hydraulic efficiency of a pelton wheel is defined as the ratio of the power delivered *P* to the wheel to the head available *H* at the nozzle entrance, then we can write.

$$P = \rho Q g H \eta_h = \frac{\pi \rho d^2 V_1^3 \eta_h}{4 \times 2C_v^2} \qquad (27.1)$$

Since [$Q = \frac{\pi d^2}{4} V_1$ and $V_1 = C_v (2gH)^{1/2}$]

The specific speed $N_{sT} = \frac{N P^{1/2}}{H^{5/4}}$

he optimum value of the overall efficiency of a Pelton turbine depends both on the values of the specific speed and the speed ratio. The Pelton wheels with a single jet operate in the specific speed range of 4-16, and therefore the ratio D/d lies between 6 to 26 as given by the Eq. (15.25b). A large value of D/d reduces the rpm as well as the mechanical efficiency of the wheel. It is possible to increase the specific speed by choosing a lower value of D/d, but the efficiency will decrease because of the close spacing of buckets. The value of D/d is normally kept between 14 and 16 to maintain high efficiency. The number of buckets required to maintain optimum efficiency is usually fixed by the empirical relation.

$$n(\text{number of buckets}) = 15 + \frac{53}{N_{sT}} \qquad (27.2)$$

Governing of PeltonTurbine **:** First let us discuss what is meant by governing of turbines in general. When a turbine drives an electrical generator or alternator, the primary requirement is that the rotational speed of the shaft and hence that of the turbine rotor has to be kept fixed. Otherwise the frequency of the electrical output will be altered. But when the electrical load changes depending upon the demand, the speed of the turbine changes automatically. This is because the external resisting torque on the shaft is altered while the driving torque due to change of momentum in the flow of fluid through the turbine remains the same. For example, when the load is increased, the speed of the turbine decreases and *vice versa* .A constancy in speed is therefore maintained by adjusting the rate of energy input to the turbine accordingly. This is usually accomplished by changing the rate of fluid flow through the turbine- the flow in increased when the load is increased and the flow is decreased when the load is decreased. This adjustment of flow with the load is known as the governing of turbines.

In case of a Pelton turbine, an additional requirement for its operation at the condition of maximum efficiency is that the ration of bucket to initial jet velocity U/V_1 has to be kept at its optimum value of about 0.46. Hence, when U is fixed. V_1 has to be fixed. Therefore the control must be made by a variation of the cross-sectional area, A, of the jet so that the flow rate changes in proportion to the change in the flow area keeping the jet velocity V_1 same. This is usually achieved by a spear valve in the nozzle (Figure 27.2a). Movement of the spear and the axis of the nozzle changes the annular area between the spear and the housing. The shape of the spear is such, that the fluid coalesces into a circular jet and then the effect of the spear movement is to vary the diameter of the jet. Deflectors are often used (Figure 27.2b) along with the spear valve to prevent the serious water hammer problem due to a sudden reduction in the rate of flow. These plates temporarily defect the jet so that the entire flow does not reach the bucket; the spear valve may then be moved slowly to its new position to reduce the rate of flow in the pipe-line gradually. If the bucket width is too small in relation to the jet diameter, the fluid is not smoothly deflected by the buckets and, in consequence, much energy is dissipated in turbulence and the efficiency drops considerably. On the other hand, if the buckets are unduly large, the effect of friction on the surfaces is unnecessarily high. The optimum value of the ratio of bucket width to jet diameter has been found to vary between 4 and 5.

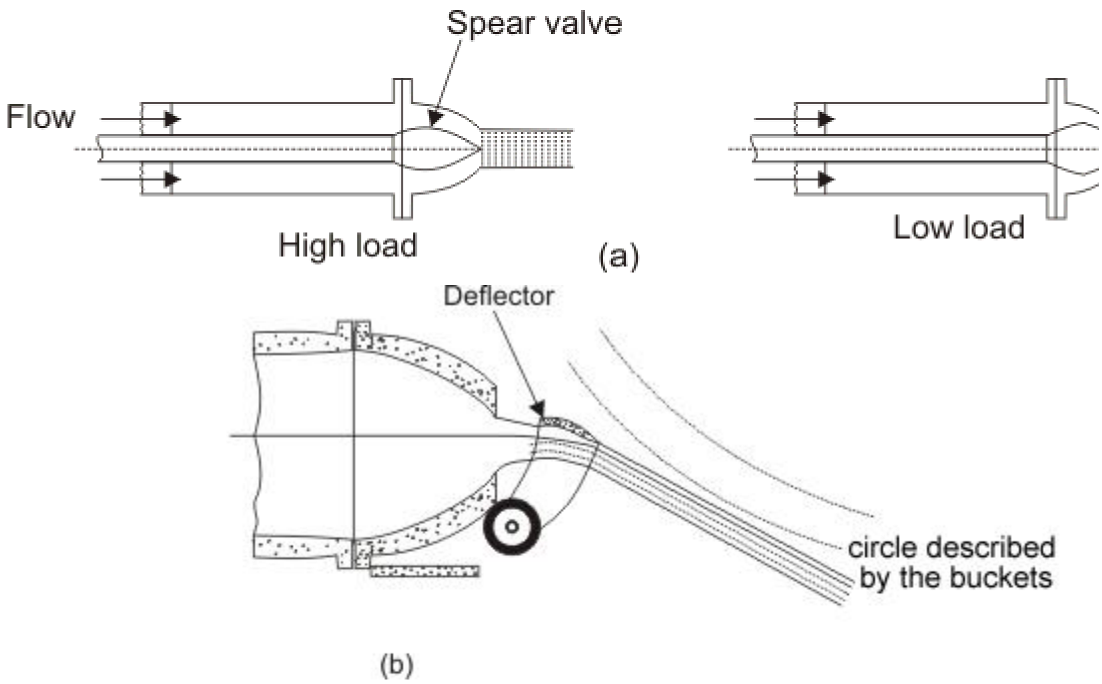


Figure 27.2 **(a) Spear valve to alter jet area in a Pelton wheel**
(b) Jet deflected from bucket

Limitation of a Pelton Turbine: The Pelton wheel is efficient and reliable when operating under large heads. To generate a given output power under a smaller head, the rate of flow through the turbine has to be higher which requires an increase in the jet diameter. The number of jets are usually limited to 4 or 6 per wheel. The increases in jet diameter in turn increases the wheel diameter. Therefore the machine becomes unduly large, bulky and slow-running. In practice, turbines of the reaction type are more suitable for lower heads.

Francis Turbine

Reaction Turbine: The principal feature of a reaction turbine that distinguishes it from an impulse turbine is that only a part of the total head available at the inlet to the turbine is converted to velocity head, before the runner is reached. Also in the reaction turbines the working fluid, instead of engaging only one or two blades, completely fills the passages in the runner. The pressure or static head of the fluid changes gradually as it passes through the runner along with the change in its kinetic energy based on absolute velocity due to the impulse action between the fluid and the runner. Therefore the cross-sectional area of flow through the passages of the fluid. A reaction turbine is usually well suited for low heads. A radial flow hydraulic turbine of reaction type was first developed by an American Engineer, James B. Francis (1815-92) and is named after him as the Francis turbine. The schematic diagram of a Francis turbine is shown in Fig. 28.1

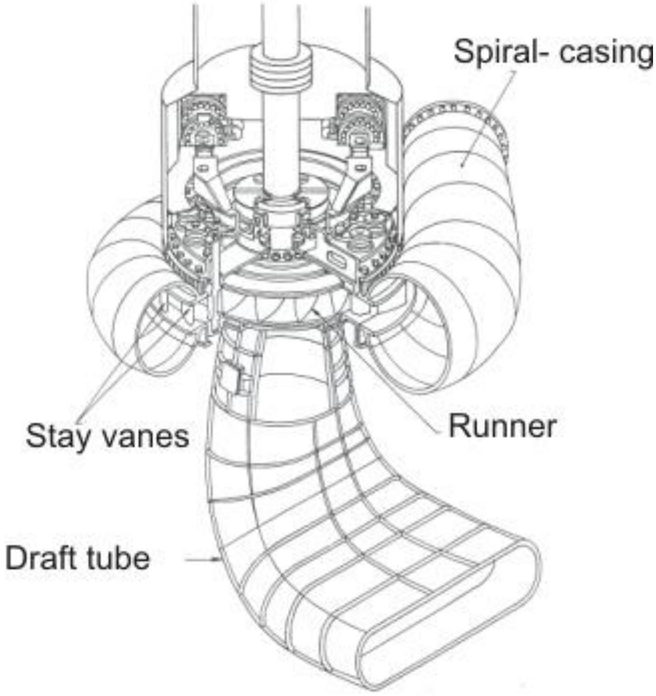


Figure 28.1 A Francis turbine

A Francis turbine comprises mainly the four components:

(i) sprical casing,

(ii) guide on stay vanes,

(iii) runner blades,

(iv) draft-tube as shown in Figure 28.1 .

Spiral Casing : Most of these machines have vertical shafts although some smaller machines of this type have horizontal shaft. The fluid enters from the penstock (pipeline leading to the turbine from the reservoir at high altitude) to a spiral casing which completely surrounds the runner. This casing is known as scroll casing or volute. The cross-sectional area of this casing decreases uniformly along the circumference to keep the fluid velocity constant in magnitude along its path towards the guide vane.

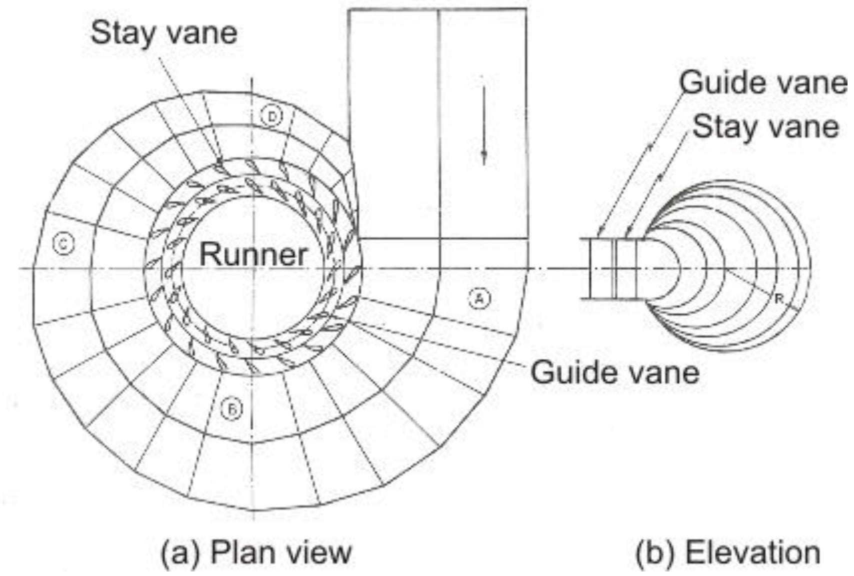


Figure 28.2 Spiral Casing

This is so because the rate of flow along the fluid path in the volute decreases due to continuous entry of the fluid to the runner through the openings of the guide vanes or stay vanes.

Guide or Stay vane:

The basic purpose of the guide vanes or stay vanes is to convert a part of pressure energy of the fluid at its entrance to the kinetic energy and then to direct the fluid on to the runner blades at the angle appropriate to the design. Moreover, the guide vanes are pivoted and can be turned by a suitable governing mechanism to regulate the flow while the load changes. The guide vanes are also known as wicket gates. The guide vanes impart a tangential velocity and hence an angular momentum to the water before its entry to the runner. The flow in the runner of a Francis turbine is not purely radial but a combination of radial and tangential. The flow is inward, i.e. from the periphery towards the centre. The height of the runner depends upon the specific speed. The height increases with the increase in the specific speed. The main direction of flow change as water passes through the runner and is finally turned into the axial direction while entering the draft tube.

Draft tube:

The draft tube is a conduit which connects the runner exit to the tail race where the water is being finally discharged from the turbine. The primary function of the draft tube is to reduce the velocity of the discharged water to minimize the loss of kinetic energy at the outlet. This permits the turbine to be set above the tail water without any appreciable drop of available head. A clear understanding of the function of the draft tube in any reaction turbine, in fact, is very important for the purpose of its design. The purpose of providing a draft tube will be better understood if we carefully study the net available head across a reaction turbine.

Net head across a reaction turbine and the purpose to providing a draft tube . The effective head across any turbine is the difference between the head at inlet to the machine and the head at outlet from it. A reaction turbine always runs completely filled with the working fluid. The tube that connects the end of the runner to the tail race is known as a draft tube and should completely to filled with the working fluid flowing through it. The kinetic energy of the fluid finally discharged into the tail race is wasted. A draft tube is made divergent so as to reduce the velocity at outlet to a minimum. Therefore a draft tube is basically a diffuser and should be designed properly with the angle between the walls of the tube to be limited to about 8 degree so as to prevent the flow separation from the wall and to reduce accordingly the loss of energy in the tube. Figure 28.3 shows a flow diagram from the reservoir via a reaction turbine to the tail race.

