Program : **B.Tech**

Subject Name: **Turbo Machinery**

Subject Code: **ME-603**

Semester: **6th**

Unit 3 - HYDRAULIC TURBINE

IMPULSE TURBINE

Figure 3.1 Typical PELTON WHEEL with 21 Buckets

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 as shown in Figure 3.1. 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 (Figure 3.2). 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.

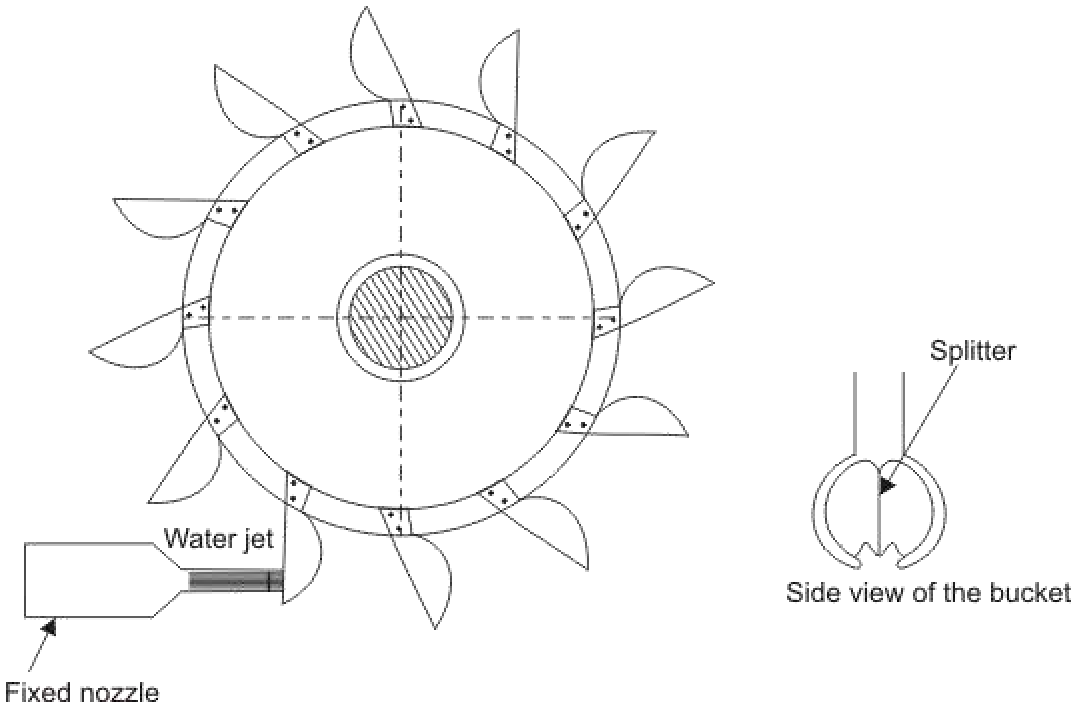
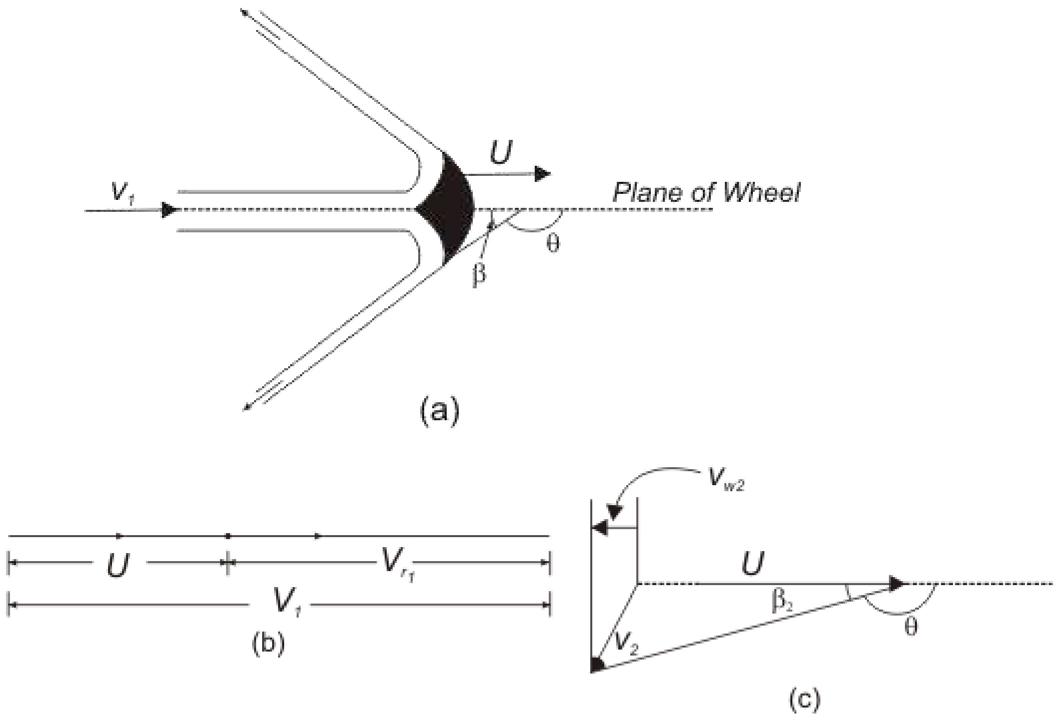


Figure 3.2 A Pelton wheel

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 165 degree. In practice, however, the deflection is limited to about 15 degree so that the water leaving a bucket may not hit the back of the following bucket. Therefore, the camber angle of the buckets is made as small as possible.

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: Figure 26.3a 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 V1 with which it strikes the bucket is given by



(a)Flow along the bucket of a pelton wheel

Figure 3.3 (b) Inlet velocity triangle

(c)Outlet velocity triangle

where, k 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 V1 is tangential, i.e. V1 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

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

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 3.3b)

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

and the tangential component of outlet velocity (Figure 26.3c)

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

$$= [V_{r_1} + V_{r_2} \cos \beta_2] U \qquad (3.1)$$

The relative velocity Vr1 becomes slightly less than V2 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 is usually expressed as Vr2/Vr1 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)

$$E/m = V_{r_1} [1 + K \cos \beta_2] U \qquad (3.2)$$

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

$$= \rho Q [1 + K \cos \beta_2](V_1 - U) U \qquad (3.3)$$

The power input to the wheel is found from the kinetic energy of the jet arriving at the wheel. Therefore the wheel efficiency of a Pelton turbine can be written as

$$= 2 [1 + K \cos \beta_2] \left[1 - \frac{U}{V_1}\right] \frac{U}{V_1} \qquad (3.4)$$

It is found that the efficiency depends on U and V1. For a given design of the bucket, i.e. for constant values of K, the efficiency becomes a function of U/V1 only.

For η to be maximum,

or, $$U / V_1 = \frac{1}{2} \qquad (3.5)$$

Therefore, the maximum wheel efficiency can be written after substituting the relation given by eqn.(3.5) in eqn.(3.4) as

$$\eta_{w\max} = 2(1-K\cos\beta_2)/2 \quad (3.6)$$

The condition given by Eq. (3.5) 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 η0 will be less than 1 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/V1 is slightly less than the theoretical value of 0.5. The value usually obtained in practice is about 0.46. The Figure 3.4 shows the variation of wheel efficiency with blade to jet speed ratio for assumed values at k=1 and 0.8. 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.

