MODULE 1

Basic Principles of Turbomachines

Next

INTRODUCTION

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 *rotodynamic machines* . 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.

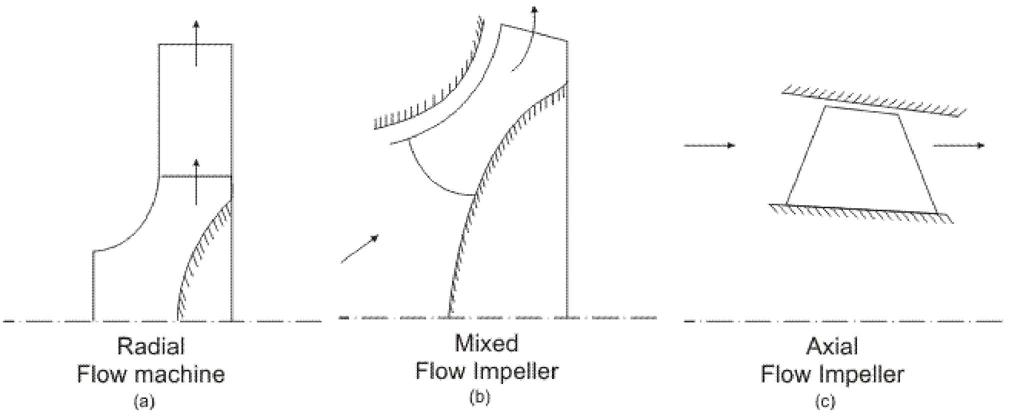


Fig. 1.1 Schematic of different types of impellers

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 absorbed by the fluid from the moving rotor.

Basic Equation of Energy Transfer in Rotodynamic Machines

The basic equation of fluid dynamics relating to energy transfer is same for all rotodynamic machines and is a simple form of " Newton 's Laws of Motion" applied to a fluid element traversing a rotor. Here we shall make use of the momentum theorem as applicable to a fluid element while flowing through fixed and moving vanes. Figure 1.2 represents diagrammatically a rotor of a generalised fluid machine, with 0-0 the axis of rotation and ω the angular velocity. Fluid enters the rotor at 1, passes through the rotor by any path and is discharged at 2. The points 1 and 2 are at radii r_1 and r_2 from the centre of the rotor, and the directions of fluid velocities at 1 and 2 may be at any arbitrary angles. For the analysis of energy transfer due to fluid flow in this situation, we assume the following:

(a) The flow is steady, that is, the mass flow rate is constant across any section (no storage or depletion of fluid mass in the rotor).

(b) The heat and work interactions between the rotor and its surroundings take place at a constant rate.

(c) Velocity is uniform over any area normal to the flow. This means that the velocity vector at any point is representative of the total flow over a finite area. This condition also implies that there is no leakage loss and the entire fluid is undergoing the same process.

The velocity at any point may be resolved into three mutually perpendicular components as shown in Fig 1.2. The axial component of velocity V_a is directed parallel to the axis of rotation , the radial component V_f is directed radially through the axis to rotation, while the tangential component V_w is directed at right angles to the radial direction and along the tangent to the rotor at that part.

The change in magnitude of the axial velocity components through the rotor causes a change in the axial momentum. This change gives rise to an axial force, which must be taken by a thrust bearing to the stationary rotor casing. The change in magnitude of radial velocity causes a change in momentum in radial direction.

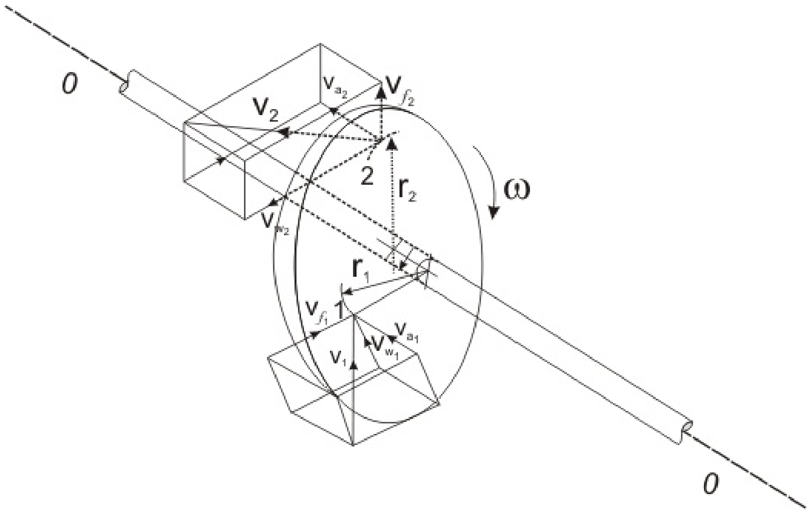


Fig 1.2 Components of flow velocity in a generalised fluid machine

However, for an axisymmetric flow, this does not result in any net radial force on the rotor. In case of a non uniform flow distribution over the periphery of the rotor in practice, a change in momentum in radial direction may result in a net radial force which is carried as a journal load. The tangential component V_w only has an effect on the angular motion of the rotor. In consideration of the entire fluid body within the rotor as a control volume, we can write from the moment of momentum theorem

$$T = m(V_{w_2} r_2 - V_{w_1} r_1) \tag{1.1}$$

where T is the torque exerted by the rotor on the moving fluid, m is the mass flow rate of fluid through the rotor. The subscripts 1 and 2 denote values at inlet and outlet of the rotor respectively. The rate of energy transfer to the fluid is then given by

$$E = T\omega = m(V_{w_2} r_2 \omega - V_{w_1} r_1 \omega) = m(V_{w_2} U_2 - V_{w_1} U_1) \tag{1.2}$$

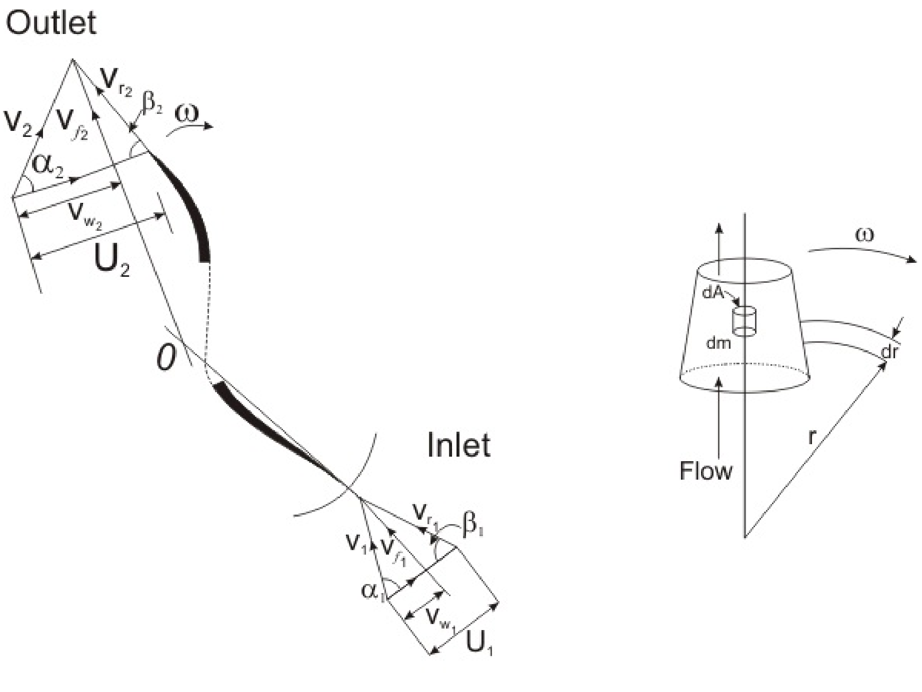
where ω is the angular velocity of the rotor and $U = \omega r$ which represents the linear velocity of the rotor. Therefore U_2 and U_1 are the linear velocities of the rotor at points 2 (outlet) and 1 (inlet) respectively (Fig. 1.2). The Eq, (1.2) is known as Euler's equation in relation to fluid machines. The Eq. (1.2) can be written in terms of head gained 'H' by the fluid as

$$H = \frac{V_{w_2} U_2 - V_{w_1} U_1}{g} \tag{1.3}$$

In usual convention relating to fluid machines, the head delivered by the fluid to the rotor is considered to be positive and vice-versa. Therefore, Eq. (1.3) written with a change in the sign of the right hand side in accordance with the sign convention as

$$H = \frac{V_{w_1} U_1 - V_{w_2} U_2}{g} \tag{1.4}$$

Components of Energy Transfer It is worth mentioning in this context that either of the Eqs. (1.2) and (1.4) is applicable regardless of changes in density or components of velocity in other directions. Moreover, the shape of the path taken by the fluid in moving from inlet to outlet is of no consequence. The expression involves only the inlet and outlet conditions. A rotor, the moving part of a fluid machine, usually consists of a number of vanes or blades mounted on a circular disc. Figure 1.3a shows the velocity triangles at the inlet and outlet of a rotor. The inlet and outlet portions of a rotor vane are only shown as a representative of the whole rotor.



(a) **(b)**

Fig 1.3 (a) Velocity triangles for a generalised rotor vane

Fig 1.3 (b) Centrifugal effect in a flow of fluid with rotation

Vector diagrams of velocities at inlet and outlet correspond to two velocity triangles, where V_r is the velocity of fluid relative to the rotor and α_1, α_2 are the angles made by the directions of the absolute velocities at the inlet and outlet respectively with the tangential direction, while β_1 and β_2 are the angles made by the relative velocities with the tangential direction. The angles β_1 and β_2 should match with vane or blade angles at inlet and outlet respectively for a smooth, shockless entry and exit of the fluid to avoid undersirable losses. Now we shall apply a simple geometrical relation as follows:

From the inlet velocity triangle,

$$V_{r_1}^2 = V_1^2 + U_1^2 - 2U_1V_1 \cos\alpha_1 = V_1^2 + U_1^2 - 2U_1V_{w_1}$$

or,
$$U_1 V_{w_1} = \frac{1}{2}(V_1^2 + U_1^2 - V_{r_1}^2) \tag{1.5}$$

Similarly from the outlet velocity triangle.

$$V_{r_2}^2 = V_2^2 + U_2^2 - 2\,U_2 V_2\,\cos\alpha_2 = V_2^2 + U_2^2 - 2U_2 V_{w_2}$$

or, $$U_2 V_{w_2} = \frac{1}{2}(V_2^2 + U_2^2 - V_{r_2}^2) \qquad (1.6)$$

Invoking the expressions of $U_1 V_{w_1}$ and $U_2 V_{w_2}$ in Eq. (1.4), we get *H* (Work head, i.e. energy per unit weight of fluid, transferred between the fluid and the rotor as) as

$$H = \frac{1}{2g}[(V_1^2 - V_2^2) + (U_1^2 - U_2^2) + (V_{r_2}^2 - V_{r_1}^2)] \qquad (1.7)$$

The Eq (1.7) is an important form of the Euler's equation relating to fluid machines since it gives the three distinct components of energy transfer as shown by the pair of terms in the round brackets. These components throw light on the nature of the energy transfer. The first term of Eq. (1.7) is readily seen to be the change in absolute kinetic energy or dynamic head of the fluid while flowing through the rotor. The second term of Eq. (1.7) represents a change in fluid energy due to the movement of the rotating fluid from one radius of rotation to another.

More About Energy Transfer in Turbomachines

Equation (1.7) can be better explained by demonstrating a steady flow through a container having uniform angular velocity ω as shown in Fig.1.3b. The centrifugal force on an infinitesimal body of a fluid of mass dm at radius r gives rise to a pressure differential dp across the thickness dr of the body in a manner that a differential force of dpdA acts on the body radially inward. This force, in fact, is the centripetal force responsible for the rotation of the fluid element and thus becomes equal to the centrifugal force under equilibrium conditions in the radial direction. Therefore, we can write

$$dp.dA = dm\,\omega^2\,r$$

with dm = dA dr ρ where ρ is the density of the fluid, it becomes

$$dp/\rho = \omega^2\,r\,dr$$

For a reversible flow (flow without friction) between two points, say, 1 and 2, the work done per unit mass of the fluid (i.e., the flow work) can be written as

$$\int_1^2 \frac{dp}{\rho} = \int_1^2 \omega^2\,r\,dr = \frac{\omega^2 r_2^2 - \omega^2 r_1^2}{2} = \frac{U_2^2 - U_1^2}{2}$$

The work is, therefore, done on or by the fluid element due to its displacement from radius r_1 to radius r_2 and hence becomes equal to the energy held or lost by it. Since the centrifugal force field is responsible for this energy transfer, the corresponding head (energy per unit weight) $U^2/2g$ is termed as centrifugal head. The transfer of energy due to a change in centrifugal head $[(U_2^2 - U_1^2)/2g]$ causes a change in the static head of the fluid.

The third term represents a change in the static head due to a change in fluid velocity relative to the rotor. This is similar to what happens in case of a flow through a fixed duct of variable cross-sectional area. Regarding the effect of flow area on fluid velocity V_r relative to the rotor, a converging passage in the direction of flow through the rotor increases the relative velocity $(V_{r2} > V_{r1})$ and hence decreases the static pressure. This usually happens in case of turbines. Similarly, a diverging passage in the direction of flow through the rotor decreases the relative velocity $(V_{r2} < V_{r1})$ and increases the static pressure as occurs in case of pumps and compressors.

The fact that the second and third terms of Eq. (1.7) correspond to a change in static head can be demonstrated analytically by deriving Bernoulli's equation in the frame of the rotor.

In a rotating frame, the momentum equation for the flow of a fluid, assumed "inviscid" can be written as

$$\rho\left[\frac{\partial \vec{v}}{\partial t} + \vec{v}\cdot\nabla\vec{v} + 2\vec{\omega}\times\vec{v} + \vec{\omega}\times(\vec{\omega}\times\vec{r})\right] = -\nabla p$$

where $\vec{v}$ is the fluid velocity relative to the coordinate frame rotating with an angular velocity $\vec{\omega}$.

We assume that the flow is steady in the rotating frame so that $\frac{\partial \vec{v}}{\partial t} = 0$. We choose a cylindrical coordinate system (r, θ, z) with z-axis along the axis of rotation. Then the momentum equation reduces to

$$\vec{v}\cdot\nabla\vec{v} + 2\omega\vec{i}_z\times\vec{v} - \omega^2 r\,\vec{i}_r = -\frac{1}{\rho}\nabla p$$

where $\vec{i}_z$ and $\vec{i}_r$ are the unit vectors along z and r direction respectively. Let $\vec{i}_s$ be a unit vector in the direction of $\vec{v}$ and s be a coordinate along the stream line. Then we can write

$$v\frac{\partial v}{\partial s}\vec{i}_s + v^2\frac{\partial \vec{i}_s}{\partial s} + 2\omega v\,\vec{i}_z\times\vec{i}_s - \omega^2 r\,i_r = -\frac{1}{\rho}\nabla p$$

More About Energy Transfer in Turbomachines

Taking scalar product with $\vec{i}_s$ it becomes

$$v\frac{\partial v}{\partial s} - \omega^2 r\frac{\partial r}{\partial s} = -\frac{1}{\rho}\frac{\partial p}{\partial s}$$

We have used $\vec{i}_s \cdot \frac{\partial \vec{i}_s}{\partial s} = 0$. With a little rearrangement, we have

$$\frac{\partial}{\partial s}\left(\frac{1}{2}v^2 - \frac{1}{2}\omega^2 r^2 + \frac{p}{\rho}\right) = 0$$

Since v is the velocity relative to the rotating frame we can replace it by V_r. Further $\omega r = U$ is the linear velocity of the rotor. Integrating the momentum equation from inlet to outlet along a streamline we have

$$\frac{1}{2}(V_{r_2}^2 - V_{r_1}^2) - \frac{1}{2}(U_2^2 - U_1^2) + \frac{p_2 - p_1}{\rho} = 0$$

or, $$\frac{1}{2}(U_1^2 - U_2^2) + \frac{1}{2}(V_{r_2}^2 - V_{r_1}^2) = \frac{p_2 - p_1}{\rho} \qquad (2.1)$$

Therefore, we can say, with the help of Eq. (2.1), that last two terms of Eq. (1.7) represent a change in the static head of fluid.

Energy Transfer in Axial Flow Machines

For an axial flow machine, the main direction of flow is parallel to the axis of the rotor, and hence the inlet and outlet points of the flow do not vary in their radial locations from the axis of rotation. Therefore, $U_1 = U_2$ and the equation of energy transfer Eq. (1.7) can be written, under this situation, as

$$H = \frac{1}{2g}[(V_1^2 - V_2^2) + (V_{r_2}^2 - V_{r_1}^2)] \qquad (2.2)$$

Hence, change in the static head in the rotor of an axial flow machine is only due to the flow of fluid through the variable area passage in the rotor.

Radially Outward and Inward Flow Machines

For radially outward flow machines, $U_2 > U_1$, and hence the fluid gains in static head, while, for a radially inward flow machine, $U_2 < U_1$ and the fluid losses its static head. Therefore, in radial flow pumps or compressors the flow is always directed radially outward, and in a radial flow turbine it is directed radially inward.

Impulse and Reaction Machines The relative proportion of energy transfer obtained by the change in static head and by the change in dynamic head is one of the important factors for classifying fluid machines. The machine for which the change in static head in the rotor is zero is known as *impulse machine* . In these machines, the energy transfer in the rotor takes place only by the change in dynamic head of the fluid. The parameter characterizing the proportions of changes in the dynamic and static head in the rotor of a fluid machine is known as degree of reaction and is defined as the ratio of energy transfer by the change in static head to the total energy transfer in the rotor.

Therefore, the degree of reaction,

$$R = \frac{\frac{1}{2g}[(U_1^2 - U_2^2) + (V_{r_2}^2 - V_{r_1}^2)]}{H} \qquad (2.3)$$

Impulse and Reaction Machines

For an impulse machine *R = 0* , because there is no change in static pressure in the rotor. It is difficult to obtain a radial flow impulse machine, since the change in centrifugal head is obvious there. Nevertheless, an impulse machine of radial flow type can be conceived by having a change in static head in one direction contributed by the centrifugal effect and an equal change in the other direction contributed by the change in relative velocity. However, this has not been established in practice. Thus for an axial flow impulse machine $U_1 = U_2$, $V_{r_1} = V_{r_2}$. For an impulse machine, the rotor can be made open, that is, the velocity V_1 can represent an open jet of fluid flowing through the rotor, which needs no casing. A very simple example of an impulse machine is a paddle wheel rotated by the impingement of water from a stationary nozzle as shown in Fig.2.1a.

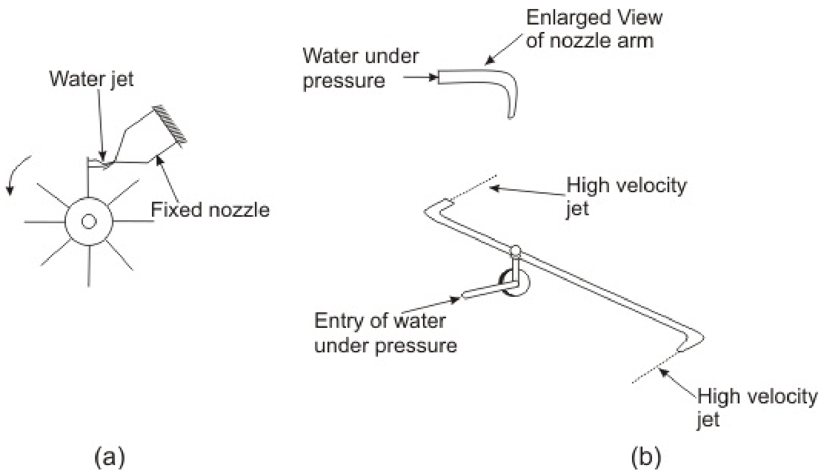


Fig 2.1 (a) Paddle wheel as an example of impulse turbine

(b) Lawn sprinkler as an example of reaction turbine

A machine with any degree of reaction must have an enclosed rotor so that the fluid cannot expand freely in all direction. A simple example of a reaction machine can be shown by the familiar lawn sprinkler, in which water comes out (Fig. 2.1b) at a high velocity from the rotor in a tangential direction. The essential feature of the rotor is that water enters at high pressure and this pressure energy is transformed into kinetic energy by a nozzle which is a part of the rotor itself.

In the earlier example of impulse machine (Fig. 2.1a), the nozzle is stationary and its function is only to transform pressure energy to kinetic energy and finally this kinetic energy is transferred to the rotor by pure impulse action. The change in momentum of the fluid in the nozzle gives rise to a reaction force but as the nozzle is held stationary, no energy is transferred by it. In the case of lawn sprinkler (Fig. 2.1b), the nozzle, being a part of the rotor, is free to move and, in fact, rotates due to the reaction force caused by the change in momentum of the fluid and hence the word ***reaction machine*** follows.

Efficiencies

The concept of efficiency of any machine comes from the consideration of energy transfer and is defined, in general, as the ratio of useful energy delivered to the energy supplied. Two efficiencies are usually considered for fluid machines-- the hydraulic efficiency concerning the energy transfer between the fluid and the rotor, and the overall efficiency concerning the energy transfer between the fluid and the shaft. The difference between the two represents the energy absorbed by bearings, glands, couplings, etc. or, in general, by pure mechanical effects which occur between the rotor itself and the point of actual power input or output.

Therefore, for a pump or compressor,

$$\eta_{hydraulic} = \eta_h = \frac{\begin{array}{l}\textit{useful energy gained by the fluid}\\ \textit{at final discharge}\end{array}}{\textit{mechanical energy supplied to rotor}} \qquad (2.4a)$$

$$\eta_{overall} = \frac{\begin{array}{l}\textit{useful energy gained by the fluid}\\ \textit{at final discharge}\end{array}}{\begin{array}{l}\textit{mechanical energy supplied to}\\ \textit{shaft at coupling}\end{array}} \qquad (2.4b)$$

For a turbine,

$$\eta_h = \frac{\textit{mechanical energy delivered by the rotor}}{\textit{energy available from the fluid}} \qquad (2.5a)$$

$$\eta_{overall} = \frac{\begin{array}{l}\textit{mechanical energy in output shaft}\\ \textit{at coupling}\end{array}}{\textit{energy available from the fluid}} \qquad (2.5b)$$

The ratio of rotor and shaft energy is represented by mechanical efficiency η_m .

Therefore

$$\eta_m = \frac{\eta_{overall}}{\eta_h} \quad (2.6)$$

Principle of Similarity and Dimensional Analysis

The principle of similarity is a consequence of nature for any physical phenomenon. By making use of this principle, it becomes possible to predict the performance of one machine from the results of tests on a geometrically similar machine, and also to predict the performance of the same machine under conditions different from the test conditions. For fluid machine, geometrical similarity must apply to all significant parts of the system viz., the rotor, the entrance and discharge passages and so on. Machines which are geometrically similar form a homologous series. Therefore, the member of such a series, having a common shape are simply enlargements or reductions of each other. If two machines are kinematically similar, the velocity vector diagrams at inlet and outlet of the rotor of one machine must be similar to those of the other. Geometrical similarity of the inlet and outlet velocity diagrams is, therefore, a necessary condition for dynamic similarity.

Let us now apply dimensional analysis to determine the dimensionless parameters, i.e., the π terms as the criteria of similarity for flows through fluid machines. For a machine of a given shape, and handling compressible fluid, the relevant variables are given in Table 3.1

Table 3.1 Variable Physical Parameters of Fluid Machine

Variable physical parameters	Dimensional formula
D = any physical dimension of the machine as a measure of the machine's size, usually the rotor diameter	L
Q = volume flow rate through the machine	$L^3 T^{-1}$
N = rotational speed (rev/min.)	T^{-1}
H = difference in head (energy per unit weight) across the machine. This may be either gained or given by the fluid depending upon whether the machine is a pump or a turbine respectively	L
ρ = density of fluid	ML^{-3}
μ = viscosity of fluid	$ML^{-1} T^{-1}$
E = coefficient of elasticity of fluid	$ML^{-1} T^{-2}$
g = acceleration due to gravity	LT^{-2}
P = power transferred between fluid and rotor (the difference between P and H is taken care of by the hydraulic efficiency η_h	$ML^2 T^{-3}$

In almost all fluid machines flow with a free surface does not occur, and the effect of gravitational force is negligible. Therefore, it is more logical to consider the energy per unit mass gH as the variable rather than H alone so that acceleration due to gravity does not appear as a separate variable. Therefore, the number of separate variables becomes eight: $D, Q, N, gH, \rho, \mu, E$ and P. Since the number of fundamental dimensions required to express these variable are three, the number of independent π terms (dimensionless terms), becomes five. Using Buckingham's π theorem with D, N and ρ as the repeating variables, the expression for the terms are obtained as,

$$\pi_1 = \frac{Q}{ND^3}, \quad \pi_2 = \frac{gH}{N^2 D^2}, \quad \pi_3 = \frac{\rho N D^2}{\mu}, \quad \pi_4 = \frac{P}{\rho N^3 D^5}, \quad \pi_5 = \frac{E/\rho}{N^2 D^2}$$

We shall now discuss the physical significance and usual terminologies of the different π terms. All lengths of the machine are proportional to D, and all areas to D^2. Therefore, the average flow velocity at any section in the machine is proportional to Q/D^2. Again, the peripheral velocity of the rotor is proportional to the product ND. The first π term can be expressed as

$$\pi_1 = \frac{Q}{ND^3} = \frac{Q/D^2}{ND} \propto \frac{\text{fluid velocity}\, V}{\text{rotor velocity}\, U}$$

Similarity and Dimensional Analysis

Thus, π_1 represents the condition for kinematic similarity, and is known as *capacity coefficient* or *discharge coefficient* The second π term π_2 is known as the *head coefficient* since it expresses the head *H* in dimensionless form. Considering the fact that *ND* $\propto$ rotor velocity, the term π_2 becomes gH/U^2 , and can be interpreted as the ratio of fluid head to kinetic energy of the rotor, Dividing π_2 by the square of π_1 we get

$$\frac{\pi_2}{\pi_1^2} = \frac{gH}{(Q/D^2)^2} \propto \frac{\textit{total fluid energy per unit mass}}{\textit{kinetic energy of the fluid per unit mass}}$$

The term π_3 can be expressed as $\rho(ND)D/\mu$ and thus represents the Reynolds number with rotor velocity as the characteristic velocity. Again, if we make the product of π_1 and π_3, it becomes $\rho(Q/D^2)D/\mu$ which represents the Reynolds's number based on fluid velocity. Therefore, if π_1 is kept same to obtain kinematic similarity, π_3 becomes proportional to the Reynolds number based on fluid velocity.

The term π_4 expresses the power *P* in dimensionless form and is therefore known as *power coefficient* . Combination of π_4 , π_1 and π_2 in the form of $\pi_4/\pi_1\pi_2$ gives $P/\rho QgH$. The term 'PQgH' represents the rate of total energy given up by the fluid, in case of turbine, and gained by the fluid in case of pump or compressor. Since *P* is the power transferred to or from the rotor. Therefore $\pi_4/\pi_1\pi_2$ becomes the hydraulic efficiency η_h for a turbine and $1/\eta_h$ for a pump or a compressor. From the fifth π term, we get

$$\frac{1}{\sqrt{\pi_5}} = \frac{ND}{\sqrt{E/\rho}}$$

Multiplying π_1, on both sides, we get

$$\frac{\pi_1}{\sqrt{\pi_5}} = \frac{Q/D^2}{\sqrt{E/\rho}} \propto \frac{\textit{fluid velocity}}{\textit{lacal asoustic velocity}}$$

Therefore, we find that $\pi_1/\sqrt{\pi_5}$ represents the well known *Mach number* , Ma.

For a fluid machine, handling incompressible fluid, the term π_5 can be dropped. The effect of liquid viscosity on the performance of fluid machines is neglected or regarded as secondary, (which is often sufficiently true for certain cases or over a limited range).Therefore the term π_3 can also be dropped.The general relationship between the different dimensionless variables (π terms) can be expressed as

$$f\left[\frac{Q}{ND^3}, \frac{gH}{N^2D^2}, \frac{E/\rho}{N^2D^2}, \frac{P}{\rho N^3D^5}\right] = 0 \qquad (3.1)$$

Therefore one set of relationship or curves of the π terms would be sufficient to describe the performance of all the members of one series.

Similarity and Dimensional Analysis

or, with another arrangement of the π terms,

$$\phi\left[\eta_h, \frac{gH}{N^2D^2}, Ma, \frac{P}{\rho N^3 D^5}\right] = 0 \qquad (3.2)$$

If data obtained from tests on model machine, are plotted so as to show the variation of dimensionless parameters $\frac{Q}{ND^3}, \frac{gH}{N^2D^2}, Ma, \frac{P}{\rho N^3D^5}$ with one another, then the graphs are applicable to any machine in the same homologous series. The curves for other homologous series would naturally be different.

Specific Speed

The performance or operating conditions for a turbine handling a particular fluid are usually expressed by the values of N, P and H, and for a pump by N, Q and H. It is important to know the range of these operating parameters covered by a machine of a particular shape (homologous series) at high efficiency. Such information enables us to select the type of machine best suited to a particular application, and thus serves as a starting point in its design. Therefore a parameter independent of the size of the machine D is required which will be the characteristic of all the machines of a homologous series. A parameter involving N, P and H but not D is obtained by dividing $(\pi_4)^{1/2}$ by $(\pi_2)^{5/4}$. Let this parameter be designated by K_{s_T} as

$$K_{s_T} = \frac{(P/\rho N^3D^5)^{1/2}}{(gH/N^2D^2)^{5/4}} = \frac{NP^{1/2}}{\rho^{1/2}(gH)^{5/4}} \qquad (3.3)$$

Similarly, a parameter involving N, Q and H but not D is obtained by divining $(\pi_1)^{1/2}$ by $(\pi_2)^{3/4}$ and is represented by K_{s_P} as

$$K_{s_P} = \frac{(Q/ND^3)^{1/2}}{(gH/N^2D^2)^{3/4}} = \frac{NQ^{1/2}}{(gH)^{3/4}} \qquad (3.4)$$

Since the dimensionless parameters K_{s_T} and K_{s_P} are found as a combination of basic π terms, they must remain same for complete similarity of flow in machines of a homologous series. Therefore, a particular value of K_{s_T} or K_{s_P} relates all the combinations of N, P and H or N, Q and H for which the flow conditions are similar in the machines of that homologous series. Interest naturally centers on the conditions for which the efficiency is a maximum. For turbines, the values of N, P and H, and for pumps and compressors, the values of N, Q and H are usually quoted for which the machines run at maximum efficiency.

The machines of particular homologous series, that is, of a particular shape, correspond to a particular value of K_s for their maximum efficient operation. Machines of different shapes have, in general, different values of K_s. Thus the parameter $K_s (K_{s_T}\ or\ K_{s_P})$ is referred to as the *shape factor* of the machines. Considering the fluids used by the machines to be incompressible, (for hydraulic turbines and pumps), and since the acceleration due to gravity dose not vary under this situation, the terms g and ρ are taken out from the expressions of K_{s_T} and K_{s_P}. The portions left as $NP^{1/2}/H^{5/4}$ and $NQ^{1/2}/H^{3/4}$ are termed, for the practical purposes, as the *specific* speed N_s for turbines or pumps. Therefore, we can write,

$$N_{s_T} \text{ (specific speed for turbines)} = NP^{1/2}/H^{5/4} \qquad (3.5)$$

$$N_{s_P} \text{ (specific speed for turbines)} = NQ^{1/2}/H^{3/4} \qquad (3.6)$$

The name specific speed for these expressions has a little justification. However a meaning can be attributed from the concept of a hypothetical machine. For a turbine, N_{s_T} is the speed of a member of the same homologous series as the actual turbine, so reduced in size as to generate unit power under a unit head of the fluid. Similarly, for a pump, N_{s_P} is speed of a hypothetical pump with reduced size but representing a homologous series so that it delivers unit flow rate at a unit head. The specific speed N_s is, therefore, not a dimensionless quantity.

The dimension of N_s can be found from their expressions given by Eqs. (3.5) and (3.6). The dimensional formula and the unit of specific speed are given as follows:

Specific speed	Dimensional formula	Unit (SI)
N_{s_T} (turbine)	$M^{1/2}\ T^{-5/2}\ L^{-1/4}$	$kg^{1/2}/\ s^{5/2}\ m^{1/4}$
N_{s_P} (pump)	$L^{3/4}\ T^{-3/2}$	$m^{3/4}\ /\ s^{3/2}$

The dimensionless parameter K_s is often known as the dimensionless specific speed to distinguish it from N_s.

MODULE 2

Gas Turbine System and Propulsion

Gas Turbine System , Centrifugal and Axial Flow Compressors

Introduction

A turbofan engine that gives propulsive power to an aircraft is shown in Figure 4.1 and the schematic of the engine is illustrated in Figure 4.2. The main components of the engine are intake, fan, compressor, combustion chamber or burnner, turbine and exhaust nozzle.

The intake is a critical part of the aircraft engine that ensures an uniform pressure and velocity at the entry to the compressor. At normal forward speed of the aircraft, the intake performs as a diffusor with rise of static pressure at the cost of kinetic energy of fluid, referred as the 'ram pressure rise'. Then the air is passed through the compressor and the high pressure air is fed to the combustion chamber, where the combustion occurs at more or less constant pressure that increases its temperature. After that the high pressure and high temperature gas is expanded through the turbine. In case of aircraft engine, the expansion in the turbine is not complete. Here the turbine work is sufficient to drive the compressor. The rest of the pressure is then expanded through the nozzle that produce the require thrust. However, in case of stationery gas turbine unit, the gas is completely expanded in the turbine. In turbofan engine the air is bypassed that has a great effect on the engine performance, which will be discussed later. Although each component have its own performance characteristics, the overall engine operates on a thermodynamic cycle.

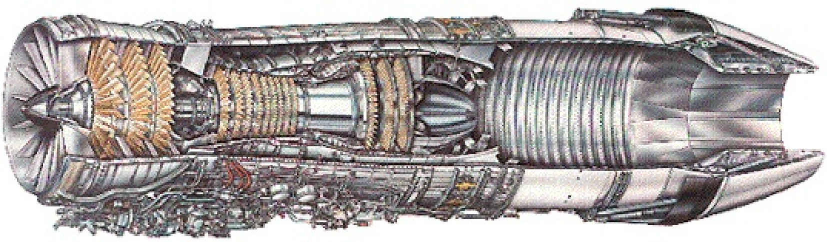


Figure 4.1 Gas Turbine (Courtesy : ae.gatech.edu)

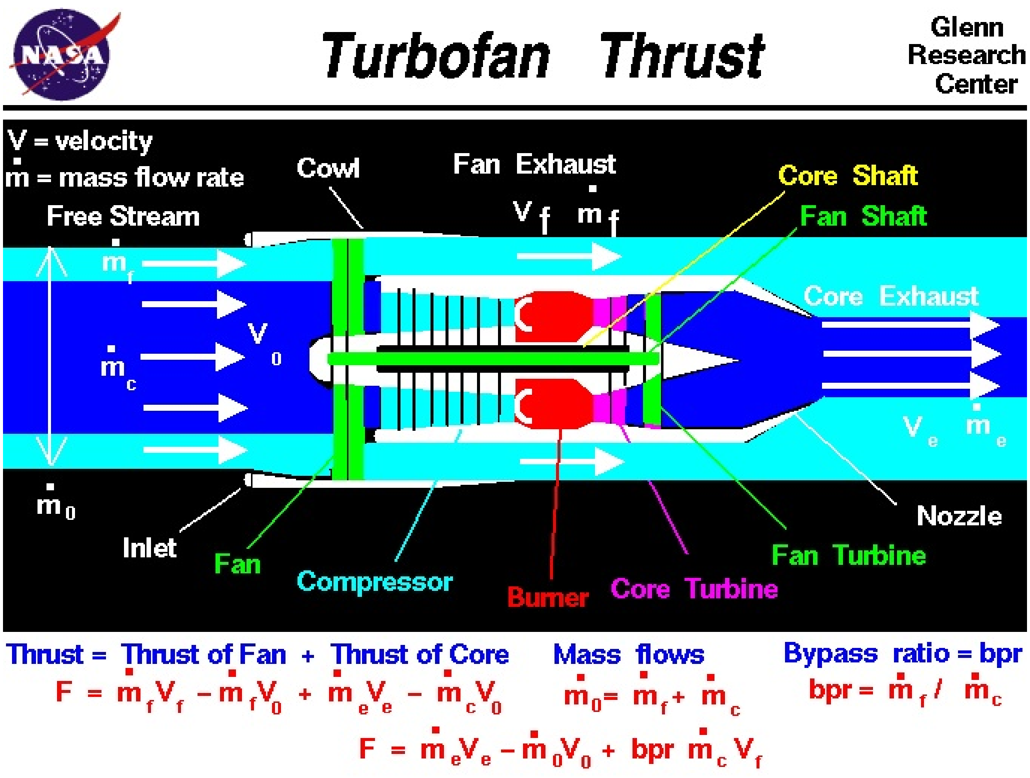


Figure 4.2 (Courtesy : NASA Glenn Research Centre)

In this chapter, we will describe the ideal gas turbine or aircraft propulsion cycles that are useful to review the performance of ideal machines in which perfection of the individual component is assumed. The specific work output and the cycle efficiency then depend only on the pressure ratio and maximum cycle temperature. Thus, this cycle analysis are very useful to find the upper limit of performance of individual components.

Following assumptions are made to analysis an ideal gas turbine cycle.

(a) The working fluid is a perfect gas with constant specific heat.

(b) Compression and expansion process are reversible and adiabatic, i.e isentropic.

(c) There are no pressure losses in the inlet duct, combustion chamber, heat exchanger, intercooler, exhaust duct and the ducts connecting the components.

(d) The mass flow is constant throughout the cycle.

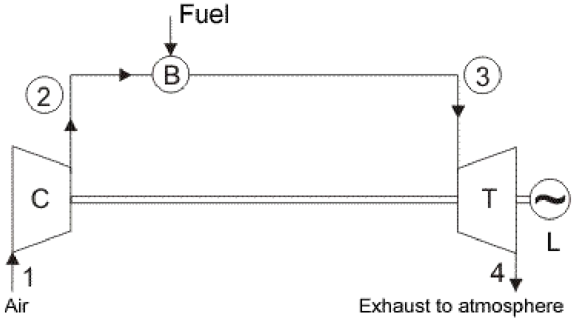
(e) The change of kinetic energy of the working fluid between the inlet and outlet of each component is negligible.

(f) The heat-exchanger, if such a component is used, is perfect.

Joule or Brayton Cycle

The ideal cycle for the simple gas turbine is the Joule or Brayton cycle which is represented by the cycle 1234 in the p-v and T-S diagram (Figure 4.3). The cycle comprises of the following process.

1-2 is the isentropic compression occuring in the compressor, 2-3 is the constant pressure heat addition in the combustion chamber, 3-4 is the isentropic expansion in the turbine releasing power output, 4-1 is the rejection of heat at constant pressure - which closes the cycle. Strictly speaking, the process 4-1 does not occur within the plant. The gases at the exit of the turbine are lost into the atmosphere; therefore it is an open cycle.



