RAJIV GANDHI PROUDYOGIKI VISHWAVIDYALAYA BHOPAL

New Scheme of Examination as per AICTE Flexible Curricula

Mechanical Engineering, VI-Semester

Departmental Elective ME- 603 (A) Turbomachinery

Unit 1: **Energy transfer in turbo machines**:

Application of first and second laws of thermodynamics to turbo machines, moment of momentum equation and Euler turbine equation, principles of impulse and reaction machines, degree of reaction, energy equation for relative velocities, one dimensional analysis only.

Unit 2: **Steam turbines**:

Impulse staging, velocity and pressure compounding, utilization factor, analysis for optimum U.F Curtis stage, and Rateau stage, include qualitative analysis, effect of blade and nozzle losses on vane efficiency, stage efficiency, analysis for optimum efficiency, mass flow and blade height. Reactions staging: Parson's stages, degree of reaction, nozzle efficiency, velocity coefficient, stator efficiency, carry over efficiency, stage efficiency, vane efficiency, conditions for optimum efficiency, speed ratio, axial thrust, reheat factor in turbines, problem of radial equilibrium, free and forced vortex types of flow, flow with constant reaction, governing and performance characteristics of steam turbines.

Unit 3: Water turbines:

Classification, Pelton, Francis and Kaplan turbines, vector diagrams and work-done, draft tubes, governing of water turbines. Centrifugal Pumps: classification, advantage over reciprocating type, definition of mano-metric head, gross head, static head, vector diagram and work done. Performance and characteristics: Application of dimensional analysis and similarity to water turbines and centrifugal pumps, unit and specific quantities, selection of machines, Hydraulic, volumetric, mechanical and overall efficiencies, Main and operating characteristics of the machines, cavitations.

Unit 4 : Rotary Fans, Blowers and Compressors:

Classification based on pressure rise, centrifugal and axial flow machines. Centrifugal Blowers Vane shape, velocity triangle, degree of reactions, slip coefficient, size and speed of machine, vane shape and stresses, efficiency, characteristics, fan laws and characteristics. Centrifugal Compressor – Vector diagrams, work done, temp and pressure ratio, slip factor, work input factor, pressure coefficient, Dimensions of inlet eye, impeller and diffuser. Axial flow Compressors- Vector diagrams, work done factor, temp and pressure ratio, degree of reaction, Dimensional Analysis, Characteristics, surging, Polytrophic and isentropic efficiencies.

Unit 5: Power transmitting turbo machines:

Application and general theory, their torque ratio, speed ratio, slip and efficiency, velocity diagrams, fluid coupling and Torque converter, characteristics, Positive displacement machines and turbo machines, their distinction. Positive displacement pumps with fixed and variable displacements, Hydrostatic systems hydraulic intensifier, accumulator, press and crane.

References:

1. Venkanna BK; turbomachinery; PHI
2. Shepherd DG; Turbo machinery
3. Csanady; Turbo machines
4. Bansal R. K; Fluid Mechanics & Fluid Machines;
5. Rogers Cohen & Sarvan Multo Gas Turbine Theory
6. Kearton W. J; Steam Turbine: Theory & Practice

RAJIV GANDHI PROUDYOGIKI VISHWAVIDYALAYA BHOPAL

New Scheme of Examination as per AICTE Flexible Curricula

Mechanical Engineering, VI-Semester

Departmental elective ME- 603 (B) Computer Aided Engineering

Unit 1: Introduction to Computer Engineering

Methods to solve engineering problems- analytical, numerical, experimental, their merits and comparison, discretization into smaller elements and effect of size/ shape on accuracy, importance of meshing, boundary conditions, Computer Aided Engineering (CAE) and design, chain-bumping-stages v/s concurrent-collaborative design cycles, computer as enabler for concurrent design and Finite Element Method (FEM), degree of freedom (DOF), mechanical systems with mass, damper and spring, stiffness constant K for tensile, bending and torsion; Practical applications of FEA in new design, optimization/ cost-cutting and failure analysis,

Unit 2: Types of Analysis

Types of analysis in CAE, static (linear/ non linear), dynamic, buckling, thermal, fatigue, crash NVH and CFD, review of normal, shear, torsion, stress-strain; types of forces and moments, tri-axial stresses, moment of inertia, how to do meshing, 1-2-3-d elements and length of elements; force stiffness and displacement matrix, Rayleigh-Ritz and Galerkin FEM; analytical and FEM solution for single rod element and two rod assembly.

Unit 3: 2 D- Meshing

Two-dimension meshing and elements for sheet work and thin shells, effect of mesh density and biasing in critical region, comparison between tria and quad elements, quality checks, jacobian, distortion, stretch, free edge, duplicate node and shell normal.

Unit 4: 3 D-Meshing

Three-dimension meshing and elements, only 3 DOF, algorithm for tria to tetra conversion, floating and fixed trias, quality checks for tetra meshing, brick meshing and quality checks, special elements and techniques, introduction to weld, bolt, bearing and shrink fit simulations, CAE and test data correlations, post processing techniques

Unit 5: Optimization

Review of linear optimization, process and product optimization, design for manufacturing (DFM) aspects in product development, use of morphing technique in FEA, classical design for infinite life and design for warranty life, warranty yard meetings and functional roles, climatic conditions and design abuses, case studies.

References:

1. Gokhle Nitin; et al; Practical Finite Element Analysis; Finite to Infinite, 686 Budhwar Peth, Pune.
2. Krishnamoorthy; Finite Element Analysis, theory and programming; TMH
3. Buchanan; Finite Element Analysis; Schaum series; TMH
4. Seshu P; Textbook of Finite Element Analysis; PHI.
5. Desai Chandrakant S et al; Introduction to finite element Method ,

RAJIV GANDHI PROUDYOGIKI VISHWAVIDYALAYA BHOPAL

New Scheme of Examination as per AICTE Flexible Curricula

Mechanical Engineering, VI-Semester

Departmental elective ME- 603 (C) Product Design

Unit 1: Introduction to product design

Product life-cycle, product policy of an organization. Selection of a profitable product, Product design process, Product analysis.

Unit 2: Value engineering in product design

Advantages, applications in product design, problem identification and selection, Analysis of functions, Anatomy of function. Primary versus secondary versus tertiary/unnecessary functions, functional analysis: Functional Analysis System Technique (FAST), Case studies.

Unit 3: Introduction to Product design tools

QFD, Computer Aided Design, Robust design, DFX, DFM. DFA, Ergonomics in product design.

Unit 4: DFMA guidelines

Product design for manual assembly, Design guidelines for metallic and non-metallic products to be manufactured by different processes such as casting, machining, injection molding etc.,

Unit-5: Rapid Prototyping

Needs of rapid prototyping, needs, advantages, working principles of SLA, LOM and SLS.

References:

1. Value Engineering: Concepts, Techniques and Applications by A.K. Mukhopadhaya
2. Rapid Prototyping: Principles and Applications by C.K. Chua
3. Engineering Design by Linda D. Schmidt

Class Notes
Turbomachinery (ME 502)

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.

Classifications 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 depends 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 party 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.

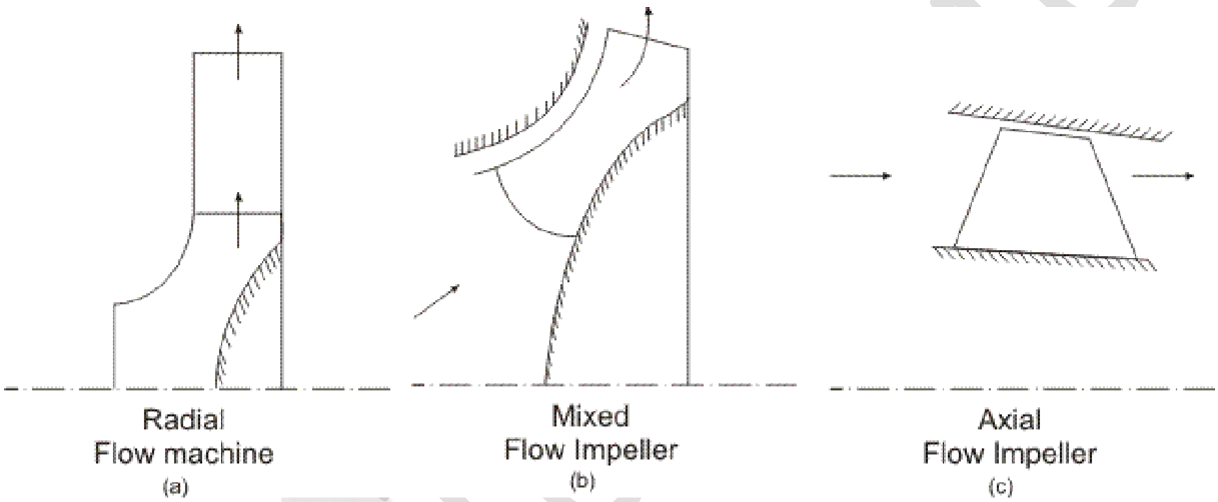


Fig. 1.1 Schematic of different types of impellers

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.

Application Of First And Second Laws Of Thermodynamics To Turbo Machines

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.

Moment Of Momentum Equation And Euler Turbine Equation

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 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 and 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 is directed parallel to the axis of rotation , the radial component is directed radially through the axis to rotation, while the tangential component 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.

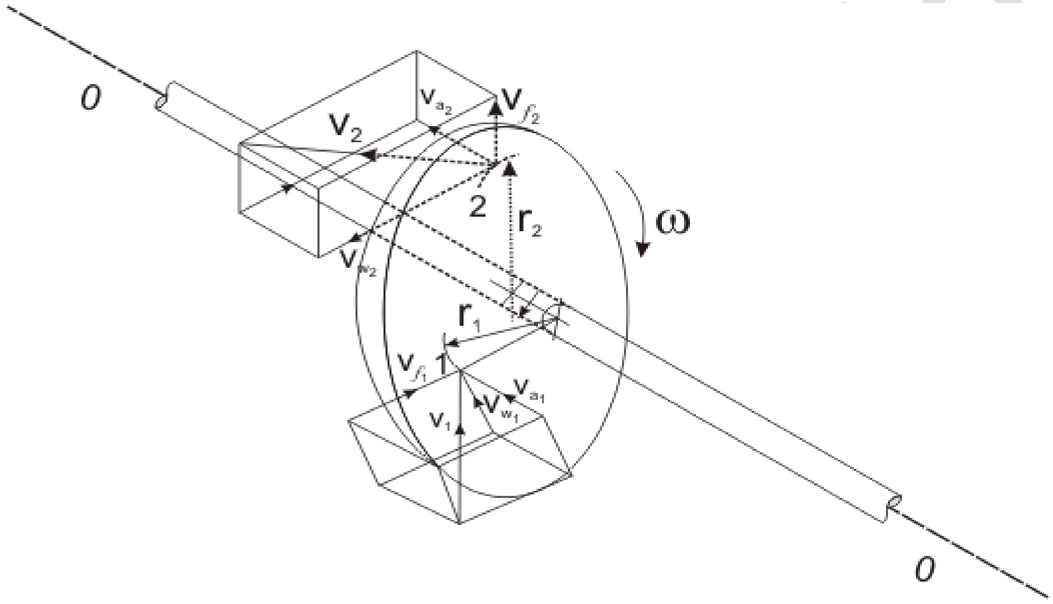


Fig 1.2 Components of flow velocity in a generalized fluid machine

However, for an axi-symmetric 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 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

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

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

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

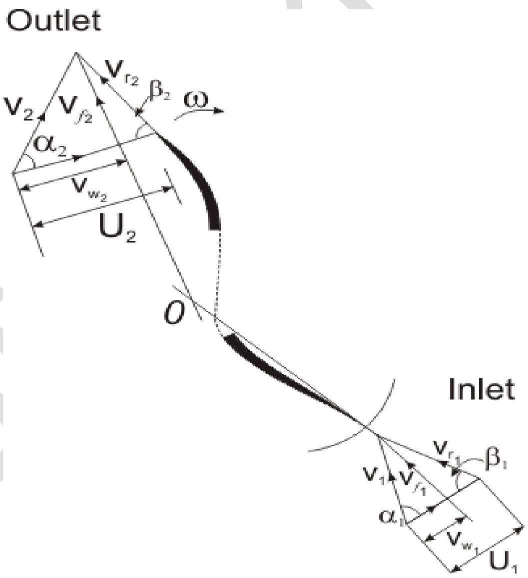
where is the angular velocity of the rotor and which represents the linear velocity of the rotor. Therefore and are the linear velocities of the rotor at points 2 (outlet) and 1 (inlet) respectively (Fig. 1.2). The Eq, (1.2) is known as Euler's equation in relation to fluid machines. The Eq. (1.2) can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

$$H = \frac{V_{w_2} U_2 - V_{w_1} U_1}{g} \qquad (1.3)$$

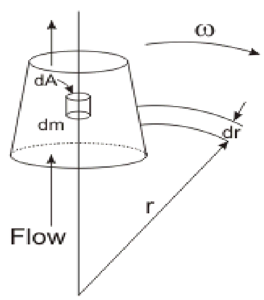
In usual convention relating to fluid machines, the head delivered by the fluid to the rotor is considered to be positive and vice-versa. Therefore, Eq. (1.3) written with a change in the sign of the right hand side in accordance with the sign convention as

$$H = \frac{V_{w_1} U_1 - V_{w_2} U_2}{g} \qquad (1.4)$$

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), (b) Velocity triangles for a generalized rotor vane, Centrifugal effect in a flow of fluid with rotation

Vector diagrams of velocities at inlet and outlet correspond to two velocity triangles, where U1 is the velocity of fluid relative to the rotor and are the angles made by the directions of the absolute velocities at the inlet and outlet respectively with the tangential direction, while and are the angles made by the relative velocities with the tangential direction. The angles α1 and α2 should match with vane or blade angles at inlet and outlet respectively for a smooth, shock-less entry and exit of the fluid to avoid undesirable losses. Now we shall apply a simple geometrical relation as follows:

From the inlet velocity triangle,

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

Similarly from the outlet velocity triangle.

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

Invoking the expressions 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

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

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 Turbo-machines

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 d*m* at radius *r* gives rise to a pressure differential d*p* across the thickness d*r* of the body in a manner that a differential force of d*p*d*A* 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 d*m* = d*A* d*r* ρ where ρ is the density of the fluid, it becomes

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

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 r1 to radius r2 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) is termed as centrifugal head. The transfer of energy due to a change in centrifugal head 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 relative to the rotor, a converging passage in the direction of flow through the rotor increases the relative velocity 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 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 "in viscid" can be written as

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

We assume that the flow is steady in the rotating frame so that . We choose a cylindrical coordinate system with z-axis along the axis of rotation. Then the momentum equation reduces to

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

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 ρ it becomes

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

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 U. Further Vr is the linear velocity of the rotor. Integrating the momentum equation from inlet to outlet along a streamline we have

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

or, (1.8)

Therefore, we can say, with the help of Eq. (1.8), 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, the equation of energy transfer Eq. (1.7) can be written, under this situation, as

$$R=\frac{\frac{1}{2g}[(U_1^2-U_2^2)+(V_{r_2}^2-V_{r_1}^2)]}{H} \qquad (1.9)$$

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 machine Vr and hence the fluid gains in static head. While, for a radially inward flow machine, V2 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.

Principles of impulse and reaction machines

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.

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 R=0. 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.1.4.

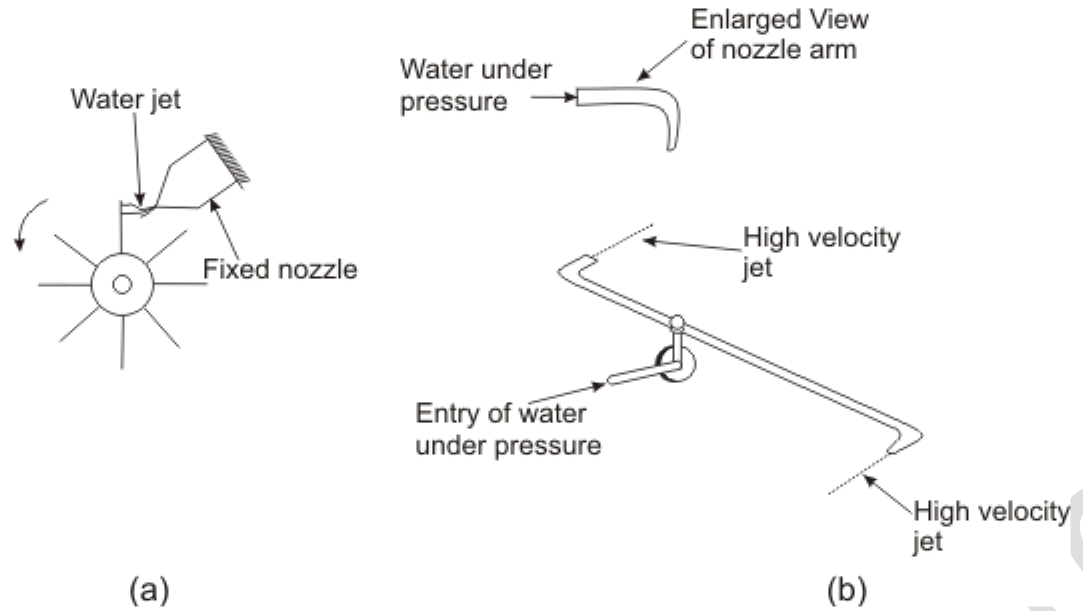


Fig 1.4 (a) Paddle wheel as an example of impulse turbine
(b) Lawn sprinkler as an example of reaction turbine

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. 1.4b) 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. 1.4a), 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. 1.4b), 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.

Degree of reaction

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,

$$\eta_{hydraulic} = \eta_h = \frac{\begin{array}{l} \text{useful energy gained by the fluid} \\ \text{at final discharge} \end{array}}{\text{mechanical energy supplied to rotor}} \tag{1.10a}$$

$$\eta_{overall} = \frac{\begin{array}{l} \text{useful energy gained by the fluid} \\ \text{at final discharge} \end{array}}{\begin{array}{l} \text{mechanical energy supplied to} \\ \text{shaft at coupling} \end{array}} \tag{1.10b}$$

For a turbine,

$$\eta_h = \frac{\text{mechanical energy delivered by the rotor}}{\text{energy available from the fluid}} \tag{1.11a}$$

$$\eta_{overall} = \frac{\begin{array}{l} \text{mechanical energy in output shaft} \\ \text{at coupling} \end{array}}{\text{energy available from the fluid}} \tag{1.11b}$$

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

$$\eta_m = \frac{\eta_{overall}}{\eta_h} \tag{1.12}$$

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.*

Unit 2 - 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. 2.1. Figure 2.2 shows the velocity diagram indicating the flow through the turbine blades.