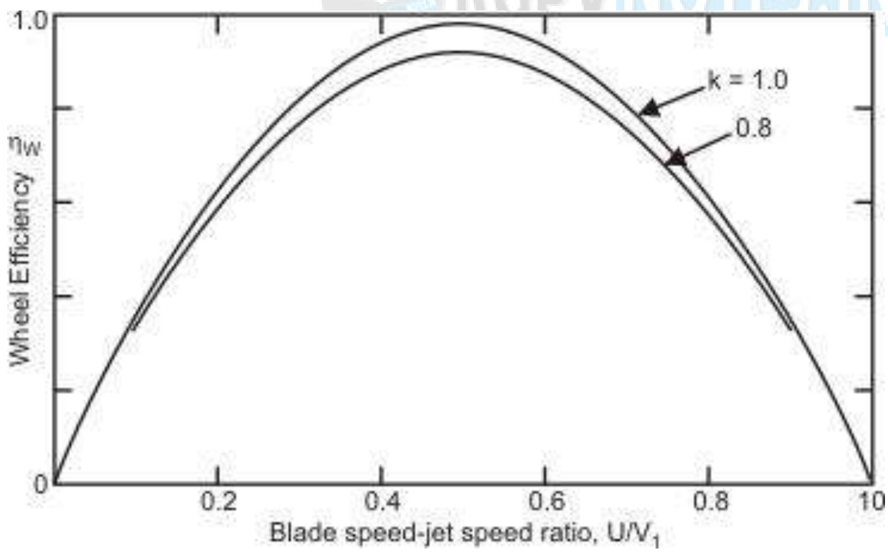


Figure 3.4 Theoretical variation of wheel efficiency for a Pelton turbine with blade speed to j speed ratio for different values of k

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.

$$P = \rho Q g H \eta_h = \frac{\pi \rho d^2 V_1^3 \eta_h}{4 \times 2 C_v^2} \qquad (3.7)$$

The 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. 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.

$$\text{n(number of buckets)} = 15 + \frac{53}{N_{sT}} \qquad (3.8)$$

Governing of Pelton Turbine: 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 V1 has to be kept at its optimum value of about 0.46. Hence, when U is fixed. U/V1 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 same. This is usually achieved by a spear valve in the nozzle (Figure 3.5a). Movement of the spear and the axis of the nozzle change 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 3.5b) 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.

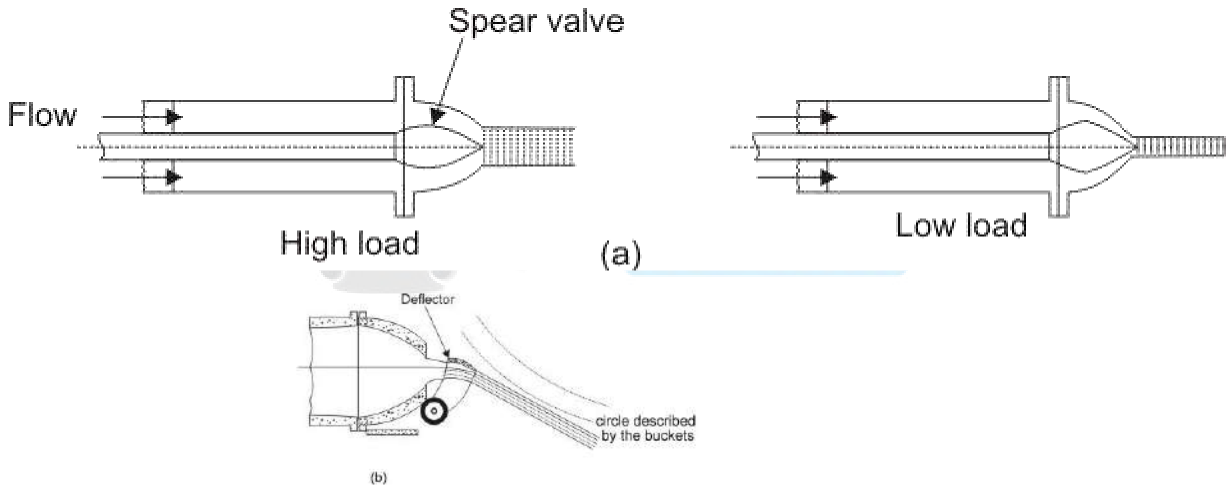


Figure 3.5 (a) Spear valve to alter jet area in a Pelton wheel
(b) Jet deflected from bucket

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 is usually limited to 4 or 6 per wheel. The increases in jet diameter in turn increase 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. 3.6

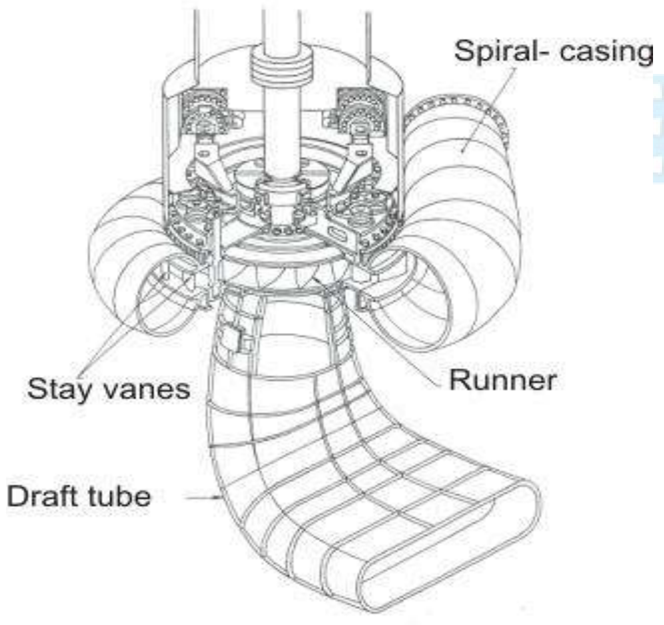


Figure 3.6 A Francis turbine

A Francis turbine comprises mainly the four components:

(i) spiral casing,

(ii) guide on stay vanes,

(iii) runner blades,

(iv) draft-tube as shown in Figure 3.6

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.

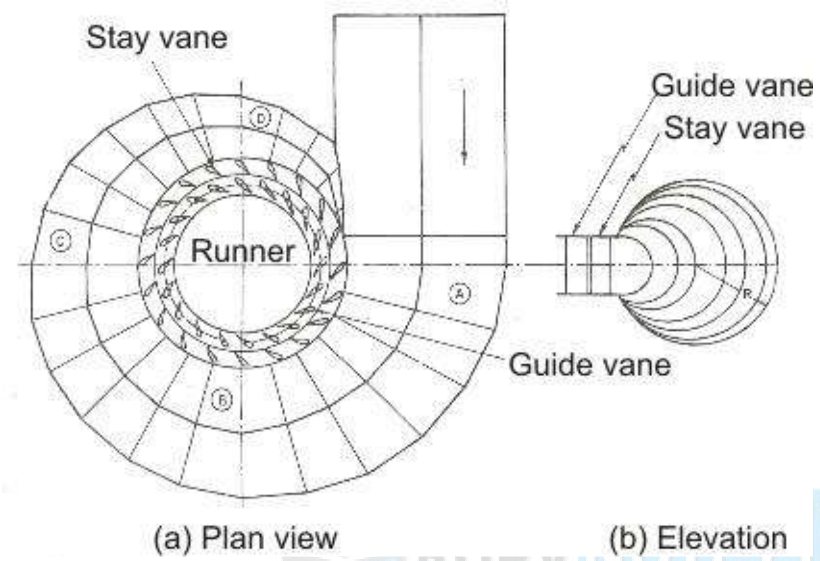


Figure 3.7 Spiral Casing

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 guides 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 3.8 shows a flow diagram from the reservoir via a reaction turbine to the tail race.

