SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – I - FLUID PROPERTIES & EQUATIONS OF MOTION – SAEA1304

I.INTRODUCTION

Fluid mechanics is the study of fluids either in motion (fluid *dynamics)* or at rest (fluid *statics)* and the subsequent effects of the fluid upon the boundaries, which may be either solid surfaces or interfaces with other fluids. Both gases and liquids are classified as fluids, and the number of fluids engineering applications is enormous: breathing, blood flow, swimming, pumps, fans, turbines, airplanes, ships, rivers, windmills, pipes, missiles, icebergs, engines, filters, jets, and sprinklers, to name a few. When you think about it, almost everything on this planet either is a fluid or moves within or near a fluid.

The study of fluids at rest is called fluid statics. The study of fluids in motion, where pressure forces are not considered, is called fluid kinematics and if the pressure forces are also considered for the fluids in motion that branch of science is carted fluid dynamics.

II. THE CONTINUUM CONCEPT OF A FLUID

Although the properties of a fluid arise from its molecular structure, engineering problems are usually concerned with the bulk behaviour of fluids. The number of molecules involved is immense, and the separation between them is normally negligible by comparison with the distances involved in the practical situation being studied. Under these conditions, it is usual to consider a fluid as a continuum – a hypothetical continuous substance – and the conditions at a point as the average of a very large number of molecules surrounding that point within a distance which is large compared with the mean intermolecular distance (although very small in absolute terms).

Quantities such as velocity and pressure can then be considered to be constant at any point, and changes due to molecular motion may be ignored. Variations in such quantities can also be assumed to take place smoothly, from point to point. This assumption breaks down in the case of rarefied gases, for which the ratio of the mean free path of the molecules to the physical dimensions of the problem is very much larger.

In this study, fluids will be assumed to be continuous substances and, when the behaviour of a small element or particle of fluid is studied, it will be assumed that it contains so many molecules that it can be treated as part of this continuum.

III. PROPERTIES OF FLUIDS

Density or Mass Density.

Density or mass density of a fluid is defined as the ratio of the mass of a fluid to its volume. Thus mass per unit volume of a fluid is called density. It is denoted by the symbol ρ (rho). The unit of mass density in SI unit is kg per cubic metre, i.e. kg/m^3. The density of liquids may be considered as constant while that of gases changes with the variation of pressure and temperature.

Mathematically, mass density is written as

$$\rho = \frac{\text{Mass of fluid}}{\text{Volume of fluid}}$$

The value of density of water is 1 gm/cm^3 or 1000 kg/m^3.

Density is highly variable in gases and increases nearly proportionally to the pressure level. Density in liquids is nearly constant; the density of water (about 1000 kg/m") increases only 1 percent if the pressure is increased by a factor of 220. Thus most liquid flows are treated analytically as nearly "incompressible."

In general, liquids are about three orders of magnitude more dense than gases at atmospheric pressure. The heaviest common liquid is mercury, and the lightest gas is hydrogen. Compare their densities at 20°C and 1 atm:

Mercury: ρ =13,580 kg/m^3 Hydrogen: ρ= 0.0838 kg/m^3

They differ by a factor of 162,000! Thus the physical parameters in various liquid and gas flows might vary considerably. The differences are often resolved by the use of *dimensional analysis.*

Specific Weight or Weight Density,

Specific weight or weight density of a fluid is the ratio between the weight of a fluid to its volume. Thus weight per unit volume of a fluid is called weight density and it is denoted by the symbol γ (lowercase Greek gamma).

Thus mathematically, $\gamma = \frac{\text{Weight of fluid}}{\text{Volume of fluid}} = \frac{\text{(Mass of fluid) x Acceleration due to gravity}}{\text{Volume of fluid}}$

$$= \frac{\text{(Mass of fluid) x g}}{\text{Volume of fluid}}$$

$\gamma = \rho\ x\ g$ $\left\{\rho = \frac{\text{Mass of fluid}}{\text{Volume of fluid}}\right\}$

Specific Volume

specific volume of a fluid is defined as the volume of a fluid occupied by a unit mass or volume per unit mass of a fluid is called specific volume. Mathematically, it is expressed as

$$\text{specific volume} = \frac{\text{Volume of fiuid}}{\text{Mass of fluid}} = \frac{1}{\frac{\text{Mass of fluid}}{\text{Volume of fiuid}}} = \frac{1}{\rho}$$

Thus specific volume is the reciprocal of mass density. It is expressed as m^3/kg. It is commonly applied to gases.

Specific Gravity.

Specific gravity is defined as the ratio of the weight density (or density) of a fluid to the weight density (or density) of a standard fluid. For liquids, the standard fluid is taken water and for gases, the standard fluid is taken air. Specific gravity is also called relative density. It is dimensionless quantity and is denoted by the symbol S.

$$\text{Mathematically}, \quad \text{S(for liquids)} = \frac{Weight\ Density\ (Density) of\ liquid}{Weight\ Density\ (Density) of\ Water}$$

$$\text{S(for Gases)} = \frac{Weight\ Density\ (Density) of\ Gas}{Weight\ Density\ (Density) of\ Air}$$

Thus weight density of a liquid = *S x* Weight density of water

= S x 1000 x 9.8I N/m^3

The density of a liquid = .S x Density of water

If the specific gravity of a fluid is known, then the density of the Fluid will be equal to specific gravity of fluid multiplied by rho density of water. For example, the specific gravity of mercury is 13.6, hence density of mercury = 1.3.6 x 1000 = 13000 kg/m^3.

VISCOSITY

Viscosity is defined as the property of a fluid which offers resistance to the movement of one layer of fluid over another adjacent layer of the fluid. When two layers of a fluid, a distance 'dy' apart, move one over the other at different velocities, say u and u + du as shown in Fig. 1, the viscosity together with relative velocity causes a shear stress acting between the fluid layers.

The top layer causes a shear stress on the adjacent lower layer while the lower layer causes a shear stress on the adjacent top layer. This shear stress is proportional to the rate of change of velocity with respect to y. It is denoted by symbol τ (Tau).

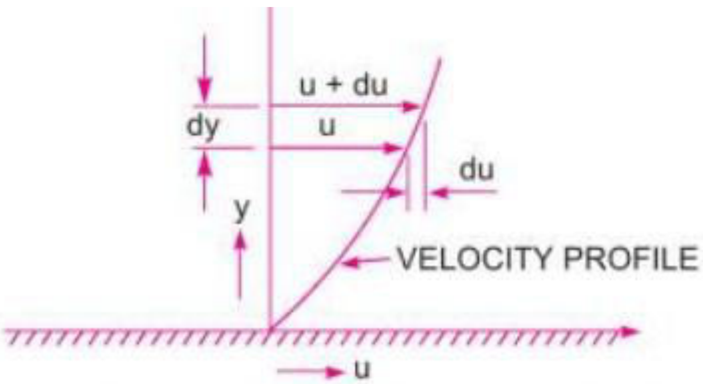


Figure 1: Velocity Variation near a solid boundary

Mathematically,

$$\tau \alpha \frac{du}{dy}$$

$$\tau = \mu \frac{du}{dy}$$

Where μ (called mu) is the constant of proportional and is known as the co-efficient of dynamic viscosity or only viscosity. $\frac{du}{dy}$ represents the rate of shear strain or rate of hear deformation or velocity gradient.

$$\mu = \frac{\tau}{\left(\frac{du}{dy}\right)}$$

Thus viscosity is also defined as the shear stress required to produce unit rate of shear strain.

Units of Viscosity

The units of viscosity is obtained by putting the dimensions of the quantities in equation

$$\mu = \frac{Shear\ stress}{\frac{Change\ of\ velocity}{Change\ of\ distance}} = \frac{Force/Area}{\left(\frac{Length}{Time}\right) X \frac{1}{Length}}$$

$$= \frac{Force/(Length)^2}{\frac{1}{Time}} = \frac{Force\ \ X\ Time}{(Length)^2} = Ns/m^2$$

Kinematic Viscosity. It is defined as the ratio between the dynamic viscosity and density of fluid. It is denoted by the Greek symbol (ϑ) called 'nu'. Thus. Mathematically,

$$\vartheta = \frac{Viscosity}{Density} = \frac{\mu}{\rho}$$

The units of kinematic viscosity is obtained as

$$\vartheta = \frac{Units\ of\ \mu}{Units\ of\ \rho} = \frac{Force\ X\ Time}{(Length)^2 X\ \frac{Mass}{(Length)^3}} = \frac{\frac{Mass\ X\ Length}{(Time)^2}\ X\ Time}{(Length)^2 X\ \frac{Mass}{(Length)^3}}$$

$$= \frac{\frac{\cancel{Mass}\ X\ \cancel{Length}}{(Time)^{\cancel{2}}}\ X\ \cancel{Time}}{\cancel{(Length)^2} X\ \frac{\cancel{Mass}}{(Length)^{\cancel{3}2}}}$$

$$= \frac{(Length)^2}{Time}$$

In MKS and SI, the unit of kinematic viscosity is metre^2/sec or m^2/sec while in CGS units it is written as cm^2/ s. In CGS units, kinematic viscosity is also known as stoke.

Thus, one stoke =cm^2/s = (1/100)^2 m^2/s =10^{-4} m^2/s

Centistoke = (1/100) stoke

Newton's Law of Viscosity.

It states that the shear stress (τ) on a fluid element layer is directly proportional to the rate of shear strain. The constant of proportionality is called the co- efficient of viscosity. Mathematically. it is expressed as given by equation

$$\tau = \mu \frac{du}{dy}$$

Fluids which obey the above relation are known as Newtonian fluids and the fluids which do not they the above relation are called Non-Newtonian fluids.

Variation of Viscosity with Temperature.

Temperature affects the viscosity. The viscosity of liquids decreases with the increase of temperature while the viscosity of gases increases with the increase of temperature. This is due to reason that the viscous forces in a fluid are due to cohesive forces and molecular momentum transfer. In liquids, the cohesive forces predominates the molecular momentum transfer due to closely packed molecules and with the increase in temperature. The cohesive force decreases with the result of decreasing viscosity. But in case of gases the cohesive forces are small and molecular momentum transfer predominates. With the increase in temperature, molecular momentum transfer increases and hence viscosity increases. The relation between viscosity and temperature for liquids and gases are:

For Liquids,

$$\mu = \mu_o \left(\frac{1}{1 + \alpha t + \beta t^2}\right) - - - - (1)$$

Where μ = Viscosity of liquid at t°C, in poise

μ_o= Viscosity of liquid at 0°C, in poise

$\propto, \beta$ = Constants for the liquid.

For water, μ_o= 1.79 X 10^{-3} poise, $\propto$ = 0.03368 and β = 0.000221.

Equation (1) shows that with the increase of temperature, the viscosity decreases

For a gas,

$$\mu = \mu_o + \alpha t - \beta t^2 - - - - - - (2)$$

For air, μ_o= 0.000017 poise, $\propto$ = 0.000000056 and β = 0.1189X10^{-9}.

Equation (2) shows that with the increase of temperature, the viscosity increases.

Types of Fluids

The fluids may be classified intn the following five types:

1. Ideal fluid,
2. Real fluid,
3. Newtonian fluid,
4. Non-Newtonian fluid, and

5. Ideal plastic fluid.

1. **Ideal Fluid.** A fluid, which is incompressible and is having no viscosity, is known as an ideal fluid. Ideal fluid is only an imaginary fluid as all the fluids. Which exist, have some viscosity.
2. **Real Fluid.** A Fluid, which possesses viscosity, is known as real fluid. All the fluids, in actual practice, are real fluids.
3. **Newtonian Fluid**. A real fluid, in which the shear stress is directly proportional to the rate of shear strain (or velocity gradient), is known as a Newtonian fluid.
4. **Non-Newtonian Fluid.** A real fluid. in which the shear stress is not proportional to the rate of shear strain (or velocity gradient), Known as a Non- Newtonian fluid.
5. **Ideal Plastic Fluid**. A fluid, in which shear stress is more than the yield value and shear stress is proportional to the rate of shear strain (or velocity gradient), is known as ideal plastic fluid.

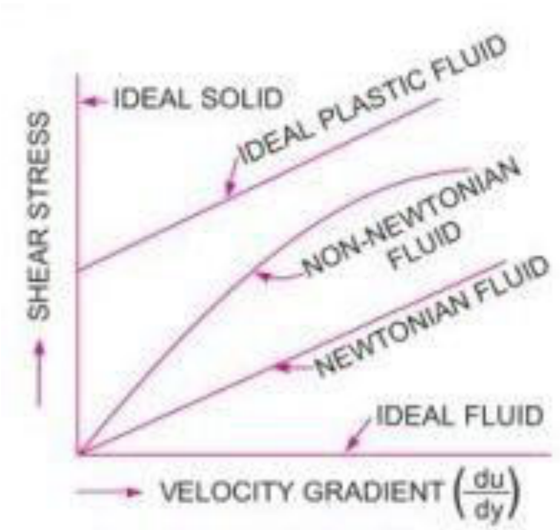


Fig.2 Types of Fluid

EQUATION OF STATE OF A PERFECT GAS

The mass density of a gas varies with its absolute pressure p and absolute temperature T.

$p = \rho RT$ -----(3)

For a perfect gas, where R is the gas constant for the gas concerned. Most gases at pressures and temperatures well removed from liquefaction follow this characteristic equation closely, but it does not apply to vapours. Units: the gas constant is measured in joules per kilogram per kelvin (J kg−1 K−1). Dimensions: $L2T^{-2}\ \Theta^{-1}$. Typical values: air, 287 J kg^{-1} K^{-1} ; hydrogen, 4110 J kg^{-1} K^{-1}.

SURFACE TENSION

Although all molecules are in constant motion, a molecule within the body of the liquid is, on average, attracted equally in all directions by the other molecules surrounding it, but, at the surface between liquid and air, or the interface between one substance and another, the upward and downward attractions are unbalanced, the surface molecules being pulled inward towards the bulk of the liquid. This effect causes the liquid surface to behave as if it were an elastic membrane under tension.

The surface tension σ is measured as the force acting across the unit length of a line drawn in the surface. It acts in the plane of the surface, normal to any line in the surface, and is the same at all points. Surface tension is constant at any given temperature for the surface of separation of two particular substances, but it decreases with increasing temperature.

The effect of surface tension is to reduce the surface of a free body of liquid to a minimum, since to expand the surface area molecules have to be brought to the surface from the bulk of the liquid against the unbalanced attraction pulling the surface molecules inwards. For this reason, drops of liquid tend to take a spherical shape in order to minimize surface area. For such a small droplet, surface tension will cause an increase of internal pressure p in order to balance the surface force.

Considering the forces acting on a diametral plane through a spherical drop of radius r, the force due to internal pressure = $p \times \pi r^2$, and the force due to surface tension around the perimeter = $2\pi r \times \sigma$.

For equilibrium, $p\pi r^2 = 2\pi r\sigma$ or $p = 2\sigma/r$. Surface tension will also increase the internal pressure in a cylindrical jet of fluid, for which $p = \sigma/r$. In either case, if r is very small, the value of p becomes very large.

For small bubbles in a liquid, if this pressure is greater than the pressure of vapour or gas in a bubble, the bubble will collapse. In many of the problems with which engineers are concerned, the magnitude of surface tension forces is very small compared with the other forces acting on the fluid and may, therefore, be neglected. However, these forces can cause serious errors in hydraulic scale models and through capillary effects. Surface tension forces can be reduced by the addition of detergents.

Exameble: Air is introduced through a nozzle into a tank of water to form a stream of bubbles. If the bubbles are intended to have a diameter of 2 mm, calculate by how much the pressure of the air at the nozzle must exceed that of the surrounding water. Assume that $\sigma = 72.7 \times 10{-}3\ \mathrm{N\ m^{-1}}$.

Solution Excess pressure,

$p = 2\sigma/r$

Putting $r = 1\ \mathrm{mm} = 10^{-3}\ \mathrm{m}$, $\sigma = 72.7 \times 10^{-3}\ \mathrm{N\ m^{-1}}$.

Excess pressure, $p = (2 \times 72.7 \times 10^{-3})/ (1 \times 10^{-3}) = 145.4\ \mathrm{N\ m^{-2}}$

CAPILLARITY

If a fine tube, open at both ends, is lowered vertically into a liquid which wets the tube, the level of the liquid will rise in the tube Fig.3 (a). If the liquid does not wet the tube, the level of liquid in the tube will be depressed below the level of the free surface outside Fig.3 (b). If θ is the angle of contact between liquid and solid and d is the tube diameter Fig.3 (a),

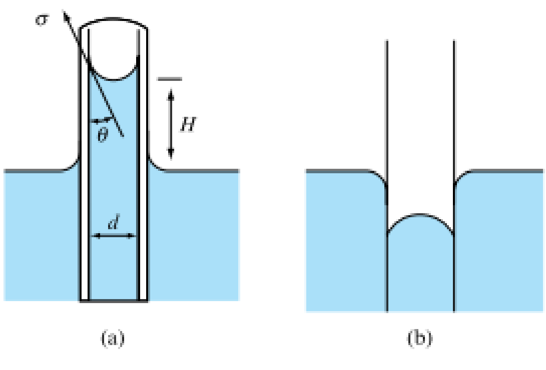


Figure 3: Capillarity

Upward pull due to surface tension = Component of surface tension acting upwards × Perimeter of tube

$$= \sigma \cos\theta \times \pi d. \quad \text{-----------------} \ (4)$$

The atmospheric pressure is the same inside and outside the tube, and, therefore, the only force opposing this upward pull is the weight of the vertical-sided column of liquid of height H, since, by definition, there are no shear stresses in a liquid at rest. Therefore, in Fig. there will be no shear stress on the vertical sides of the column of liquid under consideration.

Weight of column raised = $\rho g (\pi/4)d^2 H$, ---------- (5)

Where ρ is the mass density of the liquid. Equating the upward pull to the weight of the column, from equations (4) and (5),

$\sigma \cos\theta \times \pi d = \rho g(\pi/4)d2\ H$,

Capillary rise, $H = 4\sigma \cos\theta/\rho g d$.

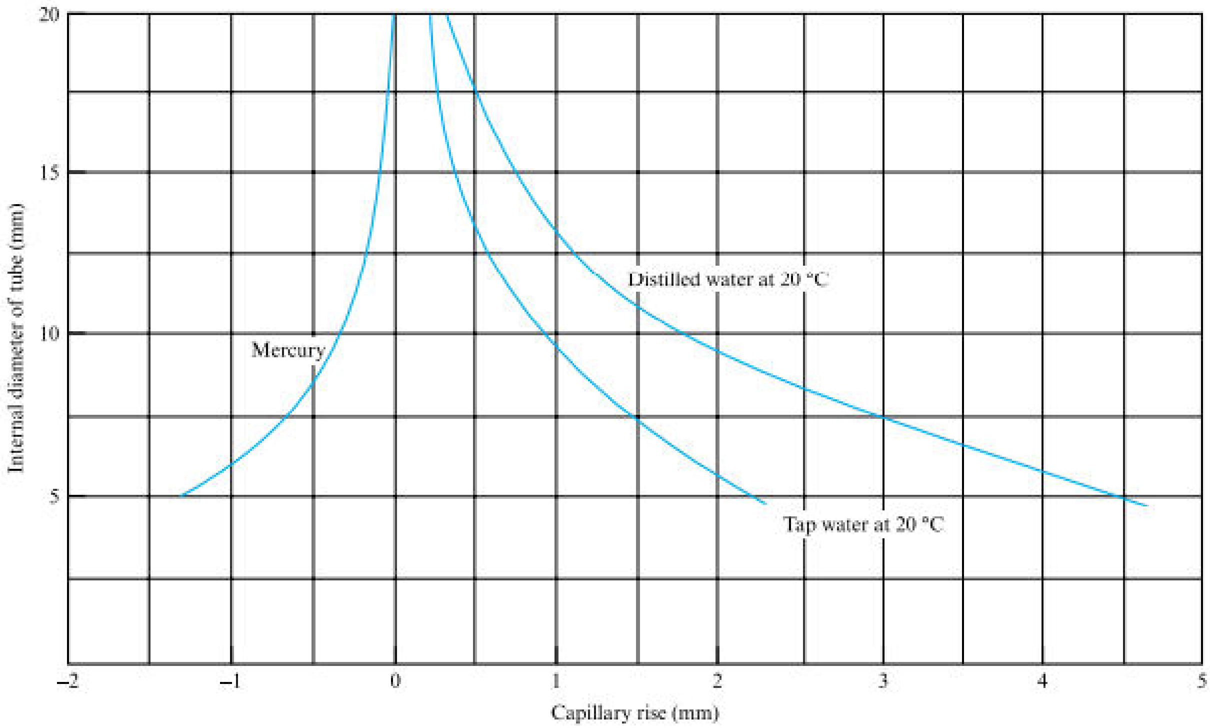


Figure 4: Capillary rise in glass tubes of circular cross section.

Capillary action is a serious source of error in reading liquid levels in fine-gauge tubes, particularly as the degree of wetting and, therefore, the contact angle θ are affected by the cleanness of the surfaces in contact. For water in a tube of 5 mm diameter, the capillary rise will be approximately 4.5 mm, while for mercury the corresponding figure would be −1.4 mm (Fig. 4). Gauge glasses for reading the level of liquids should have as large a diameter as is conveniently possible, to minimize errors due to capillarity.

VAPOUR PRESSURE

Since the molecules of a liquid are in constant agitation, some of the molecules in the surface layer will have sufficient energy to escape from the attraction of the surrounding molecules into the space above the free surface. Some of these molecules will return and condense, but others will take their place.

If the space above the liquid is confined, an equilibrium will be reached so that the number of molecules of liquid in the space above the free surface is constant. These molecules produce a partial pressure known as the vapour pressure in the space.

The degree of molecular activity increases with increasing temperature, and, therefore, the vapour pressure will also increase. Boiling will occur when the vapour pressure is equal to the pressure above the liquid. By reducing the pressure, boiling can be made to occur at temperatures well below the boiling point at atmospheric pressure: for example, if the pressure is reduced to 0.2 bar (0.2 atm), water will boil at a temperature of 60 °C.

CAVITATION

Under certain conditions, areas of low pressure can occur locally in a flowing fluid. If the pressure in such areas falls below the vapour pressure, there will be local boiling and a cloud of vapour bubbles will form. This phenomenon is known as cavitation and can cause serious problems, since the flow of liquid can sweep this cloud of bubbles on into an area of higher pressure where the bubbles will collapse suddenly. If this should occur in contact with a solid surface, very serious damage can result due to the very large force with which the liquid hits the surface.

Cavitation can affect the performance of hydraulic machinery such as pumps, turbines and propellers, and the impact of collapsing bubbles can cause local erosion of metal surfaces. Cavitation can also occur if a liquid contains dissolved air or other gases, since the solubility of gases in a liquid decreases as the pressure is reduced. Gas or air bubbles will be released in the same way as vapour bubbles, with the same damaging effects. Usually, this release occurs at higher pressures and, therefore, before vapour cavitation commences.

COMPRESSIBILITY AND THE BULK MODULUS

All materials, whether solids, liquids or gases, are compressible, i.e. the volume V of a given mass will be reduced to V – δV when a force is exerted uniformly all over its surface. If the force per unit area of surface increases from p to p + δp, the relationship between change of pressure and change of volume depends on the bulk modulus of the material:

Bulk modulus = Change in pressure/Volumetric strain.

Volumetric strain is the change in volume divided by the original volume; therefore,

$$\frac{\text{Change in volume}}{\text{Original volume}} = \frac{\text{Change in pressure}}{\text{Bulk modulus}}$$

$$-\delta V/V = \delta p/K$$

the minus sign indicating that the volume decreases as pressure increases. In the limit, as δp → 0,

$K = -V\frac{dp}{dV}$ --------(6)

Considering unit mass of a substance,

V = 1/ρ ----------(7)

Differentiating,

V dρ + ρ dV = 0

dV = −(V/ρ) dρ.

Substituting for V from equation (1.10),

$dV = -(1/\rho^2)\, d\rho$. --------------------(8)

Putting the values of V and dV obtained from equations (7) and (8) in equation (6),

$K = \rho\ (dp/d\rho)$

▶ 5.4 RATE OF FLOW OR DISCHARGE (Q)

It is defined as the quantity of a fluid flowing per second through a section of a pipe or a channel. For an incompressible fluid (or liquid) the rate of flow or discharge is expressed as the volume of fluid flowing across the section per second. For compressible fluids, the rate of flow is usually expressed as the weight of fluid flowing across the section. Thus

(*i*) For liquids the units of Q are m^3/s or litres/s

(*ii*) For gases the units of Q is kgf/s or Newton/s

Consider a liquid flowing through a pipe in which

A = Cross-sectional area of pipe

V = Average velocity of fluid across the section

Then discharge $\quad Q = A \times V.$ ...(5.1)

▶ 5.5 CONTINUITY EQUATION

The equation based on the principle of conservation of mass is called continuity equation. Thus for a fluid flowing through the pipe at all the cross-section, the quantity of fluid per second is constant. Consider two cross-sections of a pipe as shown in Fig. 5.1.

Let V_1 = Average velocity at cross-section 1-1

ρ_1 = Density at section 1-1

A_1 = Area of pipe at section 1-1

and V_2, ρ_2, A_2 are corresponding values at section, 2-2.

Then rate of flow at section 1-1 = $\rho_1 A_1 V_1$

Rate of flow at section 2-2 = $\rho_2 A_2 V_2$

According to law of conservation of mass

Rate of flow at section 1-1 = Rate of flow at section 2-2

or $$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 \quad ...(5.2)$$

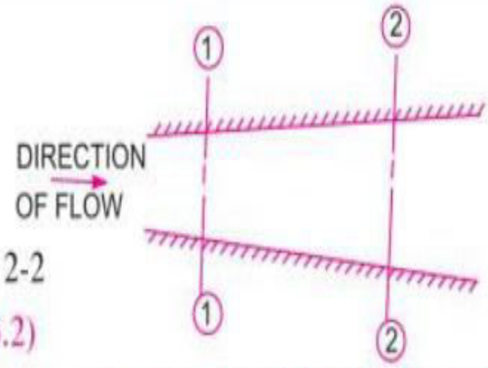


Fig. 5.1 *Fluid flowing through a pipe.*

Equation (5.2) is applicable to the compressible as well as incompressible fluids and is called **Continuity Equation**. If the fluid is incompressible, then $\rho_1 = \rho_2$ and continuity equation (5.2) reduces to

$$A_1 V_1 = A_2 V_2 \quad ...(5.3)$$

▶ 5.6 CONTINUITY EQUATION IN THREE-DIMENSIONS

Consider a fluid element of lengths dx, dy and dz in the direction of x, y and z. Let u, v and w are the inlet velocity components in x, y and z directions respectively. Mass of fluid entering the face $ABCD$ per second

$$= \rho \times \text{Velocity in } x\text{-direction} \times \text{Area of } ABCD$$

$$= \rho \times u \times (dy \times dz)$$

Then mass of fluid leaving the face $EFGH$ per second $= \rho u \; dydz + \frac{\partial}{\partial x} (\rho u \; dydz) \; dx$

∴ Gain of mass in x-direction

$$= \text{Mass through } ABCD - \text{Mass through } EFGH \text{ per second}$$

$$= \rho u \; dydz - \rho u \; dydz - \frac{\partial}{\partial x} (\rho u \; dydz)dx$$

$$= -\frac{\partial}{\partial x} (\rho u \; dydz) \; dx$$

$$= -\frac{\partial}{\partial x} (\rho u) \; dx \; dydz$$

{∵ $dydz$ is constant}

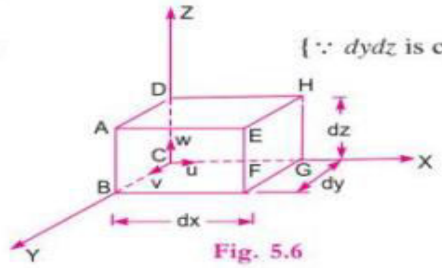


Fig. 5.6

Similarly, the net gain of mass in y-direction

$$= -\frac{\partial}{\partial y} (\rho v) \; dxdydz$$

and in z-direction

$$= -\frac{\partial}{\partial z} (\rho w) \; dxdydz$$

∴ Net gain of masses $= -\left[\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w)\right] dxdydz$

Since the mass is neither created nor destroyed in the fluid element, the net increase of mass per unit time in the fluid element must be equal to the rate of increase of mass of fluid in the element. But mass of fluid in the element is $\rho . \; dx. \; dy. \; dz$ and its rate of increase with time is $\frac{\sigma}{\partial t}$ $(\rho \; dx. \; dy. \; dz)$ or

$\frac{\partial \rho}{\partial t} . \; dx \; dy \; dz.$

Equating the two expressions,

or $$-\left[\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w)\right] dxdydz = \frac{\partial \rho}{\partial t} . \; dxdydz$$

or $$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0$$ [Cancelling $dx.dy.dz$ from both sides] ...(5.3*A*)

Equation (5.3*A*) is the continuity equation in cartesian co-ordinates in its most general form. This equation is applicable to :

(*i*) Steady and unsteady flow,

(*ii*) Uniform and non-uniform flow, and

(*iii*) Compressible and incompressible fluids.

For steady flow, $\frac{\partial \rho}{\partial t} = 0$ and hence equation (5.3*A*) becomes as

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \qquad ...(5.3B)$$

If the fluid is incompressible, then ρ is constant and the above equation becomes as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad ...(5.4)$$

Equation (5.4) is the continuity equation in three-dimensions. For a two-dimensional flow, the component $w = 0$ and hence continuity equation becomes as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0. \qquad ...(5.5)$$

5.6.1 Continuity Equation in Cylindrical Polar Co-ordinates.

The continuity equation in cylindrical polar co-ordinates (*i.e.*, r, θ, z co-ordinates) is derived by the procedure given below.

Consider a two-dimensional incompressible flow field. The two-dimensional polar co-ordinates are r and θ. Consider a fluid element $ABCD$ between the radii r and $r + dr$ as shown in Fig. 5.7. The angle subtended by the element at the centre is $d\theta$. The components of the velocity V are u_r in the radial direction and u_θ in the tangential direction. The sides of the element are having the lengths as

Side $AB = rd\theta$, $BC = dr$, $DC = (r + dr)\, d\theta$, $AD = dr$.

The thickness of the element perpendicular to the plane of the paper is assumed to be unity.

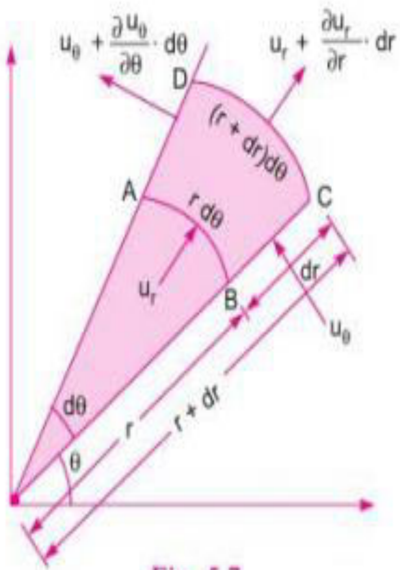


Fig. 5.7

Consider the flow in radial direction

Mass of fluid entering the face AB per unit time

$$= \rho \times \text{Velocity in } r\text{-direction} \times \text{Area}$$

$$= \rho \times u_r \times (AB \times 1) \qquad (\because \text{Area} = AB \times \text{Thickness} = rd\theta \times 1)$$

$$= \rho \times u_r \times (rd\theta \times 1) = \rho \,.\, u_r .\, rd\theta$$

Mass of fluid leaving the face CD per unit time

$$= \rho \times \text{Velocity} \times \text{Area}$$

$$= \rho \times \left(u_r + \frac{\partial u_r}{\partial r}.dr\right) \times (CD \times 1) \qquad (\because \text{Area} = CD \times 1)$$

$$= \rho \times \left(u_r + \frac{\partial u_r}{\partial r} dr\right) \times (r + dr)d\theta \qquad [\because CD = (r + dr)\, d\theta]$$

$$= \rho \times \left[u_r \times r + u_r dr + r\frac{\partial u_r}{\partial r} dr + \frac{\partial u_r}{\partial r}(dr)^2\right] d\theta$$

$$= \rho\left[u_r \times r + u_r \times dr + r\frac{\partial u_r}{\partial r}.dr\right] d\theta$$

[The term containing $(dr)^2$ is very small and has been neglected]

$\therefore$ Gain of mass in r-direction per unit time

$$= (\text{Mass through } AB - \text{Mass through } CD) \text{ per unit time}$$

$$= \rho.\, u_r.\, rd\theta - \rho\left[u_r.r + u_r.dr + r\frac{\partial u_r}{\partial r}.dr\right] d\theta$$

$$= \rho.\, u_r.\, rd\theta - \rho.\, u_r.\, r.\, d\theta - \rho\left[u_r.dr + r\frac{\partial u_r}{\partial r}.dr\right] d\theta$$

$$= -\rho\left[u_r.dr + r\frac{\partial u_r}{\partial r}.dr\right].d\theta$$

$$= -\rho\left[\frac{u_r}{r} + \frac{\partial u_r}{\partial r}\right] r.\ dr.\ d\theta$$

[This is written in this form because ($r.\ d\theta.\ dr.\ 1$) is equal to volume of element]

Now consider the flow in θ-direction

Gain in mass in θ-direction per unit time

= (Mass through *BC* – Mass through *AD*) per unit time

= [ρ × Velocity through *BC* × Area – ρ × Velocity through *AD* × Area]

$$= \left[\rho.u_\theta.dr \times 1 - \rho\left(u_\theta + \frac{\partial u_\theta}{\partial \theta}.d\theta\right) \times dr \times 1\right]$$

$$= -\rho\left(\frac{\partial u_\theta}{\partial \theta}.d\theta\right) dr \times 1 \qquad (\because \text{Area} = dr \times 1)$$

$$= -\rho\frac{\partial u_\theta}{\partial \theta}.\frac{r.d\theta.dr}{r} \qquad \text{[Multiplying and dividing by } r]$$

∴ Total gain in fluid mass per unit time

$$= -\rho\left[\frac{u_r}{r} + \frac{\partial u_r}{\partial r}\right].r.\ dr.\ d\theta - \rho\frac{\partial u_\theta}{\partial \theta}.\frac{rd\theta.\ dr}{r} \qquad ...(5.5A)$$

But mass of fluid element = ρ × Volume of fluid element

$= \rho \times [rd\theta \times dr \times 1]$

$= \rho \times rd\theta\ .\ dr$

Rate of increase of fluid mass in the element with time

$$= \frac{\partial}{\partial t}[\rho\ .\ rd\theta\ .\ dr] = \frac{\partial \rho}{\partial t}.\ rd\theta\ dr \qquad ...(5.5B)$$

($\because$ $rd\theta\ .\ dr\ .\ 1$ is the volume of element and is a constant quantity)

Since the mass is neither created nor destroyed in the fluid element, hence net gain of mass per unit time in the fluid element must be equal to the rate of increase of mass of fluid in the element.

Hence equating the two expressions given by equations (5.5 *A*) and (5.5 *B*), we get

$$-\rho\left[\frac{u_r}{r} + \frac{\partial u_r}{\partial r}\right] r\ .\ dr.\ d\theta - \rho\ \frac{\partial u_\theta}{\partial \theta}\frac{rd\theta.\ dr}{r} = \frac{\partial \rho}{\partial t}\ rd\theta\ dr$$

or

$$-\rho\left[\frac{u_r}{r} + \frac{\partial u_r}{\partial r}\right] - \rho\frac{\partial u_\theta}{\partial \theta}.\frac{1}{r} = \frac{\partial \rho}{\partial t} \qquad \text{[Cancelling } rdr\ .\ d\theta \text{ from both sides]}$$

or

$$\frac{\partial \rho}{\partial t} + \rho\left[\frac{u_r}{r} + \frac{\partial u_r}{\partial r}\right] + \rho\frac{\partial u_\theta}{\partial \theta}.\frac{1}{r} = 0 \qquad ...(5.5C)$$

Equation (5.5 *C*) is the continuity equation in polar co-ordinates for two-dimensional flow.

For steady flow $\frac{\partial \rho}{\partial t} = 0$ and hence equation (5.5 *C*) reduces to

$$\rho\left[\frac{u_r}{r} + \frac{\partial u_r}{\partial r}\right] + \rho\,\frac{\partial u_\theta}{\partial \theta}\cdot\frac{1}{r} = 0$$

or
$$\frac{u_r}{r} + \frac{\partial u_r}{\partial r} + \frac{\partial u_\theta}{\partial \theta}\cdot\frac{1}{r} = 0$$

or
$$u_r + r\frac{\partial u_r}{\partial r} + \frac{\partial u_\theta}{\partial \theta} = 0$$

or
$$\frac{\partial}{\partial r}(ru_r) + \frac{\partial}{\partial \theta}(u_\theta) = 0 \quad \left[\because \frac{\partial}{\partial r}(r\,.\,u_r) = r\,.\,\frac{\partial u_r}{\partial r} + u_r\right] \qquad ...(5.5D)$$

Equation (5.5 *D*) represents the continuity equation in polar co-ordinates for two-dimensional steady incompressible flow.

▶ 5.7 VELOCITY AND ACCELERATION

Let *V* is the resultant velocity at any point in a fluid flow. Let *u*, *v* and *w* are its component in *x*, *y* and *z* directions. The velocity components are functions of space-co-ordinates and time. Mathematically, the velocity components are given as

$$u = f_1(x, y, z, t)$$
$$v = f_2(x, y, z, t)$$
$$w = f_3(x, y, z, t)$$

and Resultant velocity, $V = ui + vj + wk = \sqrt{u^2 + v^2 + w^2}$

Let a_x, a_y and a_z are the **total acceleration** in *x*, *y* and *z* directions respectively. Then by the chain rule of differentiation, we have

$$a_x = \frac{du}{dt} = \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\cdot\frac{dz}{dt} + \frac{\partial u}{\partial t}$$

But $\frac{dx}{dt} = u, \frac{dy}{dt} = v$ and $\frac{dz}{dt} = w$

$$\left.\begin{aligned}
\therefore \quad a_x &= \frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t} \\
\text{Similarly,} \quad a_y &= \frac{dv}{dt} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t} \\
a_z &= \frac{dw}{dt} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t}
\end{aligned}\right] \qquad ...(5.6)$$

For steady flow, $\frac{\partial V}{\partial t} = 0$, where *V* is resultant velocity or $$\frac{\partial u}{\partial t} = 0, \frac{\partial v}{\partial t} = 0 \text{ and } \frac{\partial w}{\partial t} = 0$$

Hence acceleration in x, y and z directions becomes

$$\left.\begin{aligned} a_x &= \frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \\ a_y &= \frac{dv}{dt} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \\ a_z &= \frac{dw}{dt} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \end{aligned}\right] \quad \text{...(5.7)}$$

Acceleration vector $$\left.\begin{aligned} A &= a_x i + a_y j + a_z k \\ &= \sqrt{a_x^2 + a_y^2 + a_z^2}\,. \end{aligned}\right] \quad \text{...(5.8)}$$

5.7.1 Local Acceleration and Convective Acceleration.

Local acceleration is defined as the rate of increase of velocity with respect to time at a given point in a flow field. In the equation given by (5.6), the expression $\frac{\partial u}{\partial t}, \frac{\partial v}{\partial t}$ or $\frac{\partial w}{\partial t}$ is known as local acceleration.

Convective acceleration is defined as the rate of change of velocity due to the change of position of fluid particles in a fluid flow. The expressions other than $\frac{\partial u}{\partial t}, \frac{\partial v}{\partial t}$ and $\frac{\partial w}{\partial t}$ in equation (5.6) are known as convective acceleration.

6.2 EQUATIONS OF MOTION

According to Newton's second law of motion, the net force F_x acting on a fluid element in the direction of x is equal to mass m of the fluid element multiplied by the acceleration a_x in the x-direction. Thus mathematically,

$$F_x = m.a_x \quad \text{...(6.1)}$$

In the fluid flow, the following forces are present :

(*i*) F_g, gravity force.

(*ii*) F_p, the pressure force.

(*iii*) F_v, force due to viscosity.

(*iv*) F_t, force due to turbulence.

(*v*) F_c, force due to compressibility.

Thus in equation (6.1), the net force

$$F_x = (F_g)_x + (F_p)_x + (F_v)_x + (F_t)_x + (F_c)_x.$$

(*i*) If the force due to compressibility, F_c is negligible, the resulting net force

$$F_x = (F_g)_x + (F_p)_x + (F_v)_x + (F_t)_x$$

and equation of motions are called **Reynold's equations of motion.**

(*ii*) For flow, where (F_t) is negligible, the resulting equations of motion are known as **Navier-Stokes Equation.**

(*iii*) If the flow is assumed to be ideal, viscous force (F_v) is zero and equation of motions are known as **Euler's equation of motion.**

▶ 6.3 EULER'S EQUATION OF MOTION

This is equation of motion in which the forces due to gravity and pressure are taken into consideration. This is derived by considering the motion of a fluid element along a stream-line as :

Consider a stream-line in which flow is taking place in s-direction as shown in Fig. 6.1. Consider a cylindrical element of cross-section dA and length ds. The forces acting on the cylindrical element are:

1. Pressure force pdA in the direction of flow.
2. Pressure force $\left(p + \frac{\partial p}{\partial s} ds\right) dA$ opposite to the direction of flow.
3. Weight of element $\rho gdAds$.

Let θ is the angle between the direction of flow and the line of action of the weight of element.

The resultant force on the fluid element in the direction of s must be equal to the mass of fluid element × acceleration in the direction s.

The resultant force on the fluid element in the direction of s must be equal to the mass of fluid element × acceleration in the direction s.

$$\therefore \quad pdA - \left(p + \frac{\partial p}{\partial s} ds\right) dA - \rho gdAds \cos\theta$$

$$= \rho dAds \times a_s \qquad \ldots(6.2)$$

where a_s is the acceleration in the direction of s.

Now $$a_s = \frac{dv}{dt}, \text{ where } v \text{ is a function of } s \text{ and } t.$$

$$= \frac{\partial v}{\partial s}\frac{ds}{dt} + \frac{\partial v}{\partial t} = \frac{v\partial v}{\partial s} + \frac{\partial v}{\partial t} \quad \left\{\because \frac{ds}{dt} = v\right\}$$

If the flow is steady, $\frac{\partial v}{\partial t} = 0$

$$\therefore \quad a_s = \frac{v\partial v}{\partial s}$$

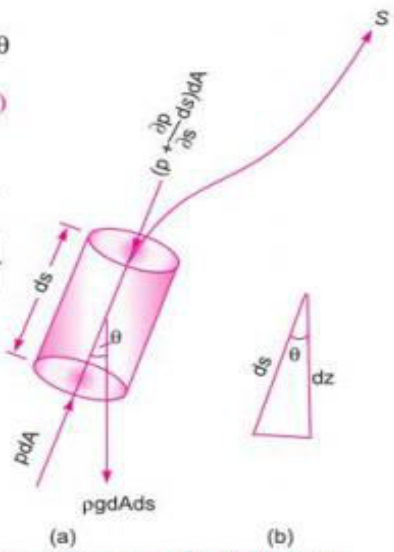


Fig. 6.1 *Forces on a fluid element.*

Substituting the value of a_s in equation (6.2) and simplifying the equation, we get

$$-\frac{\partial p}{\partial s} dsdA - \rho g\, dAds \cos\theta = \rho dAds \times \frac{\partial v}{\partial s}$$

Dividing by $\rho dsdA$, $-\frac{\partial p}{\rho \partial s} - g\cos\theta = \frac{v\partial v}{\partial s}$

or $$\frac{\partial p}{\rho \partial s} + g \cos \theta + v \frac{\partial v}{\partial s} = 0$$

But from Fig. 6.1 (*b*), we have $\cos \theta = \dfrac{dz}{ds}$

$$\therefore \quad \frac{1}{\rho}\frac{dp}{ds} + g\frac{dz}{ds} + \frac{vdv}{ds} = 0 \quad \text{or} \quad \frac{dp}{\rho} + gdz + vdv = 0$$

or $$\frac{dp}{\rho} + gdz + vdv = 0 \qquad ...(6.3)$$

Equation (6.3) is known as Euler's equation of motion.

▶ 6.4 BERNOULLI'S EQUATION FROM EULER'S EQUATION

Bernoulli's equation is obtained by integrating the Euler's equation of motion (6.3) as

$$\int \frac{dp}{\rho} + \int gdz + \int vdv = \text{constant}$$

If flow is incompressible, ρ is constant and

$$\therefore \quad \frac{p}{\rho} + gz + \frac{v^2}{2} = \text{constant}$$

or $$\frac{p}{\rho g} + z + \frac{v^2}{2g} = \text{constant}$$

or $$\frac{p}{\rho g} + \frac{v^2}{2g} + z = \text{constant} \qquad ...(6.4)$$

Equation (6.4) is a Bernoulli's equation in which

$\dfrac{p}{\rho g}$ = pressure energy per unit weight of fluid or pressure head.

$v^2/2g$ = kinetic energy per unit weight or kinetic head.

z = potential energy per unit weight or potential head.

▶ 6.6 BERNOULLI'S EQUATION FOR REAL FLUID

The Bernoulli's equation was derived on the assumption that fluid is inviscid (non-viscous) and therefore frictionless. But all the real fluids are viscous and hence offer resistance to flow. Thus there are always some losses in fluid flows and hence in the application of Bernoulli's equation, these losses have to be taken into consideration. Thus the Bernoulli's equation for real fluids between points 1 and 2 is given as

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 + h_L \qquad ...(6.5)$$

where h_L is loss of energy between points 1 and 2.

▶ 6.7 PRACTICAL APPLICATIONS OF BERNOULLI'S EQUATION

Bernoulli's equation is applied in all problems of incompressible fluid flow where energy considerations are involved. But we shall consider its application to the following measuring devices :

1. Venturimeter.
2. Orifice meter.
3. Pitot-tube.

6.7.1 Venturimeter.

A venturimeter is a device used for measuring the rate of a flow of a fluid flowing through a pipe. It consists of three parts :

(*i*) A short converging part, (*ii*) Throat, and (*iii*) Diverging part. It is based on the Principle of Bernoulli's equation.

Expression for rate of flow through venturimeter

Consider a venturimeter fitted in a horizontal pipe through which a fluid is flowing (say water), as shown in Fig. 6.9.

Let d_1 = diameter at inlet or at section (1),

p_1 = pressure at section (1)

v_1 = velocity of fluid at section (1),

a = area at section (1) = $\frac{\pi}{4} d_1^2$

and d_2, p_2, v_2, a_2 are corresponding values at section (2).

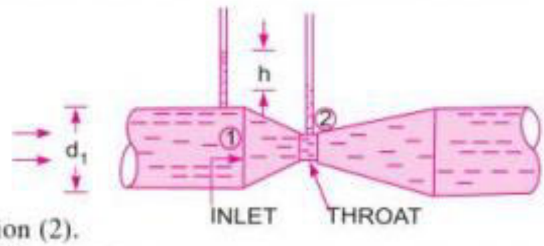


Fig. 6.9 *Venturimeter.*

Applying Bernoulli's equation at sections (1) and (2), we get

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

As pipe is horizontal, hence $z_1 = z_2$

$$\therefore \quad \frac{p_1}{\rho g} + \frac{v_1^2}{2g} = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} \quad \text{or} \quad \frac{p_1 - p_2}{\rho g} = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

But $\frac{p_1 - p_2}{\rho g}$ is the difference of pressure heads at sections 1 and 2 and it is equal to h or $\frac{p_1 - p_2}{\rho g} = h$

Substituting this value of $\frac{p_1 - p_2}{\rho g}$ in the above equation, we get

$$h = \frac{v_2^2}{2g} - \frac{v_1^2}{2g} \quad \text{...(6.6)}$$

Now applying continuity equation at sections 1 and 2

$$a_1 v_1 = a_2 v_2 \quad \text{or} \quad v_1 = \frac{a_2 v_2}{a_1}$$

Substituting this value of v_1 in equation (6.6)

$$h = \frac{v_2^2}{2g} - \frac{\left(\frac{a_2 v_2}{a_1}\right)^2}{2g} = \frac{v_2^2}{2g}\left[1 - \frac{a_2^2}{a_1^2}\right] = \frac{v_2^2}{2g}\left[\frac{a_1^2 - a_2^2}{a_1^2}\right]$$

or $$v_2^2 = 2gh \frac{a_1^2}{a_1^2 - a_2^2}$$

$\therefore$ $$v_2 = \sqrt{2gh \frac{a_1^2}{a_1^2 - a_2^2}} = \frac{a_1}{\sqrt{a_1^2 - a_2^2}} \sqrt{2gh}$$

$\therefore$ Discharge, $$Q = a_2 v_2$$

$$= a_2 \frac{a_1}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh} = \frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh} \quad \text{...(6.7)}$$

Equation (6.7) gives the discharge under ideal conditions and is called, theoretical discharge. Actual discharge will be less than theoretical discharge.

$\therefore$ $$Q_{\text{act}} = C_d \times \frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh} \quad \text{...(6.8)}$$

where C_d = Co-efficient of venturimeter and its value is less than 1.

Value of 'h' given by differential U-tube manometer

Case I. Let the differential manometer contains a liquid which is heavier than the liquid flowing through the pipe. Let

S_h = Sp. gravity of the heavier liquid
S_o = Sp. gravity of the liquid flowing through pipe
x = Difference of the heavier liquid column in U-tube

Then $$h = x\left[\frac{S_h}{S_o} - 1\right] \quad \text{...(6.9)}$$

Case II. If the differential manometer contains a liquid which is lighter than the liquid flowing through the pipe, the value of h is given by

$$h = x\left[1 - \frac{S_l}{S_o}\right] \qquad \text{...(6.10)}$$

where S_l = Sp. gr. of lighter liquid in U-tube

S_o = Sp. gr. of fluid flowing through pipe

x = Difference of the lighter liquid columns in U-tube.

Case III. Inclined Venturimeter with Differential U-tube manometer. The above two cases are given for a horizontal venturimeter. This case is related to inclined venturimeter having differential U-tube manometer. Let the differential manometer contains heavier liquid then h is given as

$$h = \left(\frac{p_1}{\rho g} + z_1\right) - \left(\frac{p_2}{\rho g} + z_2\right) = x\left[\frac{S_h}{S_o} - 1\right] \qquad \text{...(6.11)}$$

Case IV. Similarly, for inclined venturimeter in which differential manometer contains a liquid which is lighter than the liquid flowing through the pipe, the value of h is given as

$$h = \left(\frac{p_1}{\rho g} + z_1\right) - \left(\frac{p_2}{\rho g} + z_2\right) = x\left[1 - \frac{S_l}{S_o}\right] \qquad \text{...(6.12)}$$

6.7.2 Orifice Meter or Orifice Plate. It is a device used for measuring the rate of flow of a fluid through a pipe. It is a cheaper device as compared to venturimeter. It also works on the same principle as that of venturimeter. It consists of a flat circular plate which has a circular sharp edged hole called orifice, which is concentric with the pipe. The orifice diameter is kept generally 0.5 times the diameter of the pipe, though it may vary from 0.4 to 0.8 times the pipe diameter.

A differential manometer is connected at section (1), which is at a distance of about 1.5 to 2.0 times the pipe diameter upstream from the orifice plate, and at section (2), which is at a distance of about half the diameter of the orifice on the downstream side from the orifice plate.

Fig. 6.12. Orifice meter.

Let p_1 = pressure at section (1),

v_1 = velocity at section (1),

a_1 = area of pipe at section (1), and

p_2, v_2, a_2 are corresponding values at section (2). Applying Bernoulli's equation at sections (1) and (2), we get

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

or
$$\left(\frac{p_1}{\rho g} + z_1\right) - \left(\frac{p_2}{\rho g} + z_2\right) = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

But
$$\left(\frac{p_1}{\rho g} + z_1\right) - \left(\frac{p_2}{\rho g} + z_2\right) = h = \text{Differential head}$$

$\therefore$
$$h = \frac{v_2^2}{2g} - \frac{v_1^2}{2g} \quad \text{or} \quad 2gh = v_2^2 - v_1^2$$

Now section (2) is at the vena-contracta and a_2 represents the area at the vena-contracta. If a_0 is the area of orifice then, we have

$$C_c = \frac{a_2}{a_0}$$

where C_c = Co-efficient of contraction

$\therefore$
$$a_2 = a_0 \times C_c \qquad ...(ii)$$

By continuity equation, we have

$$a_1 v_1 = a_2 v_2 \quad \text{or} \quad v_1 = \frac{a_2}{a_1} v_2 = \frac{a_0\, C_c}{a_1} v_2 \qquad ...(iii)$$

Substituting the value of v_1 in equation (*i*), we get

$$v_2 = \sqrt{2gh + \frac{a_0^2 C_c^2 v_2^2}{a_1^2}}$$

or $$v_2^2 = 2gh + \left(\frac{a_0}{a_1}\right)^2 C_c^2 v_2^2 \text{ or } v_2^2 \left[1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2\right] = 2gh$$

$\therefore$ $$v_2 = \frac{\sqrt{2gh}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2}}$$

$\therefore$ The discharge $Q = v_2 \times a_2 = v_2 \times a_0\, C_c$ $\{\because a_2 = a_0 C_c \text{ from } (ii)\}$

$$= \frac{a_0 C_c \sqrt{2gh}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2}} \qquad \ldots(iv)$$

The above expression is simplified by using

$$C_d = C_c \frac{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2}}$$

$\therefore$ $$C_c = C_d \frac{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2}}$$

Substituting this value of C_c in equation (*iv*), we get

$$Q = a_0 \times C_d \frac{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2}} \times \frac{\sqrt{2gh}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2 C_c^2}}$$

$$= \frac{C_d a_0 \sqrt{2gh}}{\sqrt{1 - \left(\frac{a_0}{a_1}\right)^2}} = \frac{C_d a_0 a_1 \sqrt{2gh}}{\sqrt{a_1^2 - a_0^2}}. \qquad \ldots(6.13)$$

where C_d = Co-efficient of discharge for orifice meter.

6.7.3 Pitot-tube. It is a device used for measuring the velocity of flow at any point in a pipe or a channel. It is based on the principle that if the velocity of flow at a point becomes zero, the pressure there is increased due to the conversion of the kinetic energy into pressure energy. In its simplest form, the pitot-tube consists of a glass tube, bent at right angles as shown in Fig. 6.13.

The lower end, which is bent through 90° is directed in the upstream direction as shown in Fig. 6.13. The liquid rises up in the tube due to the conversion of kinetic energy into pressure energy. The velocity is determined by measuring the rise of liquid in the tube.

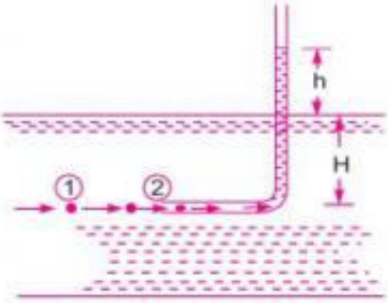


Fig. 6.13 *Pitot-tube.*

Consider two points (1) and (2) at the same level in such a way that point (2) is just as the inlet of the pitot-tube and point (1) is far away from the tube.

Let p_1 = intensity of pressure at point (1)

v_1 = velocity of flow at (1)

p_2 = pressure at point (2)

v_2 = velocity at point (2), which is zero

H = depth of tube in the liquid

h = rise of liquid in the tube above the free surface.

Applying Bernoulli's equation at points (1) and (2), we get

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

But $z_1 = z_2$ as points (1) and (2) are on the same line and $v_2 = 0$.

$$\frac{p_1}{\rho g} = \text{pressure head at (1)} = H$$

$$\frac{p_2}{\rho g} = \text{pressure head at (2)} = (h + H)$$

Substituting these values, we get

$$\therefore \quad H + \frac{v_1^2}{2g} = (h + H) \quad \therefore \quad h = \frac{v_1^2}{2g} \quad \text{or} \quad v_1 = \sqrt{2gh}$$

This is theoretical velocity. Actual velocity is given by

$$(v_1)_{act} = C_v \sqrt{2gh}$$

where C_v = Co-efficient of pitot-tube

$$\therefore \quad \text{Velocity at any point} \quad v = C_v \sqrt{2gh} \qquad ...(6.14)$$

$$(v_1)_{act} = C_v \sqrt{2gh}$$

where C_v = Co-efficient of pitot-tube

$$\therefore \quad \text{Velocity at any point} \quad v = C_v \sqrt{2gh} \qquad ...(6.14)$$

Velocity of flow in a pipe by pitot-tube. For finding the velocity at any point in a pipe by pitot-tube, the following arrangements are adopted :

1. Pitot-tube along with a vertical piezometer tube as shown in Fig. 6.14.
2. Pitot-tube connected with piezometer tube as shown in Fig. 6.15.
3. Pitot-tube and vertical piezometer tube connected with a differential *U*-tube manometer as shown in Fig. 6.16.