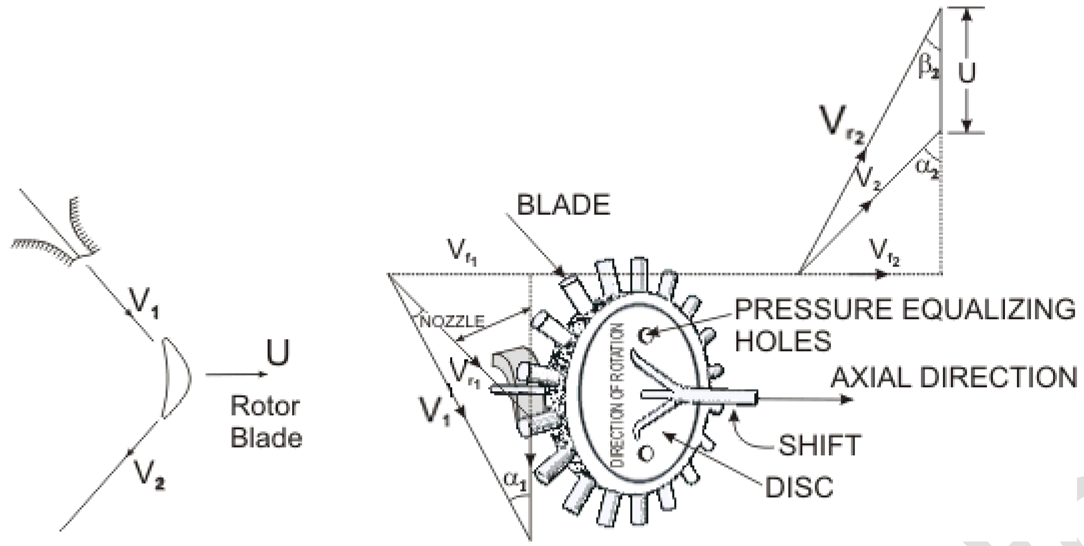


Figure 2.1 Schematic diagram of an Impulse Turbine

and V1, V2 = Inlet and outlet absolute velocity
and Vr1, Vr2 = 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).

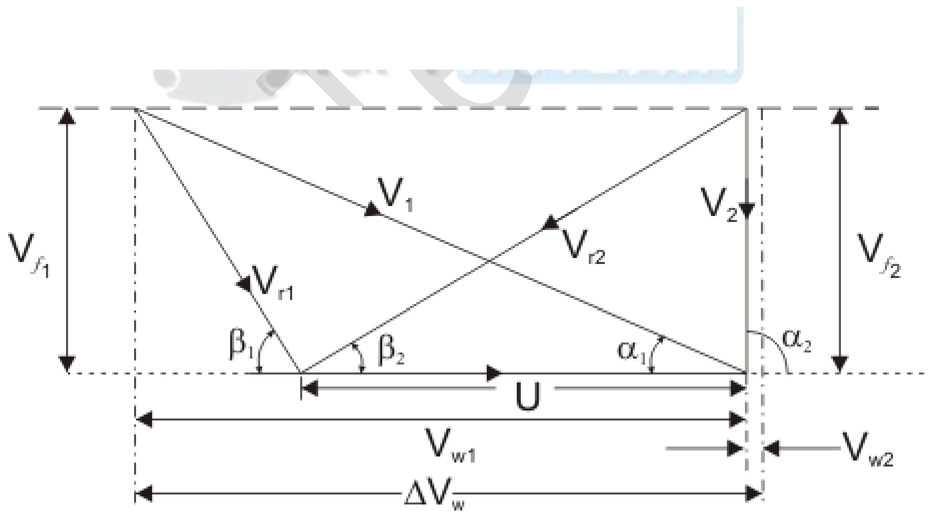


Figure 2.2 Velocity diagram of an Impulse Turbine

and β1, β2 = Inlet and outlet **blade angles**

and Vw1, Vw2 = Tangential or whirl component of absolute velocity at inlet and outlet

and Vf1, Vf2 = Axial component of velocity at inlet and outlet

Tangential force on a blade,

$$F_u = \dot{m}(V_{w1} - Vw2) \tag{2.1}$$

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

or,

$$F_u = \dot{m}\Delta V_w \tag{2.2}$$

Power developed = $F_u = \dot{m}\Delta V_w$ (2.3)

Blade efficiency or Diagram efficiency or Utilization factor is given by

$$\dot{m}U\Delta V_w \tag{2.4}$$

stage efficiency $= \eta_s = \dfrac{Work\ done\ by\ the\ rotor}{Isentropic\ enthalpy\ drop}$ (2.5)

$$\eta_s = \frac{\dot{m}U\Delta V_w}{\dot{m}(\Delta H)_{isen}} = \frac{\dot{m}U\Delta V_w}{\dot{m}\left(\frac{V_1^2}{2}\right)} \cdot \frac{\dot{m}(V_1^2/2)}{\dot{m}(\Delta H)_{isen}} \tag{2.6}$$

or,

or, $\eta_s = \eta_b \times \eta_n \qquad [\eta_n = Nozzle\ efficiency]$ (2.7)

Optimum blade speed of a single stage turbine

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

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

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

$$k = (V_{r2}/V_{r1})$$

where, k = friction coefficient

$$c = (\cos\beta_2 / \cos\beta_1) \qquad = \text{Blade speed ratio} \qquad (2.8)$$

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

$$\rho = \frac{U}{V_1} =$$ or,

$$\rho = \frac{\cos\alpha_1}{2} \qquad (2.9)$$

The maximum value of blade efficiency

$$= \frac{\cos^2\alpha_1}{2}(1+kc) \qquad (2.10)$$

For equiangular blades,

$$(\eta_b)_{\max} = \frac{\cos^2\alpha_1}{2}(1+k) \qquad (2.11)$$

If the friction over blade surface is neglected

$$(\eta_b)_{\max} = \cos^2\alpha_1 \qquad (2.12)$$

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. 2.3 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.

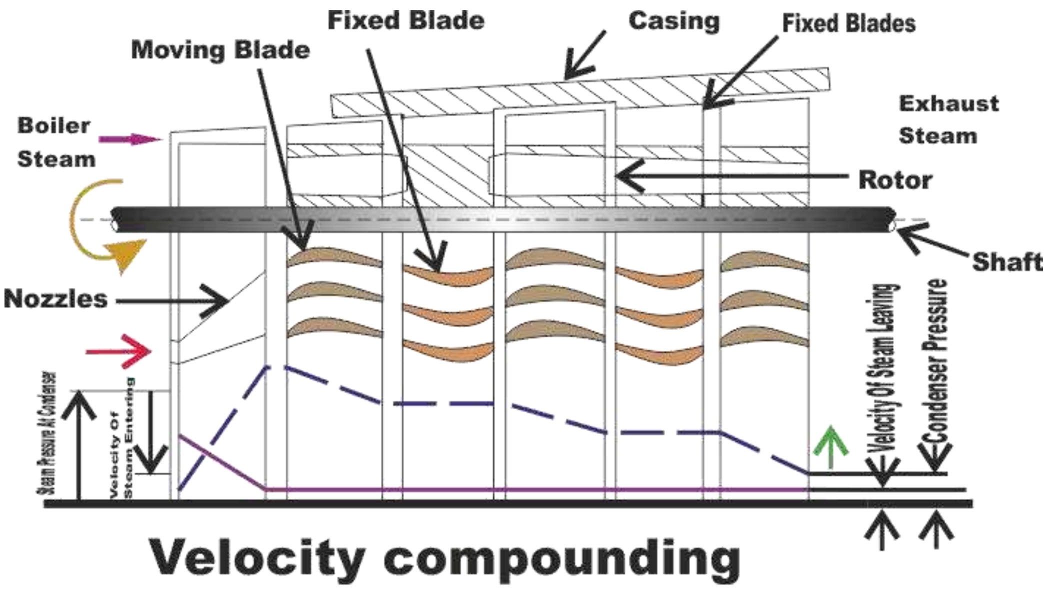


Figure 2.3 Velocity Compounding

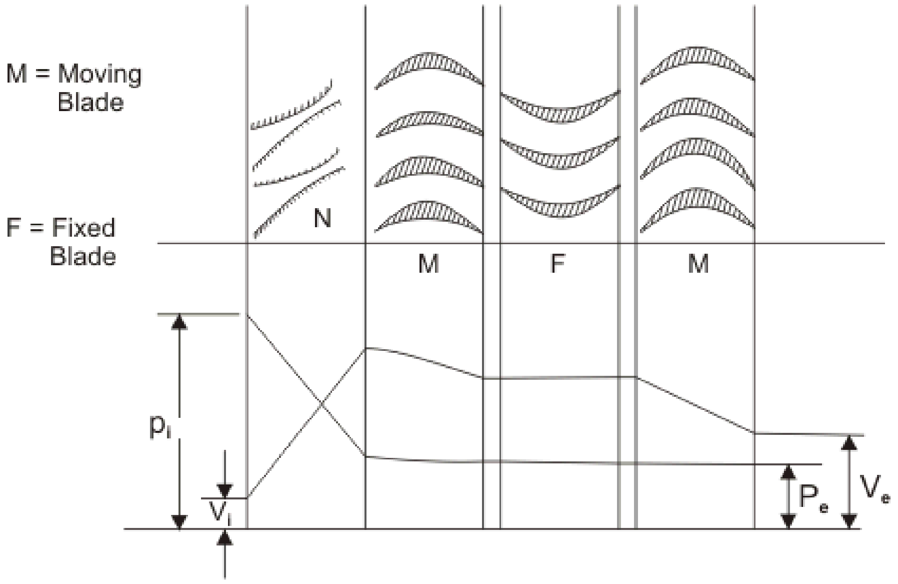
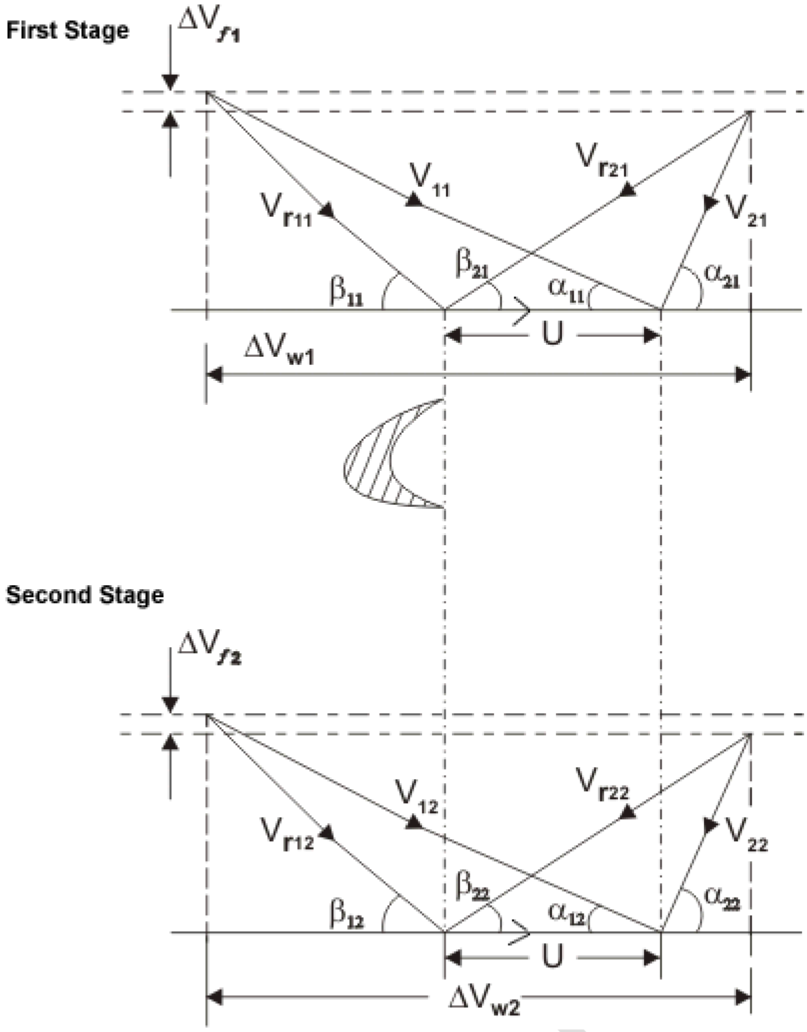


Figure 2.4 Velocity Compounding



Work done = $\dot{m}.U\,(\Delta V_{w1}+\Delta V_{w2})$ (2.13)

End thrust = $\dot{m}\,(\Delta V_{f1}+\Delta V_{f2})$ (2.14)

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 2.3.

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.

The optimum velocity ratio will depend on number of stages and is given by

- Work is not uniformly distributed (1st >2nd)

- The first 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 2.4). 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.

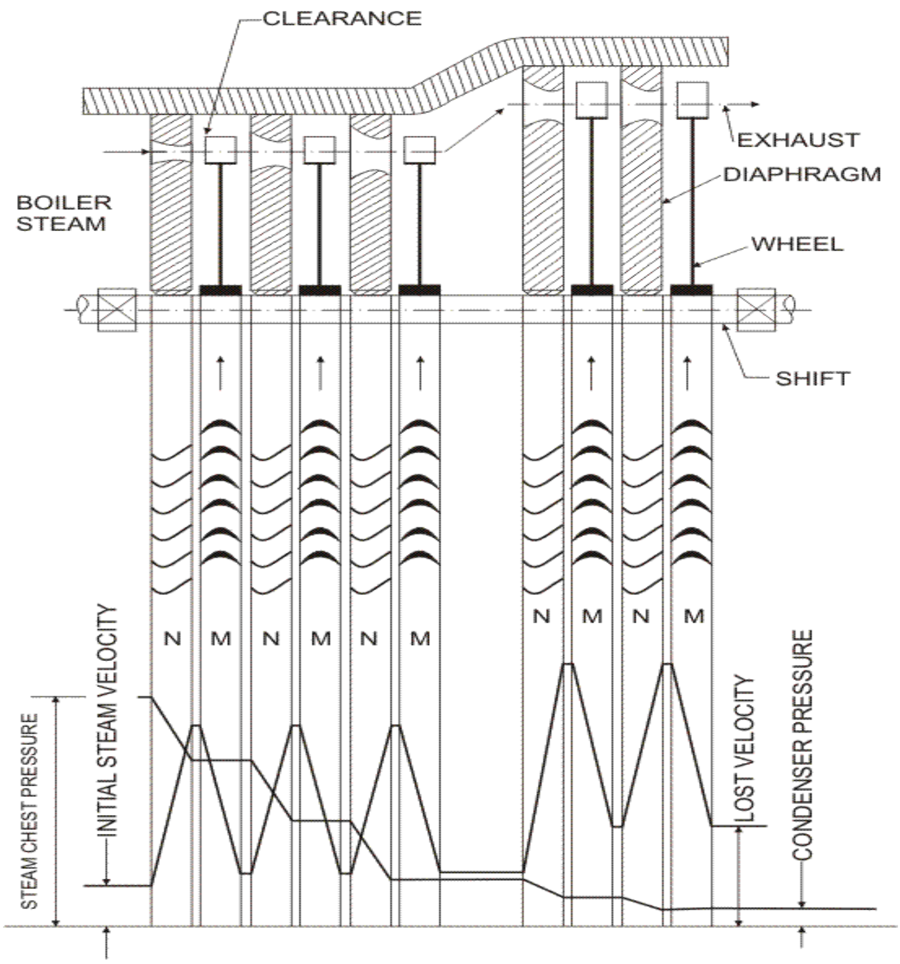


Figure 2.5 Pressure-Compounded Impulse Turbine

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

$$\underbrace{\frac{V_1^2}{2}+h_1}_{exit}=\underbrace{\frac{V_2^2}{2}+h_2}_{inlet} \quad (2.15)$$

$$\eta_N=\frac{V_1^2-\varphi V_2^2}{2(\Delta h)_{isentropic}} \quad (2.16)$$

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. 2.5) 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 2.5 shows a typical velocity diagram for the reaction stage.

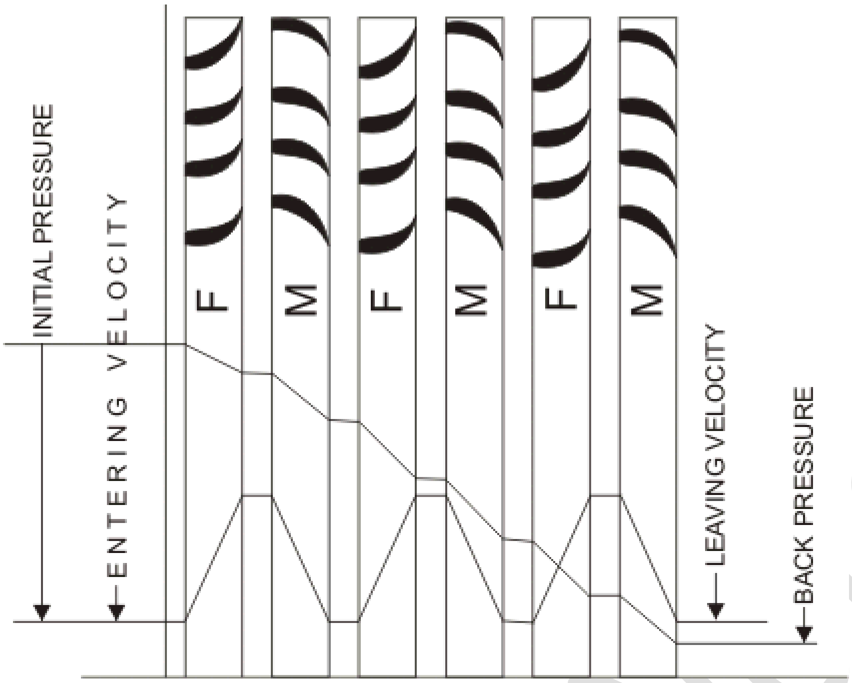


Figure 2.6 Three stages of reaction turbine indicating pressure and velocity distribution

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

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

or,

$$R = \frac{h_1 - h_2}{h_0 - h_1} \qquad (2.17)$$

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.