C- Compressor
B- Burner or Combustion Chamber
T- Turbine
L- Load

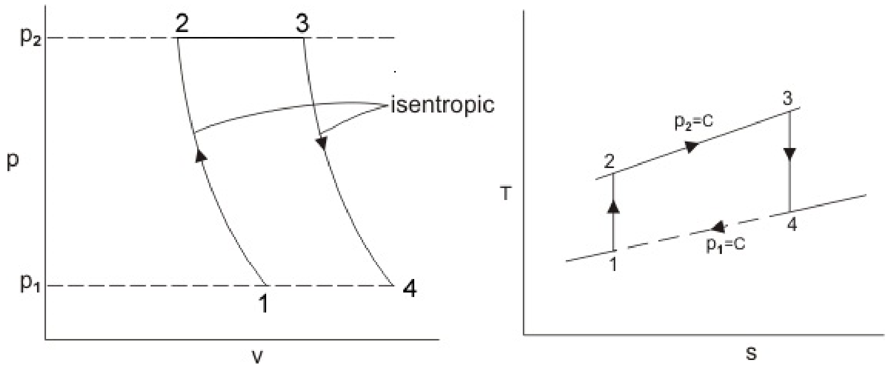


Figure 4.3 Simple gas turbine cycle.

In a steady flow isentropic process,

$$dW = dh = C_p dT$$

Thus, the

Compressor work per kg of air $W_{12} = h_2 - h_1 = C_p(T_2 - T_1)$

Turbine work per kg of air $W_{34} = h_3 - h_4 = C_p(T_3 - T_4)$

Heat supplied per kg of air $Q_{23} = h_3 - h_2 = C_p(T_3 - T_2)$

The cycle efficiency is, $\eta = \dfrac{\text{net work output}}{\text{heat supplied}} = \dfrac{W_{34} - W_{12}}{Q_{23}} = \dfrac{C_p(T_3 - T_4) - C_p(T_2 - T_1)}{C_p(T_3 - T_2)}$

or, $\eta = 1 - \dfrac{T_4 - T_1}{T_3 - T_2}$

Making use of the isentropic relation , we have,

$$\frac{T_2}{T_1} = \frac{T_3}{T_4} = \left(\frac{p_2}{p_1}\right)^{\frac{\gamma-1}{\gamma}} = (r)^{\frac{\gamma-1}{\gamma}}$$

Where, r is pressure ratio. The cycle efficiency is then given by,

$$\therefore\ \eta = 1 - \left(\frac{1}{r}\right)^{\frac{\gamma-1}{\gamma}}$$

Thus, the efficiency of a simple gas turbine depends only on the pressure ratio and the nature of the gas.
Figure 4.4 shows the relation between η and r when the working fluid is air (γ =1.4), or a monoatomic gas such as argon(γ =1.66).

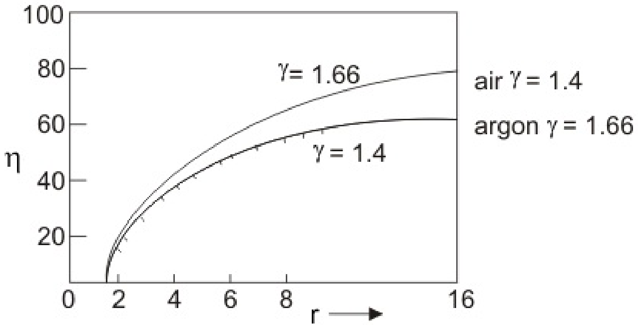


Figure 4.4 Efficiency of a simple gasturbine cycle

The specific work output w, upon which the size of plant for a given power depends, is found to be a function not only of pressure ratio but also of maximum cycle temperature T_3.

Thus, the specific work output is,

$$W = C_p(T_3 - T_4) - C_p(T_2 - T_1)$$

$$= C_p T_1\left[\frac{T_3}{T_1}\left(1-\frac{T_4}{T_3}\right)-\left(\frac{T_2}{T_1}-1\right)\right]$$

$$= C_p T_1\left[\frac{T_3}{T_1}\left\{1-\frac{1}{(\gamma)^{\gamma-1/\gamma}}\right\}-\left\{(\gamma)^{\gamma-1/\gamma}-1\right\}\right]$$

Previous Next

Let $\frac{T_3}{T_1} = t$ and $(r)^{\gamma-1/\gamma} = C$

Then $W = C_p T_1 \left[t\left(1 - \frac{1}{C}\right) - (C-1) \right]$

at $\left.\begin{array}{ll} C = 1, & W = 0 \\ C = t, & W = 0 \end{array}\right\} C = t$ means $\frac{T_3}{T_1} = (r)^{\frac{\gamma-1}{\gamma}} = \frac{T_2}{T_1}$ or $T_3 = T_2$ i.e., no heat addition

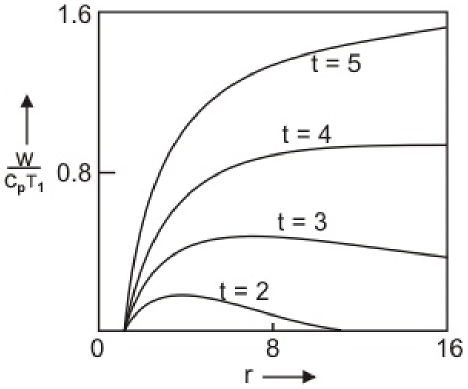


Figure 4.5 Specific work output of a simple gas turbine

To get the maximum work output for a fixed temperature ratio t and inlet temperature T_1,

$$\frac{dW}{dC} = 0 = C_p T_1 \left(\frac{t}{C^2} - 1 \right)$$

or, $$C^2 = t$$

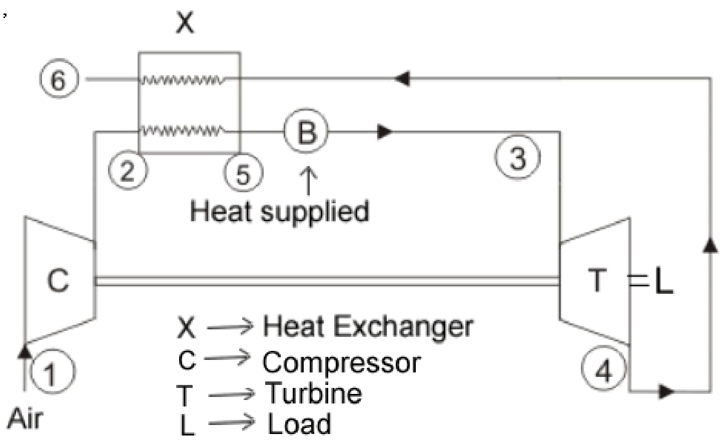
or, $$\frac{T_3}{T_1} = \frac{T_2}{T_1} \cdot \frac{T_3}{T_4}$$

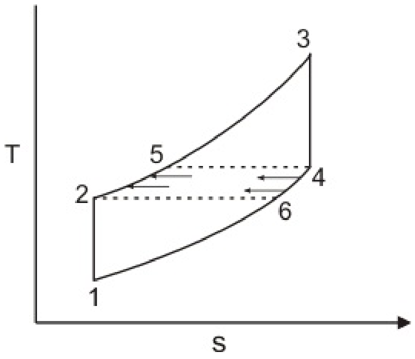
or, $$T_2 = T_4$$

Thus, the work output will be maximum when the compressor outlet temperature is equal to that of turbine. Figure 4.5 illustrates the variation of specific work output with pressure ratio for different values of temperature ratio. The work output increases with increase of T_3 for a constant value of inlet temperature T_1. However for a given temperature ratio i.e constant values of T_1 and T_3, the output becomes maximum for a particular pressure ratio.

Simple Cycle with Exhaust Heat Exchange CBTX Cycle (Regenerative cycle)

In most cases the turbine exhaust temperature is higher than the outlet temperature from the compressor. Thus the exhaust heat can be utilised by providing a heat exchanger that reduces heat input in the combustion chamber. This saving of energy increases the efficiency of the regeneration cycle keeping the specific output unchanged. A regenerative cycle is illustrated in Figure 2.6





$T_4 > T_2$ for heat exchange to take place

We assume ideal exchange $T_4 = T_5$ and $T_6 = T_2$

Figure 4.6 Simple gas turbine cycle with heat exchange

With ideal heat exchange, the cycle efficiency can be expressed as,

$\eta = \dfrac{C_p(T_3 - T_4) - C_p(T_2 - T_1)}{C_p(T_3 - T_5)}$	$\dfrac{T_2}{T_1} = \left(\dfrac{p_2}{p_1}\right)^{\frac{\gamma-1}{\gamma}} = \dfrac{T_3}{T_4}$
$= \dfrac{C_p(T_3 - T_4) - C_p(T_2 - T_1)}{C_p(T_3 - T_4)}$	or, $\dfrac{T_2}{T} = \dfrac{T_3}{T_4}$
$= 1 - \dfrac{T_2 - T_1}{T_3 - T_4} = 1 - \dfrac{T_2}{T_3}$	or, $\dfrac{T_2}{T_3} = \dfrac{T_1}{T_4}$
or, $\eta = 1 - \dfrac{T_2}{T_1} \cdot \dfrac{T_1}{T_3} = 1 - \dfrac{C}{t}$	we can write $\dfrac{T_2 - T_1}{T_3 - T_4} = \dfrac{T_2}{T_3}$

- Efficiency is more than that of simple cycle
- With heat exchange (ideal) the specific output does not change but the efficiency is increased

Previous Next

Gas Turbine Cycle with Reheat

A common method of increasing the mean temperature of heat reception is to reheat the gas after it has expanded in a part of the gas turbine. By doing so the mean temperature of heat rejection is also increased, resulting in a decrease in the thermal efficiency of the plant. However , the specific output of the plant increases due to reheat. A reheat cycle gas turbine plant is shown in Figure 5.1

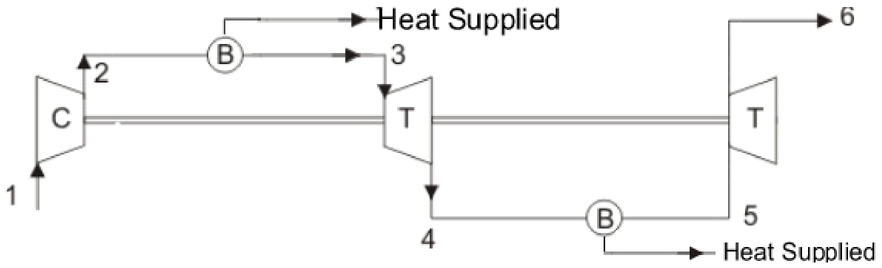


Figure 5.1 Reheat cycle gas turbine plant

The specific work output is given by,

$$= C_p(T_3 - T_4) + C_p(T_5 - T_6) - C_p(T_2 - T_1)$$

The heat supplied to the cycle is

$$= C_p(T_3 - T_2) + C_p(T_5 - T_4)$$

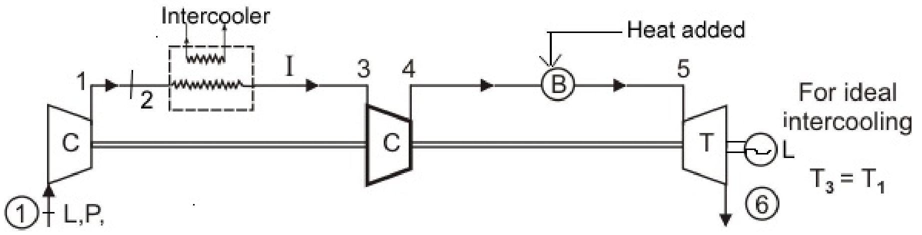
Thus, the cycle efficiency,

$$\eta = \frac{(T_3 - T_4) + (T_5 - T_6) - (T_2 - T_1)}{(T_3 - T_2) + (T_5 - T_4)}$$

Therefore, a reheat cycle is used to increase the work output while a regenerative cycle is used to enhance the efficiency.

Gas Turbine Cycle with Inter-cooling

The cooling of air between two stages of compression is known as intercooling. This reduces the work of compression and increases the specific output of the plant with a decrease in the thermal efficiency. The loss in efficiency due to intercooling can be remedied by employing exhaust heat exchange as in the reheat cycle.



Specific work output $= C_p(T_5 - T_6) - C_p(T_2 - T_1) - C_p(T_4 - T_3)$

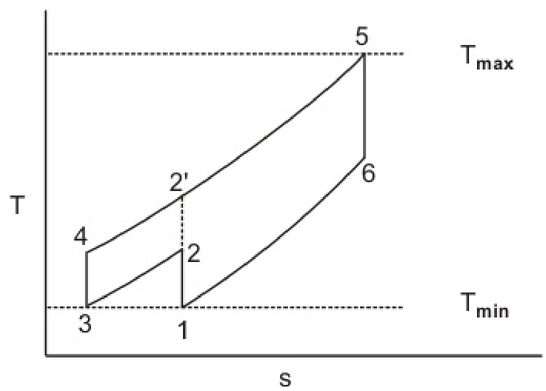


Figure 5.2 Cycle with intercooling

Heat supplied = $C_p(T_5 - T_4)$

If C_p is constant and not dependent on temperature, we can write:

$$\eta = \frac{(T_5 - T_6) - (T_2 - T_1) - (T_4 - T_3)}{(T_5 - T_4)}$$

Note $C_p(T_4 - T_3) < C_p(T_2' - T_2)$

Here heat supply and output both increases as compared to simple cycle. Because the increase in heat supply is proportionally more, η decreases.

With multiple inter-cooling and multiple reheat, the compression and expansion processes tend to be isothermal as shown in Figure 5.3

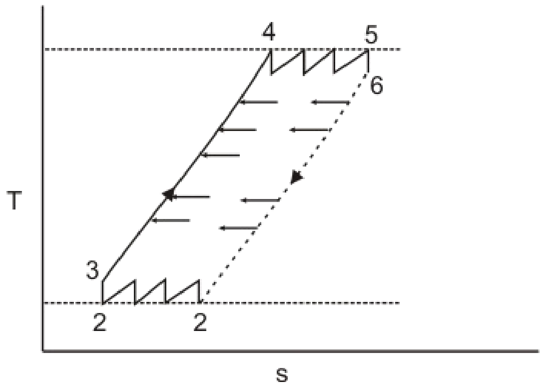


Figure 5.3 Multiple reheat and intercool cycle

The cycle tends towards the Ericsson cycle, the efficiency is same as that of the Carnot cycle

$$\eta = \frac{T_4 - T_1}{T_4} = 1 - \frac{T_1}{T_4}$$

The use of intercoolers is seldom contemplated in practice because they are bulky and need large quantities of cooling water. The main advantage of the gas turbine, that it is compact and self-contained, is then lost.

Actual Gas Turbine Cycle

- Efficiency of the compression and expansion processes will come into consideration.
- Pressure losses in the ducting, combustion and heat exchanger.
- Complete heat exchange in the regenerator is not possible.
- Mechanical losses due to bearings auxiliary etc are present.
- Specific heat of the working fluid varies with temperature.
- Mass flow throughout the cycle is not constant.

Gas Turbine Cycles for Propulsion

In aircraft gas turbine cycles, the useful power output is in the form of thrust. In case of turbojet and turbofan (Figure 4.1), the whole thrust is generated in the propelling nozzel, whereas with the turboprop most is produced by a propeller with only a small contribution from the exhaust nozzle. A turboprop and a turbojet engine is illustrated in Figure 5.4and 5.5 respectively.

(i) Turboprop

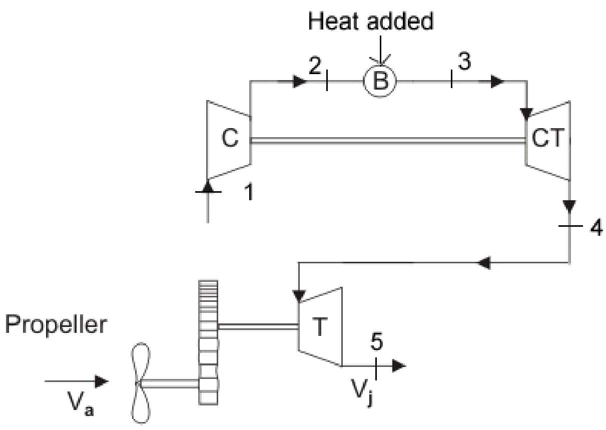


Figure 5.4 Turboprop Engine

Power must eventually be delivered to the aircraft in the form of thrust power, just as it is with a piston engine driving a propeller. The thrust power (*TP*) can be expressed in terms of shaft power (*SP*), propeller efficiency (η_p) and jet thrust *F* by

$$TP = (SP)\eta_p + FC_a$$

where C_a is the forward speed of the aircraft .

(ii) Turbjet Engine

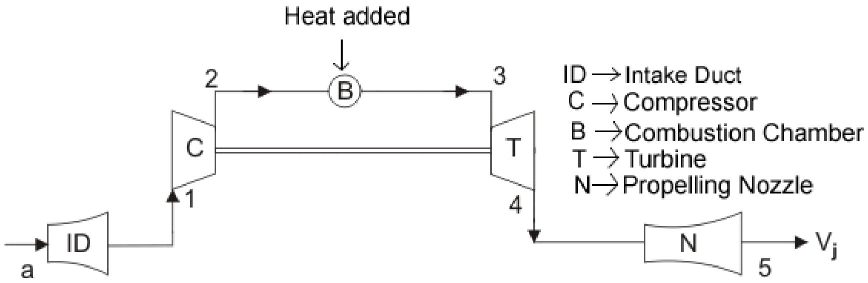


Figure 5.5 Turbojet Engine

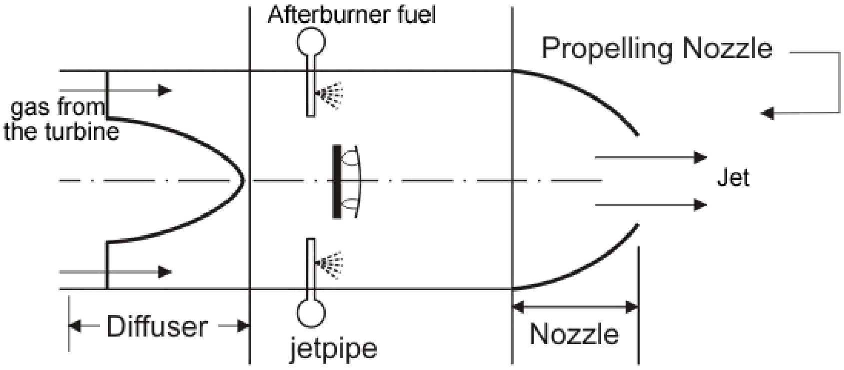
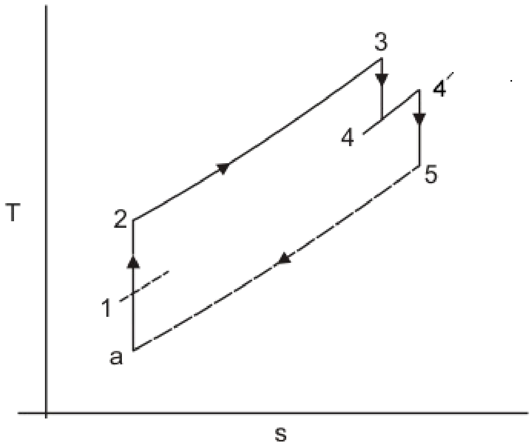


Figure 5.6 Propelling Nozzle



a-1 -> ram effect in intake.
1-2 -> isentropic compressor
2-3 -> constant pressure heat addition in the combustion chamber
3-4 -> Isentropic expansion in the turbine
4-4' ->Constant pressure heat addition in the after burner
4'-5 -> isentropic expansion in the nozzle

Figure 5.7 Turbojet Cycle

Propelling nozzle refers to the component in which the working fluid is expanded to give a high velocity jet. Between the turbine exit and propelling nozzle, there may be a jet pipe. When thrust boosting is required, an afterburner may be introduced in the jet pipe as shown in figure 5.7. Figure 5.7 indicates the ideal turbojet cycle on the T-S diagram, which is often used to evaluate the design performance of a turbojet engine.

After reviewing the thermodynamic cycle for a gas turbine or aircraft engine, characteristic features and performance of individual components such as the compressor, turbine, combustion chamber and nozzle (in case of aircraft engine) will be discussed in the following section.

Compressors

In Module 1, we discussed the basic fluid mechanical principles governing the energy transfer in a fluid machine. A brief description of different types of fluid machines using water as the working fluid was also given in Module 1.However, there exist a large number of fluid machines in practice, that use air, steam and gas (the mixture of air and products of burnt fuel) as the working fluids. The density of the fluids change with a change in pressure as well as in temperature as they pass through the machines. These machines are called 'compressible flow machines' and more popularly 'turbomachines'. Apart from the change in density with pressure, other features of compressible flow, depending upon the regimes, are also observed in course of flow of fluids through turbomachines. Therefore, the basic equation of energy transfer (Euler's equation, as discussed before) along with the equation of state relating the pressure, density and temperature of the working fluid and other necessary equations of compressible flow, are needed to describe the performance of a turbomachine. However, a detailed discussion on all types of turbomachines is beyond the scope of this book. We shall present a very brief description of a few compressible flow machines, namely, compressors, fans and blowers in this module. In practice two kinds of compressors:centrifugal and axial are generally in use.

Centrifugal Compressors

A centrifugal compressor is a radial flow rotodynamic fluid machine that uses mostly air as the working fluid and utilizes the mechanical energy imparted to the machine from outside to increase the total internal energy of the fluid mainly in the form of increased static pressure head.

During the second world war most of the gas turbine units used centrifugal compressors. Attention was focused on the simple turbojet units where low power-plant weight was of great importance. Since the war, however, the axial compressors have been developed to the point where it has an appreciably higher isentropic efficiency. Though centrifugal compressors are not that popular today, there is renewed interest in the centrifugal stage, used in conjunction with one or more axial stages, for small turbofan and turboprop aircraft engines.

A centrifugal compressor essentially consists of three components.

1. A **stationary casing**
2. A **rotating impeller** as shown in Fig. 6.1 (a) which imparts a high velocity to the air. The impeller may be single or double sided as show in Fig. 6.1 (b) and (c), but the fundamental theory is same for both.
3. A **diffuser** consisting of a number of fixed diverging passages in which the air is decelerated with a consequent rise in static pressure.

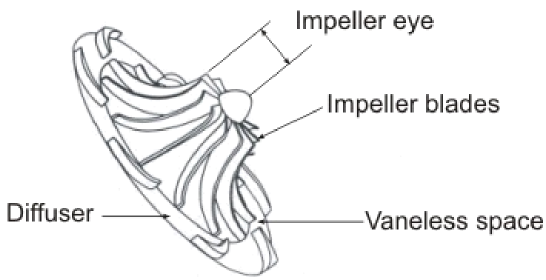
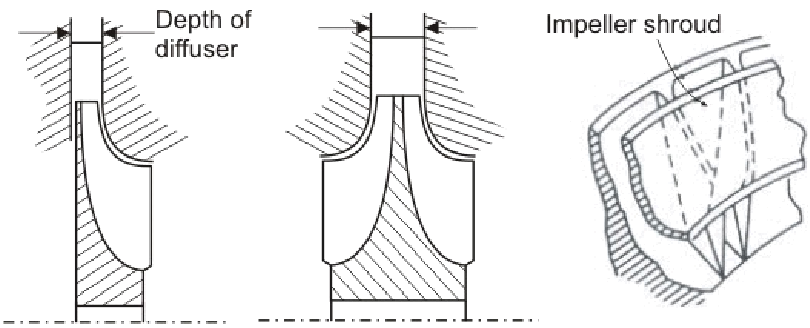


Figure 6.1(a)



(b) **(c)** **(d)**

Figure 6.1 Schematic views of a centrifugal compressor

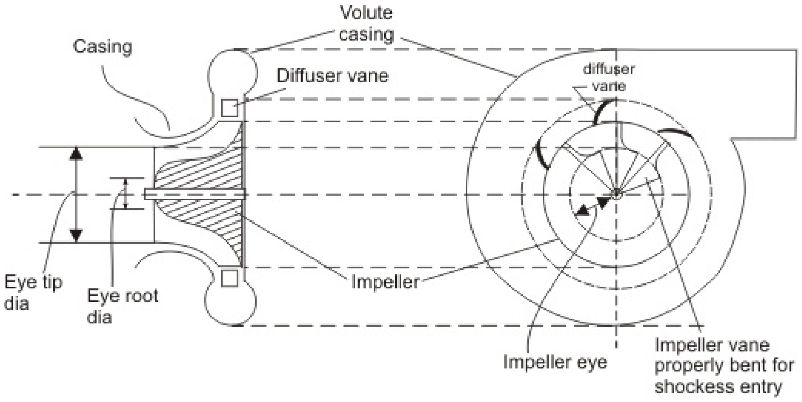


Figure 6.2 Single entry and single outlet centrifugal compresssor

Figure 6.2 is the schematic of a centrifugal compressor, where a single entry radial impeller is housed inside a volute casing.

Compressors

Principle of operation: Air is sucked into the impeller eye and whirled outwards at high speed by the impeller disk. At any point in the flow of air through the impeller the centripetal acceleration is obtained by a pressure head so that the static pressure of the air increases from the eye to the tip of the impeller. The remainder of the static pressure rise is obtained in the diffuser, where the very high velocity of air leaving the impeller tip is reduced to almost the velocity with which the air enters the impeller eye.

Usually, about half of the total pressure rise occurs in the impeller and the other half in the diffuser. Owing to the action of the vanes in carrying the air around with the impeller, there is a slightly higher static pressure on the forward side of the vane than on the trailing face. The air will thus tend to flow around the edge of the vanes in the clearing space between the impeller and the casing. This results in a loss of efficiency and the clearance must be kept as small as possible. Sometimes, a shroud attached to the blades as shown in Figure.6.1(d) may eliminate such a loss, but it is avoided because of increased disc friction loss and of manufacturing difficulties.

The straight and radial blades are usually employed to avoid any undesirable bending stress to be set up in the blades. The choice of radial blades also determines that the total pressure rise is divided equally between impeller and diffuser.

Before further discussions following points are worth mentioning for a centrifugal compresssor.

(i) The pressure rise per stage is high and the volume flow rate tends to be low. The pressure rise per stage is generally limited to 4:1 for smooth operations.

(ii) Blade geometry is relatively simple and small foreign material does not affect much on operational characteristics.

(iii) Centrifugal impellers have lower efficiency compared to axial impellers and when used in aircraft engine it increases frontal area and thus drag. Multistaging is also difficult to achieve in case of centrifugal machines.

Work done and pressure rise

Since no work is done on the air in the diffuser, the energy absorbed by the compressor will be determined by the conditions of the air at the inlet and outlet of the impeller. At the first instance, it is assumed that the air enters the impeller eye in the axial direction, so that the initial angular momentum of the air is zero. The axial portion of the vanes must be curved so that the air can pass smoothly into the eye. The angle which the leading edge of a vane makes with the tangential direction, α, will be given by the direction of the relative velocity of the air at inlet, V_{r1} , as shown in Fig. 6.3. The air leaves the impeller tip with an absolute velocity of V_2 that will have a tangential or whirl component V_{w2} . Under ideal conditions, V_2 , would be such that the whirl component is equal to the impeller speed U_2 at the tip. Since air enters the impeller in axial direction, $V_{w1} = 0$.

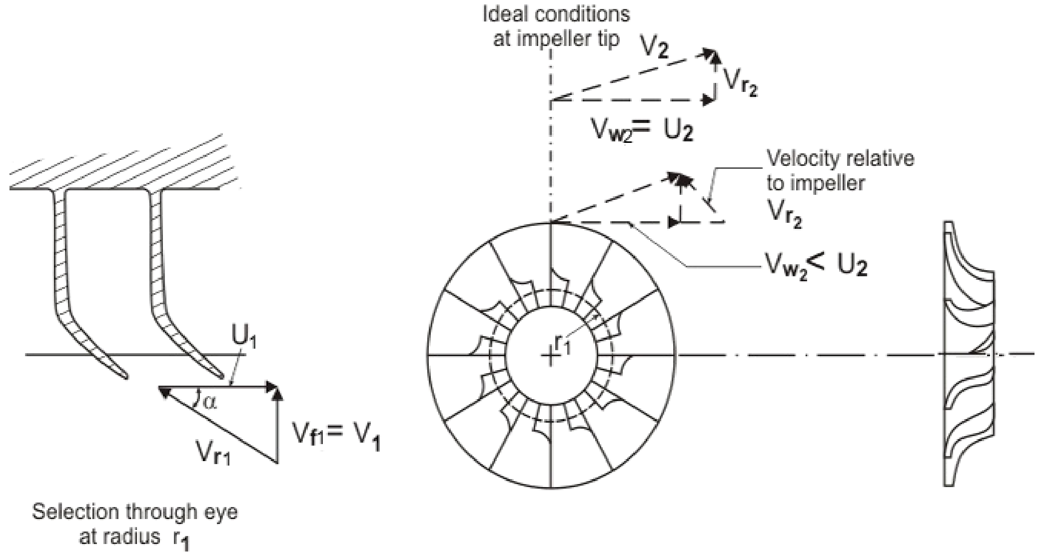


Figure 6.3 Velocity triangles at inlet and outlet of impeller blades

Under the situation of $V_{w1} = 0$ and $V_{w2} = U_2$, we can derive from Eq. (1.2), the energy transfer per unit mass of air as

$$\frac{E}{m} = U_2^2 \qquad (6.1)$$

Due to its inertia, the air trapped between the impeller vanes is reluctant to move round with the impeller and we have already noted that this results in a higher static pressure on the leading face of a vane than on the trailing face. It also prevents the air acquiring a whirl velocity equal to impeller speed. This effect is known as slip. Because of slip, we obtain $V_{w2} < U_2$. The slip factor σ is defined in the similar way as done in the case of a centrifugal pump as

$$\sigma = \frac{V_{w2}}{U_2}$$

The value of σ lies between 0.9 to 0.92. The energy transfer per unit mass in case of slip becomes

$$\frac{E}{m} = V_{w2} U_2 = \sigma U_2^2 \qquad (6.2)$$

One of the widely used expressions for σ was suggested by Stanitz from the solution of potential flow through impeller passages. It is given by

$\sigma = 1 - \frac{0.63\pi}{n}$, where n is the number of vanes.

Power Input Factor

The power input factor takes into account of the effect of disk friction, windage, etc. for which a little more power has to be supplied than required by the theoretical expression. Considering all these losses, the actual work done (or energy input) on the air per unit mass becomes

$$w = \Psi \sigma U_2^2 \tag{7.1}$$

where Ψ is the power input factor. From steady flow energy equation and in consideration of air as an ideal gas, one can write for adiabatic work *w* per unit mass of air flow as

$$w = c_p(T_{0_2} - T_{0_1}) \tag{7.2}$$

where T_{0_1} and T_{0_2} are the stagnation temperatures at inlet and outlet of the impeller, and c_p is the mean specific heat over the entire temperature range. With the help of Eq. (6.3), we can write

$$w = \Psi \sigma U_2^2 = c_p(T_{0_2} - T_{0_1}) \tag{7.3}$$

The stagnation temperature represents the total energy held by a fluid. Since no energy is added in the diffuser, the stagnation temperature rise across the impeller must be equal to that across the whole compressor. If the stagnation temperature at the outlet of the diffuser is designated by T_{0_3}, then $T_{0_3} = T_{0_2}$. One can write from Eqn. (7.3)

$$\frac{T_{0_2}}{T_{0_1}} = \frac{T_{0_3}}{T_{0_1}} = 1 + \frac{\Psi \sigma U_2^2}{c_p T_{0_1}} \tag{7.4}$$

The overall stagnation pressure ratio can be written as

$$\frac{p_{0_3}}{p_{0_1}} = \left(\frac{T_{0_{3s}}}{T_{0_1}}\right)^{\frac{\gamma}{\gamma-1}}$$

$$= \left[1 + \frac{\eta_c (T_{0_3} - T_{0_1})}{T_{0_1}}\right]^{\frac{\gamma}{\gamma-1}} \tag{7.5}$$

where, $T_{0_{3s}}$ and T_{0_3} are the stagnation temperatures at the end of an ideal (isentropic) and actual process of compression respectively (Figure 7.1), and η_c is the isentropic efficiency defined as

$$\eta_c = \frac{T_{0_{3s}} - T_{0_1}}{T_{0_3} - T_{0_1}} \tag{7.6}$$

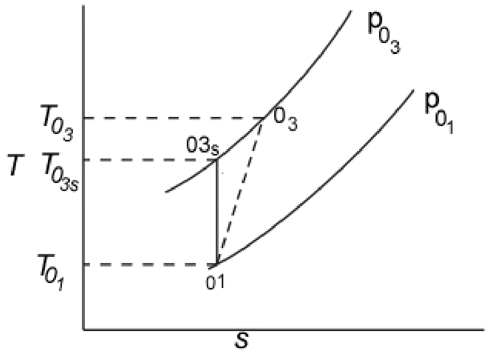


Figure 7.1 Ideal and actual processes of compression on T-s plane

Since the stagnation temperature at the outlet of impeller is same as that at the outlet of the diffuser, one can also write T_{0_2} in place of T_{0_3} in Eq. (7.6). Typical values of the power input factor lie in the region of 1.035 to 1.04. If we know η_c we will be able to calculate the stagnation pressure rise for a given impeller speed. The variation in stagnation pressure ratio across the impeller with the impeller speed is shown in Figure 7.2. For common materials, U_2 is limited to 450 m/s.

Figure 7.3 shows the inducing section of a compressor. The relative velocity V_{r1} at the eye tip has to be held low otherwise the Mach umber (based on V_{r1}) given by $M_{r1} = \frac{V_{r1}}{\sqrt{\gamma R T_1}}$ will be too high causing shock losses. Mach number M_{r1} should be in the range of 0.7-0.9. The typical inlet velocity triangles for large and medium or small eye tip diameter are shown in Figure 7.4(a) and (b) respectively.

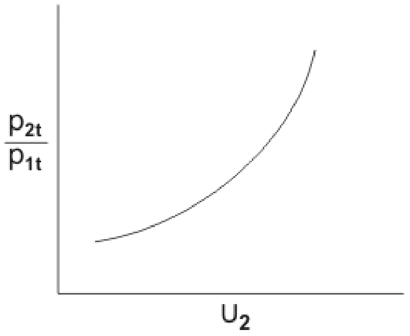


Figure 7.2 Variation in stagnation pressure ratio with impeller tip speed

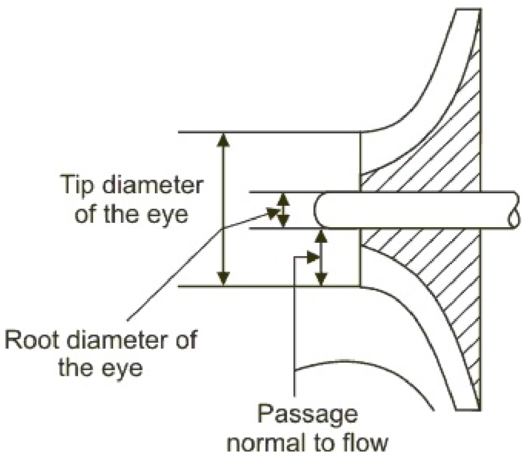


Figure 7.3 Inducing section of a centrifugal compressor

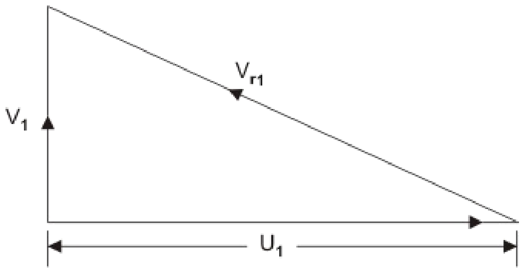


Figure 7.4 (a)

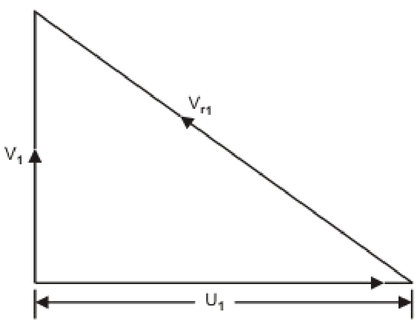


Figure 7.4(b)

Figure 7.4 Velocity triangles at the tip of eye

Diffuser

The basic purpose of a compressor is to deliver air at high pressure required for burning fuel in a combustion chamber so that the burnt products of combustion at high pressure and temperature are used in turbines or propelling nozzles (in case of an aircraft engine) to develop mechanical power. The problem of designing an efficient combustion chamber is eased if velocity of the air entering the combustion chamber is as low as possible. It is necessary, therefore to design the diffuser so that only a small part of the stagnation temperature at the compressor outlet corresponds to kinetic energy.

It is much more difficult to arrange for an efficient deceleration of flow than it is to obtain efficient acceleration. There is a natural tendency in a diffusing process for the air to break away from the walls of the diverging passage and reverse its direction. This is typically due to the phenomenon of boundary layer separation and is shown in Figure. 7.5. Experiments have shown that the maximum permissible included angle of divergence is 11° to avoid considerable losses due to flow separation.

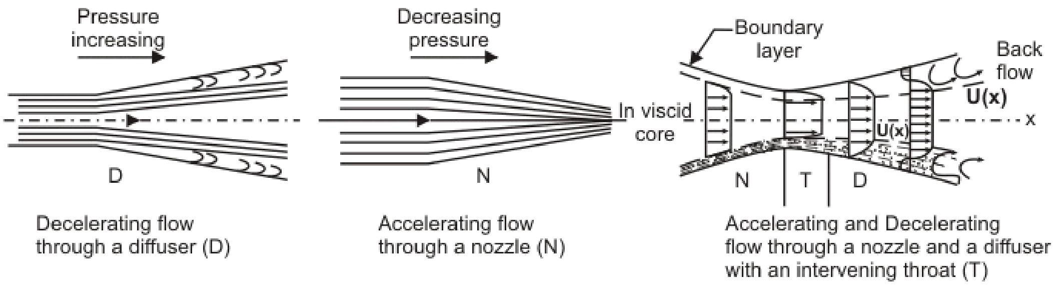


Figure 7.5 Accelerating and decelerating flows

In order to control the flow of air effectively and carry-out the diffusion process in a length as short as possible, the air leaving the impeller is divided into a number of separate streams by fixed diffuser vanes. Usually the passages formed by the vanes are of constant depth, the width diverging in accordance with the shape of the vanes. The angle of the diffuser vanes at the leading edge must be designed to suit the direction of the absolute velocity of the air at the radius of the leading edges, so that the air will flow smoothly over vanes. As there is a radial gap between the impeller tip and the leading edge of the vanes , this direction will not be that with which the air leaves the impeller tip.

To find the correct angle for diffuser vanes, the flow in the vaneless space should be considered. No further energy is supplied to the air after it leaves the impeller. If we neglect the frictional losses, the angular momentum $V_w r$ remains constant. Hence V_w decreases from impeller tip to diffuser vane, in inverse proportion to the radius. For a channel of constant depth, the area of flow in the radial direction is directly proportional to the radius. The radial velocity V_r will therefore also decrease from impeller tip to diffuser vane, in accordance with the equation of continuity. If both V_r and V_w decrease from the impeller tip then the resultant velocity V decreases from the impeller tip and some diffusion takes place in the vaneless space. The consequent increase in density means that V_r will not decrease in inverse proportion to the radius as done by V_w, and the way V_r varies must be found from the equation of continuity.

Losses in a Centrifugal Compressor

The losses in a centrifugal compressor are almost of the same types as those in a centrifugal pump. However, the following features are to be noted.

Frictional losses: A major portion of the losses is due to fluid friction in stationary and rotating blade passages. The flow in impeller and diffuser is decelerating in nature. Therefore the frictional losses are due to both skin friction and boundary layer separation. The losses depend on the friction factor, length of the flow passage and square of the fluid velocity. The variation of frictional losses with mass flow is shown in Figure. 8.1.

Incidence losses: During the off-design conditions, the direction of relative velocity of fluid at inlet does not match with the inlet blade angle and therefore fluid cannot enter the blade passage smoothly by gliding along the blade surface. The loss in energy that takes place because of this is known as incidence loss. This is sometimes referred to as shock losses. However, the word shock in this context should not be confused with the aerodynamic sense of shock which is a sudden discontinuity in fluid properties and flow parameters that arises when a supersonic flow decelerates to a subsonic one.