The total head H_1 at the entrance to the turbine can be found out by applying the Bernoulli's equation between the free surface of the reservoir and the inlet to the turbine as

$$H_0 = \frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z + h_f \qquad (28.1)$$

or,

$$H_1 = H_0 - h_f = \frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z \qquad (28.2)$$

where h_f is the head lost due to friction in the pipeline connecting the reservoir and the turbine. Since the draft tube is a part of the turbine, the net head across the turbine, for the conversion of mechanical work, is the difference of total head at inlet to the machine and the total head at discharge from the draft tube at tail race and is shown as *H* in Figure 28.3

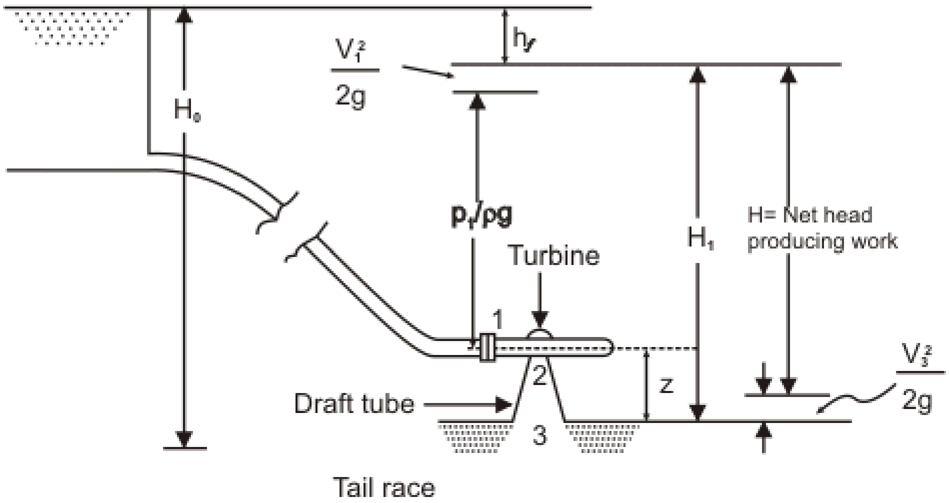


Figure 28.3 Head across a reaction turbine

Therefore, *H* = total head at inlet to machine (1) - total head at discharge (3)

$$= \frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z - \frac{V_3^2}{2g} = H_1 - \frac{V_3^2}{2g} \qquad (28.3)$$

$$= (H_0 - h_f) - \frac{V_3^2}{2g} \qquad (28.4)$$

The pressures are defined in terms of their values above the atmospheric pressure. Section 2 and 3 in Figure 28.3 represent the exits from the runner and the draft tube respectively. If the losses in the draft tube are neglected, then the total head at 2 becomes equal to that at 3. Therefore, the net head across the machine is either $(H_1 - H_3)$ or $(H_1 - H_2)$. Applying the Bernoull's equation between 2 and 3 in consideration of flow, without losses, through the draft tube, we can write.

$$\frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z = 0 + \frac{V_3^2}{2g} + 0 \qquad (28.5)$$

$$\frac{P_2}{\rho g} = -\left[z + \frac{V_2^2 - V_3^2}{2g}\right] \qquad (28.6)$$

Since $V_3 < V_2$, both the terms in the bracket are positive and hence $p_2/\rho g$ is always negative, which implies that the static pressure at the outlet of the runner is always below the atmospheric pressure. Equation (28.1) also shows that the value of the suction pressure at runner outlet depends on z, the height of the runner above the tail race and $(V_2^2 - V_3^2)/2g$, the decrease in kinetic energy of the fluid in the draft tube. The value of this minimum pressure p_2 should never fall below the vapour pressure of the liquid at its operating temperature to avoid the problem of cavitation. Therefore, we fine that the incorporation of a draft tube allows the turbine runner to be set above the tail race without any drop of available head by maintaining a vacuum pressure at the outlet of the runner.

Runner of the Francis Turbine

The shape of the blades of a Francis runner is complex. The exact shape depends on its specific speed. It is obvious from the equation of specific speed that higher specific speed means lower head. This requires that the runner should admit a comparatively large quantity of water for a given power output and at the same time the velocity of discharge at runner outlet should be small to avoid cavitation. In a purely radial flow runner, as developed by James B. Francis, the bulk flow is in the radial direction. To be more clear, the flow is tangential and radial at the inlet but is entirely radial with a negligible tangential component at the outlet. The flow, under the situation, has to make a 90^o turn after passing through the rotor for its inlet to the draft tube. Since the flow area (area perpendicular to the radial direction) is small, there is a limit to the capacity of this type of runner in keeping a low exit velocity. This leads to the design of a mixed flow runner where water is turned from a radial to an axial direction in the rotor itself. At the outlet of this type of runner, the flow is mostly axial with negligible radial and tangential components. Because of a large discharge area (area perpendicular to the axial direction), this type of runner can pass a large amount of water with a low exit velocity from the runner. The blades for a reaction turbine are always so shaped that the tangential or whirling component of velocity at the outlet becomes zero $(V_{w_2} = 0)$. This is made to keep the kinetic energy at outlet a minimum.

Figure 29.1 shows the velocity triangles at inlet and outlet of a typical blade of a Francis turbine. Usually the flow velocity (velocity perpendicular to the tangential direction) remains constant throughout, i.e. $V_{f_1} = V_{f_2}$ and is equal to that at the inlet to the draft tube.

The Euler's equation for turbine [Eq.(1.2)] in this case reduces to

$$E/m = e = V_{w_1} U_1 \qquad (29.1)$$

where, *e* is the energy transfer to the rotor per unit mass of the fluid. From the inlet velocity triangle shown in Fig. 29.1

$$V_{w_1} = V_{f_1} \cot \alpha_1 \qquad (29.2a)$$

and $$U_1 = V_{f_1} (\cot \alpha_1 + \cot \beta_1) \qquad (29.2b)$$

Substituting the values of V_{w_1} and U_1 from Eqs. (29.2a) and (29.2b) respectively into Eq. (29.1), we have

$$e = V_{f_1}^2 \cot \alpha_1 (\cot \alpha_1 + \cot \beta_1) \qquad (29.3)$$

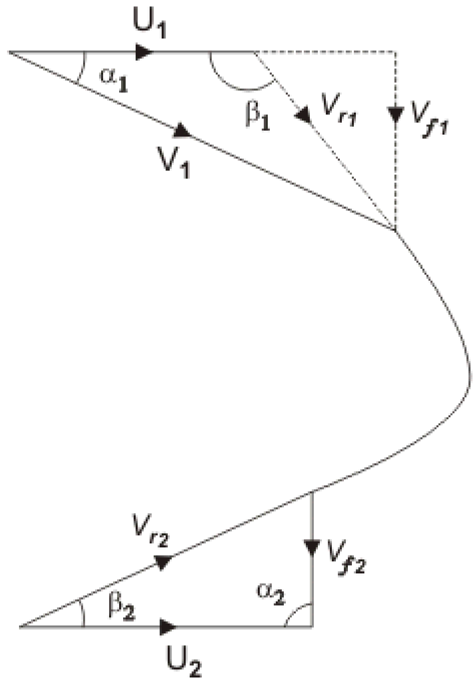


Figure 29.1 Velocity triangle for a Francis runner

The loss of kinetic energy per unit mass becomes equal to $V_{f_2}^2/2$. Therefore neglecting friction, the blade efficiency becomes

$$\eta_b = \frac{e}{e+(V_{f_2}^2/2)}$$

$$= \frac{2V_{f_1}^2 \cot\alpha_1(\cot\alpha_1 + \cot\beta_1)}{V_{f_2}^2 + 2V_{f_1}^2 \cot\alpha_1(\cot\alpha_1 + \cot\beta_1)}$$

since $V_{f_1} = V_{f_2}, \eta_b$ can be written as

$$\eta_b = 1 - \frac{1}{1+2\cot\alpha_1(\cot\alpha_1 + \cot\beta_1)}$$

The change in pressure energy of the fluid in the rotor can be found out by subtracting the change in its kinetic energy from the total energy released. Therefore, we can write for the degree of reaction.

$$R = \frac{e - \frac{1}{2}\left(V_1^2 - V_{f_2}^2\right)}{e} = 1 - \frac{\frac{1}{2}V_{f_1}^2 \cot^2\alpha_1}{e}$$

[since $V_1^2 - V_{f_2}^2 = V_1^2 - V_{f_1}^2 = V_{f_1}^2 \cot^2\alpha_1$]

Using the expression of *e* from Eq. (29.3), we have

$$R = 1 - \frac{\cot\alpha_1}{2(\cot\alpha_1 + \cot\beta_1)} \qquad (29.4)$$

The inlet blade angle β_1 of a Francis runner varies $45-120^o$ and the guide vane angle angle α_1 from $10-40^o$. The ratio of blade width to the diameter of runner B/D, at blade inlet, depends upon the required specific speed and varies from 1/20 to 2/3.

Expression for specific speed. The dimensional specific speed of a turbine, can be written as

$$N_{s_T} = \frac{NP^{1/2}}{H^{5/4}}$$

Power generated P for a turbine can be expressed in terms of available head H and hydraulic efficiency η_h as

$$P = \rho Q g H \eta_h$$

Hence, it becomes

$$N_{s_T} = N(\rho Q g \eta_h)^{1/2} H^{-3/4} \qquad (29.5)$$

Again, $N = U_1 / \pi D_1$,

Substituting U_1 from Eq. (29.2b)

$$N = \frac{V_{f_1}(\cot \alpha_1 + \cot \beta_1)}{\pi D_1} \qquad (29.6)$$

Available head H equals the head delivered by the turbine plus the head lost at the exit. Thus,

$$gH = e + (V_{f_2}^2 / 2)$$

since $$V_{f_1} = V_{f_2}$$

$$gH = e + (V_{f_1}^2 / 2)$$

with the help of Eq. (29.3), it becomes

$$gH = V_{f_1}^2 \cot \alpha_1 (\cot \alpha_1 + \cot \beta_1) + \frac{V_{f_1}^2}{2}$$

or, $$H = \frac{V_{f_1}^2}{2g}[1 + 2\cot\alpha_1(\cot\alpha_1 + \cot\beta_1)] \qquad (29.7)$$

Substituting the values of H and N from Eqs (29.7) and (29.6) respectively into the expression N_{s_T} given by Eq. (29.5), we get,

$$N_{s_T} = 2^{3/4} g^{5/4} (\rho \eta_h Q)^{1/2} \frac{V_{f_1}^{-1/2}}{\pi D_1} (\cot\alpha_1 + \cot\beta_1)[1 + 2\cot\alpha_1(\cot\alpha_1 + \cot\beta_1)^{-3}$$

Flow velocity at inlet V_{f_1} can be substituted from the equation of continuity as

$$V_{f_1} = \frac{Q}{\pi D_1 B}$$

where B is the width of the runner at its inlet

Finally, the expression for N_{s_T} becomes,

$$N_{s_T} = 2^{3/4} g^{5/4} (\rho\eta_h)^{1/2} \left(\frac{B}{\pi D_1}\right)^{1/2} (\cot\alpha_1 + \cot\beta_1)$$
$$[1 + 2\cot\alpha_1\ (\cot\alpha_1 + \cot\beta_1))^{-3/4} \qquad (29.8)$$

For a Francis turbine, the variations of geometrical parameters like $\alpha_1, \beta_1\ B/D$ have been described earlier. These variations cover a range of specific speed between 50 and 400. Figure 29.2 shows an overview of a Francis Turbine. The figure is specifically shown in order to convey the size and relative dimensions of a typical Francis Turbine to the readers.

KAPLAN TURBINE

Introduction

Higher specific speed corresponds to a lower head. This requires that the runner should admit a comparatively large quantity of water. For a runner of given diameter, the maximum flow rate is achieved when the flow is parallel to the axis. Such a machine is known as axial flow reaction turbine. An Australian engineer, Vikton Kaplan first designed such a machine. The machines in this family are called Kaplan Turbines.(Figure 30.1)

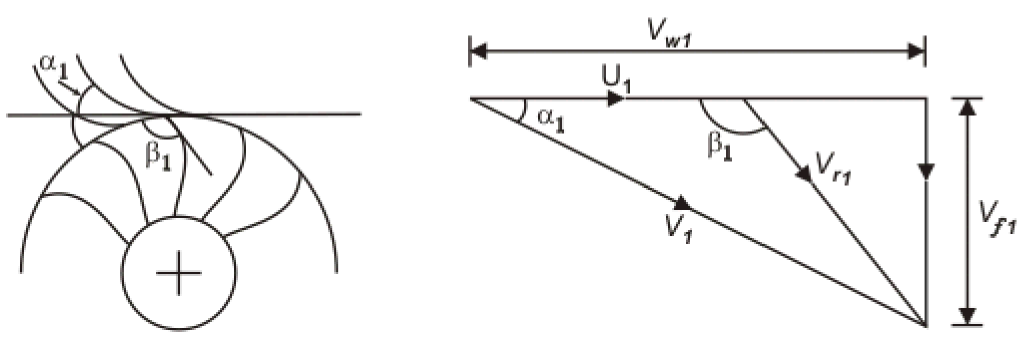
Figure 30.1 A typical Kaplan Turbine

Development of Kaplan Runner from the Change in the Shape of Francis Runner with Specific Speed

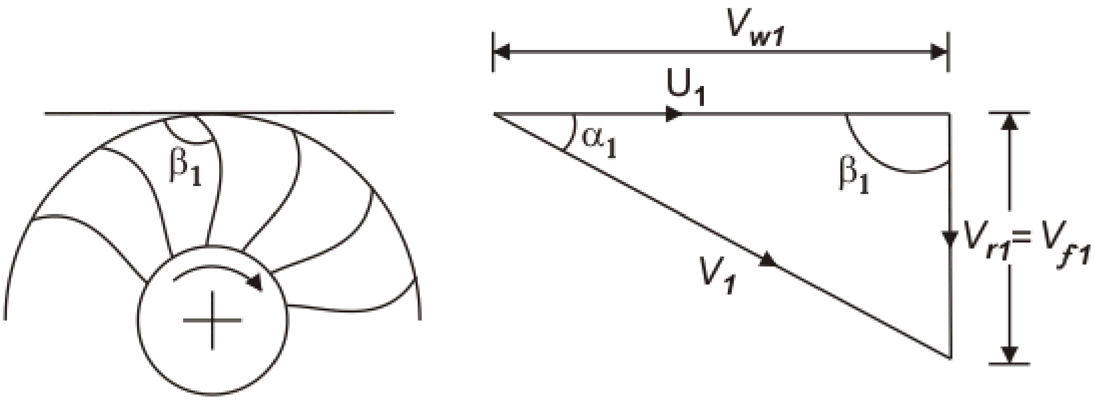
Figure 30.2 shows in stages the change in the shape of a Francis runner with the variation of specific speed. The first three types [Fig. 30.2 (a), (b) and (c)] have, in order. The Francis runner (radial flow runner) at low, normal and high specific speeds. As the specific speed increases, discharge becomes more and more axial. The fourth type, as shown in Fig.30.2 (d), is a mixed flow runner (radial flow at inlet axial flow at outlet) and is known as Dubs runner which is mainly suited for high specific speeds. Figure 30.2(e) shows a propeller type runner with a less number of blades where the flow is entirely axial (both at inlet and outlet). This type of runner is the most suitable one for very high specific speeds and is known as Kaplan runner or axial flow runner.

From the inlet velocity triangle for each of the five runners, as shown in Figs (30.2a to 30.2e), it is found that an increase in specific speed (or a decreased in head) is accompanied by a reduction in inlet velocity V_1. But the flow velocity V_{f1} at inlet increases allowing a large amount of fluid to enter the turbine. The most important point to be noted in this context is that the flow at inlet to all the runners, except the Kaplan one, is in radial and tangential directions. Therefore, the inlet velocity triangles of those turbines (Figure 30.2a to 30.2d) are shown in a plane containing the radial ant tangential directions, and hence the flow velocity V_{f1} represents the radial component of velocity.