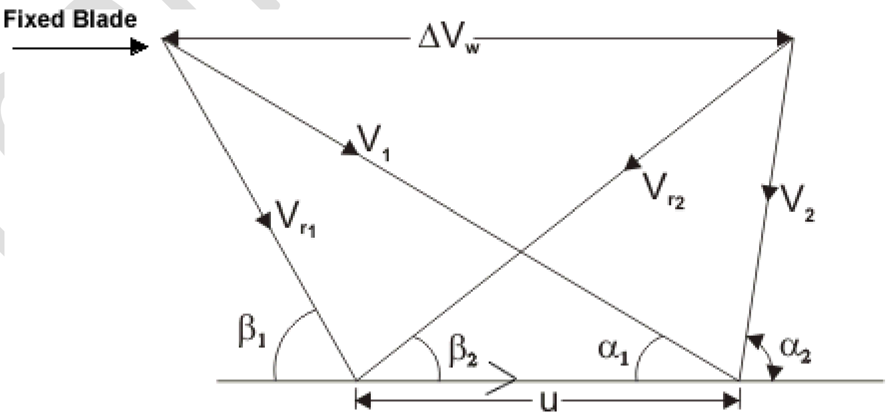


Figure 2.7 The velocity diagram of reaction blading

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)

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

(2.18)

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

$$E = (V_1^2 - U^2 + 2V_1 U \cos\alpha_1)/2 \qquad (2.19)$$

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) $= W = U\Delta V_W$

$$= U(2V_1 \cos\alpha_1 - U) \qquad (2.20)$$

Therefore, the Blade efficiency

$$= \eta_b = \frac{2U(2V_1 \cos\alpha_1 - U)}{V_1^2 - U^2 + 2V_1 U \cos\alpha_1} \qquad (2.21)$$

$$\eta_b = \frac{2\rho(2\cos\alpha_1 - P)}{1 - \rho^2 + 2\rho\cos\alpha_1} \qquad (2.22)$$

For the maximum efficiency

$$-2\rho(2\cos\alpha_1 - \rho)(-2\rho + 2\cos\alpha_1) = 0 \qquad (2.23)$$

from which finally it yields

$$\rho_{opt} = \left(\frac{U}{V_1}\right)_{opt} = \cos\alpha_1 \qquad (2.24)$$

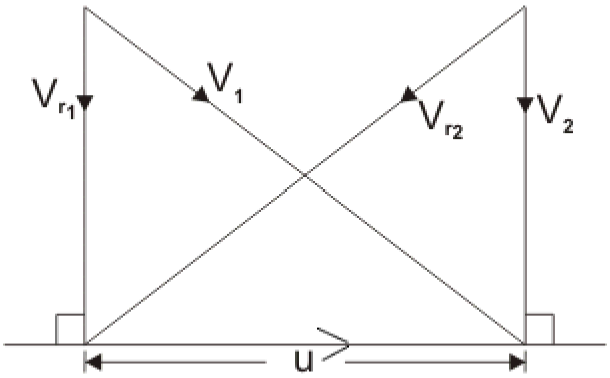


Figure 2.8 Velocity diagram for maximum efficiency

Absolute velocity of the outlet at this stage is axial (see figure 2.8). In this case, the energy transfer

$$E = U\Delta V_W = U^2 \tag{2.25}$$

$$(\eta_b)_{\max} = \frac{2\cos^2\alpha_1}{1+\cos^2\alpha_1} \tag{2.26}$$

$$(\eta_b)_{impulse} = \cos^2\alpha_1 \tag{2.27}$$

ηb 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 2.9

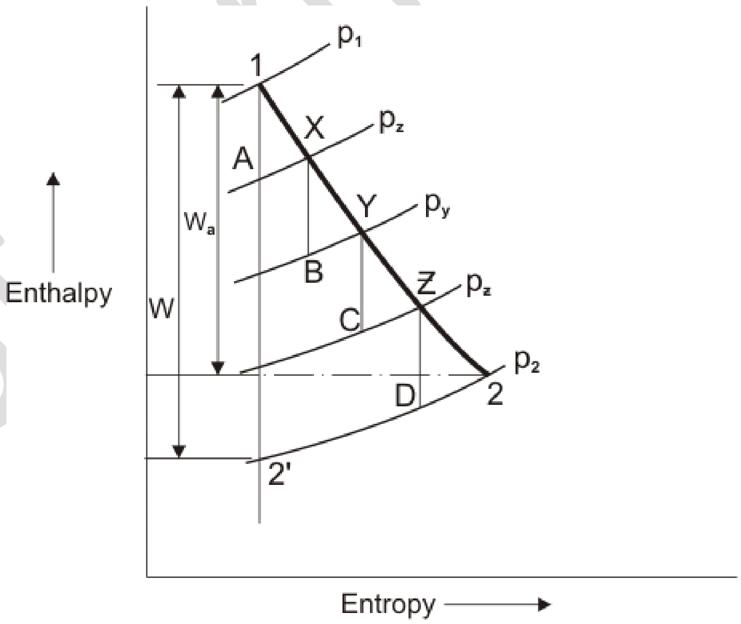


Figure 2.9 Different stage of a steam turbine

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

The overall efficiency of expansion is η0. The actual work during the expansion from 1 to 2 is

or,
$$\frac{P_1}{P_x} = \frac{P_x}{P_y} = \frac{P_y}{P_z} = \frac{P_z}{P_2} \tag{2.28}$$

or,
$$\eta_o = \frac{Wa}{W} = \frac{actual\ enthalpy\ drop\ (1-2)}{isentropic\ heat\ drop\ (1-2')} \tag{2.29}$$

R.F is 1.03 to 1.04

$$R.F = \frac{\Delta h_{1A} + \Delta h_{xB} + \Delta h_{yc} + \Delta h_{zD}}{\Delta h_{12}} \tag{2.30}$$

or,
$$\eta_s = \frac{\Delta h_{1x}}{\Delta h_{1A}} = \frac{\Delta h_{xy}}{\Delta h_{xB}} = \frac{\Delta h_{yz}}{\Delta h_{yc}} = \frac{\Delta h_{z2}}{\Delta h_{zD}} \tag{2.31}$$

We can see:

$$\eta_0 = \eta_s \times R.F \tag{2.32}$$

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

The effect depicted by Eqn (2.32) 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.

Unit 3 - HYDRAULIC TURBINE

IMPULSE TURBINE

Figure 3.1 Typical PELTON WHEEL with 21 Buckets

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

Impulse Hydraulic Turbine **: The Pelton Wheel**

The only hydraulic turbine of the impulse type in common use, is named after an American engineer Laster A Pelton, who contributed much to its development around the year 1880. Therefore this machine is known as Pelton turbine or Pelton wheel. It is an efficient machine particularly suited to high heads. The rotor consists of a large circular disc or wheel on which a number (seldom less than 15) of spoon shaped buckets are spaced uniformly round is periphery as shown in Figure 3.1. The wheel is driven by jets of water being discharged at atmospheric pressure from pressure nozzles. The nozzles are mounted so that each directs a jet along a tangent to the circle through the centres of the buckets (Figure 3.2). Down the centre of each bucket, there is a splitter ridge which divides the jet into two equal streams which flow round the smooth inner surface of the bucket and leaves the bucket with a relative velocity almost opposite in direction to the original jet.

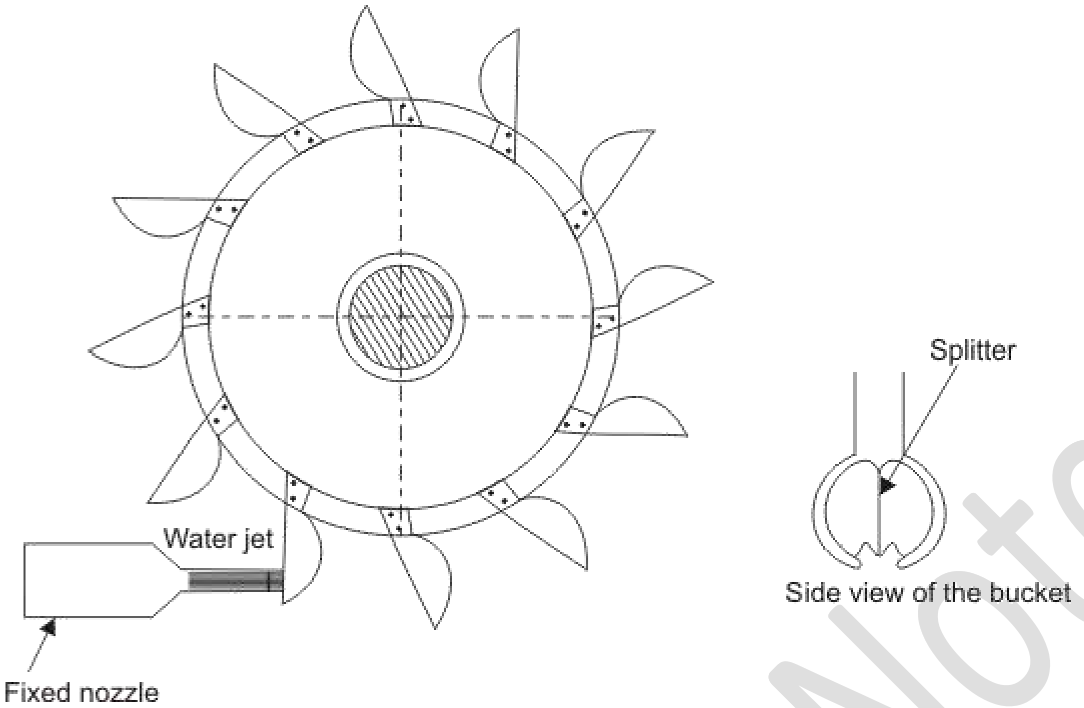


Figure 3.2 A Pelton wheel

For maximum change in momentum of the fluid and hence for the maximum driving force on the wheel, the deflection of the water jet should be 165 degree. In practice, however, the deflection is limited to about 15 degree so that the water leaving a bucket may not hit the back of the following bucket. Therefore, the camber angle of the buckets is made as small as possible.

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

Analysis of force on the bucket and power generation: Figure 26.3a shows a section through a bucket which is being acted on by a jet. The plane of section is parallel to the axis of the wheel and contains the axis of the jet. The absolute velocity of the jet V1 with which it strikes the bucket is given by

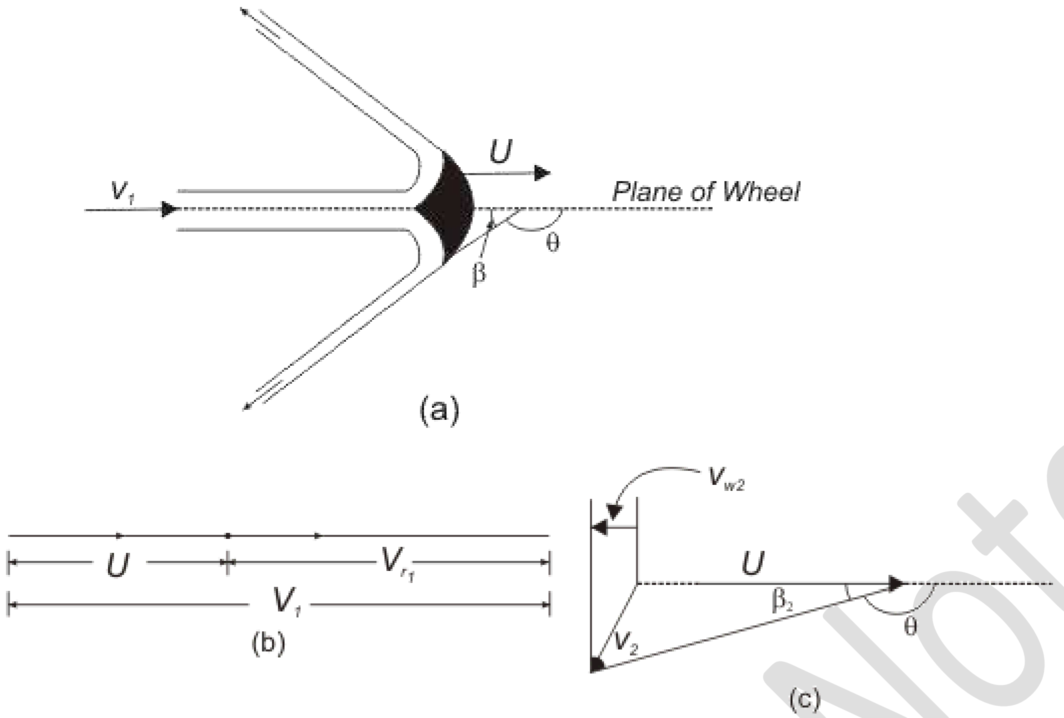


Figure 3.3 (a) Flow along the bucket of a pelton wheel

(b) Inlet velocity triangle

(c) Outlet velocity triangle

where, k is the coefficient of velocity which takes care of the friction in the nozzle. *H* is the head at the entrance to the nozzle which is equal to the total or gross head of water stored at high altitudes minus the head lost due to friction in the long pipeline leading to the nozzle. Let the velocity of the bucket (due to the rotation of the wheel) at its centre where the jet strikes be *U* . Since the jet velocity V1 is tangential, i.e. V1 and *U* are collinear, the diagram of velocity vector at inlet (Fig 26.3.b) becomes simply a straight line and the relative velocity is given by

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

It is assumed that the flow of fluid is uniform and it glides the blade all along including the entrance and exit sections to avoid the unnecessary losses due to shock. Therefore the direction of relative velocity at entrance and exit should match the inlet and outlet angles of the buckets respectively. The velocity triangle at the outlet is shown in Figure 26.3c. The bucket velocity *U* remains the same both at the inlet and outlet. With the direction of *U* being taken as positive, we can write. The tangential component of inlet velocity (Figure 3.3b)

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

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

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

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

The relative velocity Vr1 becomes slightly less than V2 mainly because of the friction in the bucket. Some additional loss is also inevitable as the fluid strikes the splitter ridge, because the ridge cannot have zero thickness. These losses are however kept to a minimum by making the inner surface of the bucket polished and reducing the thickness of the splitter ridge. The relative velocity at outlet is usually expressed as Vr2/Vr1 where, K is a factor with a value less than 1. However in an ideal case (in absence of friction between the fluid and blade surface) K=1. Therefore, we can write Eq.(26.1)

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

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

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

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

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

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

For η to be maximum,

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

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

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

The condition given by Eq. (3.5) states that the efficiency of the wheel in converting the kinetic energy of the jet into mechanical energy of rotation becomes maximum when the wheel speed at the centre of the bucket becomes one half of the incoming velocity of the jet. The overall efficiency η0 will be less than 1 because of friction in bearing and windage, i.e. friction between the wheel and the atmosphere in which it rotates. Moreover, as the losses due to bearing friction and windage increase rapidly with speed, the overall efficiency reaches it peak when the ratio U/V1 is slightly less than the theoretical value of 0.5. The value usually obtained in practice is about 0.46. The Figure 3.4 shows the variation of wheel efficiency with blade to jet speed ratio for assumed values at k=1 and 0.8. An overall efficiency of 85-90 percent may usually be obtained in large machines. To obtain high values of wheel efficiency, the buckets should have smooth surface and be properly designed. The length, width, and depth of the buckets are chosen about 2.5.4 and 0.8 times the jet diameter. The buckets are notched for smooth entry of the jet.

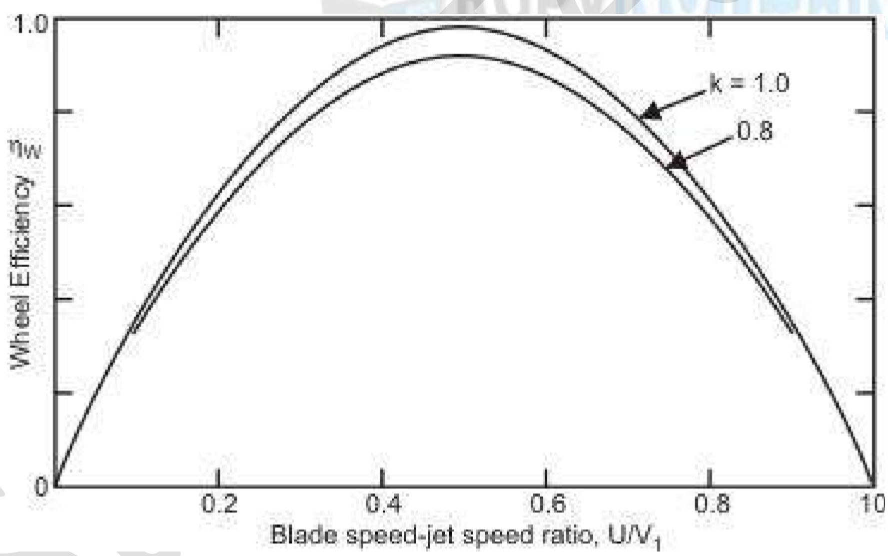


Figure 3.4 Theoretical variation of wheel efficiency for a Pelton turbine with blade speed to j speed ratio for different values of k

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

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

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

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

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

In case of a Pelton turbine, an additional requirement for its operation at the condition of maximum efficiency is that the ration of bucket to initial jet velocity V1 has to be kept at its optimum value of about 0.46. Hence, when U is fixed. U/V1 has to be fixed. Therefore the control must be made by a variation of the cross-sectional area, A, of the jet so that the flow rate changes in proportion to the change in the flow area keeping the jet velocity same. This is usually achieved by a spear valve in the nozzle (Figure 3.5a). Movement of the spear and the axis of the nozzle change the annular area between the spear and the housing. The shape of the spear is such, that the fluid coalesces into a circular jet and then the effect of the spear movement is to vary the diameter of the jet. Deflectors are often used (Figure 3.5b) along with the spear valve to prevent the serious water hammer problem due to a sudden reduction in the rate of flow. These plates temporarily defect the jet so that the entire flow does not reach the bucket; the spear valve may then be moved slowly to its new position to reduce the rate of flow in the pipe-line gradually. If the bucket width is too small in relation to the jet diameter, the fluid is not smoothly deflected by the buckets and, in consequence, much energy is dissipated in turbulence and the efficiency drops considerably. On the other hand, if the buckets are unduly large, the effect of friction on the surfaces is unnecessarily high. The optimum value of the ratio of bucket width to jet diameter has been found to vary between 4 and 5.