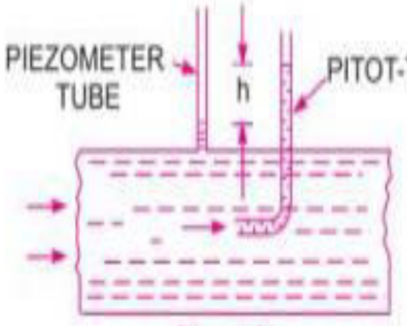


Fig. 6.14

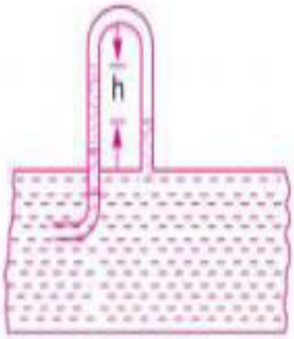


Fig. 6.15

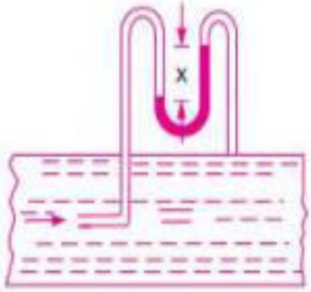


Fig. 6.16

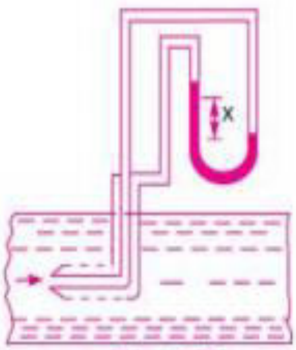


Fig. 6.17

4. Pitot-static tube, which consists of two circular concentric tubes one inside the other with some annular space in between as shown in Fig. 6.17. The outlet of these two tubes are connected to the differential manometer where the difference of pressure head 'h' is measured by knowing the difference of the levels of the manometer liquid say x. Then $h = x\left[\frac{S_g}{S_o} - 1\right]$.

▶ 6.8 THE MOMENTUM EQUATION

It is based on the law of conservation of momentum or on the momentum principle, which states that the net force acting on a fluid mass is equal to the change in momentum of flow per unit time in that direction. The force acting on a fluid mass 'm' is given by the Newton's second law of motion,

$$F = m \times a$$

where a is the acceleration acting in the same direction as force F.

But $$a = \frac{dv}{dt}$$

$\therefore$ $$F = m\frac{dv}{dt}$$

$$= \frac{d(mv)}{dt} \quad \{m \text{ is constant and can be taken inside the differential}\}$$

$\therefore$ $$F = \frac{d(mv)}{dt} \quad \text{...(6.15)}$$

Equation (6.15) is known as the momentum principle.

Equation (6.15) can be written as $F.dt = d(mv)$ ...(6.16)

which is known as the *impulse-momentum equation* and states that the impulse of a force F acting on a fluid of mass m in a short interval of time dt is equal to the change of momentum $d(mv)$ in the direction of force.

Force exerted by a flowing fluid on a pipe bend

The impulse-momentum equation (6.16) is used to determine the resultant force exerted by a flowing fluid on a pipe bend.

Consider two sections (1) and (2), as shown in Fig. 6.18.

Let v_1 = velocity of flow at section (1),

p_1 = pressure intensity at section (1),

A_1 = area of cross-section of pipe at section (1) and

v_2, p_2, A_2 = corresponding values of velocity, pressure and area at section (2).

Let F_x and F_y be the components of the forces exerted by the flowing fluid on the bend in x-and y-directions respectively. Then the force exerted by the bend on the fluid in the directions of x and y will be equal to F_x and F_y but in the opposite directions. Hence component of the force exerted by bend on the fluid in the x-direction = $-F_x$ and in the direction of $y = -F_y$. The other external forces acting on the fluid are p_1A_1 and p_2A_2 on the sections (1) and (2) respectively. Then momentum equation in x-direction is given by

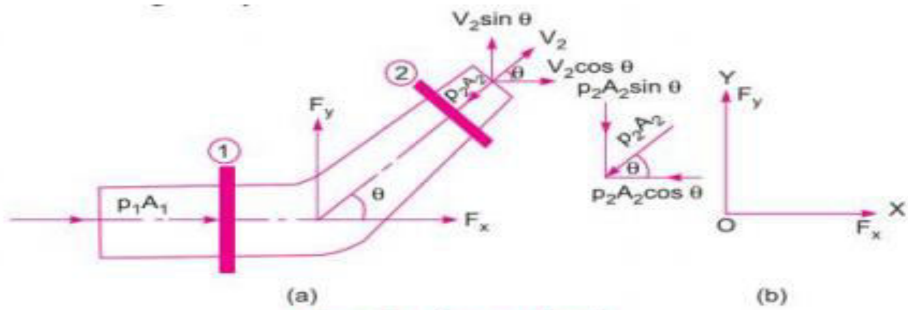


Fig. 6.18 *Forces on bend.*

Net force acting on fluid in the direction of x = Rate of change of momentum in x-direction

$\therefore \quad p_1A_1 - p_2A_2 \cos\theta - F_x$ = (Mass per sec) (change of velocity)

$= \rho Q$ (Final velocity in the direction of x − Initial velocity in the direction of x)

$$= \rho Q\,(V_2 \cos\theta - V_1) \qquad \text{...(6.17)}$$

$$\therefore \quad F_x = \rho Q\,(V_1 - V_2\cos\theta) + p_1A_1 - p_2A_2\cos\theta \qquad \text{...(6.18)}$$

Similarly the momentum equation in y-direction gives

$$0 - p_2A_2 \sin\theta - F_y = \rho Q\,(V_2\sin\theta - 0) \qquad \text{...(6.19)}$$

$$\therefore \quad F_y = \rho Q\,(-V_2\sin\theta) - p_2A_2\sin\theta \qquad \text{...(6.20)}$$

Now the resultant force (F_R) acting on the bend

$$= \sqrt{F_x^2 + F_y^2} \qquad \text{...(6.21)}$$

And the angle made by the resultant force with horizontal direction is given by

$$\tan\theta = \frac{F_y}{F_x} \qquad \text{...(6.22)}$$

MODELS OF THE FLUID: CONTROL VOLUMES AND FLUID ELEMENTS

Aerodynamics is a fundamental science, steeped in physical observation. As you proceed through this presentation, make every effort to gradually develop a “physical feel” for the material.

An important virtue of all successful aerodynamicists (indeed, of all successful engineers and scientists) is that they have good “physical intuition,” based on thought and experience, which allows them to make reasonable judgments on difficult problems.

Although this presentation is full of equations and (seemingly) esoteric concepts, now is the time for you to start developing this physical feel. With this section, we begin to build the basic equations of aerodynamics. There is a certain philosophical procedure involved with the development of hese equations, as follows:

1. Invoke three fundamental physical principles that are deeply entrenched in our macroscopic observations of nature, namely,

    a. Mass is conserved (i.e., mass can be neither created nor destroyed).

    b. Newton’s second law: force = mass × acceleration.

    c. Energy is conserved; it can only change from one form to another.

2. Determine a suitable model of the fluid. Remember that a fluid is a squishy substance, and therefore it is usually more difficult to describe than a well-defined solid body. Hence, we have to adopt a reasonable model of the fluid to which we can apply the fundamental principles stated in item 1.

3. Apply the fundamental physical principles listed in item 1 to the model of the fluid determined in item 2 in order to obtain mathematical equations which properly describe the physics of the flow. In turn, use these fundamental equations to analyse any particular aerodynamic flow problem of interest.

Finite Control Volume Approach

Consider a general flow field as represented by the streamlines in Figure 2.13. Let us imagine a closed volume drawn within a finite region of the flow.

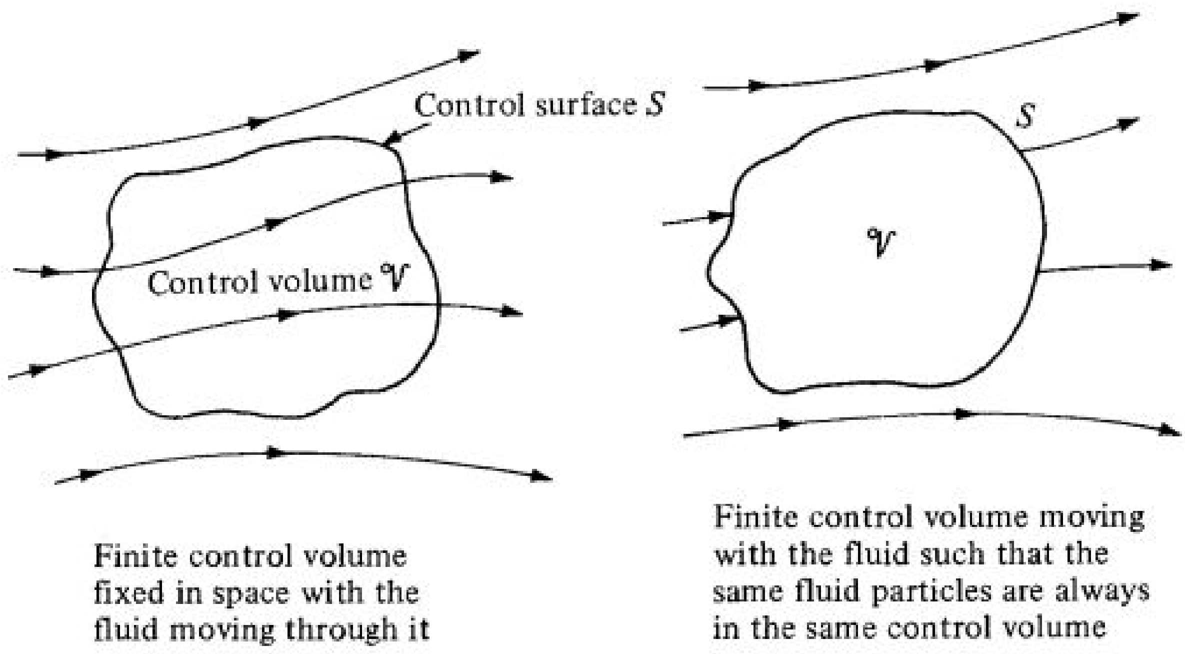


Figure 2.13 Finite control volume approach.

This volume defines a control volume V, and a control surface S is defined as the closed surface which bounds the control volume.

The control volume may be fixed in space with the fluid moving through it, as shown at the left of Figure 2.13. Alternatively, the control volume may be moving with the fluid such that the same fluid particles are always inside it, as shown at the right of Figure 2.13.

In either case, the control volume is a reasonably large, finite region of the flow. The fundamental physical principles are applied to the fluid inside the control volume, and to the fluid crossing the control surface (if the control volume is fixed in space).

Therefore, instead of looking at the whole flow field at once, with the control volume model we limit our attention to just the fluid in the finite region of the volume itself.

Infinitesimal Fluid Element Approach

Consider a general flow field as represented by the streamlines in Figure 2.14. Let us imagine an infinitesimally small fluid element in the flow, with a differential volume dV.

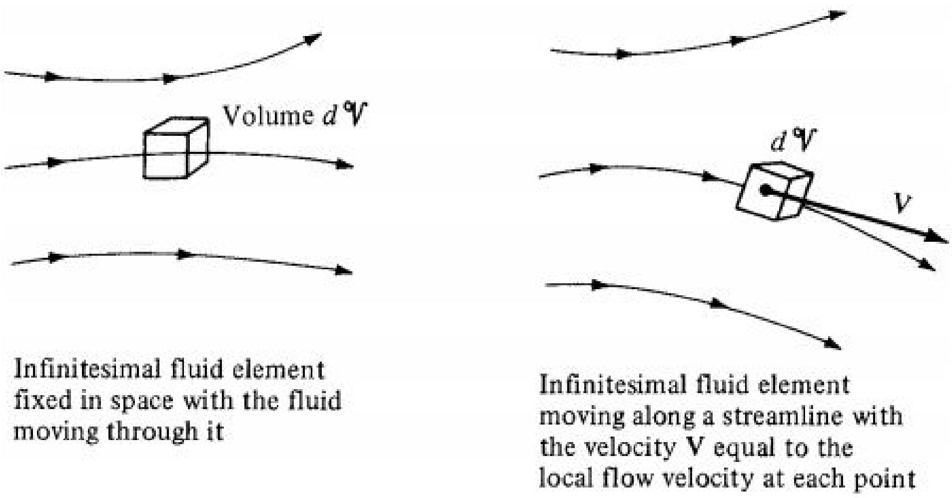


Figure. 7 Infinitesimal fluid element approach

The fluid element is infinitesimal in the same sense as differential calculus; however, it is large enough to contain a huge number of molecules so that it can be viewed as a continuous medium.

The fluid element may be fixed in space with the fluid moving through it, as shown at the left of Figure 7. Alternatively, it may be moving along a streamline with velocity V equal to the flow velocity at each point.

Again, instead of looking at the whole flow field at once, the fundamental physical principles are applied to just the fluid element itself.

Molecular Approach

In actuality, of course, the motion of a fluid is a ramification of the mean motion of its atoms and molecules. Therefore, a third model of the flow can be a microscopic approach wherein the fundamental laws of nature are applied directly to the atoms and molecules, using suitable statistical averaging to define the resulting fluid properties.

This approach is in the purview of kinetic theory, which is a very elegant method with many advantages in the long run. However, it is beyond the scope of the present book.

In summary, although many variations on the theme can be found in different texts for the derivation of the general equations of fluid flow, the flow model can usually be categorized under one of the approaches described above.

Physical Meaning of the Divergence of Velocity

In the equations to follow, the divergence of velocity, $\nabla \cdot V$, occurs frequently. Before leaving this section, let us prove the statement made earlier that $\nabla \cdot V$ is physically the time rate of change of the volume of a moving fluid element of fixed mass per unit volume of that element.

Consider a control volume moving with the fluid (the case shown on the right of Figure 7). This control volume is always made up of the same fluid particles as it moves with the flow; hence, its mass is fixed, invariant with time. However, its volume V and control surface S are changing with time as it moves to different regions of the flow where different values of ρ exist.

That is, this moving control volume of fixed mass is constantly increasing or decreasing its volume and is changing its shape, depending on the characteristics of the flow. This control volume is shown in Figure 2.15 at some instant in time. Consider an infinitesimal element of the surface d S moving at the local velocity V, as shown in Figure 2.15.

The change in the volume of the control volume ΔV, due to just the movement of dS over a time increment Δ t, is, from Figure 2.15, equal to the volume of the long, thin cylinder with base area d S and altitude (V Δ t)· n; that is,

$$\Delta V=[(V\,\Delta t)\cdot n]d\,S = (V\,\Delta t)\cdot dS \quad (2.28)$$

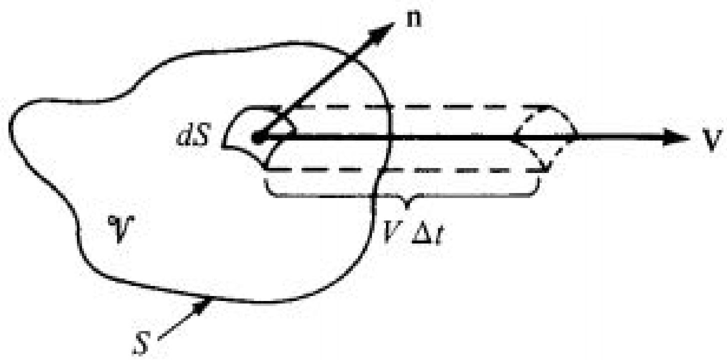


Figure 8. Moving control volume used for the physical interpretation of the divergence of velocity.

Over the time increment Δ t, the total change in volume of the whole control volume is equal to the summation of Equation (2.28) over the total control surface. In the limit as d S → 0, the sum becomes the surface integral

$$\oint\!\!\oint_S (\mathbf{V}\Delta t)\cdot \mathbf{dS}$$

If this integral is divided by Δ t, the result is physically the time rate of change of the control volume, denoted by DV/Dt; that i

$$\frac{D\mathcal{V}}{Dt} = \frac{1}{\Delta t}\oint\!\!\oint_S (\mathbf{V}\Delta t)\cdot \mathbf{dS} = \oint\!\!\oint_S \mathbf{V}\cdot \mathbf{dS} \qquad (2.29)$$

(The significance of the notation D/Dt is revealed) Applying the divergence theorem, Equation (2.26), to the right side of Equation (2.29), we have

$$\frac{D\mathcal{V}}{Dt} = \oint\!\!\oint\!\!\oint_{\mathcal{V}} (\nabla\cdot\mathbf{V})d\mathcal{V} \qquad (2.30)$$

Now let us imagine that the moving control volume in Figure 7 is shrunk to a very small volume δV, essentially becoming an infinitesimal moving fluid element as sketched on the right of Figure 2.14. Then Equation (2.30) can be written as

$$\frac{D(\delta \mathcal{V})}{Dt} = \oiiint_{\delta \mathcal{V}} (\nabla \cdot \mathbf{V}) d\mathcal{V} \tag{2.31}$$

Assume that δV is small enough such that $\nabla \cdot V$ is essentially the same value throughout δV. Then the integral in Equation (2.31) can be approximated as $(\nabla \cdot V)\delta V$. From Equation (2.31), we have

$$\frac{D(\delta \mathcal{V})}{Dt} = (\nabla \cdot \mathbf{V}) \delta \mathcal{V}$$

$$\boxed{\nabla \cdot \mathbf{V} = \frac{1}{\delta \mathcal{V}} \frac{D(\delta \mathcal{V})}{Dt}} \tag{2.32}$$

Examine Equation (2.32). It states that $\nabla \cdot V$ is physically the time rate of change of the volume of a moving fluid element, per unit volume. Hence, the interpretation of $\nabla \cdot V$, first given , Divergence of a Vector Field, is now proved.

CONTINUITY EQUATION

Consider a given area A arbitrarily oriented in a flow field as shown in Figure 8.

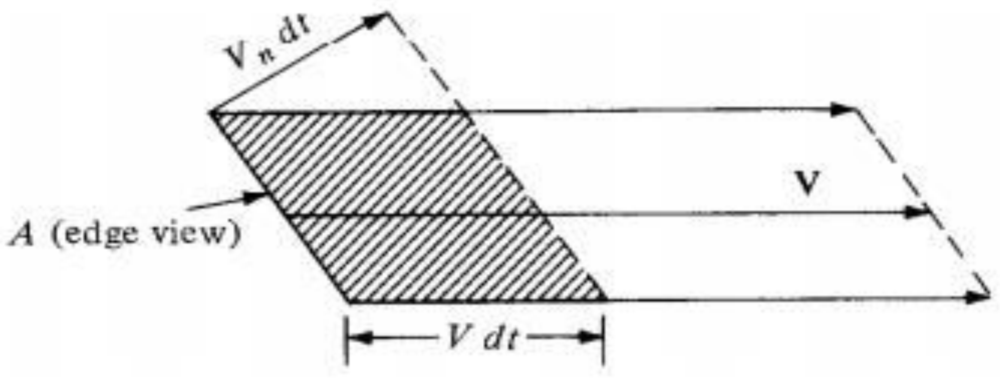


Figure 8. Sketch for discussion of mass flow through area A in a flow field.

In Figure 8, we are looking at an edge view of area A. Let A be small enough such that the flow velocity V is uniform across A.

Consider the fluid elements with velocity V that pass through A. In time dt after crossing A, they have moved a distance V dt and have swept out the shaded volume shown in Figure 8.

This volume is equal to the base area A times the height of the cylinder $V_n dt$, where V_n is the component of velocity normal to A; that is,

Volume = $(V_n dt)A$

The mass inside the shaded volume is therefore

Mass = $\rho(V_n dt)A$ (2.42)

This is the mass that has swept past A in time dt. By definition, the mass flow through A is the mass crossing A per second (e.g., kilograms per second, slugs per second). Let $\dot{m}$ denote mass flow. From Equation (2.42).

$$\dot{m} = \frac{\rho(V_n\, dt)A}{dt}$$

$$\boxed{\dot{m} = \rho V_n A} \qquad (2.43)$$

Equation (2.43) demonstrates that mass flow through A is given by the product

Area × density × component of flow velocity normal to the area

A related concept is that of mass flux, defined as the mass flow per unit area.

$$\text{Mass flux} = \frac{\dot{m}}{A} = \rho V_n$$

Typical units of mass flux are $kg/(s \cdot m^2)$ and $slug/(s \cdot ft^2)$.

- The concepts of mass flow and mass flux are important. Note from Equation (2.44) that mass flux across a surface is equal to the product of density times the component of velocity perpendicular to the surface.
- Many of the equations of aerodynamics involve products of density and velocity.
- For example, in cartesian coordinates, $V = V_x i + V_y j + V_z k = ui + vj + wk$, where u, v, and w denote the x, y, and z components of velocity, respectively. (The use of u, v, and w rather than Vx , Vy , and Vz to symbolize the x, y, and z components of velocity is quite common in aerodynamic literature; we henceforth adopt the u, v, and w notation.)
- In many of the equations of aerodynamics, you will find the products ρu, ρv, and ρw; always remember that these products are the mass fluxes in the x, y, and z directions, respectively.
- In a more general sense, if V is the magnitude of velocity in an arbitrary direction, the product ρV is physically the mass flux (mass flow per unit area) across an area oriented perpendicular to the direction of V.
- We are now ready to apply our first physical principle to a finite control volume fixed in space.

Physical principle Mass can be neither created nor destroyed.

- Consider a flow field wherein all properties vary with spatial location and time, for example, $\rho = \rho(x, y, z, t)$. In this flow field, consider the fixed finite control volume shown in Figure 9.
- At a point on the control surface, the flow velocity is V and the vector elemental surface area is dS. Also dV is an elemental volume inside the control volume. Applied to this control volume, the above physical principle means
- Net mass flow out of control volume through surface S = time rate of decrease of mass inside control volume V

(2.45a)

or $B = C$ (2.45b)

- where B andC are just convenient symbols for the left and right sides, respectively, of Equation (2.45a).
- First, let us obtain an expression for B in terms of the quantities shown in Figure 2.19.
- From Equation (2.43), the elemental mass flow across the area d S is

$$\rho V_n \, dS = \rho \mathbf{V} \cdot d\mathbf{S}$$

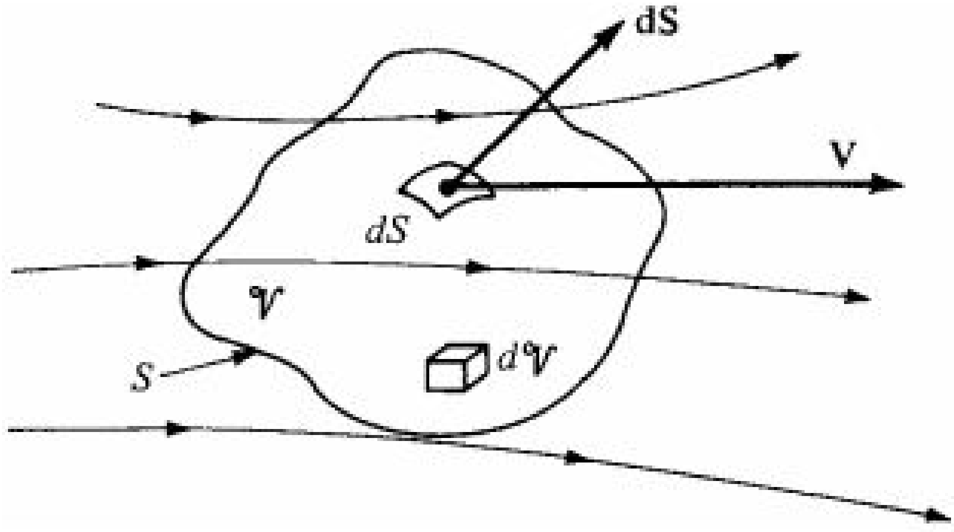


Figure 9. Finite control volume fixed in space

- Examining Figure 9, note that by convention, dS always points in a direction out of the control volume. Hence, when V also points out of the control volume (as shown in Figure 9), the product $\rho V \cdot dS$ is positive.
- Moreover, when V points out of the control volume, the mass flow is physically leaving the control volume (i.e., it is an outflow). Hence, a positive $\rho V \cdot dS$ denotes an outflow.
- In turn, when V points into the control volume, $\rho V \cdot dS$ is negative. Moreover, when V points inward, the mass flow is physically entering the control volume (i.e., it is an inflow).
- Hence, a negative $\rho V \cdot dS$ denotes an inflow. The net mass flow out of the entire control surface S is the summation over S of the elemental mass flows.

flows. In the limit, this becomes a surface integral, which is physically the left side of Equations (2.45a and b); that is,

$$B = \oiint_S \rho \mathbf{V} \cdot \mathbf{dS} \tag{2.46}$$

Now consider the right side of Equations (2.45a and b). The mass contained within the elemental volume $d\mathcal{V}$ is

$$\rho \, d\mathcal{V}$$

Hence, the total mass inside the control volume is

$$\oiiint_{\mathcal{V}} \rho \, d\mathcal{V}$$

The time rate of *increase* of mass inside $\mathcal{V}$ is then

$$\frac{\partial}{\partial t} \oiiint_{\mathcal{V}} \rho \, d\mathcal{V}$$

In turn, the time rate of *decrease* of mass inside $\mathcal{V}$ is the negative of the above; that is

$$-\frac{\partial}{\partial t} \oiiint_{\mathcal{V}} \rho \, d\mathcal{V} = C \tag{2.47}$$

Thus, substituting Equations (2.46) and (2.47) into (2.45b), we have

$$\boxed{\oiint_S \rho \mathbf{V} \cdot \mathbf{dS} = -\frac{\partial}{\partial t} \oiiint_{\mathcal{V}} \rho \, d\mathcal{V}}$$

or

$$\boxed{\frac{\partial}{\partial t} \oiiint_{\mathcal{V}} \rho \, d\mathcal{V} + \oiint_S \rho \mathbf{V} \cdot \mathbf{dS} = 0} \tag{2.48}$$

Equation (2.48) is the final result of applying the physical principle of the conservation of mass to a finite control volume fixed in space. Equation (2.48) is called the continuity equation. It is one of the most fundamental equations of fluid dynamics.

Note that Equation (2.48) expresses the continuity equation in integral form. We will have numerous opportunities to use this form; it has the advantage of relating aerodynamic phenomena over a finite region of space without being concerned about the details of precisely what is happening at a given distinct point in the flow.

On the other hand, there are many times when we are concerned with the details of a flow and we want to have equations that relate flow properties at a given point. In such a case, the integral form as expressed in Equation (2.48) is not particularly useful. However, Equation (2.48) can be reduced to another form that does relate flow properties at a given point, as follows.

To begin with, since the control volume used to obtain Equation (2.48) is fixed in space, the limits of integration are also fixed. Hence, the time derivative can be placed inside the volume integral and Equation (2.48) can be written as

$$\oiiint_{\mathcal{V}} \frac{\partial \rho}{\partial t} \, d\mathcal{V} + \oiint_{S} \rho \mathbf{V} \cdot \mathbf{dS} = 0 \tag{2.49}$$

Applying the divergence theorem, Equation (2.26), we can express the right-hand term of Equation (2.49) as

$$\oiint_{S} (\rho \mathbf{V}) \cdot \mathbf{dS} = \oiiint_{\mathcal{V}} \nabla \cdot (\rho \mathbf{V}) \, d\mathcal{V} \tag{2.50}$$

Substituting Equation (2.50) into (2.49), we obtain

$$\oiiint_{\mathcal{V}} \frac{\partial \rho}{\partial t} \, d\mathcal{V} + \oiiint_{\mathcal{V}} \nabla \cdot (\rho \mathbf{V}) \, d\mathcal{V} = 0$$

or

$$\oiiint_{\mathcal{V}} \left[\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) \right] d\mathcal{V} = 0 \tag{2.51}$$

Examine the integrand of Equation (2.51). If the integrand were a finite number, then Equation (2.51) would require that the integral over part of the control volume be equal and opposite in sign to the integral over the remainder of the control volume, such that the net integration would be zero.

However, the finite control volume is arbitrarily drawn in space; there is no reason to expect cancellation of one region by the other. Hence, the only way for the integral in Equation (2.51) to be zero for an arbitrary control volume is for the integrand to be zero at all points within the control volume. Thus, from Equation (2.51), we have

$$\boxed{\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0} \tag{2.52}$$

Equation (2.52) is the continuity equation in the form of a partial differential equation. This equation relates the flow field variables at a point in the flow, as opposed to Equation (2.48), which deals with a finite space.

It is important to keep in mind that Equations (2.48) and (2.52) are equally valid statements of the physical principle of conservation of mass.

They are mathematical representations, but always remember that they speak words—they say that mass can be neither created nor destroyed.

Note that in the derivation of the above equations, the only assumption about the nature of the fluid is that it is a continuum. Therefore, Equations (2.48) and (2.52) hold in general for the three-dimensional, unsteady flow of any type of fluid, inviscid or viscous, compressible or incompressible.

Note: It is important to keep track of all assumptions that are used in the derivation of any equation because they tell you the limitations on the final result, and therefore prevent you from using an equation for a situation in which it is not valid. In all our future derivations, develop the habit of noting all assumptions that go with the resulting equations.

It is important to emphasize the difference between unsteady and steady flows. In an unsteady flow, the flow-field variables are a function of both spatial location and time, for example,

$$\rho = \rho(x, y, z, t)$$

This means that if you lock your eyes on one fixed point in space, the density at that point will change with time. Such unsteady fluctuations can be caused by time-varying boundaries (e.g., an airfoil pitching up and down with time or the supply valves of a wind tunnel being turned off and on).

Equations (2.48) and (2.52) hold for such unsteady flows. On the other hand, the vast majority of practical aerodynamic problems involve steady flow. Here, the flow-field variables are a function of spatial location only, for example,

$$\rho = \rho(x, y, z)$$

This means that if you lock your eyes on a fixed point in space, the density at that point will be a fixed value, invariant with time. For steady flow, $\partial/\partial t = 0$, and hence Equations (2.48) and (2.52) reduce to

$$\boxed{\oiint_S \rho \mathbf{V} \cdot \mathbf{dS} = 0} \qquad (2.53)$$

$$\boxed{\nabla \cdot (\rho \mathbf{V}) = 0} \qquad (2.54)$$

MOMENTUM EQUATION

Newton's second law is frequently written as

$$F = ma \qquad (2.55)$$

where F is the force exerted on a body of mass m and a is the acceleration. However, a more general form of Equation (2.55) is

$$F = \frac{d}{dt}(mV) \qquad (2.56)$$

which reduces to Equation (2.55) for a body of constant mass. In Equation (2.56), mV is the momentum of a body of mass m. Equation (2.56) represents the second fundamental principle upon which theoretical fluid dynamics is based. Physical principle

Force = time rate of change of momentum

We will apply this principle [in the form of Equation (2.56)] to the model of a finite control volume fixed in space as sketched in Figure 2.19. Our objective is to obtain expressions for both the left and right sides of Equation (2.56) in terms of the familiar flow-field variables p, ρ, V, etc.

First, let us concentrate on the left side of Equation (2.56) (i.e., obtain an expression for F, which is the force exerted on the fluid as it flows through the control volume). This force comes from two sources:

Body forces: gravity, electromagnetic forces, or any other forces which "act at a distance" on the fluid inside V.

Surface forces: pressure and shear stress acting on the control surface S.

Let f represent the net body force per unit mass exerted on the fluid inside V. The body force on the elemental volume dV in Figure 2.19 is therefore

$\rho f\, dV$

and the total body force exerted on the fluid in the control volume is the summation of the above over the volume V:

$$\text{Body force} = \oiiint_{\mathcal{V}} \rho \mathbf{f}\, d\mathcal{V} \tag{2.57}$$

The elemental surface force due to pressure acting on the element of area d S is

$$-p\, dS$$

where the negative sign indicates that the force is in the direction opposite of dS. That is, the control surface is experiencing a pressure force that is directed into the control volume and which is due to the pressure from the surroundings, and examination of Figure 2.19 shows that such an inward-directed force is in the direction opposite of dS. The complete pressure force is the summation of the elemental forces over the entire control surface:

$$\text{Pressure force} = -\oiint_{S} p\, \mathbf{dS} \tag{2.58}$$

In a viscous flow, the shear and normal viscous stresses also exert a surface force. A detailed evaluation of these viscous stresses is not warranted at this stage of our discussion. Let us simply recognize this effect by letting $F_{viscous}$ denote the total viscous force exerted on the control surface.

We are now ready to write an expression for the left-hand side of Equation (2.56). The total force experienced by the fluid as it is sweeping through the fixed control volume is given by the sum of Equations (2.57) and (2.58) and $F_{viscous}$:

$$\mathbf{F} = \oiiint_{\mathcal{V}} \rho \mathbf{f}\, d\mathcal{V} - \oiint_{S} p\, \mathbf{dS} + \mathbf{F}_{\text{viscous}} \tag{2.59}$$

Now consider the right side of Equation (2.56). The time rate of change of momentum of the fluid as it sweeps through the fixed control volume is the sum of two terms:

$$\begin{array}{c}\text{Net flow of momentum } \textit{out} \\ \text{of control volume across surface } S\end{array} \equiv \mathbf{G} \qquad (2.60a)$$

$$\begin{array}{c}\text{Time rate of change of momentum due to} \\ \text{unsteady fluctuations of flow properties inside } \mathcal{V}\end{array} \equiv \mathbf{H} \qquad (2.60b)$$

Consider the term denoted by G in Equation (2.60a). The flow has a certain momentum as it enters the control volume in Figure 9, and, in general, it has a different momentum as it leaves the control volume (due in part to the force F that is exerted on the fluid as it is sweeping through V).

The net flow of momentum out of the control volume across the surface S is simply this outflow minus the inflow of momentum across the control surface. This change in momentum is denoted by G, as noted above. To obtain an expression for G, recall that the mass flow across the elemental area dS is (ρV · dS); hence, the flow of momentum per second across dS is

$$(\rho \mathbf{V} \cdot \mathbf{dS})\mathbf{V}$$

The net flow of momentum out of the control volume through S is the summation of the above elemental contributions, namely,

$$\mathbf{G} = \oint\!\!\!\oint_S (\rho \mathbf{V} \cdot \mathbf{dS})\mathbf{V} \qquad (2.61)$$

In Equation (2.61), recall that positive values of(ρV · dS)represent mass flow out of the control volume, and negative values represent mass flow into the control volume.

Hence, in Equation (2.61) the integral over the whole control surface is a combination of positive contributions (outflow of momentum) and negative contributions (inflow of momentum), with the resulting value of the integral representing the net outflow of momentum.

If G has a positive value, there is more momentum flowing out of the control volume per second than flowing in; conversely, if G has a negative value, there is more momentum flowing into the control volume per second than flowing out.

Now consider H from Equation (2.60b). The momentum of the fluid in the elemental volume dV shown in Figure 9 is

$$(\rho\, d\mathcal{V})\mathbf{V}$$

The momentum contained at any instant inside the control volume is therefore and its time rate of change due to unsteady flow fluctuations is

$$\oint\!\!\!\oint\!\!\!\oint_{\mathcal{V}} \rho \mathbf{V}\, d\mathcal{V}$$

$$\mathbf{H} = \frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho \mathbf{V}\, d\mathcal{V} \qquad (2.62)$$

Combining Equa e of change of momentum of the fluid as it sweeps through the fixed control volume, which in turn represents the right-hand side of Equation (2.56):

$$\frac{d}{dt}(m\mathbf{V}) = \mathbf{G} + \mathbf{H} = \oiint_{S} (\rho \mathbf{V} \cdot \mathbf{dS})\mathbf{V} + \frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho \mathbf{V}\, d\mathcal{V} \qquad (2.63)$$

Hence, from Equations (2.59) and (2.63), Newton's second law

$$\frac{d}{dt}(m\mathbf{V}) = \mathbf{F}$$

applied to a fluid flow is

$$\boxed{\frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho \mathbf{V}\, d\mathcal{V} + \oiint_{S} (\rho \mathbf{V} \cdot \mathbf{dS})\mathbf{V} = -\oiint_{S} p\, \mathbf{dS} + \oiiint_{\mathcal{V}} \rho \mathbf{f}\, d\mathcal{V} + \mathbf{F}_{\text{viscous}}} \qquad (2.64)$$

Equation (2.64) is the momentum equation in integral form. Note that it is a vector equation. Just as in the case of the integral form of the continuity equation, Equation (2.64) has the advantage of relating aerodynamic phenomena over a finite region of space without being concerned with the details of precisely what is happening at a given distinct point in the flow.

This advantage is illustrated. From Equation (2.64), we now proceed to a partial differential equation which relates flow-field properties at a point in space.

Such an equation is a counterpart to the differential form of the continuity equation given in Equation (2.52). Apply the gradient theorem, Equation (2.27), to the first term on the right side of Equation (2.64):

$$-\oiint_S p\,\mathbf{dS} = -\oiiint_{\mathcal{V}} \nabla p\, d\mathcal{V} \tag{2.65}$$

Also, because the control volume is fixed, the time derivative in Equation (2.64) can be placed inside the integral. Hence, Equation (2.64) can be written as

$$\oiiint_{\mathcal{V}} \frac{\partial(\rho\mathbf{V})}{\partial t} d\mathcal{V} + \oiint_S (\rho\mathbf{V}\cdot\mathbf{dS})\mathbf{V} = -\oiiint_{\mathcal{V}} \nabla p\, d\mathcal{V} + \oiiint_{\mathcal{V}} \rho\mathbf{f}\, d\mathcal{V} + \mathbf{F}_{\text{viscous}} \tag{2.66}$$

Recall that Equation (2.66) is a vector equation. It is convenient to write this equation as three scalar equations. Using cartesian coordinates, where

$$\mathbf{V} = u\mathbf{i} + v\mathbf{j} + w\mathbf{k}$$

the x component of Equation (2.66) is

$$\oiiint_{\mathcal{V}} \frac{\partial(\rho u)}{\partial t} d\mathcal{V} + \oiint_S (\rho\mathbf{V}\cdot\mathbf{dS})u = -\oiiint_{\mathbf{V}} \frac{\partial p}{\partial x} d\mathcal{V} + \oiiint_{\mathcal{V}} \rho f_x\, d\mathcal{V} + (F_x)_{\text{viscous}} \tag{2.67}$$

[*Note:* In Equation (2.67), the product $(\rho\mathbf{V}\cdot\mathbf{dS})$ is a scalar, and therefore has no components.] Apply the divergence theorem, Equation (2.26), to the surface integral on the left side of Equation (2.67):

$$\oiint_S (\rho\mathbf{V}\cdot\mathbf{dS})u = \oiint_S (\rho u\mathbf{V})\cdot\mathbf{dS} = \oiiint_{\mathcal{V}} \nabla\cdot(\rho u\mathbf{V})\, d\mathcal{V} \tag{2.68}$$

Substituting Equation (2.68) into Equation (2.67), we have

$$\oiiint_{\mathcal{V}} \left[\frac{\partial(\rho u)}{\partial t} + \nabla\cdot(\rho u\mathbf{V}) + \frac{\partial p}{\partial x} - \rho f_x - (\mathcal{F}_x)_{\text{viscous}}\right] d\mathcal{V} = 0 \tag{2.69}$$

where $(\mathcal{F}_x)_{\text{viscous}}$ denotes the proper form of the x component of the viscous shear stresses when placed inside the volume integral (this form will be obtained explicitly in Chapter 15). For the same reasons as stated in Section 2.4, the integrand in Equation (2.69) is identically zero at all points in the flow; hence,

$$\boxed{\frac{\partial(\rho u)}{\partial t} + \nabla\cdot(\rho u\mathbf{V}) = -\frac{\partial p}{\partial x} + \rho f_x + (\mathcal{F}_x)_{\text{viscous}}} \tag{2.70a}$$

$$\boxed{\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w \mathbf{V}) = -\frac{\partial p}{\partial z} + \rho f_z + (\mathcal{F}_z)_{\text{viscous}}} \tag{2.70c}$$

where the subscripts y and z on f and $\mathcal{F}$ denote the y and z components of the body and viscous forces, respectively. Equations (2.70*a* to *c*) are the scalar x, y, and z components of the momentum equation, respectively; they are partial differential equations that relate flow-field properties at any point in the flow.

Note that Equations (2.64) and (2.70*a* to *c*) apply to the unsteady, three-dimensional flow of any fluid, compressible or incompressible, viscous or inviscid. Specialized to a steady ($\partial/\partial t \equiv 0$), inviscid ($\mathbf{F}_{\text{viscous}} = 0$) flow with no body forces ($\mathbf{f} = 0$), these equations become

$$\boxed{\oiint_S (\rho \mathbf{V} \cdot \mathbf{dS})\mathbf{V} = -\oiint_S p\, \mathbf{dS}} \tag{2.71}$$

and

$$\nabla \cdot (\rho u \mathbf{V}) = -\frac{\partial p}{\partial x} \tag{2.72a}$$

$$\nabla \cdot (\rho v \mathbf{V}) = -\frac{\partial p}{\partial y} \tag{2.72b}$$

$$\nabla \cdot (\rho w \mathbf{V}) = -\frac{\partial p}{\partial z} \tag{2.72c}$$

Since most of the material in Chapters 3 through 14 assumes steady, inviscid flow with no body forces, we will have frequent occasion to use the momentum equation in the forms of Equations (2.71) and (2.72*a* to *c*).

The momentum equations for an inviscid flow [such as Equations (2.72*a* to *c*)] are called the *Euler equations*. The momentum equations for a viscous flow [such as Equations (2.70*a* to *c*)] are called the *Navier-Stokes equations*. We will encounter this terminology in subsequent chapters.

ENERGY EQUATION

For an incompressible flow, where ρ is constant, the primary flow-field variables are p and V. The continuity and momentum equations obtained earlier are two equations in terms of the two unknowns p and V. Hence, for a study of incompressible flow, the continuity and momentum equations are sufficient tools to do the job.

However, for a compressible flow, ρ is an additional variable, and therefore we need an additional fundamental equation to complete the system. This fundamental relation is the energy equation, to be derived in this section. In the process, two additional flow-field variables arise, namely, the internal energy e and temperature T . Additional equations must also be introduced for these variables, as will be mentioned later in this section.

The material discussed in this section is germane to the study of compressible flow.

Physical principle: Energy can be neither created nor destroyed; it can only change in form.

Consider a fixed amount of matter contained within a closed boundary.

This matter defines the system. Because the molecules and atoms within the system are constantly in motion, the system contains a certain amount of energy. For simplicity, let the system contain a unit mass; in turn, denote the internal energy per unit mass by e. The region outside the system defines the surroundings.

Let an incremental amount of heat δq be added to the system from the surroundings. Also, let δw be the work done on the system by the surroundings.

Both heat and work are forms of energy, and when added to the system, they change the amount of internal energy in the system. Denote this change of internal energy by de. From our physical principle that energy is conserved, we have for the system

$$\delta q + \delta w = de \quad \text{-- -- --} \ (2.85)$$

Equation (2.85) is a statement of the first law of thermodynamics. Let us apply the first law to the fluid flowing through the fixed control volume shown in Figure 10. Let

B1 = rate of heat added to fluid inside control volume from surroundings

B2 = rate of work done on fluid inside control volume

B3 = rate of change of energy of fluid as it flows through control volume

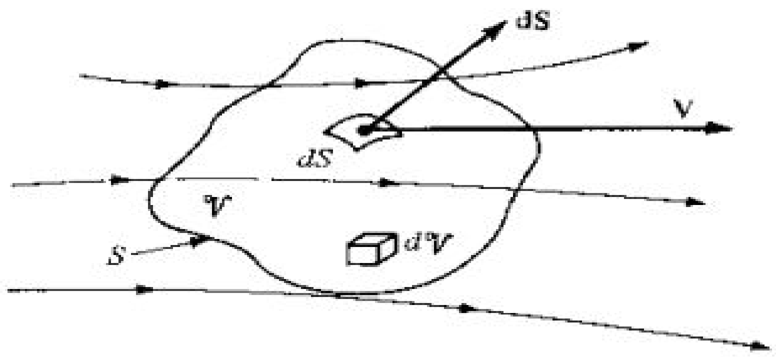


Figure 10: Finite control volume fixed in space

From the first law,

$$B1 + B2 = B3 \quad \text{-- -- -- --} \ (2.86)$$

Note that each term in Equation (2.86) involves the time rate of energy change; hence, Equation (2.86) is, strictly speaking, a power equation. However, because it is a statement of the fundamental principle of conservation of energy, the equation is conventionally termed the "energy equation." We continue this convention here.

First, consider the rate of heat transferred to or from the fluid. This can be visualized as volumetric heating of the fluid inside the control volume due to absorption of radiation originating outside the system or the local emission of radiation by the fluid itself, if the temperature inside the control volume is high enough.

In addition, there may be chemical combustion processes taking place inside the control volume, such as fuel-air combustion in a jet engine.

Let this volumetric rate of heat addition per unit mass be denoted by $\dot{q}$. Typical units for $\dot{q}$ are J/s · kg or ft · lb/s · slug. Examining Figure 10, the mass contained within an elemental volume is ρ dV; hence, the rate of heat addition to this mass is $\dot{q}(\rho\, dV)$. Summing over the complete control volume, we obtain

$$\text{Rate of volumetric heating} = \oiiint_{\mathcal{V}} \dot{q}\rho\, d\mathcal{V} \tag{2.87}$$

In addition, if the flow is viscous, heat can be transferred into the control volume by means of thermal conduction and mass diffusion across the control surface.

At this stage, a detailed development of these viscous heat-addition terms is not warranted; they are considered in detail. Rather, let us denote the rate of heat addition to the control volume due to viscous effects simply by $\dot{Q}_{\text{viscous}}$. Therefore, in Equation (2.86), the total rate of heat addition is given by Equation (2.87) plus $\dot{Q}_{\text{viscous}}$:

$$B_1 = \oiiint_{\mathcal{V}} \dot{q}\rho\, d\mathcal{V} + \dot{Q}_{\text{viscous}} \tag{2.88}$$

Before considering the rate of work done on the fluid inside the control volume, consider a simpler case of a solid object in motion, with a force F being exerted on the object, as sketched in Figure 10.

The position of the object is measured from a fixed origin by the radius vector r. In moving from position r_1 to r_2 over an interval of time dt, the object is displaced through dr. By definition, the work done on the object in time dt is F · dr. Hence, the time rate of doing work is simply F · dr/dt. However, dr/dt = V, the velocity of the moving object. Hence, we can state that

Rate of doing work on moving body = F · V

In words, the rate of work done on a moving body is equal to the product of its velocity and the component of force in the direction of the velocity.

This result leads to an expression for B2, as follows. Consider the elemental area d S of the control surface in Figure 10.

The pressure force on this elemental area is −p dS. From the above result, the rate of work done on the fluid passing through d S with velocity V is (−p dS)· V. Hence, summing over the complete control surface, we have

$$\begin{array}{r}\text{Rate of work done on fluid inside} \\ \mathcal{V} \text{ due to pressure force on } S\end{array} = -\oiint_S (p\,\mathbf{dS})\cdot\mathbf{V} \qquad (2.89)$$

In addition, consider an elemental volume dV inside the control volume, as shown in Figure 10. Recalling that f is the body force per unit mass, the rate of work done on the elemental volume due to the body force is (ρf dV)· V. Summing over the complete control volume, we obtain

$$\begin{array}{r}\text{Rate of work done on fluid} \\ \text{inside } \mathcal{V} \text{ due to body forces}\end{array} = \iiint_{\mathcal{V}} (\rho\mathbf{f}\,d\mathcal{V})\cdot\mathbf{V} \qquad (2.90)$$

If the flow is viscous, the shear stress on the control surface will also perform work on the fluid as it passes across the surface. Let us denote this contribution simply by $\dot{W}_{viscous}$. Then the total rate of work done on the fluid inside the control volume is the sum of Equations (2.89) and (2.90) and $\dot{W}_{viscous}$:

$$B_2 = -\oiint_S p\mathbf{V}\cdot\mathbf{dS} + \iiint_{\mathcal{V}} \rho(\mathbf{f}\cdot\mathbf{V})\,d\mathcal{V} + \dot{W}_{\text{viscous}} \qquad (2.91)$$

To visualize the energy inside the control volume, recall that in the first law of thermodynamics as stated in Equation (2.85), the internal energy e is due to the random motion of the atoms and molecules inside the system. Equation (2.85) is written for a stationary system.

However, the fluid inside the control volume in Figure 10 is not stationary; it is moving at the local velocity V with a consequent kinetic energy per unit mass of $V^2/2$.

Hence, the energy per unit mass of the moving fluid is the sum of both internal and kinetic energies $e+V^2/2$.

This sum is called the total energy per unit mass. We are now ready to obtain an expression for B3, the rate of change of total energy of the fluid as it flows through the control volume.

Keep in mind that mass flows into the control volume of Figure 10 bringing with it a certain total energy; at the same time mass flows out of the control volume taking with it a generally different amount of total energy.

The elemental mass flow across d S is ρV · dS, and therefore the elemental flow of total energy across d S is$(\rho V \cdot dS)(e+V^2/2)$. Summing over the complete control surface, we obtain

$$\begin{matrix}\text{Net rate of flow of total} \\ \text{energy across control surface}\end{matrix} = \oiint_S (\rho \mathbf{V} \cdot \mathbf{dS}) \left(e + \frac{V^2}{2}\right) \qquad (2.92)$$

In addition, if the flow is unsteady, there is a time rate of change of total energy inside the control volume due to the transient fluctuations of the flow-field variables. The total energy contained in the elemental volume dV is $\rho(e+ V^2/2)$ dV, and hence the total energy inside the complete control volume at any instant in time is

$$\oiiint_{\mathcal{V}} \rho \left(e + \frac{V^2}{2}\right) d\mathcal{V}$$

Therefore,

$$\begin{matrix}\text{Time rate of change of total energy} \\ \text{inside } \mathcal{V} \text{ due to transient variations} \\ \text{of flow-field variables}\end{matrix} = \frac{\partial}{\partial t} \oiiint_{\mathcal{V}} \rho \left(e + \frac{V^2}{2}\right) d\mathcal{V} \qquad (2.93)$$

In turn, B_3 is the sum of Equations (2.92) and (2.93):

$$B_3 = \frac{\partial}{\partial t} \oiiint_{\mathcal{V}} \rho \left(e + \frac{V^2}{2}\right) d\mathcal{V} + \oiint_S (\rho \mathbf{V} \cdot \mathbf{dS}) \left(e + \frac{V^2}{2}\right) \qquad (2.94)$$

Repeating the physical principle stated at the beginning of this section, the rate of heat added to the fluid plus the rate of work done on the fluid is equal to the rate of change of total energy of the fluid as it flows through the control volume (i.e., energy is conserved). In turn, these words can be directly translated into an equation by combining Equations (2.86), (2.88), (2.91), and (2.94)

$$
\boxed{\begin{aligned}&\oiiint_{\mathcal{V}} \dot{q}\rho\, d\mathcal{V} + \dot{Q}_{\text{viscous}} - \oiint_S p\mathbf{V}\cdot\mathbf{dS} + \oiiint_{\mathcal{V}} \rho(\mathbf{f}\cdot\mathbf{V})\, d\mathcal{V} + \dot{W}_{\text{viscous}} \\ &= \frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho\left(e+\frac{V^2}{2}\right) d\mathcal{V} + \oiint_S \rho\left(e+\frac{V^2}{2}\right)\mathbf{V}\cdot\mathbf{dS}\end{aligned}} \tag{2.95}
$$

Equation (2.95) is the energy equation in integral form; it is essentially the first law of thermodynamics applied to a fluid flow.

For the sake of completeness, note that if a shaft penetrates the control surface in Figure 10, driving some power machinery located inside the control volume (say, a compressor of a jet engine), then the rate of work delivered by the shaft, $\dot{W}$ shaft, must be added to the left side of Equation (2.95). Also note that the potential energy does not appear explicitly in Equation (2.95). Changes in potential energy are contained in the body force term when the force of gravity is included in f.

we can obtain a partial differential equation for total energy from the integral form given in Equation (2.95). Applying the divergence theorem to the surface integrals in Equation (2.95), collecting all terms inside the same volume integral, and setting the integrand equal to zero, we obtain

$$
\boxed{\begin{aligned}\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right] + \nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\mathbf{V}\right] &= \rho\dot{q} - \nabla\cdot(p\mathbf{V}) + \rho(\mathbf{f}\cdot\mathbf{V}) \\ &\quad + \dot{Q}'_{\text{viscous}} + \dot{W}'_{\text{viscous}}\end{aligned}} \tag{2.96}
$$

where $\dot{Q}'_{\text{viscous}}$ and $\dot{W}'_{\text{viscous}}$ represent the proper forms of the viscous terms. Equation (2.96) is a partial differential equation which relates the flow-field variables at a given point in space.

If the flow is steady ($\partial/\partial t = 0$), inviscid ($\dot{Q}_{\text{viscous}} = 0$ and $\dot{W}_{\text{viscous}} = 0$), adiabatic (no heat addition, $\dot{q} = 0$), without body forces ($f = 0$), then Equations (2.95) and (2.96) reduce to

$$
\boxed{\oiint_S \rho\left(e+\frac{V^2}{2}\right)\mathbf{V}\cdot\mathbf{dS} = -\oiint_S p\mathbf{V}\cdot\mathbf{dS}} \tag{2.97}
$$

and

$$
\boxed{\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\mathbf{V}\right] = -\nabla\cdot(p\mathbf{V})} \tag{2.98}
$$

With the energy equation, we have introduced another unknown flow-field variable e. We now have three equations, continuity, momentum, and energy, which involve four dependent variables, ρ, p, V, and e. A fourth equation can be obtained from a thermodynamic state relation for e (see Chapter 7). If the gas is calorically perfect, then

$$e = c_v T \quad \text{-- -- --} \ (2.99)$$

where c_v is the specific heat at constant volume. Equation (2.99) introduces temperature as yet another dependent variable. However, the system can be completed by using the perfect gas equation of state

$$p = \rho R T \quad \text{-- -- --} \ (2.100)$$

where R is the specific gas constant. Therefore, the continuity, momentum, and energy equations, along with Equations (2.99) and (2.100) are five independent equations for the five unknowns, ρ, p, V, e, and T . The matter of a perfect gas and related equations of state are reviewed; Equations (2.99) and (2.100) are presented here only to round out our development of the fundamental equations of fluid flow.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – II – DIMENSIONAL ANALYSIS AND FLUID FLOW IN CLOSED CONDUICTS – SAEA1304

I.INTRODUCTION

BUCKINGHAM'S Π-THEOREM

The Rayleigh's method of dimensional analysis becomes more laborious if the variables are more than the number of fundamental dimensions (M, *L,* T).

This difficulty is overcame by using Buckingham's n-theorem, which states, "If there are *n* variables (independent and dependent variables) in a physical phenomenon and if these variables contain *m* fundamental dimensions (M, *L,* T). then the variables are arranged into (n - *m)* dimensionless terms. Each term is called n-term".

Let X_1, X_2, X_3, •••, X_n are the variables involved in a physical problem. Let X_1 be the dependent variable and X_2, X_3,•••,X_n are the independent variables on which X_1 depends. Then X_1 is a function of X_2, X_3, •••, X_n and mathematically it is expressed as

$$X_1 = f(X_2, X_3, \cdots, X_n) \qquad ...(12.1)$$

Equation (12.1) can also be written as

$$f_1(X_1, X_2, X_3, \cdots, X_n) = 0. \qquad ...(12.2)$$

Equation (12.2) is a dimensionally homogeneous equation. It contains *n* variables. If there are *m* fundamental dimensions then according to Buckingham's π-theorem, equation (12.2) can be written in terms of number of dimensionless groups or π -terms in which number of π -terms is equal to (n - *m).* Hence equation (12.2) becomes as

$$f(\pi_1,, \pi_2, ..., \pi_{n-m}) = 0. \qquad ...(12.3)$$

Each of n-terms is dimensionless and is independent of the system. Division or multiplication by a constant does not change the character of the π -term. Each n-term contains $m + 1$ variables, where *m* is the number of fundamental dimensions and is also called repeating variables. Let in the above case X_2, X_3 and X_4 are repeating variables if the fundamental dimension *m* (M, *L, T)* = 3. Then each n-term is written as

$$\left.\begin{aligned} \pi_1 &= X_2^{a_1} . X_3^{b_1} . X_4^{c_1} . X_1 \\ \pi_2 &= X_2^{a_2} . X_3^{b_2} . X_4^{c_2} . X_5 \\ &\vdots \\ \pi_{n-m} &= X_2^{a_{n-m}} . X_3^{b_{n-m}} . X_4^{c_{n-m}} . X_n \end{aligned}\right\} \qquad ...(12.4)$$

Each equation is solved by the principle of dimensional homogeneity and values of a_1 , b_1 , c_1 etc., are obtained. These values are substituted in equation (12.4) and values of π_1,π_2, •••, π_{n-m} are obtained. These values of *π's* are substituted in equation (12.3). The final equation for the phenomenon isobtained by expressing any one of the π-terms as a function of others as

$$\pi_1 = \phi\,[\pi_2, \pi_3, \ldots, \pi_{n-m}]$$

or $$\pi_2 = \phi_1\,[\pi_1, \pi_3, \ldots, \pi_{n-m}] \quad \ldots(12.5)$$

METHOD OF SELECTING REPEATING VARIABLES

The number of repeating variables are equal to the number of fundamental dimensions of the problem. The choice of repeating variables is governed by the following considerations :

1. As far as possible, the dependent variable should not be selected as repeating variable.

2. The repeating variables should be chosen in such a way that one variable contains geometric property, other variable contains flow property and third variable contains fluid property.