The total head H 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

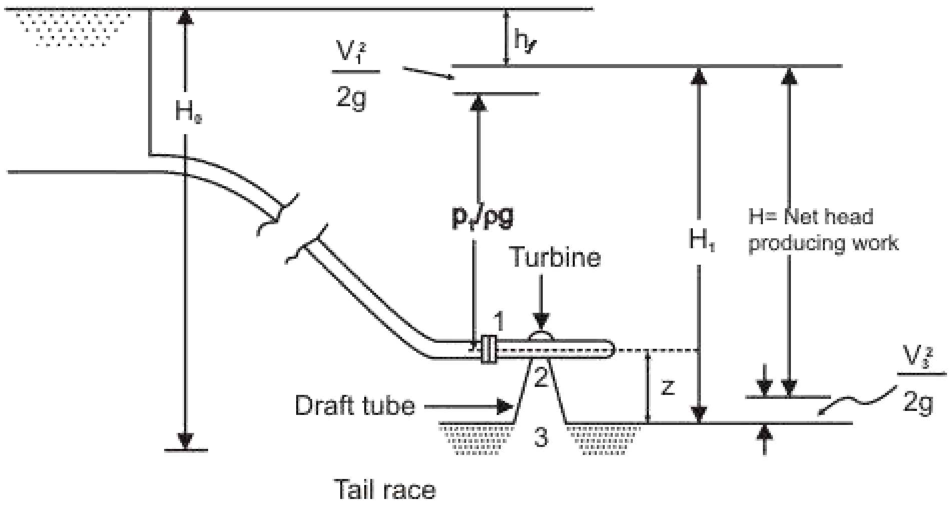


Figure 3.8 Head across a reaction turbine

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

$$= \frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z - \frac{V_3^2}{2g} = H_1 - \frac{V_3^2}{2g} \qquad (3.9)$$

$$= (H_0 - h_f) - \frac{V_3^2}{2g} \qquad (3.10)$$

The pressures are defined in terms of their values above the atmospheric pressure. Section 2 and 3 in Figure 3.8 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 H0 or H. Applying the Bernoulli's equation between 2 and 3 in consideration of flow, without losses, through the draft tube, we can write.

$$\frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z = 0 + \frac{V_3^2}{2g} + 0 \qquad (3.11)$$

$$\frac{P_2}{\rho g} = -\left[z + \frac{V_2^2 - V_3^2}{2g} \right] \qquad (3.12)$$

Since , both the terms in the bracket are positive and hence P is always negative, which implies that the static pressure at the outlet of the runner is always below the atmospheric pressure. Equation (3.12) also shows that the value of the suction pressure at runner outlet depends on z, the height of the runner above the tail race. The value of this minimum pressure P2 should never fall below the vapor 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 clearer, 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. This is made to keep the kinetic energy at outlet a minimum.

Figure 3.9 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. equal to that at the inlet to the draft tube.

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

$$E/m = e = V_{w_1} U_1 \quad (3.13)$$

Where, *e* is the energy transfer to the rotor per unit mass of the fluid. From the inlet velocity triangle shown in Fig.3.9

$$V_{w_1} = V_{f_1} \cot \alpha_1 \quad (3.14)$$

and $$U_1 = V_{f_1} (\cot \alpha_1 + \cot \beta_1) \quad (3.14\text{ b})$$

After simplification

$$e = V_{f1}^2 \cot \alpha_1 \ (\cot \alpha_1 + \cot \beta_1) \quad (3.15)$$

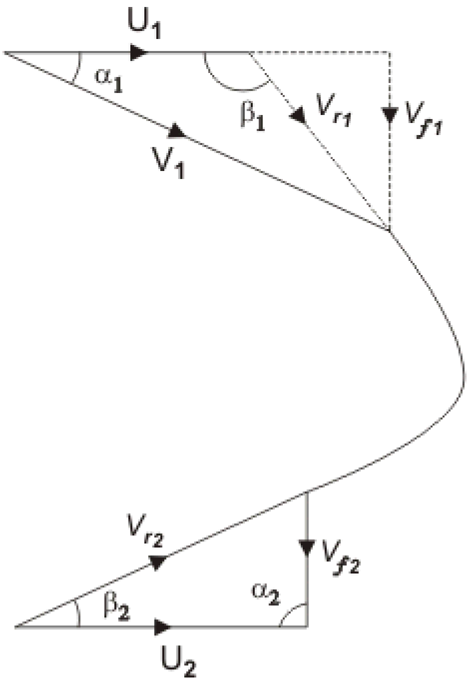


Figure 3.9 Velocity triangle for a Francis runner

The loss of kinetic energy per unit mass becomes equal to e. Therefore neglecting friction, the blade efficiency becomes

$$\eta_b = \frac{e}{e + (V_{f2}^2 / 2)}$$

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

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.

$$R = \frac{e - \frac{1}{2}\left(V_1^2 - V_{f2}^2\right)}{e} = 1 - \frac{\frac{1}{2} V_{f1}^2 \cot^2 \alpha_1}{e}$$

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

$$R = 1 - \frac{\cot \alpha_1}{2(\cot \alpha_1 + \cot \beta_1)} \quad (3.16)$$

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, Viktor Kaplan first designed such a machine. The machines isn this family are called Kaplan turbine.

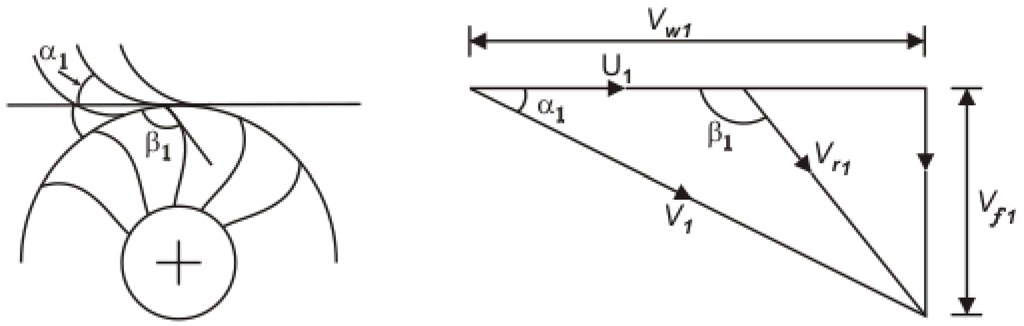
Figure 3.10 A typical Kaplan Turbine

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

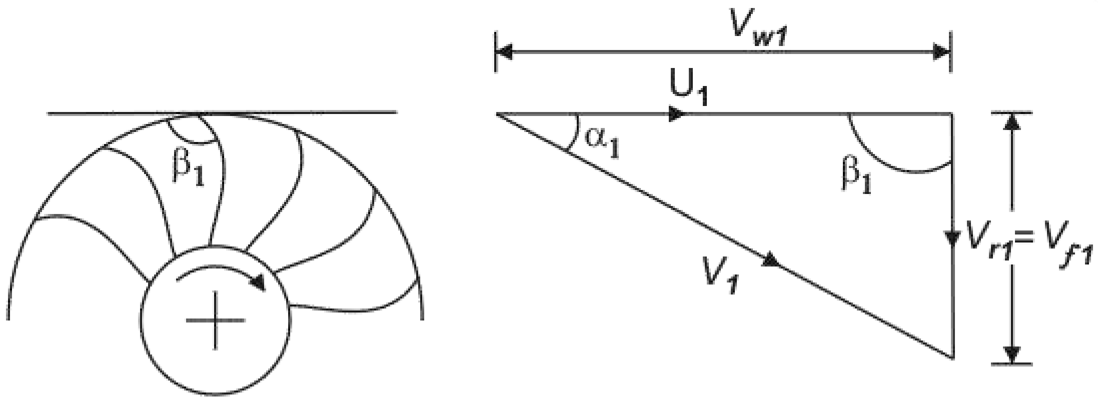
Figure 3.10 shows in stages the change in the shape of a Francis runner with the variation of specific speed. The first three types [Fig.3.11 (a), (b) and (c)] have, in order. The Francis runner (radial flow runner) at low, normal and high specific speeds. As the specific speed increases, discharge becomes more and more axial. The fourth type, as shown in Fig.3.11 (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 3.11(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 (3.11a to 3.11e), it is found that an increase in specific speed (or a decreased in head) is accompanied by a reduction in inlet velocity. But the flow velocity 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 3.11a to 3.11d) are shown in a plane containing the radial ant tangential directions, and hence the flow velocity *represents* the radial component of velocity.