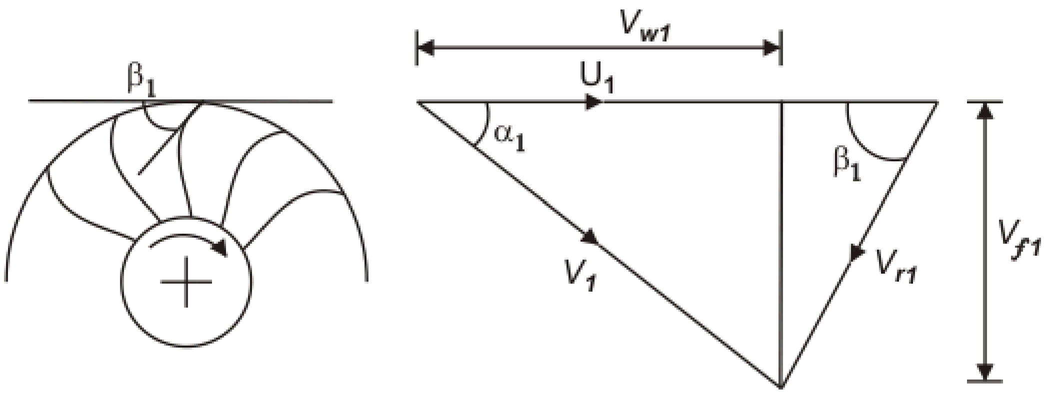
In case of a Kaplan runner, the flow at inlet is in axial and tangential directions. Therefore, the inlet velocity triangle in this case (Figure 30.2e) is shown in a place containing the axial and tangential directions, and hence the flow velocity V_{f1} represents the axial component of velocity V_a .The tangential component of velocity is almost nil at outlet of all runners. Therefore, the outlet velocity triangle (Figure 30.2f) is identical in shape of all runners. However, the exit velocity V_2 is axial in Kaplan and Dubs runner, while it is the radial one in all other runners.



(a) Francis runner for low specific speeds

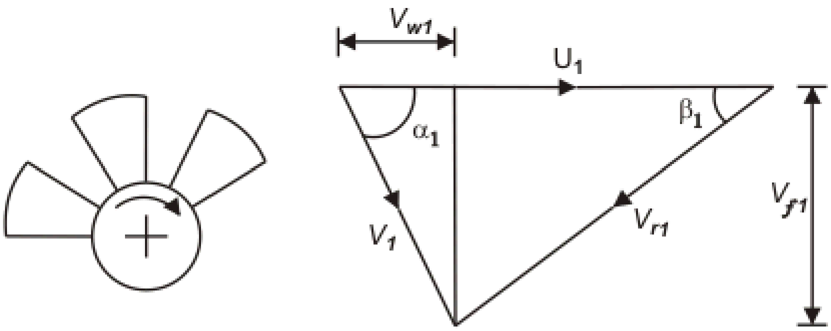


(b) Francis runner for normal specific speeds

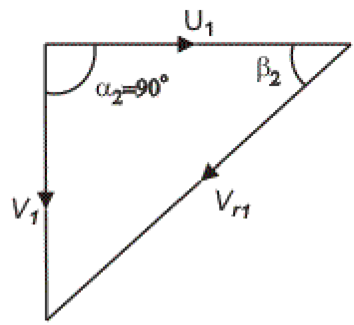


(c) Francis runner for high specific speeds

(d) Dubs runner



(e) Kalpan runner



(f) For allreaction (Francis as well as Kaplan) runners

Outlet velocity triangle

Fig. 30.2 Evolution of Kaplan runner form Francis one

Figure 30.3 shows a schematic diagram of propeller or Kaplan turbine. The function of the guide vane is same as in case of Francis turbine. Between the guide vanes and the runner, the fluid in a propeller turbine turns through a right-angle into the axial direction and then passes through the runner. The runner usually has four or six blades and closely resembles a ship's propeller. Neglecting the frictional effects, the flow approaching the runner blades can be considered to be a free vortex with whirl velocity being inversely proportional to radius, while on the other hand, the blade velocity is directly proportional to the radius. To take care of this different relationship of the fluid velocity and the blade velocity with the changes in radius, the blades are twisted. The angle with axis is greater at the tip that at the root.

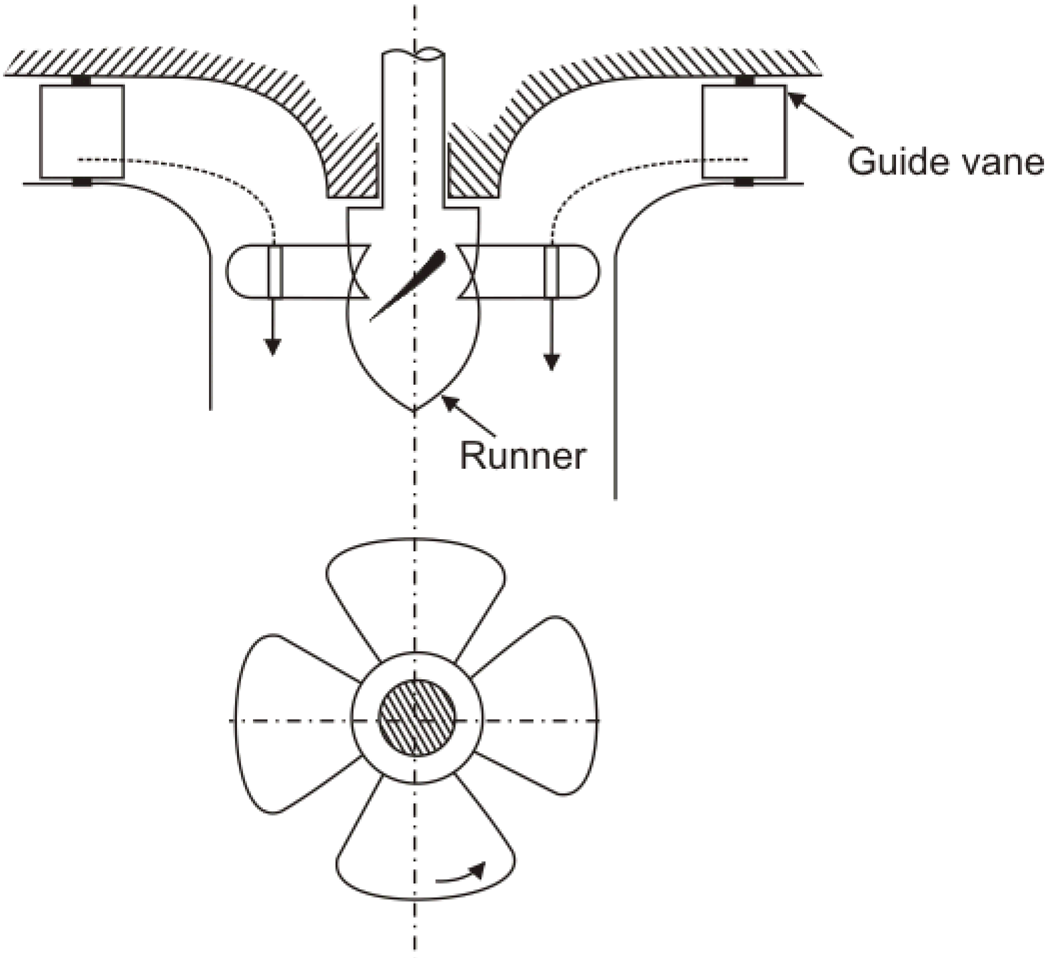


Fig. 30.3 A propeller of Kaplan turbine

Different types of draft tubes incorporated in reaction turbines The draft tube is an integral part of a reaction turbine. Its principle has been explained earlier. The shape of draft tube plays an important role especially for high specific speed turbines, since the efficient recovery of kinetic energy at runner outlet depends mainly on it. Typical draft tubes, employed in practice, are discussed as follows.

Straight divergent tube [Fig. 30.4(a)] The shape of this tube is that of frustum of a cone. It is usually employed for low specific speed, vertical shaft Francis turbine. The cone angle is restricted to 8 0 to avoid the losses due to separation. The tube must discharge sufficiently low under tail water level. The maximum efficiency of this type of draft tube is 90%. This type of draft tube improves speed regulation of falling load.

Simple elbow type (Fig. 30.4b) The vertical length of the draft tube should be made small in order to keep down the cost of excavation, particularly in rock. The exit diameter of draft tube should be as large as possible to recover kinetic energy at runner's outlet. The cone angle of the tube is again fixed from the consideration of losses due to flow separation. Therefore, the draft tube must be bent to keep its definite length. Simple elbow type draft tube will serve such a purpose. Its efficiency is, however, low(about 60%). This type of draft tube turns the water from the vertical to the horizontal direction with a minimum depth of excavation. Sometimes, the transition from a circular section in the vertical portion to a rectangular section in the horizontal part (Fig. 30.4c) is incorporated in the design to have a higher efficiency of the draft tube. The horizontal portion of the draft tube is generally inclined upwards to lead the water gradually to the level of the tail race and to prevent entry of air from the exit end.

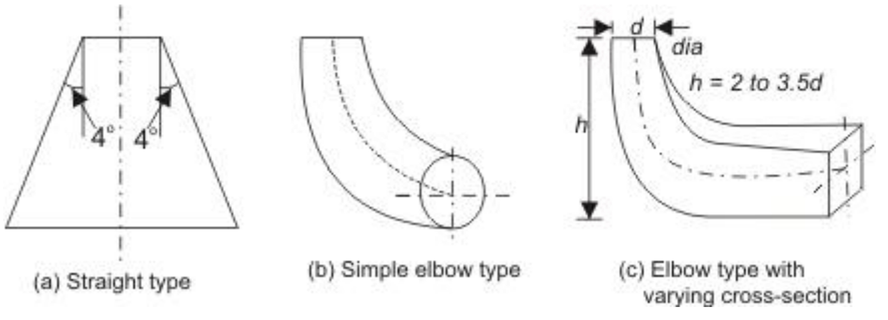


Figure 30.4 Different types of draft tubes

Cavitation in reaction turbines

If the pressure of a liquid in course of its flow becomes equal to its vapour pressure at the existing temperature, then the liquid starts boiling and the pockets of vapour are formed which create vapour locks to the flow and the flow is stopped. The phenomenon is known as cavitation. To avoid cavitation, the minimum pressure in the passage of a liquid flow, should always be more than the vapour pressure of the liquid at the working temperature. In a reaction turbine, the point of minimum pressure is usually at the outlet end of the runner blades, i.e at the inlet to the draft tube. For the flow between such a point and the final discharge into the trail race (where the pressure is atmospheric), the Bernoulli's equation can be written, in consideration of the velocity at the discharge from draft tube to be negligibly small, as

$$\frac{p_e}{pg} + \frac{V_e^2}{2g} + z = \frac{p_{atm}}{\rho g} + h_f \qquad (31.1)$$

where, p_e and V_e represent the static pressure and velocity of the liquid at the outlet of the runner (or at the inlet to the draft tube). The larger the value of V_e, the smaller is the value of p_e and the cavitation is more likely to occur. The term h_f in Eq. (31.1) represents the loss of head due to friction in the draft tube and z is the height of the turbine runner above the tail water surface. For cavitation not to occur $p_e > p_v$ where p_v is the vapour pressure of the liquid at the working temperature.

An important parameter in the context of cavitation is the available suction head (inclusive of both static and dynamic heads) at exit from the turbine and is usually referred to as the net positive suction head 'NPSH' which is defined as

$$NPSH = \frac{p_e}{\rho g} + \frac{V_e^2}{2g} - \frac{p_v}{\rho g} \qquad (31.2)$$

with the help of Eq. (31.1) and in consideration of negligible frictional losses in the draft tube $(hf = 0)$, Eq. (31.2) can be written as

$$NPSH = \frac{p_{atm}}{\rho g} - \frac{p_v}{\rho g} - z \qquad (31.3)$$

A useful design parameter σ known as Thoma'sCavitation Parameter (after the German Engineer Dietrich Thoma, who first introduced the concept) is defined as

$$\sigma = \frac{NPSH}{H} = \frac{(p_{atm}/\rho g) - (p_v/\rho g) - z}{H} \qquad (31.4)$$

For a given machine, operating at its design condition, another useful parameter σ_c, known as critical cavitaion parameter is define as

$$\sigma_c = \frac{(p_{atm}/\rho g) - (p_e/\rho g) - z}{H} \qquad (31.5)$$

Therefore, for cavitaion not to occur $\sigma > \sigma_c$ (since, $p_e > p_v$).

If either *z* or *H* is increased, σ is reduced. To determine whether cavitation is likely to occur in a particular installation, the value σ of may be calculated. When the value of σ is greater than the value of σ_c for a particular design of turbine cavitation is not expected to occur.

In practice, the value of σ_c is used to determine the maximum elevation of the turbine above tail water surface for cavitation to be avoided. The parameter of increases with an increase in the specific speed of the turbine. Hence, turbines having higher specific speed must be installed closer to the tail water level.

Performance Characteristics of Reaction Turbine

It is not always possible in practice, although desirable, to run a machine at its maximum efficiency due to changes in operating parameters. Therefore, it becomes important to know the performance of the machine under conditions for which the efficiency is less than the maximum. It is more useful to plot the basic dimensionless performance parameters (Fig. 31.1) as derived earlier from the similarity principles of fluid machines. Thus one set of curves, as shown in Fig. 31.1, is applicable not just to the conditions of the test, but to any machine in the same homologous series under any altered conditions.

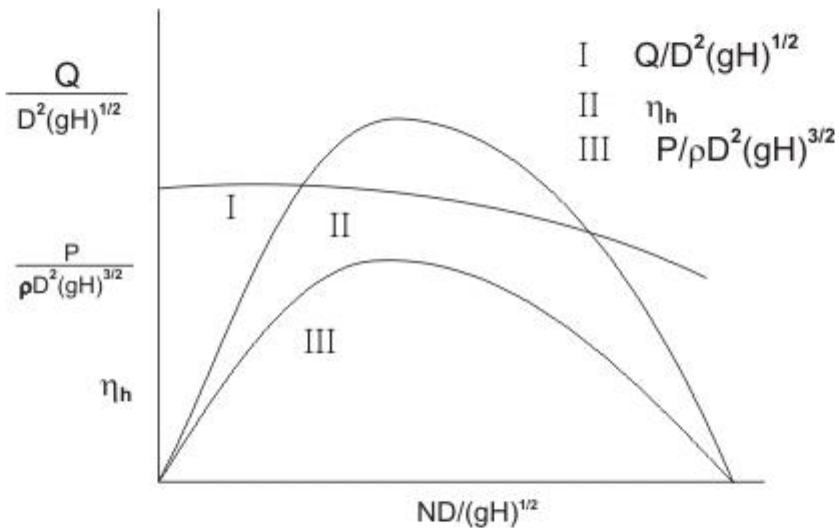


Figure 31.1 performance characteristics of a reaction turbine (in dimensionless parameters)

Figure 31.2 is one of the typical plots where variation in efficiency of different reaction turbines with the rated power is shown.