Variables with Geometric Property are

(i) Length, l (ii) d (iii) Height, H etc.

Variables with flow property are

(i) Velocity, V (ii) Acceleration etc.

Variables with fluid property :

(i) μ, (ii) p, (iii) m etc.

3. The repeating variables selected should not form a dimensionless group.

4. The repeating variables together must have the same number of fundamental dimensions.

1. No two repeating variables should have the same dimensions.

In most of fluid mechanics problems, the choice of repeating variables may be (i) *d, v,* p (ii) *l, v,* p or (iii) *l, v,* μ or (iv) *d, v,* μ.

PROCEDURE FOR SOLVING PROBLEMS BY BUCKINGHAN'S Π-THEOREM

The procedure for solving problems by Buckingham's π -theorem is explained by considering the problem :

The resisting force R of a supersonic plane during flight can be considered as dependent upon the length of the aircraft l , velocity V, air viscosity μ, air density p and bulk modulus of air K. Express the functional relationship between these variables and the resisting force.

Solution. Step 1. The resisting force R depends upon (i) l , (ii) V, (iii) μ, (iv) p and (v) K. Hence R is a function of l , V, μ, p and K. Mathematically, R = f(l, V, μ, p, K)

or it can be written as

f_1(R, l , V, μ, p, K) = 0

Total number of variables, n = 6. Number of fundamental dimensions, m = 3...(i)

m is obtained by writing dimensions of each variables as R = MLT^{-2} , V= LT^{-1} , μ = $ML^{-1}T^{-1}$, p = ML^{-3} , K = $ML^{-1}T^{-2}$

Thus as fundamental dimensions in the problem are M, L, T and hence m = 3.] Number of dimensionless n-terms = n - m = 6 - 3 = 3.

Thus three n-terms say n_1 , n_2 and n_3 are formed. Hence equation (ii) is written as

f_1 (π_1, π_2, π_3) = 0

Stet 2. Each π-term = *m* + 1 variables, where *m* is equal to 3 and also called repeating variables.

Out of six variables *R, l, V,* μ, p and *K,* three variables are to be selected as repeating variable. *R* is a dependent variable and should not be selected as a repeating variable.

Out of the five remaining variables, one variable should have geometric property, the second variable should have flow property and third one fluid property.

These requirements are fulfilled by selecting *l, V* and p as repeating variables.

The repeating variables themselves should not form a dimensionless term and should have themselves fundamental dimensions equal to *m, i.e.,* 3 here.

Dimensions of *l, V* and p are *L*, LT^{-1} , ML^{-3} and hence the three fundamental dimensions exist in *l, V* and p and they themselves do not form dimensionless group.

Step 4. Each π-term is solved by the principle of dimensional homogeneity. For the first π-term, we have

$$\pi_1 = M^0L^0T^0 = L^{a_1} . (LT^{-1})^{b_1} . (ML^{-3})^{c_1} . MLT^{-2}.$$

Equating the powers of *M*, *L*, *T* on both sides, we get

Power of *M*, $\quad 0 = c_1 + 1 \qquad \therefore \quad c_1 = -1$

Power of *L*, $\quad 0 = a_1 + b_1 - 3c_1 + 1,$

$\therefore \quad a_1 = -b_1 + 3c_1 - 1 = 2 - 3 - 1 = -2$

Power of *T*, $\quad 0 = -b_1 - 2 \qquad \therefore \quad b_1 = -2$

Substituting the values of a_1, b_1 and c_1 in equation (*iv*),

$$\pi_1 = l^{-2} . V^{-2} . \rho^{-1} . R$$

or

$$\pi_1 = \frac{R}{l^2V^2\rho} = \frac{R}{\rho l^2 V^2} \qquad ...(v)$$

Similarly for the 2nd π-term, we get $\pi_2 = M^0L^0T^0 = L^{a_2} . (LT^{-1})^{b_2} . (ML^{-3})^{c_2} . ML^{-1}T^{-1}$.

Equating the powers of M, L, T on both sides

Power of M, $\quad 0 = c_2 + 1, \quad \therefore \quad c_2 = -1$

Power of L, $\quad 0 = a_2 + b_2 - 3c_2 - 1,$

$$a_2 = -b_2 + 3c_2 + 1 = 1 - 3 + 1 = -1$$

Power of T, $\quad 0 = -b_2 - 1, \quad \therefore \quad b_2 = -1$

Substituting the values of a_2, b_2 and c_2 in π_2 of (*iv*)

$$\pi_2 = l^{-1} . V^{-1} . \rho^{-1} . \mu = \frac{\mu}{lV\rho}.$$

Similarly for the 2nd π-term, we get $\pi_2 = M^0L^0T^0 = L^{a_2} . (LT^{-1})^{b_2} . (ML^{-3})^{c_2} . ML^{-1}T^{-1}$.

Equating the powers of M, L, T on both sides

Power of M, $\quad 0 = c_2 + 1, \quad \therefore \quad c_2 = -1$

Power of L, $\quad 0 = a_2 + b_2 - 3c_2 - 1,$

$$a_2 = -b_2 + 3c_2 + 1 = 1 - 3 + 1 = -1$$

Power of T, $\quad 0 = -b_2 - 1, \quad \therefore \quad b_2 = -1$

Substituting the values of a_2, b_2 and c_2 in π_2 of (*iv*)

$$\pi_2 = l^{-1} . V^{-1} . \rho^{-1} . \mu = \frac{\mu}{lV\rho}.$$

Step 5. Substituting the values of π_1, π_2 and π_3 in equation (*iii*), we get

$$f_1\left(\frac{R}{\rho l^2V^2}, \frac{\mu}{lV\rho}, \frac{K}{V^2\rho}\right) = 0 \quad \text{or} \quad \frac{R}{\rho l^2V^2} = \phi\left[\frac{\mu}{lV\rho}, \frac{K}{V^2\rho}\right]$$

or
$$R = \rho l^2V^2\phi\left[\frac{\mu}{lV\rho}, \frac{K}{V^2\rho}\right]. \textbf{ Ans.}$$

MODEL ANALYSIS

For predicting the performance of the hydraulic structures (such as dams, spillways etc.) or hydraulic machines (such as turbines, pumps etc.), before actually constructing or manufacturing, models of the structures or machines are made and tests are performed on them to obtain the desired information.

The model is the small scale replica of the actual structure or machine. The actual structure or machine is called Prototype. It is not necessary that the models should be smaller than the prototypes (though in most of cases it is), they may be larger than the prototype. The study of models of actual machines is called Model analysis.

Model analysis is actually an experimental method of finding solutions of complex flow problems. Exact analytical solutions are possible only for a limited number of flow problems. The followings are the advantages of the dimensional and model analysis :

1. The performance of the hydraulic structure or hydraulic machine can be easily predicted, in advance, from its model.
2. With the help of dimensional analysis, a relationship between the variables influencing a flow problem in terms of dimensionless parameters is obtained. This relationship helps in conducting tests on the model.
3. The merits of alternative designs can be predicted with the help of model testing. The most economical and safe design may be, finally, adopted.
4. The tests performed on the models can be utilized for obtaining, in advance, useful information about the performance of the prototypes only if a complete similarity exists between the model and the prototype.

SIMILITUDE-TYPES OF SIMILARITIES

Similitude is defined as the similarity between the model and its prototype in every respect, which means that the model and prototype have similar properties or model and prototype are completely similar. Three types of similarities must exist between the model and prototype. They are

1. Geometric Similarity,
2. Kinematic Similarity, and
3. Dynamic Similarity.

Geometric Similarity. The geometric similarity is said to exist between the model and the prototype. The ratio of all corresponding linear dimension in the model and prototype are equal.

Let L_m = Length of model,

b_m = Breadth of model,

D_m = Diameter of model,

A_m = Area of model,

V_m = Volume of model, and

L_p, b_p, D_p, A_p, V_p = Corresponding values of the prototype.

For geometric similarity between model and prototype, we must have the relation,

$$\frac{L_P}{L_m} = \frac{b_P}{b_m} = \frac{D_P}{D_m} = L_r \qquad ...(12.6)$$

where L_r is called the scale ratio.

For area's ratio and volume's ratio the relation should be as given below :

$$\frac{A_P}{A_m} = \frac{L_P \times b_P}{L_m \times b_m} = L_r \times L_r = L_r^2 \qquad ...(12.7)$$

$$\frac{\forall_P}{\forall_m} = \left(\frac{L_P}{L_m}\right)^3 = \left(\frac{b_P}{b_m}\right)^3 = \left(\frac{D_P}{D_m}\right)^3 \qquad ...(12.8)$$

KINEMATIC SIMILARITY

Kinematic similarity means the similarity of motion between model and prototype.

Thus kinematic similarity is said to exist between the model and the prototype if the ratios of the velocity and acceleration at the corresponding points in the model and at the corresponding points in the prototype are the same.

Since velocity and acceleration are vector quantities, hence not only the ratio of magnitude of velocity and acceleration at the corresponding points in model and prototype should be same ; but the directions of velocity and accelerations at the corresponding points in the model and prototype also should be parallel.

Let

V_{P1} = Velocity of fluid at point 1 in prototype,

V_{P2} = Velocity of fluid at point 2 in prototype,

a_{p1} = Acceleration of fluid at point 1 in prototype,

a_{p2} = Acceleration of fluid at point 2 in prototype, and

V_{m1}, , V_{m2}, , a_{m1}, , a_{m2}, = Corresponding values at the corresponding points of fluid velocity and acceleration in the model.

For kinematic similarity, we must have

$$\frac{V_{P_1}}{V_{m_1}} = \frac{V_{P_2}}{V_{m_2}} = V_r \quad ...(12.9)$$

where V_r is the velocity ratio.

For acceleration, we must have $\frac{a_{P_1}}{a_{m_1}} = \frac{a_{P_2}}{a_{m_2}} = a_r$ ...(12.10)

where a_r is the acceleration ratio.

Also the directions of the velocities in the model and prototype should be same.

DYNAMIC SIMILARITY

Dynamic similarity means the similarity of forces between the model and prototype. Thus dynamic similarity is said to exist between the model and the prototype if the ratios of the corresponding forces acting at the corresponding points are equal. Also the directions of the corresponding forces at the corresponding points should be same.

Let

$(F_i)_p$ = Inertia force at a point in prototype,

$(F_v)_P$ = Viscous force at the point in prototype,

$(F_g)_p$ = Gravity force at the point in prototype, and

$(F_i)_m$, $(F_v)_w$, $(F_g)_m$ = Corresponding values of forces at the corresponding point in model.

Then for dynamic similarity, we have, Also the directions of the corresponding forces at the corresponding points in the model and proto type should be same

TYPES OF FORCES ACTING IN MOVING FLUID

For the fluid flow problems, the forces acting on a fluid mass may be any one, or a combination of the several of the following forces :

1. Inertia force, F_i
2. Viscous force, F_v
3. Gravity force, F_g
4. Pressure force, F_P
5. Surface tension force, F_s

6. Elastic force, F_e

1. Inertia Force (F_i).

It is equal to the product of mass and acceleration of the flowing fluid and acts in the direction opposite to the direction of acceleration. It is always existing in the fluid flow problems.

2. Viscous Force (F_V).

It is equal to the product of shear stress ('t) due to viscosity and surface area of the flow. It is present in fluid flow problems where viscosity is having an important role to play.

3. Gravity Force (F_g).

It is equal to the product of mass and acceleration due to gravity of the flowing fluid. It is present in case of open surface flow.

4. Pressure Force (F_P).

It is equal to the product of pressure intensity and cross-sectional area of the flowing fluid. It is present in case of pipe-flow.

5. Surface Tension Force (F_s).

It is equal to the product of surface tension and length of surface of the flowing fluid.

6. Elastic Force (F_e).

It is equal to the product of elastic stress and area of the flowing fluid.

For a flowing fluid, the above-mentioned forces may not always be present. And also the forces, which are present in a fluid flow problem, are not of equal magnitude. There are always one or two forces which dominate the other forces. These dominating forces govern the flow of fluid.

DIMENSIONLESS NUMBERS

Dimensionless numbers are those numbers which are obtained by dividing the inertia force by viscous force or gravity force or pressure force or surface tension force or elastic force. As this is a ratio of one force to the other force, it will be a dimensionless number. These dimensionless numbers are also called non-dimensional parameters. The followings are the important dimensionless numbers :

1. Reynold's number, 2. Froude's number,

3. Euler's number, 4. Weber's number,

5. Mach's number.

Reynold's Number (R_e)· It is defined as the ratio of inertia force of a flowing fluid and the viscous force of the fluid. The expression for Reynold's number is obtained as

Inertia force (F_i) $= \text{Mass} \times \text{Acceleration of flowing fluid}$

$$= \rho \times \text{Volume} \times \frac{\text{Velocity}}{\text{Time}} = \rho \times \frac{\text{Volume}}{\text{Time}} \times \text{Velocity}$$

$$= \rho \times AV \times V \quad \{\because \quad \text{Volume per sec} = \text{Area} \times \text{Velocity} = A \times V\}$$

$$= \rho AV^2 \qquad \text{...(12.11)}$$

Viscous force (F_v) $= \text{Shear stress} \times \text{Area}$ $\left\{\because \quad \tau = \mu \frac{du}{dy} \quad \therefore \quad \text{Force} = \tau \times \text{Area}\right\}$

$$= \tau \times A$$

$$= \left(\mu \frac{du}{dy}\right) \times A = \mu \cdot \frac{V}{L} \times A \qquad \left\{\because \quad \frac{du}{dy} = \frac{V}{L}\right\}$$

By definition, Reynold's number,

$$R_e = \frac{F_i}{F_v} = \frac{\rho AV^2}{\mu \cdot \frac{V}{L} \times A} = \frac{\rho VL}{\mu}$$

$$= \frac{V \times L}{(\mu / \rho)} = \frac{V \times L}{\nu} \qquad \left\{\because \quad \frac{\mu}{\rho} = \nu = \text{Kinematic viscosity}\right\}$$

In case of pipe flow, the linear dimension L is taken as diameter, d. Hence Reynold's number for pipe flow,

$$R_e = \frac{V \times d}{\nu} \quad \text{or} \quad \frac{\rho Vd}{\mu}. \qquad \text{...(12.12)}$$

Froude's Number (F_e)· The Froude's number is defined as the square root of the ratio of inertia force of a flowing fluid to the gravity force. Mathematically, it is expressed as

$$F_e = \sqrt{\frac{F_i}{F_g}}$$

where F_i from equation (12.11) $= \rho AV^2$

and F_g = Force due to gravity

$$= \text{Mass} \times \text{Acceleration due to gravity}$$

$$= \rho \times \text{Volume} \times g = \rho \times L^3 \times g \qquad \{\because \quad \text{Volume} = L^3\}$$

$$= \rho \times L^2 \times L \times g = \rho \times A \times L \times g \qquad \{\because \quad L^2 = A = \text{Area}\}$$

$$\therefore \qquad F_e = \sqrt{\frac{F_i}{F_g}} = \sqrt{\frac{\rho AV^2}{\rho ALg}} = \sqrt{\frac{V^2}{Lg}} = \frac{V}{\sqrt{Lg}} \qquad \text{...(12.13)}$$

Euler's N umber (E_u)· It is defined as the square root of the ratio of the inertia force of a flowing fluid to the pressure force. Mathematically, it is expressed as

$$E_u = \sqrt{\frac{F_i}{F_p}}$$

where F_p = Intensity of pressure × Area = $p \times A$

and $F_i = \rho A V^2$

$$\therefore \qquad E_u = \sqrt{\frac{\rho A V^2}{p \times A}} = \sqrt{\frac{V^2}{p/\rho}} = \frac{V}{\sqrt{p/\rho}} \qquad \text{...(12.14)}$$

Weber's Number (W_e)· It is defined as the square root of the ratio of the inertia force of a flowing fluid to the surface tension force. Mathematically, it is expressed as

$$\text{Weber's Number,} \qquad W_e = \sqrt{\frac{F_i}{F_s}}$$

where F_i = Inertia force = $\rho A V^2$

and F_s = Surface tension force

= Surface tension per unit length × Length = $\sigma \times L$

$$\therefore \qquad W_e = \sqrt{\frac{\rho A V^2}{\sigma \times L}} = \sqrt{\frac{\rho \times L^2 \times V^2}{\sigma \times L}} \qquad \{\because \quad A = L^2\}$$

$$= \sqrt{\frac{\rho L \times V^2}{\sigma}} = \sqrt{\frac{V^2}{\sigma/\rho L}} = \frac{V}{\sqrt{\sigma/\rho L}}. \qquad \text{...(12.15)}$$

Mach's N umber (M). Mach's number is defined as the square root of the ratio of the inertia force of a flowing fluid to the elastic force. Mathematically, it is defined as

$$M = \sqrt{\frac{\text{Inertia force}}{\text{Elastic force}}} = \sqrt{\frac{F_i}{F_e}}$$

where $F_i = \rho A V^2$

and F_e = Elastic force = Elastic stress × Area

$= K \times A = K \times L^2$ $\qquad \{\because \quad K = \text{Elastic stress}\}$

$$\therefore \qquad M = \sqrt{\frac{\rho A V^2}{K \times L^2}} = \sqrt{\frac{\rho \times L^2 \times V^2}{K \times L^2}} = \sqrt{\frac{V^2}{K/\rho}} = \frac{V}{\sqrt{K/\rho}}$$

But $\qquad \sqrt{\frac{K}{\rho}} = C$ = Velocity of sound in the fluid

$$\therefore \qquad M = \frac{V}{C}. \qquad \text{...(12.16)}$$

MODEL LAWS OR SIMILARITY LAWS

For the dynamic similarity between the model and the prototype, the ratio of the corresponding forces acting at the corresponding points in the model and prototype should be equal.

The ratio of the forces are dimensionless numbers. It means for dynamic similarity between the model and prototype, the dimensionless numbers should be same for model and the prototype.

But it is quite difficult to satisfy the condition that all the dimensionless numbers *(i.e.,* R_e, F_e, W_e, E_u and M) are the same for the model and prototype.

Hence models are designed on the basis of ratio of the force, which is dominating in the phenomenon. The laws on which the models are designed for dynamic similarity are called model laws or laws of similarity.

The followings are the model laws :

1. Reynold's model law, 2. Froude model law,

3. Euler model law, 4. Weber model law,

5. Mach model law.

Reynold's Model Law.

Reynold's model law is the law in which models are based on Reynold's number. Models based on Reynold's number includes :

i. Pipe flow

ii. Resistance experienced by sub-marines, airplanes, fully immersed bodies etc.

As defined earlier that Reynold number is the ratio of inertia force and viscous force, and hence fluid flow problems where viscous forces alone are predominant, the models are designed for dynamic similarity on Reynolds law, which states that the Reynold number for the model must be equal to the Reynold number for the prototype.

Let

V_m = Velocity of fluid in model,

P_m = Density of fluid in model,

L_m = Length or linear dimension of the model,

μ_m = Viscosity or fluid in model, and

V_p , P_P , L_p and μ_p are the corresponding values of velocity, density, linear dimension and viscosity of fluid in prototype. Then according to Reynold's model law,

$$[R_e]_m = [R_e]_P \text{ or } \frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_P V_P L_P}{\mu_P} \quad ...(12.17)$$

or $$\frac{\rho_P \cdot V_P \cdot L_P}{\rho_m \cdot V_m \cdot L_m} \times \frac{1}{\frac{\mu_P}{\mu_m}} = 1 \quad \text{or} \quad \frac{\rho_r \cdot V_r \cdot L_r}{\mu_r} = 1$$

$$\left\{\text{where} \quad \rho_r = \frac{\rho_P}{\rho_m}, V_r = \frac{V_P}{V_m} \text{ and } L_r = \frac{L_P}{L_m}, \frac{\mu_P}{\mu_m} = \mu_r\right\}$$

And also ρ_r, V_r, L_r and μ_r are called the scale ratios for density, velocity, linear dimension and viscosity.

The scale ratios for time, acceleration, force and discharge for Reynold's model law are obtained as

$$t_r = \text{Time scale ratio} = \frac{L_r}{V_r} \qquad \left\{\because \quad V = \frac{L}{t} \quad \therefore \quad t = \frac{L}{V}\right\}$$

$$a_r = \text{Acceleration scale ratio} = \frac{V_r}{t_r}$$

$$F_r = \text{Force scale ratio} = (\text{Mass} \times \text{Acceleration})_r$$

$$= m_r \times a_r = \rho_r A_r V_r \times a_r \qquad \{A_r = \text{Area ratio}\}$$

$$= \rho_r L_r^2 V_r \times a_r$$

$$Q_r = \text{Discharge scale ratio} = (\rho A V)_r$$

$$= \rho_r A_r V_r = \rho_r \cdot L_r^2 \cdot V_r.$$

Froude Model Law.

Froude model law is the law in which the models are based on Froude number which means for dynamic similarity between the model and prototype, the Froude number for both of them should be equal. Froude model law is applicable when the gravity force is only predominant force which controls the flow in addition to the force of inertia. Froude model law is applied in the following fluid flow problems :

1. Free surface flows such as flow over spillways, weirs, sluices, channels etc.,
2. Flow of jet from an orifice or nozzle,
3. Where waves are likely to be formed on surface,
4. Where fluids of different densities flow over one another.

Let

V_m = Velocity of fluid in model,

L_m = Linear dimension or length of model,

g_m = Acceleration due to gravity at a place where model is tested. and

V_p, L_p and g_p are the corresponding values of the velocity, length and acceleration due to gravity for the prototype. Then according to Froude model law,

$$(F_e)_{model} = (F_e)_{prototype} \text{ or } \frac{V_m}{\sqrt{g_m L_m}} = \frac{V_P}{\sqrt{g_P L_P}} \qquad ...(12.18)$$

If the tests on the model are performed on the same place where prototype is to operate, then $g_m = g_P$ and equation (12.18) becomes as

$$\frac{V_m}{\sqrt{L_m}} = \frac{V_P}{\sqrt{L_P}} \qquad ...(12.19)$$

or
$$\frac{V_m}{V_P} \times \frac{1}{\sqrt{\frac{L_m}{L_P}}} = 1$$

$$\frac{V_P}{V_m} = \sqrt{\frac{L_P}{L_m}} = \sqrt{L_r} \qquad \left\{\because \frac{L_P}{L_m} = L_r\right\}$$

where L_r = Scale ratio for length

$\frac{V_P}{V_m} = V_r$ = Scale ratio for velocity.

$$\therefore \qquad \frac{V_P}{V_m} = V_r = \sqrt{L_r}. \qquad ...(12.20)$$

Scale ratios for various physical quantities based on Froude model law are :

(*a*) **Scale ratio for time**

As
$$\text{time} = \frac{\text{Length}}{\text{Velocity}},$$

then ratio of time for prototype and model is

$$T_r = \frac{T_P}{T_m} = \frac{\left(\frac{L}{V}\right)_P}{\left(\frac{L}{V}\right)_m} = \frac{\frac{L_P}{V_P}}{\frac{L_m}{V_m}} = \frac{L_P}{L_m} \times \frac{V_m}{V_P} = L_r \times \frac{1}{\sqrt{L_r}} \qquad \left\{\because \frac{V_P}{V_m} = \sqrt{L_r}\right\}$$

$$= \sqrt{L_r}. \qquad ...(12.21)$$

(*b*) **Scale ratio for acceleration**

$$\text{Acceleration} = \frac{V}{T}$$

$$\therefore \qquad a_r = \frac{a_P}{a_m} = \frac{\left(\frac{V}{T}\right)_P}{\left(\frac{V}{T}\right)_m} = \frac{V_P}{T_P} \times \frac{T_m}{V_m} = \frac{V_P}{V_m} \times \frac{T_m}{T_P}$$

$$= \sqrt{L_r} \times \frac{1}{\sqrt{L_r}} \qquad \left\{\because \frac{V_P}{V_m} = \sqrt{L_r}, \frac{T_P}{T_m} = \sqrt{L_r}\right\}$$

$$= 1. \qquad \text{...(12.22)}$$

(*c*) **Scale ratio for discharge**

$$Q = A \times V = L^2 \times \frac{L}{T} = \frac{L^3}{T}$$

$$\therefore \qquad Q_r = \frac{Q_P}{Q_m} = \frac{\left(\frac{L^3}{T}\right)_P}{\left(\frac{L^3}{T}\right)_m} = \left(\frac{L_P}{L_m}\right)^3 \times \left(\frac{T_m}{T_P}\right) = L_r^{\ 3} \times \frac{1}{\sqrt{L_r}} = L_r^{\ 2.5} \qquad \text{...(12.23)}$$

(*d*) **Scale ratio for force**

As $$\text{Force} = \text{Mass} \times \text{Acceleration} = \rho L^3 \times \frac{V}{T} = \rho L^2 \cdot \frac{L}{T} \cdot V = \rho L^2 V^2$$

$\therefore$ Ratio for force, $$F_r = \frac{F_P}{F_m} = \frac{\rho_P L_P^2 V_P^2}{\rho_m L_m^2 V_m^2} = \frac{\rho_P}{\rho_m} \times \left(\frac{L_P}{L_m}\right)^2 \times \left(\frac{V_P}{V_m}\right)^2.$$

If the fluid used in model and prototype is same, then

$$\frac{\rho_P}{\rho_m} = 1 \quad \text{or} \quad \rho_P = \rho_m$$

and hence $$F_r = \left(\frac{L_P}{L_m}\right)^2 \times \left(\frac{V_P}{V_m}\right)^2 = L_r^2 \times \left(\sqrt{L_r}\right)^2 = L_r^2 \cdot L_r = L_r^3. \qquad ...(12.24)$$

(*e*) **Scale ratio for pressure intensity**

As $$p = \frac{\text{Force}}{\text{Area}} = \frac{\rho L^2 V^2}{L^2} = \rho V^2$$

$\therefore$ Pressure ratio, $$p_r = \frac{p_P}{p_m} = \frac{\rho_P V_P^2}{\rho_m V_m^2}$$

If fluid is same, then $\rho_P = \rho_m$

$$\therefore \qquad p_r = \frac{V_P^2}{V_m^2} = \left(\frac{V_P}{V_m}\right)^2 = L_r. \qquad ...(12.25)$$

(*f*) **Scale ratio for work, energy, torque, moment etc.**

$$\text{Torque} = \text{Force} \times \text{Distance} = F \times L$$

$\therefore$ Torque ratio, $$T_r^* = \frac{T_P^*}{T_m^*} = \frac{(F \times L)_P}{(F \times L)_m} = F_r \times L_r = L_r^3 \times L_r = L_r^4. \qquad ...(12.26)$$

(*g*) **Scale ratio for power**

As $$\text{Power} = \text{Work per unit time}$$

$$= \frac{F \times L}{T}$$

$\therefore$ Power ratio, $$P_r = \frac{P_P}{P_m} = \frac{\dfrac{F_P \times L_P}{T_P}}{\dfrac{F_m \times L_m}{T_m}} = \frac{F_P}{F_m} \times \frac{L_P}{L_m} \times \frac{1}{\dfrac{T_P}{T_m}}$$

$$= F_r \cdot L_r \cdot \frac{1}{T_r} = L_r^3 \cdot L_r \cdot \frac{1}{\sqrt{L_r}} = L^{3.5}. \qquad ...(12.27)$$

EULER'S MODEL LAW.

Euler's model law is the law in which the models are designed on Euler's number which means for dynamic similarity between the model and prototype, the Euler number for model and prototype should be equal.

Euler's model law is applicable when the pressure forces are alone predominant in addition to the inertia force. According to this law :

$$(E_u)_{\text{model}} = (E_u)_{\text{prototype}} \qquad \ldots(12.28)$$

V_m = Velocity of fluid in model,

P_m = Pressure of fluid in model,

ρ_m = Density of fluid in model,

V_p, P_P , ρ_P = Corresponding values in prototype, then

Substituting these values in equation (12.28), we get

$$\frac{V_m}{\sqrt{p_m / \rho_m}} = \frac{V_P}{\sqrt{p_P / \rho_P}} \qquad \ldots(12.29)$$

If fluid is same in model and prototype, then equation (12.29) becomes as

$$\frac{V_m}{\sqrt{p_m}} = \frac{V_P}{\sqrt{p_P}} \qquad \ldots(12.30)$$

Euler's model law is applied for fluid flow problems where flow is taking place in a closed pipe in which case turbulence is fully developed so that viscous forces are negligible and gravity force and surface tension force is absent.

This law is also used where the phenomenon of cavitation takes place.

WEBER MODEL LAW

Weber model law is the law in which models are based on Weber's n umber, which is the ratio of the square root of inertia force to surface tension force.

Hence where surface tension effects predominate in addition to inertia force, the dynamic similarity between the model and prototype is obtained by equating the Weber number of the model and its prototype.

Hence according to this law :

$$(W_e)_{\text{model}} = (W_e)_{\text{prototype}}, \qquad \text{where } W_e \text{ is Weber number and } = \frac{V}{\sqrt{\sigma / \rho L}}$$

V_m = Velocity of fluid in model,

J_m = Surface tensile force in model,

ρ_m = Density of fluid in model,

L_m = Length of surface in model,and

V_p, J_p, ρ_P, L_p = Corresponding values of fluid in prototype.

Then according to Weber law, we have

$$\frac{V_m}{\sqrt{\sigma_m / \rho_m L_m}} = \frac{V}{\sqrt{\sigma_P / \rho_P L_P}} \qquad \text{...(12.31)}$$

Weber model law is applied in following cases :

1. Capillary rise in narrow passages,
2. Capillary movement of water in soil,
3. Capillary waves in channels,
4. Flow over weirs for small heads.

Mach Model law

Mach model law is the law in which models are designed on Mach number, which is the ratio of the square root of inertia force to elastic force of a fluid.

Hence where the forces due to elastic compression predominate in addition to inertia force, the dynamic similarity between the model and its prototype is obtained by equating the Mach number of the model and its prototype.

Hence according to this law :

$$(M)_{\text{model}} = (M)_{\text{prototype}}$$

$$\text{where } M = \text{Mach number} = \frac{V}{\sqrt{K / \rho}}$$

V_m = Velocity of fluid in model,

K_m = Elastic stress for model,

ρ_m = Density of fluid in model, and

V_p , K_p and ρ_p = Corresponding values for prototype. Then according to Mach law,

$$= \frac{V_m}{\sqrt{K_m/\rho_m}} = \frac{V}{\sqrt{K_P/\rho_P}} \qquad \text{...(12.32)}$$

Mach model law is applied in the following cases :

1. Flow of aeroplane and projectile through air at supersonic speed, *i.e.,* at a velocity more than the velocity of sound,
2. Aerodynamic testing,
3. Under water testing of torpedoes,
4. Water-hammer problems.

MODEL TESTING OF PARTIALLY SUB-MERGED BODIES

Let us consider the testing of a ship model (ship is a partially sub-merged body) in a water-tunnel in order to find the drag force *F* or resistance experienced by a ship. The drag experienced by a ship consists of :

1. The wave resistance, which is the resistance offered by the waves on the free sea-surface, and
2. The frictional or viscous resistance, which is offered by the water on the surface of contact of the ship with water.

Thus in this case three forces namely inertia, gravity and viscous forces are present. Then for dynamic similarity between the model and its prototype, the Reynold's number (which is ratio of inertia force to viscous force) and the Froude number (which is the ratio of inertia force to gravity force) should be taken into account. This means that in this case, the Reynold model law and Froude model law should be applied.

But for Reynold model law, the condition is

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_P V_P L_P}{\mu_P}$$

Reynold number of model= Reynold number of prototype

If fluid is same for the model and prototype, then $\rho_m = \rho_p$ and $\mu_m = \mu_p$

$$\therefore \quad V_m L_m = V_P L_P$$

$$V_m = \frac{V_P L_P}{L_m} = L_r V_P \qquad \left\{\because \frac{L_P}{L_m} = L_r\right\} \quad ...(12.33)$$

For Froude model law, have from equation (12.18) as $\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_P}{\sqrt{g_P L_P}}$

If fluid is same for model and prototype and test is conducted at the same place where prototype is to operate, then $g_m = g_p$

$$\therefore \quad \frac{V_m}{\sqrt{L_m}} = \frac{V_P}{\sqrt{L_P}}$$

$$\therefore \quad V_m = \sqrt{\frac{L_m}{L_P}} \times V_P = V_P \times \frac{1}{\sqrt{\frac{L_P}{L_m}}} = V_P \times \frac{1}{\sqrt{L_r}} \quad \left\{\because \frac{L_P}{L_m} = L_r\right\} \quad ...(12.34)$$

From equations (12.33) and (12.34), we observe that the velocity of fluid in model for Reynold model law and Froude model law is different.

Thus it is quite impossible to satisfy both the laws together, which means the dynamic similarity between the model and its prototype will not exist. To overcome this difficulty, the method suggested by William Froude is adopted for testing the ship model (or partially sub-merged bodies) as :

Step 1. The total resistance experienced by a ship is equal to the wave resistance plus frictional or viscous resistance.

Let

$(R)_p$ = Total resistance experienced by prototype,

$(R_w)_P$ = Wave resistance experienced by prototype,

$(R_1)_p$ = Frictional resistance experienced by prototype, and

$(R)_m, (R_w)_m, (R_f)_m$ = Corresponding values for model.

Then, we have for prototype, $(R)_P = (R_w)_P + (R_f)_P$ ...(12.35)

and for model, $(R)_m = (R_w)_m + (R_f)_m$ ...(12.36)

Step 2. The frictional resistances for the model and the ship [*i.e.*, $(R_f)_m$ and $(R_f)_P$] are calculated from the expressions given below :

$$(R_f)_P = f_P A_P V_P^{\ n} \qquad \text{...(12.37)}$$

and

$$(R_f)_m = f_m A_m V_m^{\ n} \qquad \text{...(12.38)}$$

where f_P = Frictional resistance per unit area per unit velocity of prototype,

A_P = Wetted surface area of the prototype,

V_P = Velocity of prototype,

n = Constant, and

f_m, A_m, V_m = Corresponding values of frictional resistance, wetted area and velocity of model.

The values of f_P and f_m are determined from experiments.

Step 3. The model is tested by towing it in water contained in a towing tank such that the dynamic similarity for Froude number is satisfied *i.e.*, $(F_e)_m = (F_e)_P$. The total resistance of the model (R_m) is measured for this condition.

Step 4. The total resistance (R_m) for the model is known from step 3 and frictional resistance of the model $(R_f)_m$ is calculated from equation (12.37). Then the wave resistance for the model is known from equation (12.36) as

$$(R_w)_m = R_m - (R_f)_m \qquad \text{...(12.39)}$$

Step 5. The resistance experienced by a ship of length L, flowing with velocity V in fluid of viscosity μ, density ρ depends upon g, the acceleration due to gravity. By dimensional analysis, the expression for resistance is given by

$$\frac{R}{\rho L^2 V^2} = \phi\left[\frac{\rho V L}{\mu}, \frac{V^2}{gL}\right] = \phi\,[R_e, F_e^{\ 2}]$$

Thus resistance is a function of Reynold number (R_e) and Froude number (F_e).

For dynamic similarity for model and prototype for wave resistance only, we have

$$\frac{(R_w)_P}{\rho_P L_P^2 V_P^2} = \frac{(R_w)_m}{\rho_m L_m^2 V_m^2}$$

or wave resistance for prototype is given as

$$(R_w)_P = \frac{\rho_P}{\rho_m} \times \frac{L_P^2}{L_m^2} \times \frac{V_P^2}{V_m^2} \times (R_w)_m \qquad \text{...(12.40)}$$

But from Step 3, $\quad (F_e)_m = (F_e)_P$ or $\dfrac{V_m}{\sqrt{L_m g_m}} = \dfrac{V_P}{\sqrt{L_P g_P}}$

If the model and ship are at the same place, $g_m = g_P$

$$\therefore \qquad \frac{V_m}{\sqrt{L_m}} = \frac{V_P}{\sqrt{L_P}} \quad \text{or} \quad V_m = \sqrt{\frac{L_m}{L_P}}\ .\ V_P$$

Substituting the value of V_m in equation (12.40), we have

$$(R_w)_P = \frac{\rho_P}{\rho_m} \times \frac{L_P^2}{L_m^2} \times \frac{V_P^2}{V_P^2 \times \dfrac{L_m}{L_P}} \times (R_w)_m$$

$$= \frac{\rho_P}{\rho_m} \times \frac{L_P^3}{L_m^3} \times (R_w)_m. \qquad \text{...(12.41)}$$

Step 6. The total resistance of the ship is given by adding $(R_w)_P$ from equation (12.41) to $(R_f)_P$ given by equation (12.37) as

$$R_P = \frac{\rho_P}{\rho_m} \times \left(\frac{L_P}{L_m}\right)^3 \times (R_w)_m + f_P A_P V_P^{\ 2} \qquad \text{...(12.42)}$$

CLASSIFICATION OF MODELS

The hydraulic models are classified as :

1. Undistorted models, and
2. Distorted models.

Undistorted Models.

Undistorted models are those models which are geometrically similar to their prototypes or in other words if the scale ratio for the linear dimensions of the model and its prototype is same, the model is called undistorted model.

The behavior of the prototype can be easily predicted from the results of undistorted model.

DISTORTED MODELS.

A model is said to be distorted if it is not geometrically similar to its prototype. For a distorted model different scale ratios for the linear dimensions are adopted.

For example, in case of rivers, harbours, reservoirs etc., two different scale ratios, one for horizontal dimensions and other for vertical dimensions are taken. Thus the models of rivers, harbours and reservoirs will become as distorted models.

If for the river, the horizontal and vertical scale ratios are taken to be same so that the model is undistorted, then the depth of water in the model of the river will be very-very small which may not be measured accurately. The following are the advantage of distorted models :

1. The vertical dimensions of the model can be measured accurately.
2. The cost of the model can be reduced.
3. Turbulent flow in the model can be maintained.

- Though there are some advantages of the distorted model, yet the results of the distorted model cannot be directly transferred to its prototype. But sometimes from the distorted models very useful information can be obtained.

12.11.3 Scale Ratios for Distorted Models. As mentioned above, two different scale ratios, one for horizontal dimensions and other for vertical dimensions, are taken for distorted models.

Let $(L_r)_H$ = Scale ratio for horizontal dimension

$$= \frac{L_P}{L_m} = \frac{B_P}{B_m} = \frac{\text{Linear horizontal dimension of prototype}}{\text{Linear horizontal dimension of model}}$$

$(L_r)_V$ = Scale ratio for vertical dimension

$$= \frac{\text{Linear vertical dimension of prototype}}{\text{Linear vertical dimension of model}} = \frac{h_P}{h_m}$$

Then the scale ratios of velocity, area of flow, discharge etc., in terms of $(L_r)_H$ and $(L_r)_V$ can be obtained for distorted models as given below :

1. Scale ratio for velocity

Let V_P = Velocity in prototype

V_m = Velocity in model.

Then
$$\frac{V_P}{V_m} = \frac{\sqrt{2gh_P}}{\sqrt{2gh_m}} = \sqrt{\frac{h_P}{h_m}} = \sqrt{(L_r)_V} \qquad \left(\because \frac{h_P}{h_m} = (L_r)_V\right)$$

2. Scale ratio for area of flow

Let A_P = Area of flow in prototype = $B_P \times h_P$

A_m = Area of flow in model = $B_m \times h_m$

$$\therefore \quad \frac{A_P}{A_m} = \frac{B_P \times h_P}{B_m \times h_m} = \frac{B_P}{B_m} \times \frac{h_P}{h_m} = (L_r)_H \times (L_r)_V$$

3. Scale ratio for discharge

Let Q_P = Discharge through prototype = $A_P \times V_P$

Q_m = Discharge through model = $A_m \times V_m$

$$\therefore \quad \frac{Q_P}{Q_m} = \frac{A_P \times V_P}{A_m \times V_m} = (L_r)_H \times (L_r)_V \times \sqrt{(L_r)_V} = (L_r)_H \times [(L_r)_V]^{3/2} \quad ...(12.43)$$

FLOW THROUGH THE PIPE

Laminar flow and turbulent flow

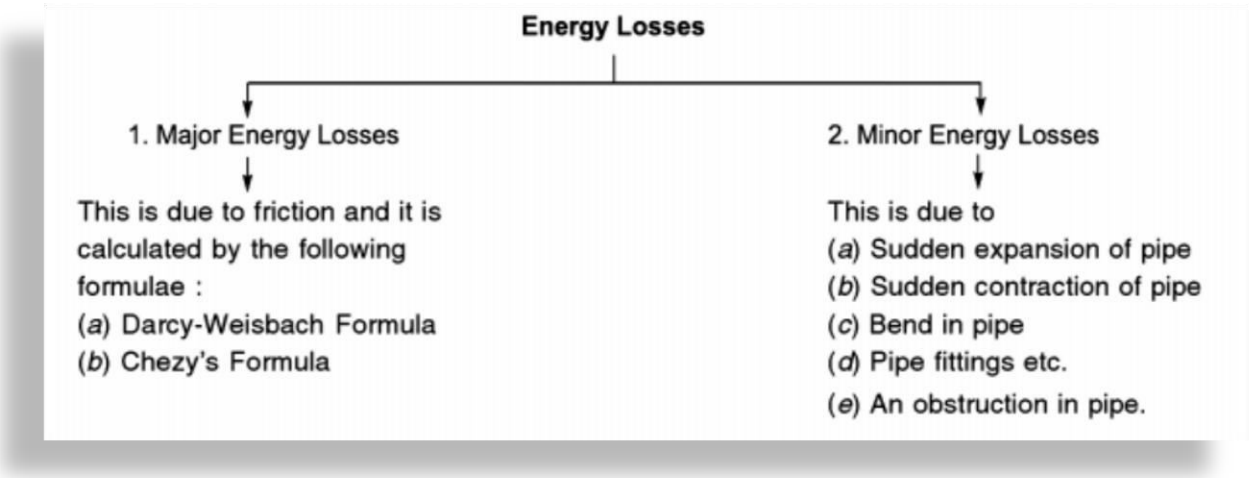
We have seen that when the Reynolds number is less than 2000 for pipe flow, the flow is known as laminar flow whereas when the Reynolds number is more than 4000, the flow is known as turbulent flow. In this chapter, the turbulent flow of fluids through pipes running full will be considered.

If the pipes are partially full as in the case of sewer lines, the pressure inside the pipe is same and equal to atmospheric pressure. Then the flow of fluid in the pipe is not under pressure.

This case will be taken in the section of flow of water through open channels. Here we will consider flow of fluids through pipes under pressure only**.**

LOSS OF ENERGY IN PIPE

When a fluid is flowing through a pipe, the fluid experiences some resistance due to which some of the energy of fluid is lost. This loss of energy is classified as :



LOSS OF ENERGY (OR HEAD) DUE TO FRICTION

(a) **Darcy-Weisbach Formula.**

The loss of head (or energy) in pipes due to friction is calculated from Darcy-Weisbach equation which is given by

$$h_f = \frac{4 \cdot f \cdot L \cdot V^2}{d \times 2g} \qquad \text{...(11.1)}$$

Where h_f= loss of head due to friction

f = co-efficient of friction which is a function of Reynolds number

L = length of pipe, V = mean velocity of flow, d = diameter of pipe

$$= \frac{16}{R_e} \text{ for } R_e < 2000 \text{ (viscous flow)}$$

$$= \frac{0.079}{R_e^{1/4}} \text{ for } R_e \text{ varying from 4000 to } 10^6$$

(b) **Chezy's Formula** for loss of head due to friction in pipes, which expression for loss of head due to friction in pipes is derived. Equation is

$$h_f = \frac{f'}{\rho g} \times \frac{P}{A} \times L \times V^2 \qquad \text{...(11.2)}$$

where h_f = loss of head due to friction, P = wetted perimeter of pipe,
A = area of cross-section of pipe, L = length of pipe,
and V = mean velocity of flow.

Now the ratio of $\frac{A}{P}\left(= \frac{\text{Area of flow}}{\text{Perimeter (wetted)}}\right)$ is called hydraulic mean depth or hydraulic radius and is denoted by m.

$\therefore$ Hydraulic mean depth, $m = \frac{A}{P} = \frac{\frac{\pi}{4}d^2}{\pi d} = \frac{d}{4}$

Substituting $\frac{A}{P} = m$ or $\frac{P}{A} = \frac{1}{m}$ in equation (11.2), we get

$$h_f = \frac{f'}{\rho g} \times L \times V^2 \times \frac{1}{m} \text{ or } V^2 = h_f \times \frac{\rho g}{f'} \times m \times \frac{1}{L} = \frac{\rho g}{f'} \times m \times \frac{h_f}{L}$$

$$\therefore \qquad V = \sqrt{\frac{\rho g}{f'} \times m \times \frac{h_f}{L}} = \sqrt{\frac{\rho g}{f'}}\sqrt{m\frac{h_f}{L}} \qquad \text{...(11.3)}$$

Let $\sqrt{\frac{\rho g}{f'}} = C$, where C is a constant known as Chezy's constant and $\frac{h_f}{L} = i$, where i is loss of head per unit length of pipe.

Substituting the values of $\sqrt{\frac{\rho g}{f'}}$ and $\sqrt{\frac{h_f}{L}}$ in equation (11.3), we get

$$V = C\sqrt{mi} \qquad \text{...(11.4)}$$

Equation (11.4) is known as Chezy's formula. Thus the loss of head due to friction in pipe from Chezy's formula can be obtained if the velocity of flow through pipe and also the value of C is known. The value of m for pipe is always equal to $d/4$.

MINOR ENERGY (HEAD) LOSSES

The loss of head or energy due to friction in a pipe is known as major loss while the loss of energy due to change of velocity of the following fluid in magnitude or direction is called minor loss of energy. The minor loss of energy (or head) includes the following cases :

1. Loss of head due to sudden enlargement,
2. Loss of head due to sudden contraction,
3. Loss of head at the entrance of a pipe,
4. Loss of head at the exit of a pipe,

5. Loss of head due to an obstruction in a pipe,
6. Loss of head due to bend in the pipe,
7. Loss of head in various pipe fittings.

In case of long pipe the above losses are small as compared with the loss of head due to friction and hence they are called minor losses and even may be neglected without serious error. But in case of a short pipe, these losses are comparable with the loss of head due to friction.

Loss of Head Due to Sudden Enlargement. Consider a liquid flowing through a pipe which has sudden enlargement as shown in Fig. 11. Consider two sections (1)-(1) and (2)-(2) before and after the enlargement.

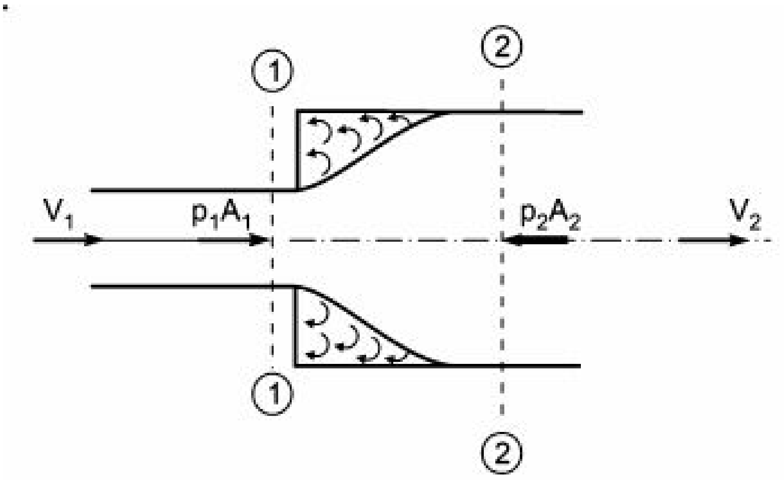


Figure 11. Sudden enlargement

Let

p_1 = pressure intensity at section 1-1, V1 = velocity of flow at section 1-1,

A_1= area of pipe at section 1-1

p_2, V_2 and A_2 = corresponding values at section 2-2.

p_2, V_2 and A_2 = corresponding values at section 2-2.

Due to sudden change of diameter of the pipe from D 1 to D2, the liquid flowing from the smaller

pipe is not able to follow the abrupt change of the boundary. Thus the flow separates from the boundary and turbulent eddies are formed as shown in Fig. 11. The loss of head (or energy) takes place due to the formation of these eddies.

Let p' = pressure intensity of the liquid eddies on the area (A 2 - A 1)

h_e = loss of head due to sudden enlargement Applying Bernoulli's equation at sections 1-1 and 2-2,

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2+\text{loss of head due to sudden enlargement}$$

But $z_1 = z_2$ as pipe is horizontal

$$\therefore \quad \frac{p_1}{\rho g}+\frac{V_1^2}{2g}=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+h_e$$

or $$h_e=\left(\frac{p_1}{\rho g}-\frac{p_2}{\rho g}\right)+\left(\frac{V_1^2}{2g}-\frac{V_2^2}{2g}\right) \qquad ...(i)$$

Consider the control volume of liquid between sections 1-1 and 2-2. Then the force acting on the liquid in the control volume in the direction of flow is given by

$$F_x = p_1A_1 + p'(A_2 - A_1) - p_2A_2$$

But experimentally it is found that $p' = p_1$

$$\therefore \quad F_x = p_1A_1 + p_1(A_2 - A_1) - p_2A_2 = p_1A_2 - p_2A_2$$
$$= (p_1 - p_2)A_2 \qquad ...(ii)$$

Momentum of liquid/sec at section 1-1 = mass × velocity

$$= \rho A_1 V_1 \times V_1 = \rho A_1 V_1^2$$

Momentum of liquid/sec at section 2-2 = $\rho A_2V_2 \times V_2 = \rho A_2V_2^2$

$\therefore$ Change of momentum/sec = $\rho A_2V_2^2 - \rho A_1V_1^2$

But from continuity equation, we have

$$A_1V_1 = A_2V_2 \text{ or } A_1 = \frac{A_2V_2}{V_1}$$

$\therefore$ Change of momentum/sec = $\rho A_2V_2^2 - \rho \times \frac{A_2V_2}{V_1} \times V_1^2 = \rho A_2V_2^2 - \rho A_2V_1V_2$

$$= \rho A_2[V_2^2 - V_1V_2] \qquad ...(iii)$$

Now net force acting on the control volume in the direction of flow must be equal to the rate of change of momentum or change of momentum per second. Hence equating (*ii*) and (*iii*)

$$(p_1 - p_2)A_2 = \rho A_2[V_2^2 - V_1V_2]$$

or $$\frac{p_1 - p_2}{\rho} = V_2^2 - V_1V_2$$

Dividing by g on both sides, we have $\frac{p_1 - p_2}{\rho g} = \frac{V_2^2 - V_1V_2}{g}$ or $\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = \frac{V_2^2 - V_1V_2}{g}$

Substituting the value of $\left(\frac{p_1}{\rho g} - \frac{p_2}{\rho g}\right)$ in equation (*i*), we get

$$h_e = \frac{V_2^2 - V_1V_2}{g} + \frac{V_1^2}{2g} - \frac{V_2^2}{2g} = \frac{2V_2^2 - 2V_1V_2 + V_1^2 - V_2^2}{2g}$$

$$= \frac{V_2^2 + V_1^2 - 2V_1V_2}{2g} = \left(\frac{V_1 - V_2}{2g}\right)^2$$

$$\therefore \qquad h_e = \frac{(V_1 - V_2)^2}{2g}. \qquad \text{...(11.5)}$$

Loss of Head due to Sudden Contraction.

Consider a liquid flowing in a pipe which has a sudden contraction in area as shown in Fig. 12. Consider two sections 1-1 and 2-2 before and after contraction.

As the liquid flows from large pipe to smaller pipe, the area of flow goes on decreasing and becomes minimum at a section C-C as shown in Fig. 12.

This section C-C is called Vena-contracta. After section C-C, a sudden enlargement of the area takes place. The loss of head due to sudden contraction is actually due to sudden enlargement from Vena-contracta to smaller pipe.

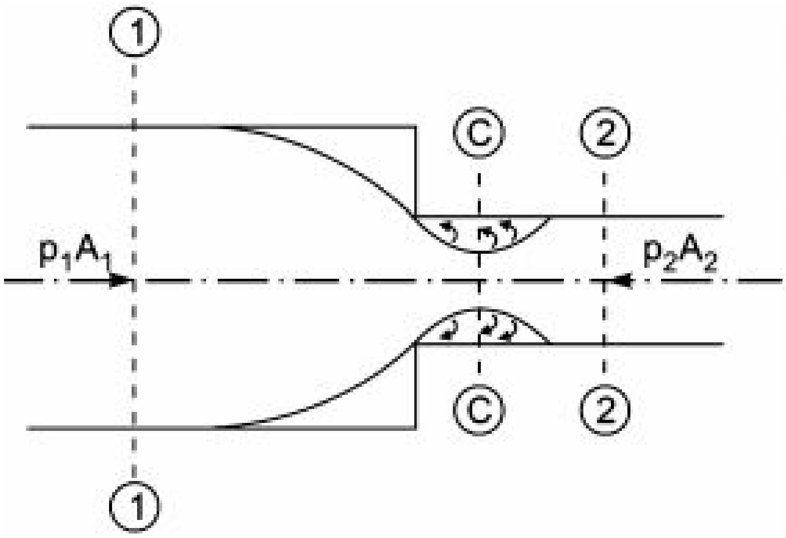


Figure 12. Sudden contraction

A_c = Area of flow at section C-C

V_e = Velocity of flow at section C-C

A_2 = Area of flow at section 2-2

V_2 = Velocity of flow at section 2-2

h_e = Loss of head due to sudden contraction.

Now h_e = actual loss of head due to enlargement from section C-C to section 2-2 and is given by equation (11.5) as

From continuity equation, we have

$$A_c V_c = A_2 V_2 \quad \text{or} \quad \frac{V_c}{V_2} = \frac{A_2}{A_c} = \frac{1}{(A_c / A_2)} = \frac{1}{C_c}$$

Substituting the value of $\frac{V_c}{V_2}$ in (*i*), we get

$$h_c = \frac{V_2^2}{2g}\left[\frac{1}{C_c} - 1\right]^2$$

$$= \frac{kV_2^2}{2g}, \text{ where } k = \left[\frac{1}{C_c} - 1\right]^2$$

If the value of C_c is assumed to be equal to 0.62, then

$$k = \left[\frac{1}{0.62} - 1\right]^2 = 0.375$$

Then h_c becomes as $\qquad h_c = \frac{kV_2^2}{2g} = 0.375\,\frac{V_2^2}{2g}$

If the value of C_c is not given then the head loss due to contraction is taken as

$$= 0.5\,\frac{V_2^2}{2g} \text{ or } h_c = 0.5\,\frac{V_2^2}{2g}. \qquad \text{...(11.7)}$$

Loss of Head at the Entrance of a Pipe

This is the loss of energy which occurs when a liquid enters a pipe which is connected to a large tank or reservoir.

This loss is similar to the loss of head due to sudden contraction. This loss depends on the form of entrance.

For a sharp edge entrance, this loss is slightly more than a rounded or bell mouthed entrance.

In practice the value of loss of head at the entrance (or inlet) of a pipe with sharp cornered entrance is taken = $0.5\,\frac{V^2}{2g}$. Where V = velocity of liquid in pipe.

This loss is denoted by h_i

$$\therefore \qquad h_i = 0.5\ \frac{V^2}{2g} \qquad ...(11.8)$$

Loss of Head at the Exit of Pipe

This is the loss of head (or energy) due to the velocity of liquid at outlet of the pipe which is dissipated either in the form of a free jet (if outlet of the pipe is free) or it is lost in the tank or reservoir (if the outlet of the pipe is connected to the tank or reservoir).

This loss is equal to $\frac{V^2}{2g}$ where V *is* the velocity of liquid at the outlet of pipe. This loss is 2g denoted h_0

$$\therefore \qquad h_o = \frac{V^2}{2g} \qquad ...(11.9)$$

where V = velocity at outlet of pipe.