Clearance and leakage losses: Certain minimum clearances are necessary between the impeller shaft and the casing and between the outlet periphery of the impeller eye and the casing. The leakage of gas through the shaft clearance is minimized by employing glands. The clearance losses depend upon the impeller diameter and the static pressure at the impeller tip. A larger diameter of impeller is necessary for a higher peripheral speed (U_2) and it is very difficult in the situation to provide sealing between the casing and the impeller eye tip.

The variations of frictional losses, incidence losses and the total losses with mass flow rate are shown in Figure.8.1

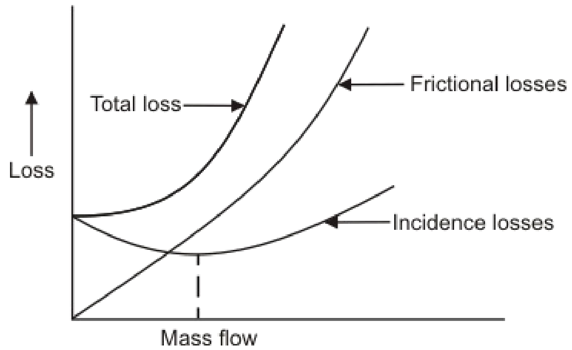


Figure 8.1 Dependence of various losses with mass flow in a centrifugal compressor

The leakage losses comprise a small fraction of the total loss. The incidence losses attain the minimum value at the designed mass flow rate. The shock losses are, in fact zero at the designed flow rate. However, the incidence losses, as shown in Fig. 8.1, comprises both shock losses and impeller entry loss due to a change in the direction of fluid flow from axial to radial direction in the vaneless space before entering the impeller blades. The impeller entry loss is similar to that in a pipe bend and is very small compared to other losses. This is why the incidence losses show a non zero minimum value (Figure. 8.1) at the designed flow rate.

Compressor characteristics

The theoretical and actual head-discharge relationships of a centrifugal compressor are same as those of a centrifugal pump as described in Module 1. However, the performance of a compressor is usually specified by curves of delivery pressure and temperature against mass flow rate for various fixed values of rotational speed at given values of inlet pressure and temperature. It is always advisable to plot such performance characteristic curves with dimensionless variables. To find these dimensionless variables, we start with a implicit functional relationship of all the variables as

$$F(D, N, m, p_{0_1}, p_{0_2}, RT_{0_1}, RT_{0_2}) = 0 \qquad (8.1)$$

where D = characteristic linear dimension of the machine, N = rotational, m = mass flow rate, p_{0_1} = stagnation pressure at compressor inlet, p_{0_2} = stagnation pressure at compressor outlet, T_{0_1} = stagnation temperature at compressor inlet, T_{0_2} = stagnation temperature at compressor outlet, and R = characteristics gas constant.

By making use of Buckingham's π theorem, we obtain the non-dimensional groups (π terms) as

$$\frac{p_{0_2}}{p_{0_1}}, \frac{T_{0_2}}{T_{0_1}}, \frac{m\sqrt{RT_{0_1}}}{D^2 p_{0_1}}, \frac{ND}{\sqrt{RT_{0_1}}}$$

The third and fourth non-dimensional groups are defined as 'non-dimensional mass flow' and 'non-dimensional rotational speed' respectively. The physical interpretation of these two non-dimensional groups can be ascertained as follows.

$$\frac{m\sqrt{RT}}{D^2 p} = \frac{\rho AV\sqrt{RT}}{D^2 p} = \frac{p}{RT}, \frac{AV\sqrt{RT}}{D^2 p} \propto, \frac{V}{\sqrt{RT}} \propto M_F$$

$$\frac{ND}{\sqrt{RT}} = \frac{U}{\sqrt{RT}} \propto M_R$$

Therefore, the 'non-dimensional mass flow' and 'non-dimensional rotational speed' can be regarded as flow Mach number, M_F and rotational speed Mach number, M_R.

When we are concerned with the performance of a machine of fixed size compressing a specified gas, R and D may be omitted from the groups and we can write

$$\text{Function}\left(\frac{p_{2t}}{p_{1t}}, \frac{T_{2t}}{T_{1t}}, \frac{m\sqrt{T_{0_1}}}{p_{0_1}}, \frac{N}{\sqrt{T_{0_1}}}\right) = 0 \qquad (8.2)$$

Though the terms $m\sqrt{T_{0_1}}/p_{0_1}$ and $N/\sqrt{T_{0_1}}$ are truly not dimensionless, they are referred as 'non-dimensional mass flow' and 'non-dimensional rotational speed' for practical purpose. The stagnation pressure and temperature ratios p_{0_2}/p_{0_1} and T_{0_2}/T_{0_1} are plotted against $m\sqrt{T_{0_1}}/p_{0_1}$ in the form of two families of curves, each curve of a family being drawn for fixed values of $N/\sqrt{T_{0_1}}$. The two families of curves represent the compressor characteristics. From these curves, it is possible to draw the curves of isentropic efficiency $\eta_C \; vs \; m\sqrt{T_{0_1}}/p_{0_1}$ for fixed values of $N/\sqrt{T_{0_1}}$. We can recall, in this context, the definition of the isentropic efficiency as

$$\eta_C = \frac{T_{0_{2s}} - T_{0_1}}{T_{0_2} - T_{0_1}} = \frac{(p_{0_2}/p_{0_1})^{\frac{\gamma-1}{\gamma}} - 1}{(T_{0_2}/T_{0_1}) - 1} \qquad (8.3)$$

Before describing a typical set of characteristics, it is desirable to consider what might be expected to occur when a valve placed in the delivery line of the compressor running at a constant speed, is slowly opened. When the valve is shut and the mass flow rate is zero, the pressure ratio will have some value. Figure 8.2 indicates a theoretical characteristics curve ABC for a constant speed.

The centrifugal pressure head produced by the action of the impeller on the air trapped between the vanes is represented by the point 'A' in Figure 8.2. As the valve is opened, flow commences and diffuser begins to influence the pressure rise, for which the pressure ratio increases. At some point *'B'*, efficiency approaches its maximum and the pressure ratio also reaches its maximum. Further increase of mass flow will result in a fall of pressure ratio. For mass flows greatly in excess of that corresponding to the design mass flow, the air angles will be widely different from the vane angles and breakaway of the air will occur. In this hypothetical case, the pressure ratio drops to unity at *'C'* , when the valve is fully open and all the power is absorbed in overcoming internal frictional resistances.

In practice, the operating point 'A' could be obtained if desired but a part of the curve between *'A'* and 'B' could not be obtained due to surging. It may be explained in the following way. If we suppose that the compressor is operating at a point 'D' on the part of characteristics curve (Figure 8.2) having a positive slope, then a decrease in mass flow will be accompanied by a fall in delivery pressure. If the pressure of the air downstream of the compressor does not fall quickly enough, the air will tend to reverse its direction and will flow back in the direction of the resulting pressure gradient. When this occurs, the pressure ratio drops rapidly causing a further drop in mass flow until the point 'A' is reached, where the mass flow is zero. When the pressure downstream of the compressor has reduced sufficiently due to reduced mass flow rate, the positive flow becomes established again and the compressor picks up to repeat the cycle of events which occurs at high frequency.

This surging of air may not happen immediately when the operating point moves to the left of 'B' because the pressure downstream of the compressor may at first fall at a greater rate than the delivery pressure. As the mass flow is reduced further, the flow reversal may occur and the conditions are unstable between 'A' and 'B'. As long as the operating point is on the part of the characteristics having a negative slope, however, decrease in mass flow is accompanied by a rise in delivery pressure and the operation is stable.

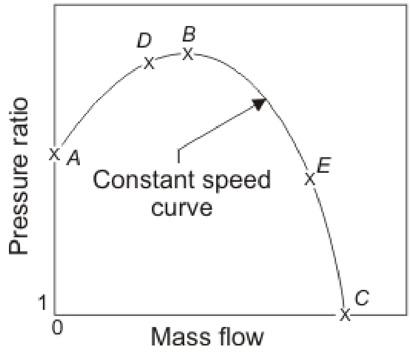


Figure 8.2 The theoretical characteristic curve

There is an additional limitation to the operating range, between 'B' and 'C'. As the mass flow increases and the pressure decreases, the density is reduced and the radial component of velocity must increase. At constant rotational speed this means an increase in resultant velocity and hence an angle of incidence at the diffuser vane leading edge. At some point say 'E', the position is reached where no further increase in mass flow can be obtained no matter how wide open the control valve is. This point represents the maximum delivery obtainable at the particular rotational speed for which the curve is drawn. This indicates that at some point within the compressor sonic conditions have been reached, causing the limiting maximum mass flow rate to be set as in the case of compressible flow through a converging diverging nozzle. Choking is said to have taken place. Other curves may be obtained for different speeds, so that the actual variation of pressure ratio over the complete range of mass flow and rotational speed will be shown by curves such as those in Figure. 8.3. The left hand extremities of the constant speed curves may be joined up to form surge line, the right hand extremities indicate choking (Figure 8.3).

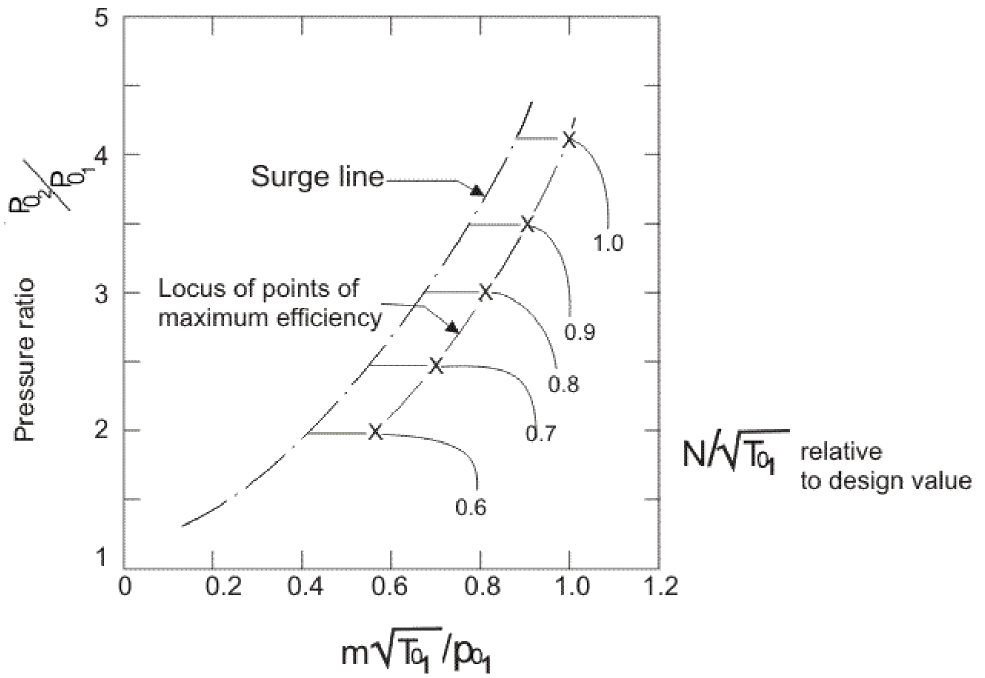


Figure 8.3 Variations of pressure ratio over the complete range of mass flow for different rotational speeds

Axial Flow Compressors

The basic components of an axial flow compressor are a rotor and stator, the former carrying the moving blades and the latter the stationary rows of blades. The stationary blades convert the kinetic energy of the fluid into pressure energy, and also redirect the flow into an angle suitable for entry to the next row of moving blades. Each stage will consist of one rotor row followed by a stator row, but it is usual to provide a row of so called inlet guide vanes. This is an additional stator row upstream of the first stage in the compressor and serves to direct the axially approaching flow correctly into the first row of rotating blades. For a compressor, a row of rotor blades followed by a row of stator blades is called a stage. Two forms of rotor have been taken up, namely drum type and disk type. A disk type rotor illustrated in Figure 9.1 The disk type is used where consideration of low weight is most important. There is a contraction of the flow annulus from the low to the high pressure end of the compressor. This is necessary to maintain the axial velocity at a reasonably constant level throughout the length of the compressor despite the increase in density of air. Figure 9.2 illustrate flow through compressor stages. In an axial compressor, the flow rate tends to be high and pressure rise per stage is low. It also maintains fairly high efficiency.

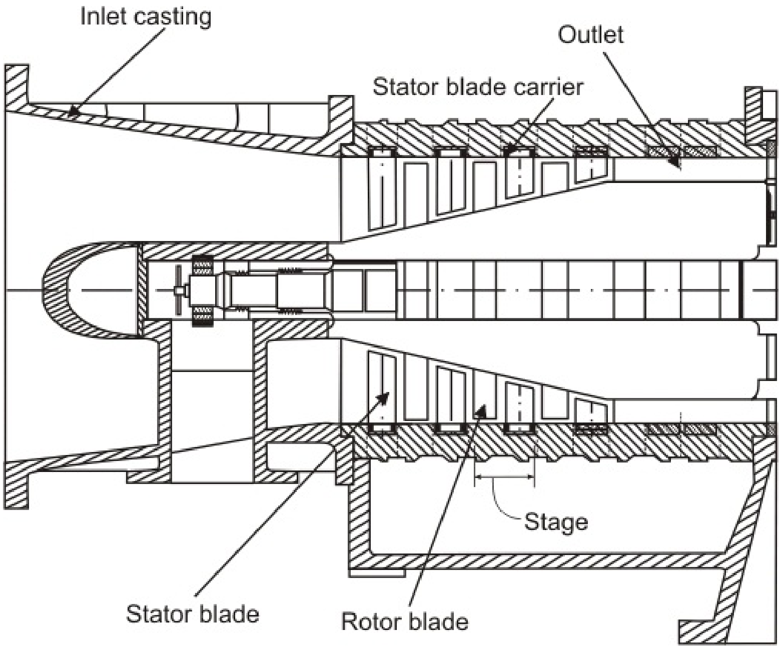


Figure 9.1 Disk type axial flow compressor

The basic principle of acceleration of the working fluid, followed by diffusion to convert acquired kinetic energy into a pressure rise, is applied in the axial compressor. The flow is considered as occurring in a tangential plane at the mean blade height where the blade peripheral velocity is U . This two dimensional approach means that in general the flow velocity will have two components, one axial and one peripheral denoted by subscript w , implying a whirl velocity. It is first assumed that the air approaches the rotor blades with an absolute velocity, V_1 , at and angle α_1 to the axial direction. In combination with the peripheral velocity U of the blades, its relative velocity will be V_{r1} at and angle β_1 as shown in the upper velocity triangle (Figure 9.3). After passing through the diverging passages formed between the rotor blades which do work on the air and increase its absolute velocity, the air will emerge with the relative velocity of V_{r2} at angle β_2 which is less than β_1 . This turning of air towards the axial direction is, as previously mentioned, necessary to provide an increase in the effective flow area and is brought about by the camber of the blades. Since V_{r2} is less than V_{r1} due to diffusion, some pressure rise has been accomplished in the rotor. The velocity V_{r2} in combination with U gives the absolute velocity V_2 at the exit from the rotor at an angle α_2 to the axial direction. The air then passes through the passages formed by the stator blades where it is further diffused to velocity V_3 at an angle α_3 which in most designs equals to α_1 so that it is prepared for entry to next stage. Here again, the turning of the air towards the axial direction is brought about by the camber of the blades.

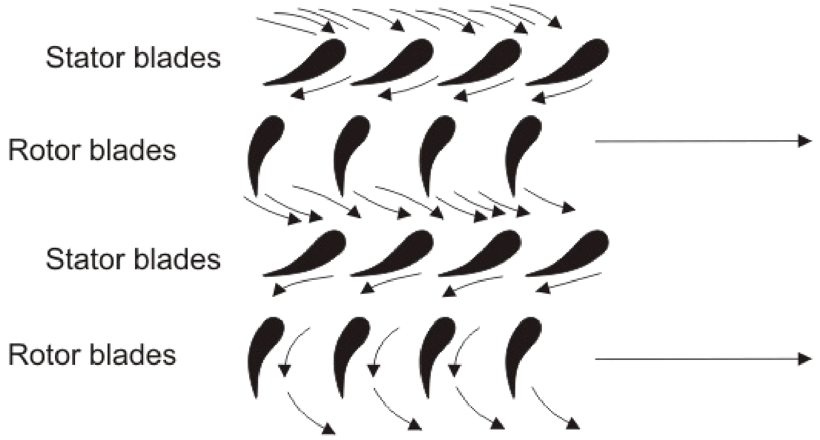


Figure 9.2 Flow through stages

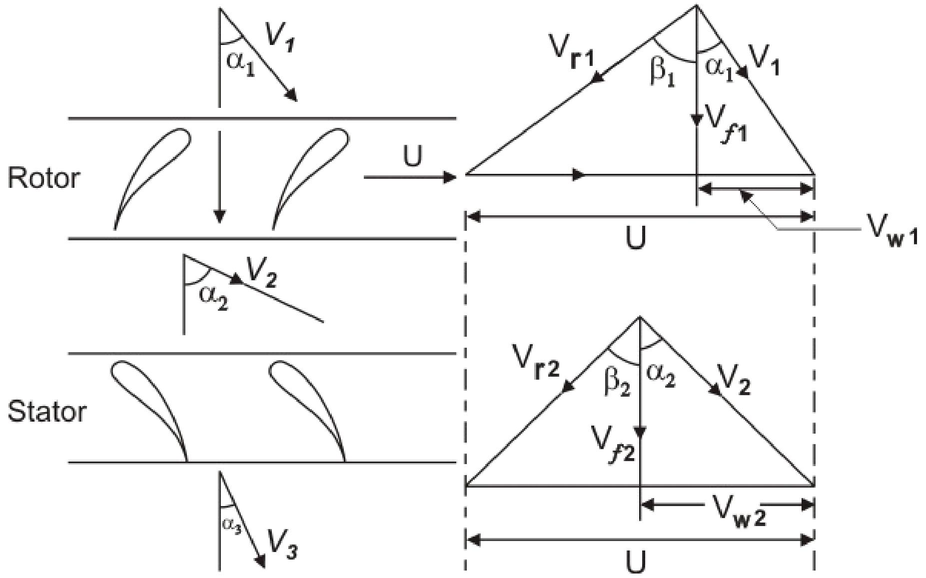


Figure 9.3 Velocity triangles

Two basic equations follow immediately from the geometry of the velocity triangles. These are:

$$\frac{U}{V_f} = \tan\alpha_1 + \tan\beta_1 \qquad (9.1)$$

$$\frac{U}{V_f} = \tan\alpha_2 + \tan\beta_2 \qquad (9.2)$$

In which $V_f = V_{f1} = V_{f2}$ is the axial velocity, assumed constant through the stage. The work done per unit mass or specific work input, w being given by

$$w = U(V_{w2} - V_{w1}) \qquad (9.3)$$

This expression can be put in terms of the axial velocity and air angles to give

$$w = UV_f(\tan\alpha_2 - \tan\alpha_1) \qquad (9.4)$$

or by using Eqs. (9.1) and (9.2)

$$w = UV_f(\tan\beta_1 - \tan\beta_2) \qquad (9.5)$$

This input energy will be absorbed usefully in raising the pressure and velocity of the air. A part of it will be spent in overcoming various frictional losses. Regardless of the losses, the input will reveal itself as a rise in the stagnation temperature of the air ΔT_0 . If the absolute velocity of the air leaving the stage V_3 is made equal to that at the entry. V_1, the stagnation temperature rise ΔT_0 will also be the static temperature rise of the stage, ΔT_s , so that

$$\Delta T_0 = \Delta T_s = \frac{UV_f}{c_p}(\tan\beta_1 - \tan\beta_2) \qquad (9.6)$$

In fact, the stage temperature rise will be less than that given in Eq. (9.6) owing to three dimensional effects in the compressor annulus. Experiments show that it is necessary to multiply the right hand side of Eq. (9.6) by a work-done factor λ which is a number less than unity. This is a measure of the ratio of actual work-absorbing capacity of the stage to its ideal value.

The radial distribution of axial velocity is not constant across the annulus but becomes increasingly peaky (Figure. 9.4) as the flow proceeds, settling down to a fixed profile at about the fourth stage. Equation (9.5) can be written with the help of Eq. (9.1) as

$$\begin{aligned} w &= U[(U - V_f\tan\alpha_1) - V_f\tan\beta_2] \qquad (9.7)\\ &= U(U - V_f(\tan\alpha_1 + \tan\beta_2)) \end{aligned}$$

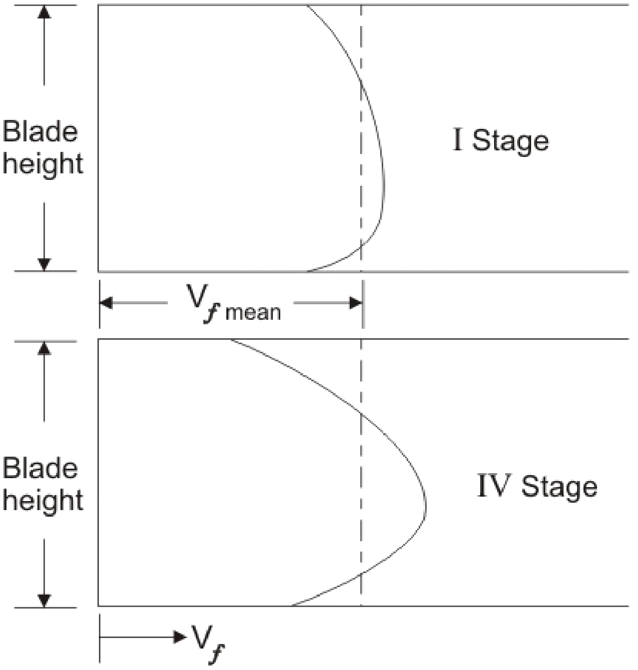


Figure 9.4 Axial velocity distibutions

Since the outlet angles of the stator and the rotor blades fix the value of α_1 and β_2 and hence the value of $(\tan\alpha_1 + \tan\beta_2)$. Any increase in V_f will result in a decrease in ϖ and vice-versa. If the compressor is designed for constant radial distribution of V_f as shown by the dotted line in Figure (9.4), the effect of an increase in V_f in the central region of the annulus will be to reduce the work capacity of blading in that area. However this reduction is somewhat compensated by an increase in ϖ in the regions of the root and tip of the blading because of the reduction of V_f *at* these parts of the annulus. The net result is a loss in total work capacity because of the adverse effects of blade tip clearance and boundary layers on the annulus walls. This effect becomes more pronounced as the number of stages is increased and the way in which the mean value varies with the number of stages. The variation of λ with the number of stages is shown in Figure. 9.5. Care should be taken to avoid confusion of the work done factor with the idea of an efficiency. If ϖ is the expression for the specific work input (Equation. 9.3), then $\lambda\varpi$ is the actual amount of work which can be supplied to the stage. The application of an isentropic efficiency to the resulting temperature rise will yield the equivalent isentropic temperature rise from which the stage pressure ratio may be calculated. Thus, the actual stage temperature rise is given by

$$\Delta T_0 = \frac{\lambda UV_f}{c_p}(\tan\beta_1 - \tan\beta_2) \qquad (9.8)$$

and the pressure ratio R_s by

$$R_s = \left[1 + \frac{\eta_s \Delta T_0}{T_{0_1}}\right]^{\frac{\gamma}{\gamma-1}} \qquad (9.9)$$

where, T_{0_1} is the inlet stagnation temperature and η_s is the stage isentropic efficiency.

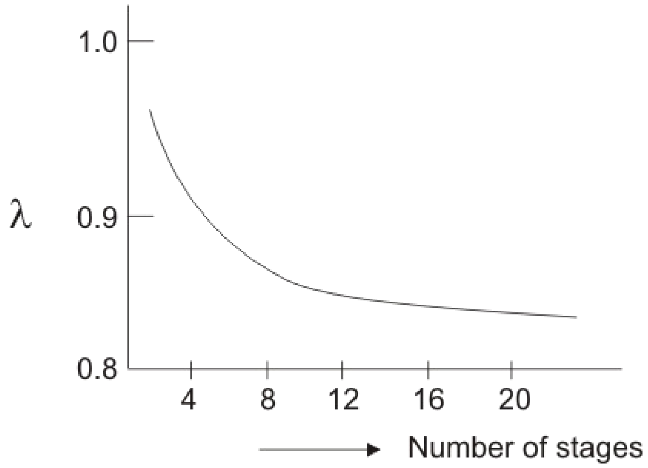


Figure 9.5 Variation of work-done factor with number of stages

Example: At the mean diameter, U = 20 m/s, $V_f = 180 m/s$, β_1 = 43.9 ° and β_2 = 13.5 °. The factor λ = 0.86 and η_s = 0.85 and inlet temperature T_{0_1} is 288 K. Calculate the pressure ratio.

$$\Delta T_0 = \frac{0.86 \times 200 \times 180}{1.005 \times 10^3}(\tan 43.9° - \tan 13.5°)$$

= 22.24 K

and $$R_s = \left[1 + \frac{0.85 \times 22.24}{288}\right]^{3.5} = 1.25$$

[c_p of air has been taken as 1005 J/kg K]

Degree of Reaction

A certain amount of distributionof pressure (a rise in static pressure) takes place as the air passes through the rotor as well as the stator; the rise in pressure through the stage is in general, attributed to both the blade rows. The term degree of reaction is a measure of the extent to which the rotor itself contributes to the increase in the static head of fluid. It is defined as the ratio of the static enthalpy rise in the rotor to that in the whole stage. Variation of c_p over the relevant temperature range will be negligibly small and hence this ratio of enthalpy rise will be equal to the corresponding temperature rise.

It is useful to obtain a formula for the degree of reaction in terms of the various velocities and air angles associated with the stage. This will be done for the most common case in which it is assumed that the air leaves the stage with the same velocity (absolute) with which it enters ($V_1 = V_3$).

This leads to $\Delta T_s = \Delta T_0$. If ΔT_A and ΔT_B are the static temperature rises in the rotor and the stator respectively,

then from Eqs (9.4),(9.5),(9.6),

$$\begin{aligned} w &= c_p(\Delta T_A + \Delta T_B) = c_p \Delta T_s \\ &= UV_f\,(\tan\beta_1 - \tan\beta_2) \\ &= UV_f\,(\tan\alpha_2 - \tan\alpha_1) \end{aligned} \tag{10.1}$$

Since all the work input to the stage is transferred to air by means of the rotor, the steady flow energy equation yields,

$$w = c_p \Delta T_A + \frac{1}{2}(V_2^2 - V_1^2)$$

With the help of Eq. (10.1), it becomes

$$c_p \Delta T_A = UV_f\,(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}(V_2^2 - V_1^2)$$

But $V_2 = V_f \sec\alpha_2$ and $V_1 = V_f \sec\alpha_1$, and hence

$$\begin{aligned} c_p \Delta T_A &= UV_f\,(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2\,(\sec^2\alpha_2 - \sec^2\alpha_1) \\ &= UV_f\,(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2\,(\tan^2\alpha_2 - \tan^2\alpha_1) \end{aligned} \tag{10.2}$$

The degree of reaction

$$\Lambda = \frac{\Delta T_A}{\Delta T_A + \Delta T_B} \tag{10.3}$$

With the help of Eq. (10.2), it becomes

$$\Lambda = \frac{UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\tan^2\alpha_2 - \tan^2\alpha_1)}{UV_f(\tan\alpha_2 - \tan\alpha_1)}$$

and

$$\Lambda = 1 - \frac{V_f}{2U}(\tan\alpha_2 + \tan\alpha_1)$$

By adding up Eq. (9.1) and Eq. (9.2) we get

$$\frac{2U}{V_f} = \tan\alpha_1 + \tan\beta_1 + \tan\alpha_2 + \tan\beta_2$$

Replacing α_1 and α_2 in the expression for Λ with β_1 and β_2 ,

$$\Lambda = \frac{V_f}{2U}(\tan\beta_1 + \tan\beta_2) \tag{10.4}$$

As the case of 50% reaction blading is important in design, it is of interest to see the result for $\Lambda = 0.5$,

$$\tan\beta_1 + \tan\beta_2 = \frac{U}{V_f}$$

and it follows from Eqs. (9.1) and (9.2) that

$$\tan\alpha_1 = \tan\beta_2,\ \text{i.e. } \alpha_1 = \beta_2 \tag{10.5a}$$

$$\tan\beta_1 = \tan\alpha_2,\ \text{i.e. } \beta_1 = \alpha_2 \tag{10.5b}$$

Furthermore since V_f is constant through the stage.

$$V_f = V_1\cos\alpha_1 = V_3\cos\alpha_3$$

And since we have initially assumed that $V_3 = V_1$, it follows that $\alpha_1 = \alpha_3$. Because of this equality of angles, namely, $\alpha_1 = \beta_2 = \alpha_3$ and $\beta_1 = \alpha_2$, blading designed on this basis is sometimes referred to as *symmetrical blading* .The 50% reaction stage is called a repeating stage.

It is to be remembered that in deriving Eq. (10.4) for Λ , we have implicitly assumed a work done factor λ of unity in making use of Eq. (10.2). A stage designed with symmetrical blading is referred to as 50% reaction stage, although Λ will differ slightly for λ.

Exercises

PROBLEMS AND SOLUTIONS FOR CYCLE, CENTRIFUGAL, AXIAL COMPRESSORS

[For all the Exercises, assume R=287J/kg K and γ = 1.4 for air]

1. Determine the pressure ratio developed and the specific work input to drive a centrifugal air compressor having an impeller diameter of 0.5 m and running at 7000 rpm. Assume zero whirl at the entry and T_{1t} = 288 K.

(1.47,33.58 kJ/kg)

2 A centrifugal compressor develops a pressure ratio of 4:1. The inlet eye of the compressor impeller is 0.3 m in diameter. The axial velocity at inlet is 120 m/s and the mass flow rate is 10 kg/s. The velocity in the delivery duct is 110 m/s. The tip speed of the impeller is 450 m/s and runs at 16,000 rpm with a total head isentropic efficiency of 80%. The inlet stagnation temperature and pressure are $101 kN/m^2$ and 300 K. Calculate (a) the static temperatures and pressures at inlet and outlet of the compressor, (b) the static pressure ratio, (c) the power required to drive the compressor.

Ans. (T_1 = 292.8 K, T_2 = 476.45 K, p = 93 kN/m 2 , p_2 = 386.9 kN/m 2 , p_2/p_1 = 4.16, p = 1.83 MW)

3. The following results were obtained from a test on a small single-sided centrifugal compressor

Compressor delivery stagnation pressure 2.97 bar
Compressor delivery stagnation temperature 429 K
Static pressure at impeller tip 1.92 bar
Mass flow 0.60 kg/s
Rotational speed 766 rev/s
Ambient conditions 0.99 bar 288 K
Determine the isentropic efficiency of the compressor.
The diameter of the impeller is 0.165 m, the axial depth of the vaneless diffuser is 0.01m and the number of impeller vanes is 17. Making use of the Stanitz equation for slip factor, calculate the stagnation pressure at the impeller tip.

Ans. (0.76, 3.13 bar)

4. A single sided centrifugal compressor is to deliver 14 kg/s of air when operating at a pressure ratio of 4:1 and a speed of 200 rev/s. The inlet stagnation conditions are 288 K and 1.0 bar. The slip factor and power input factor may be taken as 0.9 and 1.04 respectively. The overall isentropic efficiency is 0.80. Determine the overall diameter of the impeller.

Ans. (0.69m)

PROBLEMS ON AXIAL COMPRESSORS

5. Each stage of an axial flow compressor is of 50% degree of reaction and has the same mean blade speed and the same value of outlet relative velocity angle $\beta_2 = 30^o$. The mean flow coefficiency V_f / U is constant for all stages at 0.5. At entry to the first stage, the stagnation temperature is 290 K, the stagnation pressure is 101 kPa. The static pressure is 87 kPa and the flow area is $0.38\ m^2$. Determine the axial velocity, the mass flow rate and the shaft power needed to derive the compressor when there are 6 stages and the mechanical efficiency is 0.98.

Ans. (135.51 m/s, 56.20 kg/s, 10.68 MW)

6. An axial flow compressor stage has blade root, mean and tip velocities of 150, 200 and 250 m/s The stage is to be designed for a stagnation temperature rise of 20 K and an axial velocity of 150 m/s, both constant from root to tip. The work done factor is 0.93. Assuming degree of reaction 0.5 at mean radius, determine the stage air angles at root mean and tip for a free vortex design where the whirl component of velocity varies inversely with the radius

$Ans.\ (\alpha_1 = 17.04^o (= \beta_2),\ \beta_1 = 45.75^o (= \alpha_2)\ at\ mean\ radius\ \alpha_1 = 13.77^o,\ \beta_1 = 54.88^o,\ \beta_2 = 40.36^o,$
$at\ tip;\ \alpha_1 = 22.10^o,\ \beta_1 = 30.71^o,\ \beta_2 = -19.95^o,\ \alpha_2 = 53.74^o\ at\ root)$

7. An axial compressor has the following data:

Temperature and pressure at entry 300 K, 1.0 bar
Degree of reaction 50%
Mean blade ring diameter 0.4 m
Rotational speed 15,000 rpm
Blade height at entry 0.08 m
Air angles at rotor and stator exit 25°
Axial velocity 150 m/s
Work done factor 0.90
Isentropic stage efficiency 85%
Mechanical efficiency 97%

Determine (a) air angles at the rotor and stator entry (b) the mass flow rate of air (c) the power required to derive the compressor, (d) the pressure ratio developed by the stage (e) Mach number (based on relative velocities) at the rotor entry.

Ans. [(a) 25° , 58.44° (b) 17.51 kg/s, (c) 0.89 MW, (d) 1.58, (e) 0.83]

8 An axial flow compressor stage has a mean diameter of 0.6 m and runs at 15,000 rpm. If the actual temperature rise and pressure ratio developed are 30°C and 1.36 respectively, determine (a) the power required to derive the compressor while delivering 57 kg/s of air. Assume mechanical efficiency of 86% and an initial temperature of 35°C (b) the isentropic efficiency of the stage and (c) the degree of reaction if the temperature at the rotor exit is 55°C.

Ans. [(a) 2 MW, (b) 94.2%, (c) 66.6%]

MODULE 3

Cascade Theory, Axial Flow Turbine and Propu;sion System

Next

Elementary Cascade Theory

The previous module dealt with the axial flow compressors, where all the analyses were based on the flow conditions at inlet to and exit from the impeller following kinematics of flow expressed in terms of velocity triangles. However, nothing has been mentioned about layout and design of blades, which are aerofoil sections.

In the development of the highly efficient modern axial flow compressor or turbine, the study of the two-dimensional flow through a cascade of aerofoils has played an important part. An array of blades representing the blade ring of an actual turbo machinery is called the cascade. Figure 11.1a shows a compressor blade cascade tunnel. As the air stream is passed through the cascade, the direction of air is turned. Pressure and velocity measurements are made at up and downstream of cascade as shown. The cascade is mounted on a turn-table so that its angular direction relative to the direction of inflow can be changed, which enables tests to be made for a range of incidence angle. As the flow passes through the cascade, it is deflected and there will be a circulation Γ and thus the lift generated will be $\rho V_m \Gamma$ (Fig 11.1b & 11.1c). V_m is the mean velocity that makes an angle α_m with the axial direction.

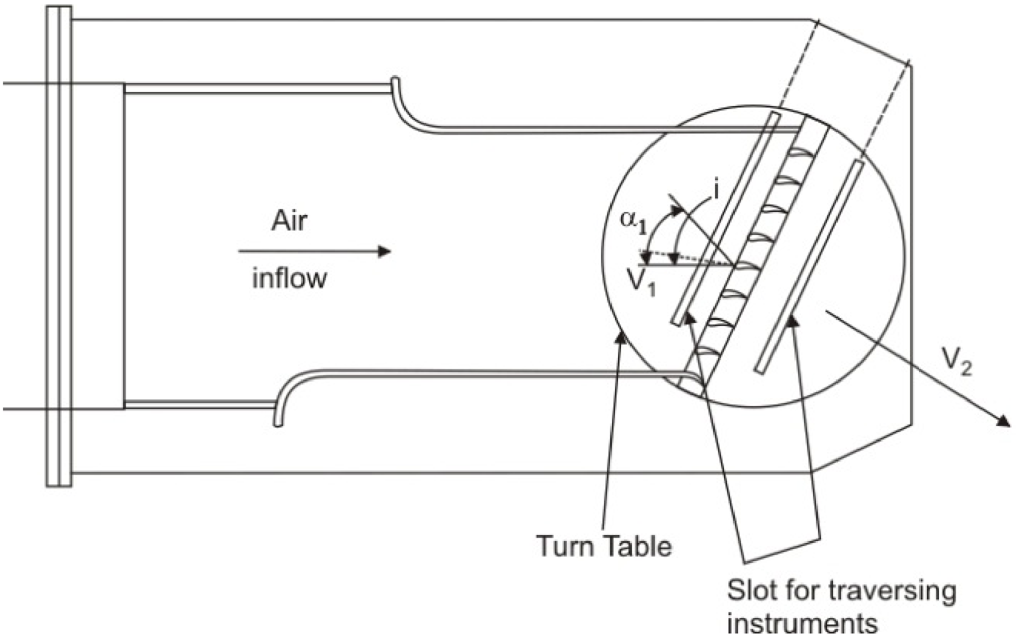


Figure 11.1a A Cascade Tunnel

Compressor cascade :

For a compressor cascade, the static pressure will rise across the cascade, i.e. $p_2 > p_1$

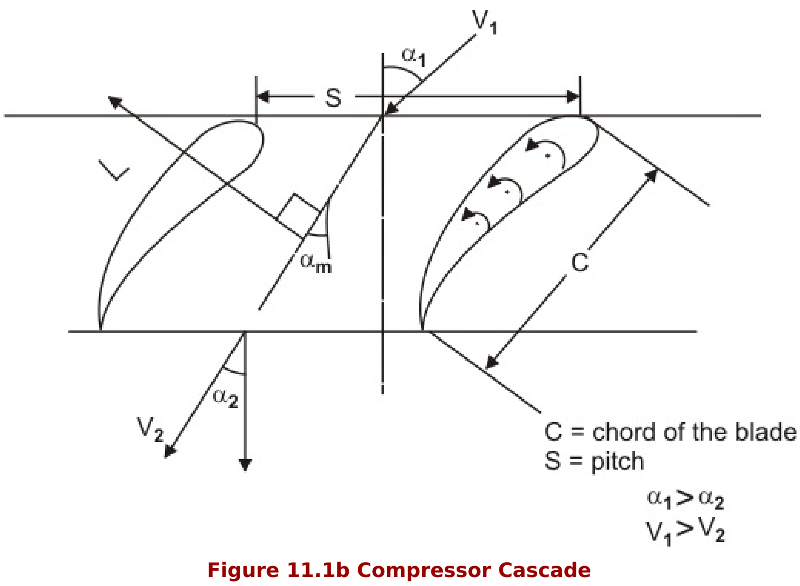


Figure 11.1b Compressor Cascade

C =chord of the blade

S = pitch

$$\alpha_1 > \alpha_2$$

$$V_1 > V_2$$

$$\tan \alpha_m = \frac{1}{2}(\tan \alpha_1 + \tan \alpha_2)$$

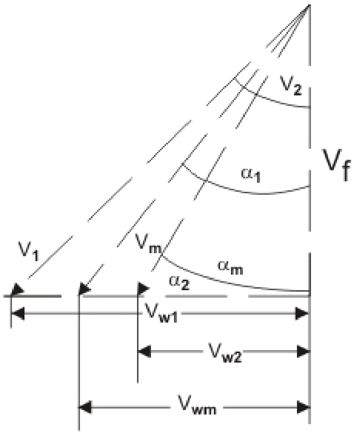


Figure 11.1c Velocity triangle

Circulation:

$$\Gamma = S(V_{w1} - V_{w2})$$

Lift = $\rho V_m \Gamma = \rho V_m S(V_{w1} - V_{w2})$

Lift coefficient, $C_L = \dfrac{L}{\frac{1}{2}\rho V_m^2 C} = \dfrac{\rho V_m S(V_{w1} - V_{w2})}{\frac{1}{2}\rho V_m^2 C}$

$$= \frac{2S}{C} * \frac{1}{V_m}(V_{w1} - V_{w2})$$

from velocity triangles,

$$V_{w1} = V_f \tan \alpha_1, \quad V_{w2} = V_f \tan \alpha_2$$

$$C_L = 2\frac{S}{C}\left(\frac{V_f}{V_m}\right)(\tan \alpha_1 - \tan \alpha_2)$$

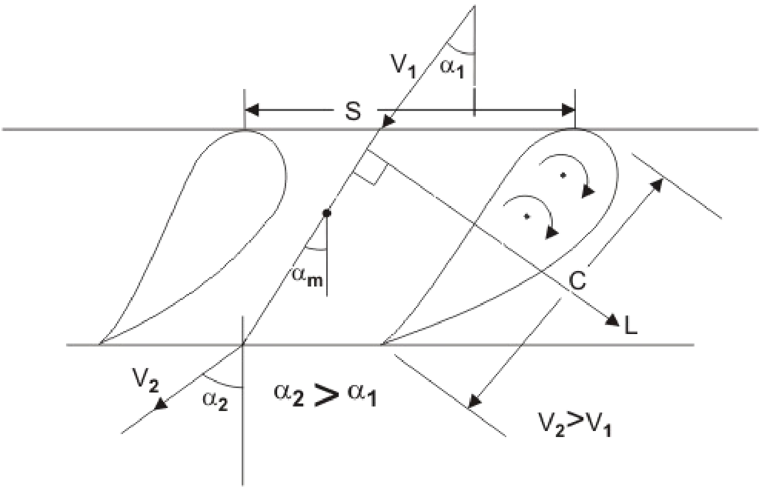
$= 2\dfrac{S}{C}(\tan \alpha_1 - \tan \alpha_2)\cos \alpha_m$, where $\tan \alpha_m = \dfrac{\tan \alpha_1 + \tan \alpha_2}{2}$

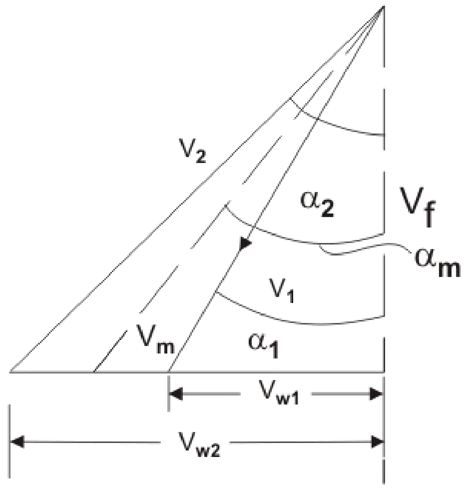
S, C -depend on the design of the cascade

α_1, α_2 - flow angles at the inlet and outlet

Lift is perpendicular to α_m line

Turbine Cascade: Static pressure will drop across the turbine cascade, i.e $p_2 < p_1$





$L = \rho V_m \Gamma$

Lift coefficient, $C_L = \dfrac{L}{\frac{1}{2}\rho V_m^2 C} = 2\dfrac{S}{C}(\tan\alpha_2 - \tan\alpha_1)\cos\alpha_m$

Above discussion is based on Kutta-Joukowski theorem

Assumption - Inviscid flow

In reality, we face viscous flow together with formation of wakes. Thus, the viscous flow is the cause of drag which in turn affect the lift .