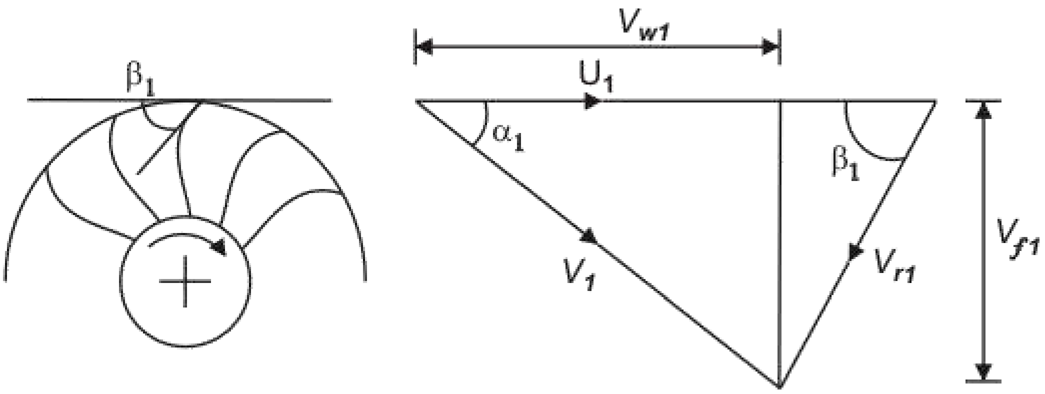
In case of a Kaplan runner, the flow at inlet is in axial and tangential directions. Therefore, the inlet velocity triangle in this case (Figure 3.11e) is shown in a place containing the axial and tangential directions, and hence the flow velocity represents the axial component of velocity .The tangential component of velocity is almost nil at outlet of all runners. Therefore, the outlet velocity triangle (Figure 3.11f) is identical in shape of all runners. However, the exit velocity is axial in Kaplan and Dubs runner, while it is the radial one in all other runners.



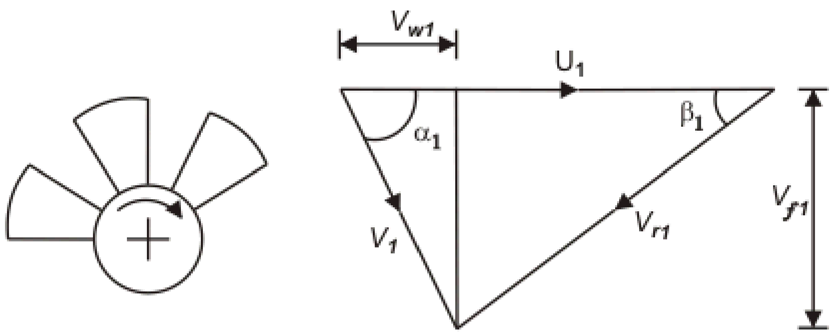
(a) Francis runner for low specific speeds



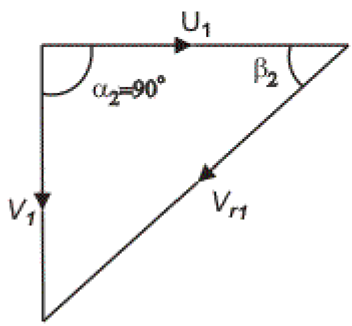
(b) Francis runner for normal specific speeds



(c) Francis runner for high specific speeds



(d) Kalpan runner



(f) Overall Velocity triangle

Figure 3.10 Velocity Triangles for Kaplan Turbine

Figure 3.11 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.

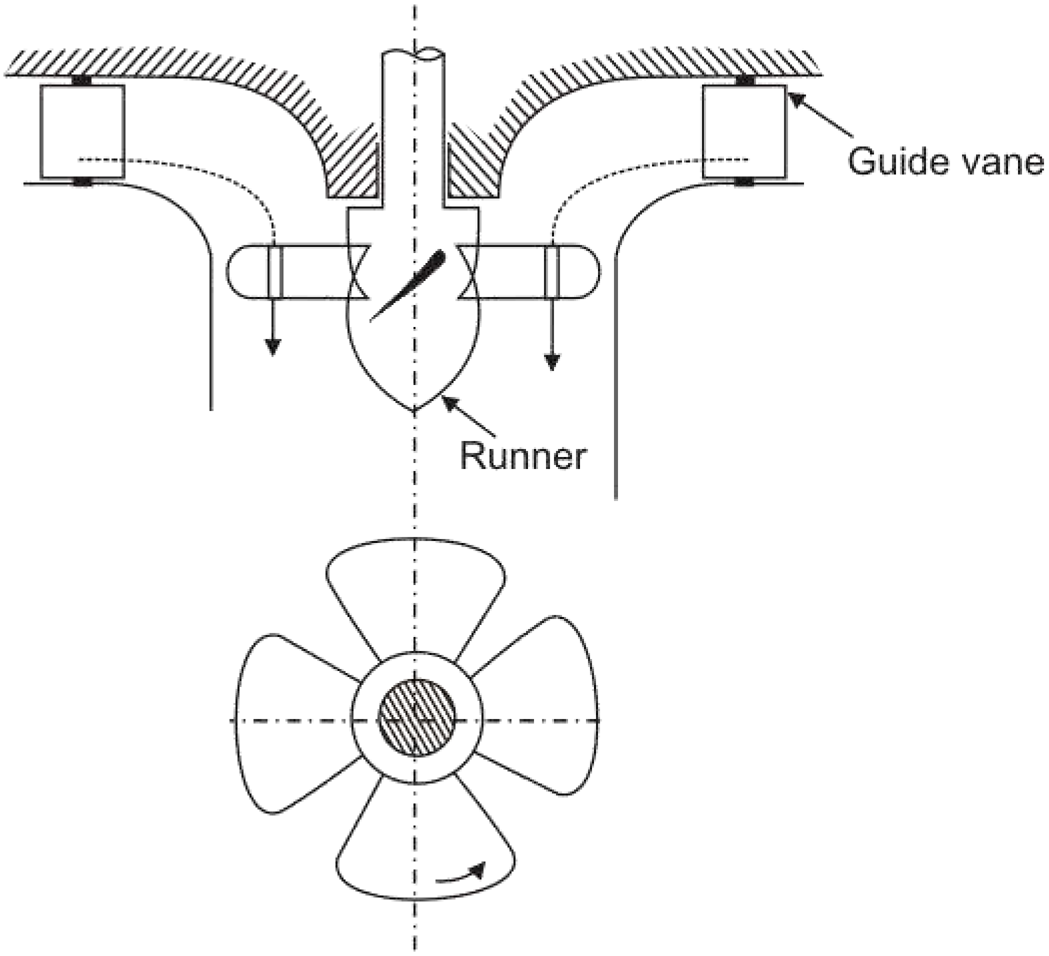


Fig. 3.11 A propeller of Kaplan turbine

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. 3.12(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. 3.12) 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.