Loss of Head Due to an Obstruction in a Pipe

Whenever there is an obstruction in a pipe, the loss of energy takes place due to reduction of the area of the cross-section of the pipe at the place where obstruction is present. There is a sudden enlargement of the area of flow beyond the obstruction due to which loss of head takes place as shown in Fig. 13 (a)

Consider a pipe of area of cross-section

A having an obstruction as

shown in Fig. 13.

Let

a = Maximum area of obstruction

A = Area of pipe

V = Velocity of liquid in pipe

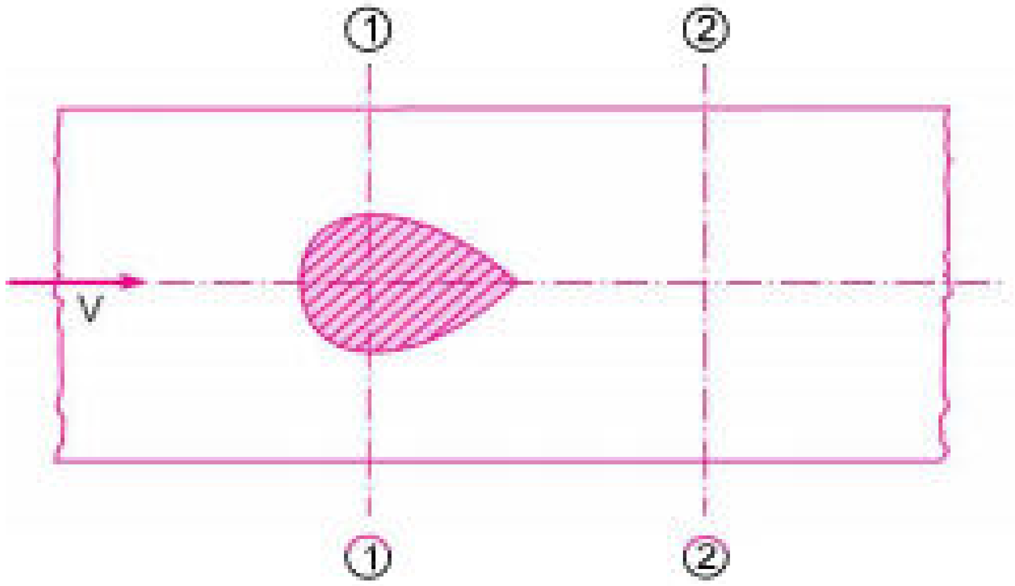


Figure.13 An obstruction in a pipe.

Then (A - *a)* = Area of flow of liquid at section 1-1 As the liquid flows and passes through section 1-1, a vena-contracta is formed beyond section 1-1, after which the stream of liquid widens again and velocity of flow at section 2-2 becomes uniform and equal to velocity, *V in* the pipe.

This situation is similar to the flow of liquid through sudden enlargement

Let

V_e = Velocity of liquid at vena-contracta.

Then loss of head due to obstruction = loss of head due to enlargement from vena-contracta to section 2-2.

$$= \frac{(V_c - V)^2}{2g} \qquad \ldots(i)$$

From continuity, we have $a_c \times V_c = A \times V$ ...(*ii*)

where a_c = area of cross-section at vena-contracta

If C_c = co-efficient of contraction

Then $$C_c = \frac{\text{area at vena - contracta}}{(A-a)} = \frac{a_c}{(A-a)}$$

$\therefore$ $$a_c = C_c \times (A-a)$$

Substituting this value in (*ii*), we get

$$C_c \times (A-a) \times V_c = A \times V \quad \therefore \quad V_c = \frac{A \times V}{C_c\,(A-a)}$$

Substituting this value of V_c in equation (*i*), we get

$$\text{Head loss due to obstruction} = \frac{(V_c - V)^2}{2g} = \frac{\left(\dfrac{A \times V}{C_c\,(A-a)} - V\right)^2}{2g} = \frac{V^2}{2g}\left(\frac{A}{C_c\,(A-a)} - 1\right)^2 \qquad \ldots(11.10)$$

Loss of Head due to Bend in Pipe

When there is any bend in a pipe, the velocity of flow changes, due to which the separation of the flow from the boundary and also formation of eddies takes place. Thus the energy is lost. Loss of head in pipe due to bend is expressed as

$$h_b = \frac{kV^2}{2g}$$

where h_b = loss of head due to bend, V = velocity of flow, k = co-efficient of bend The value of k depends on

(i) Angle of bend, *(ii)* Radius of curvature of bend, *(iii)* Diameter of pipe.

Loss of Head in Various Pipe Fittings

The loss of head in the various pipe fittings such as valves, couplings etc., is expressed as

$$= \frac{kV^2}{2g} \qquad \text{...(11.11)}$$

where V = velocity of flow, k = co-efficient of pipe fitting.

HYDRAULIC GRADIENT AND TOTAL ENERGY LINE

The concept of hydraulic gradient line and total energy line is very useful in the study of flow of fluids through pipes. They are defined as :

Hydraulic Gradient Line.

It is defined as the line which gives the sum of pressure head ( ) and datum head (z) of a flowing fluid in a pipe with respect to some reference line or it is the line which is obtained by joining the top of all vertical ordinates, showing the pressure head (p/w) of a flowing fluid in a pipe from the center of the pipe. It is briefly written as H.G.L. (Hydraulic Gradient Line).

Total Energy Line.

It is defined as the line which gives the sum of pressure head, datum head and kinetic head of a flowing fluid in a pipe with respect to some reference line.

It is also defined as the line which is obtained by joining the tops of all vertical ordinates showing the sum of pressure head and kinetic head from the centre of the pipe. It is briefly written as T.E.L. (Total Energy Line).

FLOW THROUGH SYPHON

Syphon is a long bent pipe which is used to transfer liquid from a reservoir at a higher elevation to another reservoir at a lower level when the two reservoirs are separated by a hill or high level ground as shown in Fig. 14.

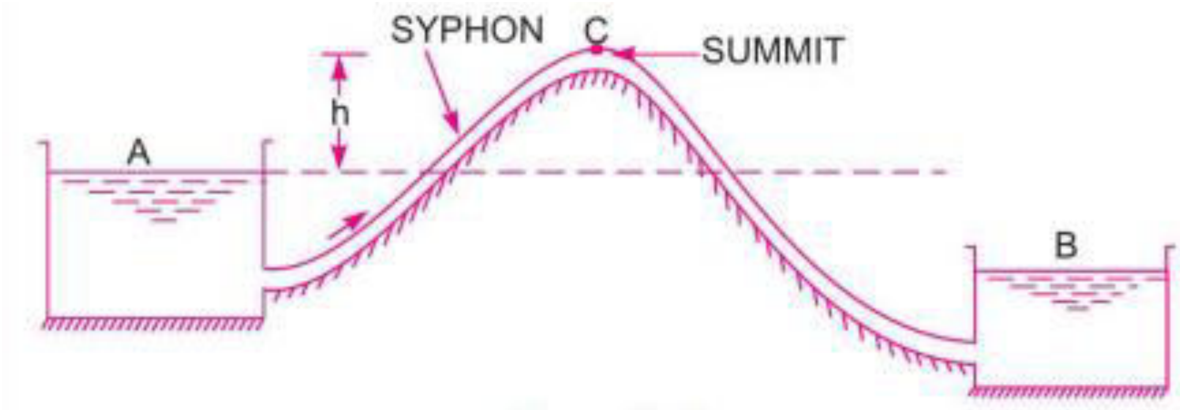


Fig. 14

The point C which is at the highest of the syphon is called the summit. As the point C is above the free surface of the water in the tank A, the pressure at C will be less than atmospheric pressure.

Theoretically, the pressure at C may be reduced to - 10.3 m of water but in actual practice this pressure is only - 7.6 m of water or 10.3 - 7.6 = 2.7 m of water absolute.

If the pressure at C becomes less than 2.7 m of water absolute, the dissolved air and other gases would come out from water and collect at the summit. The flow of water will be obstructed. Syphon is used in the following cases :

To carry water from one reservoir to another reservoir separated by a hill or ridge.

To take out the liquid from a tank which is not having any outlet.

To empty a channel not provided with any outlet sluice.

FLOW THROUGH PIPES IN SERIES OR FLOW THROUGH COMPOUND PIPES

Pipes in series or compound pipes are defined as the pipes of different lengths and different diameters connected end to end (in series) to form a pipe line as shown in Fig. 11.16.

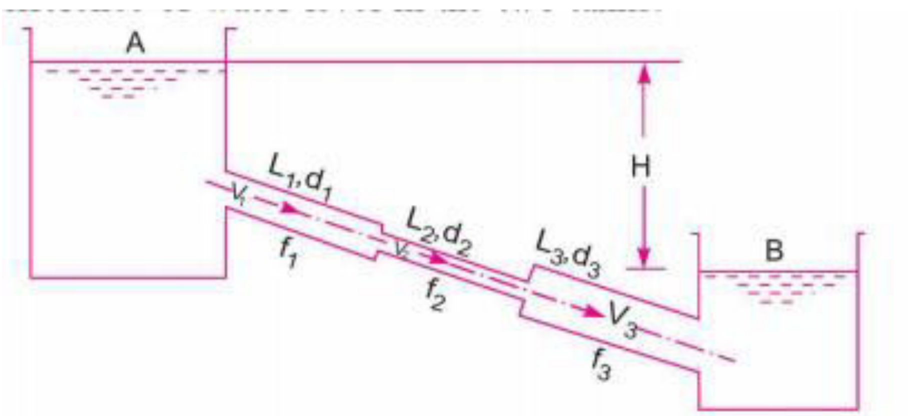


Fig. 15

Let,

L_1, L_2, L_3 = length of pipes 1, 2 and 3 respectively

d_1, d_2, d_3 = diameter of pipes 1, 2, 3 respectively

V_1, V_2, V_3 = velocity of flow through pipes 1, 2, 3

f_1 , f_2, f_3 = co-efficient of frictions for pipes 1, 2, 3

H = difference of water level in the two tanks.

The discharge passing through each pipe is same.

$$\therefore \qquad Q = A_1V_1 = A_2V_2 = A_3V_3$$

The difference in liquid surface levels is equal to the sum of the total head loss in the pipes.

$$\therefore \qquad H = \frac{0.5\,V_1^2}{2g} + \frac{4f_1L_1V_1^2}{d_1 \times 2g} + \frac{0.5\,V_2^2}{2g} + \frac{4f_2L_2V_2^2}{d_2 \times 2g} + \frac{(V_2 - V_3)^2}{2g} + \frac{4f_3L_3V_3^2}{d_3 \times 2g} + \frac{V_3^2}{2g} \quad ...(11.12)$$

If minor losses are neglected, then above equation becomes as

$$H = \frac{4f_1L_1V_1^2}{d_1 \times 2g} + \frac{4f_2L_2V_2^2}{d_2 \times 2g} + \frac{4f_3L_3V_3^2}{d_3 \times 2g} \quad ...(11.13)$$

If the co-efficient of friction is same for all pipes

i.e., $f_1 = f_2 = f_3 = f$, then equation (11.13) becomes as

$$H = \frac{4fL_1V_1^2}{d_1 \times 2g} + \frac{4fL_2V_2^2}{d_2 \times 2g} + \frac{4fL_3V_3^2}{d_3 \times 2g}$$

$$= \frac{4f}{2g}\left[\frac{L_1V_1^2}{d_1} + \frac{L_2V_2^2}{d_2} + \frac{L_3V_3^2}{d_3}\right] \quad ...(11.14)$$

EQUIVALENT PIPE

This is defined as the pipe of uniform diameter having loss of head and discharge equal to the loss of head and discharge of a compound pipe consisting of several pipes of different lengths and diameters.

The uniform diameter of the equivalent pipe is called equivalent size of the pipe. The length of equivalent pipe is equal to sum of lengths of the compound pipe consisting of different pipes.

Let L_1 = length of pipe 1 and d1 = diameter of pipe 1

L2 = length of pipe 2 and d2 = diameter of pipe 2

L3 = length of pipe 3 and d3 = diameter of pipe 3

H = total head loss

L = length of equivalent pipe

d = diameter of the equivalent pipe

Then $L = L1 + L2 + L3$

Total head loss in the compound pipe, neglecting minor losses

$$H = \frac{4f_1L_1V_1^2}{d_1 \times 2g} + \frac{4f_2L_2V_2^2}{d_2 \times 2g} + \frac{4f_3L_3V_3^2}{d_3 \times 2g} \quad \text{...(11.14A)}$$

Assuming $f_1 = f_2 = f_3 = f$

Discharge, $Q = A_1V_1 = A_2V_2 = A_3V_3 = \frac{\pi}{4}d_1^2V_1 = \frac{\pi}{4}d_2^2V_2 = \frac{\pi}{4}d_3^2V_3$

$\therefore$ $V_1 = \frac{4Q}{\pi d_1^2}, V_2 = \frac{4Q}{\pi d_2^2}$ and $V_3 = \frac{4Q}{\pi d_3^2}$

Substituting these values in equation (11.14*A*), we have

$$H = \frac{4fL_1 \times \left(\frac{4Q}{\pi d_1^2}\right)^2}{d_1 \times 2g} + \frac{4fL_2\left(\frac{4Q}{\pi d_2^2}\right)^2}{d_2 \times 2g} + \frac{4fL_3\left(\frac{4Q}{\pi d_3^2}\right)^2}{d_3 \times 2g}$$

$$= \frac{4 \times 16fQ^2}{\pi^2 \times 2g}\left[\frac{L_1}{d_1^5} + \frac{L_2}{d_2^5} + \frac{L_3}{d_3^5}\right] \quad \text{...(11.15)}$$

Head loss in the equivalent pipe, $H = \frac{4f\,.\,L\,.\,V^2}{d \times 2g}$ [Taking same value of f as in compound pipe]

where $V = \frac{Q}{A} = \frac{Q}{\frac{\pi}{4}d^2} = \frac{4Q}{\pi d^2}$

$$\therefore \quad H = \frac{4f\,.\,L\,.\left(\frac{4Q}{\pi d^2}\right)^2}{d \times 2g} = \frac{4 \times 16Q^2f}{\pi^2 \times 2g}\left[\frac{L}{d^5}\right] \quad \text{...(11.16)}$$

Head loss in compound pipe and in equivalent pipe is same hence equating equations (11.15) and (11.16), we have

$$\frac{4 \times 16fQ^2}{\pi^2 \times 2g}\left[\frac{L_1}{d_1^5} + \frac{L_2}{d_2^5} + \frac{L_3}{d_3^5}\right] = \frac{4 \times 16Q^2f}{\pi^2 \times 2g}\left[\frac{L}{d^5}\right]$$

or $$\frac{L_1}{d_1^5} + \frac{L_2}{d_2^5} + \frac{L_3}{d_3^5} = \frac{L}{d^5} \quad \text{or} \quad \frac{L}{d^5} = \frac{L_1}{d_1^5} + \frac{L_2}{d_2^5} + \frac{L_3}{d_3^5} \quad \text{...(11.17)}$$

Equation (11.17) is known as Dupuit's equation. In this equation $L = L_1 + L_2 + L_3$ and d_1, d_2 and d_3 are known. Hence the equivalent size of the pipe, *i.e.*, value of d can be obtained.

FLOW THROUGH PARALLEL PIPES

Consider a main pipe which divides into two or more branches as shown in Fig. 16 and again join together downstream to form a single pipe, then the branch pipes are said to be connected in parallel. The discharge through the main is increased by connecting pipes in parallel.

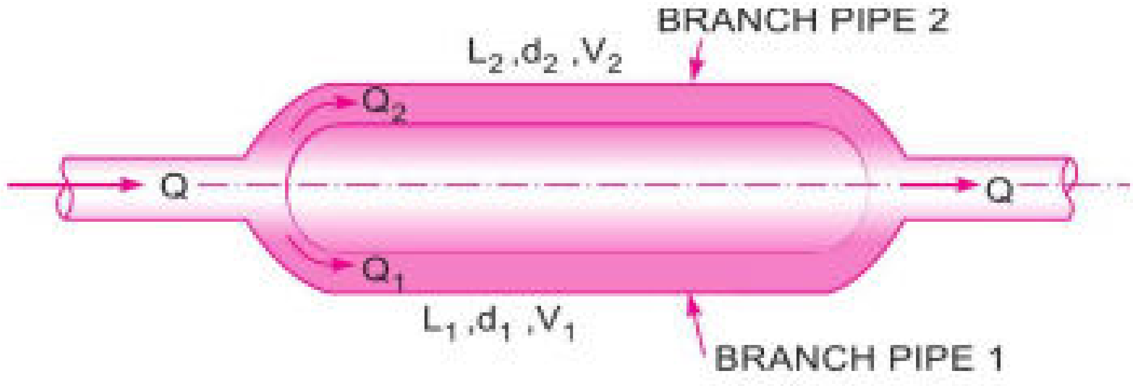


Fig.16

The rate of flow in the main pipe is equal to the sum of rate of flow through branch pipes. Hence from Fig. 16, we have

$Q = Q_1 + Q_2$

In this, arrangement, the loss of head for each branch pipe is same.

∴ Loss of head for branch pipe **1** = Loss of head for branch pipe 2

$$\text{or} \qquad \frac{4f_1L_1V_1^2}{d_1 \times 2g} = \frac{4f_2L_2V_2^2}{d_2 \times 2g} \qquad \text{...(11.19)}$$

$$\text{If} \qquad f_1 = f_2, \text{ then } \frac{L_1V_1^2}{d_1 \times 2g} = \frac{L_2V_2^2}{d_2 \times 2g} \qquad \text{...(11.20)}$$

FLOW THROUGH BRANCHED PIPES

When three or more reservoirs are connected by means of pipes, having one or more junctions, the system is called a branching pipe system. Fig. 17 shows three reservoirs at different levels connected to a single junction, by means of pipes which are called branched pipes. The lengths, diameters and co-efficient of friction of each pipes is given. It is required to find the discharge and direction of flow in each pipe.

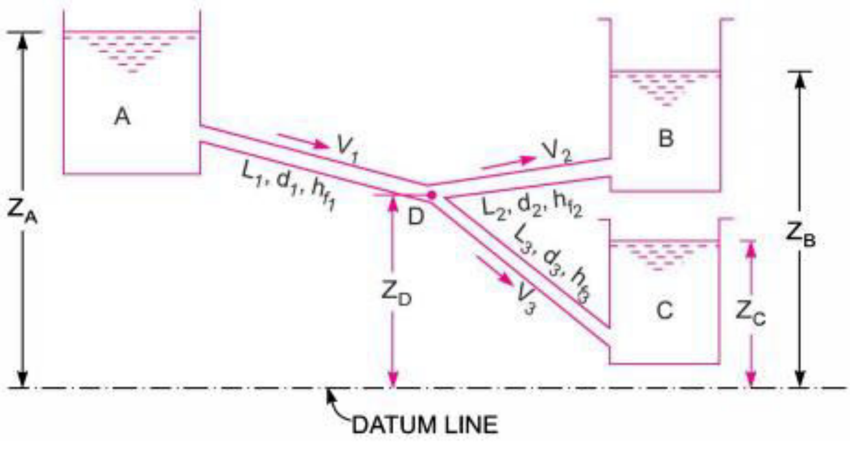


Fig.17

The basic equations used for solving such problems are :

1. Continuity equation which means the inflow of fluid at the junction should be equal to the outflow of fluid.
2. Bernoulli' s equation, and
3. Darcy-Weisbach equation

Also it is assumed that reservoirs are very large and the water surface levels in the reservoirs are constant so that steady conditions exist in the pipes. Also minor losses are assumed very small.

The flow from reservoir *A* takes place to junction *D*. The flow from junction *D* is towards reservoirs C. Now the flow from junction *D* towards reservoir *B* will take place only when piezometric head at *D* (which is equal to $\frac{p_D}{\rho g}$+ *ZD)* is more than the piezometric head at *B (i.e.,* Z_B*).* Let us consider that flow is from *D* to reservoir *B*.

For flow from *A* to *D* from Bernoulli's equation

$$Z_A = Z_D + \frac{p_D}{\rho g} + h_{f_1} \qquad \text{...}(i)$$

For flow from *D* to *B* from Bernoulli's equation

$$Z_D + \frac{p_D}{\rho g} = Z_B + h_{f_2} \qquad \text{...}(ii)$$

For flow from *D* to *C* from Bernoulli's equation

$$Z_D + \frac{p_D}{\rho g} = Z_C + h_{f_3} \qquad \text{...}(iii)$$

From continuity equation,

Discharge through *AD* = Discharge through *DB* + Discharge through *DC*

$$\therefore \qquad \frac{\pi}{4} d_1^2 V_1 = \frac{\pi}{4} d_2^2 \times V_2 + \frac{\pi}{4} d_3^2 V_3$$

or

$$d_1^2 V_1 = d_2^2 V_2 + d_3^2 V_3 \qquad \text{...}(iv)$$

There are four unknowns *i.e.*, V_1, V_2, V_3 and $\frac{p_D}{\rho g}$ and there are four equations (*i*), (*ii*), (*iii*) and (*iv*).

Hence unknown can be calculated.

POWER TRANSMISSION THROUGH PIPES

Power is transmitted through pipes by flowing water or other liquids flowing through them.

The power transmitted depends upon

(i) the weight of liquid flowing through the pipe and
(ii) the total head available at the end of the pipe.

Consider a pipe *AB* connected to a tank as shown in Fig. 18. The power available at the end *B* of the pipe and the condition for maximum transmission of power will be obtained as mentioned below

Let

L = length of the pipe,

d = diameter of the pipe,

H = total head available at

the inlet of pipe,

V = velocity of flow in pipe,

h1= loss of head due to

friction, and

f = co-efficient of friction. The head available at the outlet of the pipe, if minor losses are neglected = Total head at inlet - loss of head due to friction

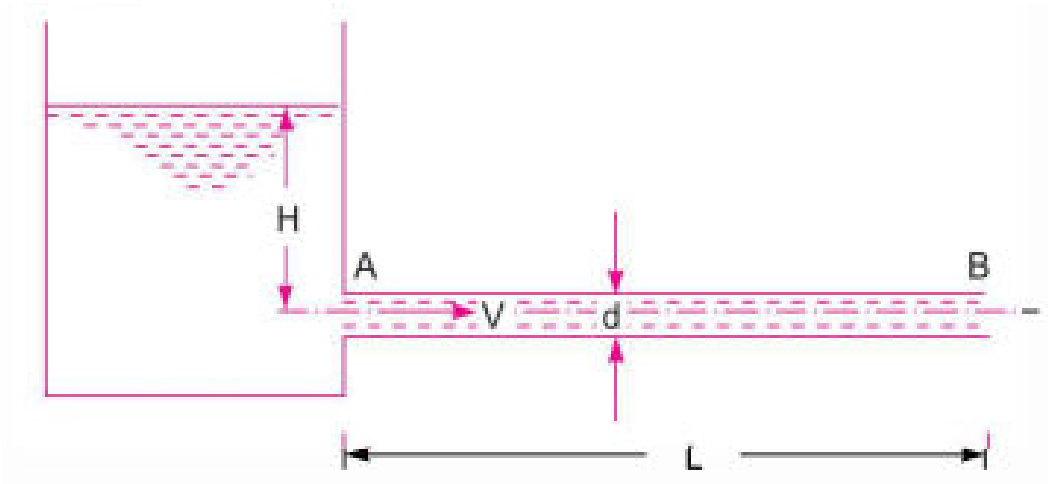


Fig.18 Power Transmission through pipe

$$= H - h_f = H - \frac{4f \times L \times V^2}{d \times 2g} \qquad \left\{\because\ h_f = \frac{4f \times L \times V^2}{d \times 2g}\right\}$$

Weight of water flowing through pipe per sec,

$$W = \rho g \times \text{volume of water per sec} = \rho g \times \text{Area} \times \text{Velocity}$$

$$= \rho g \times \frac{\pi}{4} d^2 \times V$$

$\therefore$ The power transmitted at the outlet of the pipe

$$= \text{weight of water per sec} \times \text{head at outlet}$$

$$= \left(\rho g \times \frac{\pi}{4} d^2 \times V\right) \times \left(H - \frac{4f \times L \times V^2}{d \times 2g}\right) \text{ Watts}$$

$\therefore$ Power transmitted at outlet of the pipe,

$$P = \frac{\rho g}{1000} \times \frac{\pi}{4} d^2 \times V \left(H - \frac{4fLV^2}{d \times 2g}\right) \text{ kW} \qquad \text{...(11.21)}$$

Efficiency of power transmission,

$$\eta = \frac{\text{Power available at outlet of the pipe}}{\text{Power supplied at the inlet of the pipe}}$$

$$= \frac{\text{Weight of water per sec} \times \text{Head available at outlet}}{\text{Weight of water per sec} \times \text{Head at inlet}}$$

$$= \frac{W \times (H - h_f)}{W \times H} = \frac{H - h_f}{H}. \qquad \text{...(11.22)}$$

CONDITION FOR MAXIMUM TRANSMISSION OF POWER

The condition for maximum transmission of power is obtained by differentiating equation (11.21) with respect to V and equating the same to zero.

Thus $$\frac{d}{dV}(P) = 0$$

or $$\frac{d}{dV}\left[\frac{\rho g}{1000} \times \frac{\pi}{4} d^2 \left(HV - \frac{4fLV^3}{d \times 2g}\right)\right] = 0$$

or $$\frac{\rho g}{1000} \times \frac{\pi}{4} d^2 \left(H - \frac{4 \times 3 \times f \times L \times V^2}{d \times 2g}\right) = 0$$

or $$H - 3 \times \frac{4fLV^2}{d \times 2g} = 0 \text{ or } H - 3 \times h_f = 0 \qquad \left(\because \frac{4fLV^2}{d \times 2g} = h_f\right)$$

$$\therefore \qquad H = 3h_f \text{ or } h_f = \frac{H}{3} \qquad \text{...(11.23)}$$

Equating (11.23) is the condition for maximum transmission of power. It states that power transmit ted through a pipe is maximum when the loss of head due to friction is one-third of the total head at inlet.

Maximum Efficiency of Transmission of Power

Efficiency of power transmission through pipe is given by equation (11.22) as

$$\eta = \frac{H - h_f}{H}$$

For maximum power transmission through pipe the condition is given by equation (11.23) as

$$h_f = \frac{H}{3}$$

Substituting the value of h_f in efficiency, we get maximum η,

$$\eta_{max} = \frac{H - H/3}{H} = 1 - \frac{1}{3} = \frac{2}{3} \text{ or } 66.7\%. \qquad \text{...(11.24)}$$

FLOW THROUGH NOZZLES

Fig. 19 shows a nozzle fitted at the end of a long pipe. The total energy at the end of the pipe consists of pressure energy and kinetic energy.

By fitting the nozzle at the end of the pipe, the total energy is converted into kinetic energy. Thus nozzles are used, where higher velocities of flow are required. The examples are

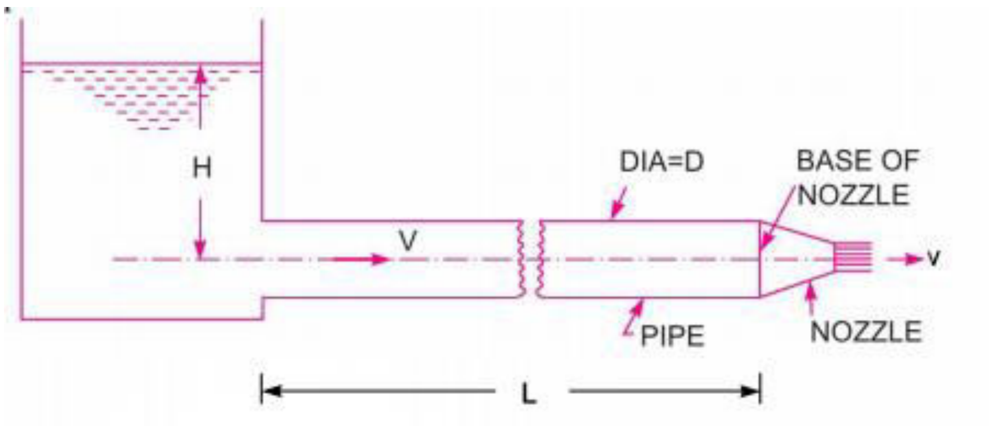


Figure 19 Nozzle fitted to a pipe

In case of Pelton turbine, the nozzle is fitted at the end of the pipe (called penstock) to increase velocity.

In case of the extinguishing fire, a nozzle is fitted at the end of the hose pipe to increase velocity.

Let D = diameter of the pipe,

L = length of the pipe,

A = area of the pipe $= \frac{\pi}{4}D^2$,

V = velocity of flow in pipe,

H = total head at the inlet of the pipe,

d = diameter of nozzle at outlet,

v = velocity of flow at outlet of nozzle,

a = area of the nozzle at outlet $= \frac{\pi}{4}d^2$,

f = co-efficient of friction for pipe.

Loss of head due to friction in pipe, $h_f = \frac{4fLV^2}{2gXD}$

∴ Head available at the end of the pipe or at the base of nozzle = Head at inlet of pipe - head lost due to friction

$$= H - h_f = \left(H - \frac{4fLV^2}{2g \times D}\right)$$

Neglecting minor losses and also assuming losses in the nozzle negligible, we have

Total head at inlet of pipe = total head (energy) at the outlet of nozzle + losses

But total head at outlet of nozzle = kinetic head = $\frac{v^2}{2g}$

$$\therefore \qquad H = \frac{v^2}{2g} + h_f = \frac{v^2}{2g} + \frac{4fLV^2}{2gD} \qquad \left(\because h_f = \frac{4fLV^2}{2gD}\right) \quad ...(i)$$

From continuity equation in the pipe and outlet of nozzle,

$$AV = av$$

$$\therefore \qquad V = \frac{av}{A}$$

Substituting this value in equation (*i*), we get

$$H = \frac{v^2}{2g} + \frac{4fL}{2gD} \times \left(\frac{av}{A}\right)^2 = \frac{v^2}{2g} + \frac{4fLa^2v^2}{2g \times D \times A^2} = \frac{v^2}{2g}\left(1 + \frac{4fLa^2}{DA^2}\right)$$

$$\therefore \qquad v = \sqrt{\frac{2gH}{\left(1 + \frac{4fL}{D} \times \frac{a^2}{A^2}\right)}} \qquad ...(11.25)$$

$\therefore$ Discharge through nozzle = $a \times v$.

11.12.1 Power Transmitted Through Nozzle.

The kinetic energy of the jet at the outlet of nozzle = $\frac{1}{2} mv^2$

Now mass of liquid at the outlet of nozzle per second = ρav

$\therefore$ Kinetic energy of the jet at the outlet per sec. = $\frac{1}{2} \rho av \times v^2 = \frac{1}{2} \rho av^3$

$\therefore$ Power in kW at the outlet of nozzle = (K.E./sec) $\times \frac{1}{1000} = \frac{\frac{1}{2}\rho av^3}{1000}$

$\therefore$ Efficiency of power transmission through nozzle,

$$\eta = \frac{\text{Power at outlet of nozzele}}{\text{Power at the inlet of pipe}} = \frac{\dfrac{\frac{1}{2}\rho a v^3}{1000}}{\dfrac{\rho g . Q . H}{1000}}$$

$$= \frac{\frac{1}{2}\rho av . v^2}{\rho g . Q . H} = \frac{\frac{1}{2}\rho av . v^2}{\rho g . av . H} \qquad \{\because Q = av\}$$

$$= \frac{v^2}{2gH} = \left[\frac{1}{1 + \frac{4fL}{D} \times \frac{a^2}{A^2}}\right] \qquad ...(11.26)$$

$$\left(\because \text{ From equation (11.25), } \frac{v^2}{2gH} = \left[\frac{1}{1 + \frac{4fL}{D}\frac{a^2}{A^2}}\right]\right)$$

11.12.2 Condition for Maximum Power Transmitted Through Nozzle. We know that, the total head at inlet of pipe = total head at the outlet of the nozzle + losses

i.e.,
$$H = \frac{v^2}{2g} + h_f \qquad \left[\begin{array}{l}\because \text{total head at outlet of nozzle} = \dfrac{v^2}{2g} \text{ and} \\ h_f = \dfrac{4 . f . L . V^2}{D \times 2g} = \text{loss of liquid in pipe}\end{array}\right]$$

$$= \frac{v^2}{2g} + \frac{4 . f . L . V^2}{D \times 2g}$$

$$\therefore \quad \frac{v^2}{2g} = \left(H - \frac{4 . f . L . V^2}{D \times 2g}\right)$$

But power transmitted through nozzle $= \dfrac{\frac{1}{2}\rho a v^3}{1000} = \dfrac{\frac{1}{2}\rho av}{1000} \times v^2 = \dfrac{\frac{1}{2}\rho av}{1000}\left[2g\left(H - \dfrac{4 . f . L . V^2}{D \times 2g}\right)\right]$

$$= \frac{\rho g a v}{1000}\left[H - \frac{4fLV^2}{D \times 2g}\right] \qquad \text{...(11.27)}$$

Now from continuity equation, $AV = av$

$$\therefore \qquad V = \frac{av}{A}$$

Substituting the value of V in equation (11.27), we get

$$\text{Power transmitted through nozzle} = \frac{\rho g a v}{1000}\left[H - \frac{4fLa^2}{D \times 2g}\frac{v^2}{A^2}\right]$$

The power (P) will be maximum, when $\dfrac{d(P)}{dv} = 0$

or $$\frac{d}{dv}\left[\frac{\rho g a v}{1000}\left(H - \frac{4fL}{D \times 2g}\frac{a^2 v^2}{A^2}\right)\right] = 0$$

or $$\frac{d}{dv}\left[\frac{\rho g a}{1000}\left(Hv - \frac{4fL}{D \times 2g}\frac{a^2 v^3}{A^2}\right)\right] = 0$$

or $$\frac{d}{dv}\left[\frac{\rho g a}{1000}\left(Hv - \frac{4fL}{D \times 2g}\frac{a^2 v^3}{A^2}\right)\right] = 0$$

or $$\left[\frac{\rho g a}{1000}\left(H - 3\frac{4fL}{D \times 2g}\frac{a^2 v^2}{A^2}\right)\right] = 0 \text{ or } H - 3 \times \frac{4fL}{D \times 2g} \times V^2 = 0 \left(\because\ V = \frac{av}{A}\right)$$

or $$H - 3h_f = 0 \qquad \left(\because\ \frac{4fLV^2}{D \times 2g} = h_f = \text{head loss in pipe}\right)$$

or $$h_f = \frac{H}{3} \qquad \text{...(11.28)}$$

Equation (11.28) gives the condition for maximum power transmitted through nozzle. It states that power transmitted through nozzle is maximum when the head lost due to friction in pipe is one-third the total head supplied at the inlet of pipe.

11.12.3 Diameter of Nozzle for Maximum Transmission of Power Through Nozzle.

For maximum transmission of power, the condition is given by equation (11.28) as, $h_f = \frac{H}{3}$

But $$h_f = \frac{4f \cdot L \cdot V^2}{D \times 2g}$$

$\therefore$ $$\frac{4fLV^2}{D \times 2g} = \frac{H}{3} \text{ or } H = 3 \times \frac{4fLV^2}{D \times 2g}$$

But H is also = total head at outlet of nozzle + losses

$$= \frac{v^2}{2g} + h_f = \frac{v^2}{2g} + \frac{4fLV^2}{D \times 2g}$$

Equating the two values of H, we get

$$3 \times \frac{4fLV^2}{D \times 2g} = \frac{v^2}{2g} + \frac{4fLV^2}{D \times 2g} \text{ or } \frac{12fLV^2}{D \times 2g} - \frac{4fLV^2}{D \times 2g} = \frac{v^2}{2g}$$

or $$\frac{8fLV^2}{D \times 2g} = \frac{v^2}{2g} \qquad ...(i)$$

But from continuity, $AV = av$ or $V = \frac{av}{A}$.

Substituting this value of V in equation (i), we get

$$\frac{8fL}{D \times 2g} \times \frac{a^2v^2}{A^2} = \frac{v^2}{2g} \text{ or } \frac{8fL}{D} \times \frac{a^2}{A^2} = 1 \qquad \left(\text{Divide by } \frac{v^2}{2g}\right) ...(ii)$$

or $$\frac{8fL}{D} \times \frac{\left(\frac{\pi}{4}d^2\right)^2}{\left(\frac{\pi}{4}D^2\right)^2} = 1 \text{ or } \frac{8fL}{D} \times \frac{d^4}{D^4} = 1 \text{ or } d^4 = \frac{D^5}{8fL}$$

$\therefore$ $$d = \left(\frac{D^5}{8fL}\right)^{1/4} \qquad ...(11.29)$$

From equation (ii), $$\frac{8fL}{D} = \frac{A^2}{a^2}$$

$\therefore$ $$\frac{A}{a} = \sqrt{\frac{8fL}{D}} \qquad ...(11.30)$$

Equation (11.30) gives the ratio of the area of the supply pipe to the area of the nozzle and hence from this equation, the diameter of the nozzle can be obtained.

SATHYABAMA

INSTITUTE OF SCIENCE AND TECHNOLOGY

(DEEMED TO BE UNIVERSITY)

Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE

www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – III –FLUID FLOW OVER BODIES AND BOUNDARY LAYER THEORY – SAEA1304

I.INTRODUCTION TO BOUNDARY LAYER FLOW

When a real fluid flows past a solid body or a solid wall, the fluid particles adhere to the boundary and condition of no slip occurs. This means that the velocity of fluid close to the boundary will be same as that of the boundary.

If the boundary is stationary, the velocity of fluid at the boundary will be zero. Farther away from the boundary, the velocity will be higher and as a result of this variation of velocity, the velocity gradient *du/dy* will exist. The velocity of fluid increases from zero velocity on the stationary boundary to free-stream velocity (*U)* of the fluid in the direction normal to the boundary. This variation of velocity from zero to free-stream velocity in the direction normal to the boundary takes place in a narrow region in the vicinity of solid boundary.

This narrow region of the fluid is called boundary layer. The theory dealing with boundary layer flows is called boundary layer theory.

According to boundary layer theory, the flow of fluid in the neighbourhood of the solid boundary may be divided into two regions as shown in Fig. 20.

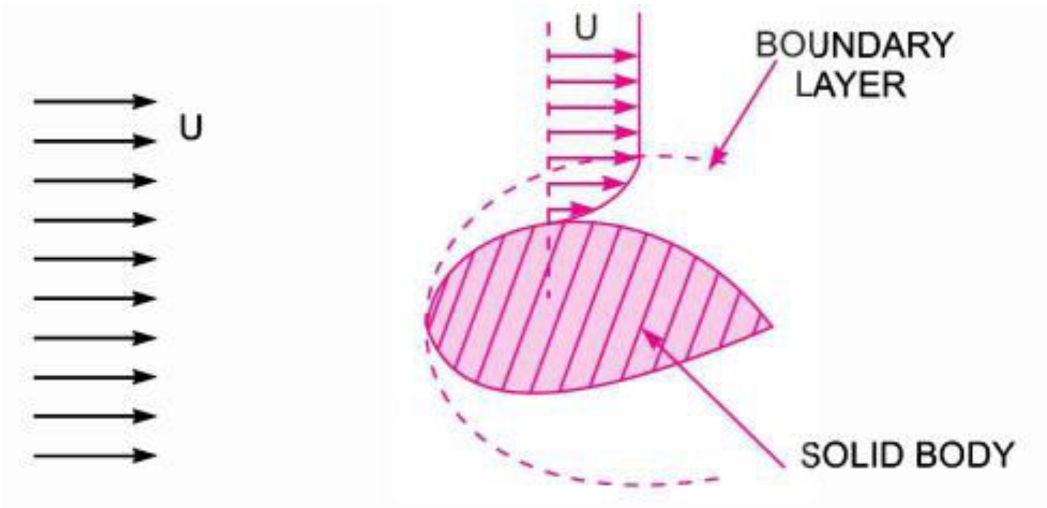


Figure 20. Flow over solid body

A very thin layer of the fluid, called the boundary layer, in the immediate neighbourhood of the solid boundary, where the variation of velocity from zero at the solid boundary to free-stream velocity in the direction normal to the boundary takes place.

In this region, the velocity gradient *du/dy* exists andhence the fluid exerts a shear stress on the wall in the direction of motion. The value of shear stress is given by

$$\tau = \mu \frac{du}{dy}$$

The remaining fluid, which is outside the boundary layer. The velocity outside the boundary layer is constant and equal to free-stream velocity.

As there is no variation of velocity in this region, the velocity gradient *du/dy* becomes zero. As a result of this the shear stress is zero.

Laminar Boundary Layer

For defining the boundary layer *(i.e.,* laminar boundary layer or turbulent boundary layer) consider the flow of a fluid, having free-stream velocity (*U),* over a smooth thin plate which is flat and placed parallel to the direction for free stream of fluid as shown in Fig. 20.

Let us consider the flow with zero pressure gradient on one side of the plate, which is stationary.

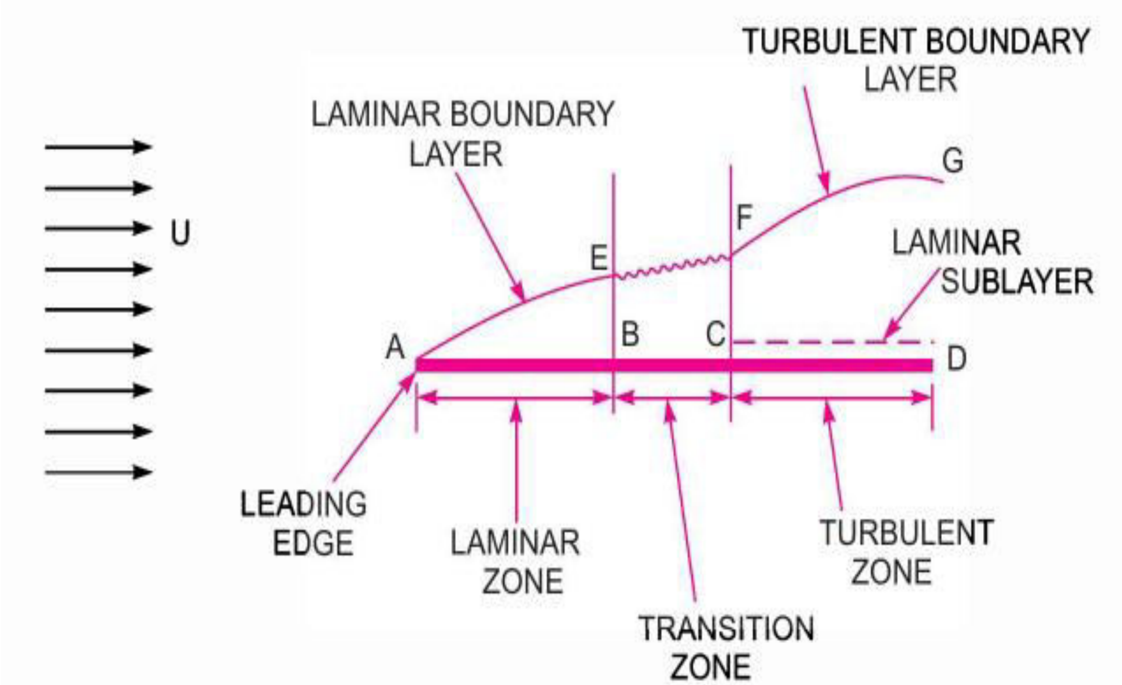


Figure 20. Flow over a plate

The velocity of fluid on the surface of the plate should be equal to the velocity of the plate. But plate is stationary and hence velocity of fluid on the surface of the plate is zero.

But at a distance away from the plate, the fluid is having certain velocity. Thus a velocity gradient is set up in the fluid near the surface of the plate.

This velocity gradient develops shear resistance, which retards the fluid. Thus the fluid with a uniform free stream velocity (*U)* is retarded in the vicinity of the solid surface of the plate and the boundary layer region begins at the sharp leading edge.

At subsequent points downstream the leading edge, the boundary layer region increases because the retarded fluid is further retarded. This is also referred as the growth of boundary layer.

Near the leading edge of the surface of the plate, where the thickness is small, the flow in the boundary layer is laminar though the main flow is turbulent. This layer of the fluid is said to be laminar boundary layer. This is shown by *AE* in Fig. 20.

The length of the plate from the leading edge, up to which laminar boundary layer exists, is called laminar zone. This is shown by distance AB. The distance of B from leading edge is obtained from Reynold number equal to 5 x 10^5 for a plate. Because up to this Reynold number the boundary layer is laminar.

The Reynold number is given by $(R_e)_x = \frac{U X x}{\vartheta}$

where

x = Distance from leading edge,

U = Free-stream velocity of fluid,

v = Kinematic viscosity of fluid,

Hence for laminar boundary layer, we have 5 x $10^5 = \frac{U X x}{\vartheta}$ ---(13.1)

If the values of U and v are known, x or the distance from the leading edge up to which laminar boundary layer exists can be calculated.

Turbulent Boundary Layer

If the length of the plate is more than the distance x, calculated from equation (13.1), the thickness of boundary layer will go on increasing in the down stream direction.

Then the laminar boundary layer becomes unstable and motion of fluid within it, is disturbed and irregular which leads to a transition from laminar to turbulent boundary layer.

This short length over which the boundary layer flow changes from laminar to turbulent is called transition zone. This is shown by distance BC in Fig. 20.

Further downstream the transition zone, the boundary layer is turbulent and continues to grow in thickness. This layer of boundary is called turbulent boundary layer, which is shown by the portion FG in Fig. 20.

Laminar Sub-layer

This is the region in the turbulent boundary layer zone, adjacent to the solid surface of the plate as shown in Fig. 20.

In this zone, the velocity variation is influenced only by viscous effects.

Though the velocity distribution would be a parabolic curve in the laminar sub-layer zone, but in view of the very small thickness we can reasonably assume that velocity variation is linear and so the velocity gradient can be considered constant.

Therefore, the shear stress in the laminar sub-layer would be constant and equal to the boundary shear stress τ_0. Thus the shear stress in the sub-layer is

$$\tau_0 = \mu \left(\frac{\partial u}{\partial y}\right)_{y=0} = \mu \frac{u}{y} \qquad \left\{\because \text{ For linear variation, } \frac{\partial u}{\partial y} = \frac{u}{y}\right\}$$

Boundary Layer Thickness (δ).

It is defined as the distance from the boundary of the solid body measured in the y-direction to the point, where the velocity of the fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by the symbol δ_O.

For laminar and turbulent zone it is denoted as :

δ_{Lam} = Thickness of laminar boundary layer,

$\delta\, t_{ur}$ = Thickness of turbulent boundary layer, and

δ' = Thickness of laminar sub-layer.

Displacement Thickness ($\delta *$)

It is defined as the distance, measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation. It is denoted by ($\delta *$).

It is also defined as :

"The distance perpendicular to the boundary, by which the free-stream is displaced due to the formation of boundary layer". Expression for $\delta *$

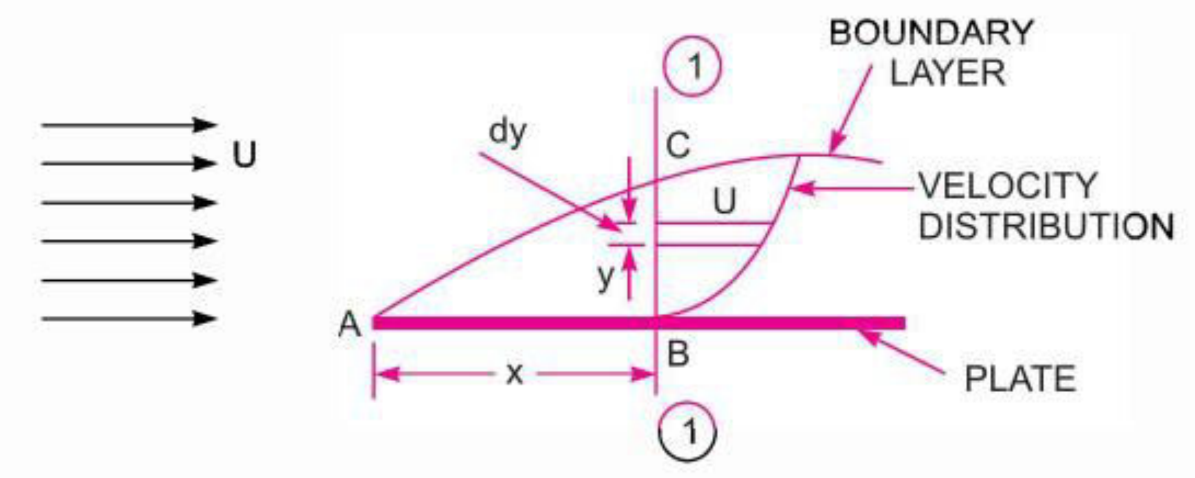


Figure 21. Displacement thickness

Consider the flow of a fluid having free-stream velocity equal to U over a thin smooth plate as shown in Fig. 21.

At a distance x from the leading edge consider a section 1-1. The velocity of fluid at B i s zero and at C, which lies on the boundary layer, is U.

Thus velocity varies from zero at *B* to *U* at C, where *BC* is equal to the thickness of boundary layer *i.e.,*

Distance *BC* = 8

At the section 1-1, consider an elemental strip.

Let

y = distance of elemental strip from the plate,

dy = thickness of the elemental strip,

u = velocity of fluid at the elemental strip,

b = width of plate.

Then area of elemental strip,

dA = *b* x *dy*

Mass of fluid per second flowing through elemental strip

= p x Velocity x Area of elemental strip

= pu x *dA* = pu x *b x dy* ...(i)

If there had been no plate, then the fluid would have been flowing with a constant velocity equal to free-stream velocity (*U)* at the section 1-1. Then mass of fluid per second flowing through elemental strip would have been

= p x Velocity x Area = p x *U* x *b x dy* *...(ii)*

As *U* is more than *u,* hence due to the presence of the plate and consequently due to the formation of the boundary layer, there will be a reduction in mass flowing per second through the elemental strip.

This reduction in mass/sec flowing through elemental strip

$$= \text{mass/sec given by equation } (ii) - \text{mass/sec given by equation } (i)$$
$$= \rho U b dy - \rho u b dy = \rho b(U - u)dy$$

∴ Total reduction in mass of fluid/s flowing through BC due to plate

$$= \int_0^\delta \rho b(U - u)dy = \rho b \int_0^\delta (U - u)dy \qquad ...(iii)$$

{if fluid is incompressible}

Let the plate is displaced by a distance δ^* and velocity of flow for the distance δ^* is equal to the free-stream velocity (*i.e.*, U). Loss of the mass of the fluid/sec flowing through the distance δ^*

$$= \rho \times \text{Velocity} \times \text{Area}$$
$$= \rho \times U \times \delta^* \times b \qquad \{\because \text{Area} = \delta^* \times b\} \;...(iv)$$

Equating equation (*iii*) and (*iv*), we get

$$\rho b \int_0^\delta (U - u)dy = \rho \times U \times \delta^* b$$

Cancelling ρb from both sides, we have

$$\int_0^\delta (U - u)dy = U \times \delta^*$$

or

$$\delta^* = \frac{1}{U}\int_0^\delta (U - u)dy = \int_0^\delta \frac{(U - u)dy}{U} \qquad \left\{\begin{array}{l}\because \; U \text{ is constant and can} \\ \text{be taken inside the integral}\end{array}\right\}$$

∴

$$\delta^* = \int_0^\delta \left(1 - \frac{u}{U}\right) dy. \qquad ...(13.2)$$

Momentum Thickness (θ).

Momentum thickness is defined as the distance, measured perpendicular to the boundary of the solid body, by which the boundary should be displaced

to compensate for the reduction in **momentum** of the flowing fluid on account of boundary layer formation. It is denoted by θ.

Consider the flow over a plate as shown in Fig. 21. Consider the section **1-1** at a distance *x* from leading edge. Take an elemental strip at a distance *y* from the plate having thickness *(dy).* The mass of fluid flowing per second through this elemental strip is given by equation (i) and is equal to *pubdy.*

Momentum of this fluid = Mass x Velocity = *(pubdy)u*

Momentum of this fluid in the absence of boundary layer = *(pubdy)U*

Loss of momentum through elemental strip = *(pubdy)U - (pubdy)* x *u = pbu(U - u)dy*

Total loss of momentum/sec through $BC = \int_0^\delta pbu(U - u)dy$ ...(13.3)

Let θ = distance by which plate is displaced when the fluid is flowing with a constant velocity U

$\therefore$ Loss of momentum/sec of fluid flowing through distance θ with a velocity U

= Mass of fluid through θ × velocity

= (ρ × area × velocity) × velocity

$= [\rho \times \theta \times b \times U] \times U$ $\{\because \text{Area} = \theta \times b\}$

$= \rho\theta b U^2$ ...(13.4)

Equating equations (13.4) and (13.3), we have

$$\rho\theta b U^2 = \int_0^\delta \rho b u(U-u)dy = \rho b \int_0^\delta u(U-u)dy \quad \{\text{If fluid is assumed incompressible}\}$$

or $$\theta U^2 = \int_0^\delta u(U-u)dy \quad \{\text{cancelling } \rho b \text{ from both sides}\}$$

or $$\theta = \frac{1}{U^2}\int_0^\delta u(U-u)dy = \int_0^\delta \frac{u(U-u)dy}{U^2}$$

$\therefore$ $$\theta = \int_0^\delta \frac{u}{U}\left[1-\frac{u}{U}\right]dy. \quad ...(13.5)$$

Energy Thickness (δ **).

It is defined as the distance measured perpendicular to the boundary of the solid body, by which the boundary should be displaced to compensate for the reduction in kinetic energy of the flowing fluid on account of boundary layer formation. It is denoted by δ **.

Consider the flow over the plate as shown in Fig. 13.3 having section **1-1** at a distance x from leading edge. The mass of fluid flowing per second through the elemental strip of thickness *'dy'* at a distance y from the plate as given by equation (i) = *pubdy*

Kinetic energy of this fluid $= \frac{1}{2} m \times \text{velocity}^2 = \frac{1}{2}(\rho u b dy)\, u^2$

Kinetic energy of this fluid in the absence of boundary layer

$$= \frac{1}{2}(\rho u b dy) U^2$$

$\therefore$ Loss of K.E. through elemental strip

$$= \frac{1}{2}(\rho u b dy)U^2 - \frac{1}{2}(\rho u b dy)\, u^2 = \frac{1}{2}\rho u b\, [U^2 - u^2]\, dy$$

$\therefore$ Total loss of K.E. of fluid passing through BC

$$= \int_0^\delta \frac{1}{2} \rho u b \, [U^2 - u^2] \, dy = \frac{1}{2} \rho b \int_0^\delta u \, (U^2 - u^2) \, dy$$

{If fluid is considered incompressible}

Let δ^{**} = distance by which the plate is displaced to compensate for the reduction in K.E.

$\therefore$ Loss of K.E. through δ^{**} of fluid flowing with velocity U

$$= \frac{1}{2} \text{(mass)} \times \text{velocity}^2 = \frac{1}{2} (\rho \times \text{area} \times \text{velocity}) \times \text{velocity}^2$$

$$= \frac{1}{2} (\rho \times b \times \delta^{**} \times U) U^2 \qquad \{\because \quad \text{Area} = b \times \delta^{**}\}$$

$$= \frac{1}{2} \rho b \delta^{**} U^3$$

Equating the two losses of K.E., we get

$$\frac{1}{2} \rho b \delta^{**} U^3 = \frac{1}{2} \rho b \int_0^\delta u \, (U^2 - u^2) \, dy$$

or

$$\delta^{**} = \frac{1}{U^3} \int_0^\delta u \, (U^2 - u^2) \, dy$$

$\therefore$

$$\delta^{**} = \int_0^\delta \frac{u}{U} \left[1 - \frac{u^2}{U^2} \right] dy. \qquad \ldots(13.6)$$

DRAG FORCE ON A FLAT PLATE DUE TO BOUNDARY LAYER

Consider the flow of a fluid having free-stream velocity equal to *U,* over a thin plate as shown in Fig. 22. The drag force on the plate can be determined if the velocity profile near the plate is known.

Consider a small length *Ax* of the plate at a distance of *x* from the leading edge as show n in Fig. 22 (a). The enlarged view of the small length of the plate is shown in Fig. 22 (b).