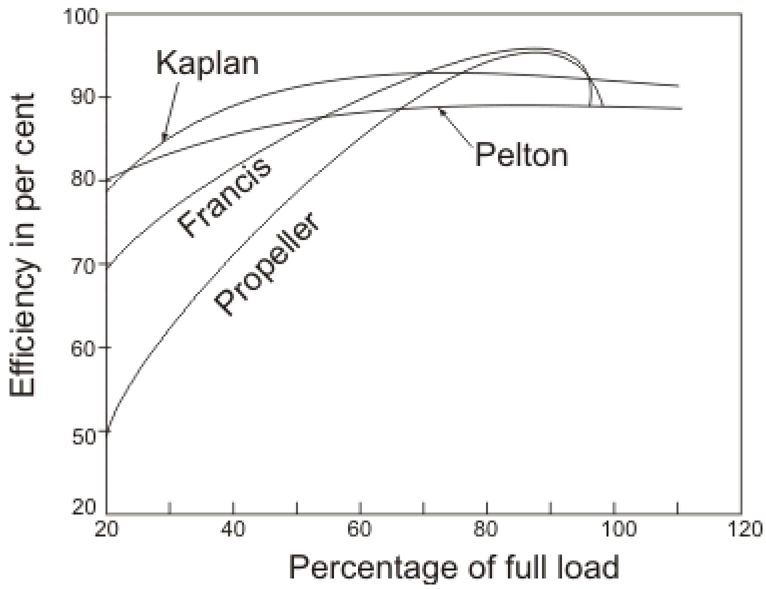


Figure 31.2 Variation of efficiency with load

Comparison of Specific Speeds of Hydraulic Turbines

Specific speeds and their ranges of variation for different types of hydraulic turbines have already been discussed earlier. Figure 32.1 shows the variation of efficiencies with the dimensionless specific speed of different hydraulic turbines. The choice of a hydraulic turbine for a given purpose depends upon the matching of its specific speed corresponding to maximum efficiency with the required specific speed determined from the operating parameters, namely, N (rotational speed), p (power) and H (available head).

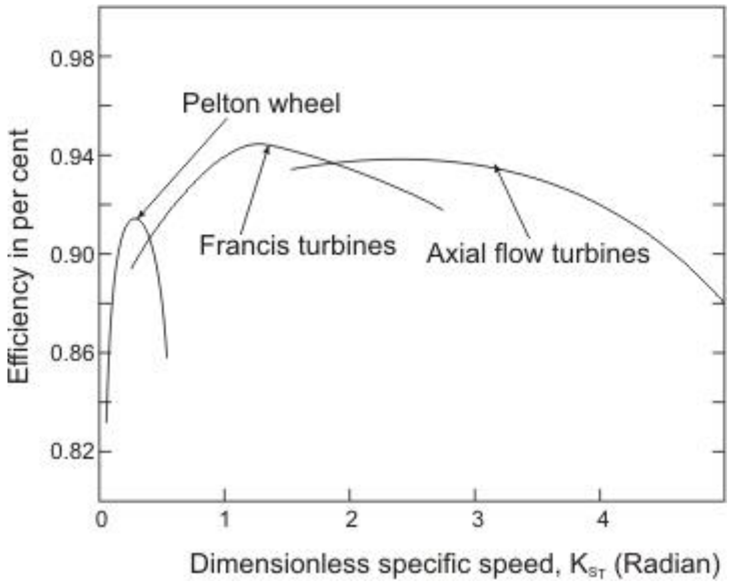


Figure 32.1 Variation of efficiency with specific speed for hydraulic turbines

Governing of Reaction Turbines Governing of reaction turbines is usually done by altering the position of the guide vanes and thus controlling the flow rate by changing the gate openings to the runner. The guide blades of a reaction turbine (Figure 32.2) are pivoted and connected by levers and links to the regulating ring. Two long regulating rods, being attached to the regulating ring at their one ends, are connected to a regulating lever at their other ends. The regulating lever is keyed to a regulating shaft which is turned by a servomotor piston of the oil

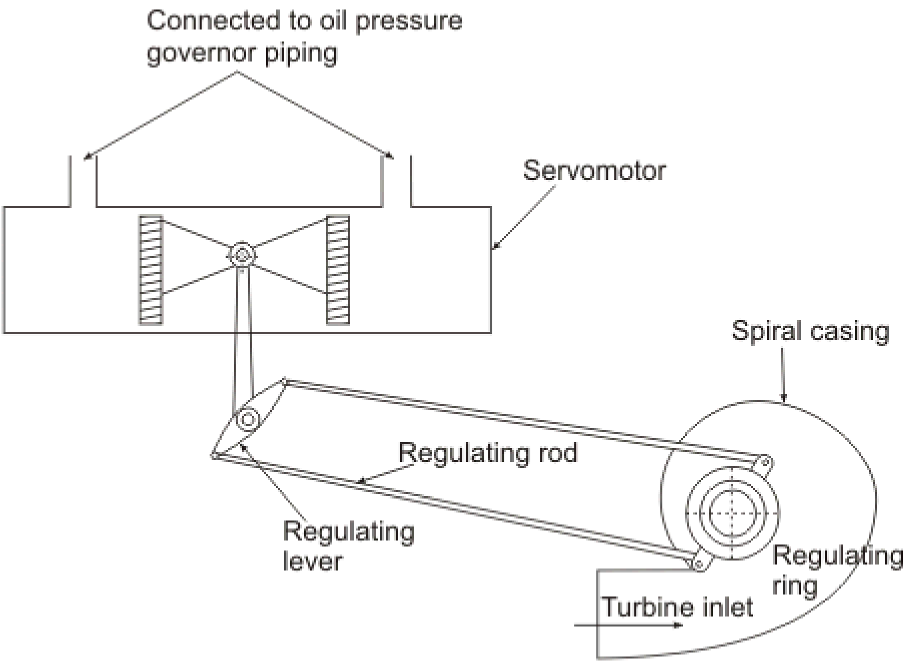


Figure 32.2 Governing of reaction turbine

EXERCISE

1) A quarter scale turbine model is tested under a head of 10.8m. The full-scale turbine is required to work under a head of 30 m and to run at 7.14 rev/s. At what speed must the model be run? If it develops 100 kW and uses 1.085 m^3 of water per second at the speed, what power will be obtained from the full scale turbine? The efficiency of the full-scale turbine being 3% greater than that of the model? What is the dimensionless specific speed of the full-scale turbine?

(Ans.17.14 rev/s,7.66MW,0.513 rev/s)

2) A Pelton wheel operates with a jet of 150mm diameter under the head of 500m. Its mean runner diameter is 2.25 m and and it rotates with speed of 375 rpm. The angle of bucket tip at outlet as 15^o coefficient of velocity is 0.98, mechanical losses equal to 3% of power supplied and the reduction in relative velocity of water while passing through bucket is 15%. Find (a) the force of jet on the bucket, (b) the power developed (c) bucket efficiency and (d) the overall efficiency.

(Ans. 165.15kN,7.3MW,90.3%,87.6%)

3) A pelton wheel works at the foot of a dam because of which the head available at the nozzle is 400m. The nozzle diameter is 160mm and the coefficient of velocity is 0.98. The diameter of the wheel bucket circle is 1.75 m and the buckets deflect the jet by 150^o. The wheel to jet speed ratio is 0.46. Neglecting friction, calculate (a) the power developed by the turbine, (b) its speed and (c) hydraulic efficiency.

[Ans. (a) 6.08 MW,(b) 435.9rpm,(c) 89.05%]

4) A Powerhouse is equipped with impulse turbines of Pelton type. Each turbine delivers a power of 14 MW when working under a head 900 m and running at 600 rpm. Find the diameter of the jet and mean diameter of the wheel. Assume that the overall efficiency is 89%, velocity coefficient of jet 0.98, and speed ratio 0.46.

(Ans.132mm. 191m)

5) A Francis turbine has a wheel diameter of 1.2 m at the entrance and 0.6m at the exit. The blade angle at the entrance is 90^0 and the guide vane angle is 15^0. The water at the exit leaves the blades without any tangential velocity. The available head is 30m and the radial component of flow velocity is constant. What would be the speed of wheel in rpm and blade angle at exit? Neglect friction.

(Ans. 268 rpm, 28.2^0)

6) In a vertical shaft inward-flow reaction turbine, the sum of the pressure and kinetic head at entrance to the spiral casing is 120 m and the vertical distance between this section and the tail race level is 3 m. The peripheral velocity of the runner at entry is 30m/s, the radial velocity of water is constant at 9m/s and discharge from the runner is without swirl. The estimated hydraulic losses are (a) between turbine entrance and exit from the guide vanes 4.8 m(b) in the runner 8.8m (c) in the draft tube 0.79 m (d) kinetic head rejected to the tail race 0.46m. Calculate the guide vane angle and the runner blade angle at inlet and the pressure heads at entry to and exit from the runner.

(Ans.14.28^0 , 59.22^0 , 47.34m, -5.88m)

Pumps

Rotodynamic Pumps

A rotodynamic pump is a device where mechanical energy is transferred from the rotor to the fluid by the principle of fluid motion through it. The energy of the fluid can be sensed from the pressur and velocity of the fluid at the delivery end of the pump. Therefore, it is essentially a turbine in reverse. Like turbines, pumps are classified according to the main direction of fluid path through them like (i) radial flow or centrifugal, (ii) axial flow and (iii) mixed flow types.

Centrifugal Pumps

The pumps employing centrifugal effects for increasing fluid pressure have been in use for more than a century.The centrifugal pump, by its principle, is converse of the Francis turbine. The flow is radially outward, and the hence the fluid gains in centrifugal head while flowing through it. Because of certain inherent advantages,such as compactness, smooth and uniform flow, low initial cost and high efficiency even at low heads, centrifugal pumps are used in almost all pumping systems. However, before considering the operation of a pump in detail, a general pumping system is discussed as follows.

General Pumping System and the Net Head Developed by a Pump

The word pumping, referred to a hydraulic system commonly implies to convey liquid from a low to a high reservoir. Such a pumping system, in general, is shown in Fig. 33.1. At any point in the system, the elevation or potential head is measured from a fixed reference datum line. The total head at any point comprises pressure head, velocity head and elevation head. For the lower reservoir, the total head at the free surface is H_A and is equal to the elevation of the free surface above the datum line since the velocity and static pressure at *A* are zero. Similarly the total head at the free surface in the higher reservoir is ($H_A + H_S$) and is equal to the elevation of the free surface of the reservoir above the reference datum.

The variation of total head as the liquid flows through the system is shown in Fig. 33.2. The liquid enters the intake pipe causing a head loss h_{in} for which the total energy line drops to point *B* corresponding to a location just after the entrance to intake pipe. The total head at *B* can be written as

$$H_B = H_A - h_{in}$$

As the fluid flows from the intake to the inlet flange of the pump at elevation z_1 the total head drops further to the point C (Figure 33.2) due to pipe friction and other losses equivalent to h_{f1} . The fluid then enters the pump and gains energy imparted by the moving rotor of the pump. This raises the total head of the fluid to a point D (Figure 33.2) at the pump outlet (Figure 33.1).

In course of flow from the pump outlet to the upper reservoir, friction and other losses account for a total head loss or h_{f2} down to a point *E* . At *E* an exit loss h_e occurs when the liquid enters the upper reservoir, bringing the total heat at point *F* (Figure 33.2) to that at the free surface of the upper reservoir. If the total heads are measured at the inlet and outlet flanges respectively, as done in a standard pump test, then

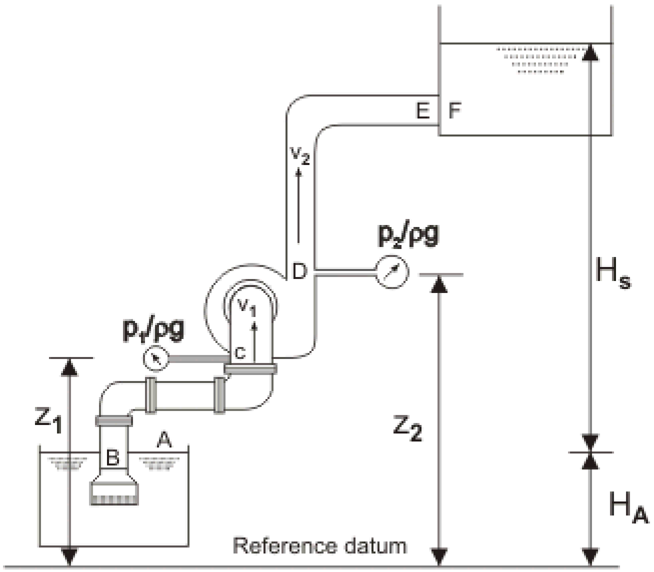


Figure 33.1 A general pumping system

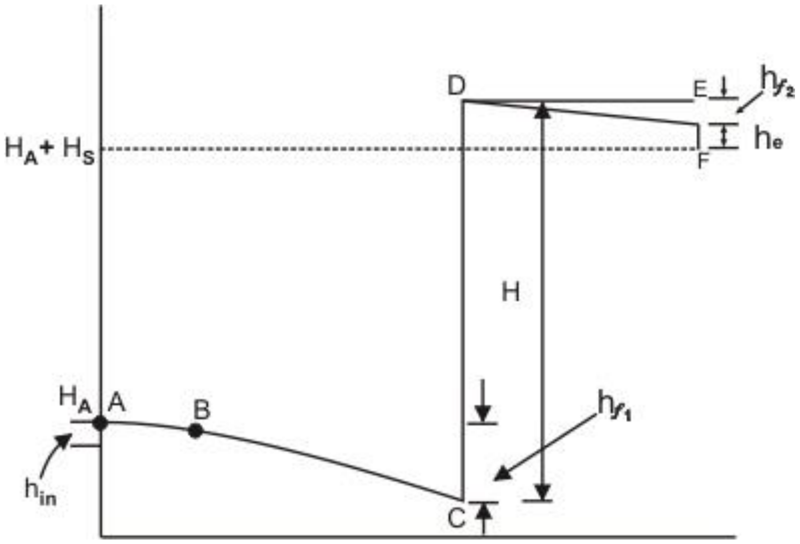


Figure 33.2 Change of head in a pumping system

Total inlet head to the pump = $(p_1 + \rho g) + (V_1^2 / 2g) + z_1$

Total outlet head of the pump = $(p_2 + \rho g) + (V_2^2 / 2g) + z_2$

where V_1 and V_2 are the velocities in suction and delivery pipes respectively.