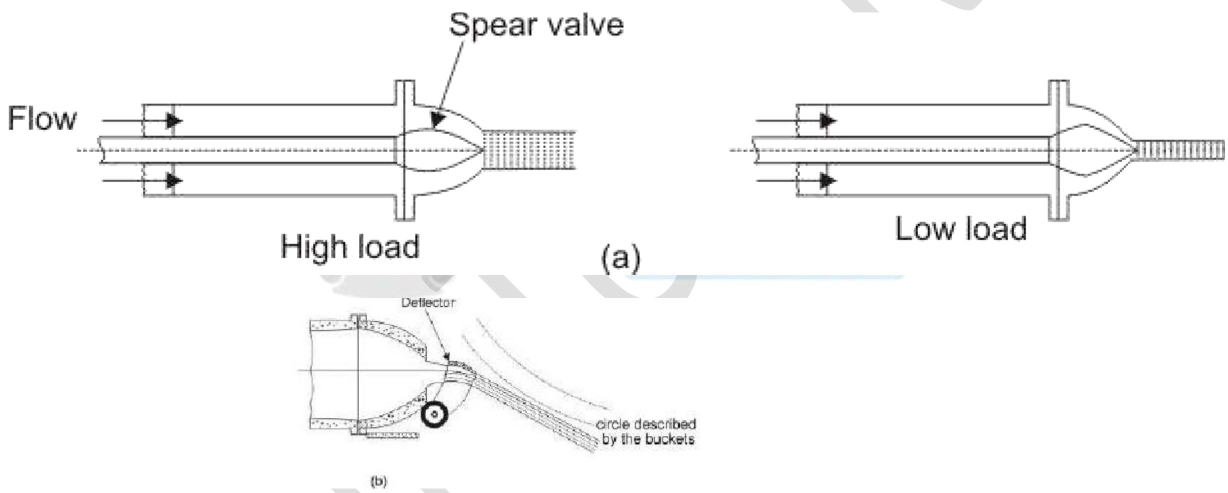


Figure 3.5 (a) Spear valve to alter jet area in a Pelton wheel
(b) Jet deflected from bucket

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

FRANCIS TURBINE

Reaction Turbine: The principal feature of a reaction turbine that distinguishes it from an impulse turbine is that only a part of the total head available at the inlet to the turbine is converted to velocity head, before the runner is reached. Also in the reaction turbines the working fluid, instead of engaging only one or two blades, completely fills the passages in the runner. The pressure or static head of the fluid changes gradually as it passes through the runner along with the change in its kinetic energy based on absolute velocity due to the impulse action between the fluid and the runner. Therefore the cross-sectional area of flow through the passages of the fluid. A reaction turbine is usually well suited for low heads. A radial flow hydraulic turbine of reaction type was first developed by an American Engineer, James B. Francis (1815-92) and is named after him as the Francis turbine. The schematic diagram of a Francis turbine is shown in Fig. 3.6

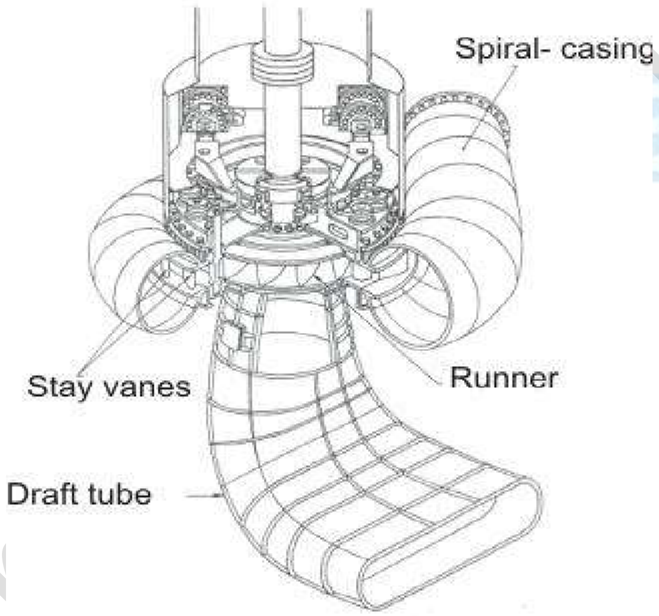


Figure 3.6 A Francis turbine

A Francis turbine comprises mainly the four components:

(i) spiral casing,

(ii) guide on stay vanes,

(iii) runner blades,

(iv) draft-tube as shown in Figure 3.6

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

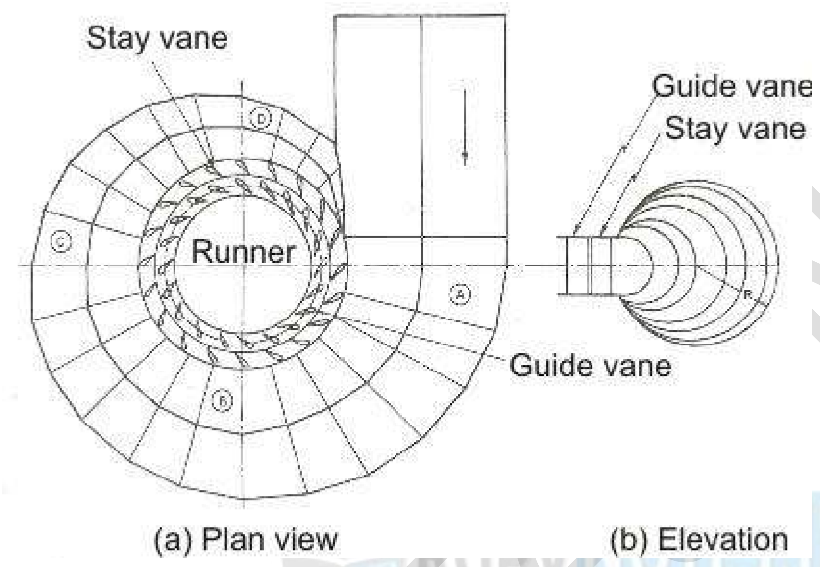


Figure 3.7 Spiral Casing

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

Guide or Stay vane:

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

Draft tube:

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

Net head across a reaction turbine and the purpose to providing a draft tube. The effective head across any turbine is the difference between the head at inlet to the machine and the head at outlet from it. A reaction turbine always runs completely filled with the working fluid. The tube that connects the end of the runner to the tail race is known as a draft tube and should completely to filled with the working fluid flowing through it. The kinetic energy of the fluid finally discharged into the tail race is wasted. A draft tube is made divergent so as to reduce the velocity at outlet to a minimum. Therefore a draft tube is basically a diffuser and should be designed properly with the angle between the walls of the tube to be limited to about 8 degree so as to prevent the flow separation from the wall and to reduce accordingly the loss of energy in the tube. Figure 3.8 shows a flow diagram from the reservoir via a reaction turbine to the tail race.

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

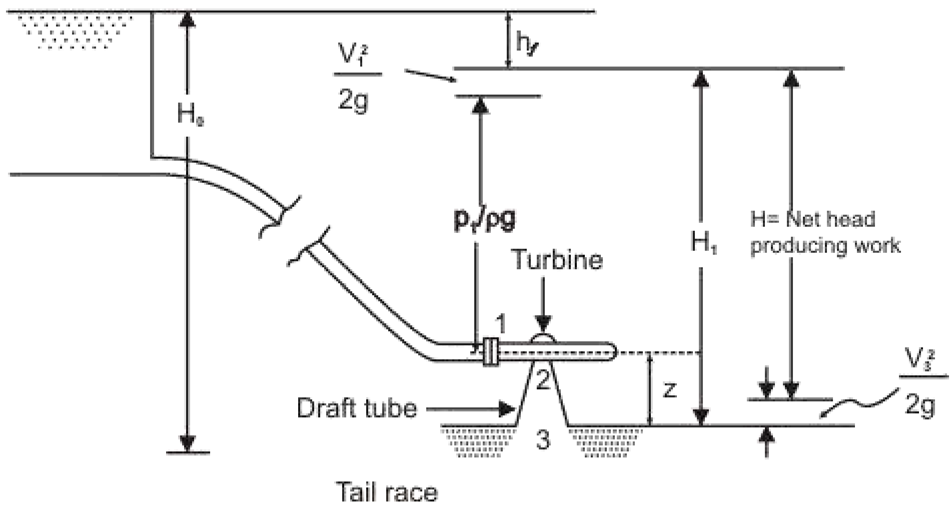


Figure 3.8 Head across a reaction turbine

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

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

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

The pressures are defined in terms of their values above the atmospheric pressure. Section 2 and 3 in Figure 3.8 represent the exits from the runner and the draft tube respectively. If the losses in the draft tube are neglected, then the total head at 2 becomes equal to that at 3. Therefore, the net head across the machine is either H0 or H. Applying the Bernoulli's equation between 2 and 3 in consideration of flow, without losses, through the draft tube, we can write.

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

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

Since , both the terms in the bracket are positive and hence P is always negative, which implies that the static pressure at the outlet of the runner is always below the atmospheric pressure. Equation (3.12) also shows that the value of the suction pressure at runner outlet depends on z, the height of the runner above the tail race. The value of this minimum pressure P2 should never fall below the vapor pressure of the liquid at its operating temperature to avoid the problem of cavitation. Therefore, we fine that the incorporation of a draft tube allows the turbine runner to be set above the tail race without any drop of available head by maintaining a vacuum pressure at the outlet of the runner.

Runner of the Francis Turbine

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

Figure 3.9 shows the velocity triangles at inlet and outlet of a typical blade of a Francis turbine. Usually the flow velocity (velocity perpendicular to the tangential direction) remains constant throughout, i.e. equal to that at the inlet to the draft tube.

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

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

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

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

and

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

After simplification

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

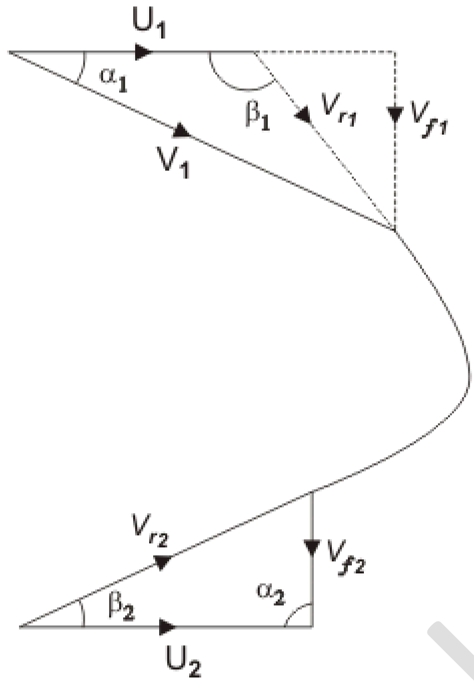


Figure 3.9 Velocity triangle for a Francis runner

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

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

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

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

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

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

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

KAPLAN TURBINE

Introduction

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

Figure 3.10 A typical Kaplan Turbine

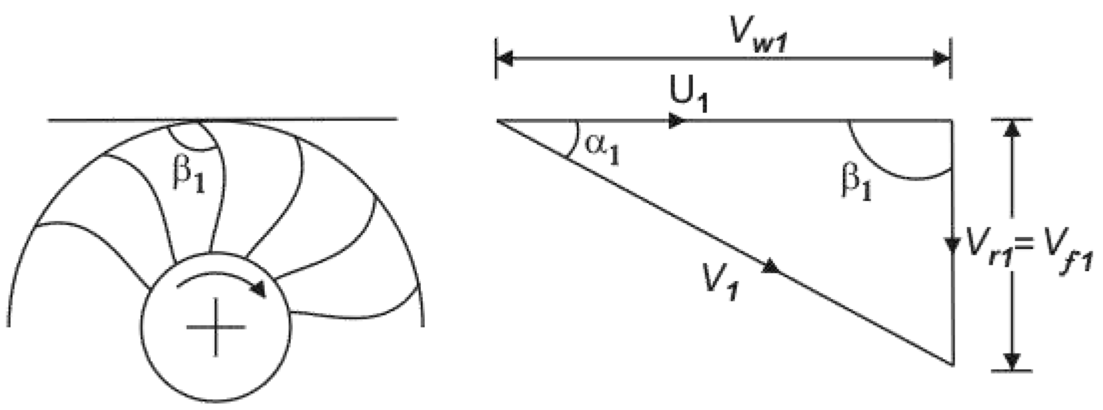
Development of Kaplan Runner from the Change in the Shape of Francis Runner with Specific Speed

Figure 3.10 shows in stages the change in the shape of a Francis runner with the variation of specific speed. The first three types [Fig.3.11 (a), (b) and (c)] have, in order. The Francis runner (radial flow runner) at low, normal and high specific speeds. As the specific speed increases, discharge becomes more and more axial. The fourth type, as shown in Fig.3.11 (d), is a mixed flow runner (radial flow at inlet axial flow at outlet) and is known as Dubs runner which is mainly suited for high specific speeds. Figure 3.11(e) shows a propeller type runner with a less number of blades where the flow is entirely axial (both at inlet and outlet). This type of runner is the most suitable one for very high specific speeds and is known as Kaplan runner or axial flow runner.

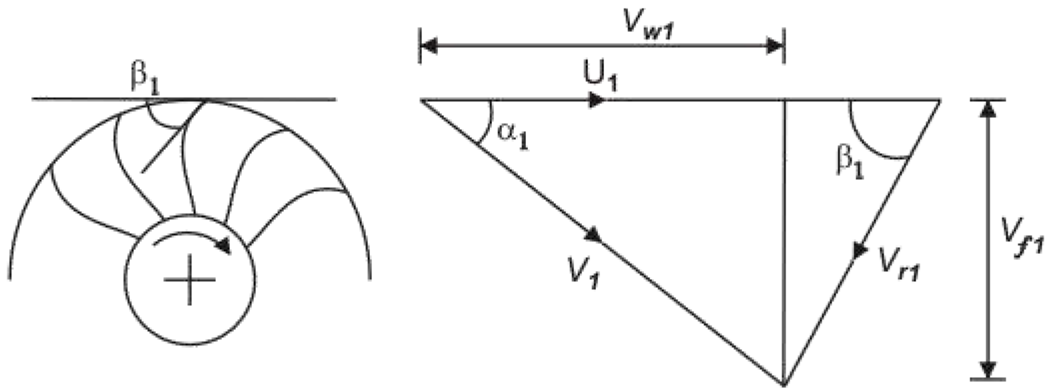
From the inlet velocity triangle for each of the five runners, as shown in Figs (3.11a to 3.11e), it is found that an increase in specific speed (or a decreased in head) is accompanied by a reduction in inlet velocity. But the flow velocity at inlet increases allowing a large amount of fluid to enter the turbine. The most important point to be noted in this context is that the flow at inlet to all the runners, except the Kaplan one, is in radial and tangential directions. Therefore, the inlet velocity triangles of those turbines (Figure 3.11a to 3.11d) are shown in a plane containing the radial ant tangential directions, and hence the flow velocity *represents* the radial component of velocity.

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

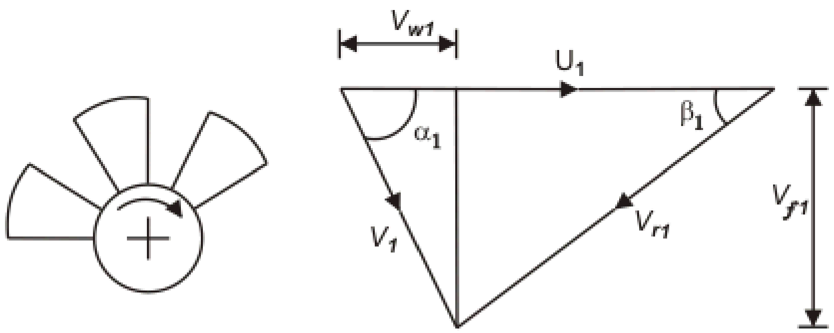
(a) Francis runner for low specific speeds



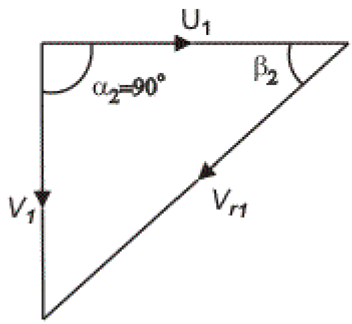
(b) Francis runner for normal specific speeds



(c) Francis runner for high specific speeds



(d) Kalpan runner



(f) Overall Velocity triangle

Figure 3.10 Velocity Triangles for Kaplan Turbine

Figure 3.11 shows a schematic diagram of propeller or Kaplan turbine. The function of the guide vane is same as in case of Francis turbine. Between the guide vanes and the runner, the fluid in a propeller turbine turns through a right-angle into the axial direction and then passes through the runner. The runner usually has four or six blades and closely resembles a ship's propeller. Neglecting the frictional effects, the flow approaching the runner blades can be considered to be a free vortex with whirl velocity being inversely proportional to radius, while on the other hand, the blade velocity is directly proportional to the radius. To take care of this different relationship of the fluid velocity and the blade velocity with the changes in radius, the blades are twisted. The angle with axis is greater at the tip that at the root.

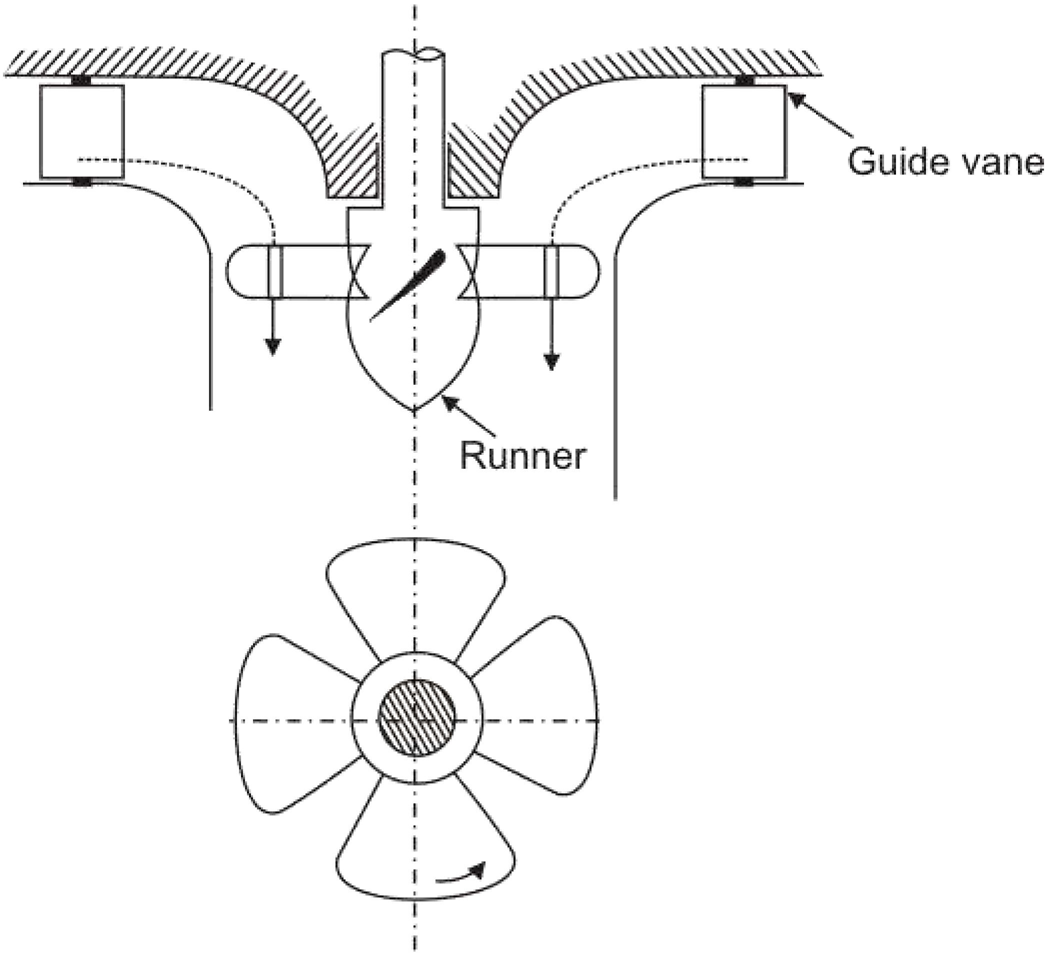


Fig. 3.11 A propeller of Kaplan turbine

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

Straight divergent tube (Fig. 3.12(a) The shape of this tube is that of frustum of a cone. It is usually employed for low specific speed, vertical shaft Francis turbine. The cone angle is restricted to 8 0 to avoid the losses due to separation. The tube must discharge sufficiently low under tail water level. The maximum efficiency of this type of draft tube is 90%. This type of draft tube improves speed regulation of falling load.