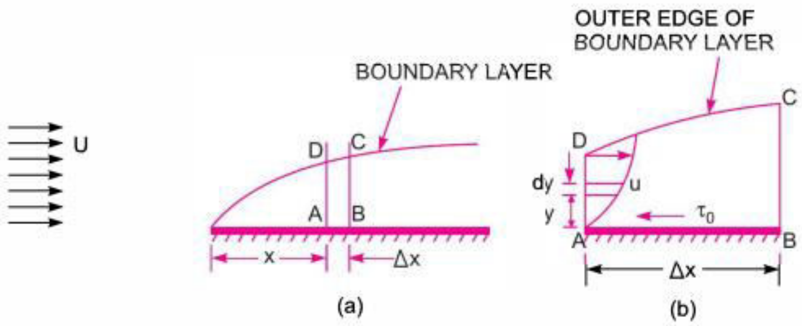


Fig. 13.4 *Drag force on a plate due to boundary layer.*

Figure 22. Drag force on a plate due to boundary layer

The shear stress τ_0 is given by $\tau_0 = \mu \left(\frac{du}{dy}\right)_{y=0}$, where $\left(\frac{du}{dy}\right)_{y=0}$ is the velocity distribution near the plate at $y = 0$.

Then drag force or shear force on a small distance Δx is given by

$$\Delta F_D = \text{shear stress} \times \text{area}$$

$$= \tau_0 \times \Delta x \times b \qquad \text{...(13.7)} \quad \{\text{Taking width of plate} = b\}$$

where ΔF_D = drag force on distance Δx

Then drag force or shear force on a small distance Δx is given by

$$\Delta F_D = \text{shear stress} \times \text{area}$$

$$= \tau_0 \times \Delta x \times b \qquad \text{...(13.7)} \quad \{\text{Taking width of plate} = b\}$$

where ΔF_D = drag force on distance Δx

The drag force ΔF_D must also be equal to the rate of change of momentum over the distance Δx.

Consider the flow over the small distance Δx. Let $ABCD$ is the control volume of the fluid over the distance Δx as shown in Fig. 13.4 (b). The edge DC represents the outer edge of the boundary layer.

Let u = velocity at any point within the boundary layer

b = width of plate

Then mass rate of flow entering through the side AD

$$= \int_0^\delta \rho \times \text{velocity} \times \text{area of strip of thickness } dy$$

$$= \int_0^\delta \rho \times u \times b \times dy \qquad \{\because \quad \text{Area of strip} = b \times dy\}$$

$$= \int_0^\delta \rho u b dy$$

Mass rate of flow leaving the side BC

$$= \text{mass through } AD + \frac{\partial}{\partial x}(\text{mass through } AD) \times \Delta x$$

$$= \int_0^\delta \rho u b dy \; \frac{\partial}{\partial x}\left[\int_0^\delta (\rho u b dy)\right] \times \Delta x$$

From continuity equation for a steady incompressible fluid flow, we have

Mass rate of flow entering AD + mass rate of flow entering DC

= mass rate of flow leaving BC

$\because$ Mass rate of flow entering DC = mass rate of flow through BC – mass rate of flow through AD

$$= \int_0^\delta \rho u b dy + \frac{\partial}{\partial x}\left[\int_0^\delta \rho u b dy\right] \times \Delta x - \int_0^\delta \rho u b dy$$

$$= \frac{\partial}{\partial x}\left[\int_0^\delta \rho u b dy\right] \times \Delta x$$

The fluid is entering through side DC with a uniform velocity U.

Now let us calculate momentum flux through control volume.

Momentum flux entering through AD

$$= \int_0^\delta \text{momentum flux through strip of thickness } dy$$

$$= \int_0^\delta \text{mass through strip} \times \text{velocity} = \int_0^\delta (\rho u b dy) \times u = \int_0^\delta \rho u^2 b dy$$

Momentum flux leaving the side $BC = \int_0^\delta \rho u^2 b dy + \frac{\partial}{\partial x}\left[\int_0^\delta \rho u^2 b dy\right] \times \Delta x$

Momentum flux entering the side DC = mass rate through DC × velocity

$$= \frac{\partial}{\partial x}\left[\int_0^\delta \rho u b dy\right] \times \Delta x \times U \qquad (\because \text{ Velocity} = U\}$$

$$= \frac{\partial}{\partial x}\left[\int_0^\delta \rho u U b dy\right] \times \Delta x$$

As U is constant and so it can be taken inside the differential and integral.

$\therefore$ Rate of change of momentum of the control volume

= Momentum flux through BC – Momentum flux through AD

– momentum flux through DC

$$= \int_0^\delta \rho u^2 b dy + \frac{\partial}{\partial x}\left[\int_0^\delta \rho u^2 b dy\right] \times \Delta x - \int_0^\delta \rho u^2 b dy - \frac{\partial}{\partial x}\left[\int_0^\delta \rho u U b dy\right] \times \Delta x$$

$$= \frac{\partial}{\partial x}\left[\int_0^\delta \rho u^2 b dy\right] \times \Delta x - \frac{\partial}{\partial x}\left[\int_0^\delta \rho u U b dy\right] \times \Delta x$$

$$= \frac{\partial}{\partial x}\left[\int_0^\delta \rho u^2 b dy - \int_0^\delta \rho u U b dy\right] \times \Delta x$$

$$= \frac{\partial}{\partial x}\left[\int_0^\delta \left(\rho u^2 b - \rho u U b\right) dy\right] \times \Delta x$$

$$= \frac{\partial}{\partial x}\left[\rho b \int_0^\delta \left(u^2 - uU\right) dy\right] \times \Delta x$$

{For incompressible fluid ρ is constant}

$$= \rho b \frac{\partial}{\partial x}\left[\int_0^\delta \left(u^2 - uU\right)dy\right] \times \Delta x \qquad \text{...(13.8)}$$

Now the rate of change of momentum on the control volume *ABCD* must be equal to the total force on the control volume in the same direction according to the momentum principle. But for a flat plate $\frac{\partial p}{\partial x} = 0$, which means there is no external pressure force on the control volume. Also the force on the side *DC* is negligible as the velocity is constant and velocity gradient is zero approximately. The only external force acting on the control volume is the shear force acting on the side *AB* in the direction from *B* to *A* as shown in Fig. 13.4 (*b*). The value of this force is given by equation (13.7) as

$$\Delta F_D = \tau_0 \times \Delta x \times b$$

$\therefore$ Total external force in the direction of rate of change of momentum

$$= -\tau_0 \times \Delta x \times b \qquad \text{...(13.9)}$$

According to momentum principle, the two values given by equations (13.9) and (13.8) should be the same.

$$\therefore \qquad -\tau_0 \times \Delta x \times b = \rho b \frac{\partial}{\partial x}\left[\int_0^\delta \left(u^2 - uU\right)dy\right] \times \Delta x$$

Cancelling $\Delta x \times b$, to both sides, we have

$$-\tau_0 = \rho \frac{\partial}{\partial x}\left[\int_0^\delta \left(u^2 - uU\right)dy\right]$$

or

$$\tau_0 = -\rho \frac{\partial}{\partial x}\left[\int_0^\delta \left(u^2 - uU\right)dy\right] = \rho \frac{\partial}{\partial x}\left[\int_0^\delta \left(uU - u^2\right)dy\right]$$

$$= \rho \frac{\partial}{\partial x}\left[\int_0^\delta U^2\left(\frac{u}{U} - \frac{u^2}{U^2}\right)dy\right] = \rho U^2 \frac{\partial}{\partial x}\left[\int_0^\delta \frac{u}{U}\left[1 - \frac{u}{U}\right]dy\right]$$

or

$$\frac{\tau_0}{\rho U^2} = \frac{\partial}{\partial x}\left[\int_0^\delta \frac{u}{U}\left[1 - \frac{u}{U}\right]dy\right] \qquad \text{...(13.10)}$$

In equation (13.10), the expression $\int_0^\delta \frac{u}{U}\left[1 - \frac{u}{U}\right]dy$ is equal to momentum thickness θ. Hence equation (13.10) is also written as

$$\frac{\tau_0}{\rho U^2} = \frac{\partial \theta}{\partial x} \qquad \text{...(13.11)}$$

Equation (13.11) is known as **Von Karman momentum integral equation** for boundary layer flows.

This is applied to :

1. Laminar boundary layers,
2. Transition boundary layers, and
3. Turbulent boundary layer flows.

For a given velocity profile in laminar zone, transition zone or turbulent zone of a boundary layer, the shear stress τ_0 is obtained from equation (13.10) or (13.11). Then drag force on a small distance Δx of the plate is obtained from equation (13.7) as

$$\Delta F_D = \tau_0 \times \Delta x \times b$$

Then total drag on the plate of length L on one side is

$$F_D = \int \Delta F_D = \int_0^L \tau_0 \times b \times dx \quad \{\text{change } \Delta x = dx\}. \qquad \text{...(13.12)}$$

13.3.1 Local Co-efficient of Drag [C_D*]. It is defined as the ratio of the shear stress τ_0 to the quantity $\frac{1}{2}\rho U^2$. It is denoted by C_D*

Hence $$C_D^* = \frac{\tau_0}{\frac{1}{2}\rho U^2}. \qquad \text{...(13.13)}$$

13.3.2 Average Co-efficient of Drag [C_D]. It is defined as the ratio of the total drag force to the quantity $\frac{1}{2}\rho A U^2$. It is also called co-efficient of drag and is denoted by C_D.

Hence $$C_D = \frac{F_D}{\frac{1}{2}\rho A U^2} \qquad \text{...(13.14)}$$

where A = Area of the surface (or plate)

U = Free-stream velocity

ρ = Mass density of fluid.

13.3.3 Boundary Conditions for the Velocity Profiles. The followings are the boundary conditions which must be satisfied by any velocity profile, whether it is in laminar boundary layer zone, or in turbulent boundary layer zone :

1. At $y = 0$, $u = 0$ and $\frac{du}{dy}$ has some finite value
2. At $y = \delta$, $u = U$
3. At $y = \delta$, $\frac{du}{dy} = 0$.

TURBULENT BOUNDARY LAYER ON A FLAT PLATE

The thickness of the boundary layer, drag force on one side of the plate and co-efficient of drag due to turbulent boundary layer on a smooth plate at zero pressure gradient are determined as in case of laminar boundary layer provided the velocity profile is known.

Blasius on the basis of experiments give the following velocity profile for turbulent boundary layer

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^n \qquad ...(13.35)$$

where $n = \frac{1}{7}$ for $R_e < 10^7$ but more than 5×10^5

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{1/7} \qquad ...(13.36)$$

Equation (13.36) is not applicable very near the boundary, where the thin laminar sub-layer of thickness δ' exists. Here velocity distribution is influenced only by viscous effects.

The value of τ_0 for flat plate is taken as $\tau_0 = 0.0225\, \rho\, U^2 \left(\frac{\mu}{\rho \delta U}\right)^{1/4}$ ...(13.37)

ANALYSIS OF TURBULENT BOUNDARY LAYER

If Reynold number is more than 5 x 10^5 and less than 10^7 the thickness of boundary layer and drag co-efficient are given as :

$$\delta = \frac{0.37x}{\left(R_{e_x}\right)^{1/5}} \text{ and } C_D = \frac{0.072}{\left(R_{e_L}\right)^{1/5}} \qquad ...(13.44)$$

where

x = Distance from the leading edge

R_{ex} = Reynold number for length x

R_{eL} = Reynold number at the end of the plate

If Reynold number is more than 107 but less than 109, Schlichting gave the empirical equation as

$$C_D = \frac{0.455}{\left(\log_{10} R_{e_L}\right)^{2.58}} \qquad \text{...(13.44}A\text{)}$$

TOTAL DRAG ON A FLAT PLATE DUE TO LAMINAR AN D TURBULENT BOUNDARY LAYER

Consider the flow over a flat plate as shown in Fig. 13.5.

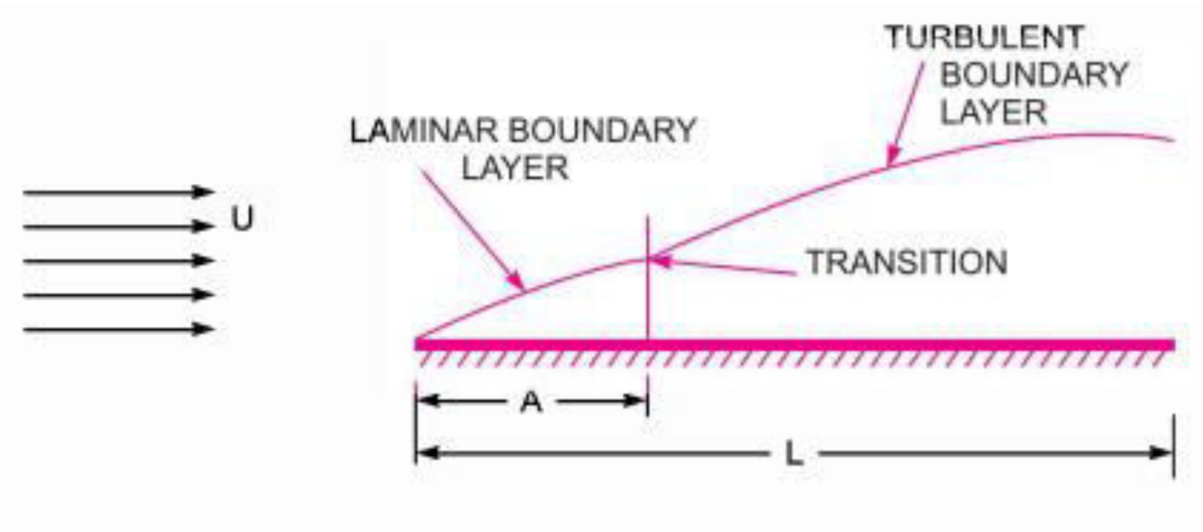


Figure 23. Drag due to laminar and turbulent boundary layer

Let

L = Total length of the plate, b = Width of plate,

A = Length of laminar boundary layer

If the length of transition region is assumed negligible, then

$L - A$ = Length of turbulent boundary layer.

We have obtained the drag on a flat plate for the laminar as well as turbulent boundary layer on the assumption that turbulent boundary layer starts from the leading edge.

This assumption is valid only when the length of laminar boundary layer is negligible. But if the length of laminar boundary layer is not negligible, then the total drag on the plate due to laminar and turbulent boundary layer is calculated as :

(1) Find the length from the leading edge upto which laminar boundary layer exists. This is done by equating $5 \times 10^5 = Ux/v$. The value of x gives the length of laminar boundary layer. Let this length is equal to A

(2) Find drag using Blasius solution for laminar boundary layer for length A.

(3) Find drag due to turbulent boundary layer for the whole length of the plate.

(4) Find the drag due to turbulent boundary layer for a length A only

Then total drag on the plate

= Drag given by (2) + Drag given by (3) - Drag given by (4)

= Drag due to laminar boundary layer for length A

+ Drag due to turbulent boundary layer for length L

- Drag due to turbulent boundary layer for length A.

SEPARATION OF BOUNDARY LAYER

When a solid body is immersed in a flowing fluid, a thin layer of fluid called the boundary layer is formed adjacent to the solid body. In this thin layer of fluid, the velocity varies from zero to free-stream velocity in the direction normal to the solid body. Along the length of the solid body, the thickness of the boundary layer increases.

The fluid layer adjacent to the solid surface has to do work against surface friction at the expense of its kinetic energy. This loss of the kinetic energy is recovered from the immediate fluid layer in contact with the layer adjacent to solid surface through momentum exchange process. Thus the velocity of the layer goes on decreasing.

Along the length of the solid body, at a certain point a stage may come when the boundary layer may not be able to keep sticking to the solid body if it cannot provide kinetic energy to overcome the resistance offered by the solid body.

In other words, the boundary layer will be separated from the surface. This phenomenon is called the boundary layer separation.

The point on the body at which the boundary layer is on the verge of separation from the surface is called point of separation.

Effect of Pressure Gradient on Boundary layer Separation

The effect of pressure gradient (dp/dx) on boundary layer separation can be explained by considering the flow over a curved surface *ABCSD* as shown in Fig. 24.

In the region *ABC* of the curved surface, the area of flow decreases and hence velocity increases. This means that flow gets accelerated in this region.

Due to the increase of the velocity, the pressure decreases in the direction of the flow and hence pressure gradient *dp/dx* is negative in this region. As long as *dp/dx* < 0, the entire boundary layer moves *dx* forward as shown in Fig. 24.

Region *CSD* of the curved surface.

The pressure is minimum at the point C. Along the region *CSD* of the curved surface, the area of flow increases and hence velocity of flow along the direction of fluid decreases. Due to decrease of velocity, the pressure increases in the direction of flow and hence pressure gradient *dp/dx* is positive or *dp/dx* > 0. Thus in the region *CSD,* the pressure gradient is *dx* *dx* positive and velocity of fluid layer along the direction of flow decreases.

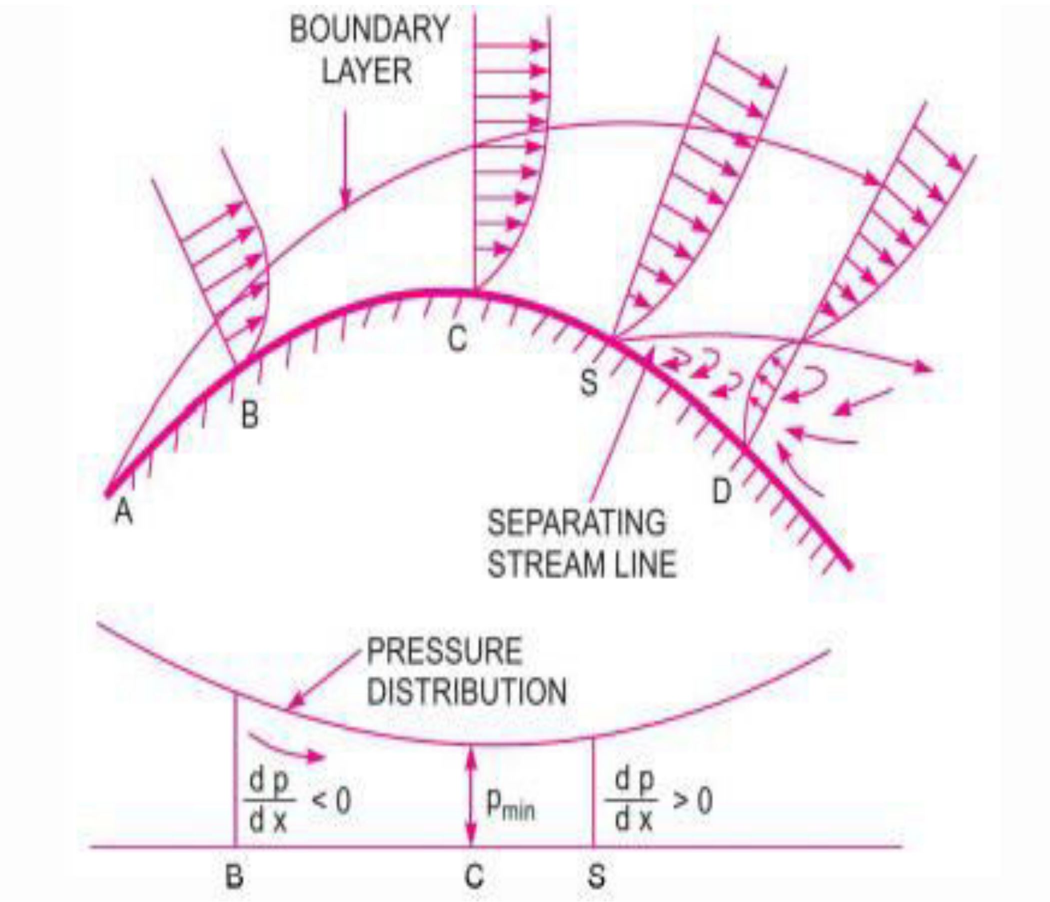


Figure 24. Effect of pressure gradient on boundary layer separation

As explained in the Art. 24, the velocity of the layer adjacent to the solid surface along the length of the solid surface goes on decreasing as the kinetic energy of the layer is used to overcome the frictional resistance of the surface.

Thus the combined effect of positive pressure gradient and surface resistance reduce the momentum of the fluid is unable to the surface. A stage comes, when the momentum of the fluid is unable to overcome the surface resistance and the boundary layer starts separating from the surface at the point S.

Downstream the point S, the flow is taking place in reverse direction and the velocity gradient becomes negative. Thus the positive pressure gradient helps in separating the boundary layer.

13.7.2 Location of Separation Point. The separation point S is determined from the condition,

$$\left(\frac{\partial u}{\partial y}\right)_{y=0} = 0 \qquad \text{...(13.46)}$$

For a given velocity profile, it can be determined whether the boundary layer has separated, or on the verge of separation or will not separate from the following conditions :

1. If $\left(\frac{\partial u}{\partial y}\right)_{y=0}$ is negative ... the flow has separated.
2. If $\left(\frac{\partial u}{\partial y}\right)_{y=0} = 0$ ... the flow is on the verge of separation.
3. If $\left(\frac{\partial u}{\partial y}\right)_{y=0}$ is positive ... the flow will not separate or flow will remain attached with the surface.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – IV – PUMPS AND TURBINES – SAEA1304

I.INTRODUCTION TO IMPACT OF JETS

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

In this presentation, the following cases of the impact of jet *i.e.,* the force exerted by the jet on a plate, will be considered :

1. Force exerted by the jet on a stationary plate when

(a) Plate is vertical to the jet,

(a) Plate is inclined to the jet, and

(b) Plate is curved.

2. Force exerted by the jet on a moving plate, when

(a) Plate is vertical to the jet,

(*b*) Plate is inclined to the jet, and

(c) Plate is curved

FORCE EXERTED BY THE JET ON A STATIONARY VERTICAL PLATE

Consider a jet of water coming out from the nozzle, strikes a flat vertical plate as shown in Fig. 17.1 Let

V = velocity of the jet,

d = diameter of the jet,

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

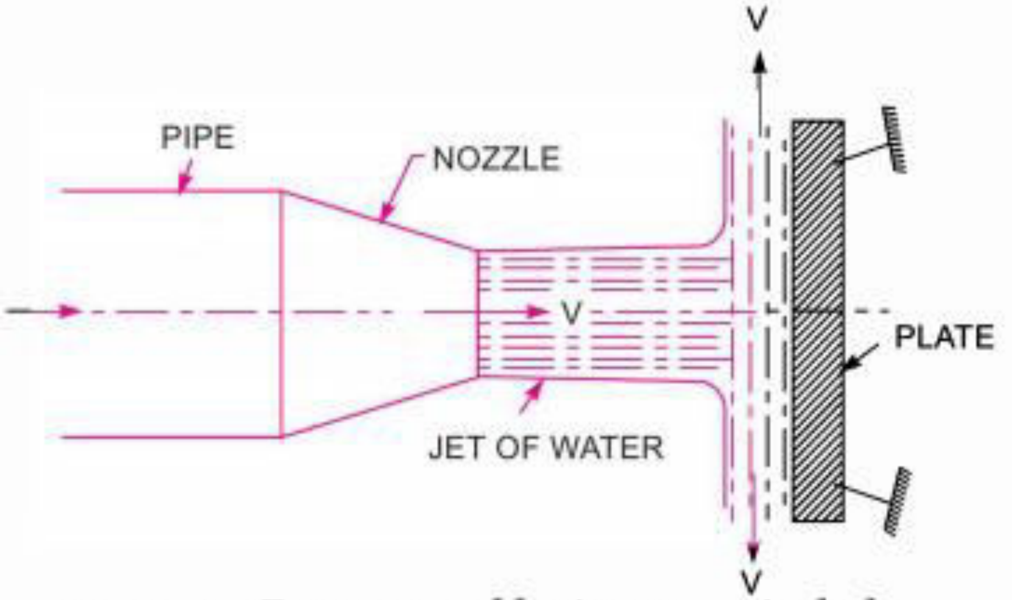


Fig. 17.1 *Force exerted by jet on vertical plate.*

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

The force exerted by the jet on the plate in the direction of jet,

$$
\begin{aligned}
F_x &= \text{Rate of change of momentum in the direction of force} \\
&= \frac{\text{Initial momentum} - \text{Final momentum}}{\text{Time}} \\
&= \frac{(\text{Mass} \times \text{Initial velocity} - \text{Mass} \times \text{Final velocity})}{\text{Time}} \\
&= \frac{\text{Mass}}{\text{Time}} [\text{Initial velocity} - \text{Final velocity}] \\
&= (\text{Mass/sec}) \times (\text{velocity of jet before striking} - \text{velocity of jet after striking}) \\
&= \rho a V[V - 0] \qquad (\because \quad \text{mass/sec} = \rho \times a\ V) \\
&= \rho a V^2 \qquad \ldots(17.1)
\end{aligned}
$$

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

Note. In equation (17. 1), if the value of density (p) is taken in S.I. units *(i.e.,* kg/m^3), the force *(F_x)* will be in Newton (N). The value of p for water in S.I. units is equal to 1000 kg/m^3.

FORCE EXERTED BY A JET ON STATIONARY INCLINED FLAT PLATE

- Let a jet of water, coming out from the nozzle, strikes an inclined flat plate as shown in Fig. 17.2.
- Let

V = Velocity of jet in the direction of *x,*

θ = Angle between the jet and plate,

a = Area of cross-section of the jet.

Then mass of water per sec striking the plate = ρ x *aV*.

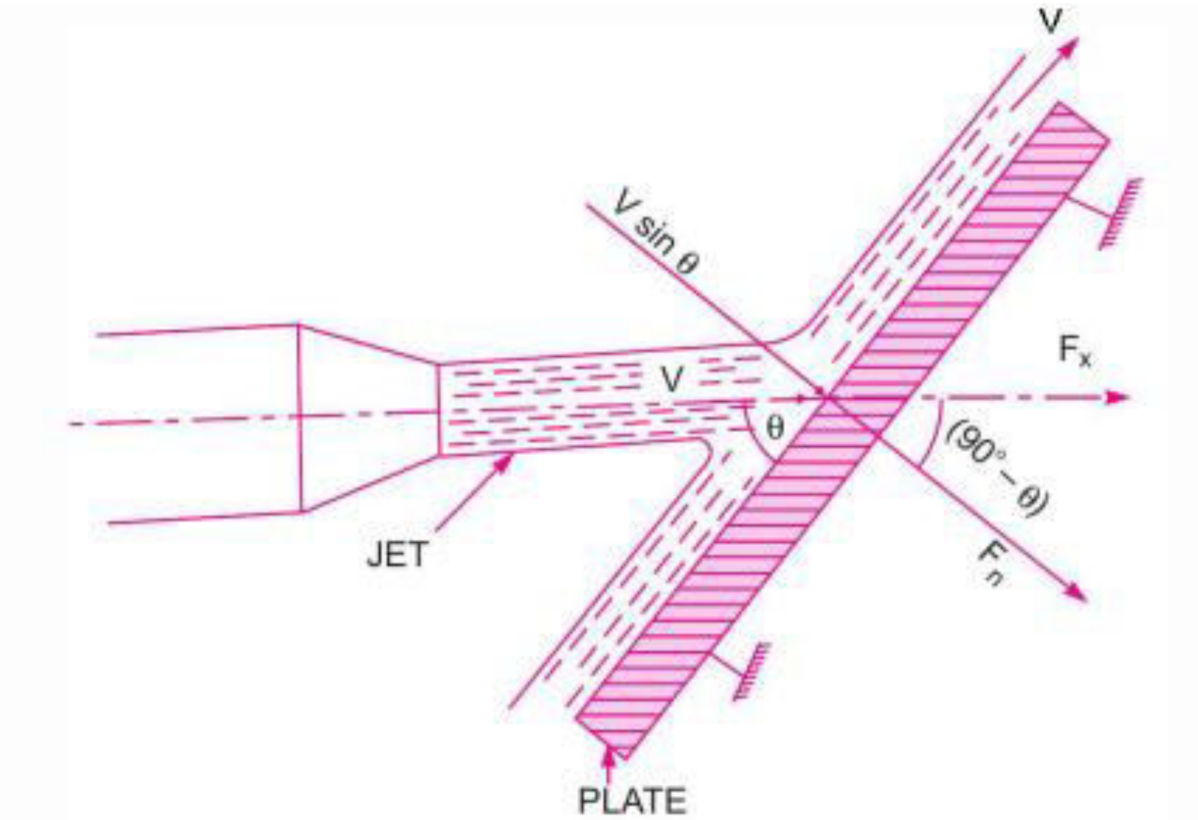


Fig. 17.2 ***Jet striking stationary inclined plate.***

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

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

Then F_n = mass of jet striking per second × [Initial velocity of jet before striking in the direction of n – Final velocity of jet after striking in the direction of n]

$$= \rho aV [V \sin\theta - 0] = \rho aV^2 \sin\theta \qquad ...(17.2)$$

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

F_x = component of F_n in the direction of flow

$$= F_n \cos(90° - \theta) = F_n \sin\theta = \rho A V^2 \sin\theta \times \sin\theta \quad (\therefore F_n = \rho aV^2 \sin\theta)$$

$$= \rho AV^2 \sin^2\theta \qquad ...(17.3)$$

And, F_y = component of F_n, perpendicular to flow

$$= F_n \sin(90° - \theta) = F_n \cos\theta = \rho AV^2 \sin\theta \cos\theta. \qquad ...(17.4)$$

FORCE EXERTED BY A JET ON STATIONARY CURVED PLATE

Jet strikes the curved plate at the Centre. Let a jet of water strikes a fixed curved plate at the Centre as shown in Fig. 17.3. The jet after striking the plate, comes out with the same velocity if the plate is smooth and there is no loss of energy due to impact of the jet, in the tangential direction of the curved plate.

The velocity at outlet of the plate can be resolved into two components, one in the direction of jet and other perpendicular to the direction of the jet.

Component of velocity in the direction of jet = - $V \cos\theta$.

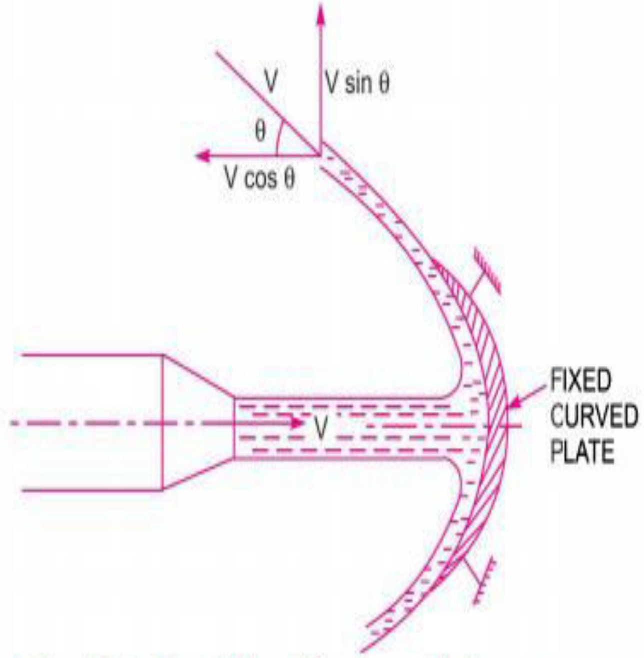


Fig. 17.3 *Jet striking a fixed curved plate at centre.*

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

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

Force exerted by the jet in the direction of jet,

$$F_x = \text{Mass per sec} \times [V_{1x} - V_{2x}]$$

where V_{1x} = Initial velocity in the direction of jet = V

V_{2x} = Final velocity in the direction of jet = $-V \cos \theta$

$$\therefore \quad F_x = \rho a V[V - (-V \cos \theta)] = \rho a V[V + V \cos \theta]$$

$$= \rho a V^2[1 + \cos \theta] \qquad \ldots(17.5)$$

Similarly, $F_y = \text{Mass per sec} \times [V_{1y} - V_{2y}]$

where V_{1y} = Initial velocity in the direction of $y = 0$

V_{2y} = Final velocity in the direction of $y = V \sin \theta$

$$\therefore \quad F_y = \rho a V[0 - V \sin \theta] = -\rho a V^2 \sin \theta \qquad \ldots(17.6)$$

–ve sign means that force is acting in the downward direction. In this case the angle of deflection of the jet $= (180° - \theta)$ ...[17.6 (*A*)]

JET STRIKES THE CURVED PLATE AT ONE END TANGENTIALLY WHEN THE PLATE IS SYMMETRICAL

Let the jet strikes the curved fixed plate at one end tangentially as shown in Fig. 17.4. Let the curved plate is symmetrical about x-axis.

Then the angle made by the tangents at the two ends of the plate will be same.

Let

V = Velocity of jet of water,

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

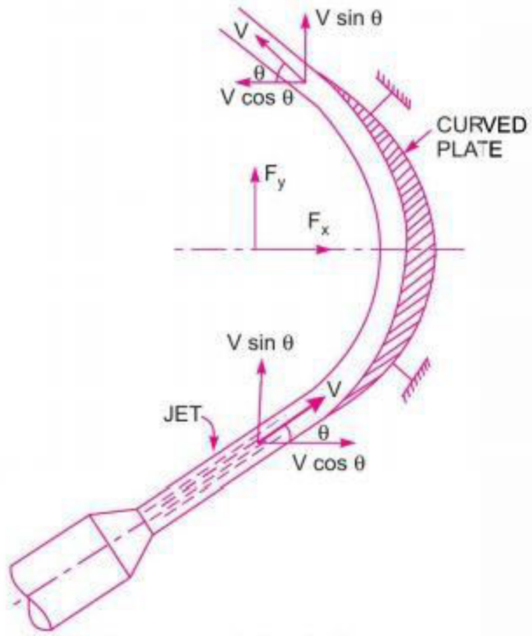


Fig. 17.4 *Jet striking curved fixed plate at one end.*

If the plate is smooth and loss of energy due to impact is zero, then the velocity of water at the outlet tip of the curved plate will be equal to *V.*

The forces exerted by the jet of water in the directions of *x* and *y* are

$$F_x = (\text{mass/sec}) \times [V_{1x} - V_{2x}]$$
$$= \rho a V[V \cos\theta - (-V\cos\theta)]$$
$$= \rho a V[V\cos\theta + V\cos\theta]$$
$$= 2\rho a V^2 \cos\theta \qquad \ldots(17.7)$$
$$F_y = \rho a V[V_{1y} - V_{2y}]$$
$$= \rho a V[V\sin\theta - V\sin\theta] = 0$$

JET STRIKES THE CURVED PLATE AT ONE END TANGENTIALLY WHEN THE PLATE IS UNSYMMETRICAL

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

Let θ = angle made by tangent at inlet tip with x-axis,

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

The two components of the velocity at inlet are

$$V_{1x} = V\cos\theta \text{ and } V_{1y} = V\sin\theta$$

The two components of the velocity at outlet are

$$V_{2x} = -V\cos\phi \text{ and } V_{2y} = V\sin\phi$$

$\therefore$ The forces exerted by the jet of water in the directions of x and y are

$$F_x = \rho aV[V_{1x} - V_{2x}] = \rho aV[V\cos\theta - (-V\cos\phi)]$$

$$= \rho aV[V\cos\theta + V\cos\phi] = \rho aV^2[\cos\theta + \cos\phi] \quad \text{...(17.8)}$$

$$F_y = \rho aV[V_{1y} - V_{2y}] = \rho aV[V\sin\theta - V\sin\phi]$$

$$= \rho aV^2[\sin\theta - \sin\phi]. \quad \text{...(17.9)}$$

FORCE EXERTED BY A JET ON A HINGED PLATE

Consider a jet of water striking a vertical plate at the centre which is hinged at *O*. Due to the force exerted by the jet on the plate, the plate will swing through some angle about the hinge as shown in Fig. 17.6

Let

x = distance of the centre of jet from hinge *0,*

θ = angle of swing about hinge,

W = weight of plate acting at C.G. of the plate.

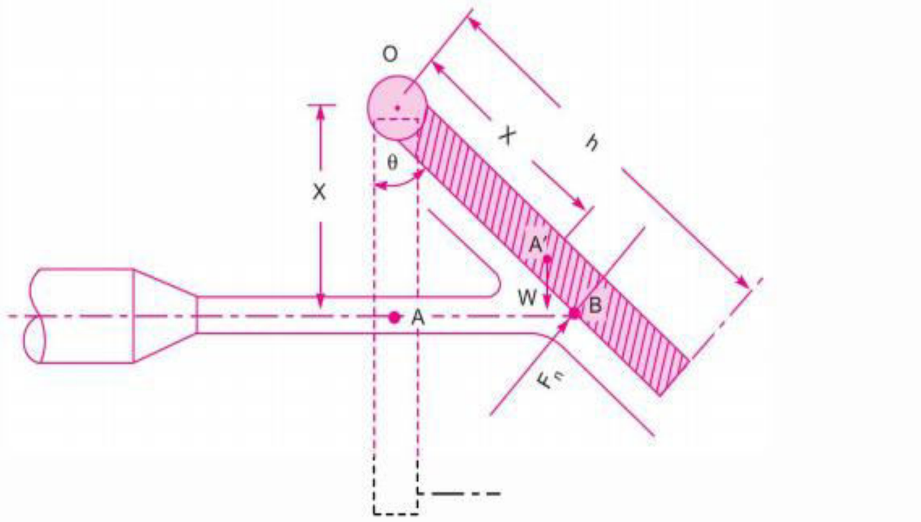


Fig. 17.6 *Force on a hinged plate.*

The dotted lines show the position of the plate, before the jet strikes the plate. The point A on the plate will be at A' after the jet strikes the plate. The distance $OA = OA' = x$. Let the weight of the plate is acting at A'.When the plate is in equilibrium after the jet strikes the plate, the moment of all the forces about the hinge must be zero. Two forces are acting on the plate. They are :

1. Force due to jet of water, normal to the plate,

$$F_n = \rho a V^2 \sin \theta'$$

where θ' = Angle between jet and plate = $(90° - \theta)$

2. Weight of the plate, W

Moment of force F_n about hinge $= F_n \times OB = \rho a V^2 \sin(90° - \theta) \times OB = \rho a V^2 \cos\theta \times OB$

$$= \rho a V^2 \cos\theta \times \frac{OA}{\cos\theta} = \rho a V^2 \times OA = \rho a V^2 \times x$$

Moment of weight W about hinge $= W \times OA' \sin\theta = W \times x \times \sin\theta$

For equilibrium of the plate, $\rho a V^2 \times x = W \times x \times \sin\theta$

$$\therefore \qquad \sin\theta = \frac{\rho a V^2}{W} \qquad \ldots(17.10)$$

From equation (17.10), the angle of swing of the plate about hinge can be calculated.

FORCE EXERTED BY A JET ON MOVING PLATES

The following cases of the moving plates will be considered :

1. Flat vertical plate moving in the direction of the jet and away from the jet,
2. Inclined plate moving in the direction of the jet, and

3. Curved plate moving in the direction of the jet or in the horizontal direction.

FORCE ON FLAT VERTICAL PLATE MOVING IN THE DIRECTION OF JET

Fig. 17.10 shows a jet of water striking a flat vertical plate moving with a uniform velocity away from the jet.

Let

V = Velocity of the jet (absolute),

a = Area of cross-section of the jet,

u = Velocity of the flat plate.

In this case, the jet does not strike the plate with a velocity V, but it strikes with a relative velocity, which is equal to the absolute velocity of jet of water min us the velocity of the plate.

Hence relative velocity of the jet with respect to plate = (V - u)

Mass of water striking the plate per sec

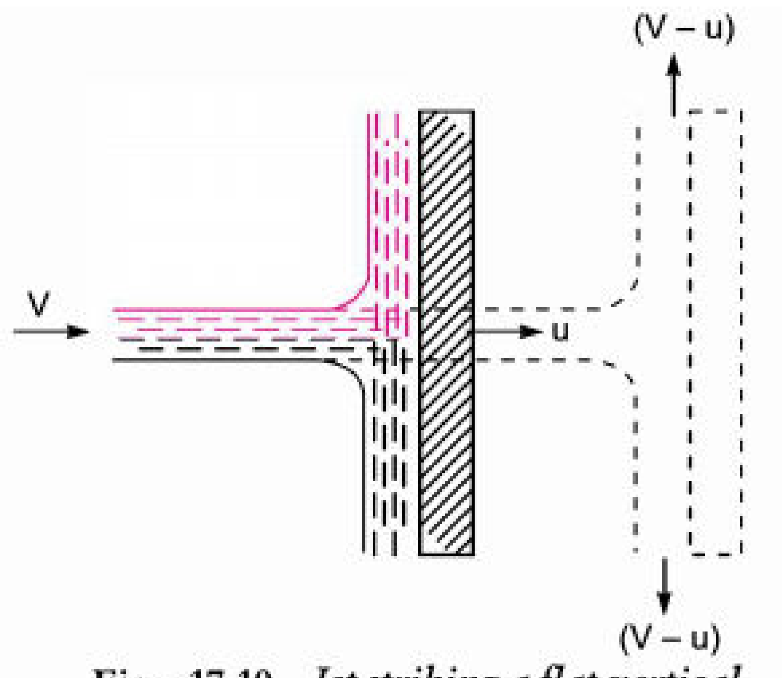


Fig. 17.10 *Jet striking a flat vertical moving plate.*

= p x Area of jet x Velocity with

which jet strikes the plate

= *pa* x [V - u]

$\therefore$ Force exerted by the jet on the moving plate in the direction of the jet,

$$F_x = \text{Mass of water striking per sec} \times [\text{Initial velocity with which water strikes} - \text{Final velocity}]$$

$$= \rho a(V-u)\,[(V-u)-0] \qquad (\because \text{ Final velocity in the direction of jet is zero})$$

$$= \rho a(V-u)^2 \qquad \text{...(17.11)}$$

In this case, the work will be done by the jet on the plate, as plate is moving. For the stationary plates, the work done is zero.

$\therefore$ Work done per second by the jet on the plate

$$= \text{Force} \times \frac{\text{Distance in the direction of force.}}{\text{Time}}$$

$$= F_x \times u = \rho a(V-u)^2 \times u \qquad \text{...(17.12)}$$

In equation (17.12), if the value of ρ for water is taken in S.I. units (*i.e.*, 1000 kg/m^3), the work done will be in N m/s. The term $\frac{\text{'Nm'}}{\text{s}}$ is equal to W (watt).

FORCE ON THE INCLINED PLATE MOVING IN THE DIRECTION OF THE JET

Let a jet of water strikes an inclined plate, which is moving with a uniform velocity in the direction of the jet as shown in Fig. 17.11.

Let

V = Absolute velocity of jet of water,

u = Velocity of the plate in the direction of jet,

a = Cross-sectional area of jet, and 0 = Angle between jet and plate.

Relative velocity of jet of water = (V - *u)*

The velocity with which jet strikes = (V - *u)*

Mass of water striking per second

= p x *a x* (V - *u)*

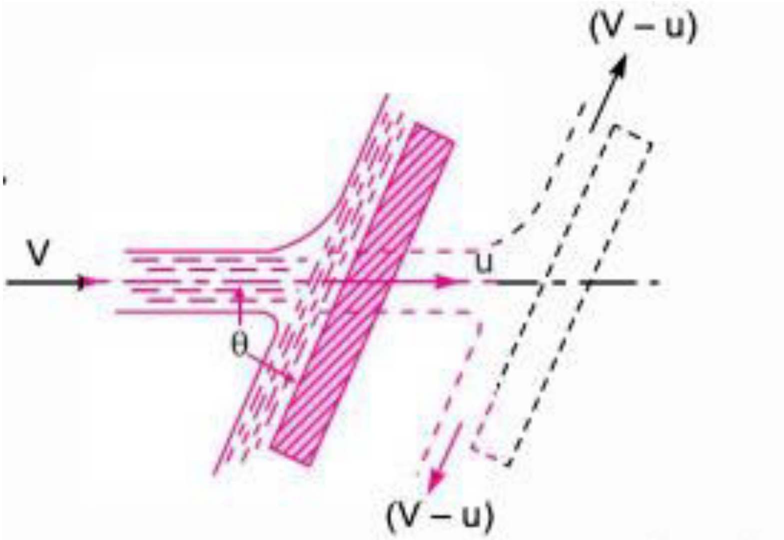


Fig. 17.11 *Jet striking an inclined moving plate.*

If the plate is smooth and loss of energy due to impact of the jet is assumed zero, the jet of water will leave the inclined plate with a velocity equal to $(V - u)$.

The force exerted by the jet of water on the plate in the direction normal to the plate is given as

$$F_n = \text{Mass striking per second} \times [\text{Initial velocity in the normal direction with which jet strikes} - \text{Final velocity}]$$

$$= \rho a (V - u) [(V - u) \sin\theta - 0] = \rho a (V - u)^2 \sin\theta \qquad \text{...(17.13)}$$

This normal force F_n is resolved into two components namely F_x and F_y in the direction of the jet and perpendicular to the direction of the jet respectively.

$$\therefore \quad F_x = F_n \sin\theta = \rho a (V - u)^2 \sin^2\theta \qquad \text{...(17.14)}$$

$$F_y = F_n \cos\theta = \rho a (V - u)^2 \sin\theta \cos\theta \qquad \text{...(17.15)}$$

$\therefore$ Work done per second by the jet on the plate

$$= F_x \times \text{Distance per second in the direction of } x$$

$$= F_x \times u = \rho a (V - u)^2 \sin^2\theta \times u = \rho a (V - u)^2 u \sin^2\theta \text{ N m/s.} \qquad \text{...(17.16)}$$

FORCE ON THE CURVED PLATE WHEN THE PLATE IS MOVING IN THE DIRECTION OF JET

Let a jet of water strikes a curved plate at the centre of the plate which is moving with a uniform velocity in the direction of the jet as shown in Fig. 17.12.

Let

V = Absolute velocity of jet,

a = Area of jet,

u = Velocity of the plate in the direction of the jet.

Relative velocity of the jet of water or the velocity with which jet strikes the curved plate = (V - u).

If plate is smooth and the loss of energy due to impact of jet is zero, then the velocity with which the jet will be leaving the curved vane = (V - u).

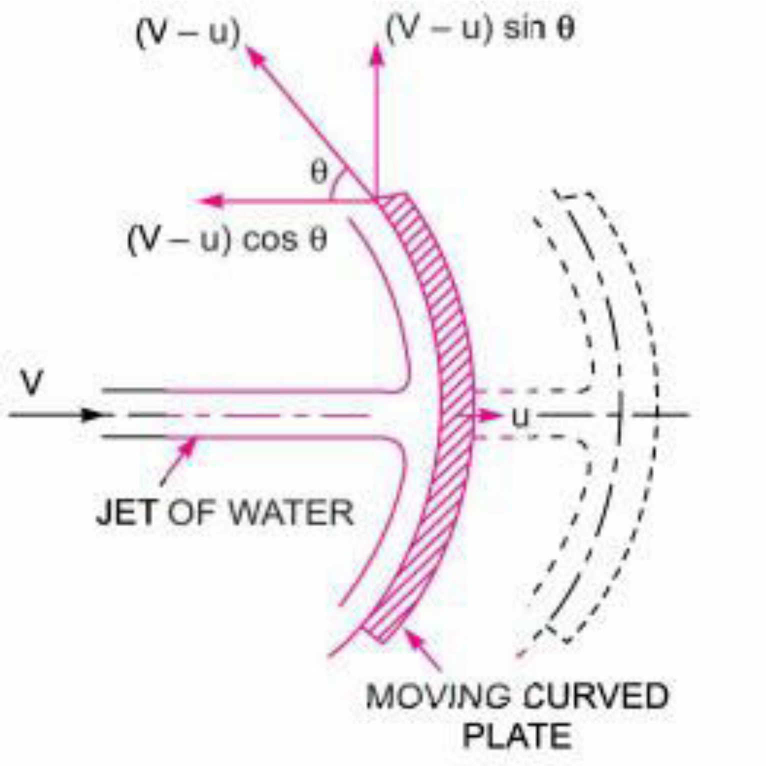


Fig. 17.12 ***Jet striking a curved moving plate.***

This velocity can be resolved into two components, one in the direction of the jet and other perpendicular to the direction of the jet.

Component of the velocity in the direction of jet = - (V - *u)* cos θ

(-ve sign is taken as at the outlet, the component is in the opposite direction of the jet).

Component of the velocity in the direction perpendicular to the direction of the jet = (V - *u)* sin θ .

Mass of the water striking the plate = $\rho \times a \times$ Velocity with which jet strikes the plate

$$= \rho a(V - u)$$

$\therefore$ Force exerted by the jet of water on the curved plate in the direction of the jet,

$$F_x = \text{Mass striking per sec} \times [\text{Initial velocity with which jet strikes the plate in the direction of jet} - \text{Final velocity}]$$

$$= \rho a(V - u)\,[(V - u) - (-(V - u)\cos\theta)]$$

$$= \rho a(V - u)\,[(V - u) + (V - u)\cos\theta]$$

$$= \rho a(V - u)^2\,[1 + \cos\theta] \qquad \text{...(17.17)}$$

Work done by the jet on the plate per second

$$= F_x \times \text{Distance travelled per second in the direction of } x$$

$$= F_x \times u = \rho a(V - u)^2\,[1 + \cos\theta] \times u$$

$$= \rho a(V - u)^2 \times u\,[1 + \cos\theta] \qquad \text{...(17.18)}$$

FORCE EXERTED BY A JET OF WATER ON AN UNSYMMETRICAL MOVING CURVED PLATE WHEN JET STRIKES TANGENTIALLY AT ONE OF THE TIPS.

Fig. 17.15 shows a jet of water striking a moving curved plate (also called vane) tangentially, at one of its tips. As the jet strikes tangentially, the loss of energy due to impact of the jet will be zero.

In this case as plate is moving, the velocity with which jet of water strikes is equal to the relative velocity of the jet with respect to the plate.

Also as the plate is moving in different direction of the jet, the relative velocity at inlet will be equal to the vector difference of the velocity of jet and velocity of the plate at inlet.

Let

V_1= Velocity of the jet at inlet.

u_1 = Velocity of the plate (vane) at inlet.

V_{r1} = Relative velocity of jet and plate at inlet.

α = Angle between the direction of the jet and direction of motion of the plate, also called guide blade angle.

θ = Angle made by the relative velocity (V_{r2}) with the direction of motion at inlet also called vane angle at inlet.

V_{w1} and V_{f1} = The components of the velocity of the jet V_1 , in the direction of motion and perpendicular to the direction of motion of the vane respectively.

V_{w1} = It is also known as velocity of whirl at inlet.

V_{f1} = It is also known as velocity of flow at inlet.

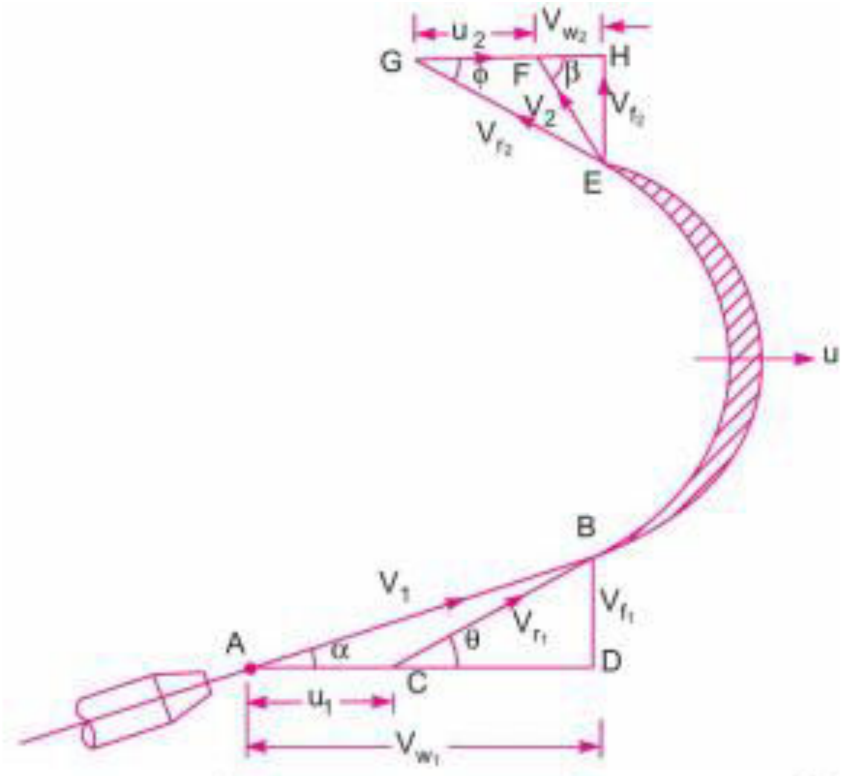


Fig. 17.15 *Jet striking a moving curved vane at one of the tips.*

V_2 = Velocity of the jet, leaving the vane or velocity of jet at outlet of the vane.

u_2 = Velocity of the vane at outlet.

V_{r2} = Relative velocity of the jet with respect to the vane at outlet.

β = Angle made by the velocity V_2 with the direction of motion of the vane at outlet.

φ = Angle made by the relative velocity V_{r2} with the direction of motion of the vane at outlet and also called vane angle at outlet.

V_{w1} and V_{f1} = Components of the velocity V_2,in the direction of motion of vane and perpendicular to the direction of motion of vane at outlet.

V_{w2} = It is also called the velocity of whirl at outlet.

V_{f2} = Velocity of flow at outlet.

The triangles *ABD* and *EGH* are called the velocity triangles at inlet and outlet. These velocity triangles are drawn as given below :

Velocity Triangle at Inlet.

Take any point *A* and draw a line $AB = V_1$ in magnitude and direction which means line *AB* makes an angle a with the horizontal line *AD*. Next draw a line $AC = u_1$ in magnitude. Join *C* to *B*.

Then *CB* represents the relative velocity of the jet at inlet. If the loss of energy at inlet due to impact is zero, then *CB* must be in the tangential direction to the vane at inlet. From *B* draw a vertical line *BD* in the downward direction to meet the horizontal line *AC* produced at *D*.

Then *BD* = Represents the velocity of flow at inlet = V_{f1}

AD = Represents the velocity of whirl at inlet = V_{w1}

ιBCD = Vane angle at inlet = θ.

Velocity Triangle at Outlet.

If the vane surface is assumed to be very smooth, the loss of energy due to friction will be zero. The water will be gliding over the surface of the vane with a relative velocity equal to V_{r1} and will come out of the vane with a relative velocity V_{r2}.

This means that the relative velocity at outlet $V_{r1} = V_{r2}$. And also the relative velocity at outlet should be in tangential direction to the vane at outlet.

Draw *EG* in the tangential direction of the vane at outlet and cut *EG* = V_{r2}• From G, draw a line *GF* in the direction of vane at outlet and equal to u_2, the velocity of the vane at outlet. Join *EF*.

Then *EF* represents the absolute velocity of the jet at outlet in magnitude and direction. From *E* draw a vertical line *EH* to meet the line *GF* produced at *H*. Then

$$EH = \text{Velocity of flow at outlet} = V_{f_2}$$
$$FH = \text{Velocity of whirl at outlet} = V_{w_2}$$
$$\phi = \text{Angle of vane at outlet.}$$
$$\beta = \text{Angle made by } V_2 \text{ with the direction of motion of vane at outlet.}$$

If the vane is smooth and is having velocity in the direction of motion at inlet and outlet equal then we have

$$u_1 = u_2 = u = \text{Velocity of vane in the direction of motion and}$$
$$V_{r_1} = V_{r_2}.$$

Now mass of water striking vane per sec = $\rho a V_{r_1}$ ...(*i*)

where a = Area of jet of water, V_{r_1} = Relative velocity at inlet.

∴ Force exerted by the jet in the direction of motion

F_x = Mass of water striking per sec × [Initial velocity with which jet strikes in the direction of motion – Final velocity of jet in the direction of motion] ...(*ii*)

But initial velocity with which jet strikes the vane = V_{r_1}

The component of this velocity in the direction of motion

$$= V_{r_1} \cos\theta = (V_{w_1} - u_1) \quad \text{(See Fig. 17.15)}$$

Similarly, the component of the relative velocity at outlet in the direction of motion = $-V_{r_2}\cos\phi$

$$= -[u_2 + V_{w_2}]$$

–ve sign is taken as the component of V_{r_2} in the direction of motion is in the opposite direction.