Therefore, the total head developed by the pump,

$$H = [(p_2 - p_1) / \rho g] + [(V_2^2 - V_1^2) / 2g] + [z_2 - z_1] \qquad (33.1)$$

The head developed H is termed as *manometrichead* . If the pipes connected to inlet and outlet of the pump are of same diameter, $V_2 = V_1$ and therefore the head developed or manometric head H is simply the gain in piezometric pressure head across the pump which could have been recorded by a manometer connected between the inlet and outlet flanges of the pump. In practice, ($z_2 - z_1$) is so small in comparison to $(p_2 - p_1) / \rho g$ that it is ignored. It is therefore not surprising o find that the static pressure head across the pump is often used to describe the total head developed by the pump. The vertical distance between the two levels in the reservoirs H_s is known as static head or static lift. Relationship between H_s, the static head and H, the head developed can be found out by applying Bernoulli's equation between *A* and *C* and between *D* and *F* (Figure 33.1) as follows:

$$0 + 0 + H_A = \frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 + h_{in} + h_{f_1} \qquad (33.2)$$

Between *D* and *F* ,

$$\frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2 = 0 + 0 + H_s + H_A + h_{f2} + h_e \qquad (33.3)$$

substituting H_A from Eq. (33.2) into Eq. (33.3), and then with the help of Eq. (33.1),

we can write

$$H = H_s + h_{in} + h_{f1} + h_{f2} + h_e$$
$$= H_s + \sum \text{losses} \qquad (33.4)$$

Therefore, we have, the total head developed by the pump = static head + sum of all the losses.

The simplest from of a centrifugal pump is shown in Figure 33.3. It consists of three important parts: (i) the rotor, usually called as impeller, (ii) the volute casing and (iii) the diffuser ring. The impeller is a rotating solid disc with curved blades standing out vertically from the face of the disc. The impeller may be single sided (Figure 33.4a) or doublesided (Figure 33.4b). A double sided impeller has a relatively small flow capacity.

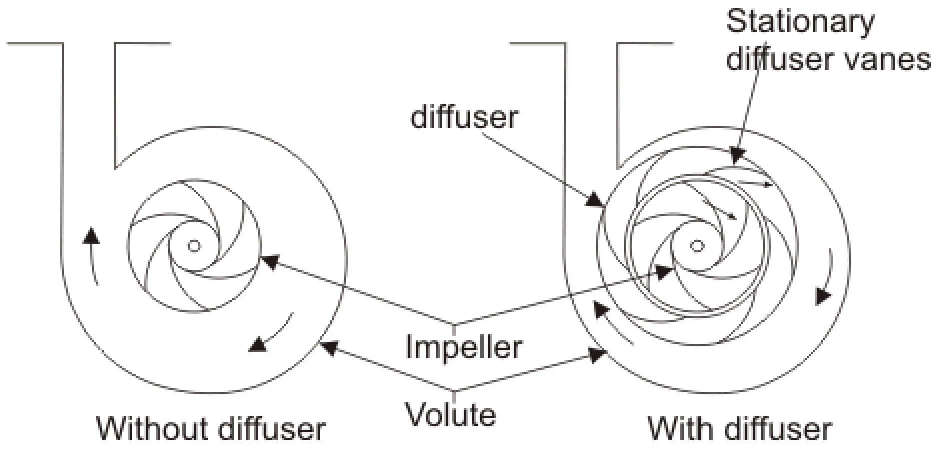
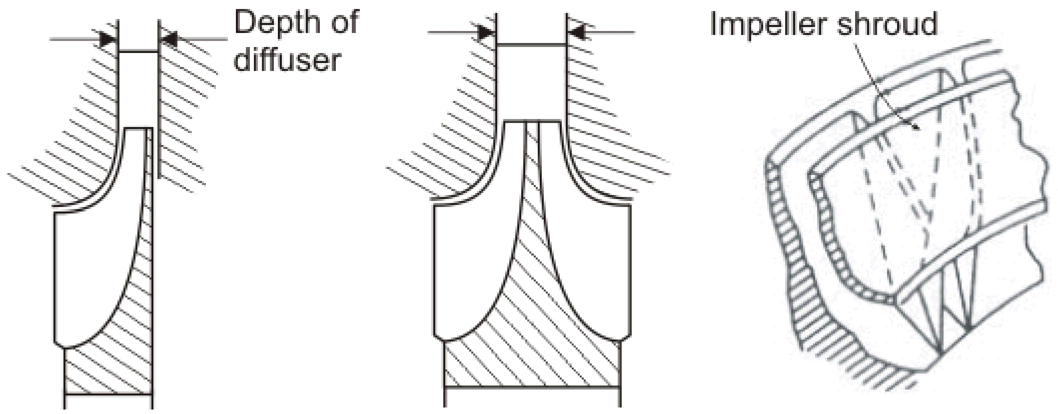


Figure 33.3 A centrifugal pump

The tips of the blades are sometimes covered by another flat disc to give shrouded blades (Figure 33.4c), otherwise the blade tips are left open and the casing of the pump itself forms the solid outer wall of the blade passages. The advantage of the shrouded blade is that flow is prevented from leaking across the blade tips from one passage to another.



(a) Single sided impeller **(b) Double sided impeller** **(c) Shrouded impeller**

Figure 33.4 Types of impellers in a centrifugal pump

As the impeller rotates, the fluid is drawn into the blade passage at the impeller eye, the centre of the impeller. The inlet pipe is axial and therefore fluid enters the impeller with very little whirl or tangential component of velocity and flows outwards in the direction of the blades. The fluid receives energy from the impeller while flowing through it and is discharged with increased pressure and velocity into the casing. To convert the kinetic energy or fluid at the impeller outlet gradually into pressure energy, diffuser blades mounted on a diffuser ring are used.

The stationary blade passages so formed have an increasing cross-sectional area which reduces the flow velocity and hence increases the static pressure of the fluid. Finally, the fluid moves from the diffuser blades into the volute casing which is a passage of gradually increasing cross-section and also serves to reduce the velocity of fluid and to convert some of the velocity head into static head. Sometimes pumps have only volute casing without any diffuser.

Figure 34.1 shows an impeller of a centrifugal pump with the velocity triangles drawn at inlet and outlet. The blades are curved between the inlet and outlet radius. A particle of fluid moves along the broken curve shown in Figure 34.1.

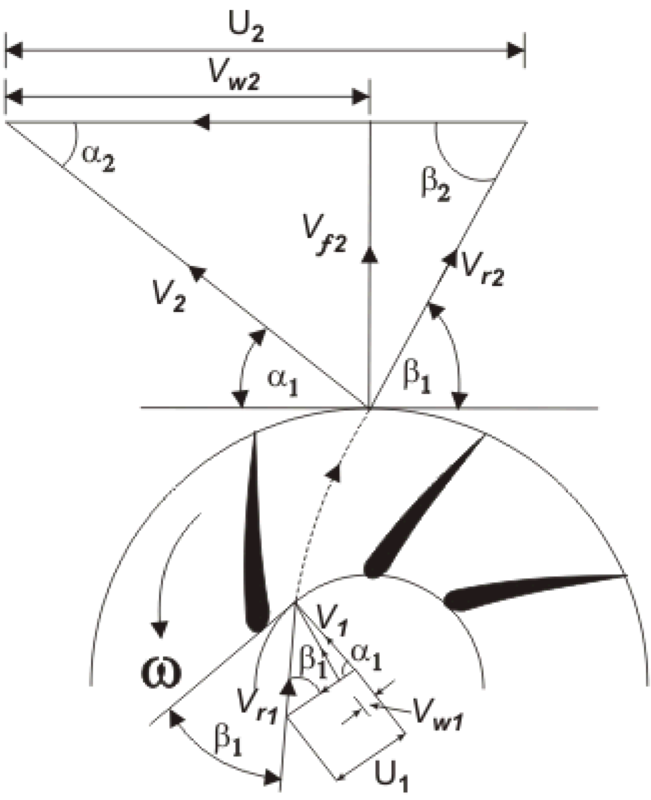


Figure 34.1 Velocity triangles for centrifugal pump Impeller

Let β_1 be the angle made by the blade at inlet, with the tangent to the inlet radius, while β_2 is the blade angle with the tangent at outlet. V_1 and V_2 are the absolute velocities of fluid at inlet an outlet respectively, while V_{r_1} and V_{r_2} are the relative velocities (with respect to blade velocity) at inlet and outlet respectively. Therefore,

$$\text{Work done on the fluid per unit weight} = (V_{w2}U_2 - V_{w1}U_1)/g \qquad (34.1)$$

A centrifugal pump rarely has any sort of guide vanes at inlet. The fluid therefore approaches the impeller without appreciable whirl and so the inlet angle of the blades is designed to produce a right-angled velocity triangle at inlet (as shown in Fig. 34.1). At conditions other than those for which the impeller was designed, the direction of relative velocity V_r does not coincide with that of a blade. Consequently, the fluid changes direction abruptly on entering the impeller. In addition, the eddies give rise to some back flow into the inlet pipe, thus causing fluid to have some whirl before entering the impeller. However, considering the operation under design conditions, the inlet whirl velocity V_{w1} and accordingly the inlet angular momentum of the fluid entering the impeller is set to zero. Therefore, Eq. (34.1) can be written as

$$\text{Work done on the fluid per unit weight} = V_{w2}U_2/g \qquad (34.2)$$

We see from this equation that the work done is independent of the inlet radius. The difference in total head across the pump known as manometrichead, is always less than the quantity $V_{w2}U_2/g$ because of the energy dissipated in eddies due to friction.

The ratio of manometric head *H* and the work head imparted by the rotor on the fluid $V_{w2}U_2/g$ (usually known as Euler head) is termed as manometricefficiency η_m. It represents the effectiveness of the pump in increasing the total energy of the fluid from the energy given to it by the impeller. Therefore, we can write

$$\eta_m = \frac{gH}{V_{w2}U_2} \qquad (34.3)$$

The overall efficiency η_0 of a pump is defined as

$$\eta_0 = \frac{\rho Q g H}{P} \qquad (34.4)$$

where, Q is the volume flow rate of the fluid through the pump, and P is the shaft power, i.e. the input power to the shaft. The energy required at the shaft exceeds $V_{w2} U_2 / g$ because of friction in the bearings and other mechanical parts. Thus a mechanical efficiency is defined as

$$\eta_{\mathrm{mech}} = \frac{\rho Q V_{w2} U_2}{P} \qquad (34.5)$$

so that

$$\eta_0 = \eta_m \times \eta_{\mathrm{mech}} \qquad (34.6)$$

Slip Factor

Under certain circumstances, the angle at which the fluid leaves the impeller may not be the same as the actual blade angle. This is due to a phenomenon known as fluid slip, which finally results in a reduction in V_{w2} the tangential component of fluid velocity at impeller outlet. One possible explanation for slip is given as follows.

In course of flow through the impeller passage, there occurs a difference in pressure and velocity between the leading and trailing faces of the impeller blades. On the leading face of a blade there is relatively a high pressure and low velocity, while on the trailing face, the pressure is lower and hence the velocity is higher. This results in a circulation around the blade and a non-uniform velocity distribution at any radius. The mean direction of flow at outlet, under this situation, changes from the blade angle at outlet β_2 to a different angle β_2' as shown in Figure 34.2 Therefore the tangential velocity component at outlet V_{w2} is reduced to V'_{w2} , as shown by the velocity triangles in Figure 34.2, and the difference ΔV_w is defined as the slip. The slip factor σ_s is defined as

$$\sigma_s = V'_{w2} / V_{w2}$$

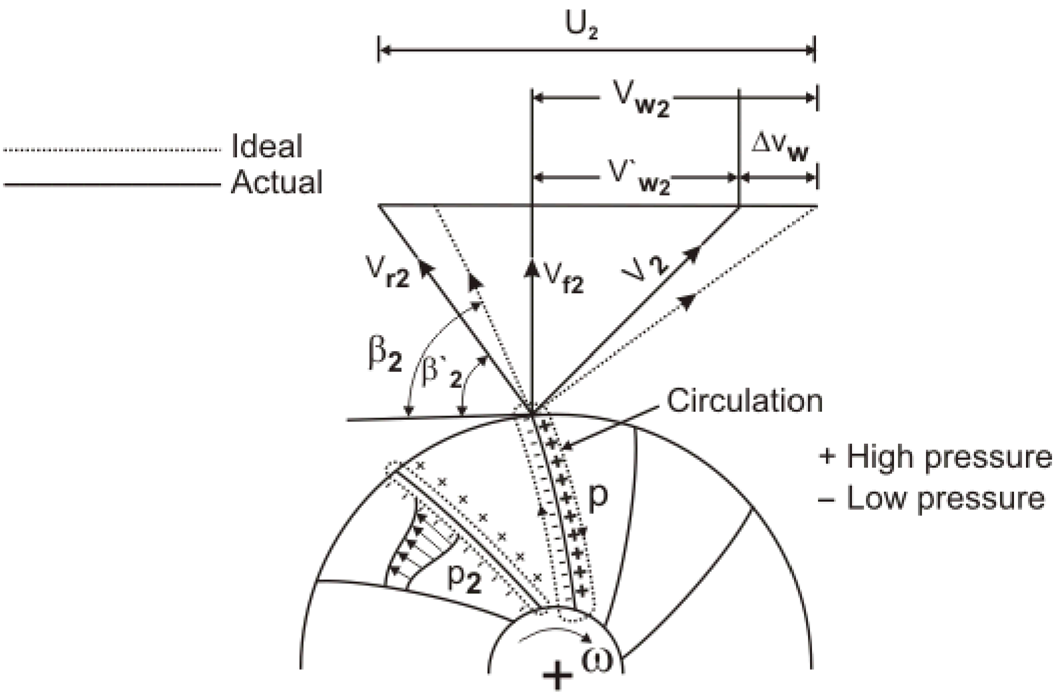


Figure 34.2 Slip and velocity in the impeller blade passage of a centrifugal

pump

With the application of slip factor σ_s, the work head imparted to the fluid (Euler head) becomes $\sigma_s V_{w2} U_2 / g$. The typical values of slip factor lie in the region of 0.9.