Effect of viscous flow

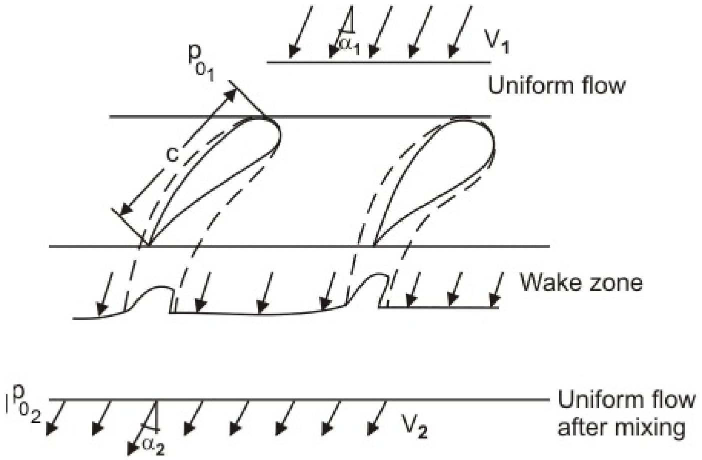
As the fluid passes through the cascade, there will be a decrease in total pressure between the inlet to the cascade and at a section well downstream of the cascade due to frictional effect on aerofoils and also losses due to mixing of blade wades (i.e. the effect of viscous flow). Since at up and downstream, the flow is uniform and kinematics of flow remains unchanged, the dynamic pressure at up and downstream remains the same. Thus loss in total pressure is the same as that of static pressure.

The loss in total pressure $\bar{\omega} = p_{0_1} - p_{0_2}$ consists of two components:

(1) Frictional loss due to the formation of boundary layer on blades.

(2) Mixing of the blade wades.

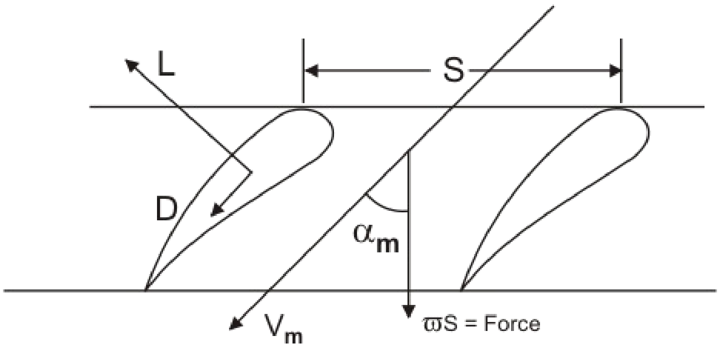
.



Compressor Cascade (Viscous Case)

In compressor cascade, due to losses in total pressure $(\tilde{\varpi})$, there will be an axial force $S\tilde{\varpi}$ as shown in figure below. Thus the drag, which is perpendicular to the lift, is defined as

$$D = \tilde{w}S\cos\alpha_m$$



The lift will be reduced due to the effect of drag which can be expressed as:

Effective lift = $\bar{L} = L - \bar{\varpi}S\sin\alpha_m = \rho V_m\Gamma - \bar{\varpi}S\sin\alpha_m$

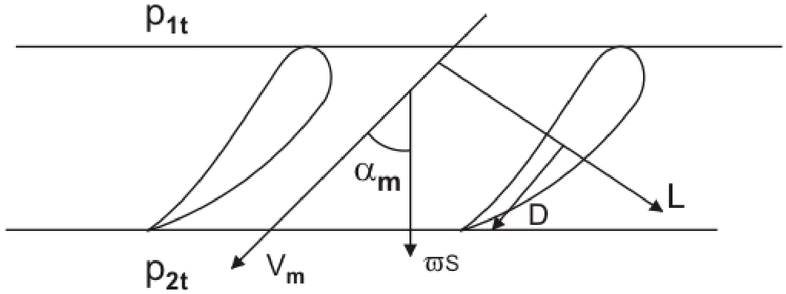
The lift has decreased due to viscosity, $\bar{L} = \rho V_m\Gamma - D\tan\alpha_m$

Actual lift coefficient $C_L = 2\frac{S}{C}(\tan\alpha_1 - \tan\alpha_2)\cos\alpha_m - C_D\tan\alpha_m$

where, C_D =drag coefficient, $C_D = \dfrac{D}{\frac{1}{2}\rho V_m^2 C}$

In the case of turbine, drag will contribute to work (and is considered as useful).

Turbine Cascade (Viscous case)



Drag = $D = \varpi S\cos\alpha_m$

Effective lift = $L + \varpi S \sin\alpha_m = \rho V_m \Gamma + \varpi S \sin\alpha_m$

Actual lift coefficient, $C_L = 2\dfrac{S}{C}(\tan\alpha_2 - \tan\alpha_1)\cos\alpha_m + C_D \tan\alpha_m$

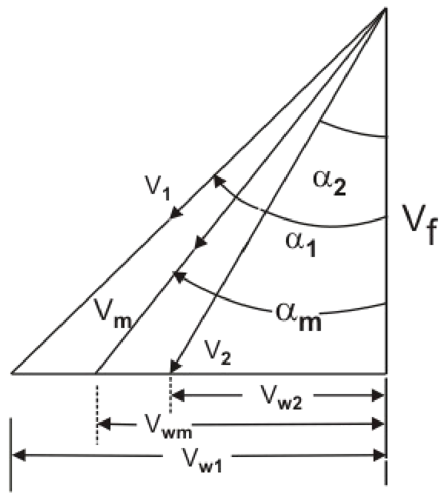
The drag increases the lift. Thus, thedrag is an useful component for work.

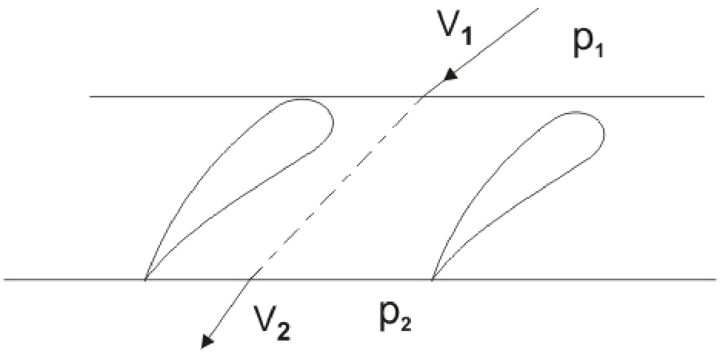
Blade efficiency (or diffusion efficiency)

For a compressor cascade, the blade efficiency is defined as:

$$\eta_b = \frac{\text{Actual rise in static pressure}}{\text{Ideal static pressure rise}}$$

Due to viscous effect, static pressure rise is reduced





$$\eta_b = \frac{(p_2 - p_1)_{\text{ideal}} - \text{loss}}{(p_2 - p_1)_{\text{ideal}}}$$

$$\eta_b = \frac{\frac{\rho}{2}(V_1^2 - V_2^2) - \varpi}{\frac{\rho}{2}(V_1^2 - V_2^2)} = 1 - \frac{\varpi}{\frac{\rho}{2}(V_1^2 - V_2^2)}$$

from velocity triangle: $V_1^2 = V_{w1}^2 + V_f^2,\ V_2^2 = V_{w2}^2 + V_f^2$

$$V_1^2 - V_2^2 = (V_{w1} + V_{w2})(V_{w1} - V_{w2})$$

$$\eta_b = 1 - \frac{\bar{\varpi}}{\frac{\rho}{2}(V_{w1}+V_{w2})(V_{w1}-V_{w2})}$$

Also we get

$$\frac{V_{w1}+V_{w2}}{2} = V_{wm}$$

$$= V_m \sin\alpha_m$$

$$\eta_b = 1 - \frac{\bar{\varpi}}{\rho V_m \sin\alpha_m} \frac{(\cos\alpha_m)S}{(V_{w1}-V_{w2})S} \cdot \frac{1}{\cos\alpha_m}$$

$$= 1 - \frac{D}{\rho V_m \Gamma \sin\alpha_m \cos\alpha_m} = 1 - \frac{D}{L\sin\alpha_m \cos\alpha_m}$$

$$= 1 - \frac{2D}{L\sin 2\alpha_m}$$

[Approximation: $L \simeq \rho\Gamma V_m$ i.e.in the expression for lift, the effect of drag is ignored]

$$(\eta_b)_{\text{comp cascade}} = 1 - \frac{2C_D}{C_L \sin 2\alpha_m}$$

η -maximum, if $\dfrac{d\eta_b}{d\alpha_m} = 0 \Rightarrow \cos 2\alpha_m = 0$

$$\alpha_m = 45°$$

The value of α_m for which efficiency is maximum, $\alpha_m = 45°$

The blade efficiency for a turbine cascade is defined as:

$$\eta_b = \frac{\text{Ideal static pressure drop } (\Delta p)_s \text{ to obtain a certain change in kinetic energy}}{\text{Actual static pressure drop to produce the same change in kinetic energy}}$$

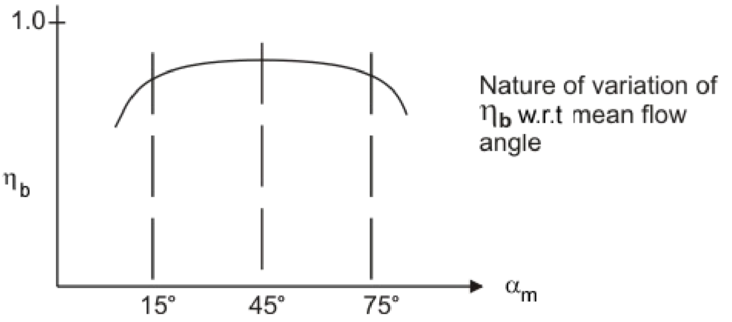
$$= \frac{\frac{\rho}{2}(V_2^2 - V_1^2)}{\frac{\rho}{2}(V_2^2 - V_1^2) + \bar{\varpi}} = \frac{1}{1 + \frac{\bar{\varpi}}{\frac{\rho}{2}(V_{w2} + V_{w1})(V_{w2} - V_{w1})}}$$

$$(\eta_b)_{\text{turbine}} = \frac{1}{1 + \frac{2C_D}{C_L \sin 2\alpha_m}}$$

For very small ratio of C_D / C_L

$$(\eta_b)_{\text{turbine}} = \left(1 + 2\frac{C_D}{C_L} * \frac{1}{\sin 2\alpha_m}\right)^{-1}$$

$(\eta_b)_{\text{turbine}} = 1 - 2\dfrac{C_D}{C_L \sin 2\alpha_m}$ which is same as the compressor cascade



Nature of variation of η_b wrt mean flow angle

Note: η_b does not vary much in the range $15° \le \alpha_m \le 75°$ provides flexibility in design.

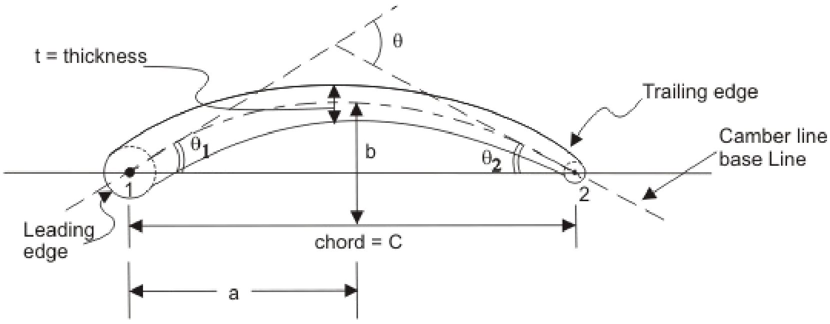
In the above derivation for blade efficiency of both the compressor and turbine cascade, the lift is assumed as $\rho \Gamma V_m$, neglecting the effect of drag. With the corrected expression of lift, actual blade efficiencies are as follows:

$$(\eta_b)_{\text{comp cascade}} = \frac{1 - \frac{C_D}{C_L}\cot \alpha_m}{1 + \frac{C_D}{C_L}\tan \alpha_m}$$

$$(\eta_b)_{\text{turb cascade}} = \frac{1 - \frac{C_D}{C_L}\tan \alpha_m}{1 + \frac{C_D}{C_L}\cot \alpha_m}$$

Cascade Nomenclature

An aerofoil is build up around a basic camber line, which is usually a curricular or a parabolic arc (figure below). An camber line is thus the skeleton of the aerofoil. A thickness *t* is distributed over the camber line with the leading and trailing edge circles that finally form an aerofoil.

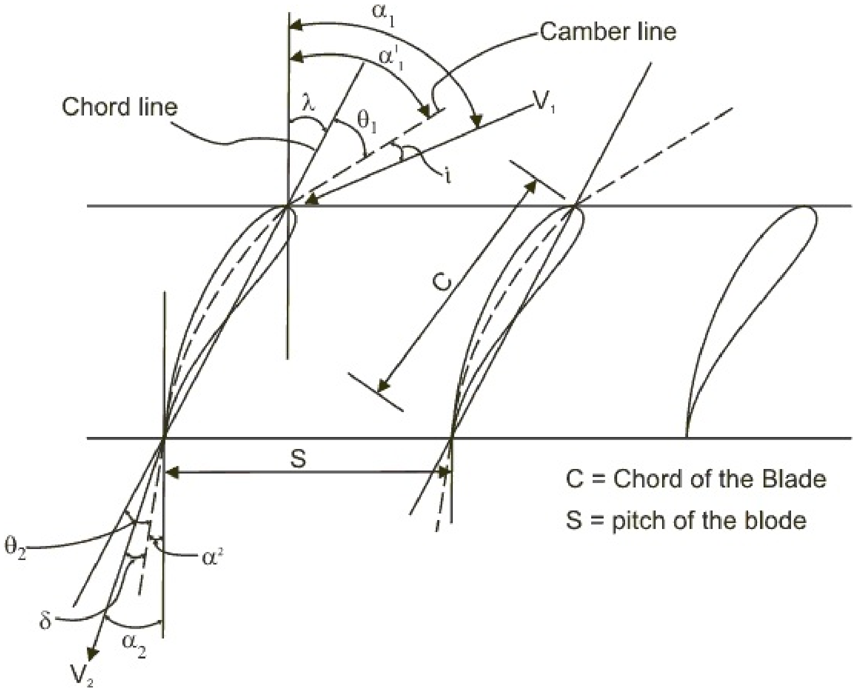


In the above figure, the dotted line indicates the camber line and 'a' is the distance from the leading edge for maximum camber and 'b' is the maximum displacement from the chord line. A cascade geometry is defined completely by the aerofoil specification, pitch-chord ratio (pitch is the spacing between two consecutive blade) and the chosen setting i.e. stagger angle λ (shown below).

θ is called the aerofoil camber angle i.e. $\theta = \theta_1 + \theta_2$. For a circular arc, $\theta_1 = \theta_2 = \theta/2$ and a/c = 0.5. For a parabolic arc a/c < 0.5.

Compressor Cascade

The different geometric angles, blade setting and their relationship with the flow angles for a compressor cascade are defined below.



λ = stagger angle (positive for a compressor cascade)

α_1' = blade inlet angle = $\lambda + \theta_1$

α_2' = blade outlet angle = $\lambda - \theta_2$

The angle of incidence 'i' is the angle made by the inlet flow V_1 with the camber line. Under a perfect situation, the flow will leave along the camber line at the trailing edge of the blade. But it does not really happen so and there is a deviation which is denoted by 'δ'. Thus, the air inlet angle,

$\alpha_1 = \lambda + \theta_1 + i$

and air outlet angle, $\alpha_2 = \lambda - \theta_2 - \delta$

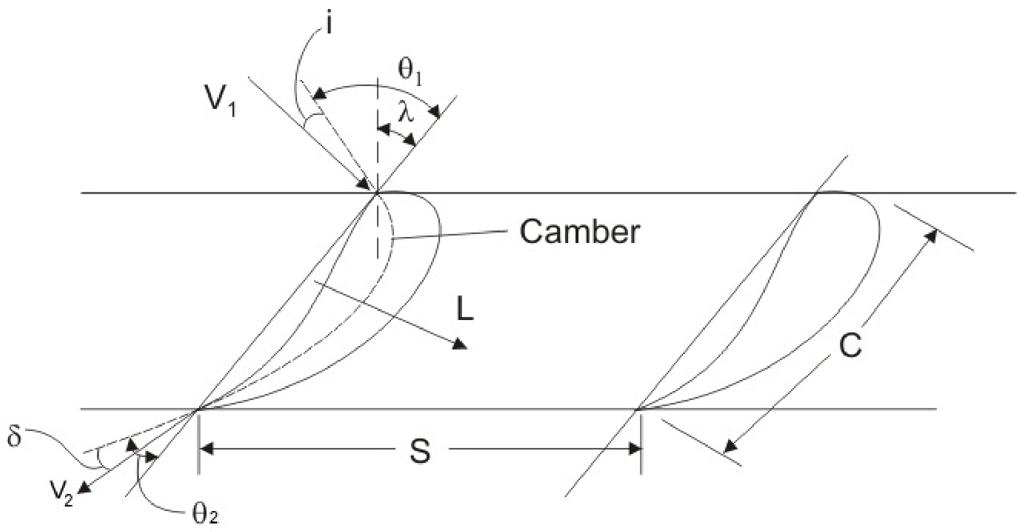
Hence, ε = deflection of flow = $\alpha_1 - \alpha_2$

$= (\theta_1 + \theta_2) + i - \delta$

$= \theta + i - \delta$

Turbine Cascade

The different geometric angles and the blade setting of a turbine cascade are shown in the figure below.



λ stagger is the stagger which is negative for a turbine cascade.

To be noted: The difference between the orientation of Compressor cascade and that of the Turbine cascade.

GAS TURBINE

Axial Flow Turbine

A gas turbine unit for power generation or a turbojet engine for production of thrust primarily consists of a compressor, combustion chamber and a turbine. The air as it passes through the compressor, experiences an increase in pressure. There after the air is fed to the combustion chamber leading to an increase in temperature. This high pressure and temperature gas is then passed through the turbine, where it is expanded and the required power is obtained.

Turbines, like compressors, can be classified into radial, axial and mixed flow machines. In the axial machine the fluid moves essentially in the axial direction through the rotor. In the radial type, the fluid motion is mostly radial. The mixed-flow machine is characterized by a combination of axial and radial motion of the fluid relative to the rotor. The choice of turbine type depends on the application, though it is not always clear that any one type is superior.

Comparing axial and radial turbines of the same overall diameter, we may say that the axial machine, just as in the case of compressors, is capable of handling considerably greater mass flow. On the other hand, for small mass flows the radial machine can be made more efficient than the axial one. The radial turbine is capable of a higher pressure ratio per stage than the axial one. However, multistaging is very much easier to arrange with the axial turbine, so that large overall pressure ratios are not difficult to obtain with axial turbines. In this chapter, we will focus on the axial flow turbine.

Generally the efficiency of a well-designed turbine is higher than the efficiency of a compressor. Moreover, the design process is somewhat simpler. The principal reason for this fact is that the fluid undergoes a pressure drop in the turbine and a pressure rise in the compressor.The pressure drop in the turbine is sufficient to keep the boundary layer generally well behaved, and the boundary layer separation which often occurs in compressors because of an adverse pressure gradient, can be avoided in turbines. Offsetting this advantage is the much more critical stress problem, since turbine rotors must operate in very high temperature gas. Actual blade shape is often more dependent on stress and cooling considerations than on aerodynamic considerations, beyond the satisfaction of the velocity-triangle requirements.

Because of the generally falling pressure in turbine flow passages, much more turning in a giving blade row is possible without danger of flow separation than in an axial compressor blade row. This means much more work, and considerably higher pressure ratio, per stage.

In recent years advances have been made in turbine blade cooling and in the metallurgy of turbine blade materials. This means that turbines are able to operate successfully at increasingly high inlet gas temperatures and that substantial improvements are being made in turbine engine thrust, weight, and fuel consumption.

GAS TURBINE

Two-dimensional theory of axial flow turbine.

An axial turbine stage consists of a row of stationary blades, called nozzles or stators, followed by the rotor, as Figure 13.1 illustrates. Because of the large pressure drop per stage, the nozzle and rotor blades may be of increasing length, as shown, to accommodate the rapidly expanding gases, while holding the axial velocity to something like a uniform value through the stage.

It should be noted that the hub-tip ratio for a high pressure gas turbine in quite high, that is, it is having blades of short lengths. Thus, the radial variation in velocity and pressure may be neglected and the performance of a turbine stage is calculated from the performance of the blading at the mean radial section, which is a two-dimensional "pitch-line design analysis ". A low-pressure turbine will typically have a much lower hub-tip ratio and a larger blade twist. A two dimensional design is not valid in this case.

In two dimensional approach the flow velocity will have two components, one axial and the other peripheral, denoted by subscripts 'f' and 'ω' respectively. The absolute velocity is denoted by V and the relative velocity with respect to the impeller by V_r . The flow conditions '1' indicates inlet to the nozzle or stator vane, '2' exit from the nozzle or inlet to the rotor and '3' exit form the rotor. Absolute angle is represented by α and relative angle by β as before.

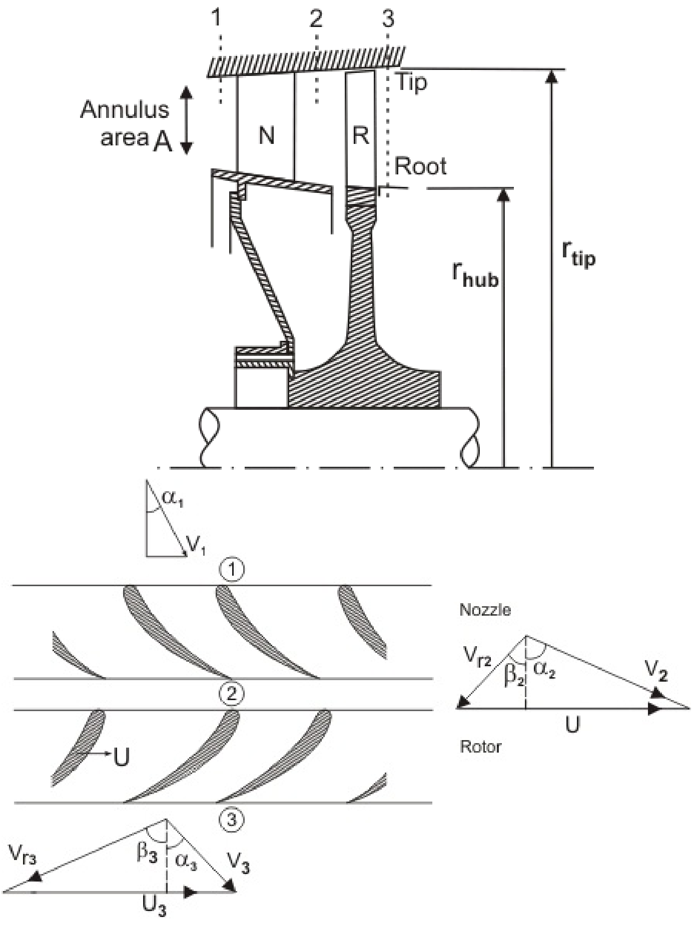


Figure 13.1 Axial Turbine Stage

A section through the mean radius would appear as in Figure.13.1. One can see that the nozzles accelerate the flow imparting an increased tangential velocity component. The velocity diagram of the turbine differs from that of the compressor in that the change in tangential velocity in the rotor, ΔV_w , is in the direction opposite to the blade speed U. The reaction to this change in the tangential momentum of the fluid is a torque on the rotor in the direction of motion. Hence the fluid does work on the rotor.

GAS TURBINE

Again applying the angular momentum relation-ship, we may show that the power output as,

$$P = \dot{m}(U_2 V_{w_2} - U_3 V_{w_3}) \qquad (13.1)$$

In an axial turbine,

$$U_2 \simeq U_3 = U \ (say)$$

The work output per unit mass flow rate is

$$W_T = U(V_{w_2} - V_{w_3})$$

Again, $$W_T = C_p(T_{o_1} - T_{o_3})$$

Defining $$\Delta T_o = T_{o_1} - T_{o_3} = T_{o_2} - T_{o_3}$$

We find that the stage work ratio is

$$\frac{\Delta T_o}{T_{o_1}} = \frac{W_T}{C_p T_{o_1}} = \frac{U(V_{w_2} - V_{w_3})}{C_p T_{o_1}} \qquad (13.2)$$

Figure 13.2 illustrates a combined (inlet to and exit from the rotor) velocity diagram of a turbine stage.

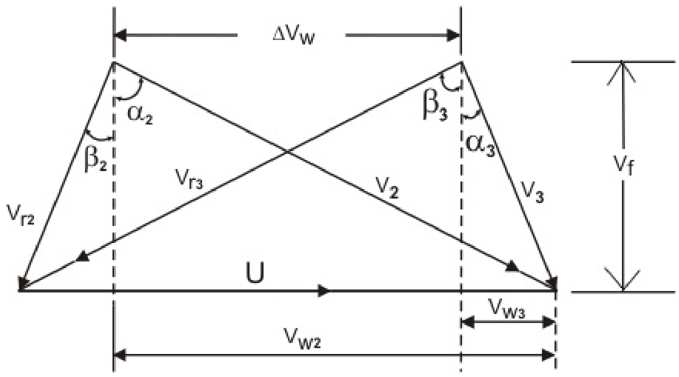


Figure 13.2 Combined velocity diagram

The velocity diagram gives the following relation:

$$\frac{U}{V_f} = \tan\alpha_2 - \tan\beta_2$$

$$= \tan\alpha_3 - \tan\beta_3$$

Thus, $$W_T = U(V_{w_2} - V_{w_3})$$

$$= UV_f[\tan\alpha_2 - \tan\alpha_3]$$

i.e, $$W_T = UV_f[\tan\alpha_2 - \tan\alpha_3] \qquad (13.3)$$

The Eq (13.3) gives the expression for W_T in terms of gas angles associated with the rotor blade.

Note that the "work-done factor" required in the case of the axial compressor is unnecessary here. This is because in an accelerating flow the effect of the growth of boundary layer along the annulus passage in much less than when there is a decelerating flow with an adverge pressure gradient.

Instead of temperature drop ratio [defined in Eq (13.2)], turbine designers generally refer to the work capacity of a turbine stage as,

$$\Psi = \frac{c_p \Delta T_o}{U^2} = \frac{V_{w_2} - V_{w_3}}{V}$$

$$= \frac{V_f}{U}[\tan\beta_2 - \tan\beta_3] \qquad (13.4)$$

Ψ is a dimensionless parameter, which is called the "blade loading capacity" or "temperature drop coefficient". In gas turbine design, V_f is kept generally constant across a stage and the ratio V_f / U is called "the flow coefficient" ϕ.

Thus, Eq (13.4) can be written as,

$$\Psi = \phi[\tan\beta_2 - \tan\beta_3] \qquad (13.5)$$

As the boundary layer over the blade surface is not very sensitive in the case of a turbine, the turbine designer has considerably more freedom to distribute the total stage pressure drop between the rotor and the stator. However, locally on the suction surface of the blade there could be a zone of an adverse pressure gradient depending on the turning and on the pitch of the blades. Thus, the boundary layer could grow rapidly or even separate in such a region affecting adversely the turbine efficiency. Figure 13.3 illustrates the schematic of flow within the blade passage and the pressure distribution over the section surface depicting a zone of diffusion. Different design groups have their own rules, learned from experience of blade testing, for the amount of diffusion which is permissible particularly for highly loaded blades.

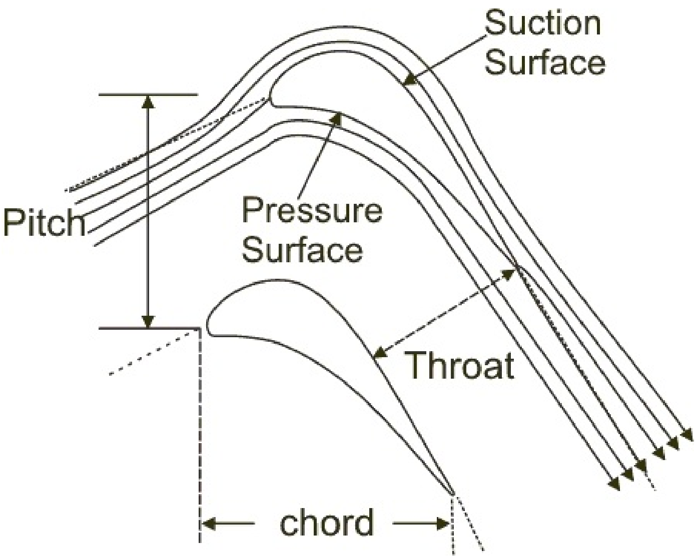


Figure 13.3a Schematic diagram of flow through a turbine blade passage

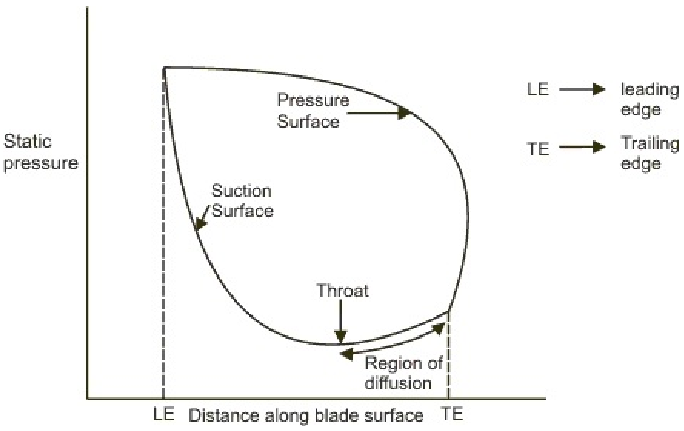


Figure13.3b Pressure distribution around a turbine blade

Degree of reaction

Another useful dimensionless parameter is the "degree for reaction" or simply the "reaction" R. It may be defined for a turbine as the fraction of overall enthalpy drop (or pressure drop) occurring in the rotor

Thus, $$R = \frac{h_2 - h_3}{h_{o_1} - h_{o_3}} \qquad (14.1)$$

or, $$\frac{T_2 - T_3}{T_{o_1} - T_{o_3}}$$

Turbine stage in which the entire pressure drop occurs in the nozzle are called "impulse stages". Stages in which a portion of the pressure drop occurs in the nozzle and the rest in the rotor are called reaction stages. In a 50% reaction turbine, the enthalpy drop in the rotor would be half of the total for the stage.

An impulse turbine stage is shown in Fig14.1, along with the velocity diagram for the common case of constant axial velocity. Since no enthalpy change occurs within the rotor, the energy equation within the rotor requires that $|V_{r_2}| = |V_{r_3}|$. If the axial velocity is held constant, then this requirement is satisfied by

$$\beta_3 = -\beta_2$$

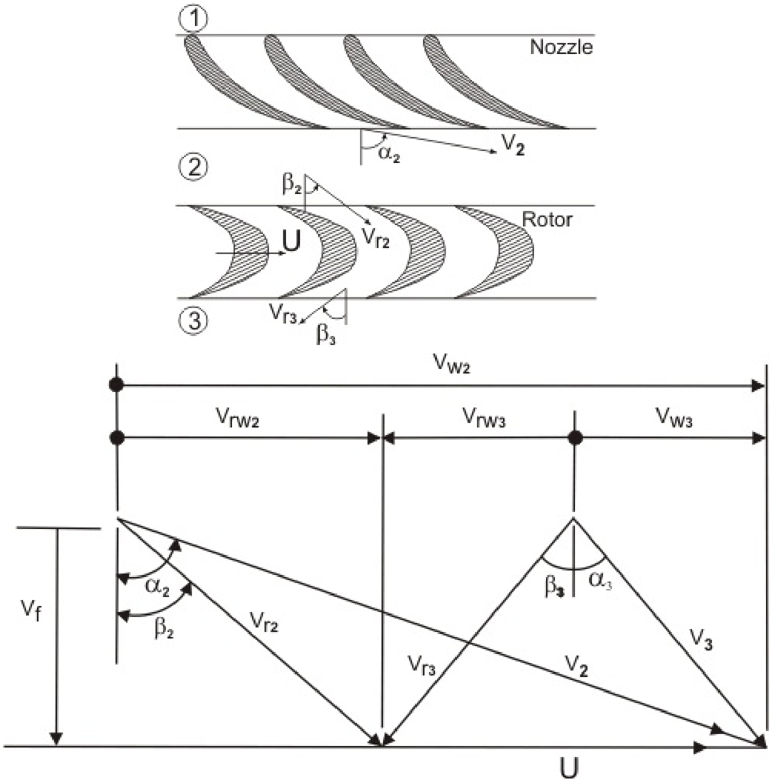


Figure 14.1 Impulse turbine stage with constant axial velocity

Degree of reaction

From the velocity diagram, we can see that

$$V_{r_{w3}} = -V_{r_{w2}}$$

i.e.

$$V_{w_2} - V_{w_3} = 2V_{r_{w2}}$$

$$= 2(V_{w_2} - U)$$

$$= 2U\left(\frac{V_f}{U}\tan\alpha_2 - 1\right)$$

Then,

$$\psi = \frac{V_{w_2} - V_{w_3}}{U}$$

$$= 2(\phi\tan\alpha_2 - 1) \qquad (14.2)$$

The Eq (14.2) illustrates the effect of the nozzle outlet angle on the impulse turbine work output.

It is evident, then, that for large power output the nozzle angle should be as large as possible. Two difficulties are associated with very large α_2. For reasonable axial velocities (i.e., reasonable flow per unit frontal area), it is evident that large α_2 creates very large absolute and relative velocities throughout the stage. High losses are associated with such velocities, especially if the relative velocity V_{r_2} is supersonic. In practice, losses seem to be minimized for values of α_2 around 70^o. In addition, one can see that for large α_2 $[\tan\alpha_2 > (2U/V_f)]$, the absolute exhaust velocity will have a swirl in the direction opposite to U. While we have not introduced the definition of turbine efficiency as yet, it is clear that, in a turbojet engine where large axial exhaust velocity is desired, the kinetic energy associated with the tangential motion of the exhaust gases is essentially a loss. Furthermore, application of the angular momentum equation over the entire engine indicates that exhaust swirl is associate with an (undesirable) net torque acting on the aircraft. Thus the desire is for axial or near-axial absolute exhaust velocity (at least for the last stage if a multistage turbine is used). For the special case of constant V_f and axial exhaust velocity $V_{w_3} = 0$ and $V_{w_2} = 2U$, the Eq.14.2 becomes,

$$\psi = 2 \qquad [\because \tan\alpha_2 = \frac{V_w}{V_f} = \frac{2U}{V_f} = 2/\phi]$$

For a given power and rotor speed, and for a given peak temperature, Eq. (14.2) is sufficient to determine approximately the mean blade speed (and hence radius) of a single-stage impulse turbine having axial outlet velocity. If , as is usually the case, the blade speed is too high (for stress limitations), or if the mean diameter is too large relative to the other engine components, it is necessary to employ a multistage turbine in which each stage does part of the work.

Degree of reaction

It has been shown that the 50% reaction compressor stage (with constant V_f) has symmetrical blading.

The same is true for the 50% reaction turbine stage. As the change in static enthalpy is same in both stator and rotor blades, the change in kinetic energy relative to each blade row must be the same. The velocity diagram for a 50% reaction stage with a constant axial velocity is shown in Fig 14.2.