Substituting the equation (*i*) and all above values of the velocities in equation (*ii*), we get

$$F_x = \rho a V_{r_1}\,[(V_{w_1} - u_1) - \{-(u_2 + V_{w_2})\}] = \rho a V_{r_1}\,[V_{w_1} - u_1 + u_2 + V_{w_2}]$$
$$= \rho a V_{r_1}\,[V_{w_1} + V_{w_2}] \quad (\because\ u_1 = u_2) \quad \text{...(iii)}$$

Equation (*iii*) is true only when angle β shown in Fig. 17.15 is an acute angle. If $\beta = 90°$, the $V_{w_2} = 0$, then equation (*iii*) becomes as,

$$F_x = \rho a V_{r_1}\,[V_{w_1}]$$

If β is an obtuse angle, the expression for F_x will become

$$F_x = \rho a V_{r_1}\,[V_{w_1} - V_{w_2}]$$

Thus in general, F_x is written as $F_x = \rho a V_{r_1}\,[V_{w_1} \pm V_{w_2}]$ ...(17.19)

Work done per second on the vane by the jet

$$= \text{Force} \times \text{Distance per second in the direction of force}$$
$$= F_x \times u = \rho a V_{r_1}\,[V_{w_1} \pm V_{w_2}] \times u \quad \text{...(17.20)}$$

∴ Work done per second per unit weight of fluid striking per second

$$= \frac{\rho a V_{r_1}\left[V_{w_1} \pm V_{w_2}\right] \times u}{\text{Weight of fluid striking/s}}\,\frac{\text{Nm/s}}{\text{N/s}} = \frac{\rho a V_{r_1}\left[V_{w_1} \pm V_{w_2}\right] \times u}{g \times \rho a V_{r_1}} = \text{Nm/N}$$

$$= \frac{1}{g}\,[V_{w_1} \pm V_{w_2}] \times u \text{ Nm/N} \quad \text{...(17.21)}$$

Work done/sec per unit mass of fluid striking per second

$$= \frac{\rho a V_{r_1}\left[V_{w_1} \pm V_{w_2}\right] \times u}{\text{Mass of fluid striking / s}} \frac{\text{Nm / s}}{\text{kg / s}} = \frac{\rho a V_{r_1}\left[V_{w_1} \pm V_{w_2}\right] \times u}{\rho a V_{r_1}} \text{ Nm/kg}$$

$$= (V_{w_1} \pm V_{w_2}) \times u \text{ Nm/kg} \quad \text{...[17.21(A)]}$$

Note. Equation (17.21) gives the work done per unit weight whereas equation [17.21(*A*)] gives the work done per unit mass.

3. Efficiency of Jet. The work done by the jet on the vane given by equation (17.20), is the output of the jet whereas the initial kinetic energy of the jet is the input. Hence, the efficiency (η) of the jet is expressed as

$$\eta = \frac{\text{Output}}{\text{Input}} = \frac{\text{Work done per second on the vane}}{\text{Initial K. E. per second of the jet}} = \frac{\rho a V_{r_1}(V_{w_1} \pm V_{w_2}) \times u}{\frac{1}{2} m V_1^2}$$

where m = mass of the fluid per second in the jet = $\rho a V_1$

V_1 = initial velocity of jet

$$\therefore \quad \eta = \frac{\rho a V_{r_1}\left[V_{w_1} \pm V_{w_2}\right] \times u}{\frac{1}{2}(\rho a V_1) \times V_1^2}. \quad \text{...[17.21(B)]}$$

FORCE EXERTED BY A JET OF WATER ON A SERIES OF VANES

The force exerted by a jet of water on a *single* moving plate (which may be flat or curved) is not practically feasible.

This case is only a theoretical one. In actual practice, a large number of plates are mounted on the circumference of a wheel at a fixed distance apart as shown in Fig. 17.22.

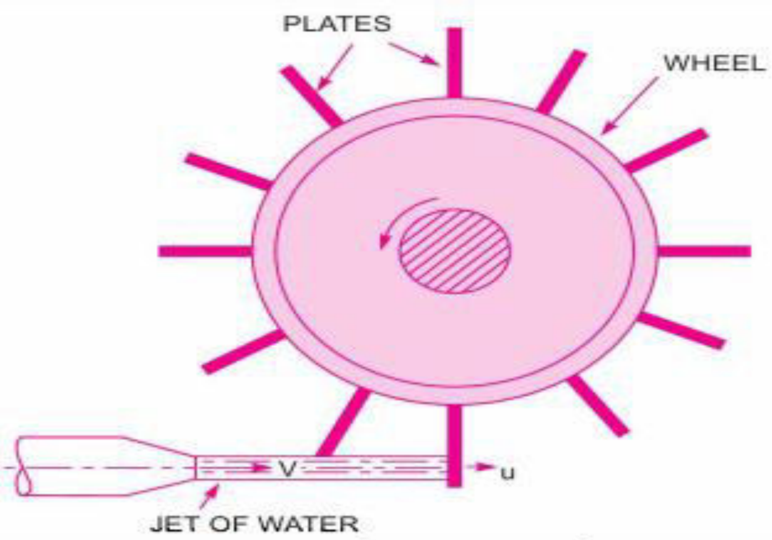


Fig. 17.22 *Jet striking a series of vanes.*

Jet strikes a plate and due to the force exerted by the jet on the plate, the wheel starts moving and the 2^{nd} plate mounted on the wheel appears before the jet, which again exerts the force on the 2^{nd} plate.

- Thus each plate appears successively before the jet and the jet exerts force on each plate. The wheel starts moving at a constant speed. Let

V = Velocity of jet,

d = Diameter of jet,

a = Cross-sectional area of jet,

$= \frac{\pi}{4} d^2$

u = Velocity of vane.

In this case the mass of water coming out from the nozzle per second is always in contact with the plates, when all the plates are considered. Hence mass of water per second striking the series of plates = $\rho a V$.

Also the jet strikes the plate with a velocity = (V - u)

After striking, the jet moves tangential to the plate and hence the velocity component in the direction of motion of plate is equal to zero.

$\therefore$ The force exerted by the jet in the direction of motion of plate,

$$F_x = \text{Mass per second [Initial velocity – Final velocity]}$$

$$= \rho a V[(V - u) - 0] = \rho a V[V - u] \quad \text{...(17.22)}$$

Work done by the jet on the series of plates per second

$$= \text{Force} \times \text{Distance per second in the direction of force}$$

$$= F_x \times u = \rho a V[V - u] \times u$$

Kinetic energy of the jet per second

$$= \frac{1}{2} mV^2 = \frac{1}{2} (\rho a V) \times V^2 = \frac{1}{2} \rho a V^3$$

$\therefore$ Efficiency,

$$\eta = \frac{\text{Work done per second}}{\text{Kinetic energy per second}} = \frac{\rho a V[V - u] \times u}{\frac{1}{2} \rho a V^3} = \frac{2u[V - u]}{V^2} \quad \text{...(17.23)}$$

Condition for Maximum Efficiency. Equation (17.23) gives the value of the efficiency of the wheel. For a given jet velocity V, the efficiency will be maximum when

$$\frac{d\eta}{du} = 0 \quad \text{or} \quad \frac{d}{du}\left[\frac{2u(V-u)}{V^2}\right] = 0 \quad \text{or} \quad \frac{d}{du}\left[\frac{2uV - 2u^2}{V^2}\right] = 0$$

or
$$\frac{2V - 2\times 2u}{V^2} = 0 \quad \text{or} \quad 2V - 4u = 0 \quad \text{or} \quad V = \frac{4u}{2} = 2u \text{ or } u = \frac{V}{2}. \quad \text{...(17.24)}$$

Maximum Efficiency. Substituting the value of $V = 2u$ in equation (17.23), we get the maximum efficiency as

$$\eta_{max} = \frac{2u\,[2u - u]}{(2u)^2} = \frac{2u \times u}{2u \times 2u} = \frac{1}{2} = 0.5 \text{ or } 50\%. \quad \text{...(17.25)}$$

Force Exerted on a Series of Radial Curved Vanes.

For a radial curved vane, the radius of the vane at inlet and outlet is different and hence the tangential velocities of the radial vane at inlet and outlet will not be equal. Consider a series of radial curved vanes mounted on a wheel as shown in Fig. 17.23.

The jet of water strikes the vanes and the wheel starts rotating at a constant angular speed.

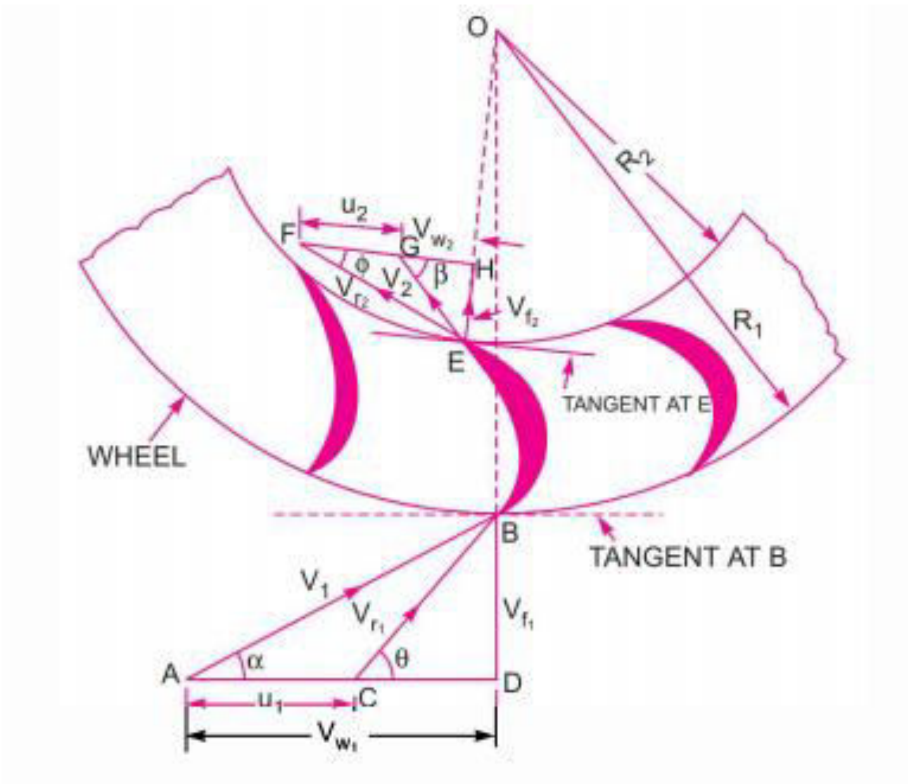


Fig. 17.23 *Series of radial curved vanes mounted on a wheel.*

Let R_1 = Radius of wheel at inlet of the vane,
R_2 = Radius of the wheel at the outlet of the vane,
ω = Angular speed of the wheel.

Then $u_1 = \omega R_1$ and $u_2 = \omega R_2$

The velocity triangles at inlet and outlet are drawn as shown in Fig. 17.23.

The mass of water striking per second for a series of vanes

= Mass of water coming out from nozzle per second

= $\rho a V_1$, where a = Area of jet and V_1 = Velocity of jet.

Momentum of water striking the vanes in the tangential direction per sec at inlet

= Mass of water per second × Component of V_1 in the tangential direction

= $\rho a V_1 \times V_{w_1}$ ($\because$ Component of V_1 in tangential direction = $V_1 \cos \alpha = V_{w_1}$)

Similarly, momentum of water at outlet per sec

= $\rho a V_1$ × Component of V_2 in the tangential direction

= $\rho a V_1 \times (-V_2 \cos \beta) = -\rho a V_1 \times V_{w_2}$ ($\because$ $V_2 \cos \beta = V_{w_2}$)

–ve sign is taken as the velocity V_2 at outlet is in opposite direction.

Now, angular momentum per second at inlet

= Momentum at inlet × Radius at inlet

= $\rho a V_1 \times V_{w_1} \times R_1$

Angular momentum per second at outlet

= Momentum of outlet × Radius at outlet

= $-\rho a V_1 \times V_{w_2} \times R_2$

Torque exerted by the water on the wheel,

T = Rate of change of angular momentum

= [Initial angular momentum per second – Final angular momentum per second]

$= \rho a V_1 \times V_{w_1} \times R_1 - (-\rho a V_1 \times V_{w_2} \times R_2) = \rho a V_1 [V_{w_1} \times R_1 + V_{w_2} R_2]$

Work done per second on the wheel

= Torque × Angular velocity = $T \times \omega$

$= \rho a V_1 [V_{w_1} \times R_1 + V_{w_2} R_2] \times \omega = \rho a V_1 [V_{w_1} \times R_1 \times \omega + V_{w_2} R_2 \times \omega]$

$= \rho a V_1 [V_{w_1} u_1 + V_{w_2} \times u_2]$ ($\because$ $u_1 = \omega R_1$ and $u_2 = \omega R_2$)

If the angle β in Fig. 17.23 is an obtuse angle then work done per second will be given as

$= \rho a V_1 [V_{w_1} u_1 - V_{w_2} u_2]$

$\therefore$ The general expression for the work done per second on the wheel

$= \rho a V_1 [V_{w_1} u_1 \pm V_{w_2} u_2]$ ...(17.26)

If the discharge is radial at outlet, then $\beta = 90°$ and work done becomes as

$= \rho a V_1 [V_{w_1} u_1]$ ($\because$ $V_{w_2} = 0$) ...(17.27)

Efficiency of the Radial Curved Vane

The work done per second on the wheel is the output of the system whereas the initial kinetic energy per second of the jet is the input. Hence, efficiency of the system is expressed as

$$\text{Efficiency,} \qquad \eta = \frac{\text{Work done per second}}{\text{Kinetic energy per second}} = \frac{\rho a V_1 \left[V_{w_1} u_1 \pm V_{w_2} u_2\right]}{\frac{1}{2}(\text{mass/sec}) \times V_1^2}$$

$$= \frac{\rho a V_1 \left[V_{w_1} u_1 \pm V_{w_2} u_2\right]}{\frac{1}{2}(\rho a V_1) \times V_1^2} = \frac{2\left[V_{w_1} u_1 \pm V_{w_2} u_2\right]}{V_1^2}. \qquad \text{...(17.28)}$$

If there is no loss of energy when water is flowing over the vanes, the work done on the wheel per second is also equal to the change in kinetic energy of the jet per second. Hence, the work done per second on the wheel is also given as

Work done per second on the wheel

= Change of K.E. per second of the jet

= (Initial K.E. per second – Final K.E. per second) of the jet

$$= \left(\frac{1}{2} m V_1^2 - \frac{1}{2} m V_2^2\right)$$

$$= \frac{1}{2} m \left(V_1^2 - V_2^2\right) = \frac{1}{2}\left(\rho a V_1^2\right)\left(V_1^2 - V_2^2\right) \qquad (\because \text{mass/second} = \rho a V_1)$$

$$\text{Hence efficiency, } \eta = \frac{\text{Work done per second on the wheel}}{\text{Initial K.E. per second of the jet}}$$

$$= \frac{V_1^2 - V_2^2}{V_1^2} = \left(1 - \frac{V_2^2}{V_1^2}\right) \qquad \text{...(17.28A)}$$

From the above equation, it is clear that for a given initial velocity of the jet (*i.e.*, V_1), the efficiency will be maximum, when V_2 is minimum. But V_2 cannot be zero as in that case the incoming jet will not move out of the vane. Equation (17.28) also gives the efficiency of the system. From this equation, it is clear that efficiency will be maximum when V_{w_2} is added to V_{w_1}. This is only posible if β is an acute* angle. Also for maximum efficiency V_{w_2} should also be maximum. This is only possible if $\beta = 0$. In that case $V_{w_2} = V_2$ and angle ϕ will be zero. But in actual practice ϕ cannot be zero. Hence for maximum efficiency, the angle ϕ should be minimum.

HYDRAULIC MACHINES- INTRODUCTION

Hydraulic machines are defined as those machines which convert either hydraulic energy (energy possessed by water) into mechanical energy (which is further converted into electrical energy) or mechanical energy into hydraulic energy.

The hydraulic machines, which convert the hydraulic energy into mechanical energy, are called *turbines* while the hydraulic machines which convert the mechanical energy into hydraulic energy are called *pumps.*

Thus the study of hydraulic machines consists of study of turbines and pumps. Turbines consists of mainly study of Pelton turbine, Francis Turbine and Kaplan Turbine while pumps consist of study of centrifugal pump and reciprocating pumps.

Turbines

Turbines are defined as the hydraulic machines which convert hydraulic energy into mechanical energy. This mechanical energy is used in running an electric generator which is directly coupled to the shaft of the turbine. Thus the mechanical energy is converted into electrical energy.

The electric power which is obtained from the hydraulic energy (energy of water) is known as *Hydroelectric power.* At present the generation of hydroelectric power is the cheapest as compared by the power generated by other sources such as oil, coal etc.

GENERAL LAYOUT OF A HYDROELECTRIC POWER PLANT

Fig. 18.1 shows a general layout of a hydroelectric power plant which consists of

I. A dam constructed across a river to store water.

II. Pipes of large diameters called penstocks, which carry water under pressure from the storage reservoir to the turbines. These pipes are made of steel or reinforced concrete.

III. Turbines having different types of vanes fitted to the wheels.

IV. Tail race, which is a channel which carries water away from the turbines after the water has worked on the turbines. The surface of water in the tail race channel is also known as tail race.

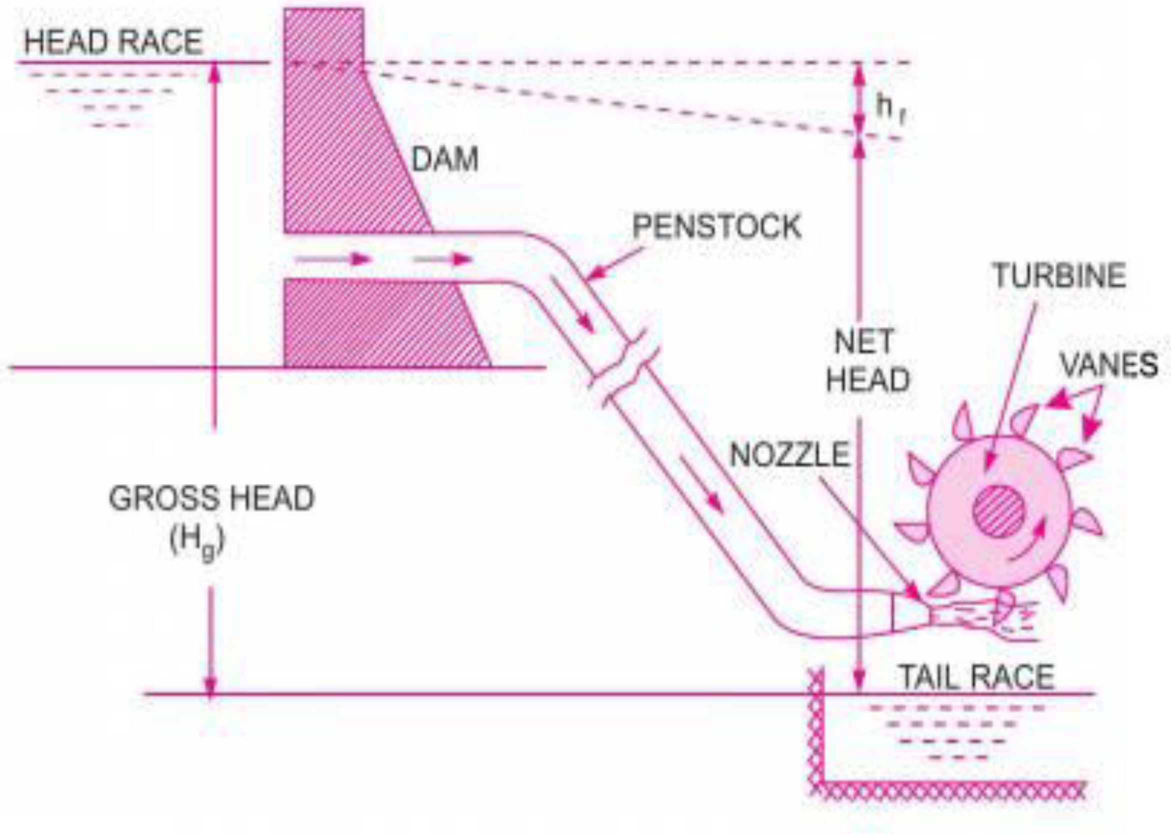


Fig. 18.1 *Layout of a hydroelectric power plant.*

DEFINITIONS OF HEADS AND EFFICIENCIES OF A TURBINE

Gross Head. The difference between the head race level and tail race level when no water is flowing is known as **Gross Head**. It is denoted by *'Hg'* in Fig. 18.1.

Net Head. It is also called effective head and is defined as the head available at the inlet of the turbine.

When water is flowing from head race to the turbine, a loss of head due to friction between the water and penstocks occurs. Though there are other losses also such as loss due to bend, pipe fittings, loss at the entrance of penstock etc., yet they are having small magnitude as compared to head loss due to friction. If *'h_f is* the head loss due to friction between penstocks and water then net heat on turbine is given by

$$H = H_g - h_f$$

where H_g = Gross head, $h_f = \dfrac{4 \times f \times L \times V^2}{D \times 2g}$,

V = Velocity of flow in penstock,

L = Length of penstock,

D = Diameter of penstock.

- Efficiencies of a Turbine. The following are the important efficiencies of a turbine.

(a)Hydraulic Efficiency, η_h (b) Mechanical Efficiency, η_m

(c) Volumetric Efficiency, η_v and (d) Overall Efficiency, η_o

Hydraulic Efficiency(η_h)

It is defined as the ratio of power given by water to the runner of a turbine (runner is a rotating part of a turbine and on the runner vanes are fixed) to the power supplied by the water at the inlet of the turbine.

The power at the inlet of the turbine is more and this power goes on decreasing as the water flows over the vanes of the turbine due to hydraulic losses as the vanes are not smooth.

Hence, the power delivered to the runner of the turbine will be less than the power available at the inlet of the turbine. Thus, mathematically, the hydraulic efficiency of a turbine is written as

$$\eta_h = \frac{\text{Power delivered to runner}}{\text{Power supplied at inlet}} = \frac{\text{R.P.}}{\text{W.P.}} \qquad \text{...(18.2)}$$

where R.P. = Power delivered to runner *i.e.*, runner power

$$= \frac{W}{g}\frac{\left[V_{w_1} \pm V_{w_2}\right] \times u}{1000} \text{ kW} \qquad \text{...for Pelton Turbine}$$

$$= \frac{W}{g}\frac{\left[V_{w_1}u_1 \pm V_{w_2}u_2\right]}{1000} \text{ kW} \qquad \text{...for a radial flow turbine}$$

W.P. = Power supplied at inlet of turbine and also called water power

$$= \frac{W \times H}{1000} \text{ kW} \qquad \text{...(18.3)}$$

where W = Weight of water striking the vanes of the turbine per second

= $\rho g \times Q$ in which Q = Volume of water/s,

V_{w_1} = Velocity of whirl at inlet,

V_{w_2} = Velocity of whirl at outlet,

u = Tangential velocity of vane,

u_1 = Tangential velocity of vane at inlet for radial vane,

u_2 = Tangential velocity of vane at outlet for radial vane,

H = Net head on the turbine.

Power supplied at the inlet of turbine in S.I.units is known as water power. It is given by

$$\text{W.P.} = \frac{\rho \times g \times Q \times H}{1000} \text{ kW} \qquad \text{...(18.3}A\text{)}$$

For water $\rho = 1000 \text{ kg/m}^3$

$$\therefore \qquad \text{W.P.} = \frac{1000 \times g \times Q \times H}{1000} = g \times Q \times H \text{ kW} \qquad \text{...(18.3}B\text{)}$$

The relation (18.3*B*) is only used when the flowing fluid is water. If the flowing fluid is other than the water, then relation (18.3*A*) is used.

Mechanical Efficiency, η_m

The power delivered by water to the runner of a turbine is transmitted to the shaft of the turbine. Due to mechanical losses, the power available at the shaft of the turbine is less than the power delivered to the runner of a turbine.

The ratio of the power available at the shaft of the turbine (known as S.P. or B.P.) to the power delivered to the runner is defined as mechanical efficiency. Hence, mathematically, it is written as

$$\eta_m = \frac{\text{Power at the shaft of the turbine}}{\text{Power delivered by water to the runner}} = \frac{\text{S.P.}}{\text{R.P.}} \qquad \text{...(18.4)}$$

Volumetric Efficiency, η_v

The volume of the water striking the runner of a turbine is slightly less than the volume of the water supplied to the turbine. Some of the volume of the water is discharged to the tail race without striking the runner of the turbine. Thus the ratio of the volume of the water actually striking the runner to the volume of water supplied to the turbine is defined as volumetric efficiency.

It is written as

$$\eta_v = \frac{\text{Volume of water actually striking the runner}}{\text{Volume of water supplied to the turbine}} \qquad \text{...(18.5)}$$

Overall Efficiency, η_o

It is defined as the ratio of power available at the shaft of the turbine to the power supplied by the water at the inlet of the turbine. It is written as :

$$\eta_o = \frac{\text{Volume available at the shaft of the turbine}}{\text{Power supplied at the inlet of the turbine}} = \frac{\text{Shaft power}}{\text{Water power}}$$

$$= \frac{\text{S.P.}}{\text{W.P.}}$$

$$= \frac{\text{S.P.}}{\text{W.P.}} \times \frac{\text{R.P.}}{\text{R.P.}} \qquad \text{(where R.P. = Power delivered to runner)}$$

$$= \frac{\text{S.P.}}{\text{R.P.}} \times \frac{\text{R.P.}}{\text{W.P.}}$$

$$= \eta_m \times \eta_h \qquad \left(\because \text{From equation (18.4)}, \frac{\text{S.P.}}{\text{R.P.}} = \eta_m \text{ and from equation (18.2)}, \frac{\text{R.P.}}{\text{W.P.}} = \eta_h \right) \qquad ...(18.6)$$

If shaft power (S.P.) is taken in kW then water power should also be taken in kW. Shaft power is commonly represented by P. But from equation (18.3A),

$$\text{Water power in kW} = \frac{\rho \times g \times Q \times H}{1000}, \text{ where } \rho = 1000 \text{ kg/m}^3$$

$$\therefore \qquad \eta_o = \frac{\text{Shaft power in kW}}{\text{Water power in kW}} = \frac{P}{\left(\frac{\rho \times g \times Q \times H}{1000}\right)} \qquad ...(18.6A)$$

where P = Shaft power.

CLASSIFICATION OF HYDRAULIC TURBINES

The hydraulic turbines are classified according to the type of energy available at the inlet of the turbine, direction of flow through the vanes, head at the inlet of the turbine and specific speed of the turbines. Thus the following are the important classifications of the turbines :

According to the type of energy at inlet :

(a)Impulse turbine, and (b) Reaction turbine.

According to the direction of flow through runner :

(a)Tangential flow turbine, (b) Radial flow turbine,

(c) Axial flow turbine, and (d) Mixed flow turbine.

According to the head at the inlet of turbine :

(a) High head turbine, (b) Medium head turbine, and (c) Low head turbine.

According to the specific speed of the turbine :

(a) Low specific speed turbine, (b) Medium specific speed turbine, and (c) High specific speed turbine.

If at the inlet of the turbine, the energy available is only kinetic energy, the turbine is known as **impulse turbine**. As the water flows over the vanes, the pressure is atmospheric from inlet to outlet of the turbine.

If at the inlet of the turbine, the water possesses kinetic energy as well as pressure energy, the turbine is known as **reaction turbine.**

As the waters flows through the runner, the water is under pressure and the pressure energy goes on changing into kinetic energy. The runner is completely enclosed in an air-tight casing and the runner and casing is completely full of water.

If the water flows along the tangent of the runner, the turbine is known as **tangential flow turbine**.

If the water flows in the radial direction through the runner, the turbine is called **radial flow turbine**.

If the water flows from outwards to inwards, radially, the turbine is known as **inward radial flow turbine**,

On the other hand, if water flows radially from inwards to outwards, the turbine is known as **outward radial flow turbine**.

If the water flows through the runner along the direction parallel to the axis of rotation of the runner, the turbine is called **axial flow turbine.**

If the water flows through the runner in the radial direction but leaves in the direction parallel to axis of rotation of the runner, the turbine is called **mixed flow turbine**.

PELTON WHEEL (OR TURBINE)

The Pelton wheel or Pelton turbine is a tangential flow impulse turbine. The water strikes the bucket along the tangent of the runner.

The energy available at the inlet of the turbine is only kinetic energy. The pressure at the inlet and outlet of the turbine is atmospheric.

This turbine is used for high heads and is named after L.A. Pelton, an American Engineer.

Fig. 18.1 shows the layout of a hydroelectric power plant in which the turbine is Pelton wheel.

The water from the reservoir flows through the penstocks at the outlet of which a nozzle is fitted.

- The nozzle increases the kinetic energy of the water flowing through the penstock.
- At the outlet of the nozzle, the water comes out in the form of a jet and strikes the buckets (vanes) of the runner. The main parts of the Pelton turbine are :

1. Nozzle and flow regulating arrangement (spear),
2. Runner and buckets,
3. Casing, and
4. Breaking jet.

Nozzle and Flow Regulating Arrangement.

The amount of water striking the buckets (vanes) of the runner is controlled by providing a spear in the nozzle as shown in Fig. 18.2.

The spear is a conical needle which is operated either by a hand wheel or automatically in an axial direction depending upon the size of the unit.

When the spear is pushed forward into the nozzle the amount of water striking the runner is reduced.

On the other hand, if the spear is pushed back, the amount of water striking the runner increases.

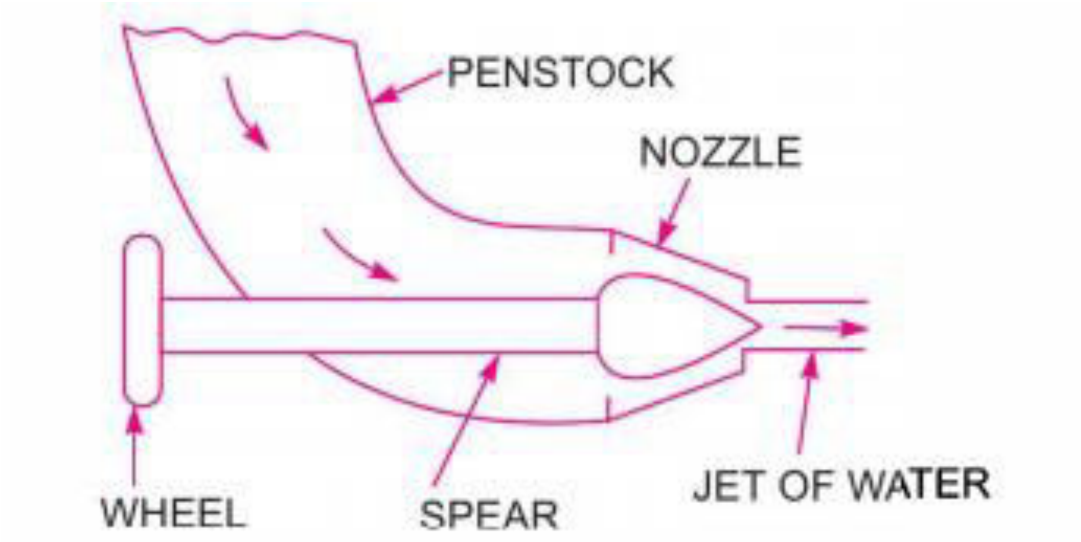


Fig. 18.2 *Nozzle with a spear to regulate flow.*

Runner with Buckets

Fig. 18.3 shows the runner of a Pelton wheel. It consists of a circular disc on the periphery of which a number of buckets evenly spaced are fixed. The shape of the buckets is of a double hemispherical cup or bowl. Each bucket is divided into two symmetrical parts by a dividing wall which is known as splitter. The jet of water strikes on the splitter. The splitter divides the jet into two equal parts and the jet comes out at the outer edge of the bucket. The buckets are shaped in such a way that the jet gets deflected through 160° or 170°. The buckets are made of cast iron, cast steel bronze or stainless steel depending upon the head at the inlet of the turbine.

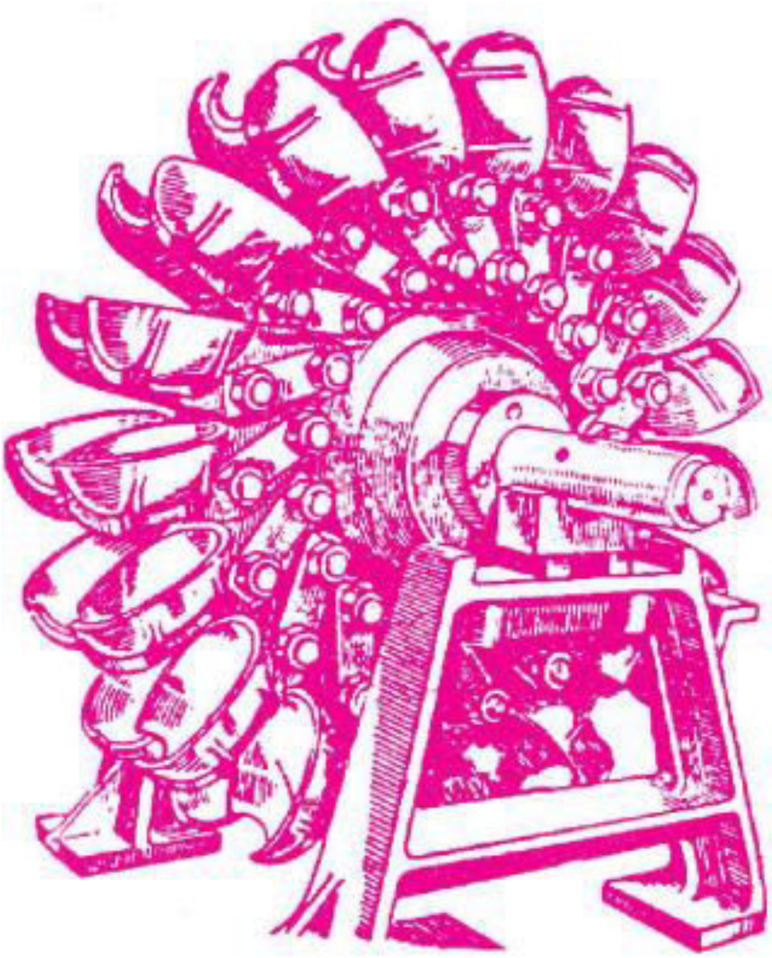


Fig. 18.3 *Runner of a pelton wheel.*

Casing

Fig. 18.4 shows a Pelton turbine with a casing. The function of the casing is to prevent the splashing of the water and to discharge water to tail race. It also acts as safeguard against accidents. It is made of cast iron or fabricated steel plates. The casing of the Pelton wheel does not perform any hydraulic function.

Breaking Jet

When the nozzle is completely closed by moving the spear in the forward direction, the amount of water striking the runner reduces to zero. But the runner due to inertia goes on revolving for a long time. To stop the runner in a short time, a small nozzle is provided which directs the jet of water on the back of the vanes. This jet of water is called breaking jet.

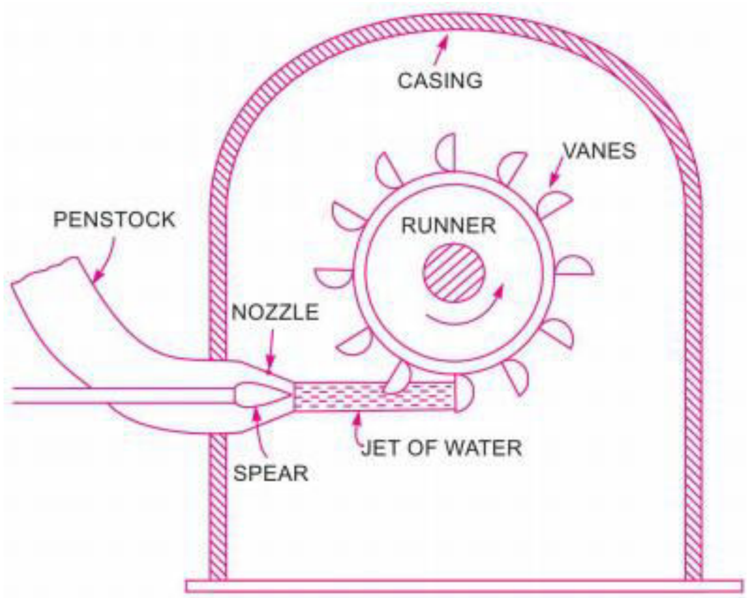


Fig. 18.4 *Pelton turbine.*

VELOCITY TRIANGLES AND WORK DONE FOR PELTON WHEEL

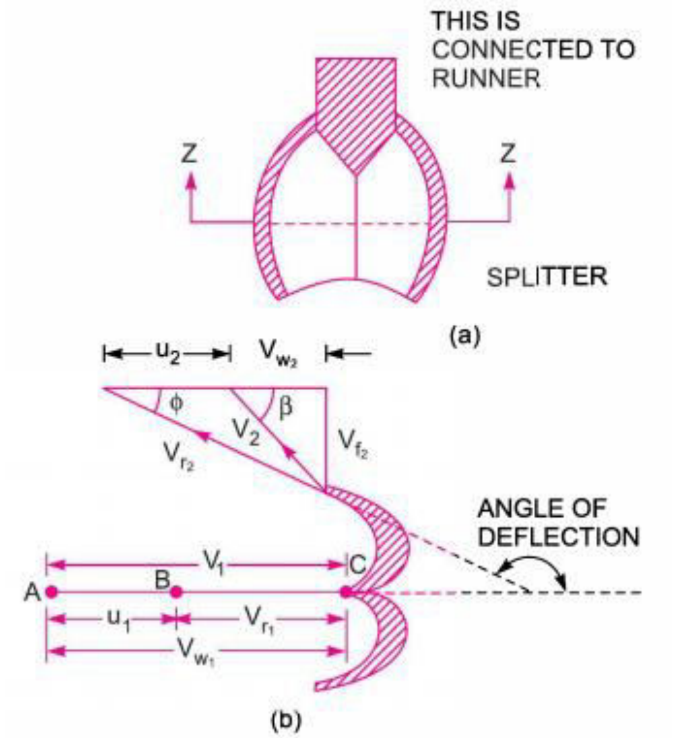


Fig. 18.5 *Shape of bucket.*

Fig. 18.5 shows the shape of the vanes or buckets of the Pelton wheel. The jet of water from the nozzle strikes the bucket at the splitter, which splits up the jet into two parts.

These parts of the jet, glides over the inner surfaces and comes out at the outer edge.

Fig. 18.5 (b) shows the section of the bucket at Z-Z. The splitter is the inlet tip and outer edge of the bucket is the outlet tip of the bucket.

The inlet velocity triangle is drawn at the splitter and outlet velocity triangle is drawn at the outer edge of the bucket.

Let H = Net head acting on the Pelton wheel

$$= H_g - h_f$$

where H_g = Gross head and $h_f = \dfrac{4fLV^2}{D^* \times 2g}$

where D^* = Dia. of Penstock, N = Speed of the wheel in r.p.m.,

D = Diameter of the wheel, d = Diameter of the jet.

Then

$$V_1 = \text{Velocity of jet at inlet} = \sqrt{2gH} \quad \text{...(18.7)}$$

$$u = u_1 = u_2 = \frac{\pi DN}{60}.$$

The velocity triangle at inlet will be a straight line where

$$V_{r_1} = V_1 - u_1 = V_1 - u$$

$$V_{w_1} = V_1$$

$$\alpha = 0° \text{ and } \theta = 0°$$

From the velocity triangle at outlet, we have

$$V_{r_2} = V_{r_1} \text{ and } V_{w_2} = V_{r_2} \cos\phi - u_2.$$

The force exerted by the jet of water in the direction of motion is given by equation (17.19) as

$$F_x = \rho a V_1 \, [V_{w_1} + V_{w_2}] \quad \text{...(18.8)}$$

As the angle β is an acute angle, +ve sign should be taken. Also this is the case of series of vanes, the mass of water striking is $\rho a V_1$ and not $\rho a V_{r_1}$. In equation (18.8), 'a' is the area of the jet which is given as

$$a = \text{Area of jet} = \frac{\pi}{4}d^2.$$

Now work done by the jet on the runner per second

$$= F_x \times u = \rho a V_1 \, [V_{w_1} + V_{w_2}] \times u \text{ Nm/s} \quad \text{...(18.9)}$$

Power given to the runner by the jet

$$= \frac{\rho a V_1 \left[V_{w_1} + V_{w_2}\right] \times u}{1000} \text{ kW} \quad \text{...(18.10)}$$

Work done/s per unit weight of water striking/s

$$= \frac{\rho a V_1 \left[V_{w_1} + V_{w_2}\right] \times u}{\text{Weight of water striking/s}}$$

$$= \frac{\rho a V_1 \left[V_{w_1} + V_{w_2}\right] \times u}{\rho a V_1 \times g} = \frac{1}{g}\left[V_{w_1} + V_{w_2}\right] \times u \quad \text{...(18.11)}$$

The energy supplied to the jet at inlet is in the form of kinetic energy and is equal to $\frac{1}{2}mV^2$

$\therefore$ K.E. of jet per second $= \frac{1}{2}(\rho a V_1) \times V_1^2$

$\therefore$ **Hydraulic efficiency**, $\eta_h = \dfrac{\text{Work done per second}}{\text{K.E. of jet per second}}$

$$= \frac{\rho a V_1 \left[V_{w_1} + V_{w_2}\right] \times u}{\frac{1}{2}(\rho a V_1) \times V_1^2} = \frac{2\left[V_{w_1} + V_{w_2}\right] \times u}{V_1^2} \qquad \text{...(18.12)}$$

Now $V_{w_1} = V_1,\ V_{r_1} = V_1 - u_1 = (V_1 - u)$

$\therefore$ $V_{r_2} = (V_1 - u)$

and $V_{w_2} = V_{r_2}\cos\phi - u_2 = V_{r_2}\cos\phi - u = (V_1 - u)\cos\phi - u$

Substituting the values of V_{w_1} and V_{w_2} in equation (18.12),

$$\eta_h = \frac{2\left[V_1 + (V_1 - u)\cos\phi - u\right] \times u}{V_1^2}$$

$$= \frac{2\left[V_1 - u + (V_1 - u)\cos\phi\right] \times u}{V_1^2} = \frac{2(V_1 - u)\left[1 + \cos\phi\right] u}{V_1^2}. \qquad \text{...(18.13)}$$

The efficiency will be maximum for a given value of V_1 when

$$\frac{d}{du}(\eta_h) = 0 \qquad \text{or} \qquad \frac{d}{du}\left[\frac{2u(V_1 - u)(1 + \cos\phi)}{V_1^2}\right] = 0$$

or $\dfrac{(1 + \cos\phi)}{V_1^2}\dfrac{d}{du}(2uV_1 - 2u^2) = 0$ or $\dfrac{d}{du}[2uV_1 - 2u^2] = 0$ $\left(\because \dfrac{1 + \cos\phi}{V_1^2} \neq 0\right)$

or $2V_1 - 4u = 0$ or $u = \dfrac{V_1}{2}$ ...(18.14)

Equation (18.14) states that hydraulic efficiency of a Pelton wheel will be maximum when the velocity of the wheel is half the velocity of the jet of water at inlet. The expression for maximum efficiency will be obtained by substituting the value of $u = \frac{V_1}{2}$ in equation (18.13).

$$\therefore \qquad \text{Max. } \eta_h = \frac{2\left(V_1 - \frac{V_1}{2}\right)(1 + \cos\phi) \times \frac{V_1}{2}}{V_1^2}$$

$$= \frac{2 \times \frac{V_1}{2}(1 + \cos\phi)\frac{V_1}{2}}{V_1^2} = \frac{(1 + \cos\phi)}{2}. \qquad \text{...(18.15)}$$

18.6.2 Points to be Remembered for Pelton Wheel

(*i*) The velocity of the jet at inlet is given by $V_1 = C_v\sqrt{2gH}$

where C_v = Co-efficient of velocity = 0.98 or 0.99

H = Net head on turbine

(*ii*) The velocity of wheel (u) is given by $u = \phi\sqrt{2gH}$

where ϕ = Speed ratio. The value of speed ratio varies from 0.43 to 0.48.

(*iii*) The angle of deflection of the jet through buckets is taken at 165° if no angle of deflection is given.

(*iv*) The mean diameter or the pitch diameter D of the Pelton wheel is given by

$$u = \frac{\pi DN}{60} \text{ or } D = \frac{60u}{\pi N}.$$

(*v*) **Jet Ratio.** It is defined as the ratio of the pitch diameter (D) of the Pelton wheel to the diameter of the jet (d). It is denoted by 'm' and is given as

$$m = \frac{D}{d} \text{ (= 12 for most cases)} \qquad \text{...(18.16)}$$

(*vi*) Number of buckets on a runner is given by

$$Z = 15 + \frac{D}{2d} = 15 + 0.5 \text{ m} \qquad \text{...(18.17)}$$

where m = Jet ratio

(*vii*) **Number of Jets.** It is obtained by dividing the total rate of flow through the turbine by the rate of flow of water through a single jet.

RADIAL FLOW REACTION TURBINES

Radial flow turbines are those turbines in which the water flows in the radial direction. The water may flow radially from outwards to inwards *(i.e.,* towards the axis of rotation) or from inwards to outwards.

If the water flows from outwards to inwards through the runner, the turbine is known as inward radial flow turbine. And if the water flows from inwards to outwards, the turbine is known as outward radial flow turbine.

Reaction turbine means that the water at the inlet of the turbine possesses kinetic energy as well as pressure energy. As the water flows through the runner, a part of pressure energy goes on changing into kinetic energy.

Thus the water through the runner is under pressure. The runner is completely enclosed in an air-tight casing and casing and the runner is always full of water.

MAIN PARTS OF A RADIAL FLOW REACTION TURBINE.

The main parts of a radial flow reaction turbine are :

- Casing,
- Guide mechanism,
- Runner, and

- Draft-tube.

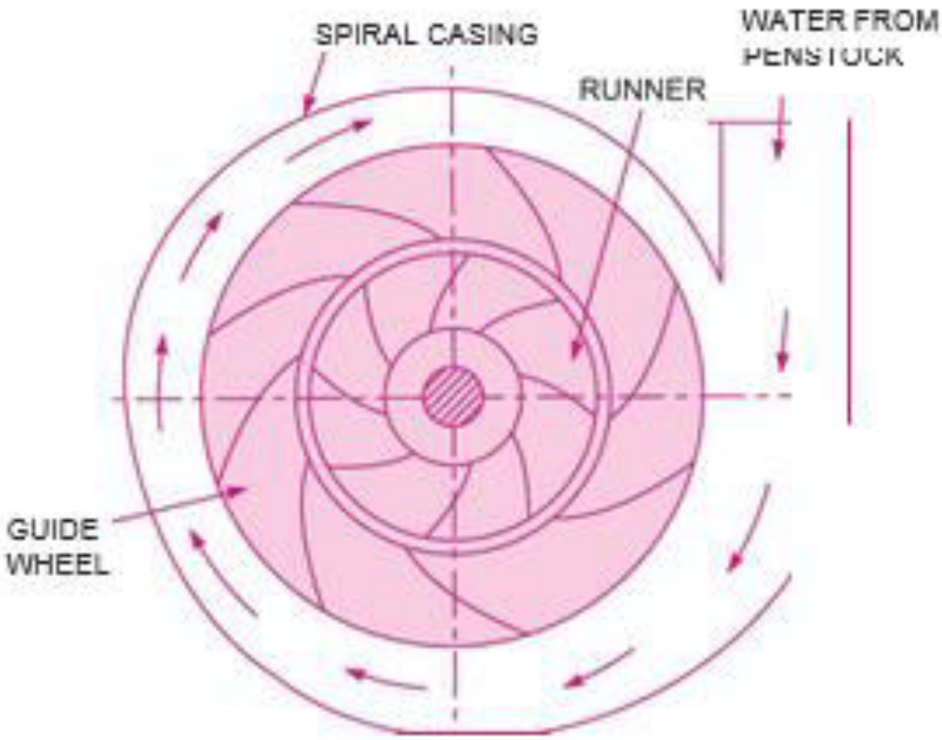


Fig. 18.10 *Main parts of a radial reaction turbines.*

Casing.

As mentioned above that in case of reaction turbine, casing and runner are always full of water. The water from the penstocks enters the casing which is of spiral shape in which area of cross section of the casing goes on decreasing gradually.

The casing completely surrounds the runner of the turbine. The casing as shown in Fig. 18.10 is made of spiral shape, so that the water may enter the runner at constant velocity throughout the circumference of the runner. The casing is made of concrete, cast steel or plate steel.

Guide Mechanism.

It consists of a stationary circular wheel all round the runner of the turbine. The stationary guide vanes are fixed on the guide mechanism. The guide vanes allow the water to strike the vanes fixed on the runner without shock at inlet.

Also by a suitable arrangement, the width between two adjacent vanes of guide mechanism can be altered so that the amount of water striking the runner can be varied.

Runner.

It is a circular wheel on which a series of radial curved vanes are fixed. The surface of the vanes are made very smooth.

The radial curved vanes are so shaped that the water enters and leaves the runner without shock. The runners are made of cast steel, cast iron or stainless steel. They are keyed to the shaft.

Draft-tube.

The pressure at the exit of the runner of a reaction turbine is generally less than atmospheric pressure.

The water at exit cannot be directly discharged to the tail race. A tube or pipe of gradually increasing area is used for discharging water from the exit of the turbine to the tail race. This tube of increasing area is called draft tube.

INWARD RADIAL FLOW TURBINE

Fig. 18.11 shows inward radial flow turbine, in which case the water from the casing enters the stationary guiding wheel. The guiding wheel consists of guide vanes which direct the water to enter the runner which consists of moving vanes.

The water flows over the moving vanes in the inward radial direction and is discharged at the inner diameter of the runner. The outer diameter of the runner is the inlet and the inner diameter is the outlet. Velocity Triangles and Work done by Water on Runner. we have discussed in detail the force exerted by the water on the radial curved vanes fixed on a wheel.

From the force exerted on the vanes, the work done by water, the horse power given by the water to the vanes and efficiency of the vanes can be obtained. Also we have drawn velocity triangles at inlet and outlet of the moving radial vanes in Fig. 17.23.

From the velocity triangles, the work done by the water on the runners, horse power and efficiency of the turbine can be obtained.

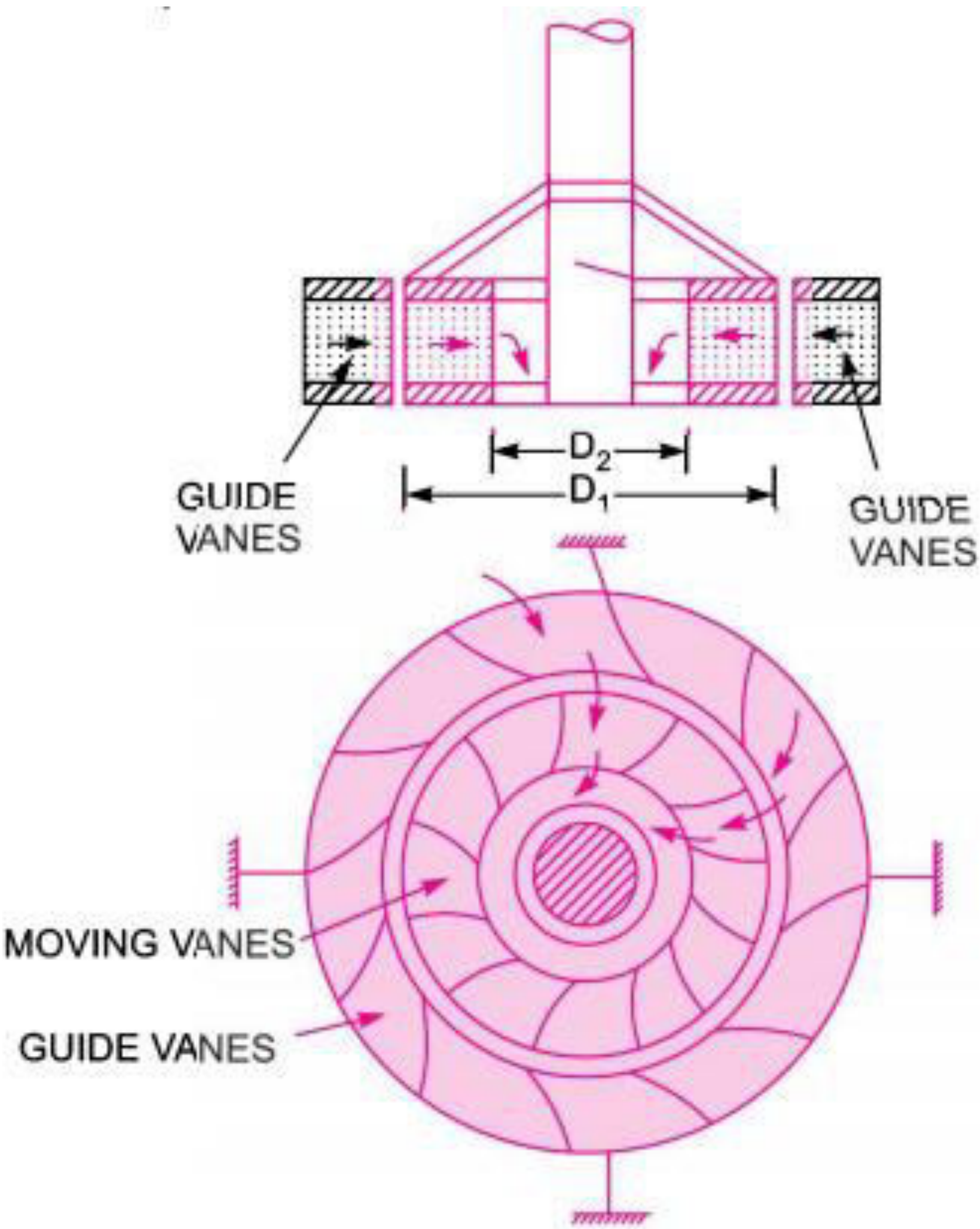


Fig. 18.11 *Inward radial flow turbine.*

The work done per second on the runner by water is given by equation (17.26) as

$$= \rho a V_1 [V_{w_1} u_1 \pm V_{w_2} u_2]$$

$$= \rho Q [V_{w_1} u_1 \pm V_{w_2} u_2] \qquad (\because \; aV_1 = Q) \quad ...(18.18)$$

The equation (18.18) also represents the energy transfer per second to the runner.

where V_{w_1} = Velocity of whirl at inlet,

V_{w_2} = Velocity of whirl at outlet,

u_1 = Tangential velocity of wheel at inlet

$= \frac{\pi D_1 \times N}{60}$, where D_1 = Outer dia. of runner,

u_2 = Tangential velocity of wheel at outlet

$= \frac{\pi D_2 \times N}{60}$, where D_2 = Inner dia. of runner, N = Speed of the turbine in .r.p.m.

The work done per second per unit weight of water per second.

$$= \frac{\text{Work done per second}}{\text{Weight of water striking per second}}$$

$$= \frac{\rho Q \left[V_{w_1} u_1 \pm V_{w_2} u_2 \right]}{\rho Q \times g} = \frac{1}{g} \left[V_{w_1} u_1 \pm V_{w_2} u_2 \right] \quad ...(18.19)$$

The equation (18.19) represents the energy transfer per unit weight/s to the runner. This equation is known by **Euler's equation** of hydrodynamics machines. This is also known as fundamental equation of hydrodynamic machines. This equation was given by Swiss scientist *L.* Euler.

In equation (18.19), +ve sign is taken if angle β is an acute angle. If β is an obtuse angle then –ve sign is taken. If $\beta = 90°$, then $V_{w_2} = 0$ and work done per second per unit weight of water striking/s become as

$$= \frac{1}{g} V_{w_1} u_1 \quad ...(18.20)$$

Hydraulic efficiency is obtained from equation (18.2) as

$$\eta_h = \frac{\text{R.P.}}{\text{W.P.}} = \frac{\frac{W}{1000 g} [V_{w_1} u_1 \pm V_{w_2} u_2]}{\frac{W \times H}{1000}} = \frac{(V_{w_1} u_1 \pm V_{w_2} u_2)}{gH} \quad ...(18.20A)$$

where R.P. = Runner power *i.e.*, power delivered by water to the runner

W.P. = Water power

If the discharge is radial at outlet, then $V_{w_2} = 0$

$$\eta_h = \frac{V_{w_1} u_1}{gH} \quad ...(18.20B)$$

18.7.3 Degree of Reaction. Degree of reaction is defined as the ratio of pressure energy change inside a runner to the total energy change inside the runner. It is represented by 'R'. Hence mathematically it can be written as

$$R = \frac{\text{Change of pressure energy inside the runner}}{\text{Change of total energy inside the runner}} \qquad \text{...(18.20C)}$$

The equation (18.19) which is the fundamental equation of hydrodynamic machines, represents the energy transfer per unit weight to the runner. This is also known as the total energy change inside the runner per unit weight.

$\therefore$ Change of total energy per unit weight inside the runner

$$= \frac{1}{g}[V_{w_1} u_1 \pm V_{w_2} u_2]$$

Let H_e = Change of total energy per unit weight inside the runner.

Then
$$H_e = \frac{1}{g}[V_{w_1} u_1 \pm V_{w_2} u_2] \qquad \text{...(18.20D)}$$

Let us find the values of $V_{w_1}u_1$ and $V_{w_2}u_2$ from inlet and outlet velocity triangles.