Losses in a Centrifugal Pump

• Mechanical friction power loss due to friction between the fixed and rotating parts in the bearing and stuffing boxes.

• Disc friction power loss due to friction between the rotating faces of the impeller (or disc) and the liquid.

• Leakage and recirculation power loss. This is due to loss of liquid from the pump and recirculation of the liquid in the impeller. The pressure difference between impeller tip and eye can cause a recirculation of a small volume of liquid, thus reducing the flow rate at outlet of the impeller as shown in Fig. (34.3).

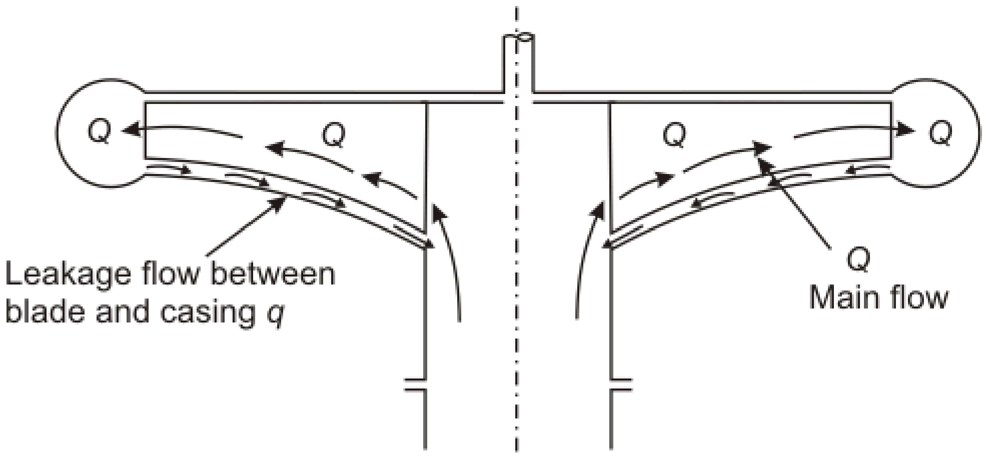


Figure 34.3 Leakage and recirculation in a centrifugal pump

Characteristics of a Centrifugal Pump

With the assumption of no whirl component of velocity at entry to the impeller of a pump, the work done on the fluid per unit weight by the impeller is given by Equation(34.2). Considering the fluid to be frictionless, the head developed by the pump will be the same san can be considered as the theoretical head developed. Therefore we can write for theoretical head developed H_{theo} as

$$H_{\text{theo}} = \frac{V_{w2}\ U_2}{g} \tag{35.1}$$

From the outlet velocity triangle figure(34.1)

$$V_{w2} = U_2 - V_{f2}\ \cot\beta_2 = U_2 - (Q/A)\cot\beta_2 \tag{35.2}$$

where Q is rate of flow at impeller outlet and A is the flow area at the periphery of the impeller. The blade speed at outlet U_2 can be expressed in terms of rotational speed of the impeller N as

$$U_2 = \pi DN$$

Using this relation and the relation given by Eq. (35.2), the expression of theoretical head developed can be written from Eq. (35.1) as

$$H_{\text{theo}} = \pi^2 D^2 N^2 - \left[\frac{\pi DN}{A}\cot\beta_2\right]Q$$

$$= K_1 - K_2 Q \tag{35.3}$$

where, $K_1 = \pi^2 D^2 N^2$ and $K_2 = (\pi DN/A)Cot\beta_2$

For a given impeller running at a constant rotational speed. K_1 and K_2 are constants, and therefore head and discharge bears a linear relationship as shown by Eq. (35.3). This linear variation of H_{theo} with Q is plotted as curve I in Fig. 35.1.

If slip is taken into account, the theoretical head will be reduced to $\sigma_s V_{w2} U_2 / g$. Moreover the slip will increase with the increase in flow rate Q. The effect of slip in head-discharge relationship is shown by the curve II in Fig. 35.1. The loss due to slip can occur in both a real and an ideal fluid, but in a real fluid the shock losses at entry to the blades, and the friction losses in the flow passages have to be considered. At the design point the shock losses are zero since the fluid moves tangentially onto the blade, but on either side of the design point the head loss due to shock increases according to the relation

$$h_{\text{shock}} = K_3 (Q_f - Q)^2 \qquad (35.4)$$

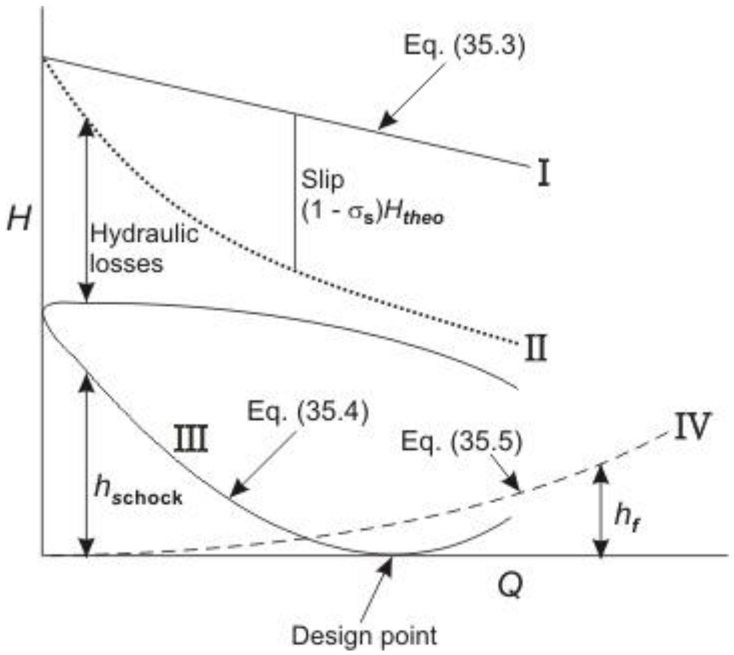


Figure 35.1 Head-discharge characteristics of a centrifugal pump

where Q_f is the off design flow rate and K_3 is a constant. The losses due to friction can usually be expressed as

$$h_f = K_4 Q^2 \qquad (35.5)$$

where, K_4 is a constant.

Equation (35.5) and (35.4) are also shown in Fig. 35.1 (curves III and IV) as the characteristics of losses in a centrifugal pump. By subtracting the sum of the losses from the head in consideration of the slip, at any flow rate (by subtracting the sum of ordinates of the curves III and IV from the ordinate of the curve II at all values of the abscissa), we get the curve V which represents the relationship of the actual head with the flow rate, and is known as head-discharge characteristic curve of the pump.

Effect of blade outlet angle

The head-discharge characteristic of a centrifugal pump depends (among other things) on the outlet angle of the impeller blades which in turn depends on blade settings. Three types of blade settings are possible (i) the forward facing for which the blade curvature is in the direction of rotation and, therefore, $\beta_2 > 90°$ (Fig. 35.2a), (ii) radial, when $\beta_2 = 90°$ (Fig. 35.2b), and (iii) backward facing for which the blade curvature is in a direction opposite to that of the impeller rotation and therefore, $\beta_2 < 90°$ (Fig. 35.2c). The outlet velocity triangles for all the cases are also shown in Figs. 35.2a, 35.2b, 35.2c. From the geometry of any triangle, the relationship between V_w, U_2 and β_2 can be written as.

$$V_{w2} = U_2 - V_{f2} \cot \beta_2$$

which was expressed earlier by Eq. (35.2).

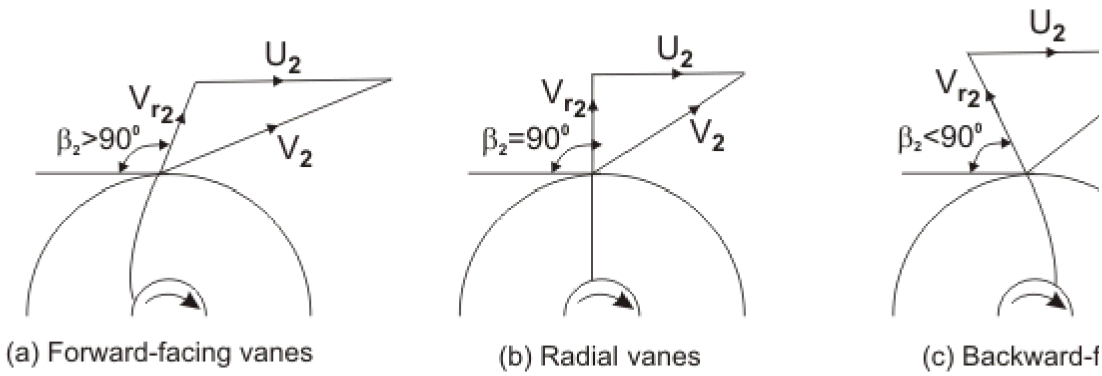


Figure 35.2 Outlet velocity triangles for different blade settings in a centrifugal pump

In case of forward facing blade, $\beta_2 > 90°$ and hence cot β_2 is negative and therefore V_{w2} is more than U_2. In case of radial blade, $\beta_2 = 90°$ and $V_{w2} = U_2$. In case of backward facing blade, $\beta_2 < 90°$ and $V_{w2} < U_2$. Therefore the sign of K_2, the constant in the theoretical head-discharge relationship given by the Eq. (35.3), depends accordingly on the type of blade setting as follows:

For forward curved blades $K_2 < 0$

For radial blades $K_2 = 0$

For backward curved blades $K_2 > 0$

With the incorporation of above conditions, the relationship of head and discharge for three cases are shown in Figure 35.3. These curves ultimately revert to their more recognized shapes as the actual head-discharge characteristics respectively after consideration of all the losses as explained earlier Figure 35.4.

For both radial and forward facing blades, the power is rising monotonically as the flow rate is increased. In the case of backward facing blades, the maximum efficiency occurs in the region of maximum power. If, for some reasons, Q increases beyond Q_D there occurs a decrease in power. Therefore the motor used to drive the pump at part load, but rated at the design point, may be safely used at the maximum power. This is known as self-limiting characteristic. In case of radial and forward-facing blades, if the pump motor is rated for maximum power, then it will be under utilized most of the time, resulting in an increased cost for the extra rating. Whereas, if a smaller motor is employed, rated at the design point, then if Q increases above Q_D the motor will be overloaded and may fail. It, therefore, becomes more difficult to decide on a choice of motor in these later cases (radial and forward-facing blades).

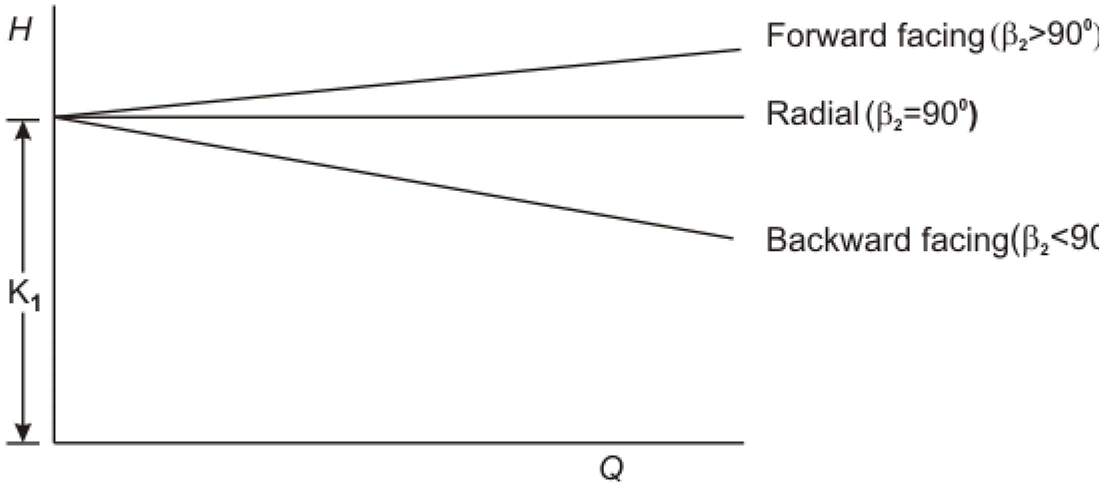


Figure 35.3 Theoretical head-discharge characteristic curves of a centrifugal pump for different blade settings

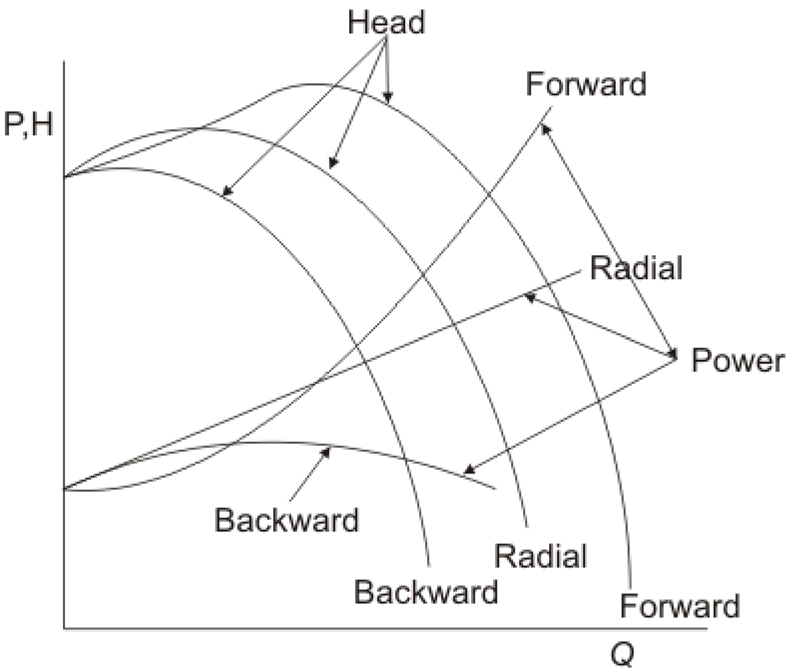


Figure 35.4 Actual head-discharge and power-discharge characteristic curves of a centrifugal pump

Flow through Volute Chambers

Apart from frictional effects, no torque is applied to a fluid particle once it has left the impeller. The angular momentum of fluid is therefore constant if friction is neglected. Thus the fluid particles follow the path of a free vortex. In an ideal case, the radial velocity at the impeller outlet remains constant round the circumference. The combination of uniform radial velocity with the free vortex ($V_w \cdot r$ =constant) gives a pattern of spiral streamlines which should be matched by the shape of the volute. This is the most important feature of the design of a pump. At maximum efficiency, about 10 percent of the head generated by the impeller is usually lost in the volute.