Simple elbow type (Fig. 3.12) The vertical length of the draft tube should be made small in order to keep down the cost of excavation, particularly in rock. The exit diameter of draft tube should be as large as possible to recover kinetic energy at runner's outlet. The cone angle of the tube is again fixed from the consideration of losses due to flow separation. Therefore, the draft tube must be bent to keep its definite length. Simple elbow type draft tube will serve such a purpose. Its efficiency is, however, low(about 60%). This type of draft tube turns the water from the vertical to the horizontal direction with a minimum depth of excavation. Sometimes, the transition from a circular section in the vertical portion to a rectangular section in the horizontal part (Fig. 30.4c) is incorporated in the design to have a higher efficiency of the draft tube. The horizontal portion of the draft tube is generally inclined upwards to lead the water gradually to the level of the tail race and to prevent entry of air from the exit end.

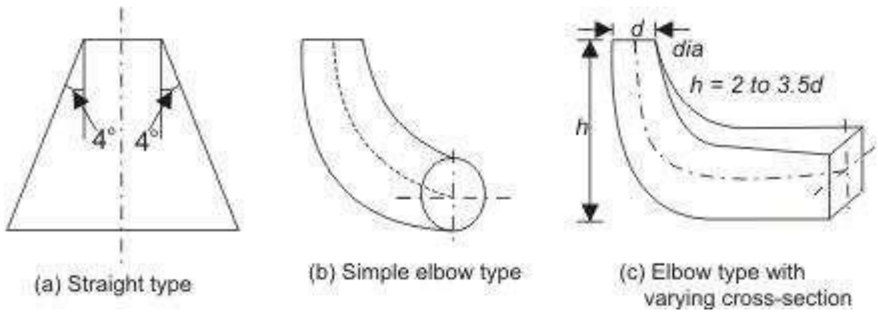


Figure 3.12 Different types of draft tubes

Performance Characteristics of Reaction Turbine

It is not always possible in practice, although desirable, to run a machine at its maximum efficiency due to changes in operating parameters. Therefore, it becomes important to know the performance of the machine under conditions for which the efficiency is less than the maximum. It is more useful to plot the basic dimensionless performance parameters (Fig. 3.13) as derived earlier from the similarity principles of fluid machines. Thus one set of curves, as shown in Fig. 3.12, is applicable not just to the conditions of the test, but to any machine in the same homologous series under any altered conditions.

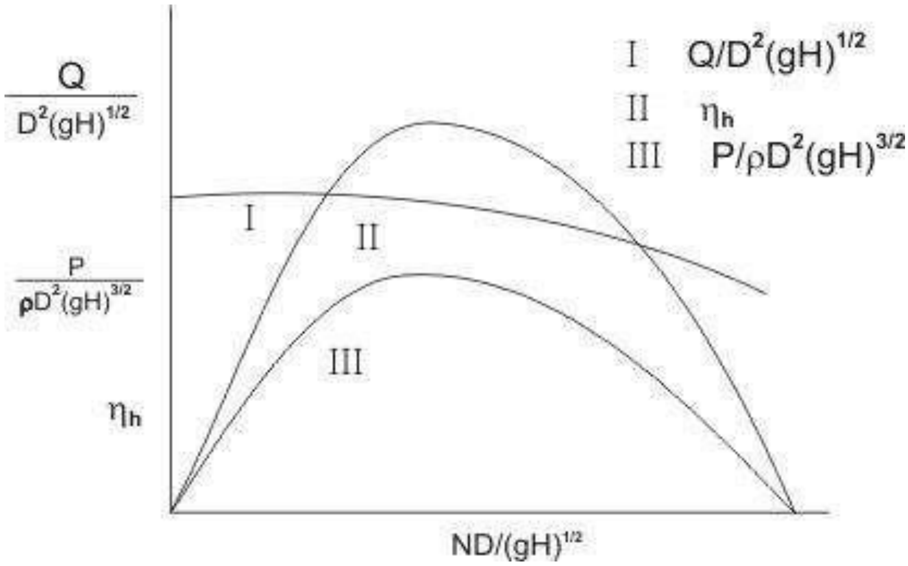


Figure 3.13 performance characteristics of a reaction turbine (in dimensionless parameters)

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

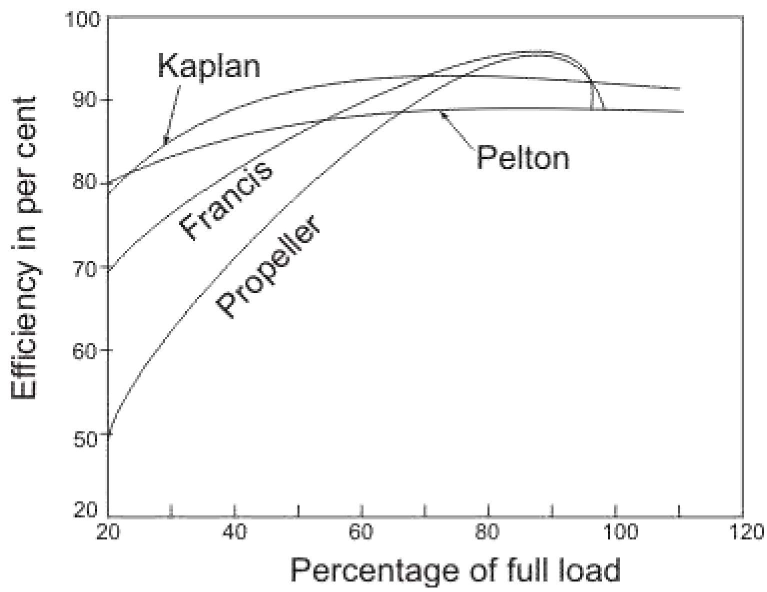


Figure 3.14 Variation of efficiency with load

Comparison of Specific Speeds of Hydraulic Turbines

Specific speeds and their ranges of variation for different types of hydraulic turbines have already been discussed earlier. Figure 3.15 shows the variation of efficiencies with the dimensionless specific speed of different hydraulic turbines. The choice of a hydraulic turbine for a given purpose depends upon the matching of its specific speed corresponding to maximum efficiency with the required specific speed determined from the operating parameters, namely, *N* (rotational speed), *p* (power) and *H* (available head).

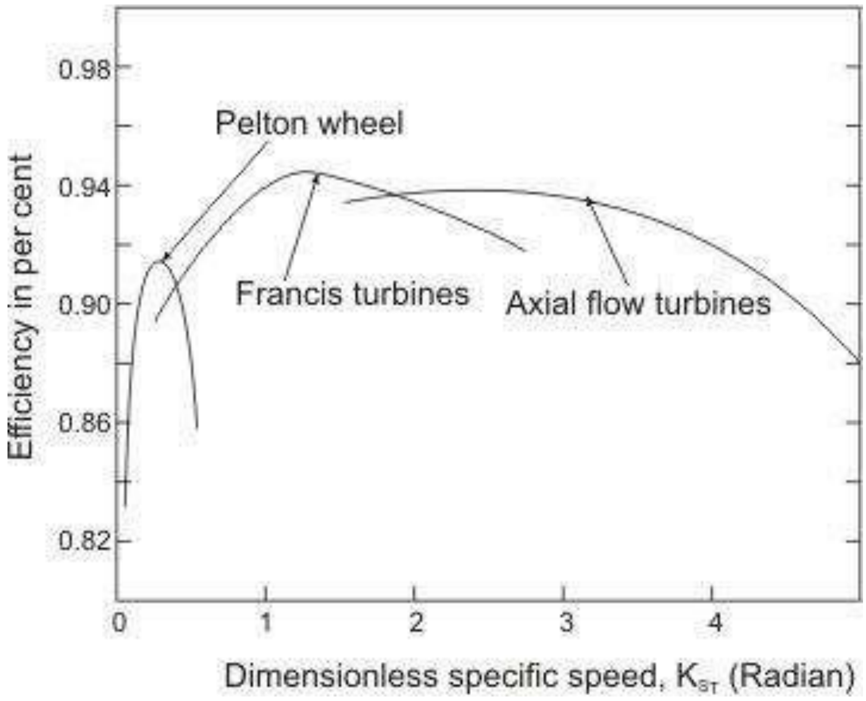


Figure 3.15 Variation of efficiency with specific speed for hydraulic turbines

Governing of Reaction Turbines Governing of reaction turbines is usually done by altering the position of the guide vanes and thus controlling the flow rate by changing the gate openings to the runner. The guide blades of a reaction turbine (Figure 3.16) are pivoted and connected by levers and links to the regulating ring. Two long regulating rods, being attached to the regulating ring at their one ends, are connected to a regulating lever at their other ends. The regulating lever is keyed to a regulating shaft which is turned by a servomotor piston of the oil.

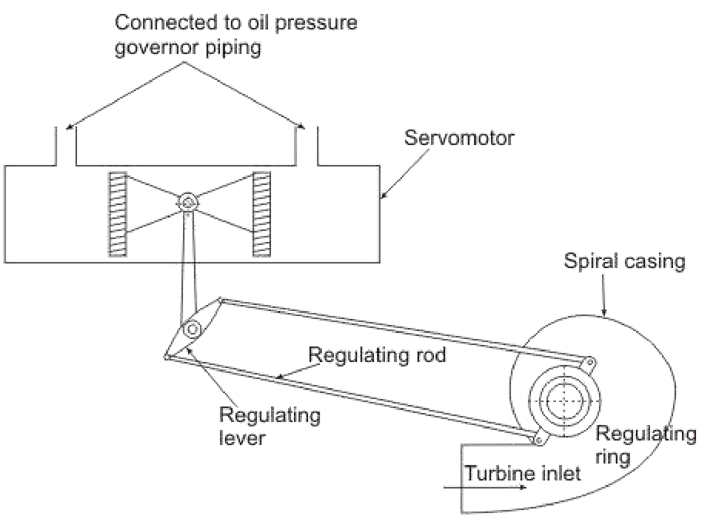


Figure 3.16 Governing of reaction turbine

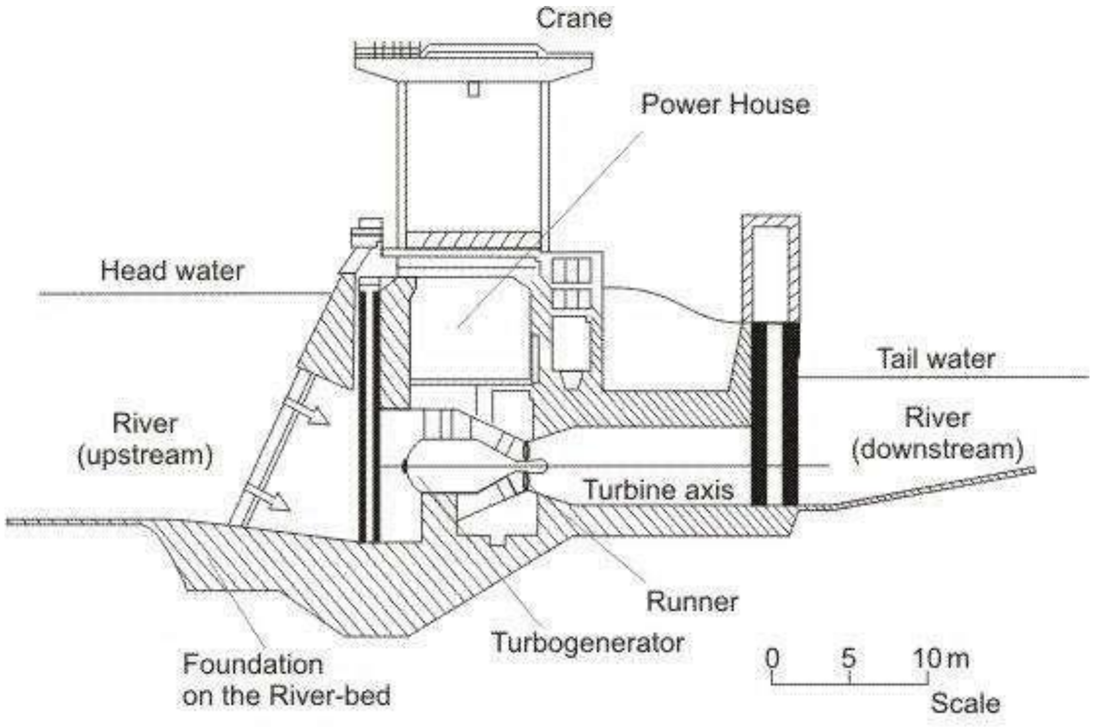


Figure 3.17 Installation of a Francis Turbine

PUMP

Rotodynamic Pumps

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

Centrifugal Pumps

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

General Pumping System and the Net Head Developed by a Pump

The word pumping, referred to a hydraulic system commonly implies to convey liquid from a low to a high reservoir. Such a pumping system, in general, is shown in Fig. 3.18. At any point in the system, the elevation or potential head is measured from a fixed reference datum line. The total head at any point comprises pressure head, velocity head and elevation head. For the lower reservoir, the total head at the free surface is H and is equal to the elevation of the free surface above the datum line since the velocity and static pressure at *A* are zero. Similarly the total head at the free surface in the higher reservoir is (Hm) and is equal to the elevation of the free surface of the reservoir above the reference datum.

The variation of total head as the liquid flows through the system is shown in Fig. 3.18. The liquid enters the intake pipe causing a head loss for which the total energy line drops to point *B* corresponding to a location just after the entrance to intake pipe.

As the fluid flows from the intake to the inlet flange of the pump at elevation, the total head drops further to the point C (Figure 3.18) due to pipe friction and other losses equivalent to hf . The fluid then enters the pump and gains energy imparted by the moving rotor of the pump. This raises the total head of the fluid to a point D (Figure 3.18) at the pump outlet (Figure 3.18).

In course of flow from the pump outlet to the upper reservoir, friction and other losses account for a total head loss or hf down to a point *E.* At *E* an exit loss He occurs when the liquid enters the upper reservoir, bringing the total heat at point *F* (Figure 3.18) to that at the free surface of the upper reservoir. If the total heads are measured at the inlet and outlet flanges respectively, as done in a standard pump test, then

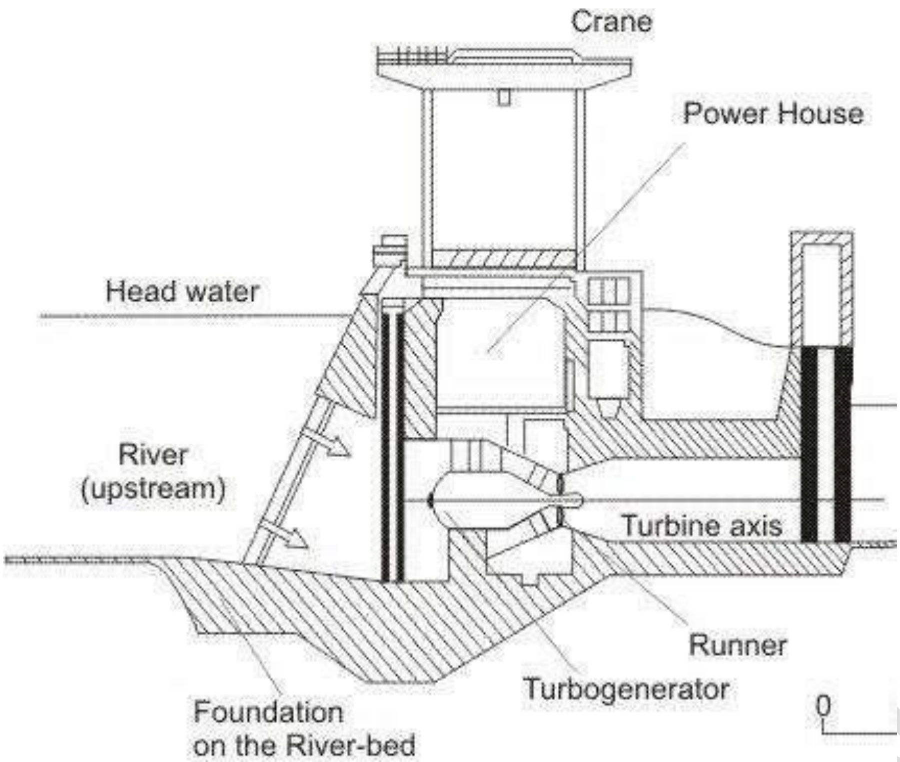


Figure 3.18 A general pumping system

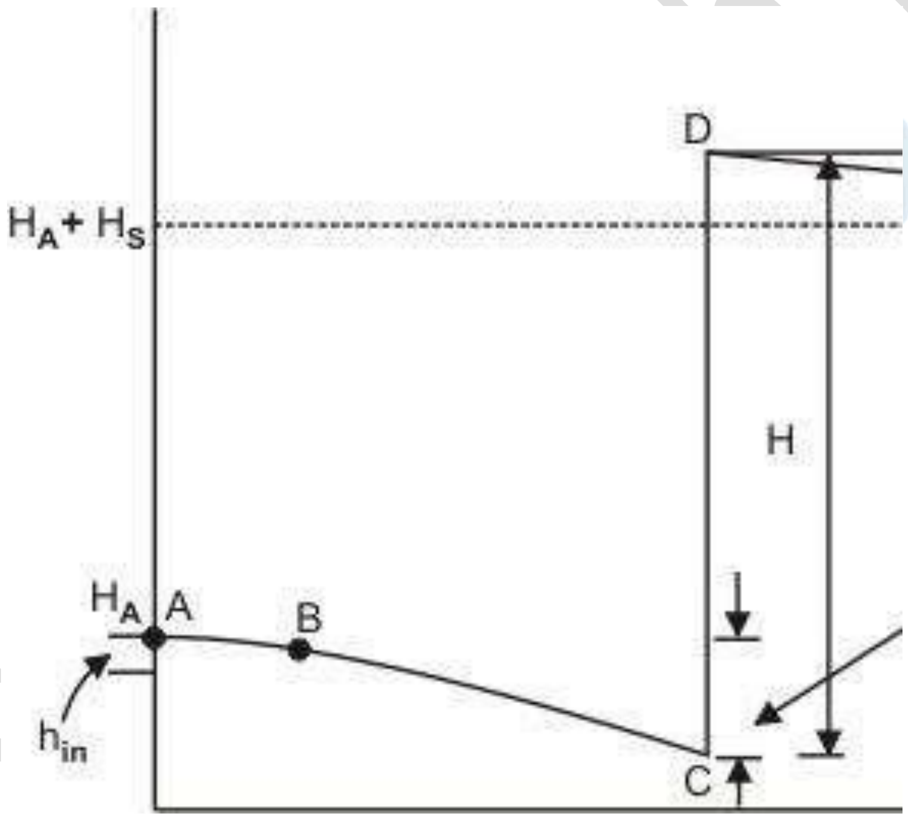


Figure 3.19 Change of head in a pumping system

Total inlet head to the pump = H_A

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

The head developed H is termed as manometric head.