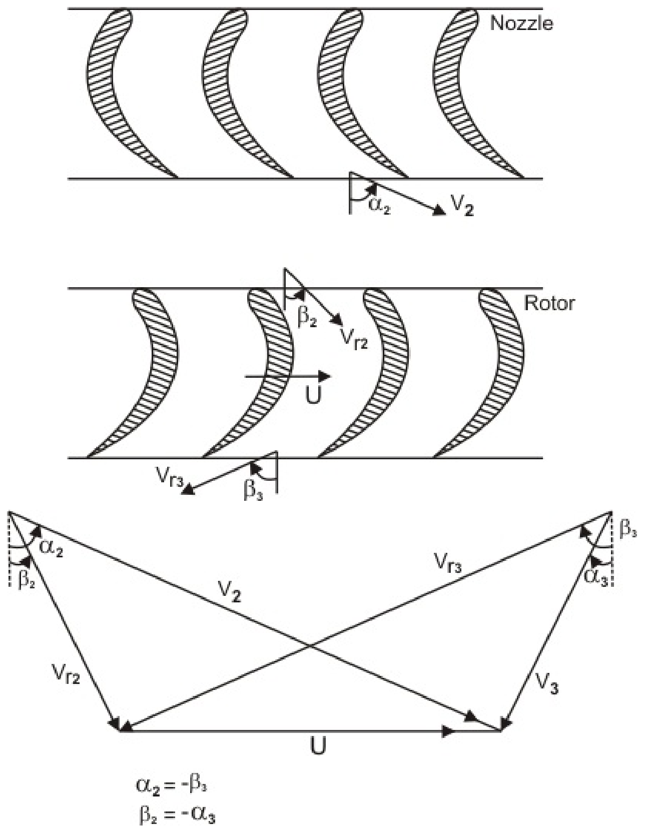


Figure 14.2 Fifty -percent reaction turbine with constant axial velocity

Since the velocity diagram is symmetrical,

$$V_{r_{w2}} = V_{w_2} - U$$

$$= -V_{w_3}$$

i.e

$$V_{w_3} = -(V_{w_2} - U)$$

or,

$$V_{w_2} - V_{w_3} = 2V_{w_2} - U$$

or,

$$\psi = \frac{V_{w_2} - V_{w_3}}{U}$$

$$= 2\frac{V_f}{U}\tan\alpha_2 - 1$$

or,

$$\psi = 2\phi\tan\alpha_2 - 1 \qquad (14.3)$$

Degree of reaction

Again the desirability of large α_2 is indicated and the same limitations are encountered, so that typical values of α_2 are near 70^o. For the special case of axial outlet velocity and constant V_f, α_3 and β_2 are zero and the velocity diagram becomes a rectangle. The stage work output is then

$$\psi = 1$$

Thus, for the same blade speed and for axial outlet velocities, the impulse stage work is twice that of the 50% reaction stage. However, we can expect the impulse stage to have somewhat greater loss, since the average fluid velocity in the stage is higher and the boundary layer on the suction side of the rotor blades may be significantly thicker and closer to separation, depending on the turning angle and blade spacing. The 50% reaction stage is not uniquely desirable, of course. One can use any degree of reaction (greater then zero) to design a turbine of acceptable performance.

The gas flow angles at inlet and exit of blades can be expressed in terms of Ψ, ϕ and R.

For the rotor blade, the relative total enthalpy remains constant and we have,

$$h_2 + \frac{V_{r_2}^2}{2} = h_3 + \frac{V_{r_3}^2}{2}$$

or,

$$h_2 - h_3 = \frac{V_{r_3}^2}{2} - \frac{V_{r_2}^2}{2}$$

If the axial velocity is the same upstream and downstream of the rotor, then

$$h_2 - h_3 = \frac{(V_{r_{w_3}} - V_{r_{w_2}})(V_{r_{w_3}} + V_{r_{w_2}})}{2}$$

The Eq.(14.1) becomes,

$$R = \frac{(V_{r_{w3}} - V_{r_{w_2}})(V_{r_{w_3}} + V_{r_{w2}})}{(V_{w_2} - V_{w_3})}$$

Again from the velocity triangle (Fig 13.2),

$$V_{w_2} - V_{w_3} = V_{r_{w2}} - V_{r_{w3}}$$

Thus,

$$R = -\left(\frac{V_{r_{w3}} + V_{r_{w2}}}{2U}\right) \qquad (14.4)$$

$$= -\frac{1}{2}\frac{V_f}{U}(\tan\beta_2 + \tan\beta_3)$$

$$R = -\frac{1}{2}\phi\,(\tan\beta_2 + \tan\beta_3) \qquad (14.5)$$

Solving Eq.13.5 and Eq.14.5, we have

$$\tan\beta_2 = (\psi - 2R)/2\phi \qquad (14.6)$$

$$\tan\beta_3 = (\psi + 2R)/2\phi \qquad (14.7)$$

and from geometric relation

$$\tan\alpha_2 = \tan\beta_2 + \frac{1}{\phi} \qquad (14.8)$$

$$\tan\alpha_3 = \tan\beta_3 + \frac{1}{\phi} \qquad (14.9)$$

Hence, from given values of Ψ, ϕ and R we can estimate gas flow angles and the blade layout.

Again, combining Eq.14.5 and Eq.14.8, we have

$$R = \frac{1}{2}[1 - \phi(\tan\alpha_2 + \tan\beta_3)] \qquad (14.10)$$

Which is the expression for R in terms of the exit air angles. For the special case of symmetrical blading , $\alpha_2 = -\beta_3$ and we have $R = 1/2$. For the case of $V_{r_{w3}} = -V_{r_{w2}}$, we have $R = 0$. Now for the special case of zero exit swirl, $V_{w_3} = 0$ and it follows that $V_{r_{w3}} = V_f \tan\beta_3 = -U$, i.e. $\tan\beta_3 = -\frac{1}{\phi}$ and Eq. 14.10 because

$$R = 1 - \frac{1}{2}\phi\tan\alpha_2 \qquad (14.11)$$

Again for zero exit swirl, the blade loading capacity, Eq.13.5 reduces to

$$\psi = \phi\tan\alpha_2 \qquad (14.12) \text{ since } [\alpha_3 = 0]$$

Equations (14.11) and (14.12) have been used in plotting Fig (14.3), which pertains to design conditions only.

Here we see that for a given stator outlet angle, the impulse stage requires a much higher axial velocity ratio than does the 50% reaction stage. In the impulse stage all flow velocities are higher, and that is one reason why its efficiency is lower than that of the 50% reaction stage.

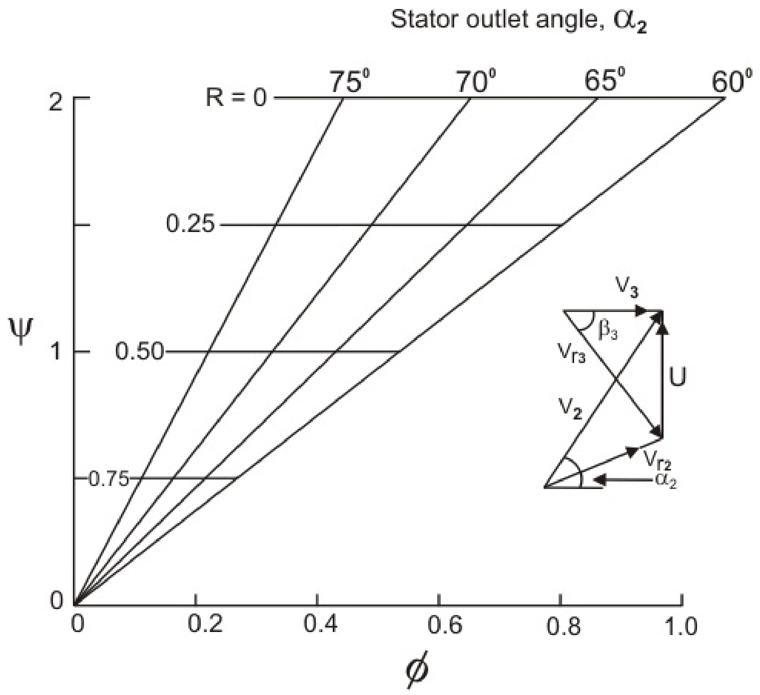


Figure 14.3 Work capacity Ψ and degree of reaction R of axial turbine stages design for zero exit swirl.

STAGE EFFICENCY

The aerodynamic losses in the turbine differ with the stage configuration, that is, the degree of reaction. Improved efficiency is associated with higher reaction, which tends to mean less work per stage and thus a large number of stages for a given overall pressure ratio.

The understanding of aerodynamic losses is important to design, not only in the choice of blading type (impulse or reaction) but also in devising ways to control these losses, for example, methods to control the clearance between the tip of the turbine blade and the outer casing wall. The choices of blade shape, aspect ratio, spacing, Reynolds number, Mach number and flow incidence angle can all affect the losses and hence the efficiency of turbine stages.

Two definitions of efficiency are in common usage: the choice between them depends on the application for which the turbine is used. For many conventional applications, useful turbine output is in the form of shift power and the kinetic energy of the exhaust, $V_3^2/2$, is considered as a loss. In this case, ideal work would be $C_P(T_{01}-T_{3s})$ and a total to static turbine efficiency, η_{ts} , based on the inlet and exit static conditions, is used.

Thus, $$\eta_{ts}=\frac{T_{o1}-T_{o3}}{T_{o1}-T_{3s}} \qquad (15.1)$$

The ideal (isentropic) to actual expansion process in turbines is illustrated in Fig 15.1.

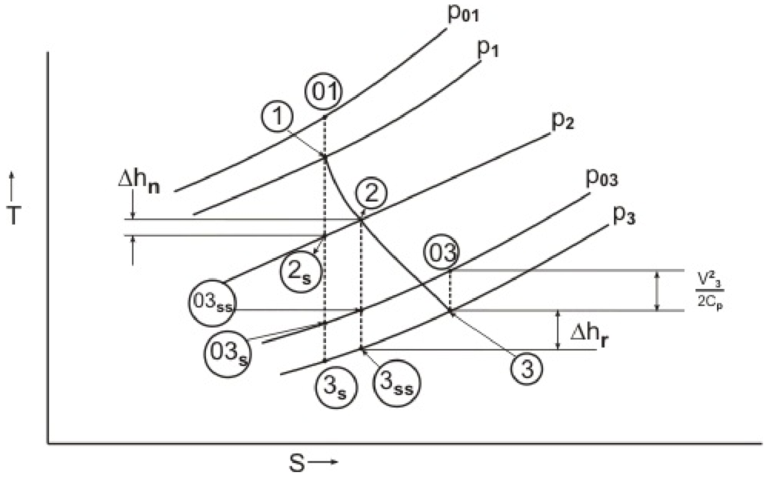


Figure 15.1 T-S diagram: expansion in a turbine

Further,

$$\eta_{ts}=\frac{T_{o1}-T_{o3}}{T_{o1}[1-(P_3/P_{o1})^{(\gamma-1)/\gamma}]}$$

$$=\frac{1-(T_{o3}/T_{o1})}{1-(p_3/p_{o1})^{(\gamma-1)/\gamma}} \qquad (15.2)$$

In some applications, particularly in turbojets, the exhaust kinetic energy is not considered a loss since the exhaust gases are intended to emerge at high velocity. The ideal work in this case is then $C_P(T_{01}-T_{03s})$ rather than $C_P(T_{01}-T_{3s})$. This requires a different definition of efficiency, the total-to-total turbine efficiency η_{tt}, defined by

$$\eta_{tt}=\frac{T_{01}-T_{03}}{T_{01}-T_{03s}}=\frac{1-(T_{03}/T_{01})}{1-(p_{03}/p_{01})^{(\gamma-1)/\gamma}} \qquad (15.3)$$

One can compare η_{tt} & η_{ts} by making the approximation,

$$T_{03s}-T_{3s}\cong T_{03}-T_3=V_3^2/2C_p,$$

and using Eqs. (15.2) and (15.3) it can be shown that

$$\eta_{tt}=\frac{\eta_{ts}}{1-V_3^2[2C_p(T_{01}-T_{3s})]}$$

Thus $$\eta_{tt}>\eta_{ts}$$

The actual turbine work can be expressed as,

$$W_t = \eta_{tt} C_p T_{01} [1 - (\frac{P_{03}}{P_{01}})^{(\gamma-1)/r}]. \quad (15.4)$$

or, $$W_t = \eta_{ts} C_p T_{01} [1 - (\frac{P_3}{P_{01}})^{(\gamma-1)/r}].$$

STAGE EFFICENCY

Despite the development in computational methods to predict the flow field in turbine blade passages, the estimation of stage losses and thus efficiency is still a matter of considerable difficulty. In addition to the primary flow through the blade passage, there are secondary flows which move fluid across the blade passages under the action of centrifugal and coriolis forces; blade loading effects causing incidence and deviation; leakage between the moving blade tip and the stationary shroud; the boundary layers and wakes shed by blades; and for transonic blades, shock waves in the blade passage and at the trailing edges. Another class of effects is the unsteady generated mainly by the interaction of adjacent blade rows. All these contribute to losses.In general, cascade tests of different blade geometries are performed.The results from these cascade tests can be correlated to define the loss coefficients for the stator (nozzle) and rotor blade of turbines.

With reference to the Fig15.1, the effects of loss and thus irreversibility through the stator and rotor are expressed by differences in static enthalpies, $(h_2 - h_{2s})$ and $(h_3 - h_{3ss})$ respectively. Non-dimensional "enthalpy loss coefficient" for the nozzle can be defined as,

$$h_2 - h_{2s} = \frac{1}{2} V_2^2 \zeta_N$$

Similarly, the "enthalpy loss coefficient" for the rotor,

$$h_3 - h_{3ss} = \frac{1}{2} V_{r_3}^2 \zeta_R$$

Thus, the expressions for the efficiency can be approximated as:

$$\eta_{tt} = \left[1 + \frac{\zeta_R V_{r_3}^2 + \zeta_N V_2^2}{2(h_1 - h_3)} \right]^{-1} \qquad (15.5)$$

$$\eta_{ts} = \left[1 + \frac{\zeta_R V_{r_3}^2 + \zeta_N V_2^2 + V_1^2}{2(h_1 - h_3)} \right]^{-1} \qquad (15.6)$$

While designing a turbine stage for a particular application, the restriction arises from the view point of blade stress rather than from the aerodynamics to achieve the maximum possible efficiency. In short, the blade speed is limited by the blade stress, particularly in high temperature applications. The turbine designers will often work to a maximum value of blade speed defined by temperature and material properties. Thus in modern times, the turbine blade cooling in very vital, which determines the life of an engine (particularly for the turbojet engine). In many applications, the characteristics of the compressor which the turbine drives also impose limits on the turbine speed.

Turbine Performance

For a given design of turbine operating with a given fluid at sufficiently high Reynolds member, it can be shown from the dimensional analysis as,

$$\frac{P_{0_2}}{P_{0_3}} = f\left(\frac{\dot{m}\sqrt{RT_{0_2}}}{P_{0_2} D^2}, \frac{\Omega D}{\sqrt{\gamma R T_{0_2}}}\right),$$

where, stagnation states 02 and 03 are at the turbine inlet and outlet, respectively. Figure (15.2) shows the overall performance of a particular single-stage turbine.

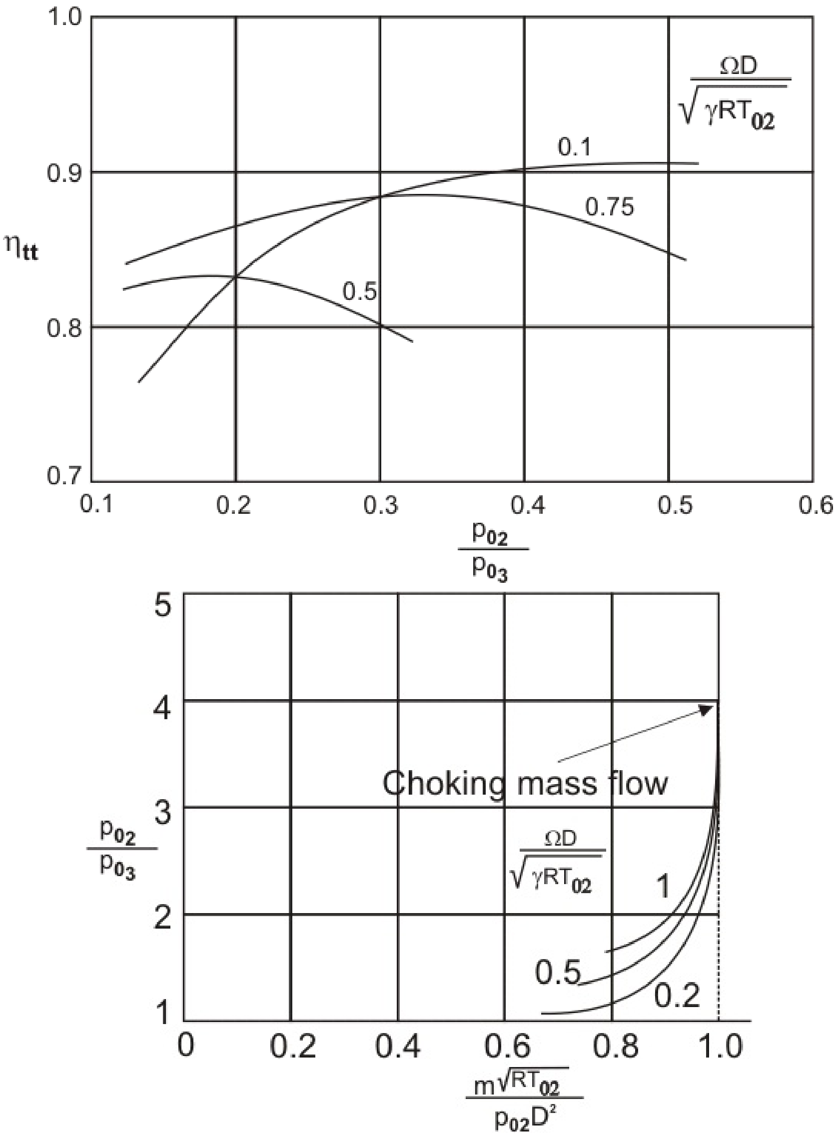


Figure15.2 Typical characteristics of a turbine stage

One can see that pressure ratios greater than those for compressor stages can be obtained with satisfactory efficiency.

The performance of turbines is limited principally by two factors: compressibility and stress. Compressibility limits the mass flow that can pass through a given turbine and, as we will see, stress limits the wheel speed U. The work per stage, for example, depends on the square of the wheel speed. However, the performance of the engine depends very strongly on the maximum temperature. Of course, as the maximum temperature increases, the allowable stress level diminishes; hence in the design of the engine there must be a compromise between maximum temperature and maximum rotor tip speed U.

For given pressure ratio and adiabatic efficiency, the turbine work per unit mass is proportional to the inlet stagnation temperature. Since, in addition, the turbine work in a jet or turboshaft engine is commonly two or three times the useful energy output of the engine, a 1% increase in turbine inlet temperature can produce a 2% or 3% increase in engine output. This considerable advantage has supplied the incentive for the adoption of fairly elaborate methods for cooling the turbine nozzle and rotor blades.

A brief note on Gas Turbine Combustors

Over a period of five decades, the basic factors influencing the design of combustion systems for gas turbines have not changed, although recently some new requirements have evolved. The key issues may be summarized as follows.

• The temperature of the gases after combustion must be comparatively controlled to suit the highly stressed turbine materials. Development of improved materials and methods of blade cooling, however, has enabled permissible combustor outlet temperatures to rise from about 1100K to as much as 1850 K for aircraft applications.

• At the end of the combustion space the temperature distribution must be of known form if the turbine blades are not to suffer from local overheating. In practice, the temperature can increase with radius over the turbine annulus, because of the strong influence of temperature on allowable stress and the decrease of blade centrifugal stress from root to tip.

• Combustion must be maintained in a stream of air moving with a high velocity in the region of 30-60 m/s, and stable operation is required over a wide range of air/fuel ratio from full load to idling conditions. The air/fuel ratio might vary from about 60:1 to 120:1 for simple cycle gas turbines and from 100:1 to 200:1 if a heat-exchanger is used. Considering that the stoichiometric ratio is approximately 15:1, it is clear that a high dilution is required to maintain the temperature level dictated by turbine stresses

• The formation of carbon deposits ('coking') must be avoided, particularly the hard brittle variety. Small particles carried into the turbine in the high-velocity gas stream can erode the blades and block cooling air passages; furthermore, aerodynamically excited vibration in the combustion chamber might cause sizeable pieces of carbon to break free resulting in even worse damage to the turbine.

• In aircraft gas turbines, combustion must be stable over a wide range of chamber pressure because of the substantial change in this parameter with a altitude and forward speed. Another important requirement is the capability of relighting at high altitude in the event of an engine flame-out.

• Avoidance of smoke in the exhaust is of major importance for all types of gas turbine; early jet engines had very smoky exhausts, and this became a serious problem around airports when jet transport aircraft started to operate in large numbers. Smoke trails in flight were a problem for military aircraft, permitting them to be seen from a great distance. Stationary gas turbines are now found in urban locations, sometimes close to residential areas.

• Although gas turbine combustion systems operate at extremely high efficiencies, they produce pollutants such as oxides of nitrogen (NO_x) , carbon monoxide (CO) and unburned hydrocarbons (UHC) and these must be controlled to very low levels. Over the years, the performance of the gas turbine has been improved mainly by increasing the compressor pressure ratio and turbine inlet temperature (TIT). Unfortunately this results in increased production of NO_x . Ever more stringent emissions legislation has led to significant changes in combustor design to cope with the problem.

Probably the only feature of the gas turbine that eases the combustion designer's problem is the peculiar interdependence of compressor delivery air density and mass flow which leads to the velocity of the air at entry to the combustion system being reasonably constant over the operating range.

For aircraft applications there are the additional limitations of small space and low weight, which are, however, slightly offset by somewhat shorter endurance requirements. Aircraft engine combustion chambers are normally constructed of light-gauge, heat-resisting alloy sheet (approx. 0.8 mm thick), but are only expected to have a life of some 10000 hours. Combustion chambers for industrial gas turbine plant may be constructed on much sturdier lines but, on the other hand, a life of about 100000 hours is required. Refractory linings are sometimes used in heavy chambers, although the remarks made above regarding the effects of hard carbon deposits breaking free apply with even greater force to refractory material.

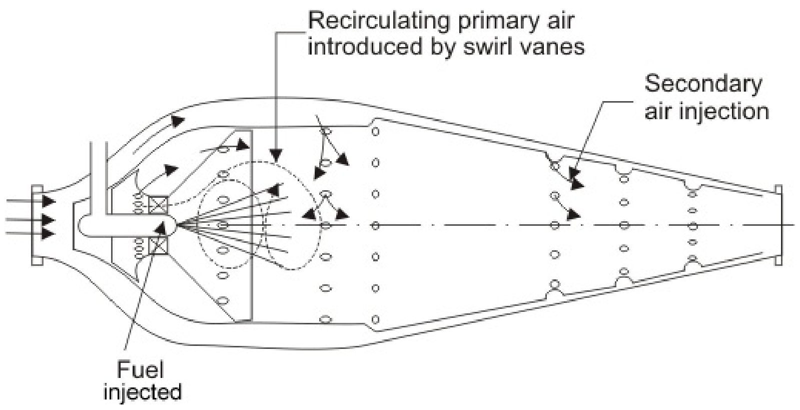


Figure 16.1 Combustion chamber with swirl vanes

Figure 16.1 indicates the schematic of a combustion chamber. The primary air is introduced through twisted radial vanes known as 'swirl vanes', that results in a vortex motion with a low-pressure region along the axis of the chamber. The fuel is injected in the same direction of air. The vortex motion is some time enhanced by injecting the secondary air through short tangential chutes in the flame tube. The burning gases tends to flow towards the region of low pressure and some portion of them swept round towards the jet of fuel as indicated by the arrow. The objective is to obtain a stable flame.

Aircraft Engines and Propulsion System

The modern aircraft engine have the ability to actuate massive airstream and thus to produce high thrust. The engine airflow rate is perhaps 50 times the fuel flow rate, and the term air breathing engine is quite appropriate. Thus, a continuous stream of air flows through the air-breathing engine. The air is compressed, mixed with fuel, ignited, expanded through a turbine and then expelled as the exhaust gas.

The following four types of aircraft engines are generally used

- Turbojet Engine
- Turboprop Engine
- Turbofan Engine
- Ramjet Engine

At low speeds, propeller propulsion is more efficient than jet propulsion. Conventional propellers, however, become inefficient and noisy at flight speeds higher than 0.5 or 0.6 times the speed of sound. In contrast, turbojet and turbofan engines can function efficiently and quietly at flight speeds as high as 0.85 times the speed of sound. Turbojets can also operate at supersonic flight speeds. Ramjet, which is the simplest of all air-breathing engines can operate at a higher speed than turbojet engines and is mostly suitable for supersonic flight.

1. Turbojet Engine

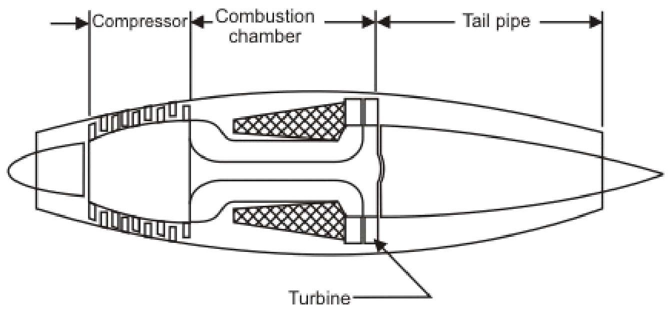


Figure 16.2 Turbojet

The turbojet engine consists of a gas turbine, the output of which is used solely to provide power to the compressor. The compressor and the turbine are normally mounted on common shaft. Air is taken into the engine through an approximate diffuser duct, passes through the compressor and enters the combustions chamber, where it is mixed and burned with fuel.

Most common fuels are hydrocarbons (Aviation kerosene). The ratio of fuel to air is determined by the maximum allowable gas temperature permitted by the turbine. Normally, a considerable excess air is used. The hot high pressure gases are then expanded through the turbine to a pressure which is higher than the ambient atmosphere, and yet sufficiently lower than the combustion chamber pressure, to produce just enough power in the turbine to run the compressor. After leaving the turbine, the gas is expanded to the ambient pressure through an appropriate nozzle. As this occurs, the gas is accelerated to a velocity, which is greater than the incoming velocity of the ingested air, and therefore produces a propulsive thrust.

2. Turboprop Engine

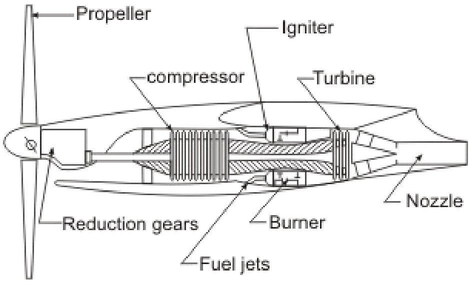


Figure 17.1 Turboprop

In this engine, a higher proportion of the total available pressure drop from the combustion chamber to the atmosphere is taken through the turbine and a smaller proportion through the propulsive nozzle. This strategy produces excess power in the turbine. The excess power is used not only to drive the compressor, but also to drive a propeller, in the same way as in the conventional reciprocating engines. Here the major portion of the thurst is generated by the propeller.

3. Turbofan Engine

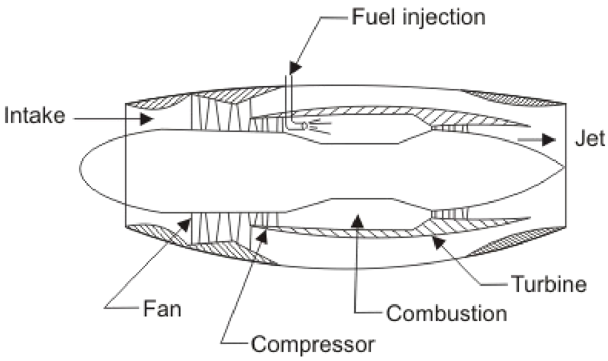


Figure 17.2 Turbofan

This is another variety of gas-turbine aircraft engine. This is very similar to the turboprop in principle, except that a fan is used instead of a propeller, and this fan is contained within a duct as shown in the above figure. Most airliners use modern turbofan engines because of their high thrust and good fuel efficiency. Figure below shows the picture of Boeing 747 aircraft that uses a turbofan engine. In a turbofan engine , the air is sucked by the engine inlet. Some of the incoming air passes through the fan and continues on into the core compressor and then the burner, where it is mixed with fuel and combustion occurs. The hot exhaust passes through the core and fan turbines and then out the nozzle, as in a basic turbojet. The rest of the incoming air passes through the fan and bypasses, or goes around the engine, just like the air through a propeller. The air that goes through the fan has a velocity that is slightly increased from free stream. So a turbofan gets some of its thrust from the core and some of its thrust from the fan. The ratio of the air that goes around the engine to the air that goes through the core is called the **bypass ratio**. Because the fuel flow rate for the core is changed only a small amount by the addition of the fan, a turbofan generates more thrust for nearly the same amount of fuel used by the core. This means that a turbofan is very fuel efficient. In fact, high bypass ratio turbofans are nearly as fuel efficient as turboprops.

Boeing 747:Typical example of a Turbofan Engine

4. Ramjet Engine

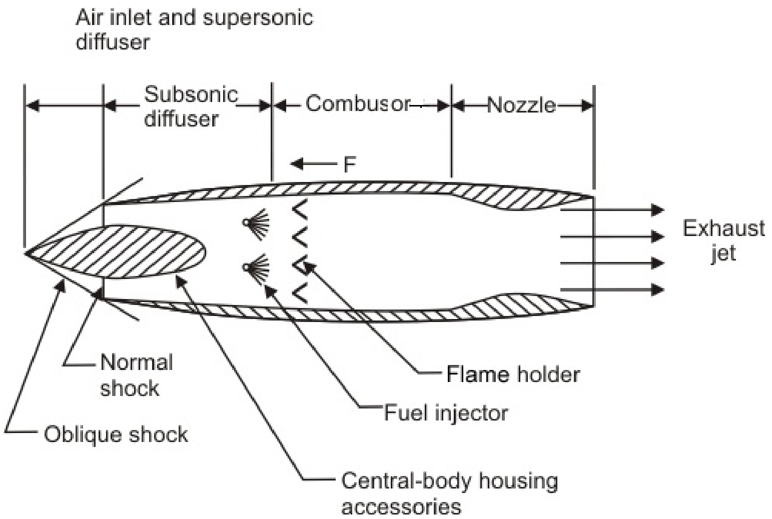


Figure 17.3 Ramjet

At higher forward speeds, the ram pressure of the air is already very large, and the necessity for a compressor tends to disappear. A turbojet engine minus the compressor and turbine, but with a combustion chamber, is known as a ramjet engine. Such engines simply consists of

1. A duct designed to diffuse the incoming air, slowing its velocity and raising its pressure
2. A combustor, designed to heat the air, normally by combustion with a liquid fuel
3. A nozzle, designed to expand and accelerate the heated gases rearwards

The ramjet engine does not accelerate itself from a standing start but requires some other form of propulsion, usually a rocket, to accelerate it to near its operating speed.

MODULE 4

Steam Turbine

Next

Steam Turbine

Introduction

A steam turbine converts the energy of high-pressure, high temperature steam produced by a steam generator into shaft work. The energy conversion is brought about in the following ways:

1. The high-pressure, high-temperature steam first expands in the nozzles emanates as a high velocity fluid stream.
2. The high velocity steam coming out of the nozzles impinges on the blades mounted on a wheel. The fluid stream suffers a loss of momentum while flowing past the blades that is absorbed by the rotating wheel entailing production of torque.
3. The moving blades move as a result of the impulse of steam (caused by the change of momentum) and also as a result of expansion and acceleration of the steam relative to them. In other words they also act as the nozzles.

A steam turbine is basically an assembly of nozzles fixed to a stationary casing and rotating blades mounted on the wheels attached on a shaft in a row-wise manner. In 1878, a Swedish engineer, Carl G. P. de Laval developed a simple impulse turbine, using a convergent-divergent (supersonic) nozzle which ran the turbine to a maximum speed of 100,000 rpm. In 1897 he constructed a velocity-compounded impulse turbine (a two-row axial turbine with a row of guide vane stators between them.

Auguste Rateau in France started experiments with a de Laval turbine in 1894, and developed the pressure compounded impulse turbine in the year 1900.

In the USA , Charles G. Curtis patented the velocity compounded de Lavel turbine in 1896 and transferred his rights to General Electric in 1901.

In England , Charles A. Parsons developed a multi-stage axial flow reaction turbine in 1884.

Steam turbines are employed as the prime movers together with the electric generators in thermal and nuclear power plants to produce electricity. They are also used to propel large ships, ocean liners, submarines and to drive power absorbing machines like large compressors, blowers, fans and pumps.

Turbines can be condensing or non-condensing types depending on whether the back pressure is below or equal to the atmosphere pressure.

Flow Through Nozzles

A *nozzle* is a duct that increases the velocity of the flowing fluid at the expense of pressure drop. A duct which decreases the velocity of a fluid and causes a corresponding increase in pressure is a *diffuser* . The same duct may be either a nozzle or a diffuser depending upon the end conditions across it. If the cross-section of a duct decreases gradually from inlet to exit, the duct is said to be convergent. Conversely if the cross section increases gradually from the inlet to exit, the duct is said to be divergent. If the cross-section initially decreases and then increases, the duct is called a convergent-divergent nozzle. The minimum cross-section of such ducts is known as throat. A fluid is said to be *compressible* if its density changes with the change in pressure brought about by the flow. If the density does not changes or changes very little, the fluid is said to be incompressible. Usually the gases and vapors are compressible, whereas liquids are *incompressible* .

Nozzle, Steam Nozzle and Steam Turbine

STAGNATION, SONIC PROPERTIES AND ISENTROPIC EXPANSION IN NOZZLE

The stagnation values are useful reference conditions in a compressible flow. Suppose the properties of a flow (such as T, p, ρ etc.) are known at a point. The stagnation properties at a point are defined as those which are to be obtained if the local flow were imagined to cease to zero velocity isentropically. The stagnation values are denoted by a subscript zero. Thus, the stagnation enthalpy is defined as

$$h_0 = h + \frac{1}{2}V^2$$

For a calorically perfect gas, this yields,

$$c_p T_0 = c_p T + \frac{1}{2}V^2 \qquad (18.1)$$

which defines the stagnation temperature. It is meaningful to express the ratio of (T_0 / T) in the form

$$\frac{T_0}{T} = 1 + \frac{V^2}{2c_p T} = 1 + \frac{\gamma - 1}{2} \cdot \frac{V^2}{\gamma R T}$$

$$\text{or,} \quad \frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} Ma^2 \qquad (18.2)$$

If we know the local temperature (T) and Mach number (Ma), we can fine out the stagnation temperature T_0. Consequently, isentropic relations can be used to obtain stagnation pressure and stagnation density as.

$$\frac{p_0}{p} = \left(\frac{T_0}{T}\right)^{\frac{\gamma}{\gamma-1}} = \left[1 + \frac{\gamma - 1}{2} Ma^2\right]^{\frac{\gamma}{\gamma-1}} \qquad (18.3)$$

$$\frac{\rho_0}{\rho} = \left(\frac{T_0}{T}\right)^{\frac{1}{\gamma-1}} = \left[1 + \frac{\gamma - 1}{2} Ma^2\right]^{\frac{\gamma}{\gamma-1}} \qquad (18.4)$$

In general, the stagnation properties can vary throughout the flow field.

However, if the flow is adiabatic, then $h + \frac{V^2}{2}$ is constant throughout the flow. It follows that the h_0, T_0 and a_0 are constant throughout an adiabatic flow, even in the presence of friction. Here a is the speed of sound and the suffix signifies the stagnation condition. It is understood that all stagnation properties are constant along an isentropic flow. If such a flow starts from a large reservoir where the fluid is practically at rest, then the properties in the reservoir are equal to the stagnation properties everywhere in the flow (Fig. 18.1).

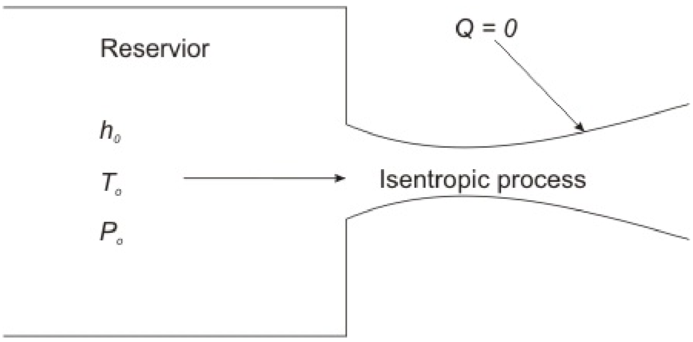


Fig 18.1 An isentropic process starting from a reservoir

There is another set of conditions of comparable usefulness where the flow is sonic, Ma=1.0. These sonic, or critical properties are denoted by asterisks: p^*, ρ^*, a^*, and. T^*. These properties are attained if the local fluid is imagined to expand or compress isentropically until it reachers Ma=1.

We have already discussed that the total enthalpy, hence T_0, is conserved so long the process is adiabatic, irrespective of frictional effects. In contrast, the stagnation pressure p_0 and density ρ_0 decrease if there is friction.

From Eq.(18.1), we note that

$$V^2 = 2c_p (T_0 - T)$$

$$\text{or,} \quad V = \left[\frac{2\gamma R}{\gamma - 1}(T_0 - T)\right]^{\frac{1}{2}} \qquad (18.5a)$$

is the relationship between the fluid velocity and local temperature (T), in an adiabatic flow. The flow can attain a maximum velocity of

$$V_{\max} = [\frac{2\gamma R T_0}{\gamma - 1}]^{\frac{1}{2}} \qquad (18.5b)$$

As it has already been stated, the unity Mach number, Ma=1, condition is of special significance in compressible flow, and we can now write from Eq.(18.2), (18.3) and (18.4).

$$\frac{T_0}{T*} = \frac{1+\gamma}{2} \qquad (18.6a)$$

$$\frac{p_0}{p*} = (\frac{1+\gamma}{2})^{\frac{\gamma}{\gamma-1}} \qquad (18.6b)$$

$$\frac{\rho_0}{\rho*} = (\frac{1+\gamma}{2})^{\frac{\gamma}{\gamma-1}} \qquad (18.6c)$$

For diatomic gases, like air $\gamma = 1.4$, the numerical values are

$$\frac{T*}{T_0} = 0.8333, \quad \frac{p*}{p_0} = 0.5282, \quad and \quad \frac{\rho*}{\rho_0} = 0.6339$$

The fluid velocity and acoustic speed are equal at sonic condition and is

$$V* = a* = [\gamma R T*]^{1/2} \qquad (18.7a)$$

or, $$V* = [\frac{2\gamma}{\gamma+1} R T_0]^{\frac{1}{2}} \qquad (18.7b)$$

We shall employ both stagnation conditions and critical conditions as reference conditions in a variety of one dimensional compressible flows.

Effect of Area Variation on Flow Properties in Isentropic Flow

In considering the effect of area variation on flow properties in isentropic flow, we shall concern ourselves primarily with the velocity and pressure. We shall determine the effect of change in area, *A,* on the velocity *V*, and the pressure *p*.

From Bernoulli's equation, we can write

$$\frac{dp}{\rho} + d\left(\frac{V^2}{2}\right) = 0$$

$$\text{or,} \quad dp = -\rho V\, dV$$

Dividing by ρV^2, we obtain

$$\frac{dp}{\rho V^2} = -\frac{dV}{V} \qquad (19.1)$$

A convenient differential form of the continuity equation can be obtained from Eq. (14.50) as

$$\frac{dA}{A} = -\frac{dV}{V} - \frac{d\rho}{\rho}$$

Substituting from Eq. (19.1),

$$\frac{dA}{A} = \frac{dp}{\rho V^2} - \frac{d\rho}{\rho}$$

or,

$$\frac{dA}{A} = \frac{dp}{\rho V^2}\left[1 - \frac{V^2}{dp/d\rho}\right] \qquad (19.2)$$

Invoking the relation ($a^2 = \frac{dp}{d\rho}$) for isentropic process in Eq. (19.2), we get

$$\frac{dA}{A} = \frac{dp}{\rho V^2}\left[1 - \frac{V^2}{a^2}\right] = \frac{dp}{\rho V^2}\left[1 - Ma^2\right] \qquad (19.3)$$

From Eq. (19.3), we see that for Ma<1 an area change causes a pressure change of the same sign, i.e. positive d*A* means positive d*p* for Ma<1. For Ma>1, an area change causes a pressure change of opposite sign.