Vanned Diffuser

A vanned diffuser, as shown in Fig. 36.1, converts the outlet kinetic energy from impeller to pressure energy of the fluid in a shorter length and with a higher efficiency. This is very advantageous where the size of the pump is important. A ring of diffuser vanes surrounds the impeller at the outlet. The fluid leaving the impeller first flows through a vaneless space before entering the diffuser vanes. The divergence angle of the diffuser passage is of the order of 8-10 ° which ensures no boundary layer separation. The optimum number of vanes are fixed by a compromise between the diffusion and the frictional loss. The greater the number of vanes, the better is the diffusion (rise in static pressure by the reduction in flow velocity) but greater is the frictional loss. The number of diffuser vanes should have no common factor with the number of impeller vanes to prevent resonant vibration.

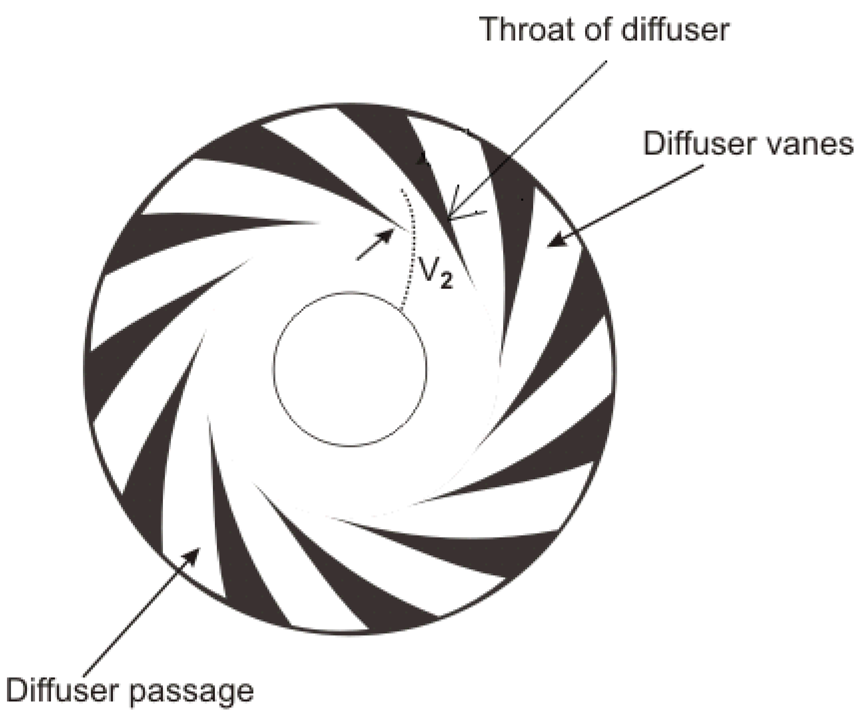


Figure 36.1 A vanned diffuser of a centrifugal pump

Cavitation in centrifugal pumps

Cavitation is likely to occur at the inlet to the pump, since the pressure there is the minimum and is lower than the atmospheric pressure by an amount that equals the vertical height above which the pump is situated from the supply reservoir (known as sump) plus the velocity head and frictional losses in the suction pipe. Applying the Bernoulli's equation between the surface of the liquid in the sump and the entry to the impeller, we have

$$\frac{p_i}{\rho g} + \frac{V_i^2}{2g} + z = \frac{p_A}{\rho g} - h_f \qquad (36.1)$$

where, p_i is the pressure at the impeller inlet and p_A is the pressure at the liquid surface in the sump which is usually the atmospheric pressure, *Z1* is the vertical height of the impeller inlet from the liquid surface in the sump, h_f is the loss of head in the suction pipe. Strainers and non-return valves are commonly fitted to intake pipes. The term h_f must therefore include the losses occurring past these devices, in addition to losses caused by pipe friction and by bends in the pipe.

In the similar way as described in case of a reaction turbine, the net positive suction head 'NPSH' in case of a pump is defined as the available suction head (inclusive of both static and dynamic heads) at pump inlet above the head corresponding to vapor pressure.

Therefore,

$$\text{NPSH} = \frac{p_i}{\rho g} + \frac{V_i^2}{2g} - \frac{p_v}{\rho g} \qquad (36.2)$$

Again, with help of Eq. (36.1), we can write

$$\text{NPSH} = \frac{p_A}{\rho g} - \frac{p_v}{\rho g} - z - h_f$$

The Thomas cavitation parameter s and critical cavitation parameter σ_c are defined accordingly (as done in case of reaction turbine) as

$$\sigma = \frac{\text{NPSH}}{\text{H}} = \frac{(p_A / \rho g) - (p_v / \rho g) - z - h_f}{H} \qquad (36.3)$$

and $$\sigma_c = \frac{(p_A / \rho g) - (p_i / \rho g) - z - h_f}{H} \qquad (36.4)$$

We can say that for cavitation not to occur,

$$\sigma > \sigma_c \, (\text{i.e. } p_i > p_v)$$

In order that s should be as large as possible, *z* must be as small as possible. In some installations, it may even be necessary to set the pump below the liquid level at the sump (i.e. with a negative vale of *z*) to avoid cavitation.

Variation of Pump Diameter

A variation in pump diameter may also be examined through the similarly laws. For a constant speed,

$$Q_1 / Q_1^3 = Q_2 / D_2^3$$

and $$H_1 / D_1^2 = H_2 / D_2^2$$

or, $$H \propto Q^{2/3} \tag{38.1}$$

Pumps in Series and Parallel

When the head or flow rate of a single pump is not sufficient for a application, pumps are combined in series or in parallel to meet the desired requirements. Pumps are combined in series to obtain an increase in head or in parallel for an increase in flow rate. The combined pumps need not be of the same design. Figures 38.1 and 38.2 show the combined *H-Q* characteristic for the cases of identical pumps connected in series and parallel respectively. It is found that the operating point changes in both cases. Fig. 38.3 shows the combined characteristic of two different pumps connected in series and parallel.

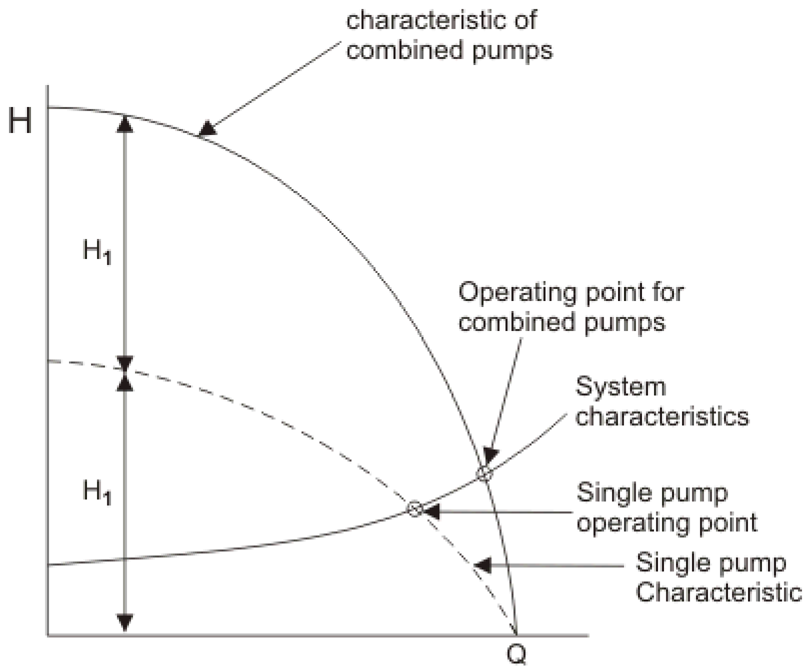


Figure 38.1 Two similar pumps connected in series

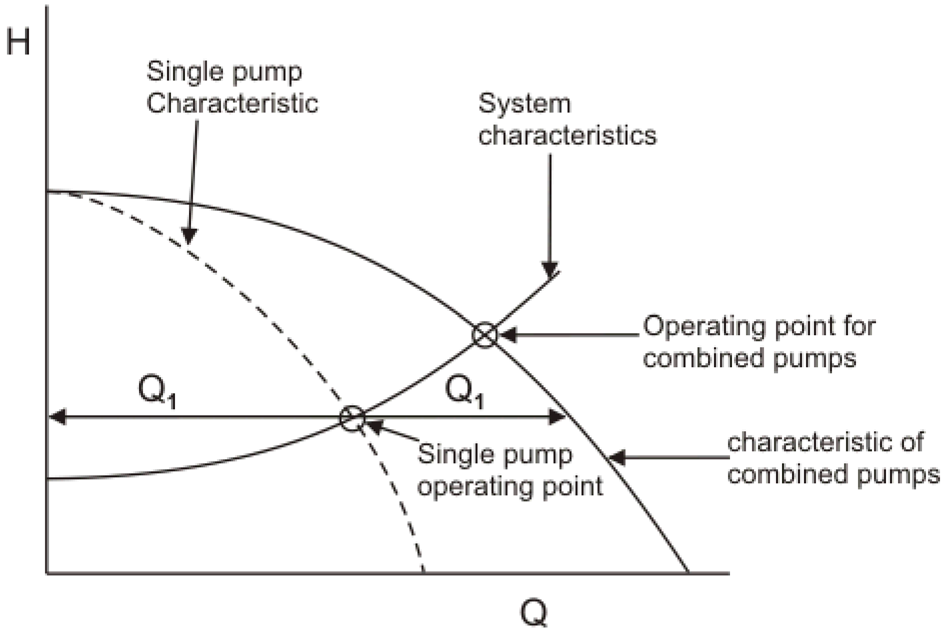


Figure 38.2 Two similar pumps connected in parallel

Specific Speed of Centrifugal Pumps

The concept of specific speed for a pump is same as that for a turbine. However, the quantities of interest are *N*, *H* and *Q* rather than *N*, *H* and *P* like in case of a turbine.

For pump

$$N_{s_p} = NQ^{1/2} / H^{3/4} \qquad (38.2)$$

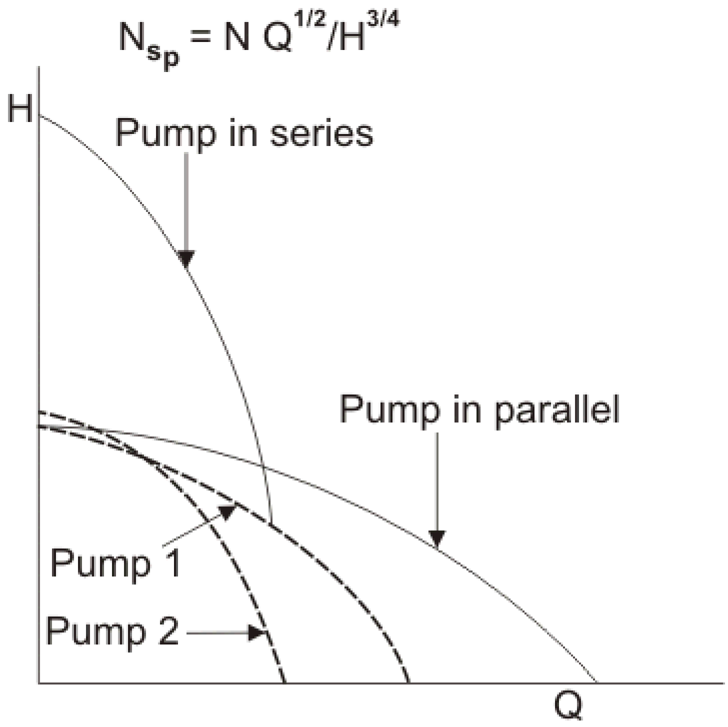


Figure 38.3 Two different pumps connected in series and parallel

The effect of the shape of rotor on specific speed is also similar to that for turbines. That is, radial flow (centrifugal) impellers have the lower values of N_{s_p} compared to those of axial-flow designs. The impeller, however, is not the entire pump and, in particular, the shape of volute may appreciably affect the specific speed. Nevertheless, in general, centrifugal pumps are best suited for providing high heads at moderate rates of flow as compared to axial flow pumps which are suitable for large rates of flow at low heads. Similar to turbines, the higher is the specific speed, the more compact is the machine for given requirements. For multistage pumps, the specific speed refers to a single stage.

Problems

1) The impeller of a centrifugal pump is 0.5m in diameter and rotates at 1200 rpm. Blades are curved back to an angle of 30 ° to the tangent at outlet tip. If the measured velocity of flow at outlet is 5 m/s, find the work input per kg of water per second. Find the theoretical maximum lift to which the water can be raised if the pump is provided with whirlpool chamber which reduces the velocity of water by 50%.

(Ans. 72.78m, 65.87m)

2) The impeller of a centrifugal pump is 0.3m in diameter and runs at 1450rpm. The pressure gauges on suction and delivery sides show the difference of 25m. The blades are curved back to an angle of 30 ° . The velocity of flow through impeller, being constant, equals to 2.5m/s, find the manometric efficiency of the pump. If the frictional losses in impeller amounts to 2m, find the fraction of total energy which is converted into pressure energy by impeller. Also find the pressure rise in pump casing.

(Ans. 58.35%, 54.1%, 1.83m of water)

3) A centrifugal pump is required to work against a head of 20m while rotating at the speed of 700 rpm. If the blades are curved back to an angle of 30 ° to tangent at outlet tip and velocity of flow through impeller is 2 /s, calculate the impeller diameter when (a) all the kinetic energy at impeller outlet is wasted and (b) when 50% of this energy is converted into pressure energy in pump casing.

(Ans. 0.55m, 0.48m)

4) During a laboratory test on a pump, appreciable cavitation began when the pressure plus the velocity head at inlet was reduced to 3.26m while the change in total head across the pump was 36.5m and the discharge was 48 litres/s. Barometric pressure was 750 mm of Hg and the vapour pressure of water 1.8kPa. What is the value of σ_c ? If the pump is to give the same total head and discharge in location where the normal atmospheric pressure is 622mm of Hg and the vapour pressure of

water is 830 Pa, by how much must the height of the pump above the supply level be reduced?