Now from inlet velocity triangle, we know that [Refer to Fig. 18.11(*a*)]

$$V_{w_1} = u_1 + V_{r_1x}, \quad \text{where } V_{r_1x} = V_{r_1} \cos\theta = \sqrt{V_{r_1}^2 - V_{f_1}^2}$$

$$= u_1 + \sqrt{V_{r_1}^2 - V_{f_1}^2}$$

$$= u_1 + \sqrt{V_{r_1}^2 - (V_1^2 - V_{w_1}^2)} \quad [\because \text{From triangle } ABC,\ V_{f_1}^2 = V_1^2 - V_{w_1}^2]$$

$$\therefore \quad (V_{w_1} - u_1) = \sqrt{V_{r_1}^2 - (V_1^2 - V_{w_1}^2)}$$

Squaring both sides, we get

$$(V_{w_1} - u_1)^2 = V_{r_1}{}^2 - (V_1^2 - V_{w_1}^2)$$

or
$$V_{w_1}^2 + u_1{}^2 - 2V_{w_1} u_1 = V_{r_1}{}^2 - V_1^2 + V_{w_1}^2$$

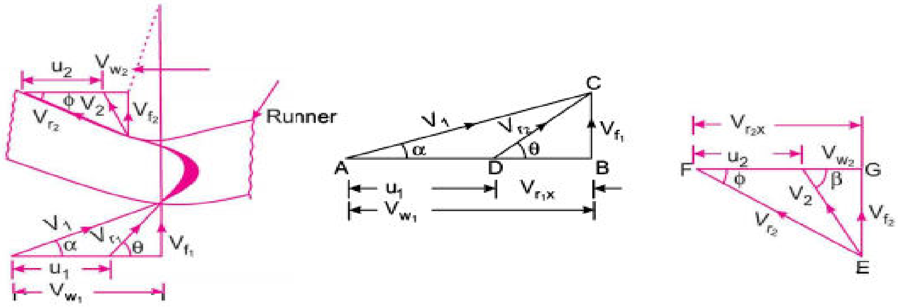


Fig. 18.11 (*a*)

or
$$V_{w_1}^2 + u_1{}^2 - V_{r_1}^2 + V_1^2 - V_{w_1}^2 = 2V_{w_1} u_1$$

or
$$u_1{}^2 - V_{r_1}{}^2 + V_1^2 = 2V_{w_1} u_1$$

or
$$2V_{w_1} u_1 = u_1{}^2 - V_{r_1}^2 + V_1^2$$

or
$$V_{w_1} u_1 = \frac{1}{2}[u_1^2 - V_{r_1}^2 + V_1^2] \qquad \text{...(i)}$$

Similarly from outlet triangle, we know that [Refer to Fig. 18.11(a)]

$$V_{w_2} = V_{r_2x} - u_2$$

$$= \sqrt{V_{r_2}{}^2 - V_{f_2}^2} - u_2, \text{ where } V_{r_2x} = V_{r_2}\cos\theta = \sqrt{V_{r_2}^2 - V_{f_2}^2}$$

$$= \sqrt{V_{r_2}{}^2 - (V_2^2 - V_{w_2}^2)} - u_2 \qquad \therefore \qquad V_{f_2}^2 = V_2^2 - V_{w_2}^2$$

$$\therefore \qquad V_{w_2} + u_2 = \sqrt{V_{r_2}^2 - V_2^2 + V_{w_2}^2}$$

Squaring both sides, we get

$$(V_{w_2} + u_2)^2 = V_{r_2}^2 - V_2^2 + V_{w_2}^2$$

or $$V_{w_2}^2 + u_2^2 + 2V_{w_2}u_2 = V_{r_2}^2 - V_2^2 + V_{w_2}^2$$

or $$2V_{w_2}u_2 = V_{r_2}^2 - V_2^2 + V_{w_2}^2 - V_{w_2}^2 - u_2^2$$

or $$2V_{w_2}u_2 = V_{r_2}^2 - V_2^2 - u_2^2$$

or $$V_{w_2}u_2 = \frac{1}{2}[V_{r_2}^2 - V_2^2 - u_2^2] \qquad \text{...}(ii)$$

In the above case of velocity triangles under consideration, the change of total energy per unit weight inside the runner is equal to $\frac{1}{g}\,[V_{w_1}u_1 + V_{w_2}u_2]$

Substituting the values of $V_{w_1}u_1$ and $V_{w_2}u_2$ from equations (i) and (ii) into equation (18.20 D), we get Change of total energy per unit weight inside the runner as

$$H_e = \frac{1}{g}\left[\frac{1}{2}\left(u_1^2 - V_{r_1}{}^2 + V_1^2\right) + \frac{1}{2}\left(V_{r_2}{}^2 - V_2^2 - u_2^2\right)\right]$$

$$= \frac{1}{2g}\left[\left(u_1^2 - u_2^2\right) + \left(V_1^2 - V_2^2\right) + \left(V_{r_2}{}^2 - V_{r_1}{}^2\right)\right]$$

$$= \frac{V_1^2 - V_2^2}{2g} + \frac{u_1^2 - u_2^2}{2g} + \frac{V_{r_2}{}^2 - V_{r_1}{}^2}{2g} \qquad \text{...(18.20}E\text{)}$$

The above equation consists of three terms. The first term represents the change in kinetic energy of the fluid per unit weight and the second term represents the change of energy per unit weight due to centrifugal action. The third term represents the change in static pressure energy per unit weight, as per Bernoulli's equation applied to relative flow through runner passage by reducing the rotating system into stationary system. We know that the energy change due to centrifugal action takes place in the form of pressure energy. [When a container containing a liquid is rotated, then due to centrifugal action there is change of pressure energy *i.e.*, $h = \frac{\Delta p}{\rho g} = \frac{u_2^2 - u_1^2}{2g}$]. Hence, the last two terms in equation (18.20E) represents the change in pressure energy inside the runner passage per unit weight.

$\therefore$ Change in pressure energy inside the runner per unit weight $= \frac{u_1^2 - u_2^2}{2g} + \frac{V_{r_2}^2 - V_{r_1}^2}{2g}$ ...(*iii*)

Now the equation (18.20*C*) becomes as

$$R = \frac{\text{Change of pressure energy inside the runner per unit weight}}{\text{Change of total energy inside the runner per unit weight}}$$

$$= \left(\frac{\left(u_1^2 - u_2^2\right)}{2g} + \frac{V_{r_2}^2 - V_{r_1}^2}{2g}\right) \Big/ \left[\left(\frac{V_1^2 - V_2^2}{2g}\right) + \left(\frac{u_1^2 - u_2^2}{2g}\right) + \left(\frac{V_{r_2}^2 - V_{r_1}^2}{2g}\right)\right]$$

or $$R = \frac{\left(u_1^2 - u_2^2\right) + \left(V_{r_2}^2 - V_{r_1}^2\right)}{\left(V_1^2 - V_2^2\right) + \left(u_1^2 - u_2^2\right) + \left(V_{r_2}^2 - V_{r_1}^2\right)} \quad ...(18.20F)$$

or $$R = \frac{\left(V_1^2 - V_2^2\right) + \left(u_1^2 - u_2^2\right) + \left(V_{r_2}^2 - V_{r_1}^2\right) - \left(V_1^2 - V_2^2\right)}{\left(V_1^2 - V_2^2\right) + \left(u_1^2 - u_2^2\right) + \left(V_{r_2}^2 - V_{r_1}^2\right)}$$

$$= 1 - \frac{\left(V_1^2 - V_2^2\right)}{\left(V_1^2 - V_2^2\right) + \left(u_1^2 - u_2^2\right) + \left(V_{r_2}^2 - V_{r_1}^2\right)} \quad ...(18.20G)$$

From equation (18.20*E*) , we know that

$$H_e = \frac{V_1^2 - V_2^2}{2g} + \frac{u_1^2 - u_2^2}{2g} + \frac{V_{r_2}^2 - V_{r_1}^2}{2g}$$

or $$2gH_e = (V_1^2 - V_2^2) + (u_1^2 - u_2^2) + (V_{r_2}^2 - V_{r_1}^2)$$

Now the equation (18.20*G*) can be written as

$$R = 1 - \frac{\left(V_1^2 - V_2^2\right)}{2g\ H_e} \quad ...(18.20H)$$

Values of *R* for Pelton turbine and other actual reaction turbines

(*i*) *For a Pelton turbine,*

$$u_1 = u_2 \text{ and } V_{r_2} = V_{r_1}$$

$\therefore$ From equation (18.20*G*)

$$R = 1 - \frac{\left(V_1^2 - V_2^2\right)}{\left(V_1^2 - V_2^2\right)} = 1 - 1 = 0$$

(*ii*) *For an actual reaction turbine,* generally, the angle β is 90° so that the loss of kinetic energy at outlet is minimum (*i.e.*, V_2 is minimum).

Hence in outlet velocity triangle, V_{w_2} becomes zero (*i.e.*, $V_{w_2} = 0$). Also $V_2 = V_{f_2}$ [Refer to Fig. 18.11(*b*)]

Also there is not much change in velocity of flow. This means $V_{f_1} = V_{f_2}$

From equation (18.20*D*), we know that

$$H_e = \frac{1}{g}[V_{w_1}u_1 + V_{w_2}u_2]$$

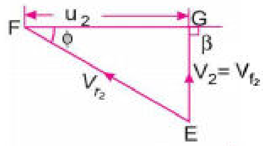


Fig. 18.11 (*b*)

$$= \frac{1}{g} V_{w_1} u_1 \qquad \left(\because \; V_{w_2} = 0\right)$$

$$= \frac{1}{g}[V_{f_1} \cot \alpha] [V_{f_1} \cot \alpha - V_{f_1} \cot \theta] \qquad \text{[Refer to Fig. 18.11}(a)]$$

$$[\because \; V_{w_1} = V_{f_1} \cot \alpha \text{ and } u_1 = V_{w_1} - V_{f_1} \cot \theta = V_{f_1} \cot \alpha - V_{f_1} \cot \theta]$$

$$= \frac{1}{g} V_{f_1}^2 \cot \alpha [\cot \alpha - \cot \theta]$$

Now $$V_1^2 - V_2^2 = (V_{f_1} \operatorname{cosec} \alpha)^2 - V_{f_2}^2 \qquad (\because \; V_2 = V_{f_2})$$

$$= V_{f_1}^2 \operatorname{cosec}^2 \alpha - V_{f_1}^2 \qquad (\because \; V_{f_2} = V_{f_1})$$

or $$V_1^2 - V_2^2 = V_{f_1}^2 (\operatorname{cosec}^2 \alpha - 1)$$

$$= V_{f_1}^2 \cot^2 \alpha \qquad (\because \; \operatorname{cosec}^2 \alpha - 1 = \cot^2 \alpha)$$

Substituting the value of H_e and $(V_1^2 - V_2^2)$ in equation (18.20H), we get

$$R = 1 - \frac{V_{f_1}^2 \cot^2 \alpha}{2g \times [\frac{1}{g} V_{f_1}^2 \cot \alpha (\cot \alpha - \cot \theta)]}$$

$$= 1 - \frac{\cot \alpha}{2(\cot \alpha - \cot \theta)} \qquad \text{...(18.20}I)$$

18.7.4 Definitions. The following terms are generally used in case of reaction radial flow turbines which are defined as :

(*i*) **Speed Ratio.** The speed ratio is defind as = $\frac{u_1}{\sqrt{2gH}}$

where u_1 = Tangential velocity of wheel at inlet.

(*ii*) **Flow Ratio.** The ratio of the velocity of flow at inlet (V_{f_1}) to the velocity given $\sqrt{2gH}$ is known as flow ratio or it is given as

$$= \frac{V_{f_1}}{\sqrt{2gH}}, \text{ where } H = \text{Head on turbine}$$

(*iii*) **Discharge of the Turbine.** The discharge through a reaction radial flow turbine is given by

$$Q = \pi D_1 B_1 \times V_{f_1} = \pi D_2 \times B_2 \times V_{f_2} \qquad \text{...(18.21)}$$

where D_1 = Diameter of runner at inlet,
B_1 = Width of runner at inlet,
V_{f_1} = Velocity of flow at inlet, and
D_2, B_2, V_{f_2} = Corresponding values at outlet.

If the thickness of vanes are taken into consideration, then the area through which flow takes place is given by $(\pi D_1 - n \times t)$

where n = Number of vanes on runner and t = Thickness of each vane

The discharge Q, then is given by $Q = (\pi D_1 - n \times t)\, B_1 \times V_{f_1}$ ...(18.22)

(*iv*) The head (H) on the turbine is given by $H = \dfrac{p_1}{\rho \times g} + \dfrac{V_1^2}{2g}$ ...(18.23)

where p_1 = Pressure at inlet.

(*v*) **Radial Discharge.** This means the angle made by absolute velocity with the tangent on the wheel is 90° and the component of the whirl velocity is zero. Radial discharge at outlet means $\beta = 90°$ and $V_{w_2} = 0$, while radial discharge at outlet means $\alpha = 90°$ and $V_{w_1} = 0$.

(*vi*) If there is no loss of energy when water flows through the vanes then we have

$$H - \frac{V_2^2}{2g} = \frac{1}{g}[V_{w_1} u_1 \pm V_{w_2} u_2]. \qquad ...(18.24)$$

OUTWARD RADIAL FLOW REACTION TURBINE.

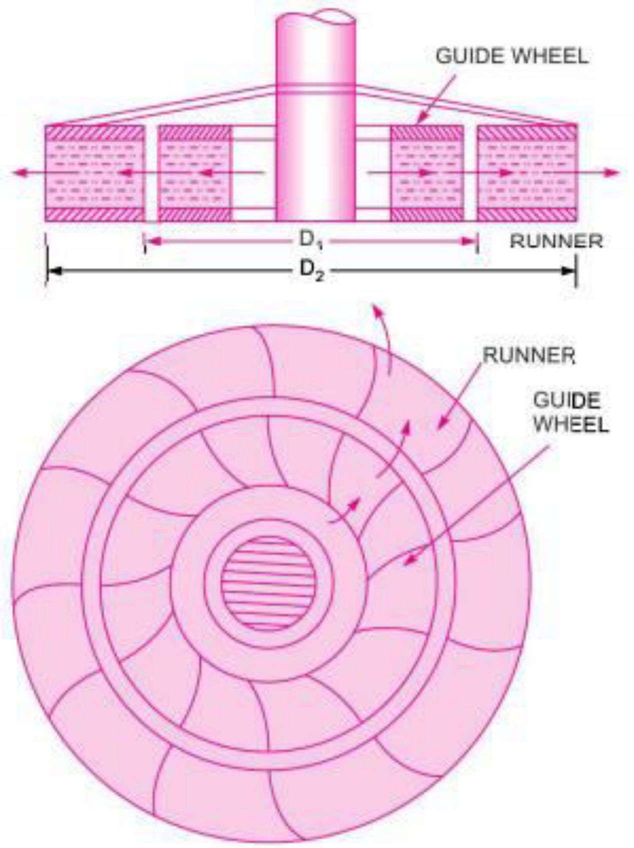


Fig. 18.18 *Outward radial flow turbine.*

Fig. 18.18 shows outward radial flow reaction turbine in which the water from casing enters the stationary guide wheel. The guide wheel consists of guide vanes which direct water to enter the runner which is around the stationary guide wheel. The water flows through the vanes of the runner in the outward radial direction and is discharged at the outer diameter of the runner. The inner diameter of the runner is inlet and outer diameter is the outlet.

The velocity triangles at inlet and outlet will be drawn by the same procedure as adopted for inward flow turbine. The work done by the water on the runner per second, the horse power developed and hydraulic efficiency will be obtained from the velocity triangles. In this case as inlet of the runner is at the inner diameter of the runner, the tangential velocity at inlet will be less than that of at outlet, *i.e.,*

$$u_1 < u_2 \text{ as } D_1 < D_2$$

FRANCIS TURBINE

The inward flow reaction turbine having radial discharge at outlet is known as Francis Turbine, after the name of J.B. Francis, an American engineer who in the beginning designed inward radial flow reaction type of turbine.

In the modern Francis turbine, the water enters the runner of the turbine in the radial direction at outlet and leaves in the axial direction at the inlet of the runner.

Thus the modern Francis Turbine is a mixed flow type turbine The velocity triangle at inlet and outlet of the Francis turbine are drawn in the same way as in case of inward flow reaction turbine.

As in case of Francis turbine, the discharge is radial at outlet, the velocity of whirl at outlet *(i.e.,* V_{w2} *)* will be zero. Hence the work done by water on the runner per second will be

$$= \rho Q[V_{w_1} u_1]$$

And work done per second per unit weight of water striking/s $= \frac{1}{g}\left[V_{w_1} u_1\right]$

Hydraulic efficiency will be given by, $\eta_h = \frac{V_{w_1} u_1}{gH}$.

18.8.1 Important Relations for Francis Turbines. The following are the important relations for Francis Turbines :

1. The ratio of width of the wheel to its diameter is given as $n = \frac{B_1}{D_1}$. The value of n varies from 0.10 to .40.

2. The flow ratio is given as,

Flow ratio = $\frac{V_{f_1}}{\sqrt{2gH}}$ and varies from 0.15 to 0.30.

3. The speed ratio = $\frac{u_1}{\sqrt{2gH}}$ varies from 0.6 to 0.9.

AXIAL FLOW REACTION TURBINE

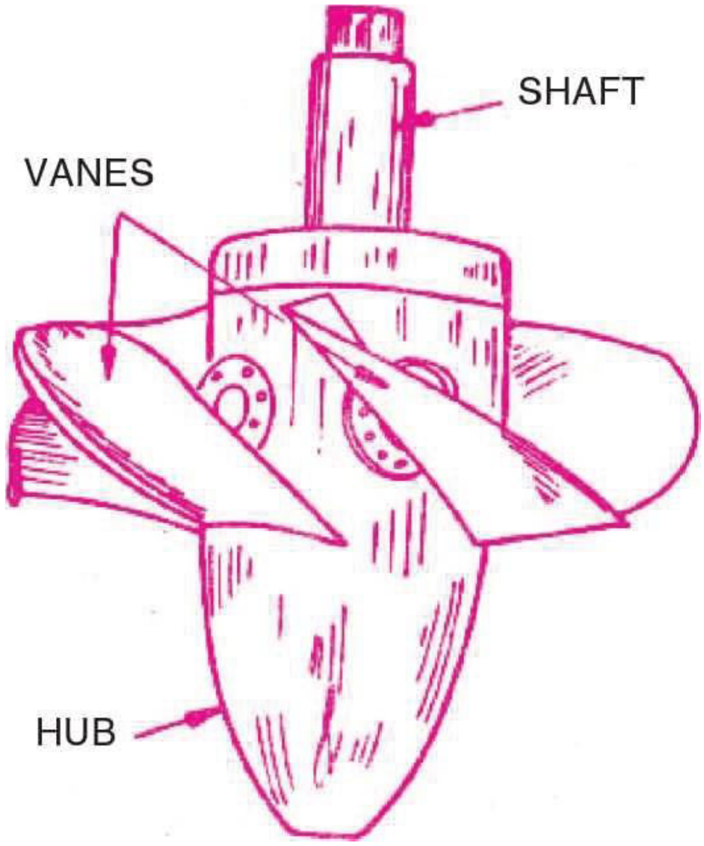
If the water flows parallel to the axis of the rotation of the shaft, the turbine is known as axial flow turbine. And if the head at the inlet of the turbine is the sum of pressure energy and kinetic energy and during the flow of water through runner a part of pressure energy is converted into kinetic energy, the turbine is known as reaction turbine.

For the axial flow reaction turbine, the shaft of the turbine is vertical. The lower end of the shaft is made larger which is known as 'hub' or 'boss'. The vanes are fixed on the hub and hence hub acts as a runner for axial flow reaction turbine. The following are the important type of axial flow reaction turbines:

1.Propeller Turbine, and 2. Kaplan Turbine.

When the vanes are fixed to the hub and they are not adjustable, the turbine is known as propeller turbine. But if the vanes on the hub are adjustable, the turbine is known as a *Kaplan Turbine,* after the name of V Kaplan, an Austrian Engineer.

This turbine is suitable where a large quantity of water at low head is available. Fig. 18.25 shows the runner of a Kaplan turbine, which consists of a hub fixed to the shaft. On the hub, the adjustable vanes are fixed as shown in Fig. 18.25.



The main parts of a Kaplan turbine are :

1. Scroll casing,
2. Guide vanes mechanism,
3. Hub with vanes or runner of the turbine, and
4. Draft tube.

Fig. 18.26 shows all main parts of a Kaplan turbine. The water from penstock enters the scroll casing and then moves to the guide vanes. From the guide vanes, the water turns through 90° and flows axially through the runner as shown in Fig. 18.26. The discharge through the runner is obtained as

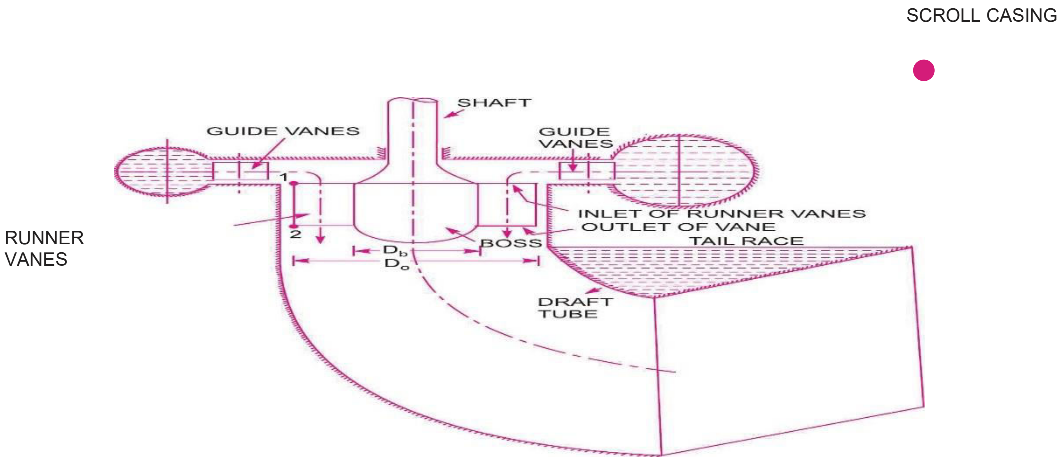
$$Q = \frac{\pi}{4}\left(D_o^2 - D_b^3\right) \times V_{f_1} \qquad \text{...(18.25)}$$

where D_o = Outer diameter of the runner,

D_b = Diameter of hub, and

V_{f_1} = Velocity of flow at inlet.

The inlet and outlet velocity triangles are drawn at the extreme edge of the runner vane corresponding to the points 1 and 2 as shown in Fig. 18.26.



18.9.1 Some Important Point for Propeller (Kaplan Turbine). The following are the important points for propeller or Kaplan turbine :

1. The peripheral velocity at inlet and outlet are equal

$\therefore$ $$u_1 = u_2 = \frac{\pi D_o N}{60}, \text{ where } D_o = \text{Outer dia. of runner}$$

2. Velocity of flow at inlet and outlet are equal

$\therefore$ $$V_{f_1} = V_{f_2}.$$

3. Area of flow at inlet = Area of flow at outlet

$$= \frac{\pi}{4}\left(D_o^2 - D_b^2\right).$$

CENTRIFUGAL PUMPS

The hydraulic machines which convert the mechanical energy into hydraulic energy are called pumps. The hydraulic energy is in the form of pressure energy. If the mechanical energy is converted into pressure energy by means of centrifugal force acting on the fluid, the hydraulic machine is called centrifugal pump.

The centrifugal pump acts as a reverse of an inward radial flow reaction turbine. This means that the flow in centrifugal pumps is in the radial outward directions.

The centrifugal pump works on the principle of forced vortex flow which means that when a certain mass of liquid is rotated by an external torque, the rise in pressure head of the rotating liquid takes place. The rise in pressure head at any point of the rotating liquid is proportional to the square of tangential velocity of the liquid at that point. *i.e.,* rise in pressure head.

$$= \frac{V^2}{2g} \text{ or } \frac{\omega^2 r^2}{2g}$$

Thus at the outlet of the impeller, where radius is more, the rise in pressure head will be more and the liquid will be discharged at the outlet with a high pressure head. Due to this high pressure head, the liquid can be lifted to a high level.

MAIN PARTS OF A CENTRIFUGAL PUMP

The following are the main parts of a centrifugal pump :

- Impeller.
- Casing.
- Suction pipe with a foot valve and a strainer.
- Delivery pipe

All the main parts of the centrifugal pump are shown in Fig. 19.1.

Impeller.

- The rotating part of a centrifugal pump is called 'impeller'.
- It consists of a series of backward curved vanes. The impeller is mounted on a shaft which is connected to the shaft of an electric motor.

Casing.

- The casing of a centrifugal pump is similar to the casing of a reaction turbine.
- It is an air tight passage surrounding the impeller and is designed in such a way that the kinetic energy of the water discharged at the outlet of the impeller is converted into pressure energy before the water leaves the casing and enters the delivery pipe. The following three types of the casings are commonly adopted :
- Volute casing as shown in Fig. 19.1.
- Vortex casing as shown in Fig. 19.2 (a).

- Casing with guide blades as shown in Fig. 19.2 (b).

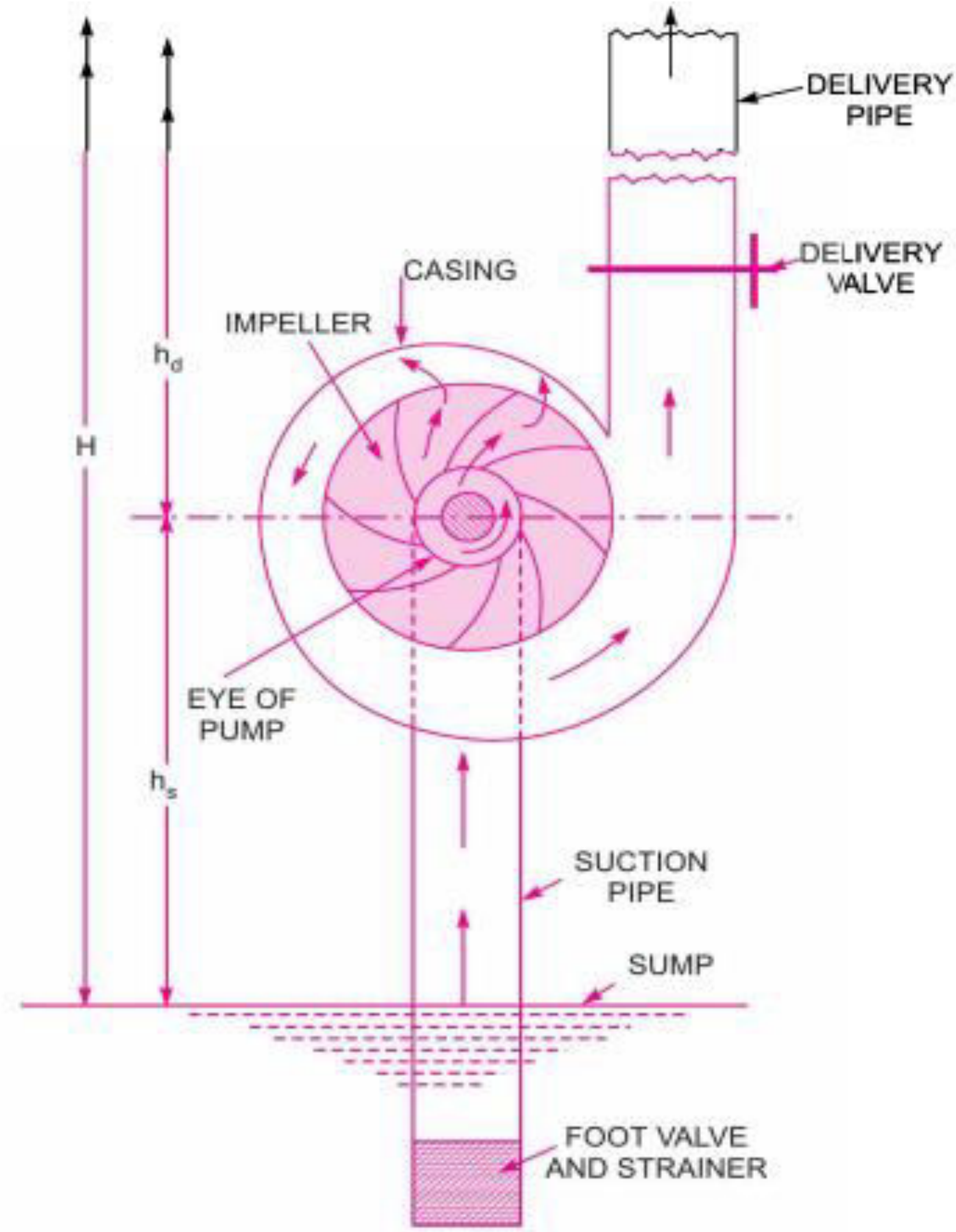


Fig. 19.1 *Main parts of a centrifugal pump.*

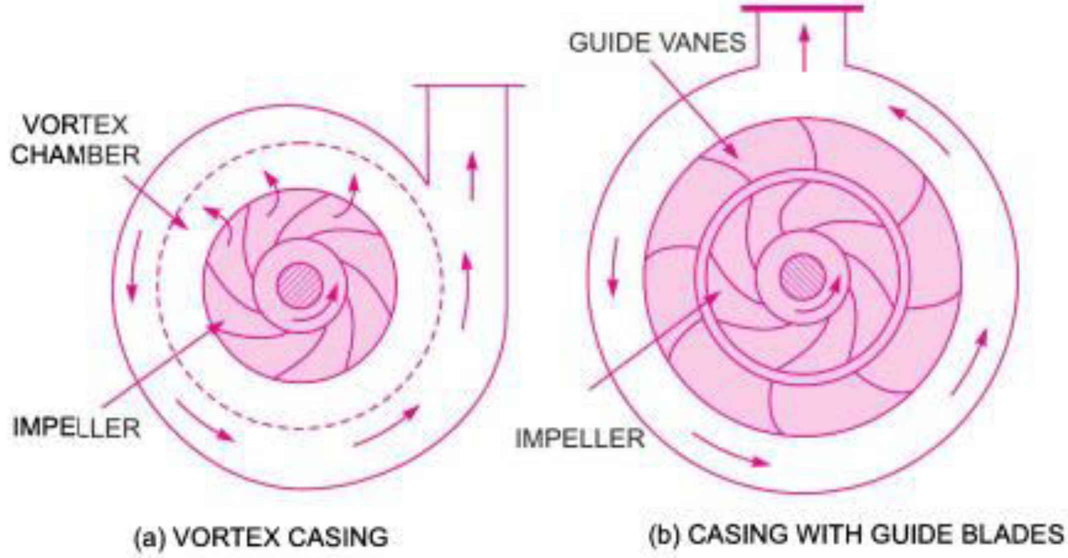


Fig. 19.2 *Different types of casing.*

Volute Casing.

- Fig 19.1 shows the volute casing, which surrounds the impeller. It is of spiral type in which area of flow increases gradually. The increase in area of flow decreases the velocity of flow.
- The decrease in velocity increases the pressure of the water flowing through the casing.
- It has been observed that in case of volute casing, the efficiency of the pump increases slightly as a large amount of energy is lost due to the formation of eddies in this type of casing.

Vortex Casing.

- If a circular chamber is introduced between the casing and the impeller as shown in Fig. 19.2 (a), the casing is known as Vortex Casing.
- By introducing the circular chamber, the loss of energy due to the formation of eddies is reduced to a considerable extent. Thus the efficiency of the pump is more than the efficiency when only volute casing is provided.

Casing with Guide Blades.

- This casing is shown in Fig. 19.2 (b) in which the impeller is surrounded by a series of guide blades mounted on a ring which is known as diffuser.
- The guide vanes are designed in such a way that the water from the impeller enters the guide vanes without stock.

- Also the area of the guide vanes increases, thus reducing the velocity of flow through guide vanes and consequently increasing the pressure of water.
- The water from the guide vanes then passes through the surrounding casing which is in most of the cases concentric with the impeller as shown in Fig. 19.2 (b).

Suction Pipe with a Foot valve and a Strainer.

- A pipe whose one end is connected to the inlet of the pump and other end dips into water in a sump is known as suction pipe.
- A foot valve which is a non-return valve or one-way type of valve is fitted at the lower end of the suction pipe.
- The foot valve opens only in the upward direction. A strainer is also fitted at the lower end of the suction pipe.

Delivery Pipe.

- A pipe whose one end is connected to the outlet of the pump and other end delivers the water at a required height is known as delivery pipe.

WORK DONE BY THE CENTRIFUGAL PUMP (OR BY IMPFLLER) ON WATER

In case of the centrifugal pump, work is done by the impeller on the water. The expression for the work done by the impeller on the water is obtained by drawing velocity triangles at inlet and outlet of the impeller in the same way as for a turbine.

The water enters the impeller radially at inlet for best efficiency of the pump, which means the absolute velocity of water at inlet makes an angle of 90° with the direction of motion of the impeller at inlet.

Hence angle $\alpha = 90°$ and $V_{w1} = 0$. For drawing the velocity triangles, the I same notations are used as that for turbines. Fig. 19.3 shows the velocity triangles at the inlet and outlet tips of the vanes fixed to an impeller.

N = Speed of the impeller in r.p.m.,

D_1 = Diameter of impeller at inlet,

u_1 = Tangential velocity of impeller at inlet = $\frac{\pi D_1 N}{60}$

D_2 = Diameter of impeller at outlet,

u_2 = Tangential velocity of impeller at outlet= $\frac{\pi D_2 N}{60}$

V_1 = Absolute velocity of water at inlet,

V_{r1} = Relative velocity of water at inlet,

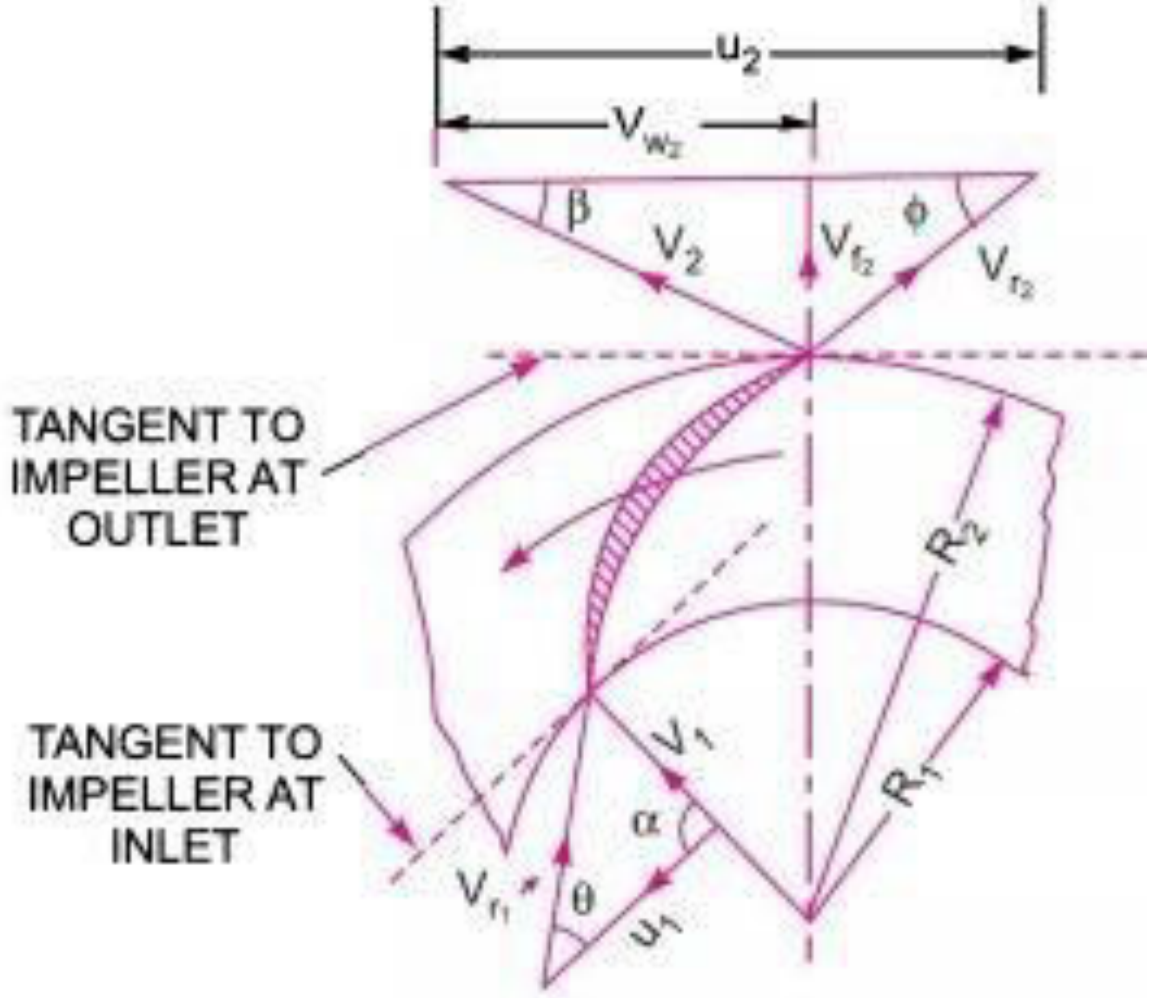


Fig. 19.3 *Velocity triangles at inlet and outlet.*

α = Angle made by absolute velocity (V1) at inlet with the direction of motion of vane,

θ = Angle made by relative velocity (V_{r1}) at inlet with the direction of motion of vane, and V_2. V_{r2} ,β and φ are the corresponding values at outlet.

As the water enters the impeller radially which means the absolute velocity of water at inlet is in the radial direction and hence angle α = 90° and V_{w1} = 0.

A centrifugal pump is the reverse of a radially inward flow reaction turbine. But in case of a radially

inward flow reaction turbine, the work done by the water on the runner per second per unit weight of the water striking per second is given by equation (18.19) as

$$= \frac{1}{g}[V_{w_1}u_1 - V_{w_2}u_2]$$

∴ Work done by the impeller on the water per second per unit weight of water striking per second

$$= -\text{[Work done in case of turbine]}$$

$$= -\left[\frac{1}{g}\left(V_{w_1}u_1 - V_{w_2}u_2\right)\right] = \frac{1}{g}\left[V_{w_2}u_2 - V_{w_1}u_1\right]$$

$$= \frac{1}{g}V_{w_2}u_2 \qquad (\because \; V_{w_1} = 0 \text{ here}) \quad ...(19.1)$$

Work done by impeller on water per second

$$= \frac{W}{g}.V_{w_2}u_2 \qquad ...(19.2)$$

where W = Weight of water = $\rho \times g \times Q$

where Q = Volume of water

and Q = Area × Velocity of flow = $\pi D_1 B_1 \times V_{f_1}$

$$= \pi D_2 B_2 \times V_{f_2} \qquad ...(19.2A)$$

where B_1 and B_2 are width of impeller at inlet and outlet and V_{f_1} and V_{f_2} are velocities of flow at inlet and outlet.

Equation (19.1) gives the head imparted to the water by the impeller or energy given by impeller to water per unit weight per second.

DEFINITIONS OF HEADS AND EFFICIENCIES OF A CENTRIFUGAL PUMP

Suction Head (h_s). It is the vertical height of the centre line of the centrifugal pump above the water surface in the tank or pump from which water is to be lifted as shown in Fig. 19.1. This height is also called suction lift and is denoted by 'h_s'

Delivery Head (h_d). The vertical distance between the centre line of the pump and the water surface in the tank to which water is delivered is known as delivery head. This is denoted by 'h_d'.

Static Head (H_S). The sum of suction head and delivery head is known as static head. This is represented by 'H_S' and is written as $H_s = h_s + h_d$.

Manometric Head (H_m). The manometric head is defined as the head against which a centrifugal pump has to work. It is denoted by 'H_m'. It is given by the following expressions :

(*a*) H_m = Head imparted by the impeller to the water – Loss of head in the pump

$$= \frac{V_{w_2} u_2}{g} - \text{Loss of head in impeller and casing} \quad \text{...(19.4)}$$

$$= \frac{V_{w_2} u_2}{g} \text{ ...if loss of pump is zero} \quad \text{...(19.5)}$$

(*b*) H_m = Total head at outlet of the pump – Total head at the inlet of the pump

$$= \left(\frac{P_o}{\rho g} + \frac{V_o^2}{2g} + Z_o\right) - \left(\frac{p_i}{\rho g} + \frac{V_i^2}{2g} + Z_i\right) \quad \text{...(19.6)}$$

where $\frac{p_o}{\rho g}$ = Pressure head at outlet of the pump = h_d

$\frac{V_o^2}{2g}$ = Velocity head at outlet of the pump

= Velocity head in delivery pipe = $\frac{V_d^2}{2g}$

Z_o = Vertical height of the outlet of the pump from datum line, and

$\frac{p_i}{\rho g}, \frac{V_i^2}{2g}$, Z_i = Corresponding values of pressure head, velocity head and datum head at the inlet of the pump,

i.e., h_s, $\frac{V_s^2}{2g}$ and Z_s respectively.

(*c*) $$H_m = h_s + h_d + h_{f_s} + h_{f_d} + \frac{V_d^2}{2g} \quad \text{...(19.7)}$$

where h_s = Suction head, h_d = Delivery head,

h_{f_s} = Frictional head loss in suction pipe, h_{f_d} = Frictional head loss in delivery pipe, and

V_d = Velocity of water in delivery pipe.

5. Efficiencies of a Centrifugal Pump. In case of a centrifugal pump, the power is transmitted from the shaft of the electric motor to the shaft of the pump and then to the impeller. From the impeller, the power is given to the water. Thus power is decreasing from the shaft of the pump to the impeller and then to the water. The following are the important efficiencies of a centrifugal pump :

(*a*) Manometric efficiency, η_{man} (*b*) Mechanical efficiency, η_m and

(*c*) Overall efficiency, η_o.

(*a*) **Manometric Efficiency (η_{man}).** The ratio of the manometric head to the head imparted by the impeller to the water is known as manometric efficiency. Mathematically, it is written as

$$\eta_{man} = \frac{\text{Manometric head}}{\text{Head imparted by impeller to water}}$$

$$= \frac{H_m}{\left(\frac{V_{w_2} u_2}{g}\right)} = \frac{gH_m}{V_{w_2} u_2} \qquad \text{...(19.8)}$$

The power at the impeller of the pump is more than the power given to the water at outlet of the pump. The ratio of the power given to water at outlet of the pump to the power available at the impeller, is known as manometric efficiency.

The power given to water at outlet of the pump $= \frac{WH_m}{1000}$ kW

$$\text{The power at the impeller} = \frac{\text{Work done by impeller per second}}{1000} \text{kW}$$

$$= \frac{W}{g} \times \frac{V_{w_2} \times u_2}{1000} \text{kW}$$

$$\eta_{man} = \frac{\frac{W \times H_m}{1000}}{\frac{W}{g} \times \frac{V_{w_2} \times u_2}{1000}} = \frac{g \times H_m}{V_{w_2} \times u_2}.$$

(*b*) **Mechanical Efficiency (η_m).** The power at the shaft of the centrifugal pump is more than the power available at the impeller of the pump. The ratio of the power available at the impeller to the power at the shaft of the centrifugal pump is known as mechanical efficiency. It is written as

$$\eta_m = \frac{\text{Power at the impeller}}{\text{Power at the shaft}}$$

$$\text{The power at the impeller in kW} = \frac{\text{Work done by impeller per second}}{1000}$$

$$= \frac{W}{g} \times \frac{V_{w_2} u_2}{1000} \qquad \text{[Using equation (19.2)]}$$

$$\eta_m = \frac{\frac{W}{g}\left(\frac{V_{w_2} u_2}{1000}\right)}{\text{S.P.}} \qquad \text{...(19.9)}$$

where S.P. = Shaft power.

(*c*) **Overall Efficiency (η_0).** It is defined as ratio of power output of the pump to the power input to the pump. The power output of the pump in kW

$$= \frac{\text{Weight of water lifted} \times H_m}{1000} = \frac{WH_m}{1000}$$

Power input to the pump = Power supplied by the electric motor

= S.P. of the pump.

$$\therefore \qquad \eta_o = \frac{\left(\dfrac{WH_m}{1000}\right)}{\text{S.P.}} \qquad \text{...(19.10)}$$

$$\text{Also} \qquad \eta_o = \eta_{man} \times \eta_m. \qquad \text{...(19.11)}$$

BASIC PROPELLER PRINCIPLES

The aircraft propeller consists of two or more blades and a central hub to which the blades are attached. Each blade of an aircraft propeller is essentially a rotating wing.

As a result of their construction, the propeller blades are like airfoils and produce forces that create the thrust to pull, or push, the aircraft through the air.

The engine furnishes the power needed to rotate the propeller blades through the air at high speeds, and the propeller transforms the rotary power of the engine into forward thrust.

A cross-section of a typical propeller blade is shown in Figure 4-35. This section or blade element is an airfoil comparable to a cross-section of an aircraft wing.

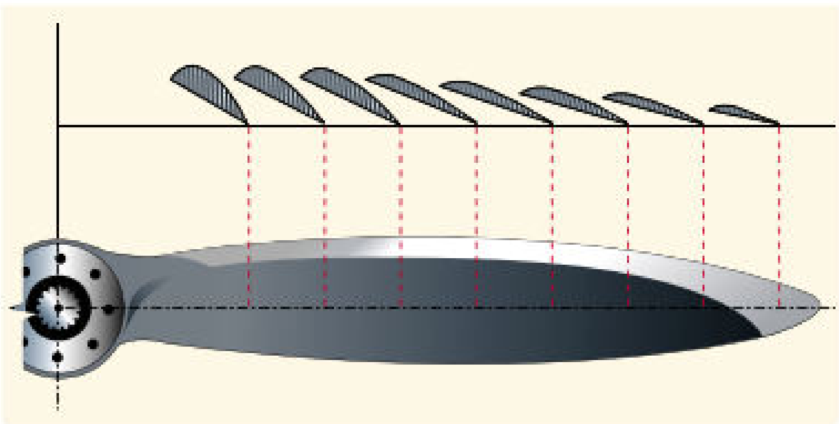


Figure 4-35. *Airfoil sections of propeller blade.*

One surface of the blade is cambered or curved, similar to the upper surface of an aircraft wing, while the other surface is flat like the bottom surface of a wing.

The chord line is an imaginary line drawn through the blade from its leading edge to its trailing edge. As in a wing, the leading edge is the thick edge of the blade that meets the air as the propeller rotates.

Blade angle, usually measured in degrees, is the angle between the chord of the blade and the plane of rotation and is measured at a specific point along the length of the blade.

[Figure 4-36] Because most propellers have a flat blade "face," the chord line is often drawn along the face of the propeller blade. Pitch is not blade angle, but because pitch is largely determined by blade angle, the two terms are often used interchangeably. An increase or decrease in one is usually associated with an increase or decrease in the other.

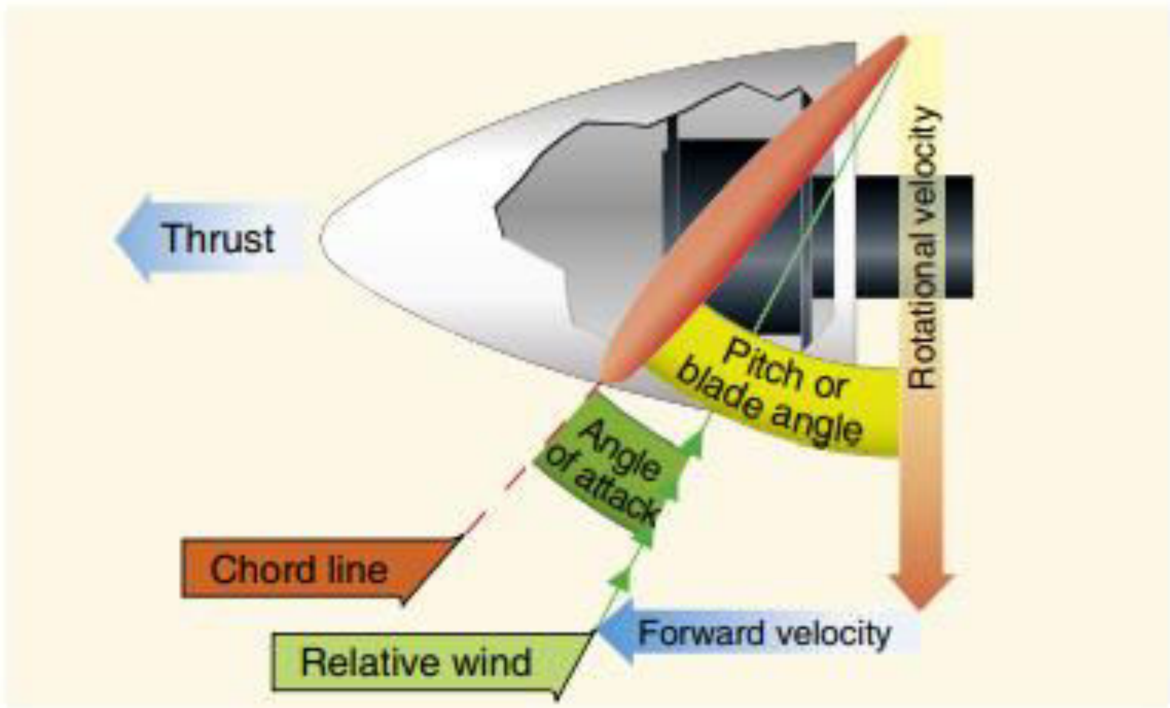


Figure 4-36. *Propeller blade angle.*

The pitch of a propeller may be designated in inches. A propeller designated as a "74-48" would be 74 inches in length and have an effective pitch of 48 inches.

The pitch is the distance in inches, which the propeller would screw through the air in one revolution if there were no slippage. When specifying a fixed-pitch propeller for a new type of aircraft, the manufacturer usually selects one with a pitch that operates efficiently at the expected cruising speed of the aircraft.

Every fixed-pitch propeller must be a compromise because it can be efficient at only a given combination of airspeed and revolutions per minute (rpm). Pilots cannot change this combination in flight.

When the aircraft is at rest on the ground with the engine operating, or moving slowly at the beginning of takeoff, the propeller efficiency is very low because the propeller is restrained from advancing with sufficient speed to permit its fixed-pitch blades to reach their full efficiency.In this situation, each propeller blade is turning through the air at an AOA that produces relatively little thrust for the amount of power required to turn it.

To understand the action of a propeller, consider first its motion, which is both rotational and forward. As shown by the vectors of propeller forces in Figure 4-36, each section of a propeller blade moves downward and forward. The angle at which this air (relative wind) strikes the propeller blade is its AOA.

The air deflection produced by this angle causes the dynamic pressure at the engine side of the propeller blade to be greater than atmospheric pressure, thus creating thrust.The shape of the blade also creates thrust because it is cambered like the airfoil shape of a wing. As the air flows past the propeller, the pressure on one side is less than that on the other.

As in a wing, a reaction force is produced in the direction of the lesser pressure. The airflow over the wing has less pressure, and the force (lift) is upward. In the case of the propeller, which is mounted in a vertical instead of a horizontal plane, the area of decreased pressure is in front of the propeller, and the force (thrust) is in a forward direction.

Aerodynamically, thrust is the result of the propeller shape and the AOA of the blade. Thrust can be considered also in terms of the mass of air handled by the propeller. In these terms, thrust equals mass of air handled multiplied by slipstream velocity minus velocity of the aircraft. The power expended in producing thrust depends on the rate of air mass movement.

On average, thrust constitutes approximately 80 percent of the torque (total horsepower absorbed by the propeller). The other 20 percent is lost in friction and slippage.

For any speed of rotation, the horsepower absorbed by the propeller balances the horsepower delivered by the engine. For any single revolution of the propeller, the amount of air handled depends on the blade angle, which determines how big a "bite" of air the propeller takes.

Thus, the blade angle is an excellent means of adjusting the load on the propeller to control the engine rpm. The blade angle is also an excellent method of adjusting the AOA of the propeller. On constant-speed propellers, the blade angle must be adjusted to provide the most efficient AOA at all engine and aircraft speeds. Lift versus drag curves, which are drawn for propellers, as well as wings, indicate that the most efficient AOA is small, varying from +2° to +4°. The actual blade angle necessary to maintain this small AOA varies with the forward speed of the aircraft.

Fixed-pitch and ground-adjustable propellers are designed for best efficiency at one rotation and forward speed. They are designed for a given aircraft and engine combination. A propeller may be used that provides the maximum efficiency for takeoff, climb, cruise, or high-speed flight. Any change in these conditions results in lowering the efficiency of both the propeller and the engine.

Since the efficiency of any machine is the ratio of the useful power output to the actual power input, propeller efficiency is the ratio of thrust horsepower to brake horsepower. Propeller efficiency varies from 50 to 87 percent, depending on how much the propeller "slips."

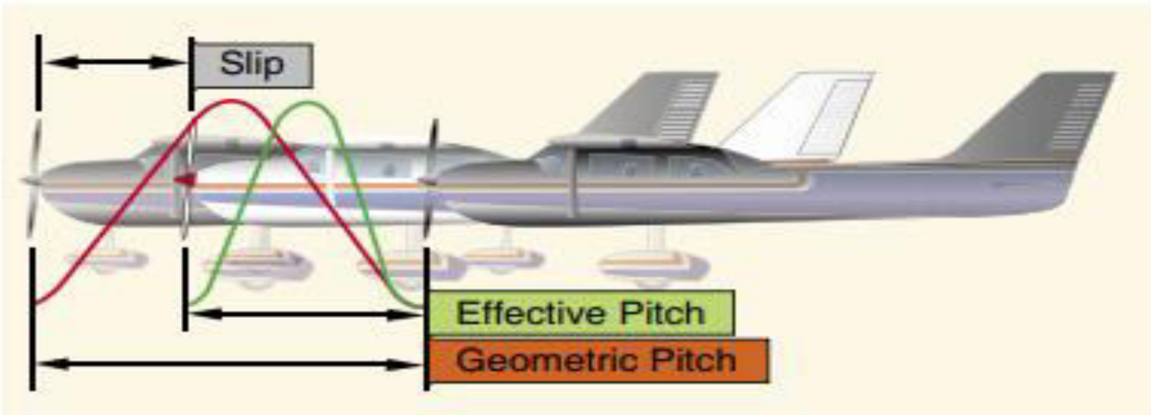


Figure 4-37. *Propeller slippage.*

Propeller slip is the difference between the geometric pitch of the propeller and its effective pitch. [Figure 4-37] Geometric pitch is the theoretical distance a propeller should advance in one revolution; effective pitch is the distance it actually advances. Thus, geometric or theoretical pitch is based on no slippage, but actual or effective pitch includes propeller slippage in the air.

The reason a propeller is "twisted" is that the outer parts of the propeller blades, like all things that turn about a central point, travel faster than the portions near the hub.

[Figure 4-38] If the blades had the same geometric pitch throughout their lengths, portions near the hub could have negative AOAs while the propeller tips would be stalled at cruise speed.

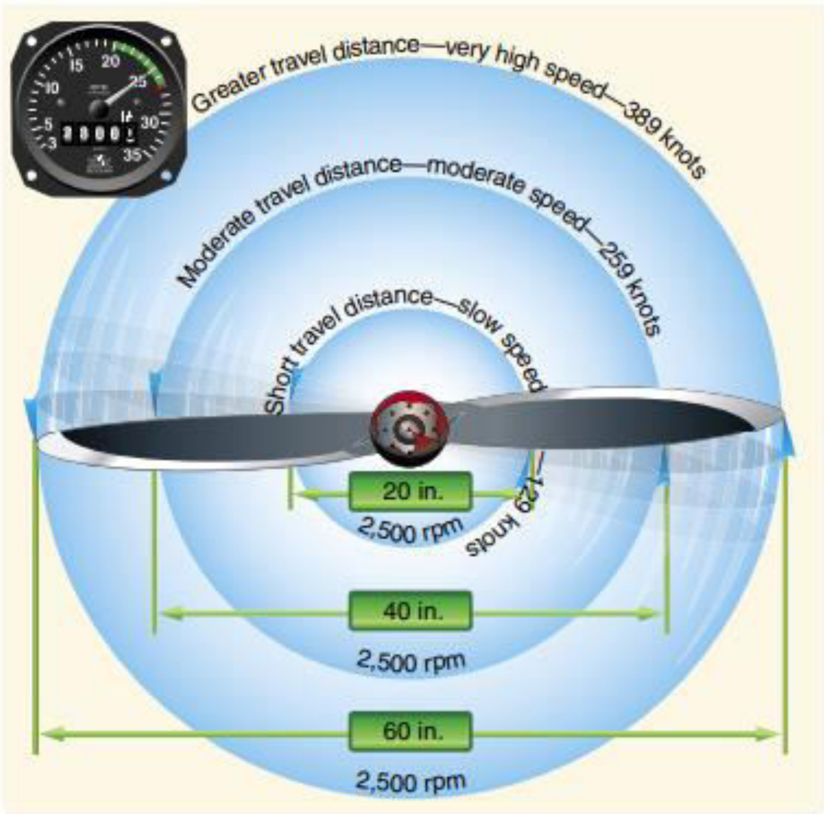


Figure 4-38. *Propeller tips travel faster than the hub.*

Twisting or variations in the geometric pitch of the blades permits the propeller to operate with a relatively constant AOA along its length when in cruising flight. Propeller blades are twisted to change the blade angle in proportion to the differences in speed of rotation along the length of the propeller, keeping thrust more nearly equalized along this length.