Again, substituting from Eq.(19.1) into Eq. (19.3), we obtain

$$\frac{dA}{A} = -\frac{dV}{V}\left[1 - Ma^2\right] \qquad (19.4)$$

From Eq. (19.4), we see that Ma<1 an area change causes a velocity change of opposite sign, i.e. positive d*A* means negative d*V* for Ma<1. For Ma>1, an area change causes a velocity change of same sign.

These results are summarized in Fig.19.1, and the relations (19.3) and (19.4) lead to the following important conclusions about compressible flows:

1. At subsonic speeds (Ma<1) a decrease in area increases the speed of flow. A subsonic nozzle should have a convergent profile and a subsonic diffuser should possess a divergent profile. The flow behaviour in the regime of Ma<1 is therefore qualitatively the same as in incompressible flows.
2. In supersonic flows (Ma>1), the effect of area changes are different. According to Eq. (19.4), a supersonic nozzle must be built with an increasing area in the flow direction. A supersonic diffuser must be a converging channel. Divergent nozzles are used to produce supersonic flow in missiles and launch vehicles.

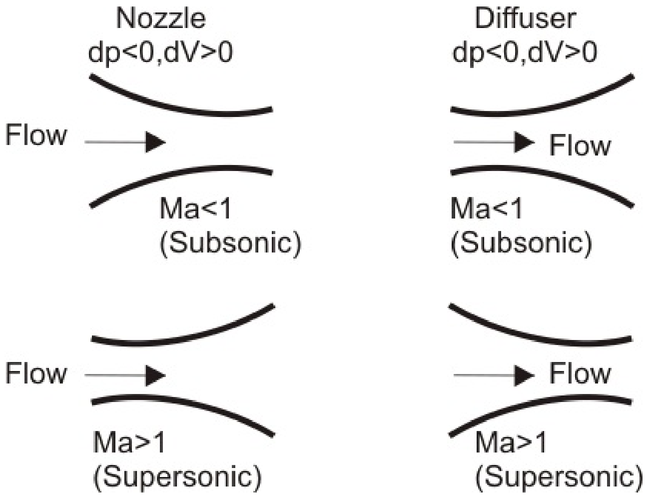


Fig 19.1 Shapes of nozzles and diffusersin subsonic and supersonic regimes

Suppose a nozzle is used to obtain a supersonic stream staring from low speeds at the inlet (Fig.19.2). Then the Mach number should increase from Ma=0 near the inlet to Ma>1 at the exit. It is clear that the nozzle must converge in the subsonic portion and diverge in the supersonic portion. Such a nozzle is called a *convergent-divergent nozzle.* A convergent-divergent nozzle is also called a *de Laval nozzle*, after Carl G.P. de Laval who first used such a configuration in his steam turbines in late nineteenth century (this has already been mentioned in the introductory note). From Fig.**19.2** it is clear that the Mach number must be unity at the throat, where the area is neither increasing nor decreasing. This is consistent with Eq. (19.4) which shows that dV can be non-zero at the throat only if Ma=1. It also follows that the sonic velocity can be achieved only at the throat of a nozzle or a diffuser.

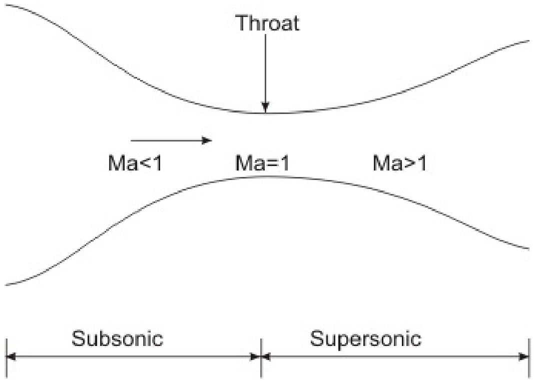


Fig 19.2 A convergent-divergent nozzle

The condition, however, does not restrict that *Ma* must necessarily be unity at the throat, According to Eq. (19.4), a situation is possible where $Ma \neq 1$ at the throat if dV=0 there. For an example, the flow in a convergent-divergent duct may be subsonic everywhere with *Ma* increasing in the convergent portion and decreasing in the divergent portion with $Ma \neq 1$ at the throat (see Fig.19.3). The first part of the duct is acting as a nozzle, whereas the second part is acting as a diffuser. Alternatively, we may have a convergent-divergent duct in which the flow is supersonic everywhere with *Ma* decreasing in the convergent part and increasing in the divergent part and again $Ma \neq 1$ at the throat (see Fig. 19.4).

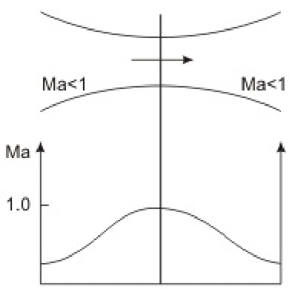


Fig 19.3 Convergent-divergent duct with $Ma \neq 1$ at throat

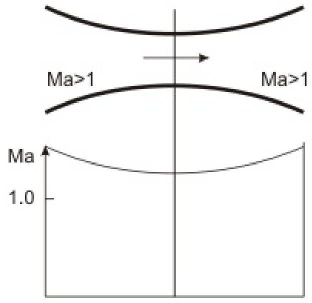
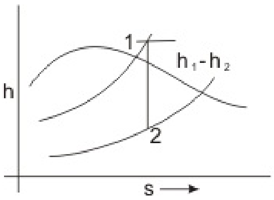
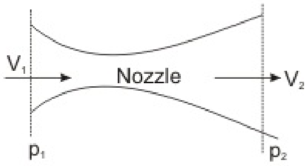


Fig 19.4 Convergent-divergent duct with $Ma \neq 1$ at throat

Iscentropic Flow of a vapor or gas through a nozzle





First law of thermodynamics:

$$h_1 + \frac{V_1^2}{2} = h_2 + \frac{V_2^2}{2}$$

$$V_2 \approx \sqrt{2(h_1 - h_2)} \qquad \text{(if } V_1 \ll V_2 \text{)}$$

Where $(h_1 - h_2)$ is enthalpy drop across the nozzle

Again we know, Tds = dh - νdp

For the isentropic flow, dh = νdp

$$\text{or,} \quad \int_1^2 dh = \int_1^2 v dp$$

$$\text{or,} \quad (h_1 - h_2) = -\int_1^2 v dp \qquad (20.1)$$

Assuming that the pressure and volume of steam during expansion obey the law pν^n = constant, where n is the isentropic index

$$-\int_1^2 v dp = -\int_1^2 (p_1 v_1{}^n)^{\frac{1}{n}} p^{-\frac{1}{n}} dp = -\int_1^2 (p_2 v_2^n)^{\frac{1}{n}} p^{-\frac{1}{n}} dp$$

$$= -\left\{ p_2^{\frac{1}{n}} v_2 \left[\frac{p^{1-\frac{1}{n}}}{1-\frac{1}{n}} \right]_1^2 \right\}$$

$$= -\frac{n}{n-1} \left\{ p_2^{\frac{1}{n}} v^2 \left[p_2^{\frac{n-1}{n}} - p_1^{\frac{n-1}{n}} \right] \right\}$$

$$= -\frac{n}{n-1} \{ p_2 v_2 - p_1^{1/2} v_1 p_1^{\frac{n-1}{n}} \}$$

$$= \frac{n}{n-1} \{ p_1 v_1 - p_2 v_2 \} \qquad (20.2)$$

Now, mass flow rate

$$\dot{m} = \rho_2 A_2 V_2$$

$$\frac{\dot{m}}{A_2} = \rho_2 V_2 = \frac{V_2}{v_2}$$

Therefore, the mass flow rate at the exit of the nozzle

$$\frac{\dot{m}}{A_2} = \frac{1}{v_2} \sqrt{\frac{2n}{n-1} p_1 v_1 \{1 - (\frac{p_2}{p_1})^{\frac{n-1}{n}}\}}$$

$$= \sqrt{\frac{2n}{n-1} \frac{p_1}{v_1} [(\frac{p_2}{p_1})^{\frac{2}{n}} - (\frac{p_2}{p_1})^{\frac{n+1}{n}}\}} \qquad (20.3)$$

The exit pressure, p_2 determines the $\dot{m}$ for a given inlet condition. The mass flow rate is maximum when,

$$\frac{d}{dy}[y^{\frac{2}{n}} - y^{\frac{n+1}{n}}] = 0 \quad ; \quad y = \frac{p_2}{p_1}$$

$$y = \left[\frac{2}{n+1}\right]^{\frac{n}{n-1}}$$

For maximum $\dot{m}$,

$$\frac{p_{or}}{p_1} = \frac{p^*}{p_1} = \frac{p_2}{p_1} = \left[\frac{2}{n+1}\right]^{\frac{n}{n-1}} \qquad (20.4)$$

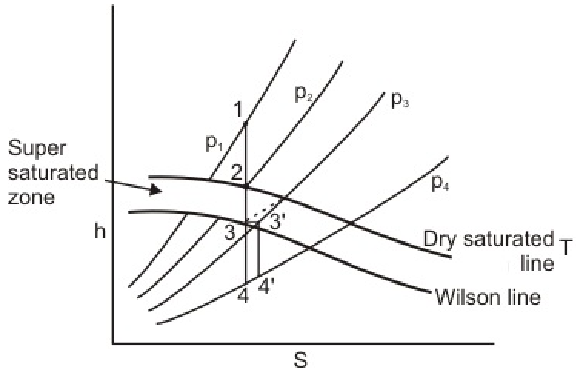
n = $\gamma = 1.4$,	for diatomic gases
$= 1.3$,	for super saturated steam
$= 1.135$,	for dry saturated steam
$= 1.035 + 0.1x$,	for wet steam with dryness fraction x

For , $n = 1.4$, $p^* = 0.528 p_1$ (50%drop in inlet pressure)

$n = 1.3$, $p^* = 0.546 p_1$

If we compare this with the results of sonic properties, as described in the earlier section, we shall observe that the critical pressure occurs at the throat for Ma = 1. The critical pressure ratio is defined as the ratio of pressure at the throat to the inlet pressure, for checked flow when Ma = 1

Steam Nozzles



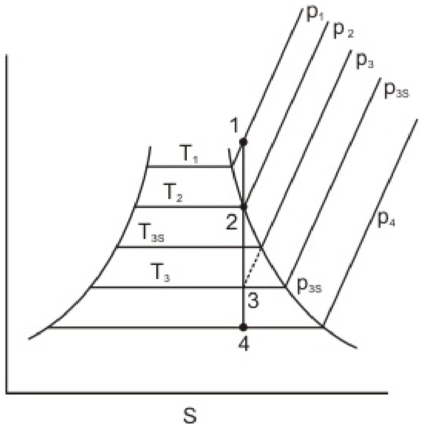


Figure 21.1 Super Saturated Expansion of Steam in a Nozzle

- The process 1-2 is the isentropic expansion. The change of phase will begin to occur at point 2
- vapour continues to expand in a dry state
- Steam remains in this unnatural superheated state untit its density is about eight times that of the saturated vapour density at the same pressure
- When this limit is reached, the steam will suddenly condense
- Point 3 is achieved by extension of the curvature of constant pressure line p_3 from the superheated region which strikes the vertical expansion line at 3 and through which Wilson line also passes. The point 3 corresponds to a metastable equilibrium state of the vapour.
- The process 2-3 shows expansion under super-saturation condition which is not in thermal equilibrium
- It is also called under cooling
- At any pressure between p_2 and p_3 i.e., within the superheated zone, the temperature of the vapous is lower than the saturation temperature corresponding to that pressure
- Since at 3, the limit of supersaturation is reached, the steam will now condense instantaneously to its normal state at the constant pressure, and constant enthalpy which is shown by the horizontal line $33'$ where $3'$ is on normal wet area pressure line of the same pressure p_3.
- $3' - 4'$ is again isentropic, expansion in thermal equilibrium.
- To be noted that 4 and $4'$ are on the same pressure line.

Thus the effect of supersaturation is to reduce the enthalpy drop slightly during the expansion and consequently a corresponding reduction in final velocity. The final dryness fraction and entropy are also increased and the measured discharge is greater than that theoretically calculated.

Degree of super heat = $\dfrac{p_3}{p_{3s}}$

p_3 = limiting saturation pressure

P_{3s} = saturation pressure at temperature T_3 shown on T-s diagram

degree of undercooling - T_{3s} - T_3

T_{3s} is the saturation temperature at p_3

T_3 = Supersaturated steam temperature at point 3 which is the limit of supersaturation.

$$\frac{\dot{m}}{A_2} = \sqrt{\frac{2n}{n-1}\cdot\frac{p_1}{v_1}\left[\left(\frac{2}{n+1}\right)^{\frac{n}{n-1}\cdot\frac{2}{n}} - \left(\frac{2}{n+1}\right)^{\frac{n}{n-1}\cdot\frac{n+1}{n}}\right]} \qquad (21.1)$$

$$\frac{\dot{m}}{A_2} = \sqrt{\frac{2n}{n-1}\cdot\frac{p_1}{v_1}\left[\left(\frac{2}{n+1}\right)^{\frac{2}{n-1}} - \left(\frac{2}{n+1}\right)^{\frac{n+1}{n-1}}\right]} \qquad (21.2)$$

Supersaturated vapour behaves like supersaturated steam and the index to expansion, $n = 1.3$

STEAM TURBINES

Turbines

- We shall consider steam as the working fluid
- Single stage or Multistage
- Axial or Radial turbines
- Atmospheric discharge or discharge below atmosphere in condenser
- Impulse/and Reaction turbine

Impulse Turbines

Impulse turbines (single-rotor or multirotor) are simple stages of the turbines. Here the impulse blades are attached to the shaft. Impulse blades can be recognized by their shape. They are usually symmetrical and have entrance and exit angles respectively, around 20 ° . Because they are usually used in the entrance high-pressure stages of a steam turbine, when the specific volume of steam is low and requires much smaller flow than at lower pressures, the impulse blades are short and have constant cross sections.

The Single-Stage Impulse Turbine

The *single-stage impulse turbine* is also called the *de Laval turbine* after its inventor. The turbine consists of a single rotor to which impulse blades are attached. The steam is fed through one or several convergent-divergent nozzles which do not extend completely around the circumference of the rotor, so that only part of the blades is impinged upon by the steam at any one time. The nozzles also allow governing of the turbine by shutting off one or more them.

The velocity diagram for a single-stage impulse has been shown in Fig. 22.1. Figure 22.2 shows the velocity diagram indicating the flow through the turbine blades.

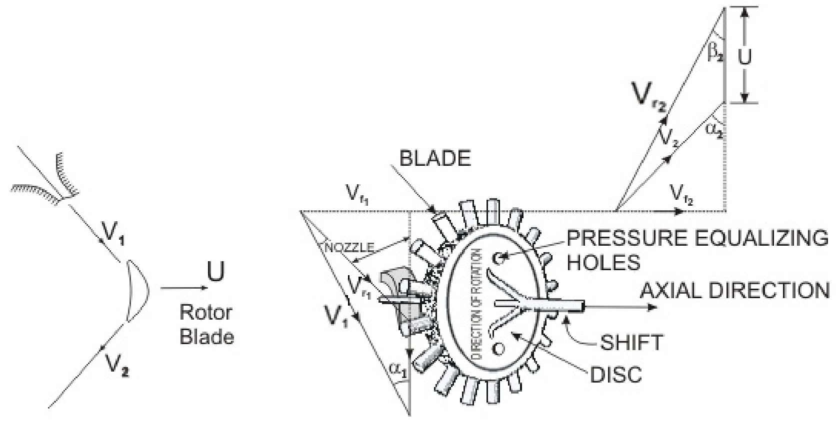


Figure 22.1 Schematic diagram of an Impulse Trubine

V_1 and V_2 = Inlet and outlet absolute velocity

V_{r_1} and V_{r_2} = Inlet and outlet relative velocity (Velocity relative to the rotor blades.)

U = mean blade speed

α_1 = nozzle angle, α_2 = absolute fluid angle at outlet

It is to be mentioned that all angles are with respect to the tangential velocity (in the direction of U)

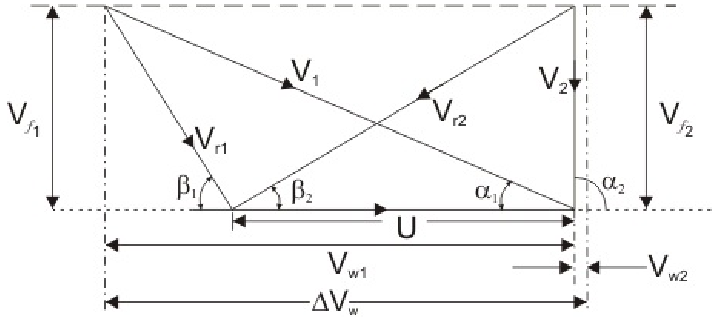


Figure 22.2 Velocity diagram of an Impulse Turbine

β_1 and β_2 = Inlet and outlet **blade angles**

Vw_1 and Vw_2 = Tangential or whirl component of absolute velocity at inlet and outlet

V_{f1} and V_{f2} = Axial component of velocity at inlet and outlet

Tangential force on a blade,

$$F_u = \dot{m}(V_{w1} - Vw2) \qquad (22.1)$$

(mass flow rate X change in velocity in tangential direction)

or,

$$F_u = \dot{m}\Delta V_w \qquad (22.2)$$

Power developed = $\dot{m}U\Delta V_w$ (22.3)

Blade efficiency or Diagram efficiency or Utilization factor is given by

$$\eta_b = \frac{\dot{m}\cdot U\cdot\Delta V_w}{m(V_1^2/2)} = \frac{\text{Workdone}}{\text{K.E supplied}}$$

or,

$$\eta_b = \frac{2U\Delta V_w}{V_1^2} \qquad (22.4)$$

$$\text{stage efficiency } = \eta_s = \frac{\textit{Work done by the rotor}}{\textit{Isentropic enthalpy drop}} \quad (23.1)$$

$$\eta_s = \frac{\dot m U \Delta V_w}{\dot m (\Delta H)_{isen}} = \frac{\dot m U \Delta V_w}{\dot m \left(\frac{V_1^2}{2}\right)} \cdot \frac{\dot m (V_1^2/2)}{\dot m (\Delta H)_{isen}} \quad (23.2)$$

or,

$$\text{or, } \eta_s = \eta_b \times \eta_n \qquad [\eta_n = \textit{Nozzle efficiency}] \quad (23.3)$$

Optimum blade speed of a single stage turbine

$$\Delta V_w = V_{r1}\cos\beta_1 + V_{r2}\cos\beta_2$$

$$= V_{r1}\cos\beta_1 + \left(1 + \frac{V_{r2}}{V_{r1}} \cdot \frac{\cos\beta_2}{\cos\beta_1}\right)$$

$$= (V_1 \cos\alpha_1 - U) + (1 + kc) \quad (23.4)$$

where, $k = (V_{r2}/V_{r1})$ = friction coefficient

$$c = (\cos\beta_2/\cos\beta_1)$$

$$\eta_b = \frac{2U\Delta V_w}{V_1^2} = 2\frac{U}{V_1}\left(\cos\alpha_1 - \frac{U}{V_1}\right)(1+kc)$$

$$\rho = \frac{U}{V_1} = \frac{\text{Blade speed}}{\text{Fluid velocity at the blade inlet}} = \text{Blade speed ratio} \quad (23.5)$$

η_b is maximum when $\frac{d\eta_b}{d\rho} = 0$ also $\frac{d^2\eta_b}{d\rho} = -4(1+kc)$

$$\text{or, } \quad \frac{d}{d\rho}\{2(\rho\cos\alpha_1 - \rho^2)(1+kc)\} = 0$$

$$\text{or, } \quad \rho = \frac{\cos\alpha_1}{2} \quad (23.6)$$

α_1 is of the order of 18^0 to 22^0

Now, $(\rho)_{opt} = \left(\frac{U}{V_1}\right)_{opt} = \frac{\cos\alpha_1}{2}$ (For single stage impulse turbine)

$\therefore$ The maximum value of blade efficiency

$$(\eta_b)_{\max} = 2(\rho\cos\alpha_1 - \rho^2)(1+kc)$$

$$= \frac{\cos^2\alpha_1}{2}(1+kc) \quad (23.7)$$

For equiangular blades,

$$(\eta_b)_{\max} = \frac{\cos^2\alpha_1}{2}(1+k) \quad (23.8)$$

If the friction over blade surface is neglected

$$(\eta_b)_{\max} = \cos^2\alpha_1 \quad (23.9)$$

Compounding in Impulse Turbine

If high velocity of steam is allowed to flow through one row of moving blades, it produces a rotor speed of about 30000 rpm which is too high for practical use.

It is therefore essential to incorporate some improvements for practical use and also to achieve high performance. This is possible by making use of more than one set of nozzles, and rotors, in a series, keyed to the shaft so that either the steam pressure or the jet velocity is absorbed by the turbine in stages. This is called compounding. Two types of compounding can be accomplished: (a) velocity compounding and (b) pressure compounding

Either of the above methods or both in combination are used to reduce the high rotational speed of the single stage turbine.

The Velocity - Compounding of the Impulse Turbine

The velocity-compounded impulse turbine was first proposed by C.G. Curtis to solve the problems of a single-stage impulse turbine for use with high pressure and temperature steam. The *Curtis stage* turbine, as it came to be called, is composed of one stage of nozzles as the single-stage turbine, followed by two rows of moving blades instead of one. These two rows are separated by one row of fixed blades attached to the turbine stator, which has the function of redirecting the steam leaving the first row of moving blades to the second row of moving blades. A Curtis stage impulse turbine is shown in Fig. 23.1 with schematic pressure and absolute steam-velocity changes through the stage. In the Curtis stage, the total enthalpy drop and hence pressure drop occur in the nozzles so that the pressure remains constant in all three rows of blades.

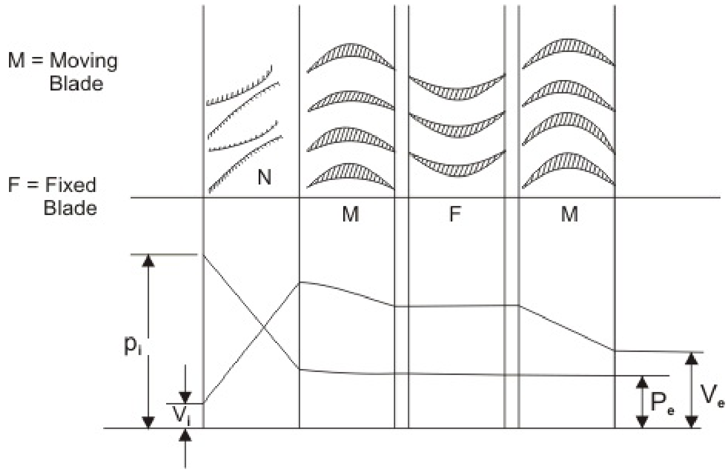


Figure 23.1 Velocity Compounding arrangement

Velocity is absorbed in two stages. In fixed (static) blade passage both pressure and velocity remain constant. Fixed blades are also called guide vanes. Velocity compounded stage is also called **Curtis stage.** The velocity diagram of the velocity-compound Impulse turbine is shown in Figure 23.2.

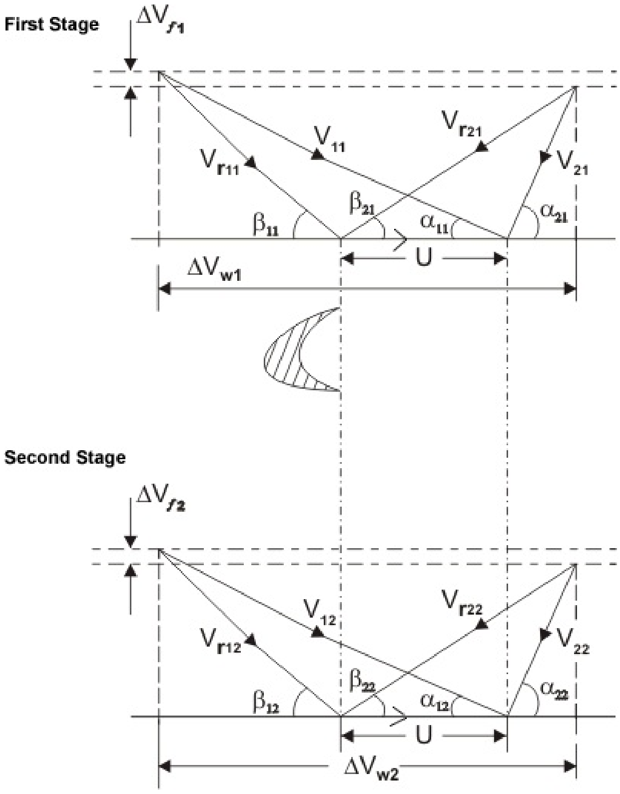


Figure 23.2 Velocity diagrams for the Velocity-Compounded Impulse turbine

The fixed blades are used to guide the outlet steam/gas from the previous stage in such a manner so as to smooth entry at the next stage is ensured.

K, the blade velocity coefficient may be different in each row of blades

$$\text{Work done} = \dot{m}.U\,(\Delta V_{w1} + \Delta V_{w2}) \qquad (23.10)$$

$$\text{End thrust} = \dot{m}\,(\Delta V_{f1} + \Delta V_{f2}) \qquad (23.11)$$

The optimum velocity ratio will depend on number of stages and is given by $P_{opt} = \dfrac{\cos\alpha_{11}}{2n}$

- Work is not uniformly distributed (1st >2nd)
- The fist stage in a large (power plant) turbine is velocity or pressure compounded impulse stage.

Pressure Compounding or Rateau Staging

The Pressure - Compounded Impulse Turbine

To alleviate the problem of high blade velocity in the single-stage impulse turbine, the total enthalpy drop through the nozzles of that turbine are simply divided up, essentially in an equal manner, among many single-stage impulse turbines in series (Figure 24.1). Such a turbine is called a *Rateau turbine* , after its inventor. Thus the inlet steam velocities to each stage are essentially equal and due to a reduced Δh.

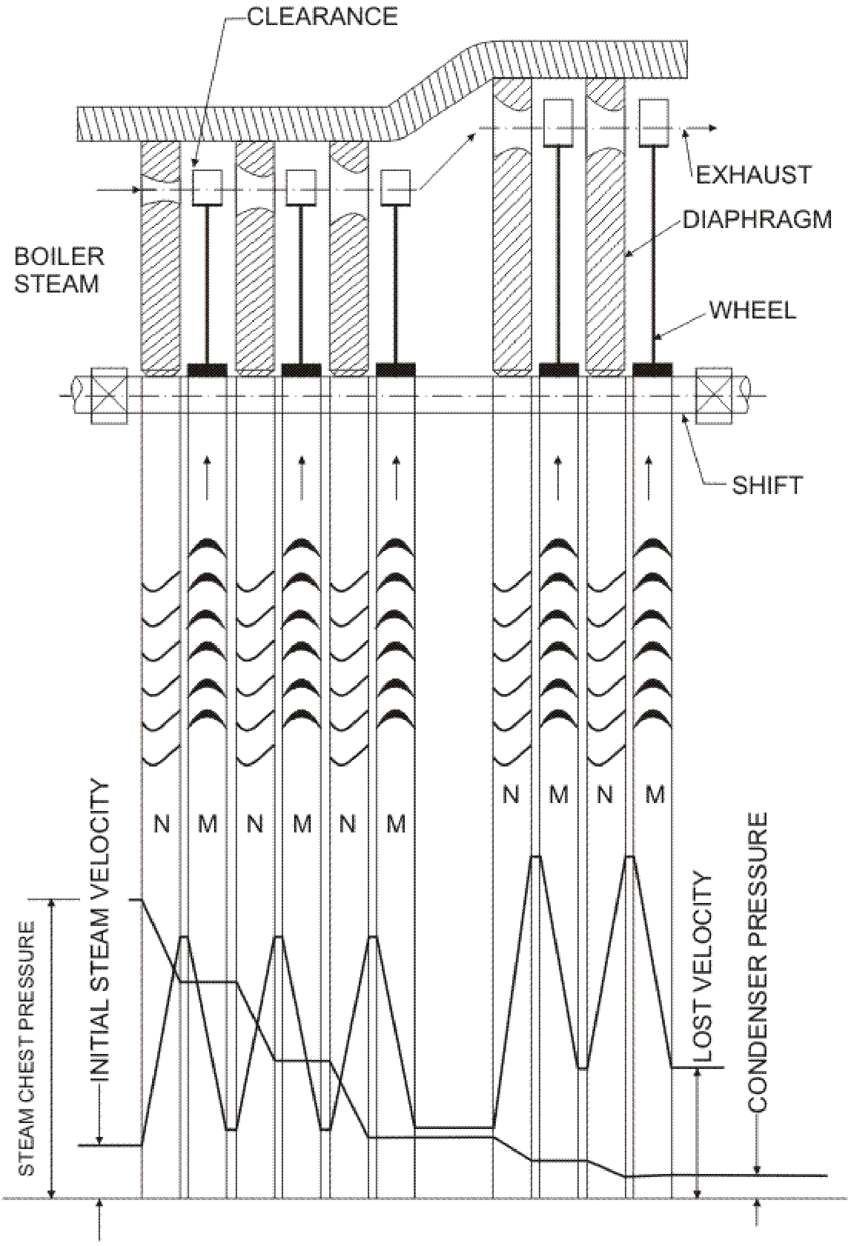


Figure 24.1 Pressure-Compounded Impulse Turbine

Pressure drop - takes place in more than one row of nozzles and the increase in kinetic energy after each nozzle is held within limits. Usually convergent nozzles are used

We can write

$$\underbrace{\frac{V_1^2}{2}+h_1}_{exit} = \underbrace{\frac{V_2^2}{2}+h_2}_{inlet} \qquad (24.1)$$

$$\eta_N = \frac{V_1^2 - \varphi V_2^2}{2(\Delta h)_{isentropic}} \qquad (24.2)$$

where ϕ is carry over coefficient

Reaction Turbine

A ***reaction turbine***, therefore, is one that is constructed of rows of fixed and rows of moving blades.

The fixed blades act as nozzles. The moving blades move as a result of the impulse of steam received (caused by change in momentum) and also as a result of expansion and acceleration of the steam relative to them. In other words, they also act as nozzles. The enthalpy drop per stage of one row fixed and one row moving blades is divided among them, often equally. Thus a blade with a 50 percent degree of reaction, or a 50 percent reaction stage, is one in which half the enthalpy drop of the stage occurs in the fixed blades and half in the moving blades. The pressure drops will not be equal, however. They are greater for the fixed blades and greater for the high-pressure than the low-pressure stages.

The moving blades of a reaction turbine are easily distinguishable from those of an impulse turbine in that they are not symmetrical and, because they act partly as nozzles, have a shape similar to that of the fixed blades, although curved in the opposite direction. The schematic pressure line (Fig. 24.2) shows that pressure continuously drops through all rows of blades, fixed and moving. The absolute steam velocity changes within each stage as shown and repeats from stage to stage. Figure 24.3 shows a typical velocity diagram for the reaction stage.

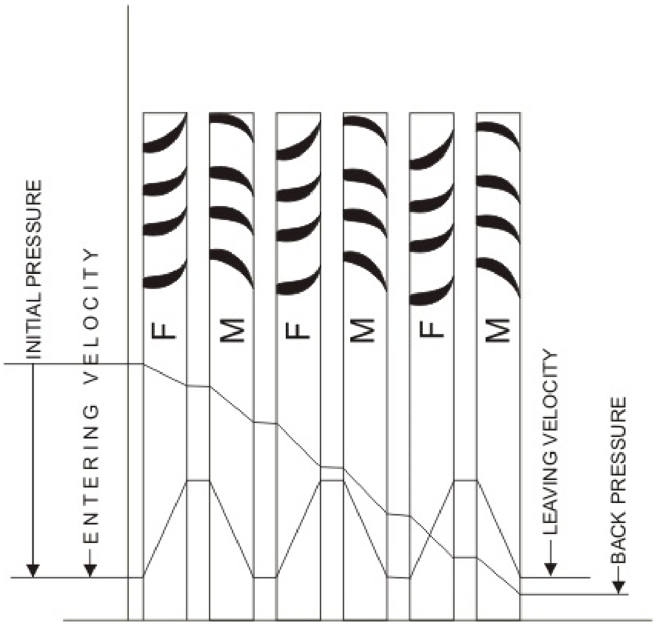


Figure 24.2 Three stages of reaction turbine indicating pressure and velocity distribution

Pressure and enthalpy drop both in the fixed blade or **stator** and in the moving blade or **Rotor**

$$\text{Degree of Reaction} = \frac{\textit{Enthalpy drop in Rotor}}{\textit{Enthalpy drop in Stage}}$$

$$\text{or,} \qquad R = \frac{h_1 - h_2}{h_0 - h_1} \qquad (24.3)$$

A very widely used design has half degree of reaction or 50% reaction and this is known as Parson's Turbine. This consists of symmetrical stator and rotor blades.

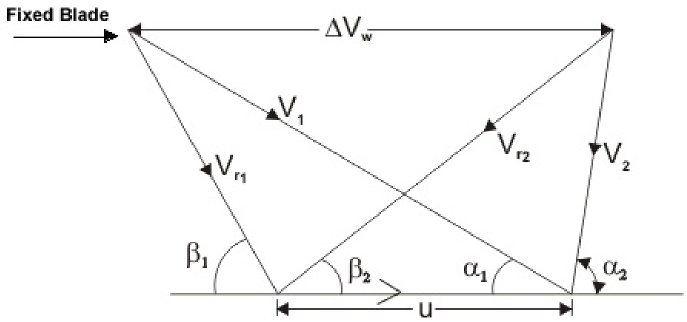


Figure 24.3 The velocity diagram of reaction blading

The velocity triangles are symmetrical and we have

$$\alpha_1 = \beta_2 \quad , \quad \beta_1 = \alpha_2$$

$$V_1 = V_{r2} \quad , \quad V_{r1} = V_2$$

Energy input per stage (unit mass flow per second)

$$E = \frac{V_1^2}{2} + \frac{V_{r2}^2 - V_{r1}^2}{2}$$

$$E = V_1^2 - \frac{V_{r1}^2}{2} \qquad (24.4)$$

$$E = V_1^2 - \frac{V_1^2}{2} - \frac{U^2}{2} + \frac{2V_1 U \cos\alpha_1}{2}$$

$$E = (V_1^2 - U^2 + 2V_1 U \cos\alpha_1)/2 \qquad (24.5)$$

From the inlet velocity triangle we have,

$$V_{r1}^2 = V_1^2 - U^2 - 2V_1 U \cos\alpha_1$$

Work done (for unit mass flow per second) $= W = U\Delta V_W$

$$= U(2V_1 \cos\alpha_1 - U) \qquad (24.6)$$

Therefore, the Blade efficiency

$$= \eta_b = \frac{2U(2V_1 \cos\alpha_1 - U)}{V_1^2 - U^2 + 2V_1 U \cos\alpha_1} \qquad (24.7)$$

Reaction Turbine, Continued

Put $\rho = \frac{U}{V_1}$ then

$$\eta_b = \frac{2\rho(2\cos\alpha_1 - P)}{1-\rho^2 + 2\rho\cos\alpha_1} \qquad (25.1)$$

For the maximum efficiency $\frac{d\eta_b}{d\rho} = 0$ and we get

$$(1-\rho^2 + 2\rho\cos\alpha_1)(4\cos\alpha_1 - 4\rho) - 2\rho(2\cos\alpha_1 - \rho)(-2\rho + 2\cos\alpha_1) = 0 \quad (25.2)$$

from which finally it yields

$$\rho_{opt} = \left(\frac{U}{V_1}\right)_{opt} = \cos\alpha_1 \qquad (25.3)$$

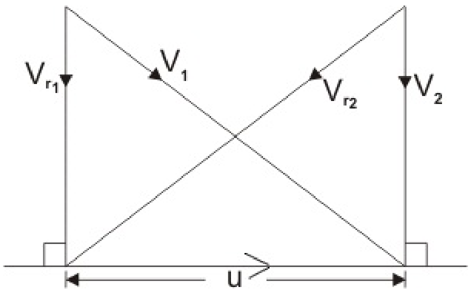


Figure 25.1 Velocity diagram for maximum efficiency

Absolute velocity of the outlet at this stage is axial (see figure 25.1). In this case, the energy transfer

$$E = U\Delta V_W = U^2 \qquad (25.4)$$

$(\eta_b)_{maximum}$ can be found out by putting the value of $\rho = \cos\alpha_1$ in the expression for blade efficiency

$$(\eta_b)_{\max} = \frac{2\cos^2\alpha_1}{1+\cos^2\alpha_1} \qquad (25.5)$$

$$(\eta_b)_{impulse} = \cos^2\alpha_1 \qquad (25.6)$$

η is greater in reaction turbine. Energy input per stage is less, so there are more number of stages.

Stage Efficiency and Reheat factor

The Thermodynamic effect on the turbine efficiency can be best understood by considering a number of stages between two stages 1 and 2 as shown in Figure 25.2

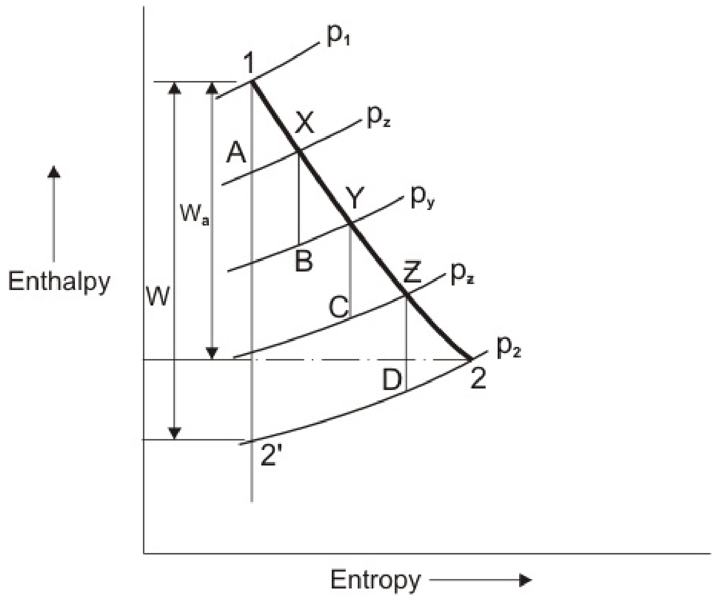


Figure 25.2 Different stage of a steam turbine

The total expansion is divided into four stages of the same efficiency (η_s) and pressure ratio.

$$\frac{P_1}{P_x} = \frac{P_x}{P_y} = \frac{P_y}{P_z} = \frac{P_z}{P_2} \quad (25.7)$$

The overall efficiency of expansion is η_o. The actual work during the expansion from 1 to 2 is

$$Wa = \eta_o W$$

or, $$\eta_o = \frac{Wa}{W} = \frac{actual\ enthalpy\ drop\ (1-2)}{isentropic\ heat\ drop\ (1-2')} \quad (25.8)$$

$$\text{Reheat factor (R.F.)} = \frac{Cumulative\ \ enthalpy\ drop\ (isentropic)}{Isentropic\ \ enthalpy\ \ drop\ \ (overall)}$$

or, $$R.F = \frac{\Delta h_{1A} + \Delta h_{xB} + \Delta h_{yc} + \Delta h_{zD}}{\Delta h_{12}} \quad (25.9)$$

R.F is 1.03 to 1.04

If η_s remains same for all the stages or η_s is the mean stage efficiency.

$$\eta_s = \frac{\Delta h_{1x}}{\Delta h_{1A}} = \frac{\Delta h_{xy}}{\Delta h_{xB}} = \frac{\Delta h_{yz}}{\Delta h_{yc}} = \frac{\Delta h_{z2}}{\Delta h_{zD}} \quad (25.10)$$

or, $$\eta_s = \frac{\Delta h_{1x} + \Delta h_{xy} + \Delta h_{yz} + \Delta h_{z2}}{\Delta h_{1A} + \Delta h_{xB} + \Delta h_{yc} + \Delta h_{zD}} \quad (25.11)$$

$$= \frac{actual\ \ enthalpy\ \ drop}{Cumulative\ enthalpy\ drop\ (isentropic)}$$

We can see:

$$\eta_0 = \eta_s \times R.F \quad (25.12)$$

This makes the overall efficiency of the turbine greater than the individual stage efficiency.