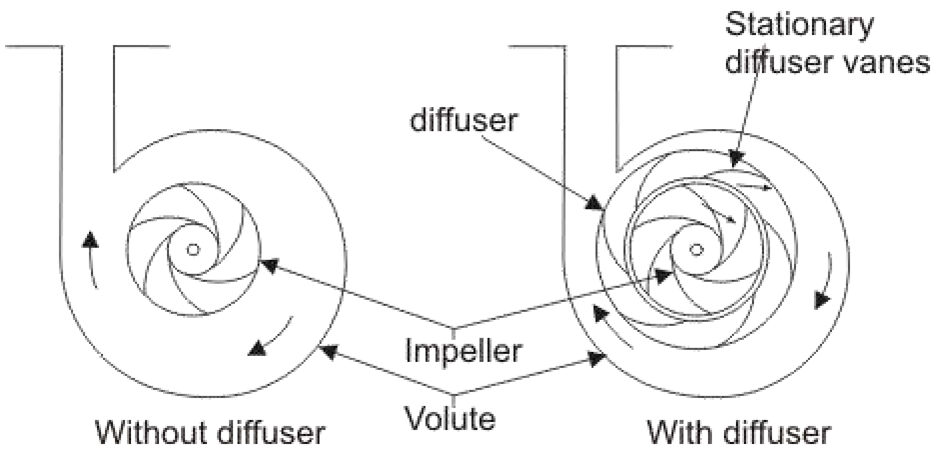
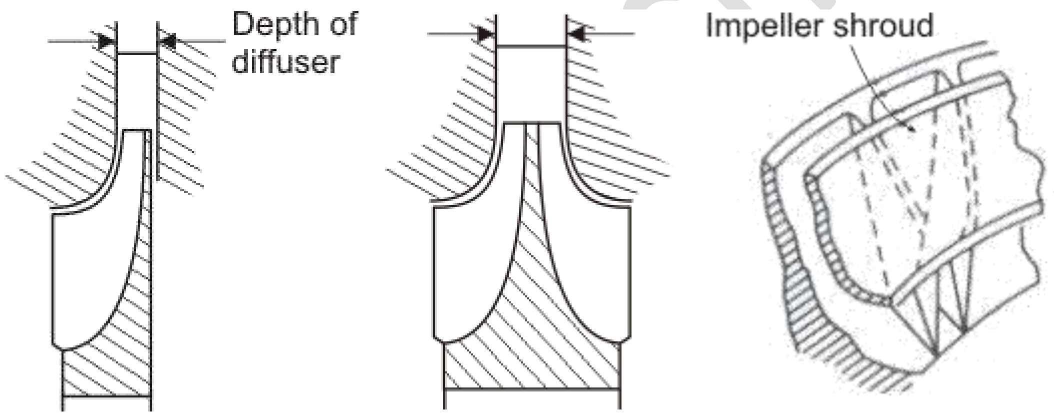


Figure 3.20 A centrifugal pump

The tips of the blades are sometimes covered by another flat disc to give shrouded blades (Figure 3.21c), otherwise the blade tips are left open and the casing of the pump itself forms the solid outer wall of the blade passages. The advantage of the shrouded blade is that flow is prevented from leaking across the blade tips from one passage to another.



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

Figure 3.21 Types of impellers in a centrifugal pump

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

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

Figure 3.21 shows an impeller of a centrifugal pump with the velocity triangles drawn at inlet and outlet. The blades are curved between the inlet and outlet radius. A particle of fluid moves along the broken curve shown in Figure 3.22.

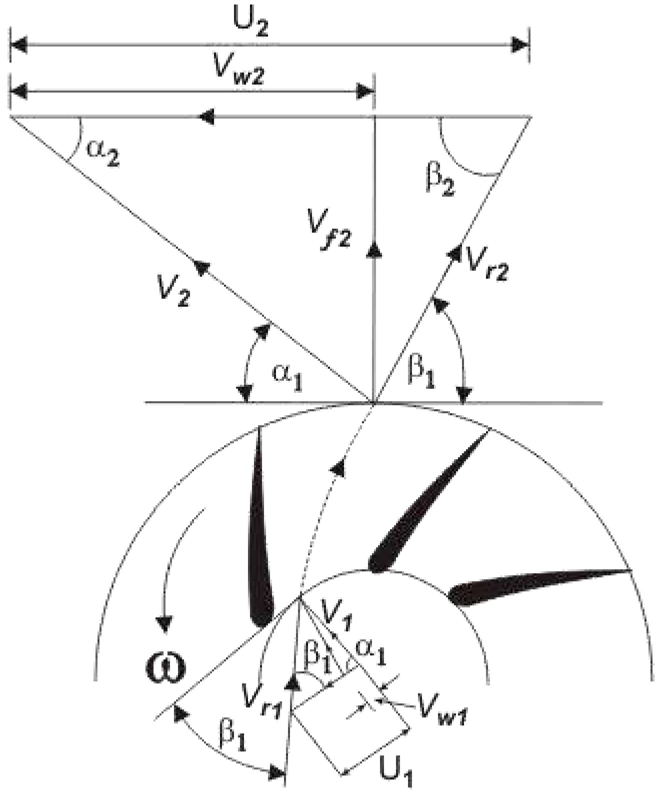


Figure 3.23 Velocity triangles for centrifugal pump Impeller

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

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

A centrifugal pump rarely has any sort of guide vanes at inlet. The fluid therefore approaches the impeller without appreciable whirl and so the inlet angle of the blades is designed to produce a right-angled velocity triangle at inlet (as shown in Fig. 3.23). At conditions other than those for which the impeller was designed, the direction of relative velocity does not coincide with that of a blade. Consequently, the fluid changes direction abruptly on entering the impeller. In addition, the eddies give rise to some back flow into the inlet pipe, thus causing fluid to have some whirl before entering the impeller. However, considering the operation under design conditions, the inlet whirl velocity Vw and accordingly the inlet angular momentum of the fluid entering the impeller is set to zero. Therefore, Eq. (3.17) can be written as

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

We see from this equation that the work done is independent of the inlet radius. The difference in total head across the pump known as manometric head, is always less than the quantity because of the energy dissipated in eddies due to friction.

The ratio of manometric head *H* and the work head imparted by the rotor on the fluid (usually known as Euler head) is termed as manometric efficiency . It represents the effectiveness of the pump in increasing the total energy of the fluid from the energy given to it by the impeller.

Slip Factor

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

In course of flow through the impeller passage, there occurs a difference in pressure and velocity between the leading and trailing faces of the impeller blades. On the leading face of a blade there is relatively a high pressure and low velocity, while on the trailing face, the pressure is lower and hence the velocity is higher. This results in a circulation around the blade and a non-uniform velocity distribution at any radius. The mean direction of flow at outlet, under this situation, changes from the blade angle at outlet to a different angle as shown in Figure 3.23 Therefore the tangential velocity component at outlet is reduced and the difference is defined as the slip.

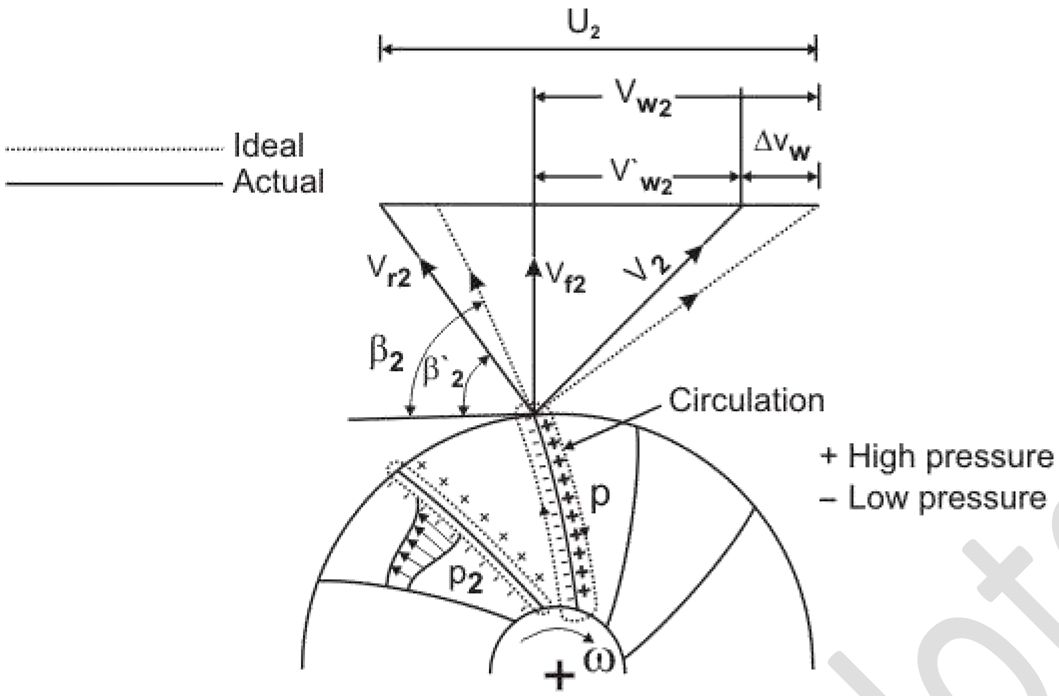


Figure 3.24 Slip and velocity in the impeller blade passage of a centrifugal pump

Losses in a Centrifugal Pump

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

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

Leakage and recirculation power loss. This is due to loss of liquid from the pump and recirculation of the liquid in the impeller. The pressure difference between impeller tip and eye can cause a recirculation of a small volume of liquid, thus reducing the flow rate at outlet of the impeller as shown in Fig. (3.24).

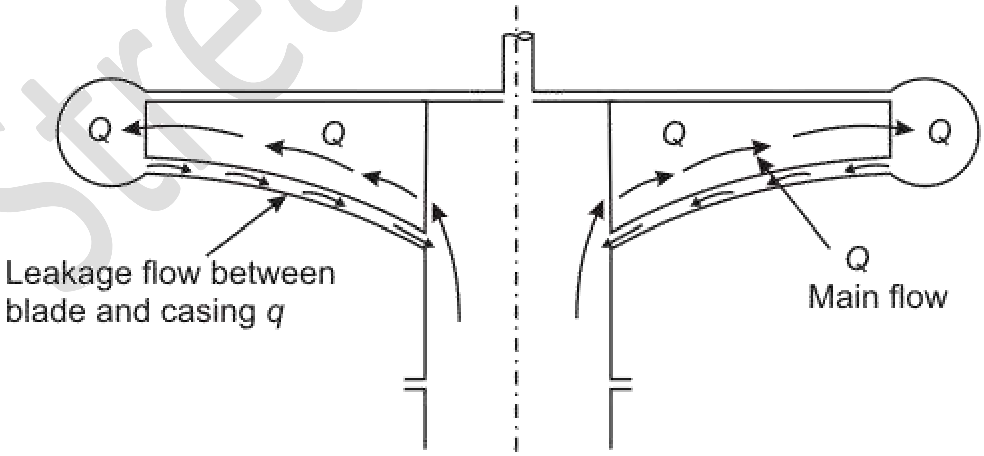


Figure 3.25 Leakage and recirculation in a centrifugal pump

Effect of blade outlet angle

The head-discharge characteristic of a centrifugal pump depends (among other things) on the outlet angle of the impeller blades which in turn depends on blade settings. Three types of blade settings are possible (i) the forward facing for which the blade curvature is in the direction of rotation and, therefore, (Fig. 3.26a), (ii) radial (Fig. 3.26b), and (iii) backward facing for which the blade curvature is in a direction opposite to that of the impeller rotation (Fig. 3.26c). The outlet velocity triangles for all the cases are also shown in Figs. 3.26a, 3.26b, 3.26c. Which was expressed earlier by Eq. (3.18)

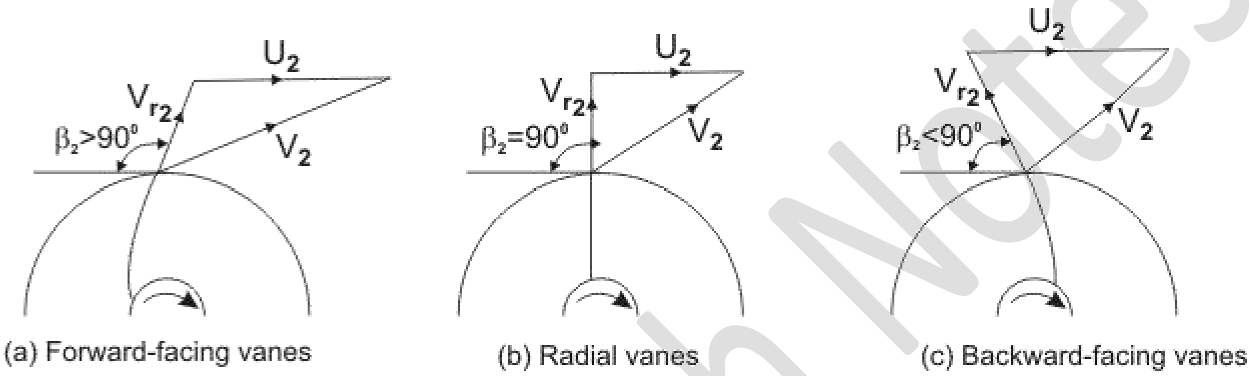


Figure 3.26 Outlet velocity triangles for different blade settings in a centrifugal pump

With the incorporation of above conditions, the relationship of head and discharge for three cases are shown in Figure 3.25. These curves ultimately revert to their more recognized shapes as the actual head-discharge characteristics respectively after consideration of all the losses as explained earlier Figure 3.26

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

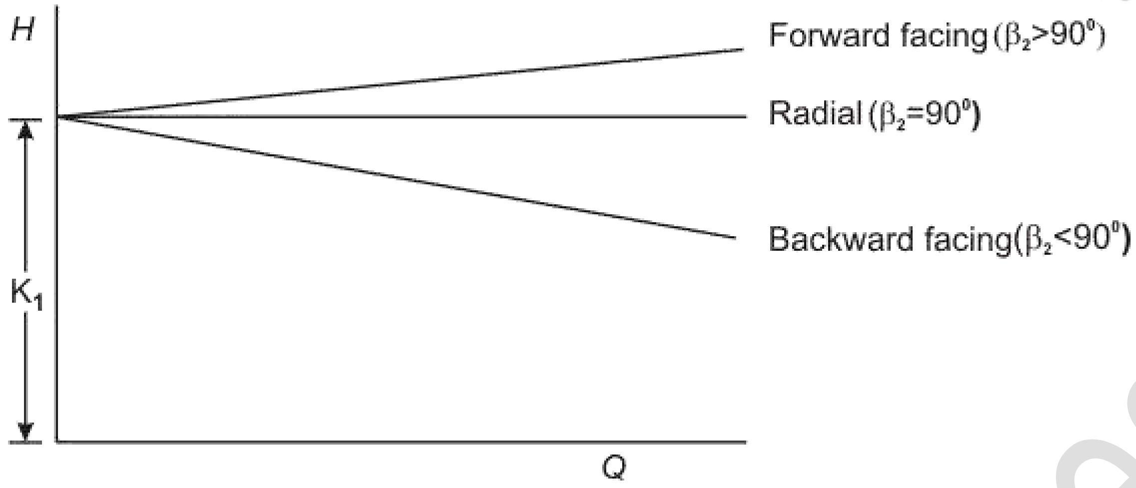


Figure 3.27 Theoretical head-discharge characteristic curves of a centrifugal pump for different blade settings

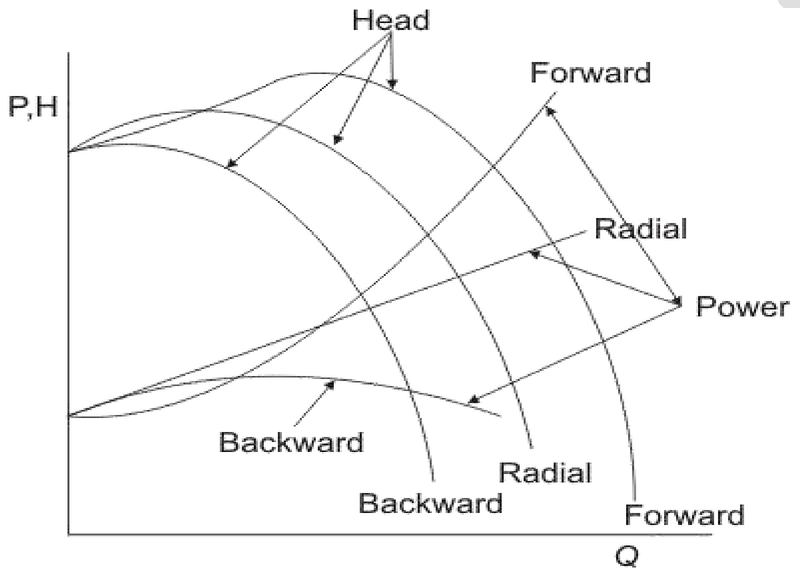


Figure 3.28 Actual head-discharge and power-discharge characteristic curves of a centrifugal pump

Unit 4 - ROTARY COMPRESSORS, FAN AND BLOWER

COMPRESSORS

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 changes 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.

Classification based on pressure rise

A centrifugal compressor essentially consists of three components.

1. A **stationary casing**
2. A **rotating impeller** as shown in Fig. 4.1 (a) which imparts a high velocity to the air. The impeller may be single or double sided as show in Fig. 4.1 (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.

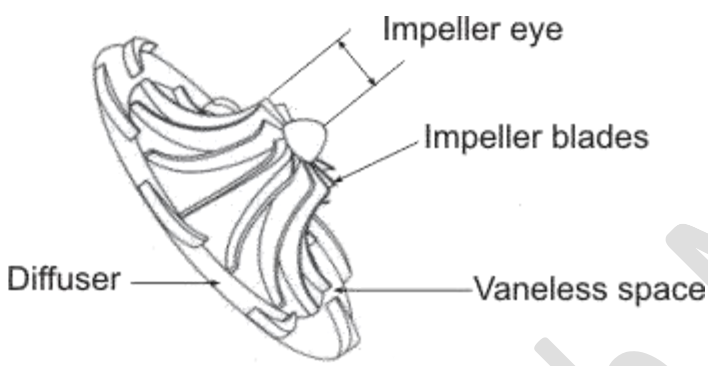
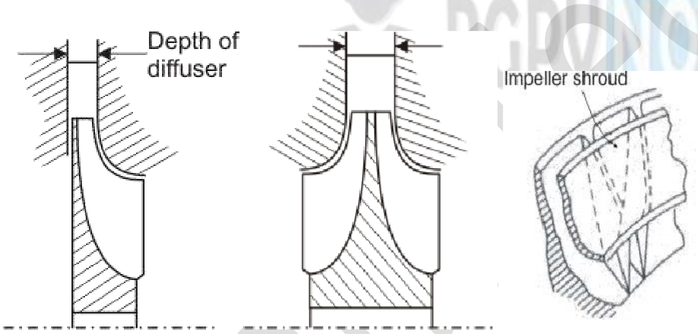


Figure 4.1(a)



(b) (c) (d)

Figure 4.1 Schematic views of a centrifugal compressor

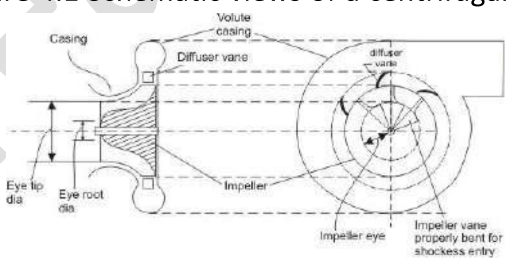


Figure 4.2 Single entry and single outlet centrifugal compressor

Figure 4.2 is the schematic of a centrifugal compressor, where a single entry radial impeller is housed inside a volute casing.