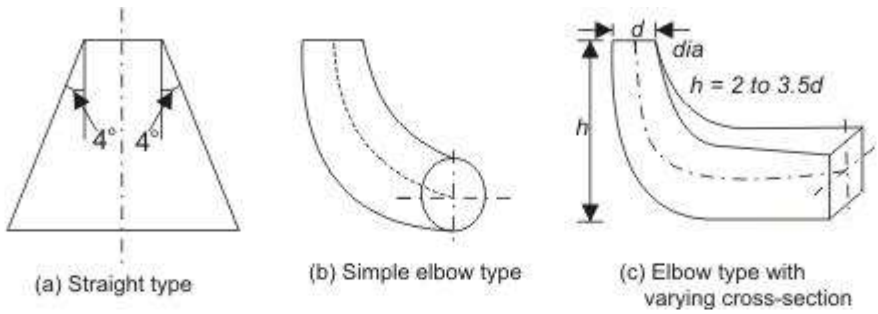


Figure 3.12 Different types of draft tubes

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. 3.13) as derived earlier from the similarity principles of fluid machines. Thus one set of curves, as shown in Fig. 3.12, is applicable not just to the conditions of the test, but to any machine in the same homologous series under any altered conditions.

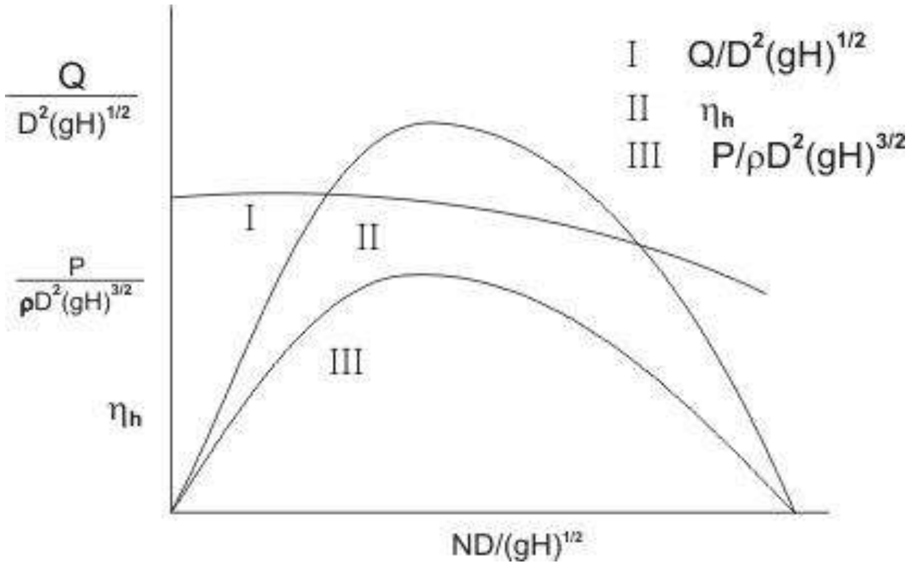


Figure 3.13 performance characteristics of a reaction turbine (in dimensionless parameters)

Figure 3.14 is one of the typical plots where variation in efficiency of different reaction turbines with the rated power is shown.

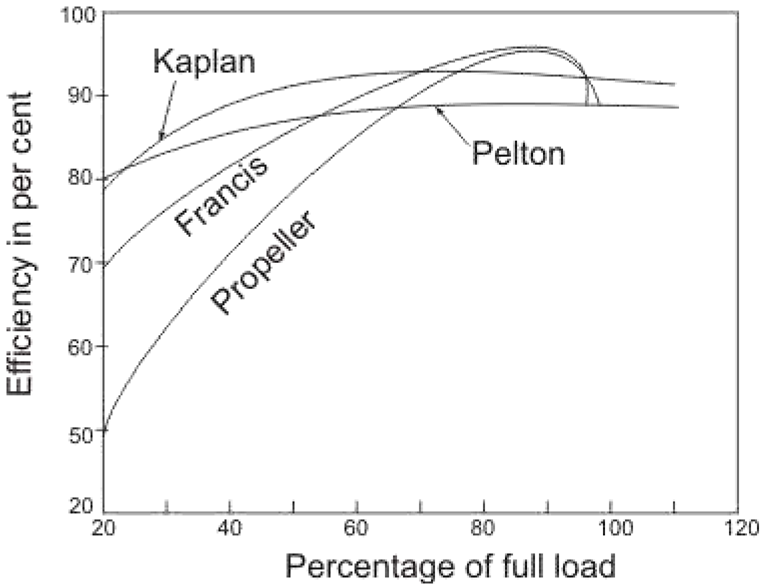


Figure 3.14 Variation of efficiency with load

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 3.15 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).

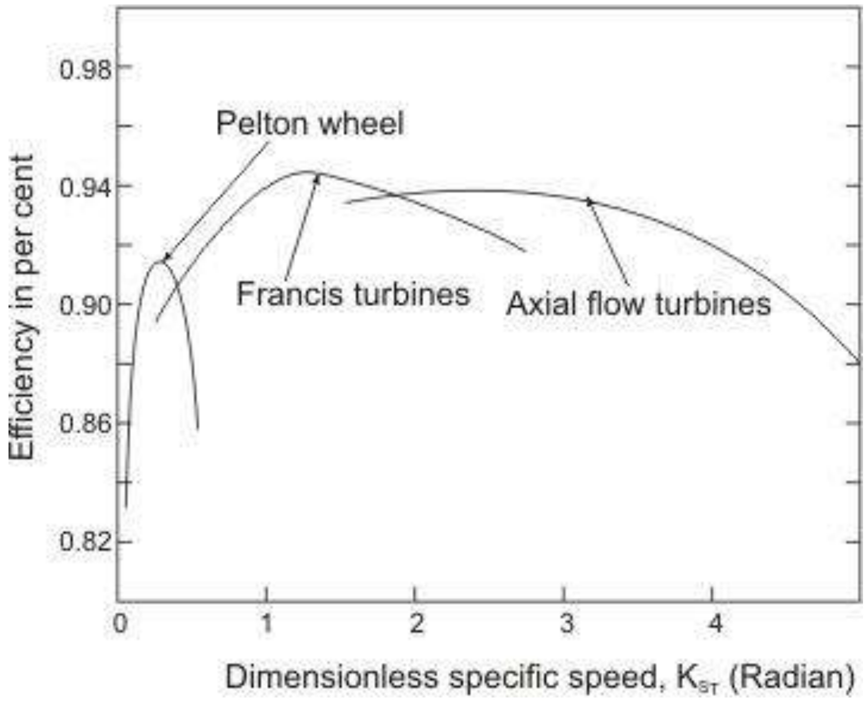


Figure 3.15 Variation of efficiency with specific speed for hydraulic turbines

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 3.16) 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.