The effect depicted by Eqn (25.12) is due to the thermodynamic effect called "reheat". This does not imply any heat transfer to the stages from outside. It is merely the reappearance of stage losses an increased enthalpy during the constant pressure heating (or reheating) processes AX, BY, CZ and D2.

Exercise Problems (for Steam Turbines)

Q1. The adiabatic enthalpy drop in a given stage of a multi-stage impulse turbine is 22.1 KJ/kg of steam. The nozzle outlet angle is 16^o, and the efficiency of the nozzle, defined as the ratio of the actual gain of kinetic energy in the nozzle to adiabatic heat drop, is 92%. The mean diameter of the blades is 1473.2 mm and the revolution per minutes is 1500. Given that the carry over factor ϕ is 0.88, and that the blades are equiangular (the blade velocity coefficient is 0.87). Calculate the steam velocity at the outlet from nozzles, blade angles, and gross stage efficiency.

Q2. The following particulars relate to a two row velocity compounded impulse wheel which forms a first stage of a combination turbine.

Steam velocity at nozzle outlet = 579.12m/s

Mean blade velocity = 115.82m/s

Nozzle outlet angle = 16^o

Outlet angle first row of moving blades = 18^o

Outlet angle fixed guide blades = 22^o

Outlet angle, second row of moving blades = 36^o

Steam flow rate = 2.4 kg/s

The ratio of the relative velocity at outlet to that at inlet is 0.84 for all blades. Determine for each row of moving blades the following

- The velocity of whirl
- The tangential thrust on blades
- The axial thrust on the blades
- The power developed

What is the efficiency of the wheel as a whole?

Q3. A velocity compounded impulse wheel has two rows of moving blades with a mean diameter of 711.2 mm. The speed of rotation is 3000rpm, the nozzle angle is 16^o and the estimated steam velocity at the nozzle outlet is 554.73m/s. The mass flow rate of the steam passing through the blades is 5.07 kg/s.

Assuming that the energy loss in each row of blades (moving and fixed) is 24% of the kinetic energy of the steam entering the blades and referred to as the relative velocity, and that the outlet angles of the blades are:

(1) first row of moving blades 18^o,

(2) intermediate guide blade 22^o,

(3) second row of moving blades is 38^0 , draw the diagram of relative velocities and derive the following.

- Blade inlet angles
- Power developed in each row of blades
- Efficiency of the wheel as a whole

Q4. The following particulars refer to a stage of an impulse-reaction turbine.

Outlet angle of fixed blades = 20^o

Outlet angle of moving blades = 30^o

Radial height of fixed blades =100mm

Radial height of moving blades =100mm

Mean blade velocity = 138m/s

Ratio of blade speed to steam speed = 0.625

Specific volume of steam at fixed blade outlet =1.235 m^3 / kg

Specific volume of steam at moving blade outlet =1.305 m^3 / kg

Calculation the degree of reaction, the adiabatic heat drop in pair of blade rings, and the gross stage efficiency, given the following coefficients which may be assumed to be the same in both fixed and moving blades : $\eta_m = 0.9, \phi = 0.86.$

Q5. Steam flows into the nozzles of a turbine stage from the blades of preceding stage with a velocity of 100m/s and issues from the nozzles with a velocity of 325 m/s at angle of 20^o to the wheel plane. Calculate the gross stage efficiency for the following data:

Mean blade velocity=180m/s

Expansion efficiency for nozzles and blades = 0.9

Carry over factor for nozzles and blades = 0.9

Degree of reaction = 0.26

Blade outlet angle = 28^o

MODULE 5

Hydraulic Turbines (Pelton Wheel, Francis Turbine and Kaplan Turbine)

IMPULSE TURBINE

Figure 26.1 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 26.1. 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 26.2). 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.

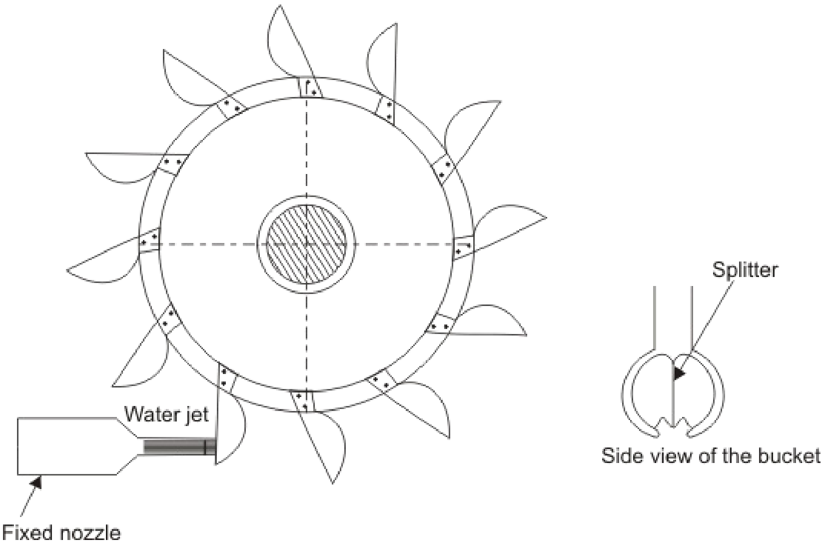


Figure 26.2 A Pelton wheel

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 180^o. In practice, however, the deflection is limited to about 165^o so that the water leaving a bucket may not hit the back of the following bucket. Therefore, the camber angle of the buckets is made as $165^o(\theta = 165^o)$. Figure(26.3a)

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 V_1 with which it strikes the bucket is given by

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

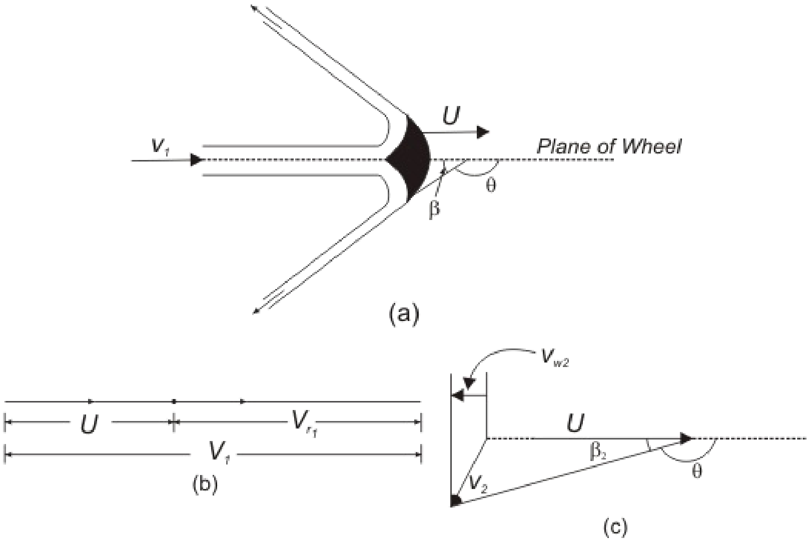


Figure 26.3 **(a)Flow along the bucket of a pelton wheel**
(b) Inlet velocity triangle
(c)Outlet velocity triangle

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)

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

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

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

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

$$E/m = [V_{w_1} - V_{w_2}]\, U$$

$$= [V_{r_1} + V_{r_2} \cos \beta_2]\, U \qquad (26.1)$$

(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.

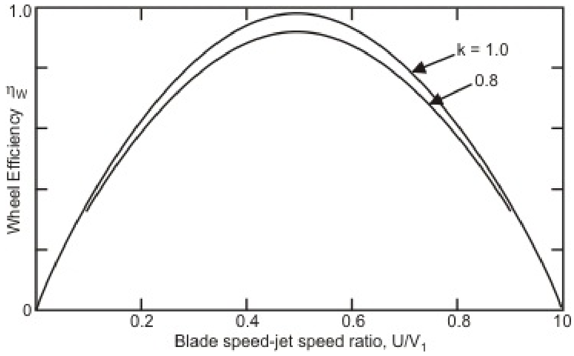


Figure 27.1 Theoretical variation of wheel efficiency for a Pelton turbine with blade speed to jet speed ratio for different values of k

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.

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

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

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

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 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.

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

Govering 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 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.

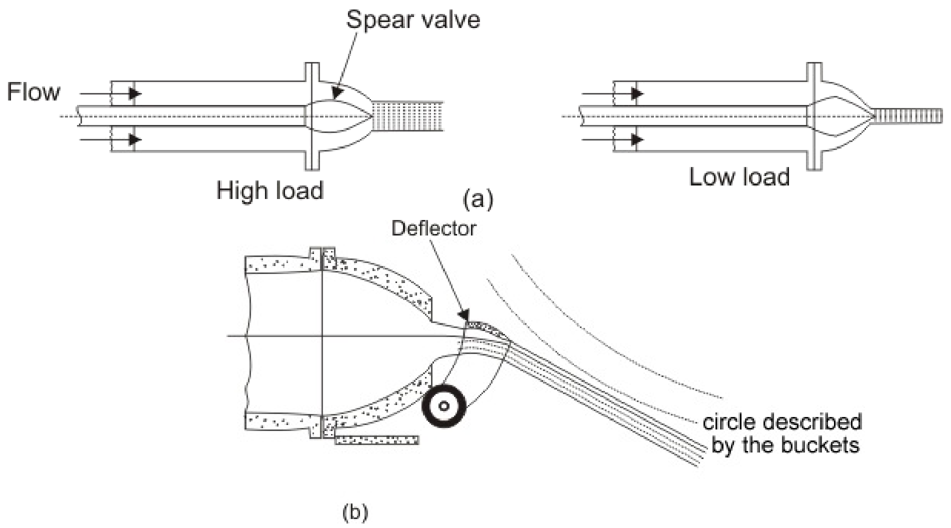


Figure 27.2 **(a) Spear valve to alter jet area in a Pelton wheel**
(b) Jet deflected from bucket

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

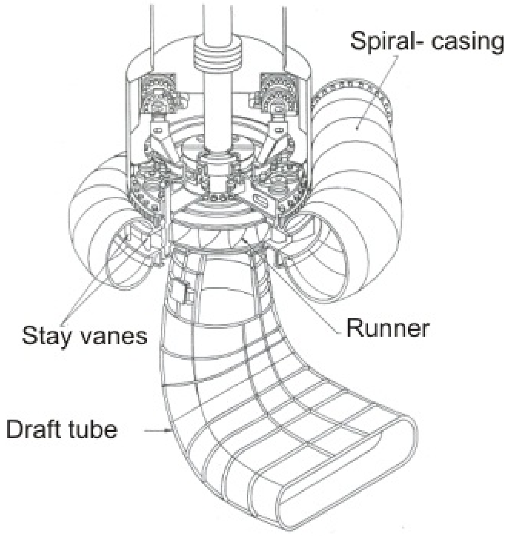


Figure 28.1 A Francis turbine

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.

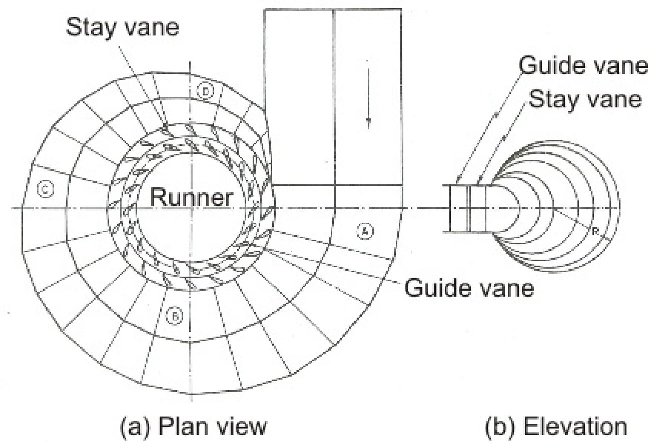


Figure 28.2 Spiral Casing

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

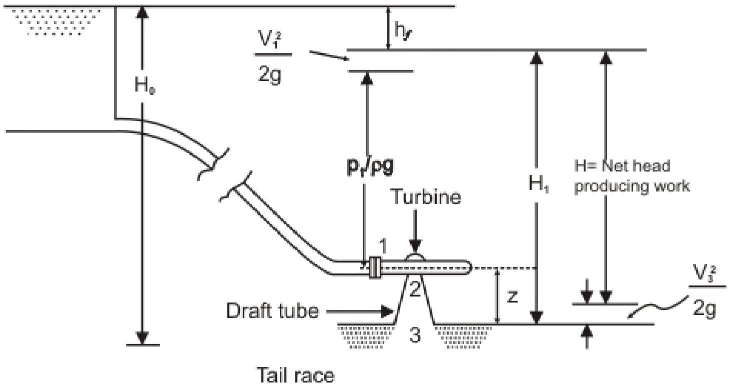


Figure 28.3 Head across a reaction turbine

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

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

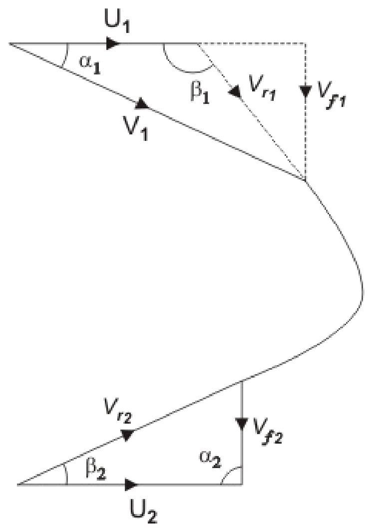


Figure 29.1 Velocity triangle for a Francis runner

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 $V_{f_1} = V_{f_2}$, η_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 $V_1^2 - V_{f2}^2 = V_1^2 - V_{f1}^2 = V_{f1}^2 \cot^2 \alpha_1$]

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,

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

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 α_1, β_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.

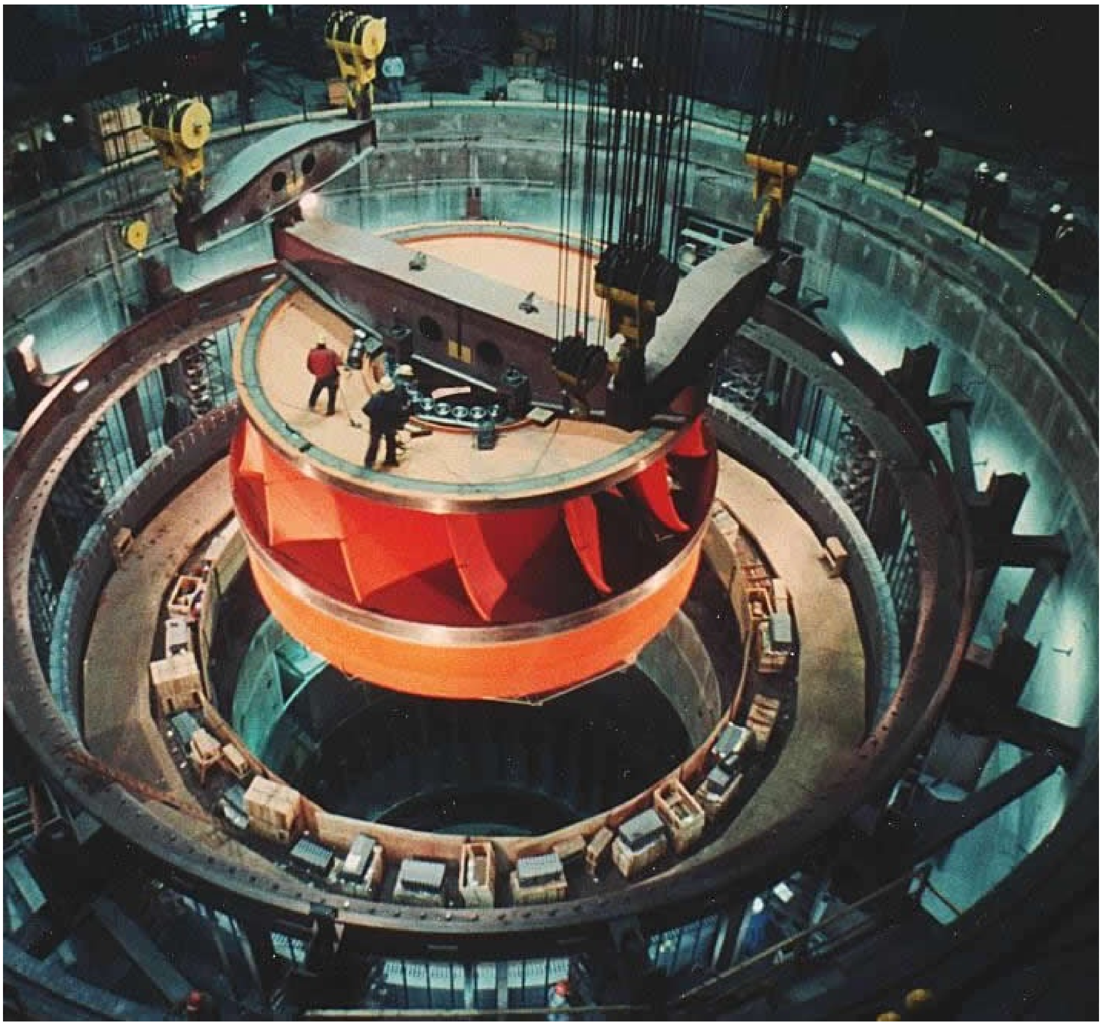


Figure 29.2 Installation of a Francis Turbine

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)

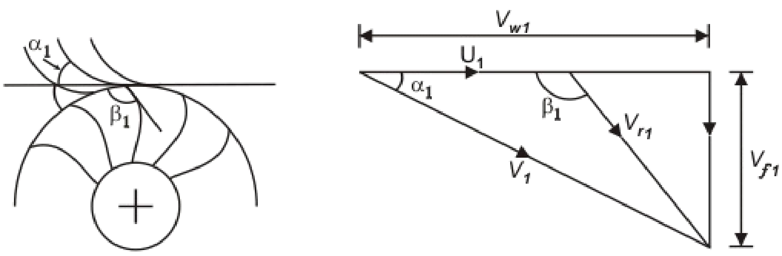
Figure 30.1 A typical Kaplan Turbine

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

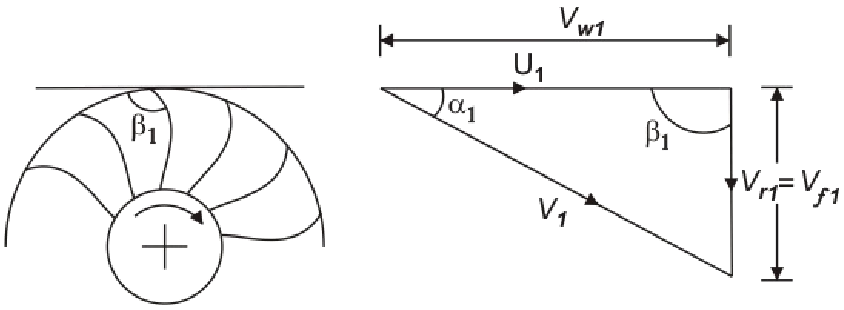
Figure 30.2 shows in stages the change in the shape of a Francis runner with the variation of specific speed. The first three types [Fig. 30.2 (a), (b) and (c)] have, in order. The Francis runner (radial flow runner) at low, normal and high specific speeds. As the specific speed increases, discharge 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.

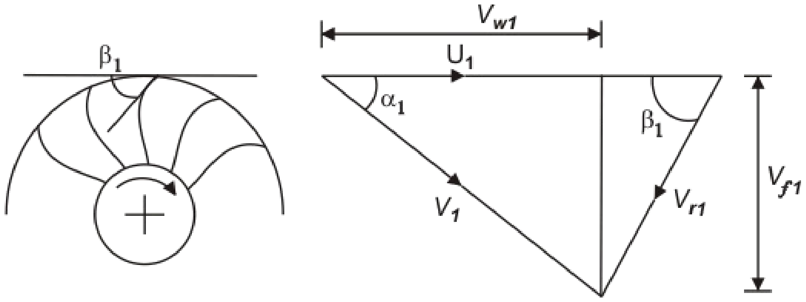
In case of a Kaplan runner, the flow at inlet is in axial and tangential directions. Therefore, the inlet velocity triangle in this case (Figure 30.2e) is shown in a place containing the axial and tangential directions, and hence the flow velocity V_{f1} represents the axial component of velocity V_a .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

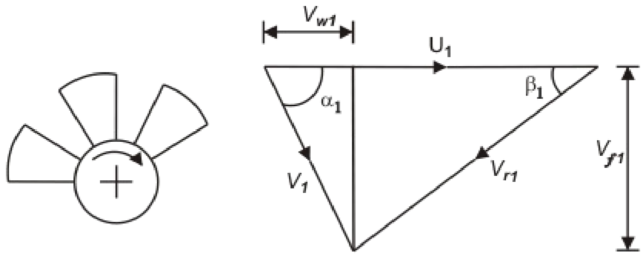


(b) Francis runner for normal specific speeds

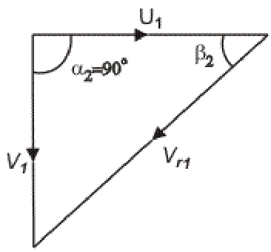


(c) Francis runner for high specific speeds

(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.

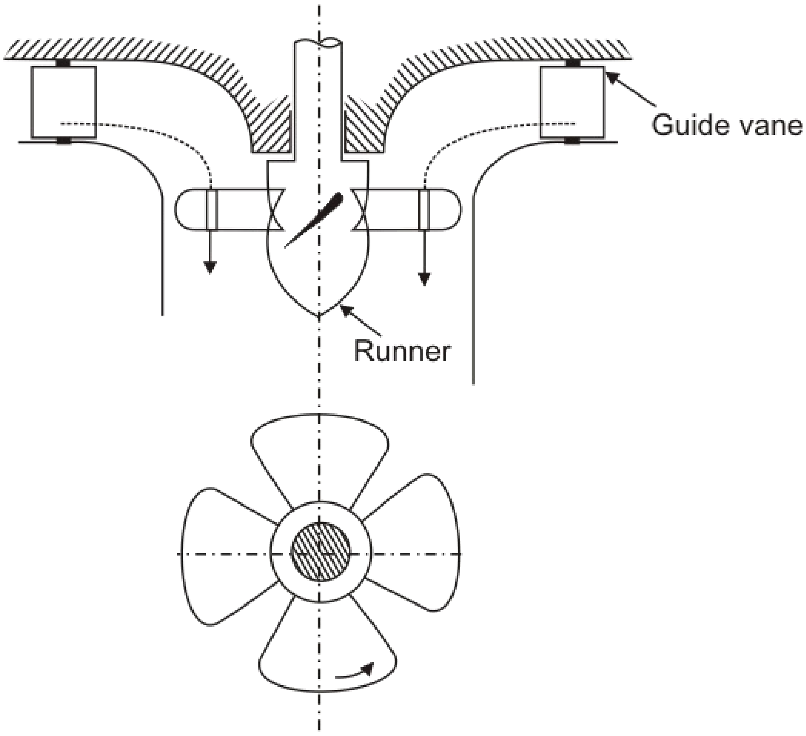


Fig. 30.3 A propeller of Kaplan turbine

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.

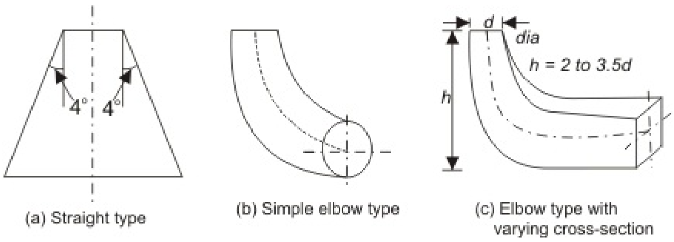


Figure 30.4 Different types of draft tubes

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

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

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's Cavitation 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.

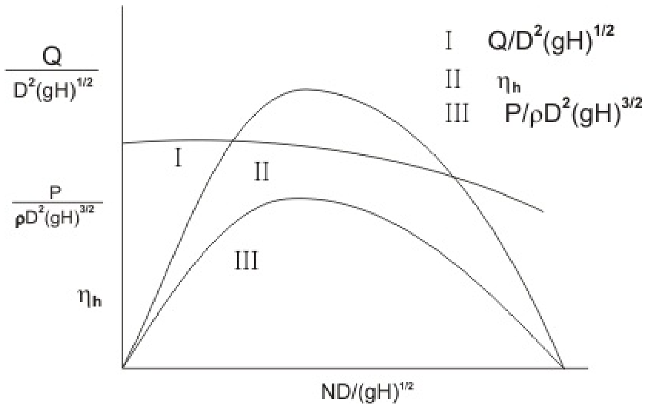


Figure 31.1 performance characteristics of a reaction turbine (in dimensionless parameters)

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

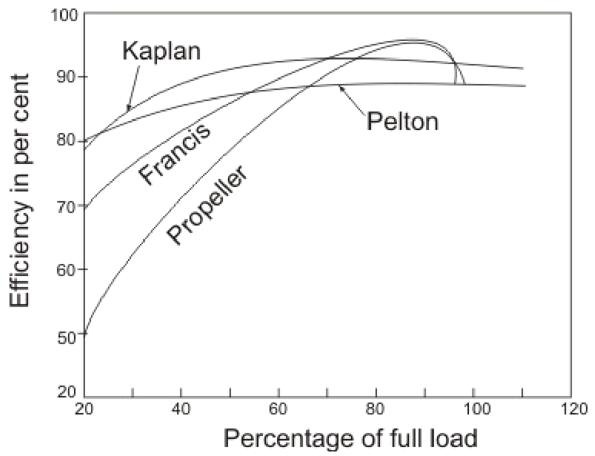


Figure 31.2 Variation of efficiency with load

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).

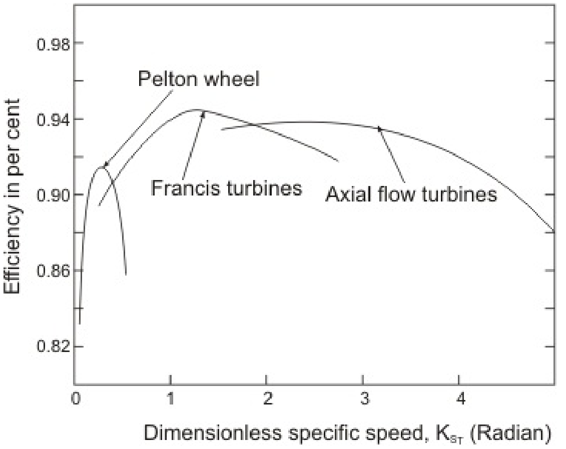


Figure 32.1 Variation of efficiency with specific speed for hydraulic turbines

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

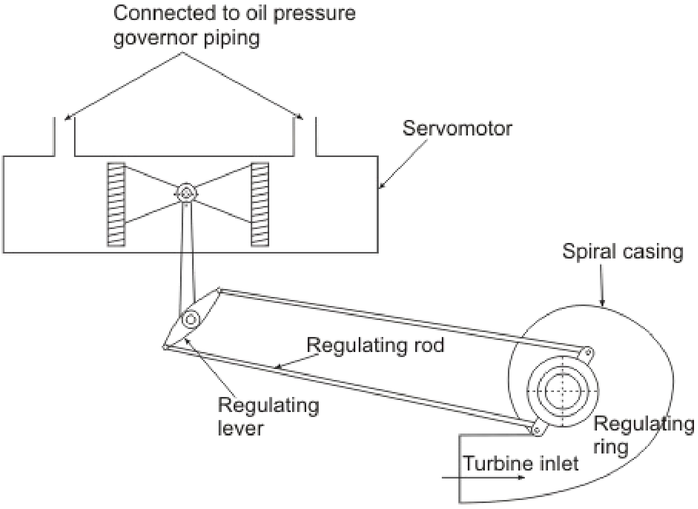


Figure 32.2 Governing of reaction turbine

Bulb Turbine

The bulb turbine is a reaction turbine of Kaplan type which is used for extremely low heads. The characteristic feature of this turbine is that the turbine components as well as the generator are housed inside a bulb, from which the name is developed. The main difference from the Kaplan turbine is that the water flows in a mixed axial-radial direction into the guide vane cascade and not through a scroll casing. The giude vane spindles are normally inclined to 60^0 in relation to the turbine shaft and thus results in a conical guide vane cascade contrary to other types of turbines. The runner of a bulb turbine may have different numbers of blades depending on the head and water flow. The bulb turbines have higher full-load efficiency and higher flow capacity as compared to Kaplan turbine. It has a relatively lower construction cost. The bulb turbines can be utilized to tap electrical power from the fast flowing rivers on the hills. Figure 32.3 shows the schematic of a Bulb Turbine Power Plant.

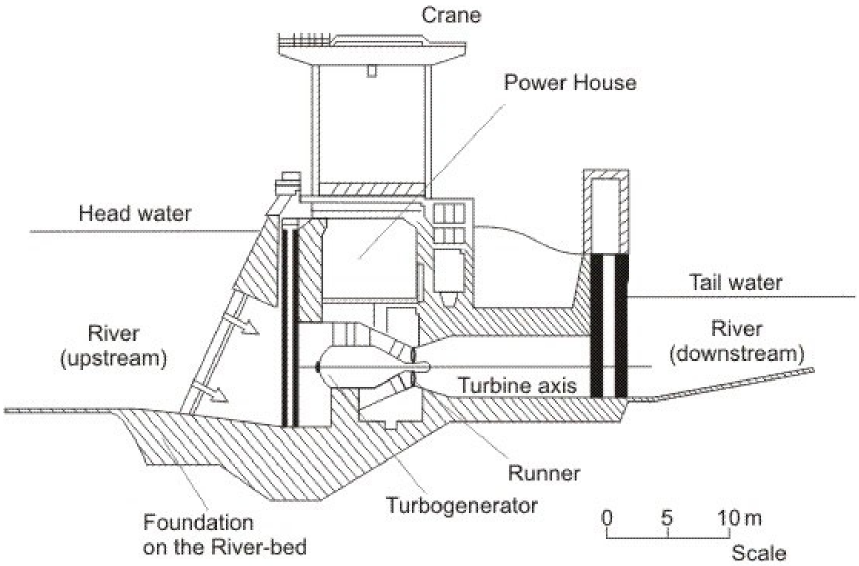


Figure 32.3 Schematic of Bulb Turbine Power Generating Station

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)

7) The following data refer to an elbow type draft tube:

Area of circular inlet = $25m^2$

Area of rectangular outlet = $116m^2$

Velocity of water at inlet to draft tube = 10 m/s

The frictional head loss in the draft tube equals to 10% of the inlet velocity head.

Elevation of inlet plane above tail race level = 0.6m

Determine:

a) Vacuum or negative head at inlet

b) Power thrown away in tail race

(Ans.4.95 m vac, 578kW)

8) Show that when vane angle at inlet of a Francis turbine is 90 o and the velocity of flow is constant, the hydraulic efficiency is given by $2/(2+\tan^2\alpha)$, where α is the guide blade angle.

9) A conical type draft tube attached to a Francis turbine has an inlet diameter of 3 m and its area at outlet is $20m^2$. The velocity of water at inlet, which is 5 m above tail race level, is 5 m/s. Assuming the loss in draft tube equals to 50% of velocity head at outlet, find (a) the pressure head at the top of the draft tube (b) the total head at the top of the draft tube taking tail race level as datum (c) power lost in draft tube.

(Ans. 6.03 m vac, 0.24m, 0.08m)

MODULE 6

Pumps

Next

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

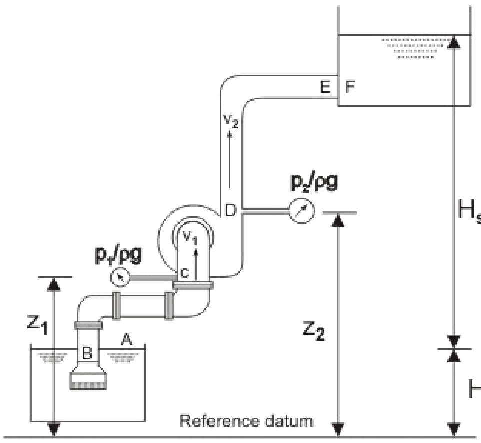


Figure 33.1 A general pumping system

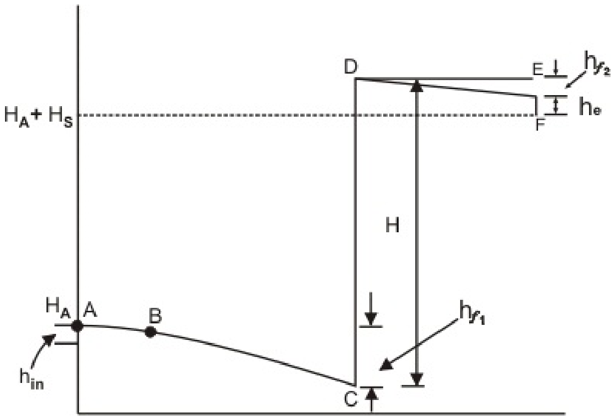


Figure 33.2 Change of head in a pumping system

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 *manometric head* . 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

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

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.

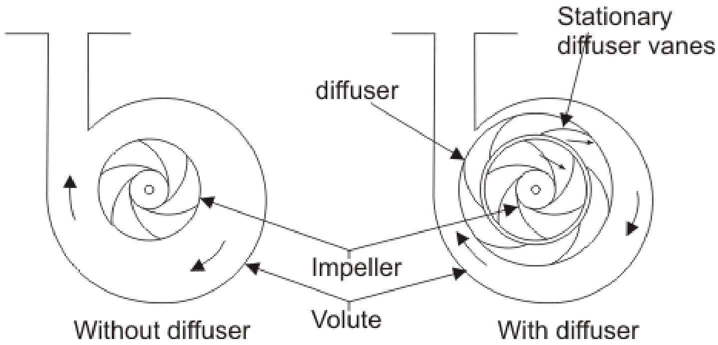
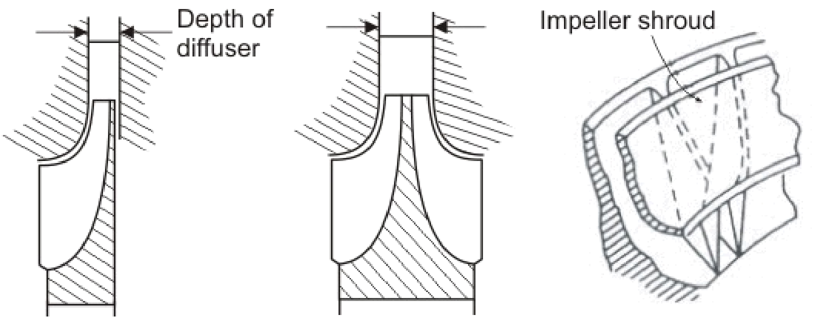


Figure 33.3 A centrifugal pump

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.

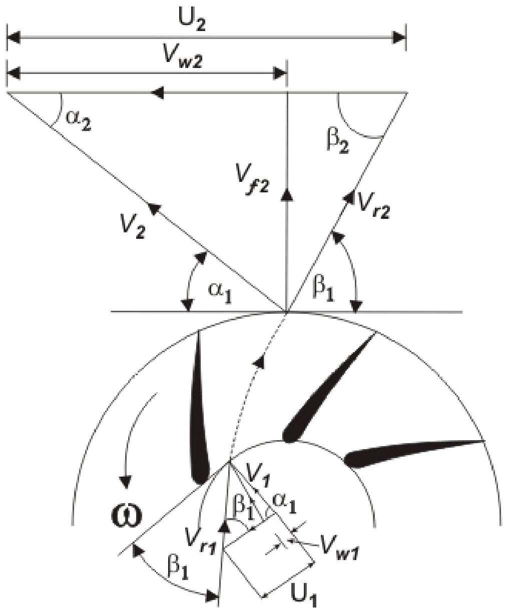


Figure 34.1 Velocity triangles for centrifugal pump Impeller

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 manometric head, 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 manometric efficiency 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

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

so that

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

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

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

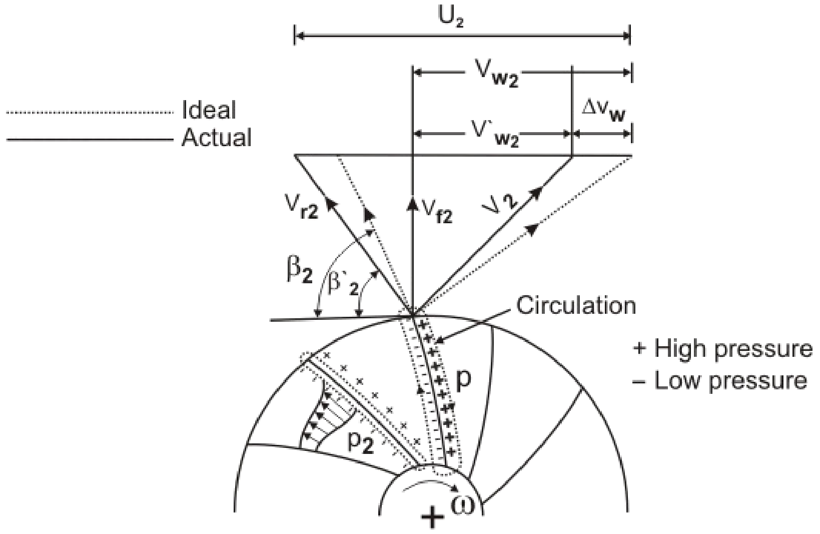


Figure 34.2 Slip and velocity in the impeller blade passage of a centrifugal pump

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).