(Ans. 0.084, 1.65m)

Model Solution

Problem 1

1) The peripheral speed at impeller outlet

$$U_2 = \frac{\pi \times 0.5 \times 1200}{60} = 31.4\,\text{m/s}$$

$V_{f2} = 5\,\text{m/s}$ (given)

Work input per unit weight of

Water = $\dfrac{V_{w2}\,U_2}{g} = \dfrac{(31.4 - 5\cot 30^\circ) \times 31.4}{9.81}$

=72.78m

Under ideal condition (without loss), the total head developed by the pump = 72.78 m

Absolute velocity of water at the outlet

$$V_2 = \sqrt{(31.4 - 5\cot 30^\circ)^2 + 5^2}$$

=23.28 m/s

At the whirlpool chamber,

The velocity of water at delivery = 0.5 ´ 23.28m/s

Therefore the pressure head at impeller outlet

$$=72.78 - \frac{(0.5\times 23.28)^2}{2\times 9.81}$$

= 65.87m

Hence, we theoretical maximum lift = 65.87m

RECIPROCATING PUMPS

Reciprocating pump is a device which converts the mechanical energy into hydraulic energy by sucking the liquid into a cylinder. In this pump, a piston is reciprocating, which uses thrust on the liquid and increases its hydraulic energy.

Reciprocating pump is also known a called a positive displacement pump. Because it discharges a definite quantity of liquid. It is often used where a small quantity of liquid is to handled and where delivery pressure is quite large

Parts of Reciprocating Pump

The following are the main parts of the reciprocating pump.

1. Cylinder
2. Suction Pipe
3. Delivery Pipe
4. Suction valve
5. Delivery valve
6. Air vessels

Cylinder

A cylinder, in which piston is moving to and fro. The moment of the piston obtains by a connecting rod, which connects the piston and crank.

Suction Pipe

In which the source of water connects the cylinder together. The suction pipe allows the water to flow in the cylinder.

Delivery Pipe

After the process, the source of water leaves the cylinder and discharge through the delivery pipe.

Suction Valve

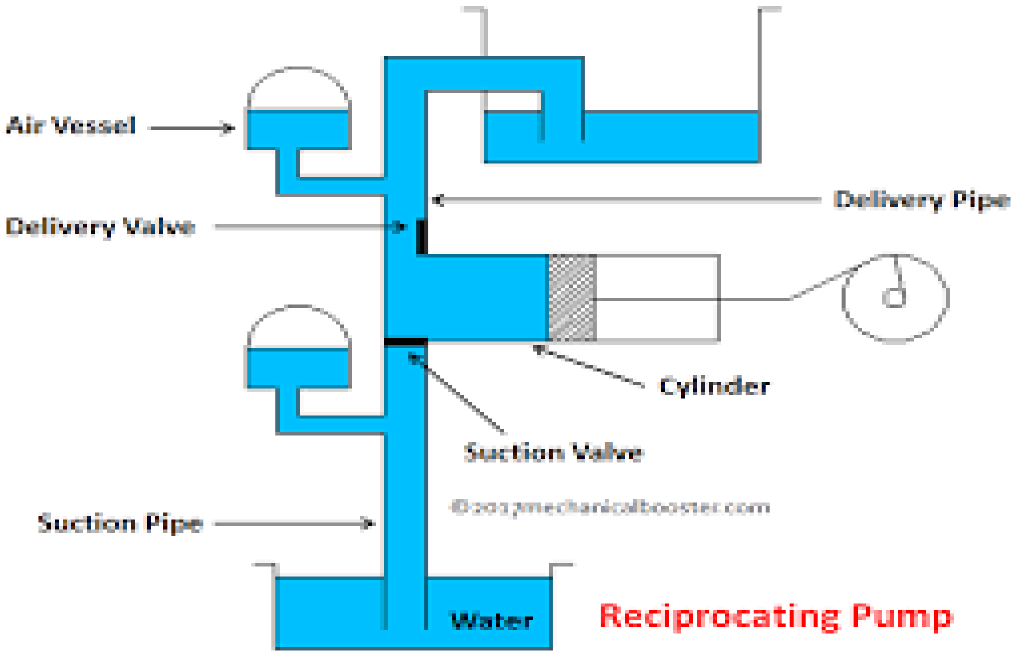
In this valve, the flow of water enters from the suction pipe into the cylinder.

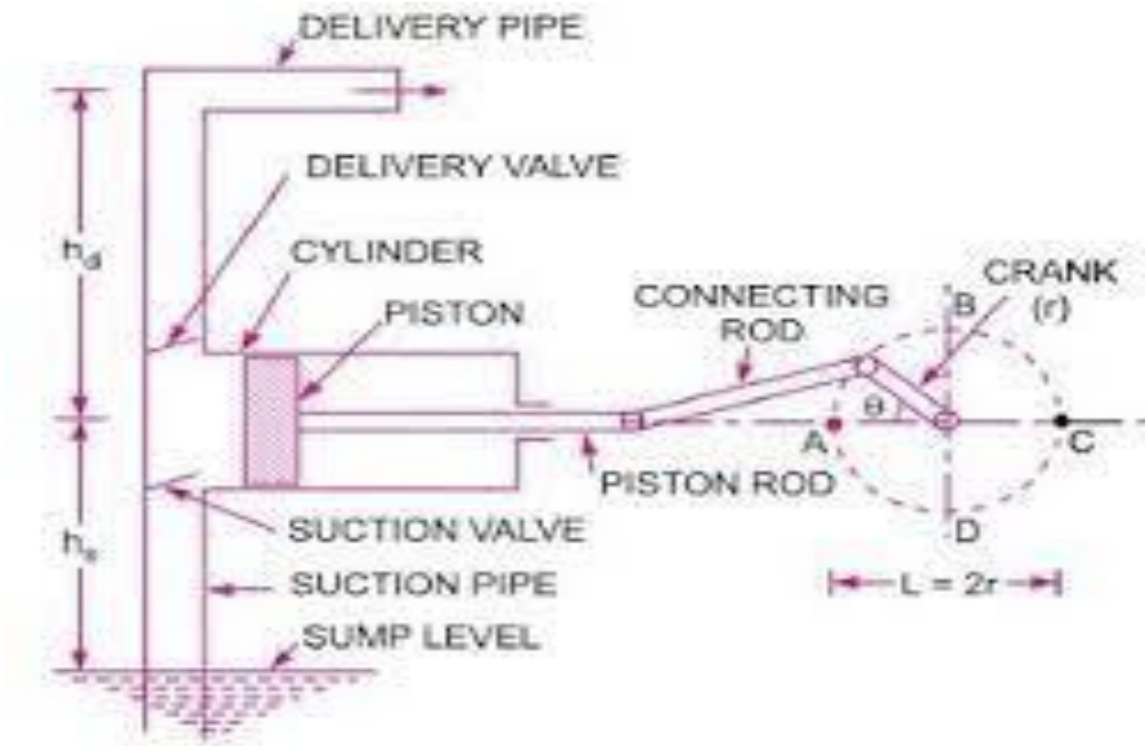
Delivery Valve

In this valve, the flow of water discharge from the cylinder into the delivery pipe

Air Vessels

It is a closed chamber made up cast iron. Having to two ends one ends is open at its base through which the water flows into the vessel cylinder. The air vessels fitted to the suction pipe and delivery pipe of this pump to get a uniform discharge.





Functions of Air Vessels

- The air vessels use to get the continuous flow of water at a uniform rate.
- To reduce the amount of work in overcoming the frictional resistance in the suction pipe and delivery pipe.
- To run the pump at high speed with separation.

Types of Reciprocating Pump

The following are the types of reciprocating pump according to the source of work and mechanism.

1. Simple hand-operated reciprocating pump
2. Power-operated deep well reciprocating pump
3. Single-acting reciprocating pump
4. Double-acting reciprocating pump

5. Triple-acting reciprocating pump
6. Pump with air vessels
7. Pump without air vessels

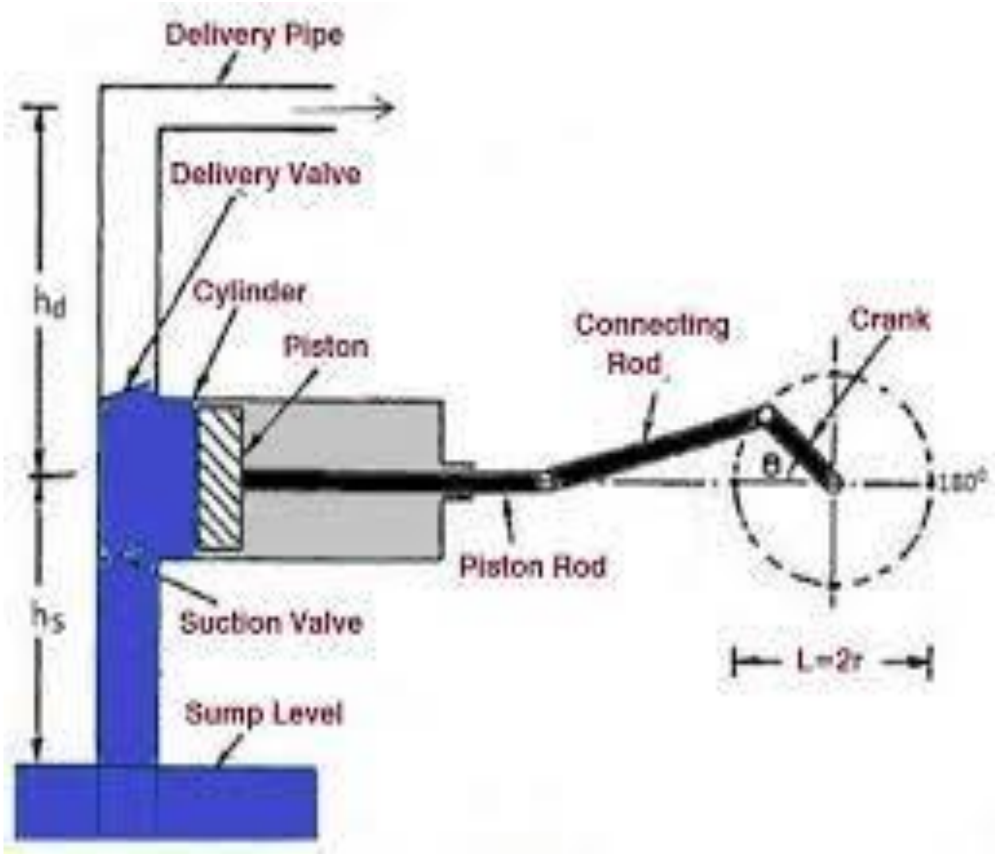
Working Principle of Reciprocating Pump

1. Single-acting reciprocating pump and
2. A double-acting reciprocating pump.

1. Single Acting Reciprocating Pump

In this pump, A cylinder, in which a piston moves forward and backward. The piston is reciprocating by means of the connecting rod. The connecting rod connects the piston and the rotating crank. The crank is rotating by means of an electric motor.

The suction and delivery pipes with suction and delivery valve are arranged to the cylinder.



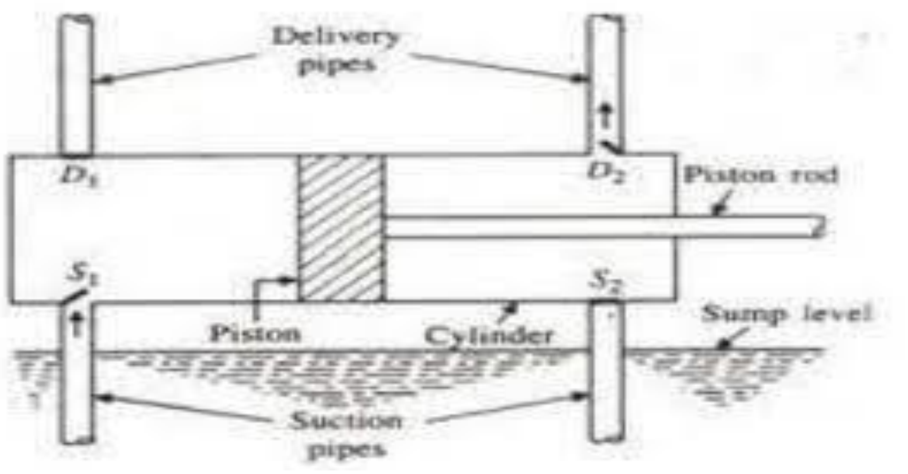
- **The suction valve** allows the water to the cylinder and
- **The delivery valve** leaves the water from the cylinder.

As the crank rotates, during the **first stroke of the piston** (called suction stroke), the water enters into the cylinder. In a suction stroke, the crank is rotating from A to C (from 0° to 180°) the piston is moving towards the right side of the cylinder. Due to this, the vacuum creates in the cylinder. This vacuum causes the suction valve to open and the water enters the cylinder.

In the next stroke called delivery stroke, the water leaves the cylinder. In the delivery stroke, the crank is rotating from C to A (from 180° to 360°) the piston is moving to the left side of the cylinder. Due to this, the pressure of the liquid increases inside the cylinder. This pressure causes the suction valve to close and delivery valve to open. Then the water is forced into the delivery pipe and raised to a required height.

2. Double Acting Reciprocating Pump

In this, the water is acting on both sides of the piston as shown in the figure.



Thus two suction pipes and two delivery pipes are required for a double-acting pump. When there is a suction stroke on one side of the piston, at the same time there is a delivery stroke on the other side of the piston.

Hence for one complete revolution of the crank, there is two delivery stroke and the water is delivered to the pipes by the pump during these two delivery strokes.

Application of Reciprocating Pumps

Following are the applications of reciprocating pumps:

1. The reciprocating pump is used in oil drilling operations.
2. It is useful in pneumatic pressure systems.
3. Mostly used in light oil pumping.
4. It is used for feeding small boilers condensate return.

QUESTIONS:

1.What is the difference between centrifugal pump and reciprocating pump?

2.Why slip occurs in reciprocating pump?

3.What is the Reciprocating pump start up procedure?

4.What is priming why it is not done on reciprocating pumps?

5.How can the slip of a reciprocating pump be negative?

6.What leads to the failure of reciprocating pumps?

7.Doescavitation occurs in reciprocating pump?

8.What is reciprocating pump and its principle?

9.Is NPSH required for reciprocating pump?

10.What is the physical difference between reciprocating compressor and reciprocating pump?

11.Why a reciprocating pump produce high head but low discharge?

12.Why a centrifugal pump is more efficient than reciprocating pump?

13.Why is a reciprocating pump not coupled directly to a motor?

14.What is the main reason of cavitations in reciprocating pump?

15.Is compressor pump a type of reciprocating pump?

16.How are reciprocating pumps classified?

17.Why does not centrifugal pump give a constant flow but reciprocating pump does?

18.What is the working of a reciprocating pump?

are the two diff