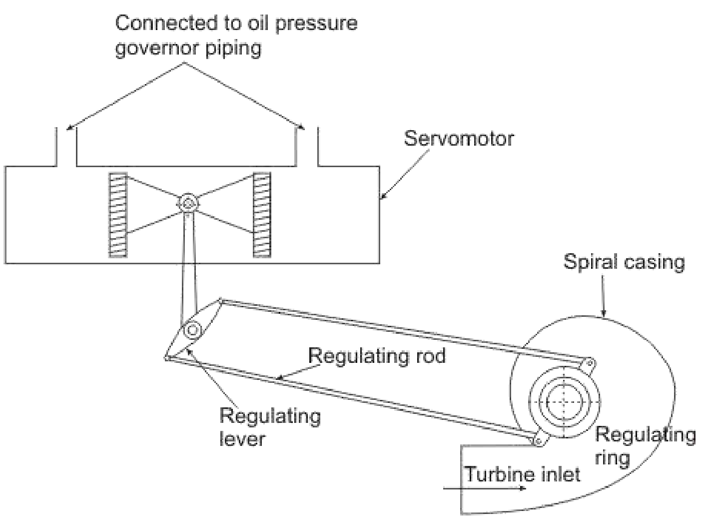


Figure 3.16 Governing of reaction turbine

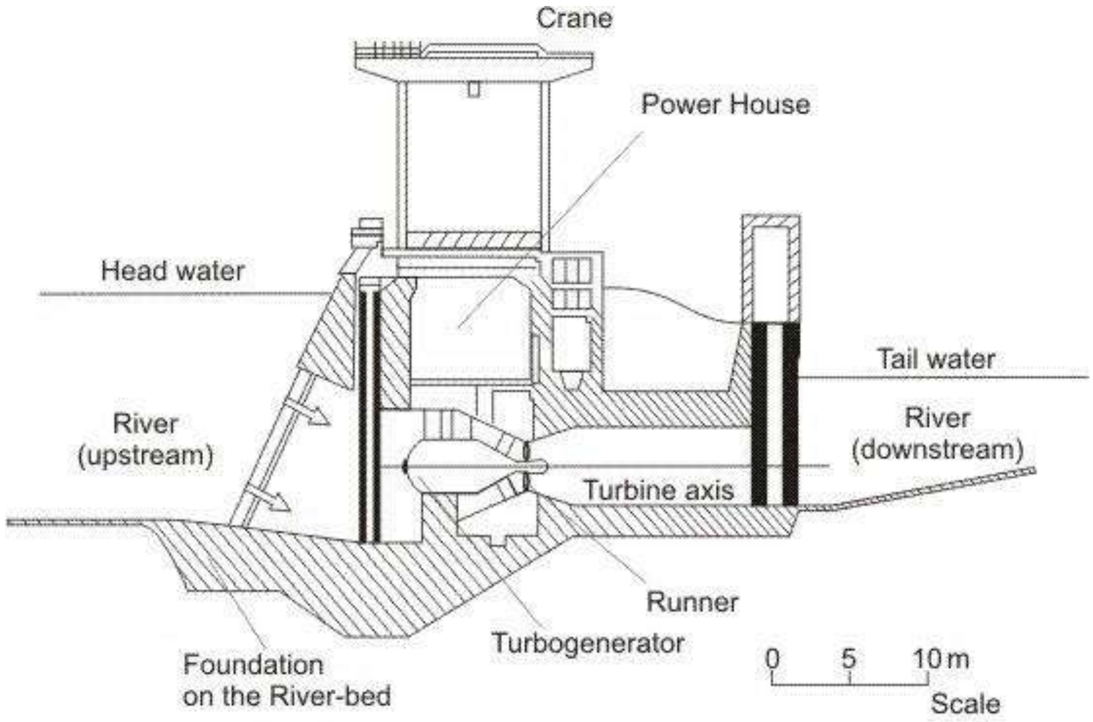


Figure 3.17 Installation of a Francis Turbine

PUMP

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. 3.18. 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 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 (Hm) 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. 3.18. The liquid enters the intake pipe causing a head loss for which the total energy line drops to point *B* corresponding to a location just after the entrance to intake pipe.

As the fluid flows from the intake to the inlet flange of the pump at elevation, the total head drops further to the point C (Figure 3.18) due to pipe friction and other losses equivalent to hf . 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 3.18) at the pump outlet (Figure 3.18).

In course of flow from the pump outlet to the upper reservoir, friction and other losses account for a total head loss or hf down to a point *E.* At *E* an exit loss He occurs when the liquid enters the upper reservoir, bringing the total heat at point *F* (Figure 3.18) 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

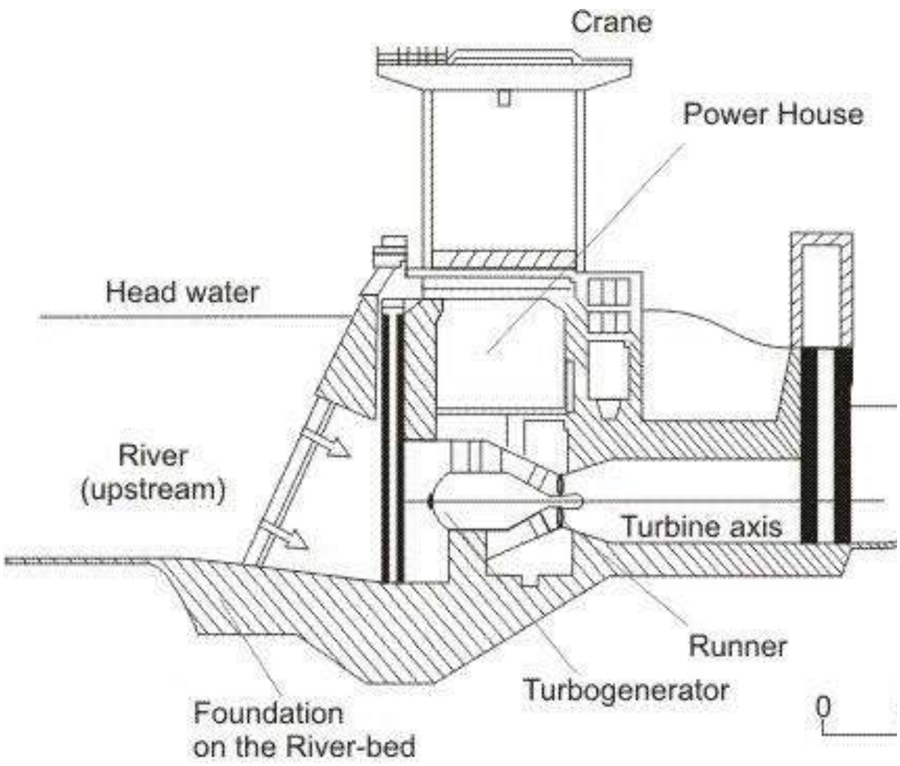


Figure 3.18 A general pumping system

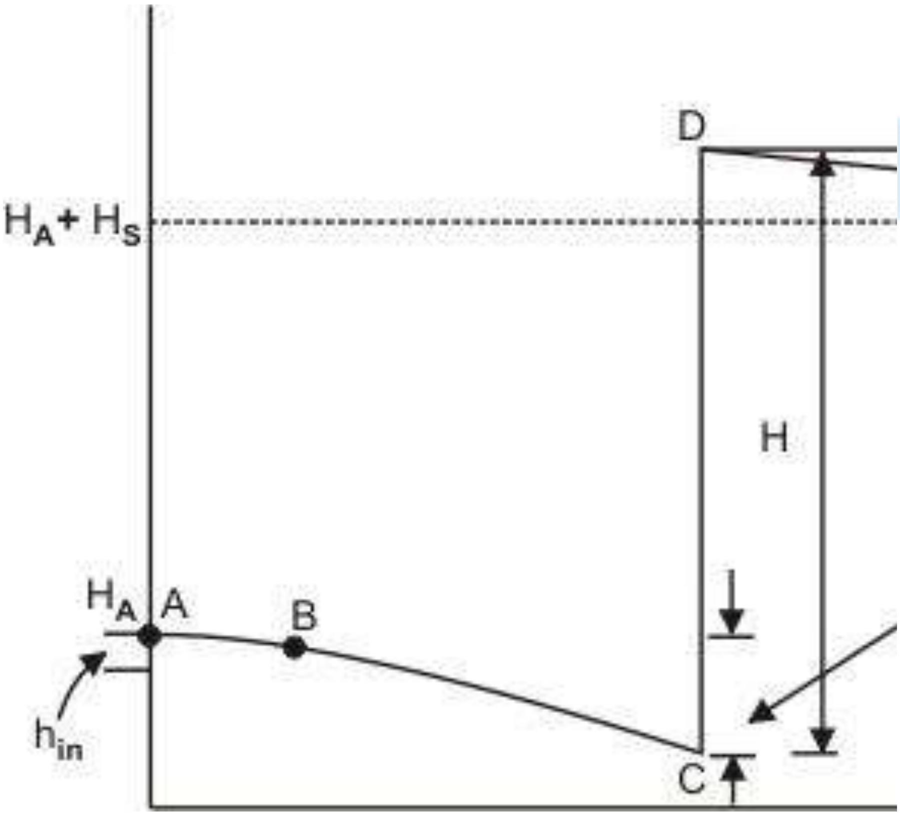


Figure 3.19 Change of head in a pumping system

Total inlet head to the pump = H_A

Total outlet head of the pump = $H_A + H_S$

The head developed H is termed as manometric head.