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 result 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 4.1(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 compressor.

(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.

CENTRIFUGAL AND AXIAL FLOW 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 Vr, as shown in Fig. 4.3. The air leaves the impeller tip with an absolute velocity of V2 that will have a tangential or whirl component Vw .Under ideal conditions Vw, would be such that the whirl component is equal to the impeller speed V1 at the tip. Since air enters the impeller in axial direction, α.

Centrifugal Blowers Vane shape

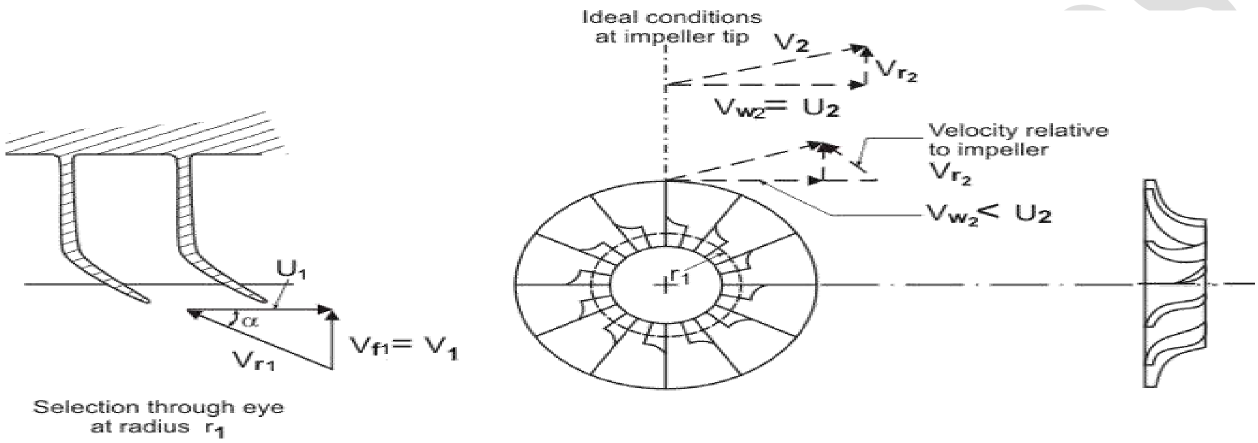


Figure 4.3 Velocity triangles at inlet and outlet of impeller blades

The energy transfer per unit mass of air is given as

$$\frac{E}{m} = U_2^2 \qquad (4.1)$$

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 Vw2 < U2. 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

$$\frac{E}{m} = V_{w2} U_2 = \sigma U_2^2 \quad (4.2)$$

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

$$w = \Psi \sigma U_2^2 \quad (4.3)$$

Where *w* 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

$$w = c_p (T_{0_2} - T_{0_1}) \quad (4.4)$$

where T_{01} and T_{02} are the stagnation temperatures at inlet and outlet of the impeller, and Cp is the mean specific heat over the entire temperature range. With the help of Eq. (4.3), we can write

$$w = \Psi \sigma U_2^2 = c_p (T_{0_2} - T_{0_1}) \quad (4.5)$$

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 T03, Then One can write from Eqn. (4.5)

$$\frac{T_{0_2}}{T_{0_1}} = \frac{T_{0_3}}{T_{0_1}} = 1 + \frac{\Psi \sigma U_2^2}{c_p T_{0_1}} \quad (4.6)$$

The overall stagnation pressure ratio can be written as

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

$$= \left[1 + \frac{\eta_c (T_{0_3} - T_{0_1})}{T_{0_1}}\right]^{\frac{\gamma}{\gamma-1}} \qquad (4.7)$$

where, T03 and T03s are the stagnation temperatures at the end of an ideal (isentropic) and actual process of compression respectively (Figure 7.1), and ηc is the isentropic efficiency defined as

$$\eta_c = \frac{T_{0_{3s}} - T_{0_1}}{T_{0_3} - T_{0_1}} \qquad (4.8)$$

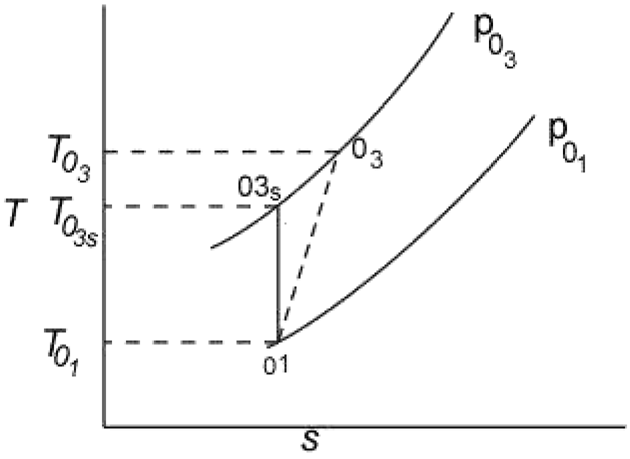


Figure 4.4 Ideal and actual processes of compression on T-s plane

Since the stagnation temperature at the outlet of impeller is same as that at the outlet of the diffuser, one can also write T03s in place of T03 in Eq. (4.8). Typical values of the power input factor lie in the region of 1.035 to 1.04. If we know T03 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 4.4. For common material, it is limited to 450 m/s.

Figure 4.5 shows the inducing section of a compressor. The relative velocity Vr at the eye tip has to be held low otherwise the Mach umber given by M will be too high causing shock losses.

Mach number M 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 4.7(a) and (b) respectively.

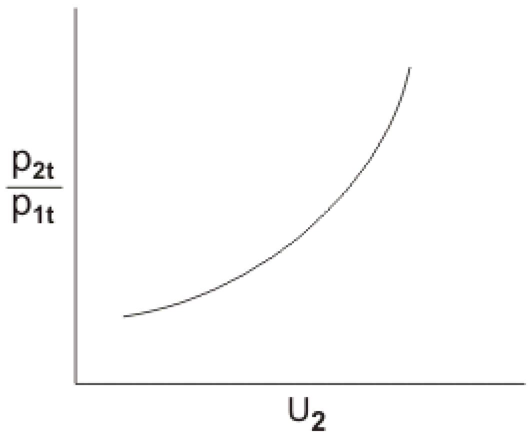


Figure 4.5 Variation in stagnation pressure ratio with impeller tip speed

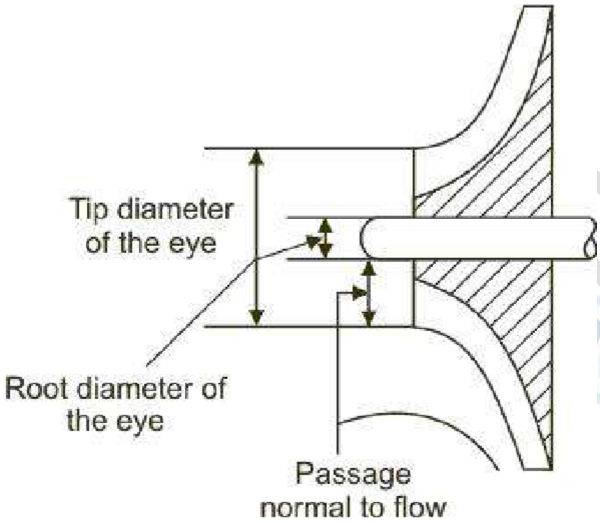


Figure 4.6 Inducing section of a centrifugal compressor

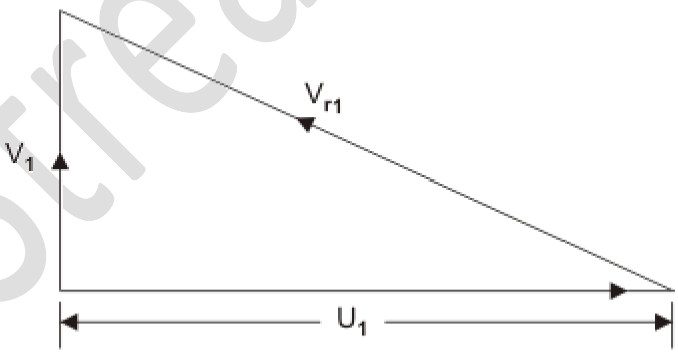


Figure 4.7 (a)

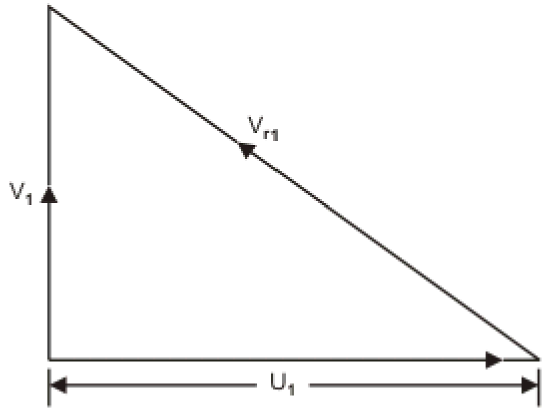


Figure 4.7 (b)

Figure 4.7 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. 4.8. Experiments have shown that the maximum permissible included angle of divergence is 11° to avoid considerable losses due to flow separation.

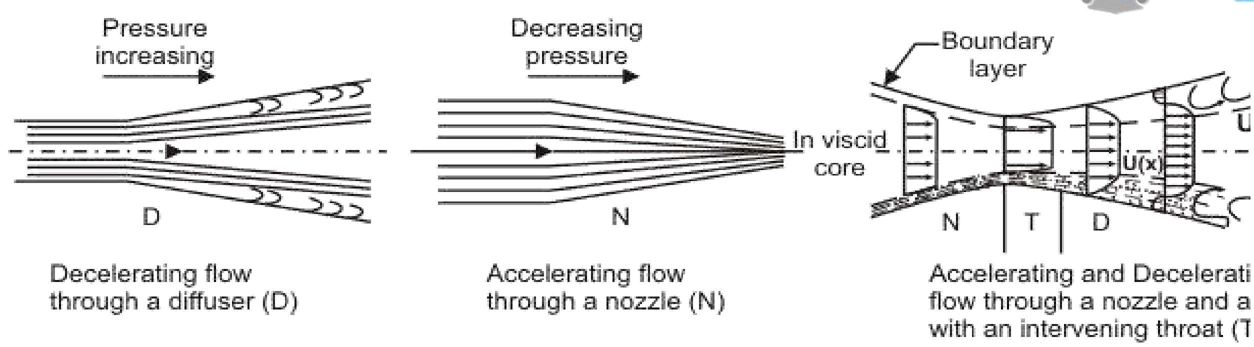


Figure 4.8 Accelerating and decelerating flows

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 vane less 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 ω remains constant. Hence h0 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 Vr will therefore also decrease from impeller tip to diffuser vane, in accordance with the equation of continuity. If both h0 and Vr decrease from the impeller tip then the resultant velocity *V* decreases from the impeller tip and some diffusion takes place in the vane less space. The consequent increase in density means that P2 will not decrease in inverse proportion to the radius and the way V2 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. 4.9.

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 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 4.9

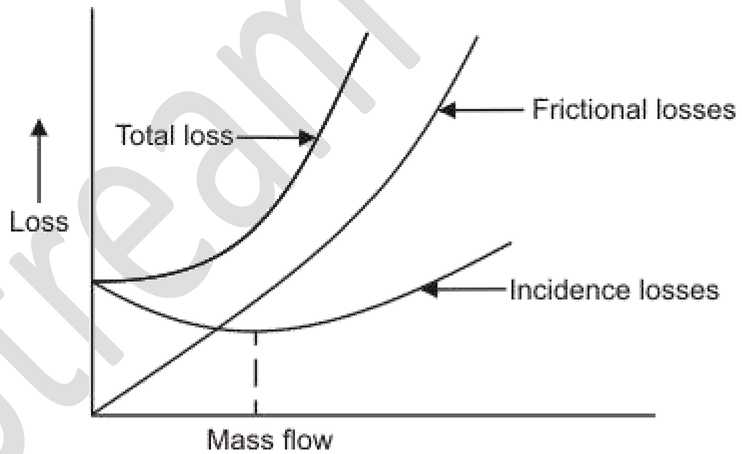


Figure 4.9 Dependence of various losses with mass flow in a centrifugal compressor

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. 4.9, 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 vane-less 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 nonzero minimum value (Figure. 4.9) 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

$$F(D, N, m, p_{0_1}, p_{0_2}, RT_{0_1}, RT_{0_2}) = 0 \quad (4.9)$$

Where *D* = characteristic linear dimension of the machine,

N = rotational,

m = mass flow rate,

P01 = stagnation pressure at compressor inlet,

P02 =stagnation pressure at compressor outlet,

T01 = stagnation temperature at compressor inlet,

T02 = 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 and rotational speed Mach number, .

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

$$\text{Function}\left(\frac{p_{2t}}{p_{1t}}, \frac{T_{2t}}{T_{1t}}, \frac{m\sqrt{T_{01}}}{p_{01}}, \frac{N}{\sqrt{T_{01}}}\right) = 0 \qquad (4.10)$$

Though the terms P1t and P2t 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 T1t and T2t are plotted against h in the form of two families of curves, each curve of a family being drawn for fixed value. The two families of curves represent the compressor characteristics. From these curves, it is possible to draw the curves of isentropic efficiency ηc. We can recall, in this context, the definition of the isentropic efficiency as

$$\eta_C = \frac{T_{02s} - T_{01}}{T_{02} - T_{01}} = \frac{(p_{02}/p_{01})^{\frac{\gamma-1}{\gamma}} - 1}{(T_{02}/T_{01}) - 1} \qquad (4.11)$$

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 4.10 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 4.10. 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 4.10) 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.

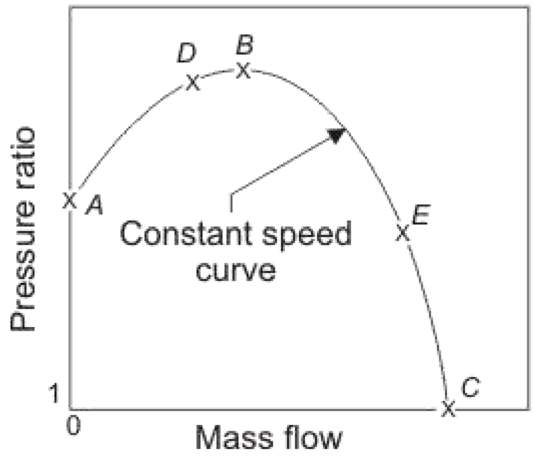


Figure 4.10 The theoretical characteristic curve

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 4.11. The left hand extremities of the constant speed curves may be joined up to form surge line, the right hand extremities indicate choking (Figure 4.11).

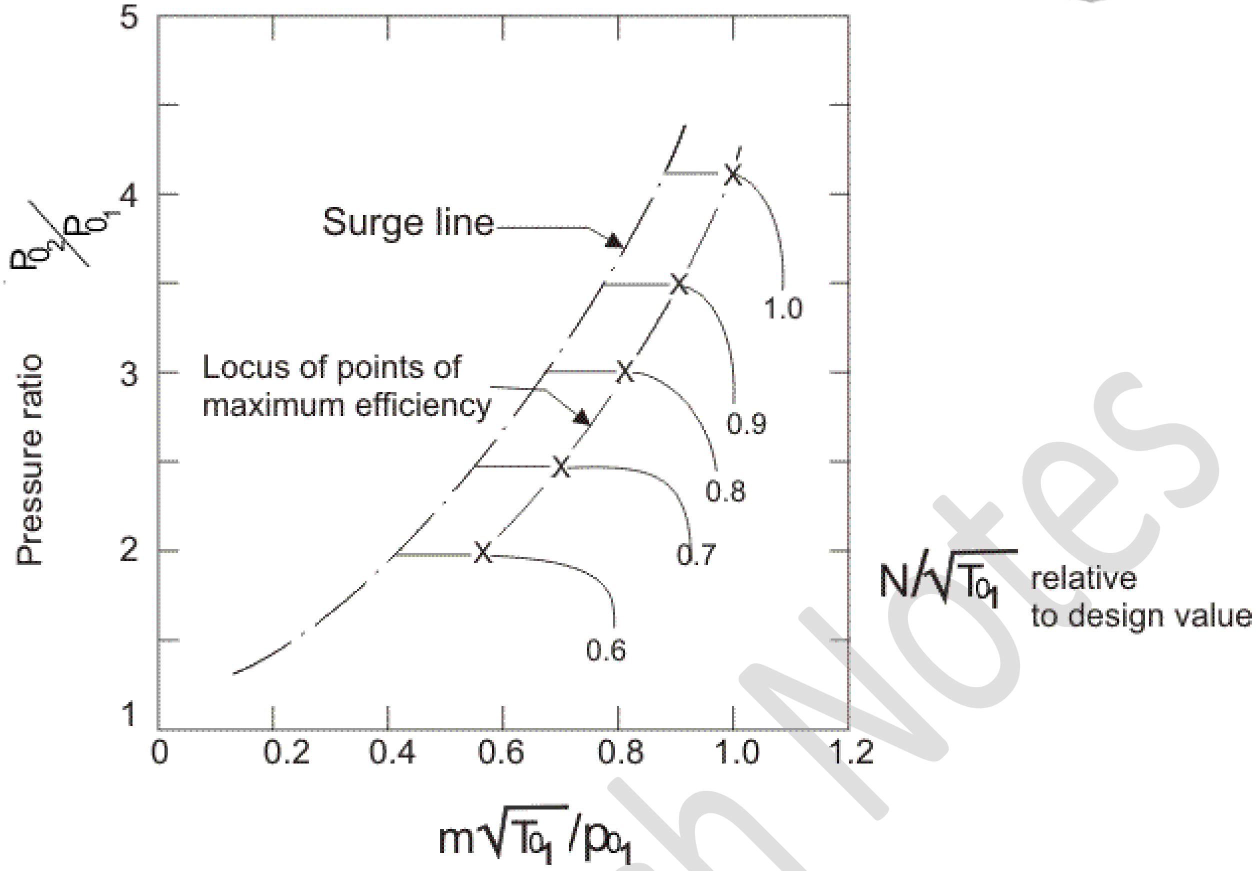


Figure 4.11 Variations of pressure ratio over the complete range of mass flow for different rotational speeds

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 4.11 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 4.12 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.