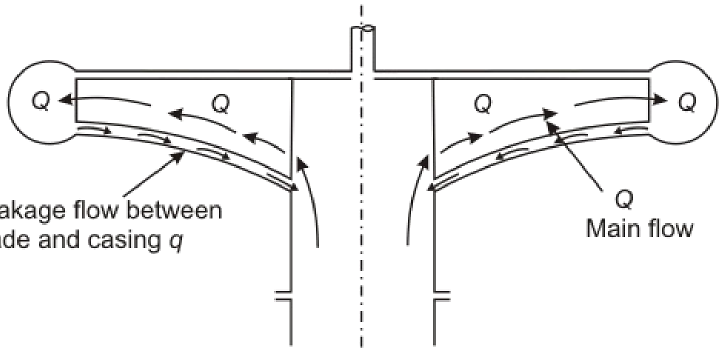


Figure 34.3 Leakage and recirculation in a centrifugal pump

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

$$H_{\text{theo}} = \frac{V_{w2}\, U_2}{g} \qquad (35.1)$$

From the outlet velocity triangle figure(34.1)

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

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

$$U_2 = \pi D N$$

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

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

$$= K_1 - K_2 Q \qquad (35.3)$$

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

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

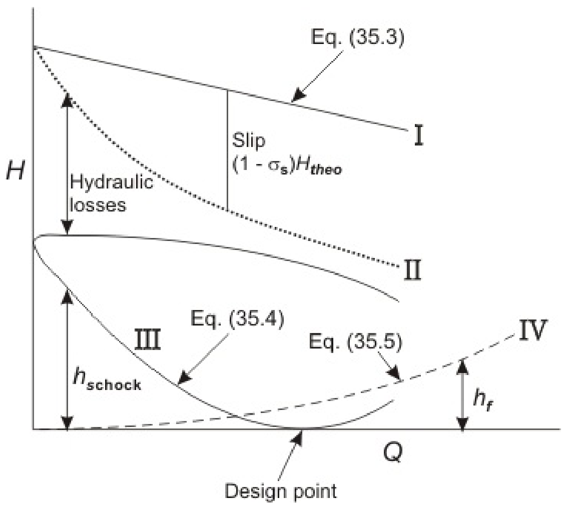


Figure 35.1 Head-discharge characteristics of a centrifugal pump

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).

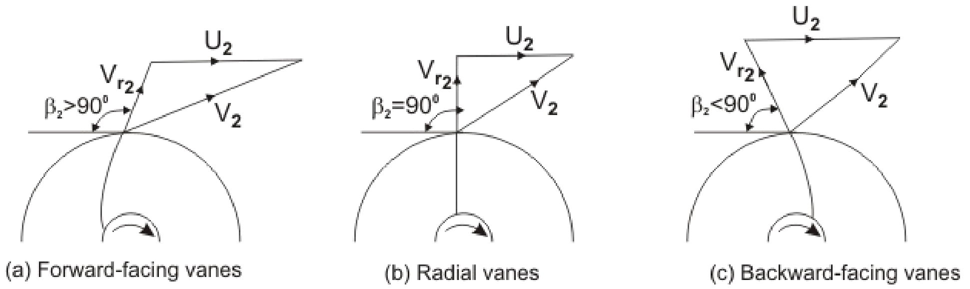


Figure 35.2 Outlet velocity triangles for different blade settings in a centrifugal pump

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).

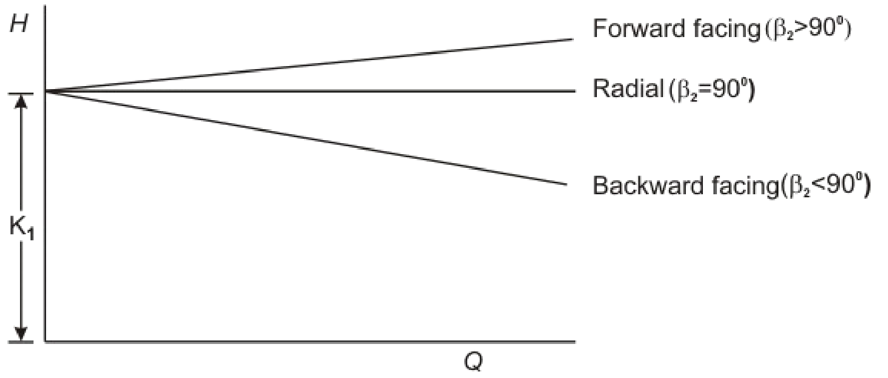


Figure 35.3 Theoretical head-discharge characteristic

curves of a centrifugal pump for different blade settings

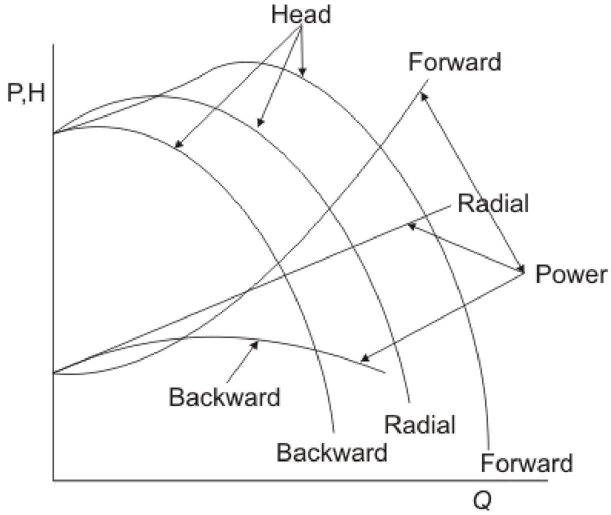


Figure 35.4 Actual head-discharge and power-discharge characteristic curves of a centrifugal pump

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.

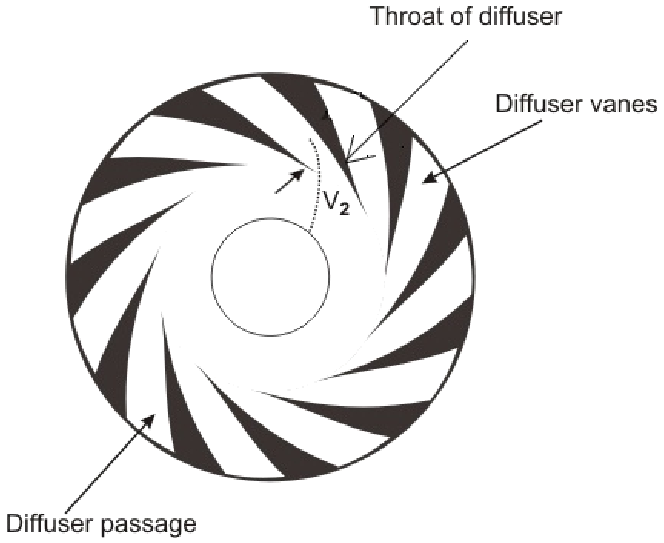


Figure 36.1 A vanned diffuser of a centrifugal pump

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,

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

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

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

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

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

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.

Previous Next

Axial Flow or Propeller Pump

The axial flow or propeller pump is the converse of axial flow turbine and is very similar to it an appearance. The impeller consists of a central boss with a number of blades mounted on it. The impeller rotates within a cylindrical casing with fine clearance between the blade tips and the casing walls. Fluid particles, in course of their flow through the pump, do not change their radial locations. The inlet guide vanes are provided to properly direct the fluid to the rotor. The outlet guide vanes are provided to eliminate the whirling component of velocity at discharge. The usual number of impeller blades lies between 2 and 8, with a hub diameter to impeller diameter ratio of 0.3 to 0.6.

The Figure 37.1 shows an axial flow pump. The flow is the same at inlet and outlet. an axial flow pumps develops low head but have high capacity. the maximum head for such pump is of the order of 20m.The section through the blade at X-X (Figure 37.1) is shown with inlet and outlet velocity triangles in Figure 37.2.

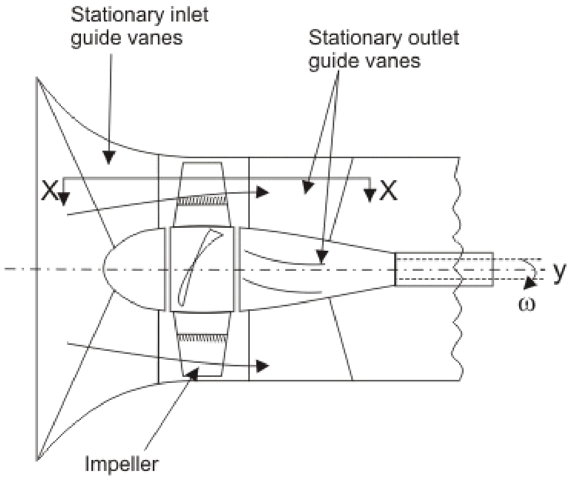


Figure 37.1 A propeller of an axial flow pump

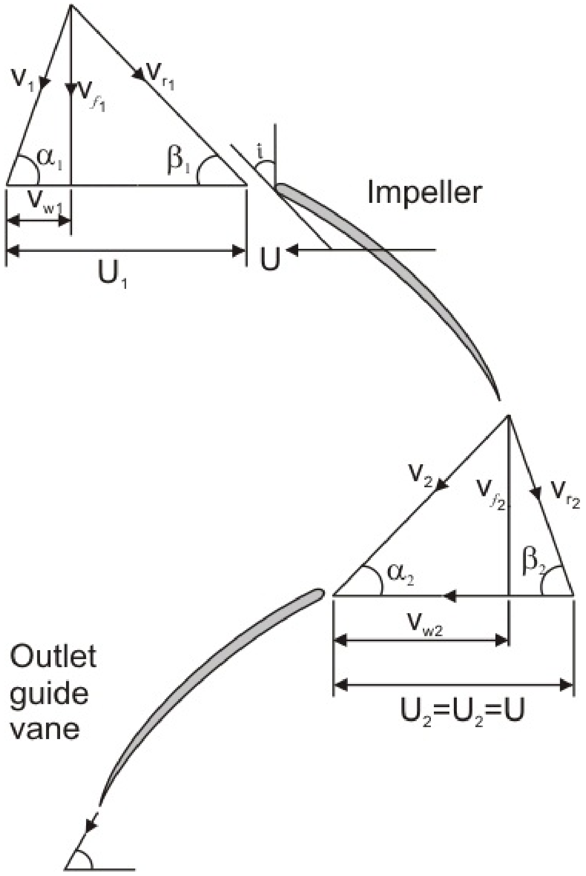


Figure 37.2 Velocity triangles of an axial flow pump

Analysis

The blade has an aerofoil section. The fluid does not impinge tangentially to the blade at inlet, rather the blade is inclined at an angle of incidence
(i) to the relative velocity at the inlet V_1 . If we consider the conditions at a mean radius r_m then

$$u_2 = u_1 = u = wr_m$$

where w is the angular velocity of the impeller.

Work done on the fluid per unit weight = $u(V_{w2} - V_{w1})/g$

For maximum energy transfer , $V_{w1} = 0$, i.e $\alpha_1 = 90^0$ .Again , from the outlet velocity triangle,

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

Assuming a constant flow from inlet to outlet

$$V_{f1} = V_{f2} = V_f$$

Then, we can write

Maximum energy transfer to the fluid per unit weight

$$= u(u - V_f \cot \beta_2)/g \qquad (37.1)$$

For constant energy transfer over the entire span of the blade from hub to tip , the right hand side of Equation (37.1) has to be same for all values of r . It is obvious that u^2 increases with radius r , therefore an equal increase in $uV_f \cot \beta_2$ must take place, and since V_f is constant then $\cot \beta_2$ must increase. Therefore , the blade must be twisted as the radius changes.

Matching of Pump and System Characteristics

The design point of a hydraulic pump corresponds to a situation where the overall efficiency of operation is maximum. However the exact operating point of a pump, in practice, is determined from the matching of pump characteristic with the headloss-flow, characteristic of the external system (i.e. pipe network, valve and so on) to which the pump is connected.

Let us consider the pump and the piping system as shown in Fig. 15.18. Since the flow is highly turbulent, the losses in pipe system are proportional to the square of flow velocities and can, therefore, be expressed in terms of constant loss coefficients. Therefore, the losses in both the suction and delivery sides can be written as

$$h_1 = fl_1 V_1^2 / 2gd_1 + K_1 V_1^2 / 2g \qquad (37.2a)$$

$$h_2 = fl_2 V_2^2 / 2gd_2 + K_2 V_2^2 / 2g \qquad (37.2b)$$

where, h_1 is the loss of head in suction side and h_2 is the loss of head in delivery side and *f* is the Darcy's friction factor, l_1, d_1 and l_2, d_2 are the lengths and diameters of the suction and delivery pipes respectively, while V_1 and V_2 are accordingly the average flow velocities. The first terms in Eqs. (37.1a) and (37.1b) represent the ordinary friction loss (loss due to friction between fluid ad the pipe wall), while the second terms represent the sum of all the minor losses through the loss coefficients K_1 and K_2 which include losses due to valves and pipe bends, entry and exit losses, etc. Therefore the total head the pump has to develop in order to supply the fluid from the lower to upper reservoir is

$$H = H_s + h_1 + h_2 \qquad (37.3)$$

Now flow rate through the system is proportional to flow velocity. Therefore resistance to flow in the form of losses is proportional to the square of the flow rate and is usually written as

$$h_1 + h_2 = \text{system resistance} = KQ^2 \qquad (37.4)$$

where *K* is a constant which includes, the lengths and diameters of the pipes and the various loss coefficients. System resistance as expressed by Eq. (37.4), is a measure of the loss of head at any particular flow rate through the system. If any parameter in the system is changed, such as adjusting a valve opening, or inserting a new bend, etc., then *K* will change. Therefore, total head of Eq. (37.2) becomes,

$$H = H_s + KQ^2 \qquad (37.5)$$

The head *H* can be considered as the total opposing head of the pumping system that must be overcome for the fluid to be pumped from the lower to the upper reservoir.

The Eq. (37.4) is the equation for system characteristic, and while plotted on *H-Q* plane (Figure 37.3), represents the system characteristic curve. The point of intersection between the system characteristic and the pump characteristic on *H-Q* plane is the operating point which may or may not lie at the design point that corresponds to maximum efficiency of the pump. The closeness of the operating and design points depends on how good an estimate of the expected system losses has been made. It should be noted that if there is no rise in static head of the liquid (for example pumping in a horizontal pipeline between two reservoirs at the same elevation), H_s is zero and the system curve passes through the origin.

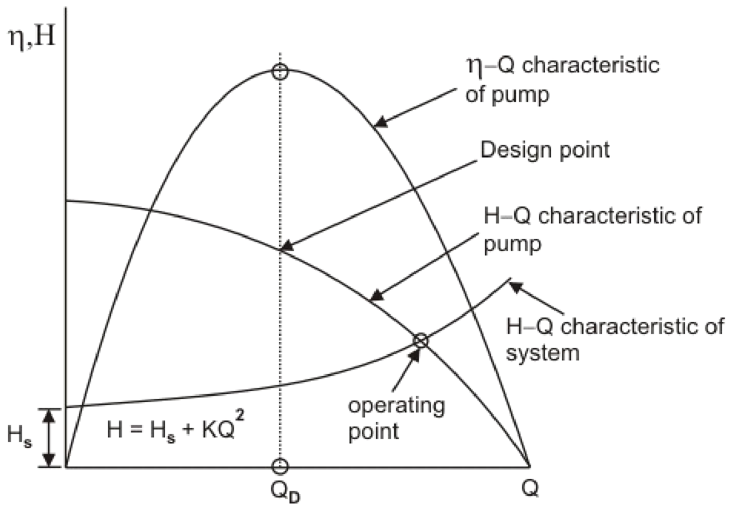


Figure 37.3 H-Q Characteristics of pump and system

Effect of Speed Variation

Head-Discharge characteristic of a given pump is always referred to a constant speed. If such characteristic at one speed is know, it is possible to predict the characteristic at other speeds by using the principle of similarity. Let *A, B, C* are three points on the characteristic curve (Fig. 37.4) at speed N_1 .

For points *A, B* and *C* , the corresponding heads and flows at a new speed N_2 are found as follows:

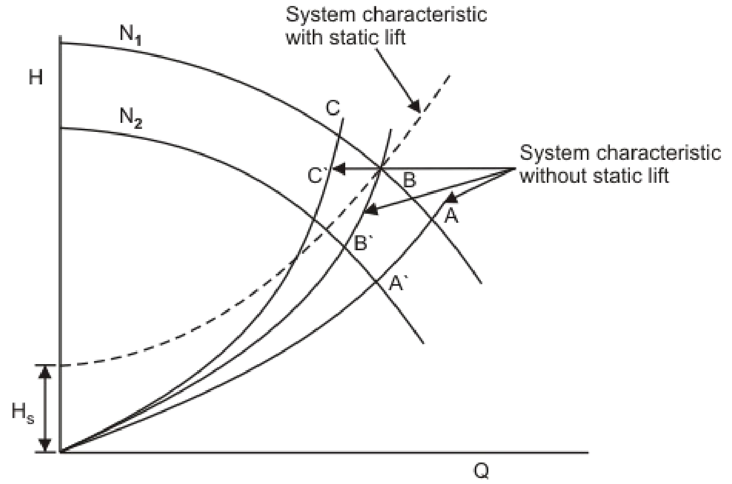


Figure 37.4 Effect of speed variation on operating point of a centrifugal pump

From the equality of π_1 term [Eq. (3.1)] gives

$$Q_1 / N_1 = Q_2 / N_2 \quad \text{(since for a given pump } D \text{ is constant)} \quad (37.6)$$

and similarly, equality of π_2 term [Eq. (3.1)] gives

$$H_1 / N_1^2 = H_2 / N_2^2 \quad (37.7)$$

Applying Eqs. (37.6) and (37.7) to points *A, B* and *C* the corresponding points A' B' and C' are found and then the characteristic curve can be drawn at the new speed N_2

Thus,

$$Q_2 = Q_1 N_2 / N_1 \text{ and } H_2 = H_1 (N_2)^2 / (N_1)^2$$

which gives

$$\frac{H_2}{H_1} = \frac{Q_2^2}{Q_1^2}$$

or $$H \propto Q^2 \quad (37.8)$$

Equation (37.8) implies that all corresponding or similar points on Head-Discharge characteristic curves at different speeds lie on a parabola passing through the origin. If the static lift H_s becomes zero, then the curve for system characteristic and the locus of similar operating points will be the same parabola passing through the origin. This means that, in case of zero static life, for an operating point at speed N_1 , it is only necessary to apply the similarity laws directly to find the corresponding operating point at the new speed since it will lie on the system curve itself (Figure 37.4).

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,
$$H \propto Q^{2/3} \qquad (38.1)$$

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.

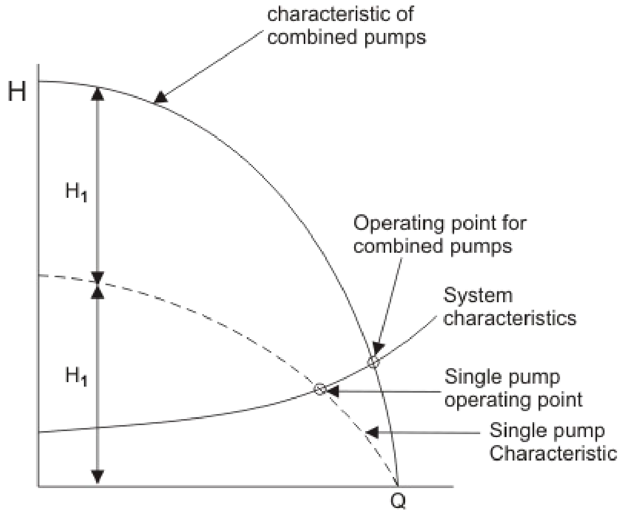


Figure 38.1 Two similar pumps connected in series

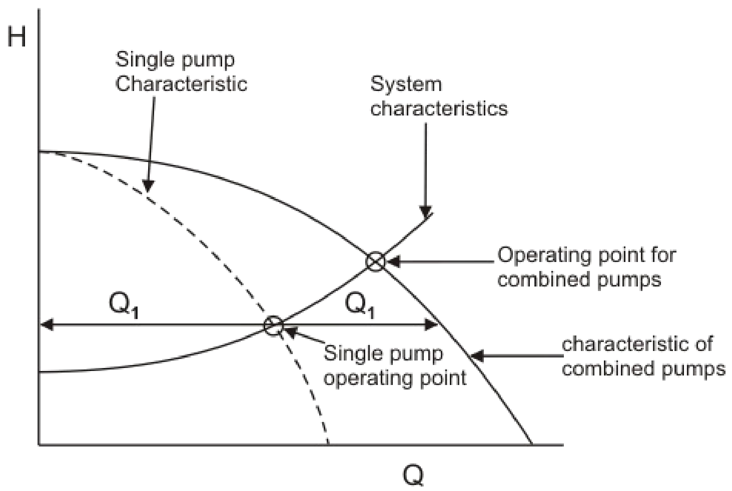


Figure 38.2 Two similar pumps connected in parallel

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

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

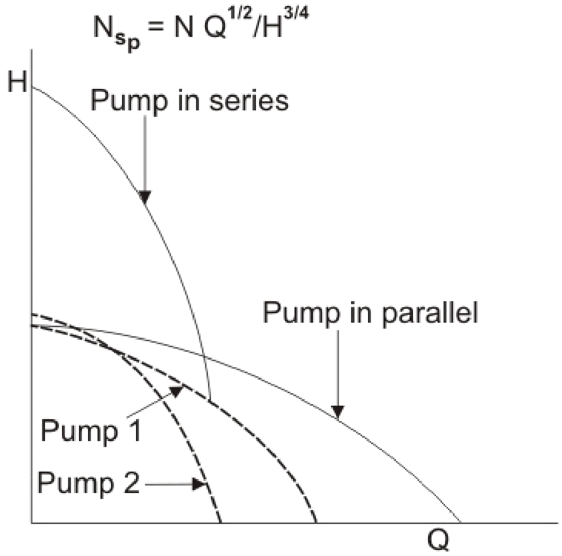


Figure 38.3 Two different pumps connected in series and parallel

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

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

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

Work input per unit weight of

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

=72.78m

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

Absolute velocity of water at the outlet

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

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

= 65.87m

Hence, we theoretical maximum lift = 65.87m

MODULE 7

Fans and Blowers

Next

Fans and blowers (Fig. 39.1) are turbomachines which deliver air at a desired high velocity (and accordingly at a high mass flow rate) but at a relatively low static pressure. The pressure rise across a fan is extremely low and is of the order of a few millimeters of water gauge.The upper limit of pressure rise is of the order of 250mm of water gauge.The rise in static pressure across a blower is relatively higher and is more than 1000 mm of water gauge that is required to overcome the pressure losses of the gas during its flow through various passages. A blower may be constructed in multistages for still higher discharge pressure.

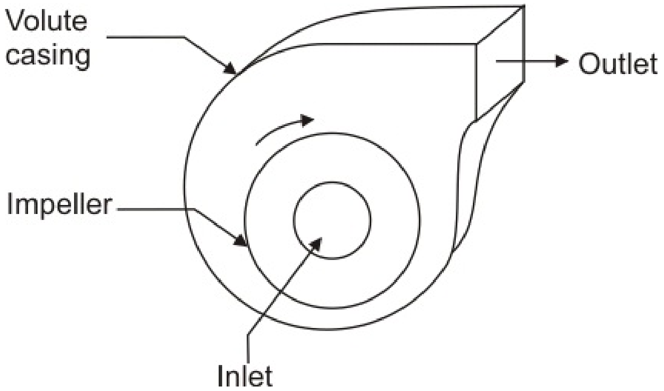


Figure 39.1 A centrifugal fan or blower

A large number of fans and blowers for relatively high pressure applications are of centrifugal type. The main components of a centrifugal blower are shown in Fig. 39.2. A blower consists of an impeller which has blades fixed between the inner and outer diameters. The impeller can be mounted either directly on the shaft extension of the prime mover or separately on a shaft supported between two additional bearings. Air or gas enters the impeller axially through the inlet nozzle which provides slight acceleration to the air before its entry to the impeller. The action of the impeller swings the gas from a smaller to a larger radius and delivers the gas at a high pressure and velocity to the casing. The flow from the impeller blades is collected by a spiral-shaped casing known as *volute casing* or *spiral casing* . The casing can further increase the static pressure of the air and it finally delivers the air to the exit of the blower.

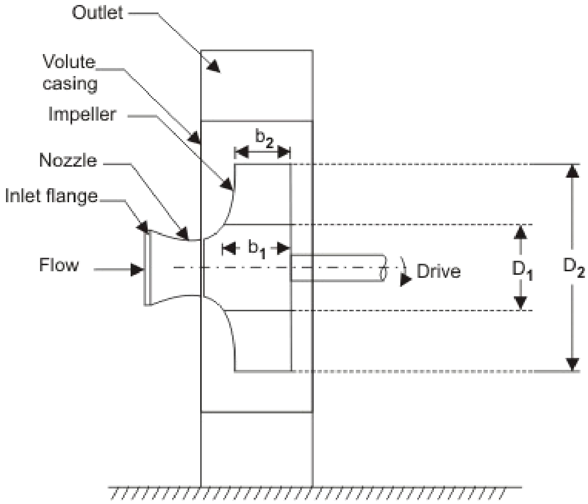


Figure 39.2 Main components of a centrifugal blower

The centrifugal fan impeller can be fabricated by welding curved or almost straight metal blades to the two side walls (shrouds) of the rotor. The casings are made of sheet metal of different thickness and steel reinforcing ribs on the outside. Suitable sealing devices are used between the shaft and the casing.

A centrifugal fan impeller may have backward swept blades, radial tipped blades or forward swept blades as shown in Fig. 39.3. The inlet and outlet velocity triangles are also shown accordingly in the figure. Under ideal conditions, the directions of the relative velocity vectors V_{r1} and V_{r2} are same as the blade angles at the entry and the exit. A zero whirl at the inlet is assumed which results in a zero angular momentum at the inlet. The backward swept blades are employed for lower pressure and lower flow rates. The radial tipped blades are employed for handling dust-laden air or gas because they are less prone to blockage, dust erosion and failure. The radial-tipped blades in practice are of forward swept type at the inlet as shown in Fig. 39.3. The forward-swept blades are widely used in practice. On account of the forward-swept blade tips at the exit, the whirl component of exit velocity (V_{w2}) is large which results in a higher stage pressure rise.

The following observations may be noted from figure 39.3.

$V_{w2} < U_2$, if $\beta_2 < 90^0$, backward swept blades

$V_{w2} = U_2$, if $\beta_2 = 90^0$, radial blades

$V_{w2} > U_2$, if $\beta_2 > 90^0$, forward swept blades

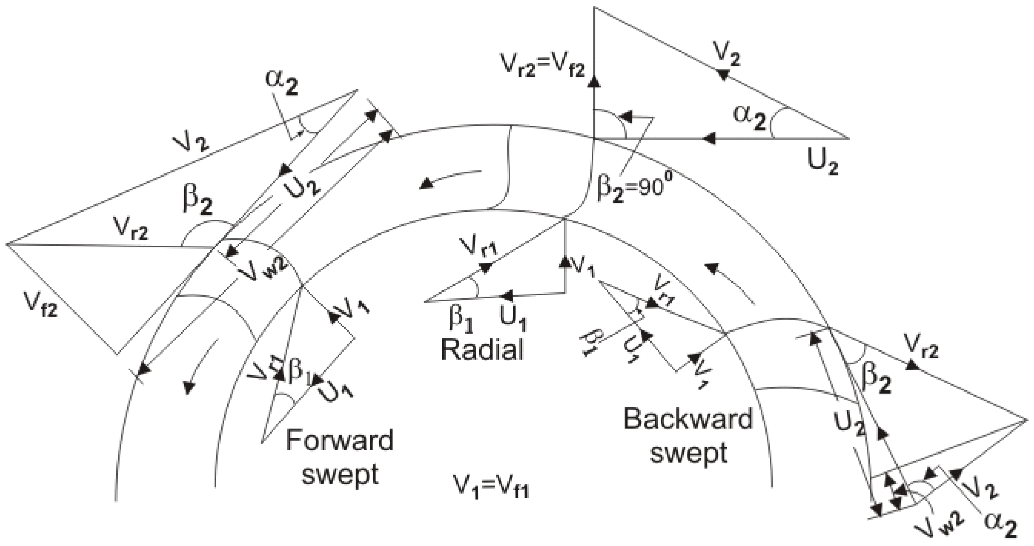


Figure 39.3 Velocity triangles at inlet and outlet of different types of blades of an impeller of a centrifugal blower

Parametric Calculations

The mass flow rate through the impeller is given by

$$\dot{m} = \rho_1 Q_1 = \rho_2 Q_2 \qquad (39.1)$$

The areas of cross sections normal to the radial velocity components V_{f1} and V_{f2} are $A_1 = \pi D_1 b_1$ and $A_2 = \pi D_2 b_2$

$$m = \rho_1 V_{f1}(\pi D_1 b_1) = \rho_2 V_{f2}(\pi D_2 b_2) \qquad (39.2)$$

The radial component of velocities at the impeller entry and exit depend on its width at these sections. For small pressure rise through the impeller stage, the density change in the flow is negligible and the flow can be assumed to be almost incompressible. For constant radial velocity

$$V_{f1} = V_{f2} = V_f \qquad (39.3)$$

Eqs. (39.2) and (39.3) give

$$b_1 / b_2 = D_2 / D_1 \qquad (39.4)$$

Work

The work done is given by Euler's Equation (refer to Module-1) as

$$w = U_2 V_{w2} - U_1 V_{w1} \qquad (39.5)$$

It is reasonable to assume zero whirl at the entry. This condition gives

$\alpha_1 = 90°$, $V_{w1} = 0$ and hence, $U_1 V_{w1} = 0$

Therefore we can write,

$$V_1 = V_{f1} = V_{f2} = U_1 \tan \beta_1 \qquad (39.6)$$

Equation (39.5) gives

$$w = U_2 V_{w2} = U_2^2 \left(\frac{V_{w2}}{U_2}\right) \qquad (39.7)$$

For any of the exit velocity triangles (Figure 39.3)

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

$$\frac{V_{w2}}{U_2} = \left[1 - \frac{V_{f2} \cot \beta_2}{U_2}\right] \qquad (39.8)$$

Eq. (39.7) and (39.8)

$$w = U_2^2 [1 - \varphi \cot \beta_2] \qquad (39.9)$$

where $\varphi = (V_{f2} / U_2)$ is known as flow coefficient

Head developed in meters of air = $H_a = \dfrac{U_2 V_{w2}}{g}$ (39.10)

Equivalent head in meters of water = $H_w = \dfrac{\rho_a H_a}{\rho_w}$ (39.11)

where ρ_a and ρ_w are the densities of air and water respectively.

Assuming that the flow fully obeys the geometry of the impeller blades, the specific work done in an isentropic process is given by

$$(\Delta h_0) = U_2 (1 - \varphi \cot \beta_2) \qquad (39.12)$$

The power required to drive the fan is

$$P = m(\Delta h_0) = mU_2 V_{w2} = mU_2^2 (1 - \varphi \cot \beta_2)$$

$$= mc_p(\Delta T_0) \tag{39.13}$$

The static pressure rise through the impeller is due to the change in centrifugal energy and the diffusion of relative velocity component. Therefore, it can be written as

$$p_2 - p_1 = (\Delta p) = \frac{1}{2}\rho(U_2^2 - U_1^2) + \frac{1}{2}\rho(V_{r1}^2 - V_{r2}^2) \qquad (39.13)$$

The stagnation pressure rise through the stage can also be obtained as:

$$(\Delta p_0) = \frac{1}{2}\rho(U_2^2 - U_1^2) + \frac{1}{2}\rho(V_{r1}^2 - V_{r2}^2) + \frac{1}{2}\rho(V_2^2 - V_1^2) \qquad (39.14)$$

From (39.13) and (39.14) we get

$$(\Delta p_0) = (\Delta p) + \frac{1}{2}\rho(V_2^2 - V_1^2) \qquad (39.15)$$

From any of the outlet velocity triangles (Fig. 39.3),

$$\frac{V_2}{\sin\beta_2} = \frac{U_2}{\sin\{\pi - (\alpha_2 + \beta_2)\}}$$

or, $$\frac{V_2}{\sin\beta_2} = \frac{U_2}{\sin(\alpha_2 + \beta_2)} \qquad (39.16)$$

or, $$V_{w2} = V_2\cos\alpha_2 = \frac{U_2\sin\beta_2\cos\alpha_2}{\sin(\alpha_2 + \beta_2)}$$

$$\frac{V_{w2}}{U_2} = \frac{\sin\beta_2\cos\alpha_2}{\sin\alpha_2\cos\beta_2 + \cos\alpha_2\sin\beta_2}$$

or, $$\frac{V_{w2}}{U_2} = \frac{\tan\beta_2}{\tan\alpha_2 + \tan\beta_2}$$

Work done per unit mass is also given by (from (39.7) and (40.4)):

$$w = U_2^2\left(\frac{\tan\beta_2}{\tan\alpha_2 + \tan\beta_2}\right) \qquad (39.17)$$

Efficiency

On account of losses, the isentropic work $\frac{1}{\rho}(\Delta p_0)$ is less than the actual work.

Therefore the stage efficiency is defined by

$$\eta_s = \frac{(\Delta p_0)}{\rho U_2 V_{w2}} \qquad (39.18)$$

Number of Blades

Too few blades are unable to fully impose their geometry on the flow, whereas too many of them restrict the flow passage and lead to higher losses. Most of the efforts to determine the optimum number of blades have resulted in only empirical relations given below

(i) $$n = \frac{8.5\sin\beta_2}{1 - D_1/D_2} \qquad (39.19)$$

(ii) $$n = 6.5\left(\frac{D_2 + D_1}{D_2 - D_1}\right)\sin\frac{1}{2}(\beta_1 + \beta_2) \qquad (39.20)$$

(iii) $$n = \frac{1}{3}\beta_2 \qquad (39.21)$$

Impeller Size

The diameter ratio (D_1/D_2) of the impeller determines the length of the blade passages. The smaller the ratio the longer is the blade passage. The following value for the diameter ratio is often used by the designers

$$\frac{D_1}{D_2} = 1.2(\varphi)^{1/3} \qquad (39.22)$$

where

$$\varphi = V_{f2}/U_2$$

The following relation for the blade width to diameter ratio is recommended:

$$b_1 / D_1 = 0.2 \qquad (39.23)$$

If the rate of diffusion in a parallel wall impeller is too high, the tapered shape towards the outer periphery, is preferable..

The typical performance curves describing the variation of head, power and efficiency with discharge of a centrifugal blower or fan are shown in Figure 39.4

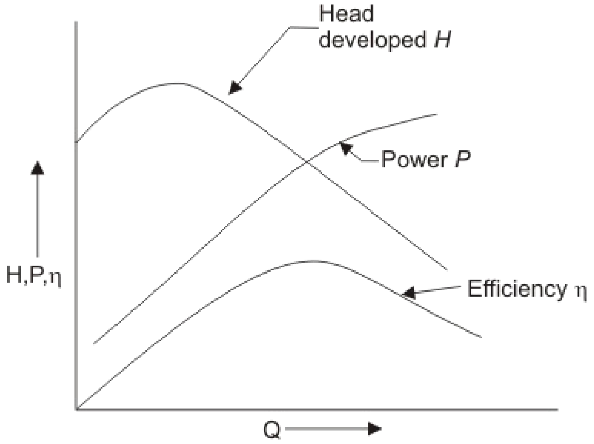


Figure39.4 Performance characteristic curves of a centrifugal blower or fan

Fan Laws

The relationships of discharge Q , head H and Power P with the diameter D and rotational speed N of a centrifugal fan can easily be expressed from the dimensionless performance parameters determined from the principle of similarity of rotodynamic machines as described before . These relationships are known as Fan Laws described as follows

$$Q = K_q D^3 N \qquad (40.1)$$

$$H = \frac{K_h D^2 N^2 \rho}{g} \qquad (40.2)$$

$$P = \frac{K_p D^5 N^3 \rho}{g} \qquad (40.3)$$

where $K_q, K_h,$ and K_p are constants.

For the same fan, the dimensions get fixed and the laws are

$$\frac{Q_1}{Q_2} = \frac{N_1}{N_2}$$

$$\frac{H_1}{H_2} = \left(\frac{N_1}{N_2}\right)^2 \text{ and } \frac{P_1}{P_2} = \left(\frac{N_1}{N_2}\right)^3$$

For the different size and other conditions remaining same, the laws are

$$\frac{Q_1}{Q_2} = \left(\frac{D_1}{D_2}\right)^3, \ \frac{H_1}{H_2} = \left(\frac{D_1}{D_2}\right)^2 \text{ and } \frac{P_1}{P_2} = \left(\frac{D_1}{D_2}\right)^5 \qquad (40.4)$$

These relationships are known as the Fan-laws. The Fan-lows can be summarized as

For the same fan:

Discharge $\propto$ Speed

Head developed $\propto$ (Speed)^2

Power $\propto$ (Speed)^3

For the fans of different sizes:

Discharge $\propto$ (Diameter)^3

Head developed $\propto$ (Diameter)^2

Power $\propto$ (Diameter)^5

Performance of Fans

For all three cases (backward, radial and forward swept blades) in Figure 39.3, we can write

$$V_{w2} = U_2 - V_{f2} \cot \beta_2 \qquad (40.5)$$

The work done is given by Euler's equation (refer to Modue-1) as

$$W = U_2 V_{w2} - U_1 V_{w1} \qquad (40.6)$$

Noting that $V_{w1} = 0$ (zero whirl at the entry) we can write

$$\frac{\Delta p}{\rho} = U_2 V_{w2} = U_2 (U_2 - V_{f2} \cot \beta_2)$$

or,
$$\frac{\Delta p}{\rho} = U_2^2 - U_2 V_{f2} \cot \beta_2 \qquad (40.7)$$

The volume flow rate (assuming no density change between the inlet and outlet)

$$Q = \pi D_1 V_{f1} b_1 = \pi D_2 V_{f2} b_2$$

Thus

$$V_{f2} = \frac{Q}{\pi D_2 b_2}$$

By substitution in (40.7)

$$\frac{\Delta p}{\rho} = U_2^2 - (QU_2)/(\pi D_2 b_2) \cot \beta_2 \qquad (40.8)$$

Previous Next

Let us define, Pressure Coefficient,

$$\Psi = \Delta p \Big/ \left(\frac{1}{2}\rho U_2^2\right) \qquad (40.9)$$

Volume Coefficient,

$$\phi = 4Q \big/ (\pi D_2^2 U_2) \qquad (40.10)$$

and Power Coefficient,

$$\mathbb{P} = \Psi\phi \qquad (40.11)$$

Substitution of (40.8) in the above yields

$$\frac{\Delta p}{\frac{1}{2}(\rho U_2^2)} = 2\left(1 - \frac{Q}{\pi D_2 b_2 U_2}\cot\beta_2\right) \qquad (40.12)$$

or,

$$\Psi = 2\left(1 - \frac{\phi D_2}{4b_2}\cot\beta_2\right) \qquad (40.13)$$

$$\mathbb{P} = 2\phi\left(1 - \frac{\phi D_2}{4b_2}\cot\beta_2\right) \qquad (40.14)$$

Equations (40.13) and (40.14) are plotted in Fig 40.1 for different values of volume coefficient ϕ with β_2 as a parameter.

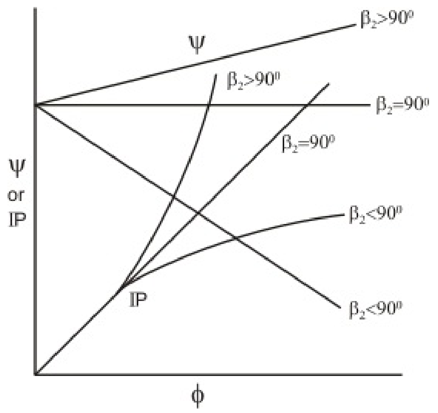


Figure 40.1 Performance curves of a Fan

The characteristics in Figure 40.1 depict the following

(i) Forward curved fans $(\beta_2 > 90^0)$ develop the highest pressure for a given impeller diameter and speed.

(ii) Power requirement of a forward curved fan increases steeply for a small change in flow rate.

(iii) Pressure developed decreases fast with increasing flow rate in a backward curved fan

In conclusion, the forward curved fans have large volume discharge and pressure rise but they demand higher power. However, forward curved fans are unstable for off-design operating conditions.

Backward curved fans are very efficient and the drooping power characteristic makes them suitable for a better off-design performance

Radial curved fans are preferred for dust-laden fluids. Due to their shape, the solid particles are not stuck and deposited on the blade surface.