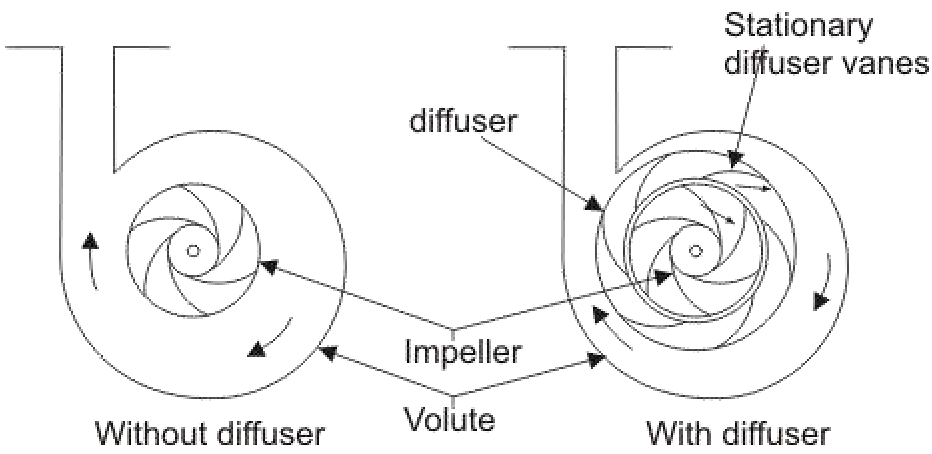
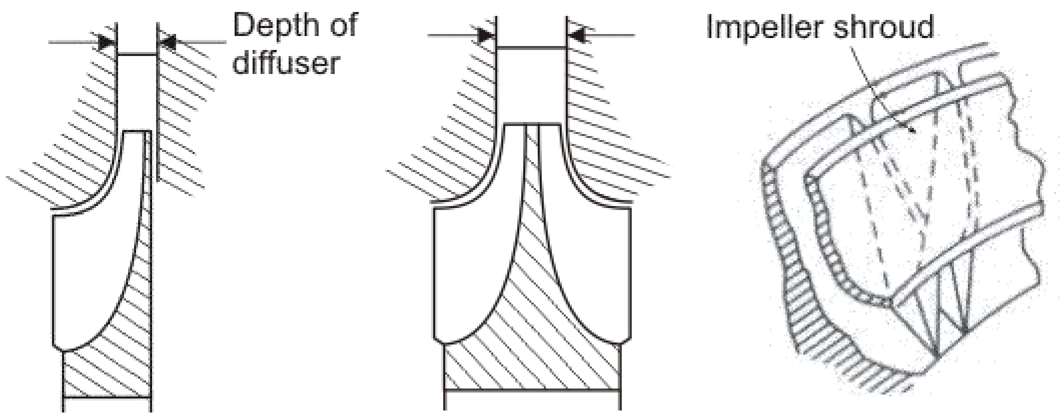


Figure 3.20 A centrifugal pump

The tips of the blades are sometimes covered by another flat disc to give shrouded blades (Figure 3.21c), 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 3.21 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 3.21 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 3.22.

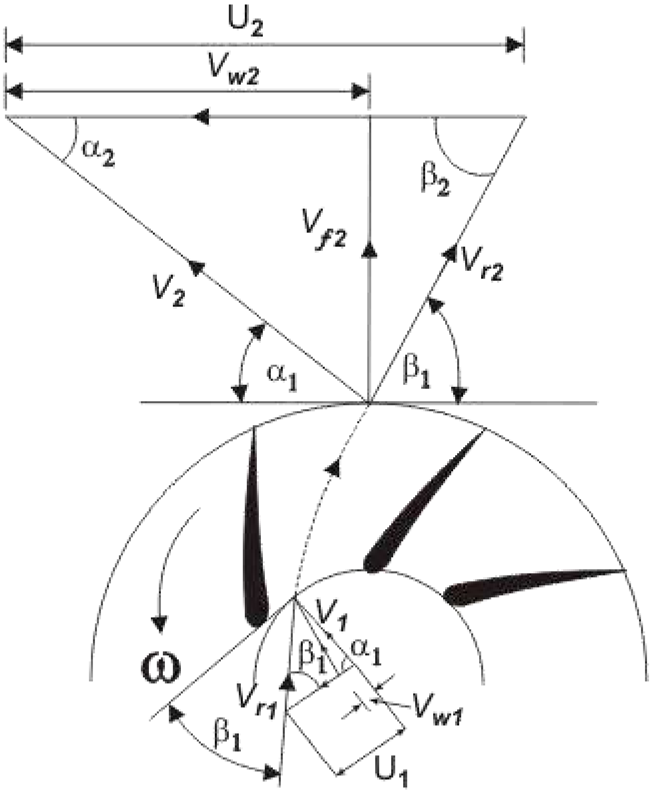


Figure 3.23 Velocity triangles for centrifugal pump Impeller

Let α 1 be the angle made by the blade at inlet, with the tangent to the inlet radius, while β1 is the blade angle with the tangent at outlet. V1 and V2 are the absolute velocities of fluid at inlet an outlet respectively, while Vr1 and Vr2 are the relative velocities (with respect to blade velocity) at inlet and outlet respectively. Therefore,

Work done on the fluid per unit weight = e/m (3.17)

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. 3.23). At conditions other than those for which the impeller was designed, the direction of relative velocity 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 Vw and accordingly the inlet angular momentum of the fluid entering the impeller is set to zero. Therefore, Eq. (3.17) can be written as

Work done on the fluid per unit weight = e/m (3.18)

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 manometric head, is always less than the quantity 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 (usually known as Euler head) is termed as manometric efficiency . It represents the effectiveness of the pump in increasing the total energy of the fluid from the energy given to it by the impeller.

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 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 to a different angle as shown in Figure 3.23 Therefore the tangential velocity component at outlet is reduced and the difference is defined as the slip.

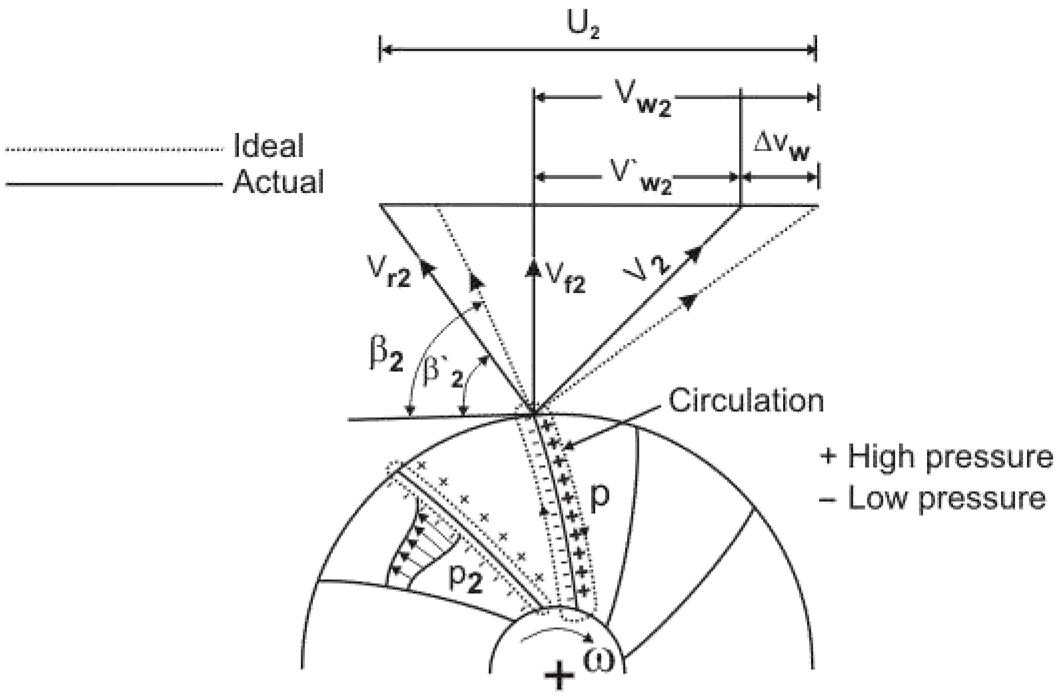


Figure 3.24 Slip and velocity in the impeller blade passage of a centrifugal pump

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. (3.24).