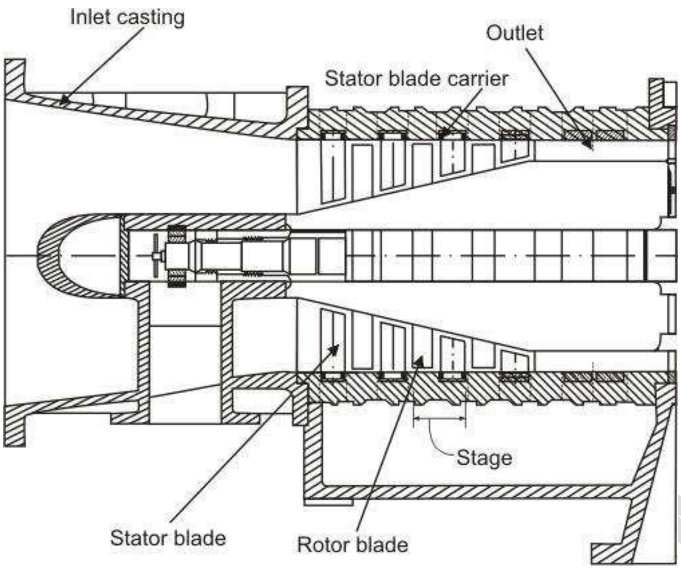


Figure 4.11 Disk type axial flow compressor

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, V1, and angle α to the axial direction. In combination with the peripheral velocity U of the blades, its relative velocity will be Vr at angle β as shown in the upper velocity triangle (Figure 4.12). 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 Vr at angle β which is less than 30 degree. 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 β is less than 30 degree due to diffusion, some pressure rise has been accomplished in the rotor. The velocity V1 in combination with U gives the absolute velocity V2 at the exit from the rotor at an angle α to the axial direction. The air then passes through the passages formed by the stator blades where it is further diffused to velocity Vr2 at an angle β2 which in most designs equals to V2 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.

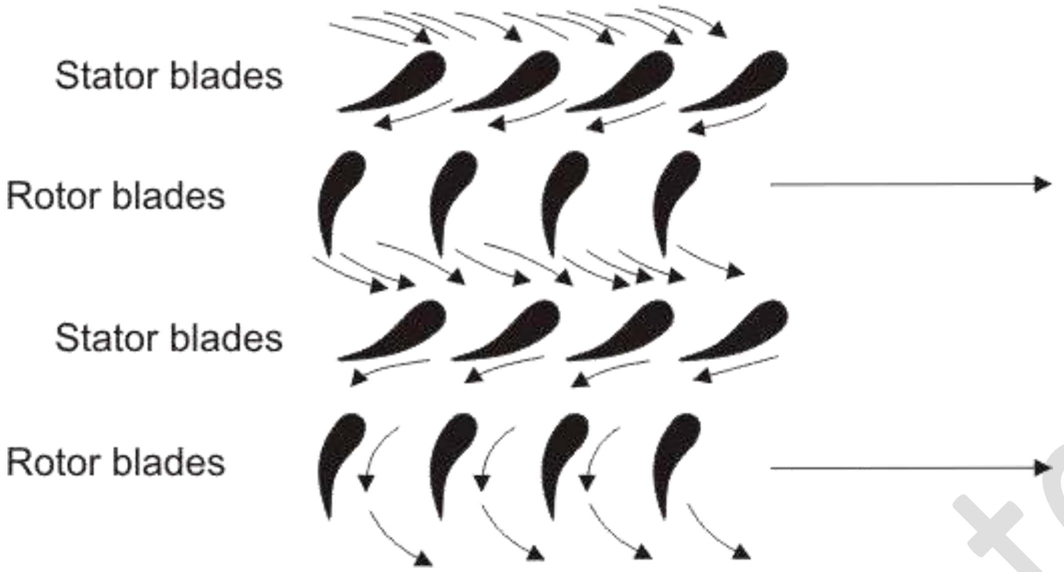


Figure 4.12 Flow through stages

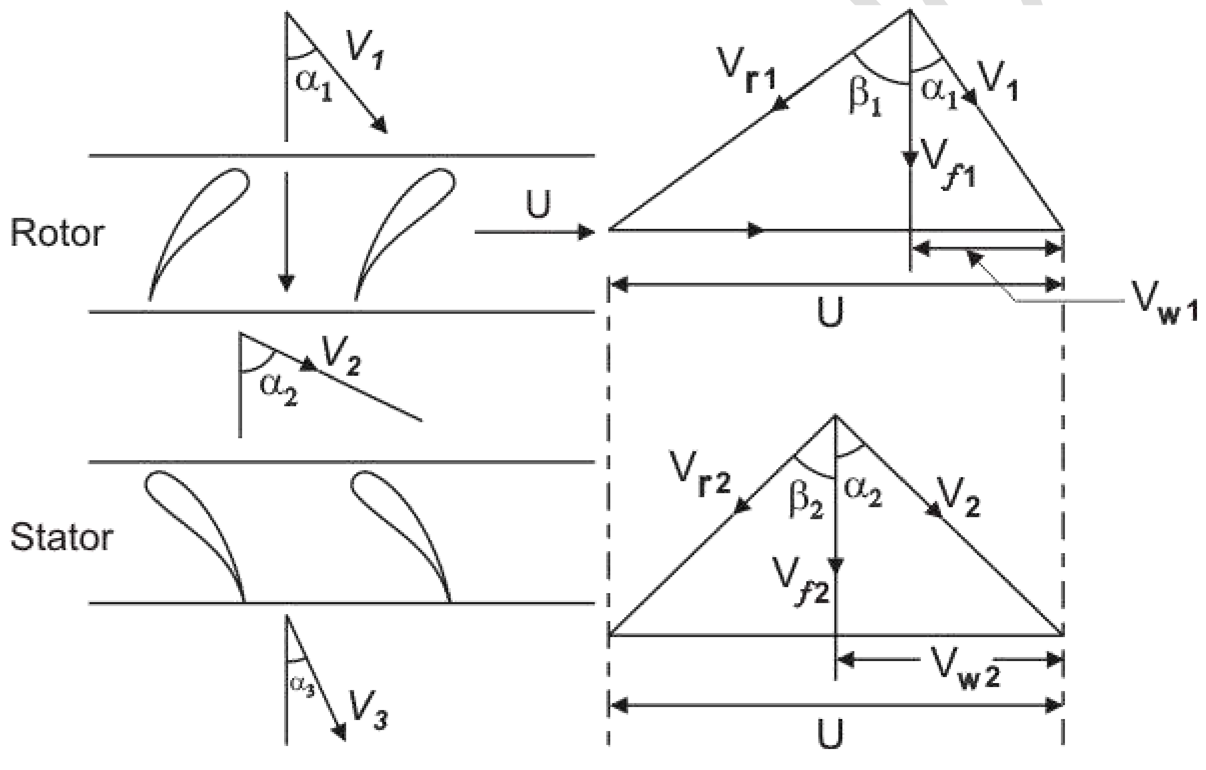


Figure 4.13 Velocity triangles

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

$$\frac{U}{V_f} = \tan \alpha_1 + \tan \beta_1 \tag{4.11}$$

$$\frac{U}{V_f} = \tan \alpha_2 + \tan \beta_2 \tag{4.12}$$

In which Vf is the axial velocity, assumed constant through the stage. The work done per unit mass or specific work input, *w* being given by

$$w = U(V_{w2} - V_{w1}) \quad (4.13)$$

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

$$w = UV_f(\tan \alpha_2 - t\,\mathrm{an}\, \alpha_1) \quad (4.14)$$

or by using Eqs. (4.11) and (4.12)

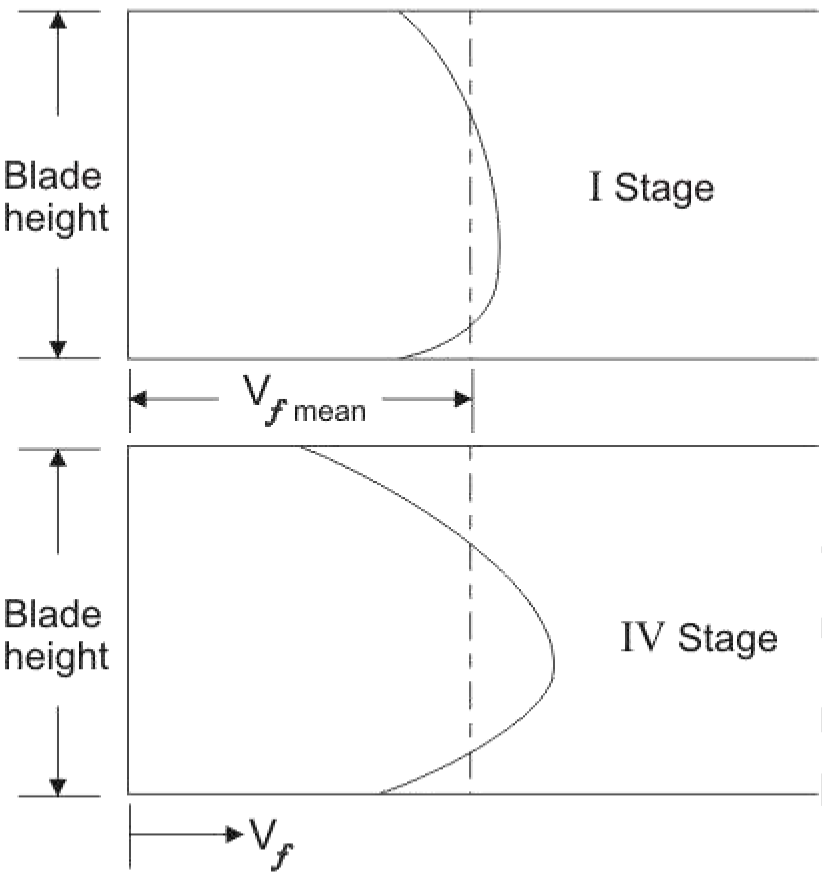
$$\Delta T_0 = \Delta T_s = \frac{UV_f}{c_p}(\tan \beta_1 - \tan \beta_2) \quad (4.15)$$

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 T02 *.* If the absolute velocity of the air leaving the stage V2 is made equal to that at the entry. V1, the stagnation temperature rise T02 will also be the static temperature rise of the stage, T03 so that

$$\begin{aligned} w &= U[(U - V_f \tan \alpha_1) - V_f \tan \beta_2] \\ &= U(U - V_f (\tan \alpha_1 + \tan \beta_2)) \end{aligned} \quad (4.16)$$

In fact, the stage temperature rise will be less than that given in Eq. (4.16) owing to three dimensional effects in the compressor annulus. Experiments show that it is necessary to multiply the right hand side of Eq. (4.16) 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. 4.14) as the flow proceeds, settling down to a fixed profile at about the fourth stage. Equation (4.15) can be written with the help of Eq. (4.11) as



(4.17)

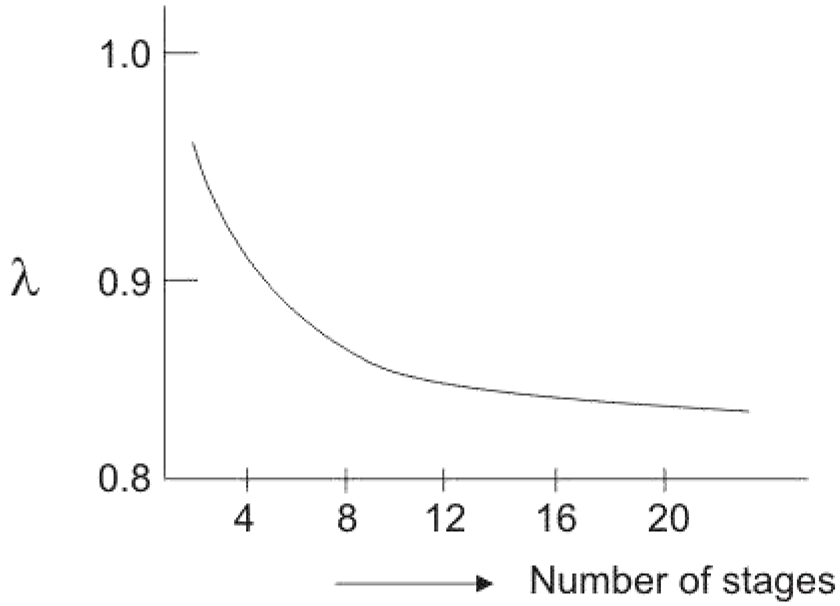


Figure 4.14 Axial velocity distribution

Since the outlet angles of the stator and the rotor blades fix the value of λ and number of stages and hence the value of V3. Any increase in number of stages will result in a decrease in λ and vice-versa. If the compressor is designed for constant radial distribution of V as shown by the dotted line in Figure (4.14), the effect of an increase in Vf 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 Vf in the regions of the root and tip of the blading because of the reduction of V *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. Care should be taken to avoid confusion of the work done factor with the idea of efficiency. If w is the expression for the specific work input (Equation. 4.13), then W 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.

DEGREE OF REACTION

A certain amount of distribution of 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 Vf 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 (V3).

$$\begin{aligned} w &= c_p(\Delta T_A + \Delta T_B) = c_p \Delta T_s \\ &= UV_f(\tan\beta_1 - \tan\beta_2) \\ &= UV_f(\tan\alpha_2 - \tan\alpha_1) \end{aligned} \quad (4.18)$$

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

$$c_p \Delta T_A = UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\sec^2\alpha_2 - \sec^2\alpha_1)$$
$$= UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\tan^2\alpha_2 - \tan^2\alpha_1)$$

$$c_p \Delta T_A = UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\sec^2\alpha_2 - \sec^2\alpha_1)$$
$$= UV_f(\tan\alpha_2 - \tan\alpha_1) - \frac{1}{2}V_f^2(\tan^2\alpha_2 - \tan^2\alpha_1) \quad (4.19)$$

The degree of reaction

$$\Lambda = \frac{\Delta T_A}{\Delta T_A + \Delta T_B} \quad (4.20)$$

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

By adding up Eq. (4.19) and Eq. (4.20) we get

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

$$\Lambda = \frac{V_f}{2U}(\tan\beta_1 + \tan\beta_2) \quad (4.21)$$

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

i.e. $\tan\alpha_1 = \tan\beta_2,$ (4.22a) i.e. $$V_f = V_1 \cos \alpha_1 = V_3 \cos \alpha_3 \qquad (4.22b)$$

Unit 5 - POWER TRANSMITTING TURBO MACHINES

Application and general theory

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 party radial and partly axial, the term *mixed-flow machine* is used. Figure 5.1 (a) (b) and (c) are the schematic diagrams of various types of impellers based on the flow direction.

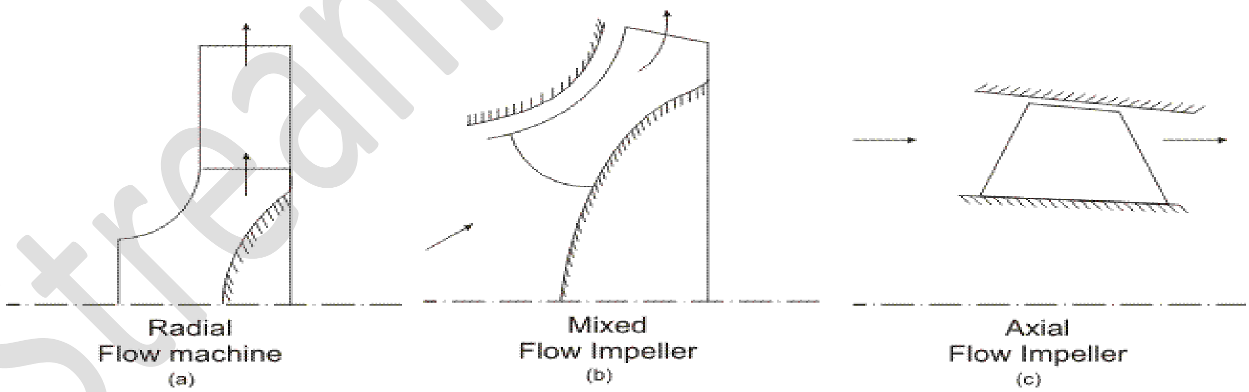


Fig. 5.1 Schematic of different types of impellers

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.

Torque ratio, speed ratio, slip and efficiency

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.

FLUID COUPLING AND TORQUE CONVERTER,

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. 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.

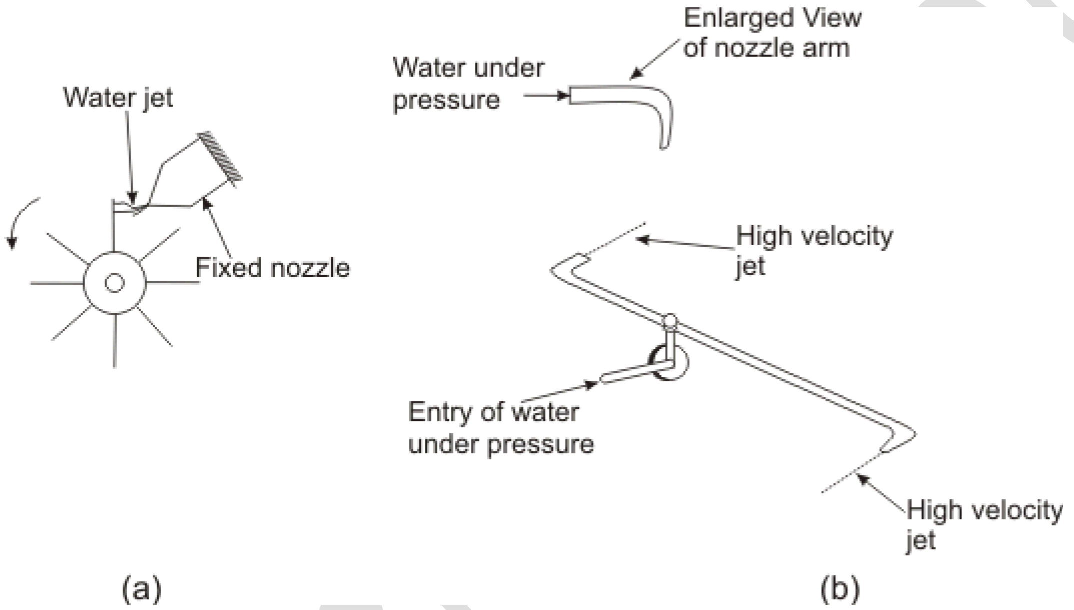


Fig 5.2 (a) Paddle wheel as an example of impulse turbine

(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. 5.2b) 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. 5.2a), 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. 5.2b), 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.

Hydrostatic systems

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 rotors itself and the point of actual power input or output.

Therefore, for a pump or compressor,

$$\eta_{hydraulic} = \eta_h = \frac{\begin{array}{l}\text{useful energy gained by the fluid}\\ \text{at final discharge}\end{array}}{\text{mechanical energy supplied to rotor}} \qquad (5.1a)$$

$$\eta_{overall} = \frac{\begin{array}{l}\text{useful energy gained by the fluid}\\ \text{at final discharge}\end{array}}{\begin{array}{l}\text{mechanical energy supplied to}\\ \text{shaft at coupling}\end{array}} \qquad (5.1b)$$

For a turbine,

$$\eta_h = \frac{\text{mechanical energy delivered by the rotor}}{\text{energy available from the fluid}} \qquad (5.2a)$$

$$\eta_{overall} = \frac{\begin{array}{l}\text{mechanical energy in output shaft}\\ \text{at coupling}\end{array}}{\text{energy available from the fluid}} \qquad (5.2b)$$

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

Therefore

$$\eta_m = \frac{\eta_{overall}}{\eta_h} \qquad (5.3)$$