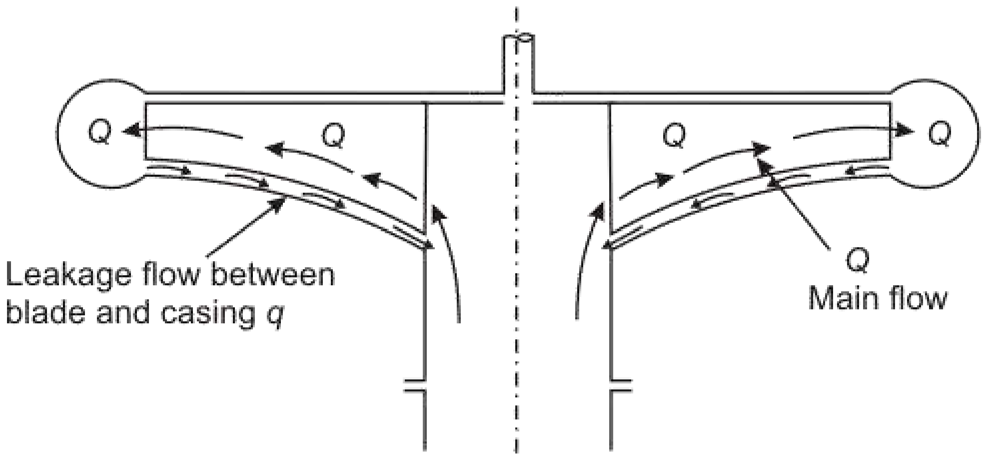


Figure 3.25 Leakage and recirculation in a centrifugal pump

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, (Fig. 3.26a), (ii) radial (Fig. 3.26b), and (iii) backward facing for which the blade curvature is in a direction opposite to that of the impeller rotation (Fig. 3.26c). The outlet velocity triangles for all the cases are also shown in Figs. 3.26a, 3.26b, 3.26c. Which was expressed earlier by Eq. (3.18)

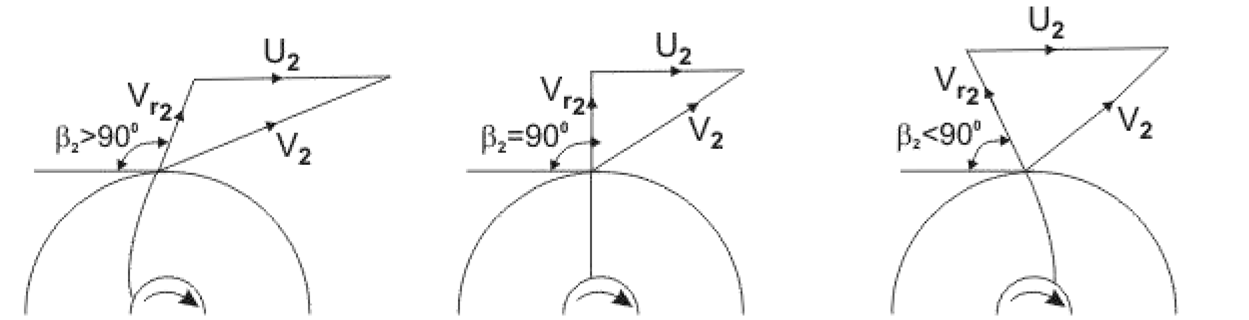


Figure 3.26 Outlet velocity triangles for different blade settings in a centrifugal pump

With the incorporation of above conditions, the relationship of head and discharge for three cases are shown in Figure 3.25. 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 3.26

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 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 underutilized 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 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).

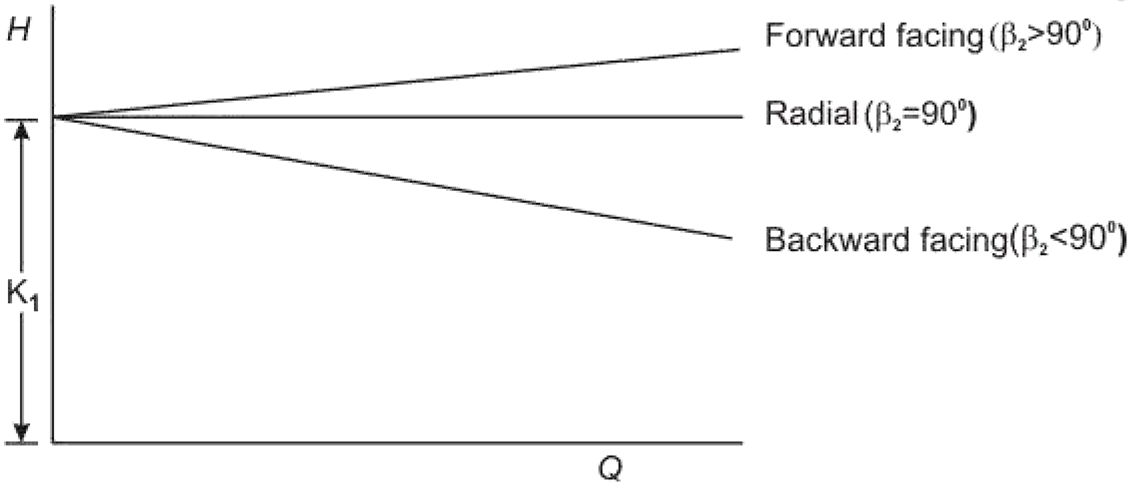


Figure 3.27 Theoretical head-discharge characteristic curves of a centrifugal pump for different blade settings

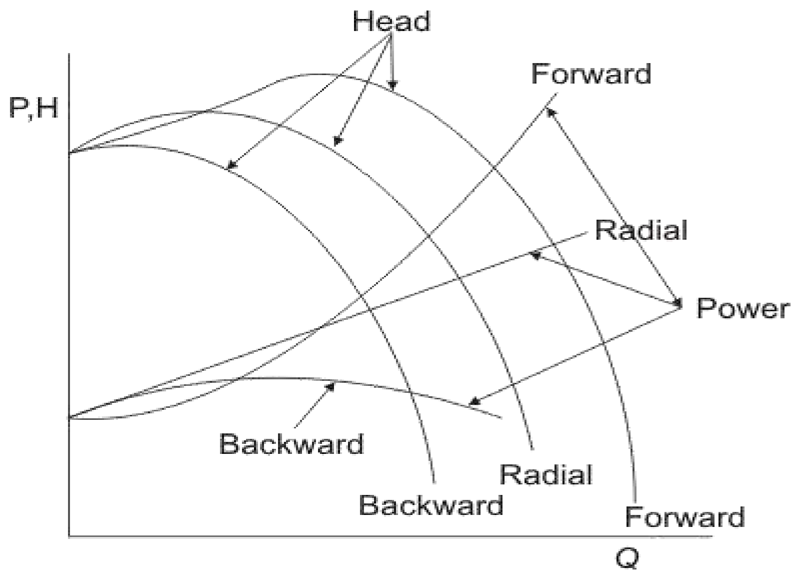


Figure 3.28 Actual head-discharge and power-discharge characteristic curves of a centrifugal pump