Usually 1° to 4° provides the most efficient lift/drag ratio, but in flight the propeller AOA of a fixed-pitch propeller varies—normally from 0° to 15°. This variation is caused by changes in the relative airstream, which in turn results from changes in aircraft speed. Thus, propeller AOA is the product of two motions: propeller rotation about its axis and its forward motion.

A constant-speed propeller automatically keeps the blade angle adjusted for maximum efficiency for most conditions encountered in flight. During takeoff, when maximum power and thrust are required, the constant-speed propeller is at a low propeller blade angle or pitch. The low blade angle keeps the AOA small and efficient with respect to the relative wind. At the same time, it allows the propeller to handle a smaller mass of air per revolution.

This light load allows the engine to turn at high rpm and to convert the maximum amount of fuel into heat energy in a given time. The high rpm also creates maximum thrust because, although the mass of air handled per revolution is small, the rpm and slipstream velocity are high, and with the low aircraft speed, there is maximum thrust.

After liftoff, as the speed of the aircraft increases, the constantspeed propeller automatically changes to a higher angle (or pitch). Again, the higher blade angle keeps the AOA small and efficient with respect to the relative wind. The higher blade angle increases the mass of air handled per revolution. This decreases the engine rpm, reducing fuel consumption and engine wear, and keeps thrust at a maximum.

After the takeoff climb is established in an aircraft having a controllable-pitch propeller, the pilot reduces the power output of the engine to climb power by first decreasing the manifold pressure and then increasing the blade angle to lower the rpm.

At cruising altitude, when the aircraft is in level flight and less power is required than is used in takeoff or climb, the pilot again reduces engine power by reducing the manifold pressure and then increasing the blade angle to decrease the rpm.

Again, this provides a torque requirement to match the reduced engine power. Although the mass of air handled per revolution is greater, it is more than offset by a decrease in slipstream velocity and an increase in airspeed. The AOA is still small because the blade angle has been increased with an increase in airspeed.

TORQUE AND P-FACTOR

To the pilot, "torque" (the left turning tendency of the airplane) is made up of four elements which cause or produce a twisting or rotating motion around at least one of the airplane's three axes. These four elements are:

1. Torque reaction from engine and propeller,

2. Corkscrewing effect of the slipstream,
3. Gyroscopic action of the propeller, and
4. Asymmetric loading of the propeller (P-factor).

Torque Reaction

Torque reaction involves Newton's Third Law of Physics— for every action, there is an equal and opposite reaction.

As applied to the aircraft, this means that as the internal engine parts and propeller are revolving in one direction, an equal force is trying to rotate the aircraft in the opposite direction. [Figure 4-39]

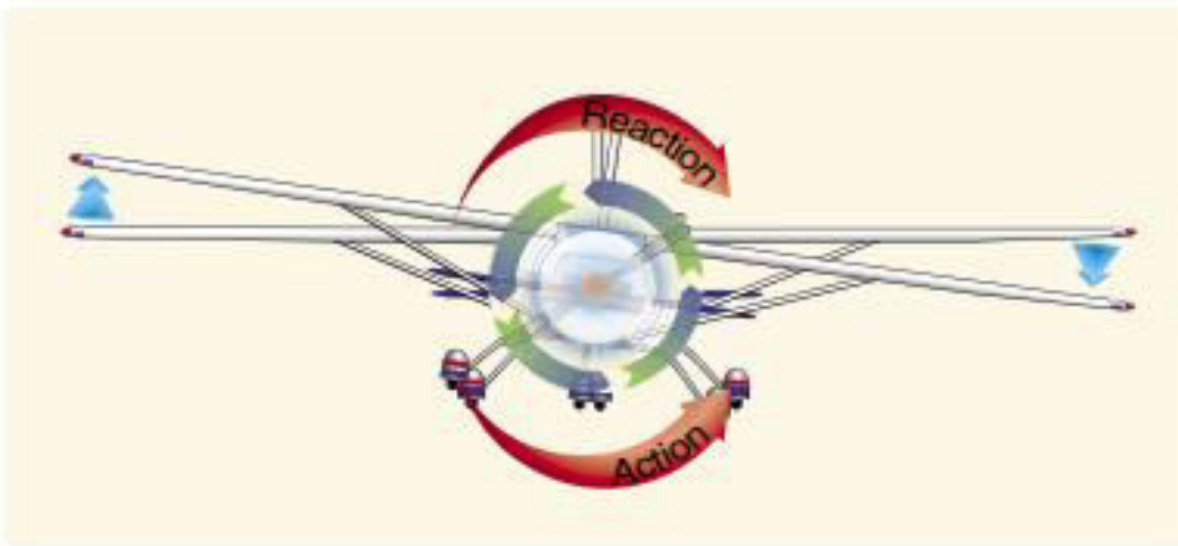


Figure 4-39. *Torque reaction.*

When the aircraft is airborne, this force is acting around the longitudinal axis, tending to make the aircraft roll. To compensate for roll tendency, some of the older aircraft are rigged in a manner to create more lift on the wing that is being forced downward.

The more modern aircraft are designed with the engine offset to counteract this effect of torque.

NOTE: Most United States built aircraft engines rotate the propeller clockwise, as viewed from the pilot's seat. The discussion here is with reference to those engines.

Generally, the compensating factors are permanently set so that they compensate for this force at cruising speed, since most of the aircraft's operating lift is at that speed. However, aileron trim tabs permit further adjustment for other speeds.

When the aircraft's wheels are on the ground during the takeoff roll, an additional turning moment around the vertical axis is induced by torque reaction.

As the left side of the aircraft is being forced down by torque reaction, more weight is being placed on the left main landing gear. This results in more ground friction, or drag, on the left tire than on the right, causing a further turning moment to the left.

The magnitude of this moment is dependent on many variables. Some of these variables are:

1. Size and horsepower of engine,
2. Size of propeller and the rpm,
3. Size of the aircraft, and
4. Condition of the ground surface.

This yawing moment on the takeoff roll is corrected by the pilot's proper use of the rudder or rudder trim

Corkscrew Effect

The high-speed rotation of an aircraft propeller gives a corkscrew or spiraling rotation to the slipstream.

Figure 4-40. *Corkscrewing slipstream.*

At high propeller speeds and low forward speed (as in the takeoffs and approaches to power-on stalls), this spiraling rotation is very compact and exerts a strong sideward force on the aircraft's vertical tail surface. [Figure 4-40]

When this spiraling slipstream strikes the vertical fin it causes a turning moment about the aircraft's vertical axis. The more compact the spiral, the more prominent this force is.

As the forward speed increases, however, the spiral elongates and becomes less effective.The corkscrew flow of the slipstream also causes a rolling moment around the longitudinal axis.

Note that this rolling moment caused by the corkscrew flow of the slipstream is to the right, while the rolling moment caused by torque reaction is to the left—in effect one may be counteracting the other.

However, these forces vary greatly and it is the pilot's responsibility to apply proper corrective action by use of the flight controls at all times. These forces must be counteracted regardless of which is the most prominent at the time.

Gyroscopic Action

Before the gyroscopic effects of the propeller can be understood, it is necessary to understand the basic principle of a gyroscope. All practical applications of the gyroscope are based upon two fundamental properties of gyroscopic action: rigidity in space and precession.

The one of interest for this discussion is precession. Precession is the resultant action, or deflection, of a spinning rotor when a deflecting force is applied to its rim. As can be seen in Figure 4-41, when a force is applied, the resulting force takes effect 90° ahead of and in the direction of rotation.

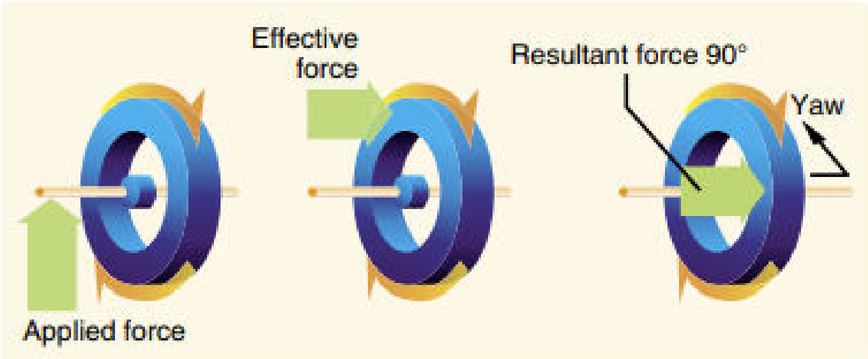


Figure 4-41. *Gyroscopic precession.*

The rotating propeller of an airplane makes a very good gyroscope and thus has similar properties.

Any time a force is applied to deflect the propeller out of its plane of rotation, the resulting force is 90° ahead of and in the direction of rotation and in the direction of application, causing a pitching moment, a yawing moment, or a combination of the two depending upon the point at which the force was applied.

This element of torque effect has always been associated with and considered more prominent in tailwheel-type aircraft, and most often occurs when the tail is being raised during the takeoff roll.

[Figure 4-42] This change in pitch attitude has the same effect as applying a force to the top of the propeller's plane of rotation. The resultant force acting 90° ahead causes a yawing moment to the left around the vertical axis.

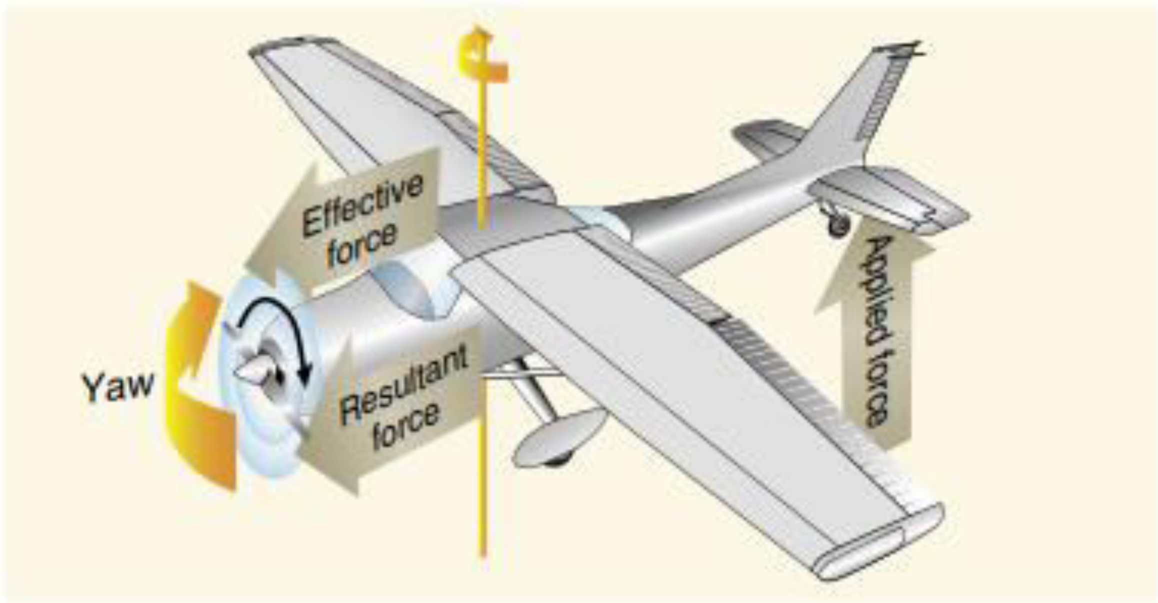


Figure 4-42. *Raising tail produces gyroscopic precession.*

The magnitude of this moment depends on several variables, one of which is the abruptness with which the tail is raised (amount of force applied).

However, precession, or gyroscopic action, occurs when a force is applied to any point on the rim of the propeller's plane of rotation;

the resultant force will still be 90° from the point of application in the direction of rotation. Depending on where the force is applied, the airplane is caused to yaw left or right, to pitch up or down, or a combination of pitching and yawing.

It can be said that, as a result of gyroscopic action, any yawing around the vertical axis results in a pitching moment, and any pitching around the lateral axis results in a yawing moment. To correct for the effect of gyroscopic action, it is necessary for the pilot to properly use elevator and rudder to prevent undesired pitching and yawing.

Asymmetric Loading (P-Factor)

When an aircraft is flying with a high AOA, the "bite" of the downward moving blade is greater than the "bite" of the upward moving blade.

This moves the center of thrust to the right of the prop disc area, causing a yawing moment toward the left around the vertical axis.

To prove this explanation is complex because it would be necessary to work wind vector problems on each blade while considering both the AOA of the aircraft and the AOA of each blade.

This asymmetric loading is caused by the resultant velocity, which is generated by the combination of the velocity of the propeller blade in its plane of rotation and the velocity of the air passing horizontally through the propeller disc.

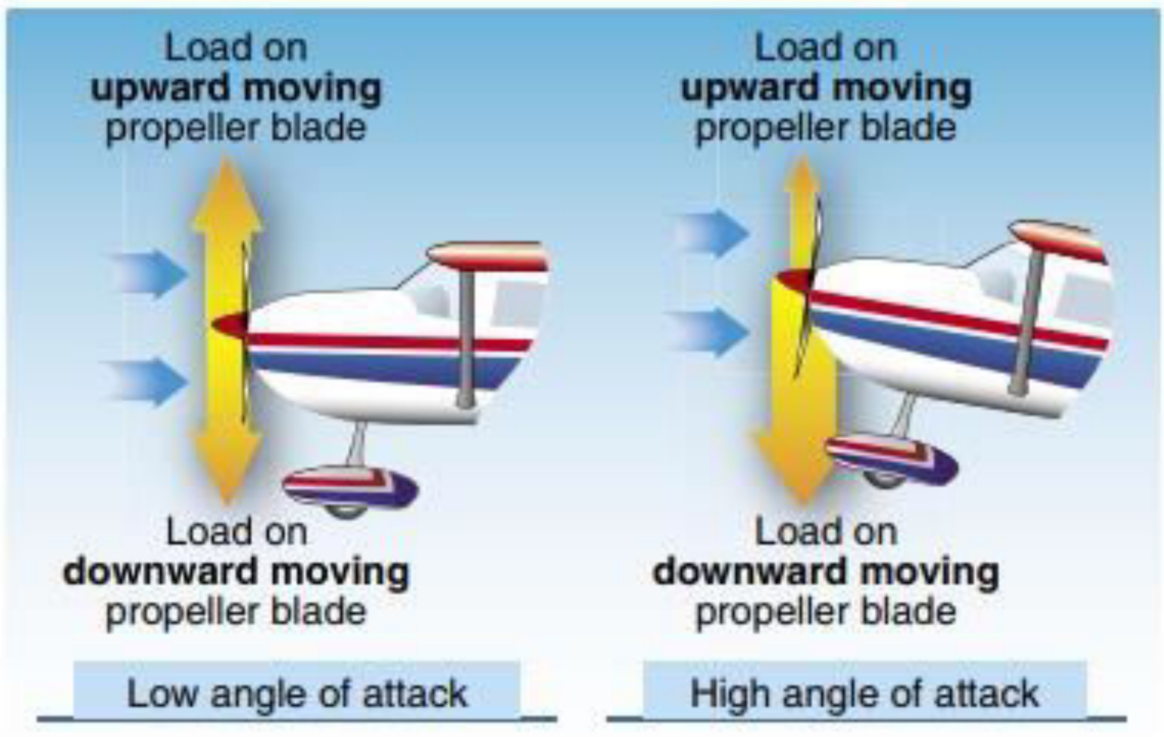


Figure 4-43. *Asymmetrical loading of propeller (P-factor).*

With the aircraft being flown at positive AOAs, the right (viewed from the rear) or downswinging blade, is passing through an area of resultant velocity which is greater than that affecting the left or upswinging blade.

Since the propeller blade is an airfoil, increased velocity means increased lift. The downswinging blade has more lift and tends to pull (yaw) the aircraft's nose to the left.

When the aircraft is flying at a high AOA, the downward moving blade has a higher resultant velocity, creating more lift than the upward moving blade.

[Figure 4-43] This might be easier to visualize if the propeller shaft was mounted perpendicular to the ground (like a helicopter).

If there were no air movement at all, except that generated by thepropeller itself, identical sections of each blade would have the same airspeed.

With air moving horizontally across this vertically mounted propeller, the blade proceeding forward into the flow of air has a higher airspeed than the blade retreating with the airflow.

Thus, the blade proceeding into the horizontal airflow is creating more lift, or thrust, moving the center of thrust toward that blade. Visualize rotating the vertically mounted propeller shaft to shallower angles relative to the moving air (as on an aircraft).

This unbalanced thrust then becomes proportionately smaller and continues getting smaller until it reaches the value of zero when the propeller shaft is exactly horizontal in relation to the moving air.

If there were no air movement at all, except that generated by thepropeller itself, identical sections of each blade would have the same airspeed.

With air moving horizontally across this vertically mounted propeller, the blade proceeding forward into the flow of air has a higher airspeed than the blade retreating with the airflow.

Thus, the blade proceeding into the horizontal airflow is creating more lift, or thrust, moving the center of thrust toward that blade. Visualize rotating the vertically mounted propeller shaft to shallower angles relative to the moving air (as on an aircraft).

This unbalanced thrust then becomes proportionately smaller and continues getting smaller until it reaches the value of zero when the propeller shaft is exactly horizontal in relation to the moving air.

Propeller

The propeller is a rotating airfoil, subject to induced drag, stalls, and other aerodynamic principles that apply to any airfoil. It provides the necessary thrust to pull, or in some cases push, the aircraft through the air.

The engine power is used to rotate the propeller, which in turn generates thrust very similar to the manner in which a wing produces lift.

The amount of thrust produced depends on the shape of the airfoil, the angle of attack of the propeller blade, and the revolutions per minute (rpm) of the engine. The propeller itself is twisted so the blade angle changes from hub to tip. The greatest angle of incidence, or the highest pitch, is at the hub while the smallest angle of incidence or smallest pitch is at the tip. [Figure 6-6]

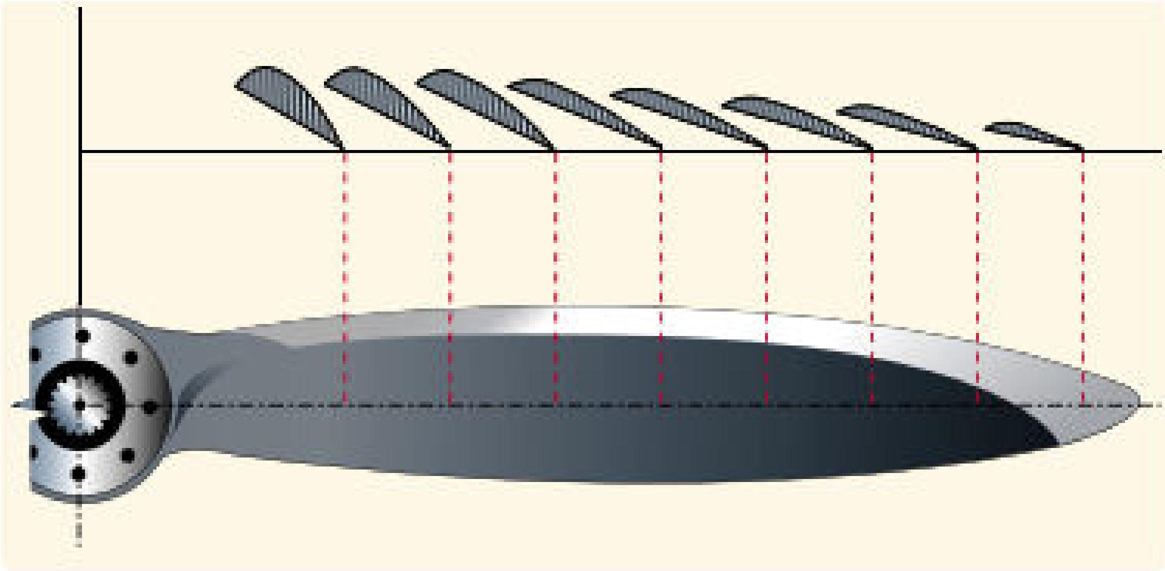


Figure 6-6. *Changes in propeller blade angle from hub to tip.*

The reason for the twist is to produce uniform lift from the hub to the tip. As the blade rotates, there is a difference in the actual speed of the various portions of the blade.

The tip of the blade travels faster than the part near the hub, because the tip travels a greater distance than the hub in the same length of time.

[Figure 6-7] Changing the angle of incidence (pitch) from the hub to the tip to correspond with the speed produces uniform lift throughout the length of the blade.

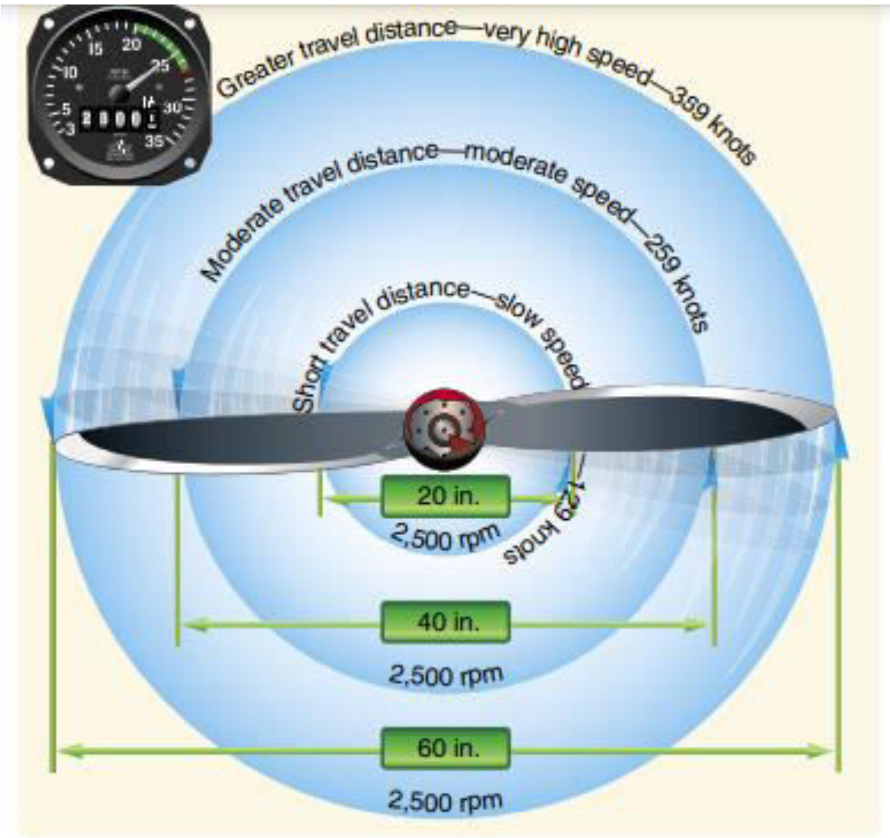


Figure 6-7. *Relationship of travel distance and speed of various portions of propeller blade.*

- A propeller blade designed with the same angle of incidence throughout its entire length would be inefficient because as airspeed increases in flight, the portion near the hub would have a negative angle of attack while the blade tip would be stalled.
- Small aircraft are equipped with either one of two types of propellers. One is the fixed pitch, and the other is the adjustable pitch.

Fixed-Pitch Propeller

- A propeller with fixed blade angles is a fixed-pitch propeller. The pitch of this propeller is set by the manufacturer and cannot be changed.
- Since a fixed-pitch propeller achieves the best efficiency only at a given combination of airspeed and rpm, the pitch setting is ideal for neither cruise nor climb.
- Thus, the aircraft suffers a bit in each performance category. The fixed-pitch propeller is used when low weight, simplicity, and low cost are needed.

- There are two types of fixed-pitch propellers: climb and cruise. Whether the airplane has a climb or cruise propeller installed depends upon its intended use. The climb propeller has a lower pitch, therefore less drag.
- Less drag results in higher rpm and more horsepower capability, which increases performance during takeoffs and climbs, but decreases performance during cruising flight.
- The cruise propeller has a higher pitch, therefore more drag. More drag results in lower rpm and less horsepower capability, which decreases performance during takeoffs and climbs, but increases efficiency during cruising flight.
- The cruise propeller has a higher pitch, therefore more drag. More drag results in lower rpm and less horsepower capability, which decreases performance during takeoffs and climbs, but increases efficiency during cruising flight.
- The propeller is usually mounted on a shaft, which may be an extension of the engine crankshaft. In this case, the rpm of the propeller would be the same as the crankshaft rpm. On some engines, the propeller is mounted on a shaft geared to the engine crankshaft. In this type, the rpm of the propeller is different than that of the engine.
- In a fixed-pitch propeller, the tachometer is the indicator of engine power. [Figure 6-8] A tachometer is calibrated in hundreds of rpm and gives a direct indication of the engine and propeller rpm.
- The instrument is color coded, with a green arc denoting the maximum continuous operating rpm. Some tachometers have additional markings to reflect engine and/or propeller limitations.
- The manufacturer's recommendations should be used as a reference to clarify any misunderstanding of tachometer markings.

Figure 6-8. *Engine rpm is indicated on the tachometer.*

- The rpm is regulated by the throttle, which controls the fuel/air flow to the engine. At a given altitude, the higher the tachometer reading, the higher the power output of the engine.
- When operating altitude increases, the tachometer may not show correct power output of the engine.
- For example, 2,300 rpm at 5,000 feet produces less horsepower than 2,300 rpm at sea level because power output depends on air density. Air density decreases as altitude increases and a decrease in air density (higher density altitude) decreases the power output of the engine.
- As altitude changes, the position of the throttle must be changed to maintain the same rpm. As altitude is increased, the throttle must be opened further to indicate the same rpm as at a lower altitude.

Adjustable-Pitch Propeller

- The adjustable-pitch propeller was the forerunner of the constant-speed propeller. It is a propeller with blades whose pitch can be adjusted on the ground with the engine not running, but which cannot be adjusted in flight.
- It is also referred to as a ground adjustable propeller. By the 1930s, pioneer aviation inventors were laying the ground work for automatic

pitch-change mechanisms, which is why the term sometimes refers to modern constant-speed propellers that are adjustable in flight.

- The first adjustable-pitch propeller systems provided only two pitch settings: low and high. Today, most adjustable-pitch propeller systems are capable of a range of pitch settings.
- A constant-speed propeller is a controllable-pitch propeller whose pitch is automatically varied in flight by a governor maintaining constant rpm despite varying air loads.
- It is the most common type of adjustable-pitch propeller. The main advantage of a constant-speed propeller is that it converts a high percentage of brake horsepower (BHP) into thrust horsepower (THP) over a wide range of rpm and airspeed combinations.
- A constant-speed propeller is more efficient than other propellers because it allows selection of the most efficient engine rpm for the given conditions. An aircraft with a constant-speed propeller has two controls: the throttle and the propeller control.
- The throttle controls power output and the propeller control regulates engine rpm. This in turn regulates propeller rpm which is registered on the tachometer.
- Once a specific rpm is selected, a governor automatically adjusts the propeller blade angle as necessary to maintain the selected rpm.
- For example, after setting the desired rpm during cruising flight, an increase in airspeed or decrease in propeller load will cause the propeller blade angle to increase as necessary to maintain the selected rpm.
- A reduction in airspeed or increase in propeller load will cause the propeller blade angle to decrease.
- The propeller’s constant-speed range, defined by the high and low pitch stops, is the range of possible blade angles for a constant-speed propeller. As long as the propeller blade angle is within the constant-speed range and not against either pitch stop, a constant engine rpm will be maintained.
- If the propeller blades contact a pitch stop, the engine rpm will increase or decrease as appropriate, with changes in airspeed and propeller load.
- For example, once a specific rpm has been selected, if aircraft speed decreases enough to rotate the propeller blades until they contact the low

pitch stop, any further decrease in airspeed will cause engine rpm to decrease the same way as if a fixed-pitch propeller were installed. The same holds true when an aircraft equipped with a constant-speed propeller accelerates to a faster airspeed.

- As the aircraft accelerates, the propeller blade angle increases to maintain the selected rpm until the high pitch stop is reached. Once this occurs, the blade angle cannot increase any further and engine rpm increases.
- On aircraft equipped with a constant-speed propeller, power output is controlled by the throttle and indicated by a manifold pressure gauge. The gauge measures the absolute pressure of the fuel/air mixture inside the intake manifold and is more correctly a measure of manifold absolute pressure (MAP).
- At a constant rpm and altitude, the amount of power produced is directly related to the fuel/air flow being delivered to the combustion chamber. As the throttle setting is increased, more fuel and air flows to the engine and MAP increases.
- When the engine is not running, the manifold pressure gauge indicates ambient air pressure (i.e., 29.92 inches mercury (29.92 "Hg)).
- When the engine is started, the manifold pressure indication will decrease to a value less than ambient pressure (i.e., idle at 12 "Hg).
- Engine failure or power loss is indicated on the manifold gauge as an increase in manifold pressure to a value corresponding to the ambient air pressure at the altitude where the failure occurred. [Figure 6-9]
- The manifold pressure gauge is color coded to indicate the engine’s operating range. The face of the manifold pressure gauge contains a green arc to show the normal operating range, and a red radial line to indicate the upper limit of manifold pressure.
- For any given rpm, there is a manifold pressure that should not be exceeded. If manifold pressure is excessive for a given rpm, the pressure within the cylinders could be exceeded, placing undue stress on the cylinders.
- If repeated too frequently, this stress can weaken the cylinder components and eventually cause engine failure. As a general rule, manifold pressure (inches) should be less than the rpm

Figure 6-9. *Engine power output is indicated on the manifold pressure gauge.*

- A pilot can avoid conditions that overstress the cylinders by being constantly aware of the rpm, especially when increasing the manifold pressure.
- Conform to the manufacturer's recommendations for power settings of a particular engine to maintain the proper relationship between manifold pressure and rpm.
- When both manifold pressure and rpm need to be changed, avoid engine overstress by making power adjustments in the proper order:
- When power settings are being decreased, reduce manifold pressure before reducing rpm. If rpm is reduced before manifold pressure, manifold pressure will automatically increase, possibly exceeding the manufacturer's tolerances.

- When power settings are being increased, reverse the order—increase rpm first, then manifold pressure.
- To prevent damage to radial engines, minimize operating time at maximum rpm and manifold pressure, and avoid operation at maximum rpm and low manifold pressure.
- The engine and/or airframe manufacturer's recommendations should be followed to prevent severe wear, fatigue, and damage to high-performance reciprocating engines.

SATHYABAMA
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)
Accredited "A" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

SCHOOL OF MECHANICAL ENGINEERING

DEPARTMENT OF AERONAUTICAL ENGINEERING

UNIT – V - BASIC AERODYNAMICS – SAE1304

I.INTRODUCTION TO TYPES OF FLUID FLOW

The fluid flow is classified as :

i. Steady and unsteady flows ;

ii. Uniform and non-uniform flows ;

iii. Laminar and turbulent flows ;

iv. Compressible and incompressible flows ;

v. Rotational and irrotational flows ; and

vi. One, two and three-dimensional flows.

Steady and Unsteady Flows.

Steady flow is defined as that type of flow in which the fluid characteristics like velocity, pressure, density, etc., at a point do not change with time. Thus for steady flow, mathematically, we have

$$\left(\frac{\partial V}{\partial t}\right)_{x_0,\, y_0,\, z_0} = 0,\ \left(\frac{\partial p}{\partial t}\right)_{x_0,\, y_0,\, z_0} = 0, \left(\frac{\partial \rho}{\partial t}\right)_{x_0,\, y_0,\, z_0} = 0$$

where (x0, *y 0, z0)* is a fixed point in fluid field.

Unsteady flow is that type of flow, in which the velocity, pressure or density at a point changes with respect to time. Thus, mathematically, for unsteady flow

$$\left(\frac{\partial V}{\partial t}\right)_{x_0,\, y_0,\, z_0} \neq 0,\ \left(\frac{\partial p}{\partial t}\right)_{x_0,\, y_0,\, z_0} \neq 0 \text{ etc.}$$

Uniform and Non-uniform Flows. Uniform flow is defined as that type of flow in which the velocity at any given time does not change with respect to space *(i.e.,* length of direction of the flow). Mathematically, for uniform flow

$$\left(\frac{\partial V}{\partial s}\right)_{t = \text{constant}} = 0$$

where ∂v = Change of velocity

∂s = Length of flow in the direction S.

Non-uniform flow is that type of flow in which the velocity at any given time changes with respect to space. Thus, mathematically, for non-uniform flow

$$\left(\frac{\partial V}{\partial s}\right)_{t = \text{constant}} \neq 0.$$

Laminar and Turbulent Flows.

Laminar flow is defined as that type of flow in which the fluid particles move along well-defined paths or stream line and all the stream-lines are straight and parallel.

Thus the particles move in laminas or layers gliding smoothly over the adjacent layer. This type of flow is also called stream-line flow or viscous flow.

Turbulent flow is that type of flow in which the fluid particles move in a *zig-zag* way. Due to the movement of fluid particles in a *zig-zag* way, the eddies formation takes place which are responsiblefor high energy loss. For a pipe flow, the type of flow is determined by a non-dimensional number $\frac{VD}{\vartheta}$ called the Reynold number,

where

D = Diameter of pipe

V = Mean velocity of flow in pipe and

v = Kinematic viscosity of fluid.

If the Reynold number is less than 2000, the flow is called laminar. If the Reynold number is more than 4000, it is called turbulent flow. If the Reynold number lies between 2000 and 4000, the flow may be laminar or turbulent.

Compressible and Incompressible Flows.

Compressible flow is that type of flow in which the density of the fluid changes from point to point or in other words the density (p) is not constant for the fluid. Thus, mathematically, for compressible flow

p ≠Constant

Incompressible flow is that type of flow in which the density is constant for the fluid flow. Liquids are generally incompressible while gases are compressible. Mathematically, for incompressible flow

p = Constant.

Rotational and lrrotational Flows. Rotational flow is that type of flow in which the fluid particles while flowing along stream-lines, also rotate about their own axis. And if the fluid particles while flowing along stream-lines, do not rotate about their own axis then that type of flow is called irrotational flow.

One-, Two- and Three-Dimensional Flows. One-dimensional flow is that type of flow in which the flow parameter such as velocity is a function of time and one space co-ordinate only, say *x.* For a steady one-dimensional flow, the velocity is a function of one-space-co-ordinate only. The variation of velocities in other two mutually perpendicular directions is assumed negligible. Hence mathematically, for one-dimensional flow

$u = f(x)$, $v = 0$ and $w = 0$

where *u, v* and w are velocity components in *x, y* and *z* directions respectively.

Two-dimensional flow is that type of flow in which the velocity is a function of time and two rectangular space co-ordinates say *x* and *y.* For a steady two-dimensional flow the velocity is a function of two space co-ordinates only. The variation of velocity in the third direction is negligible. Thus, mathematically for two-dimensional flow

$u = f_1(x, y)$, $v = f_2(x, y)$ and $w = 0$.

Three-dimensional flow is that type of flow in which the velocity is a function of time and three mutually perpendicular directions. But for a steady three-dimensional flow the fluid parameters are functions of three space co-ordinates (x, *y* and *z)* only. Thus, mathematically, for three-dimensional flow

$u = f_1(x, y, z)$, $v = f_2(x, y, z)$ and $w = f_3(x, y, z)$.

VELOCITY AND ACCELERATION

Let *V is* the resultant velocity at any point in a fluid flow. Let *u, v* and *w* are its component in *x, y* and *z* directions. The velocity components are functions of space-co-ordinates and time. Mathematically, the velocity components are given as

$$u = f_1(x, y, z, t)$$

$$v = f_2(x, y, z, t)$$

$$w = f_3(x, y, z, t)$$

and Resultant velocity,

$$V = u_i + v_j + w_k = \sqrt{u^2 + v^2 + w^2}$$

Let a_x, a_y and a_z are the **total acceleration** in *x, y* and *z* directions respectively. Then by the chain rule of differentiation, we have

$$a_x = \frac{du}{dt} = \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\cdot\frac{dz}{dt} + \frac{\partial u}{\partial t}$$

But $\quad \frac{dx}{dt} = u, \frac{dy}{dt} = v \text{ and} \frac{dz}{dt} = w$

$\therefore$

$$\left.\begin{aligned} a_x &= \frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t} \\ \text{Similarly,}\quad a_y &= \frac{dv}{dt} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t} \\ a_z &= \frac{dw}{dt} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t} \end{aligned}\right] \qquad \text{...(5.6)}$$

For steady flow, $\frac{\partial V}{\partial t} = 0$, where *V* is resultant velocity or $$\frac{\partial u}{\partial t} = 0, \frac{\partial v}{\partial t} = 0 \text{ and} \frac{\partial w}{\partial t} = 0$$

Hence acceleration in x, y and z directions becomes

$$\left.\begin{aligned} a_x &= \frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \\ a_y &= \frac{dv}{dt} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} \\ a_z &= \frac{dw}{dt} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} \end{aligned}\right] \quad \text{...(5.7)}$$

Acceleration vector $$\left.\begin{aligned} A &= a_x i + a_y j + a_z k \\ &= \sqrt{a_x^2 + a_y^2 + a_z^2}. \end{aligned}\right] \quad \text{...(5.8)}$$

5.7.1 Local Acceleration and Convective Acceleration. **Local acceleration** is defined as the rate of increase of velocity with respect to time at a given point in a flow field. In the equation given by (5.6), the expression $\frac{\partial u}{\partial t}, \frac{\partial v}{\partial t}$ or $\frac{\partial w}{\partial t}$ is known as local acceleration.

Convective acceleration is defined as the rate of change of velocity due to the change of position of fluid particles in a fluid flow. The expressions other than $\frac{\partial u}{\partial t}, \frac{\partial v}{\partial t}$ and $\frac{\partial w}{\partial t}$ in equation (5.6) are known as convective acceleration.

▶ 5.8 VELOCITY POTENTIAL FUNCTION AND STREAM FUNCTION

5.8.1 Velocity Potential Function. It is defined as a scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction. It is defined by ϕ (Phi). Mathematically, the velocity, potential is defined as $\phi = f(x, y, z)$ for steady flow such that

$$\left.\begin{aligned} u &= -\frac{\partial \phi}{\partial x} \\ v &= -\frac{\partial \phi}{\partial y} \\ w &= -\frac{\partial \phi}{\partial z} \end{aligned}\right\} \quad \text{...(5.9)}$$

where u, v and w are the components of velocity in x, y and z directions respectively.

The velocity components in cylindrical polar co-ordinates in terms of velocity potential function are given by

$$\left.\begin{aligned} u_r &= \frac{\partial \phi}{\partial r} \\ u_\theta &= \frac{1}{r}\frac{\partial \phi}{\partial \theta} \end{aligned}\right\} \quad \text{...(5.9A)}$$

where u_r = velocity component in radial direction (*i.e.*, in r direction)

and u_θ = velocity component in tangential direction (*i.e.*, in θ direction)

The continuity equation for an incompressible steady flow is $\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0.$

Substituting the values of u, v and w from equation (5.9), we get

$$\frac{\partial}{\partial x}\left(-\frac{\partial \phi}{\partial x}\right)+\frac{\partial}{\partial y}\left(-\frac{\partial \phi}{\partial y}\right)+\frac{\partial}{\partial z}\left(-\frac{\partial \phi}{\partial z}\right)=0$$

or

$$\frac{\partial^2 \phi}{\partial x^2}+\frac{\partial^2 \phi}{\partial y^2}+\frac{\partial^2 \phi}{\partial z^2}=0. \qquad ...(5.10)$$

Equation (5.10) is a Laplace equation.

For two-dimension case, equation (5.10) reduces to $\frac{\partial^2 \phi}{\partial x^2}+\frac{\partial^2 \phi}{\partial y^2}=0.$ ...(5.11)

If any value of ϕ that satisfies the Laplace equation, will correspond to some case of fluid flow.

Properties of the Potential Function. The rotational components* are given by

$$\omega_z=\frac{1}{2}\left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right)$$

$$\omega_y=\frac{1}{2}\left(\frac{\partial u}{\partial z}-\frac{\partial w}{\partial x}\right)$$

$$\omega_x=\frac{1}{2}\left(\frac{\partial w}{\partial y}-\frac{\partial v}{\partial z}\right)$$

Substituting the values, of u, v and w from equation (5.9) in the above rotational components, we get

$$\omega_z=\frac{1}{2}\left[\frac{\partial}{\partial x}\left(-\frac{\partial \phi}{\partial y}\right)-\frac{\partial}{\partial y}\left(-\frac{\partial \phi}{\partial x}\right)\right]=\frac{1}{2}\left[-\frac{\partial^2 \phi}{\partial x \partial y}+\frac{\partial^2 \phi}{\partial y \partial x}\right]$$

$$\omega_y=\frac{1}{2}\left[\frac{\partial}{\partial z}\left(-\frac{\partial \phi}{\partial x}\right)-\frac{\partial}{\partial x}\left(-\frac{\partial \phi}{\partial z}\right)\right]=\frac{1}{2}\left[-\frac{\partial^2 \phi}{\partial z \partial x}+\frac{\partial^2 \phi}{\partial x \partial z}\right]$$

and

$$\omega_x=\frac{1}{2}\left[\frac{\partial}{\partial y}\left(-\frac{\partial \phi}{\partial z}\right)-\frac{\partial}{\partial z}\left(-\frac{\partial \phi}{\partial y}\right)\right]=\frac{1}{2}\left[-\frac{\partial^2 \phi}{\partial y \partial z}+\frac{\partial^2 \phi}{\partial z \partial y}\right]$$

If ϕ is a continuous function, then $\frac{\partial^2 \phi}{\partial x \partial y}=\frac{\partial^2 \phi}{\partial y \partial x}$; $\frac{\partial^2 \phi}{\partial z \partial x}=\frac{\partial^2 \phi}{\partial x \partial z}$; etc.

If ϕ is a continuous function, then $\frac{\partial^2 \phi}{\partial x \partial y} = \frac{\partial^2 \phi}{\partial y \partial x}$; $\frac{\partial^2 \phi}{\partial z \partial x} = \frac{\partial^2 \phi}{\partial x \partial z}$; etc.

$$\therefore \qquad \omega_z = \omega_y = \omega_x = 0.$$

When rotational components are zero, the flow is called irrotational. Hence the properties of the potential function are :

1. If velocity potential (ϕ) exists, the flow should be irrotational.

2. If velocity potential (ϕ) satisfies the Laplace equation, it represents the possible steady incompressible irrotational flow.

5.8.2 Stream Function. It is defined as the scalar function of space and time, such that its partial derivative with respect to any direction gives the velocity component at right angles to that direction. It is denoted by ψ (*Psi*) and defined only for two-dimensional flow. Mathematically, for steady flow it is defined as $\psi = f(x, y)$ such that

$$\left.\begin{aligned} \frac{\partial \psi}{\partial x} &= v \\ \text{and} \qquad \frac{\partial \psi}{\partial y} &= -u \end{aligned}\right\} \qquad \ldots(5.12)$$

The velocity components in cylindrical polar co-ordinates in terms of stream function are given as

$$u_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \text{ and } u_\theta = -\frac{\partial \psi}{\partial r} \qquad \ldots(5.12A)$$

where u_r= radial velocity and u_θ = tangential velocity

The continuity equation for two-dimensional flow is $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0.$

Substituting the values of u and v from equation (5.12), we get

$$\frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial y}\right) + \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial x}\right) = 0 \text{ or } -\frac{\partial^2 \psi}{\partial x \partial y} + \frac{\partial^2 \psi}{\partial x \partial y} = 0.$$

Hence existence of ψ means a possible case of fluid flow. The flow may be rotational or irrotational.

The rotational component ω_z is given by $\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$.

Substituting the values of u and v from equation (5.12) in the above rotational component, we get

$$\omega_z = \frac{1}{2}\left[\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial x}\right) - \frac{\partial}{\partial y}\left(-\frac{\partial \psi}{\partial y}\right)\right] = \frac{1}{2}\left[\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2}\right]$$

For irrotational flow, $\omega_z = 0$. Hence above equation becomes as $\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial \psi}{\partial y^2} = 0$

which is Laplace equation for ψ.

The **properties** of stream function (ψ) are :

1. If stream function (ψ) exists, it is a possible case of fluid flow which may be rotational or irrotational.

2. If stream function (ψ) satisfies the Laplace equation, it is a possible case of an irrotational flow.

5.8.3 Equipotential Line. A line along which the velocity potential ϕ is constant, is called equipotential line.

For equipotential line $\quad \phi = \text{Constant}$

$\therefore \quad d\psi = 0$

But $\quad \phi = f(x, y)$ for steady flow

$$\therefore \quad d\phi = \frac{\partial \phi}{\partial x} dx + \frac{\partial \phi}{\partial y} dy$$

$$= -udx - vdy \qquad \left\{\because \frac{\partial \phi}{\partial x} = -u, \frac{\partial \phi}{\partial x} = -v\right\}$$

$$= -(udx + vdy).$$

For equipotential line, $\quad d\phi = 0$

or $\quad -(udx + vdy) = 0$ or $udx + vdy = 0$

$$\therefore \quad \frac{dy}{dx} = -\frac{u}{v} \qquad \text{...(5.13)}$$

But $\quad \dfrac{dy}{dx}$ = Slope of equipotential line.

5.8.4 Line of Constant Stream Function

$\psi = \text{Constant}$

$\therefore \quad d\psi = 0$

$$\text{But} \quad d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy = +vdx - udy \qquad \left\{\because \frac{\partial \psi}{\partial x} = v; \frac{\partial \psi}{\partial y} = -u\right\}$$

For a line of constant stream function

$= d\psi = 0$ or $vdx - udy = 0$

$$\text{or} \quad \frac{dy}{dx} = \frac{v}{u} \qquad \text{...(5.14)}$$

But $\dfrac{dy}{dx}$ is slope of stream line.

From equations (5.13) and (5.14) it is clear that the product of the slope of the equipotential line and the slope of the stream line at the point of intersection is equal to – 1. Thus the equipotential lines are othogonal to the stream lines at all points of intersection.

5.8.5 Flow Net. A grid obtained by drawing a series of equipotential lines and stream lines is called a flow net. The flow net is an important tool in analysing two-dimensional irrotational flow problems.

5.8.6 Relation between Stream Function and Velocity Potential Function

From equation (5.9),

we have $$u = -\frac{\partial \phi}{\partial x} \text{ and } v = -\frac{\partial \phi}{\partial y}$$

From equation (5.12), we have $u = -\frac{\partial \psi}{\partial y}$ and $v = \frac{\partial \psi}{\partial x}$

Thus, we have $$u = -\frac{\partial \phi}{\partial x} = -\frac{\partial \psi}{\partial y} \text{ and } v = -\frac{\partial \phi}{\partial y} = \frac{\partial \psi}{\partial x}$$

Hence
$$\left.\begin{aligned} \frac{\partial \phi}{\partial x} &= \frac{\partial \psi}{\partial y} \\ \text{and} \quad \frac{\partial \phi}{\partial y} &= -\frac{\partial \psi}{\partial x} \end{aligned}\right\} \qquad \ldots(5.15)$$

Problem 5.10 *The velocity potential function (ϕ) is given by an expression*

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3 y}{3} + y^2$$

(*i*) *Find the velocity components in x and y direction.*
(*ii*) *Show that ϕ represents a possible case of flow.*

Solution. Given : $$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3 y}{3} + y^2$$

The partial derivatives of ϕ w.r.t. x and y are

$$\frac{\partial \phi}{\partial x} = -\frac{y^3}{3} - 2x + \frac{3x^2 y}{3} \qquad \ldots(1)$$

and $$\frac{\partial \phi}{\partial y} = -\frac{3xy^2}{3} + \frac{x^3}{3} + 2y \qquad \ldots(2)$$

(*i*) The velocity components u and v are given by equation (5.9)

$$u = -\frac{\partial \phi}{\partial x} = -\left[-\frac{y^3}{3} - 2x + \frac{3x^2 y}{3}\right] = \frac{y^3}{3} + 2x - x^2 y$$

$\therefore$ $$u = \mathbf{\frac{y^2}{3} + 2x - x^2 y.\ Ans.}$$

$\therefore$ $$v = -\frac{\partial \phi}{\partial y} = -\left[-\frac{3xy^2}{3} + \frac{x^3}{3} + 2y\right] = \frac{3xy^2}{3} - \frac{x^3}{3} - 2y = \mathbf{xy^2 - \frac{x^3}{3} - 2y.}$$

Ans.

(*ii*) The given value of ϕ, will represent a possible case of flow if it satisfies the Laplace equation, *i.e.*,

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0$$

From equations (1) and (2), we have

Now $$\frac{\partial \phi}{\partial x} = -y^3/3 - 2x + x^2 y$$

$\therefore$ $$\frac{\partial^2 \phi}{\partial x^2} = -2 + 2xy$$

and $$\frac{\partial \phi}{\partial y} = -xy^2 + \frac{x^3}{3} + 2y$$

$\therefore$ $$\frac{\partial^2 \phi}{\partial y^2} = -2xy + 2$$

$\therefore$ $$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = (-2 + 2xy) + (-2xy + 2) = 0$$

$\therefore$ Laplace equation is satisfied and hence ϕ represent a possible case of flow. **Ans.**

IDEAL FLOW (POTENTIAL FLOW)

Ideal fluid is a fluid which is incompressible and inviscid. Incompressible fluid is a fluid for which density (p)remains constant. Inviscid fluid is a fluid for which viscosity (μ) is zero. Hence a fluid for which density is constant and viscosity is zero, is known as an ideal fluid.

The shear stress is given by, $= \mu \frac{\partial u}{\partial y}$. Hence for ideal fluid the shear stress will be zero as μ = 0 for ideal fluid. Also the shear force (which is equal to shear stress multiplied by area) will be zero in

case of ideal or potential flow. The ideal fluids will be moving with uniform velocity. All the fluid particles will be moving with the same velocity.

The concept of ideal fluid simplifies the typical mathematical analysis. Fluids such as water and air have low viscosity. Also when the speed of air is appreciably lower than that of sound in it, the compressibility is so low that air is assumed to be incompressible. Hence under certain conditions, certain real fluids such as water and air may be treated like ideal fluids.

IMPORTANT CASES OF POTENTIAL FLOW

The following are the important cases of potential flow :

i. Uniform flow,

ii. Source flow,

iii. Sink flow,

iv. Free-vortex flow,

v. Superimposed flow.

UNIFORM FLOW

In a uniform flow, the velocity remains constant. All the fluid particles are moving with the same velocity. The uniform flow may be :

(i) Parallel to x-axis *(ii)* Parallel to y-axis.

5.13.1 Uniform Flow Parallel to x-Axis. Fig. 5.27 (*a*) shows the uniform flow parallel to *x*-axis. In a uniform flow, the velocity remains constant. All the fluid particles are moving with the same velocity.

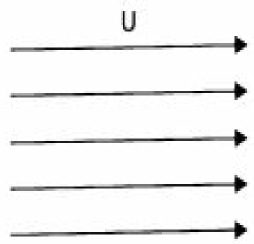


Fig. 5.27 (*a*)

Let U = Velocity which is uniform or constant along x-axis

u and v = Components of uniform velocity U along x and y-axis.

For the uniform flow, parallel to x-axis, the velocity components u and v are given as

$$u = U \text{ and } v = 0 \qquad \text{...(5.28)}$$

But the velocity u in terms of stream function is given by,

$$u = \frac{\partial \psi}{\partial y}$$

and in terms of velocity potential the velocity u is given by,

$$u = \frac{\partial \phi}{\partial x}$$

$$\therefore \quad u = \frac{\partial \psi}{\partial y} = \frac{\partial \phi}{\partial x} \quad \text{...(5.29)}$$

Similarly, it can be shown that $v = -\frac{\partial \psi}{\partial x} = \frac{\partial \phi}{\partial y}$ ...(5.29*A*)

But $u = U$ from equation (5.28). Substituting $u = U$ in equation (5.29), we have

$$\therefore \quad U = \frac{\partial \psi}{\partial y} = \frac{\partial \phi}{\partial x} \quad \text{...(5.30)}$$

or $$U = \frac{\partial \psi}{\partial y} \text{ and also } U = \frac{\partial \phi}{\partial x}$$

First part gives $d\psi = U\,dy$ whereas second part gives $d\phi = U\,dx$.

Integration of these parts gives as

$$\psi = Uy + C_1 \text{ and } \phi = Ux + C_2$$

where C_1 and C_2 are constant of integration.

Now let us plot the stream lines and potential lines for uniform flow parallel to *x*-axis.

Plotting of Stream lines. For stream lines, the equation is

$$\psi = U \times y + C_1$$

Let $\psi = 0$, where $y = 0$. Substituting these values in the above equation, we get

$$0 = U \times 0 + C_1 \text{ or } C_1 = 0$$

Hence the equation of stream lines becomes as

$$\psi = U.\,y \quad \text{...(5.31)}$$

The stream lines are straight lines parallel to *x*-axis and at a distance *y* from the *x*-axis as shown in Fig. 5.28. In equation (5.31), *U*. *y* represents the volume flow rate (*i.e.*, m^3/s) between *x*-axis and that stream line at a distance *y*.

Note. The thickness of the fluid stream perpendicular to the plane is assumed to be unity. Then $y \times 1$ or *y* represents the area of flow. And *U* . *y* represents the product of velocity and area. Hence *U*. *y* represents the volume flow rate.

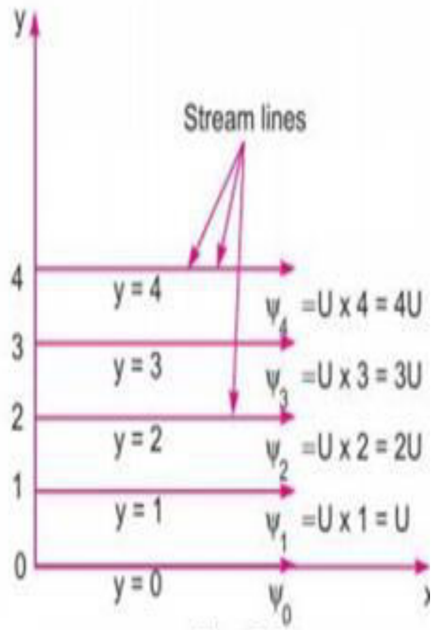


Fig. 5.28

Plotting of potential lines. For potential lines, the equation is

$$\phi = U \,.\, x + C_2 \qquad \ldots(5.32)$$

Let $\phi = 0$, where $x = 0$. Substituting these values in the above equation, we get $C_2 = 0$. Hence equation of potential lines becomes as

$$\phi = U.\, x$$

The above equation shows that potential lines are straight lines parallel to y-axis and at a distance of x from y-axis as shown in Fig. 5.29.

Fig. 5.30 shows the plot of stream lines and potential lines for uniform flow parallel to x-axis. The stream lines and potential lines intersect each other at right angles.

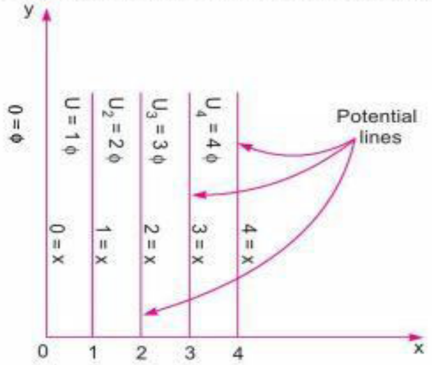


Fig. 5.29

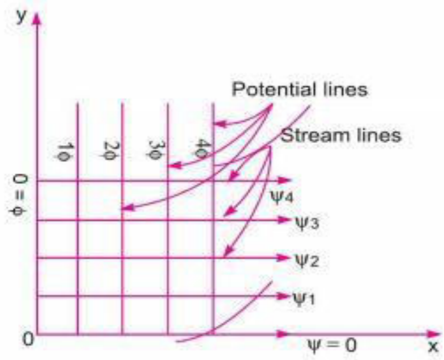


Fig. 5.30

5.13.2 Uniform Potential Flow Parallel to y-Axis.

Fig. 5.31 shows the uniform potential flow parallel to y-axis in which U is the uniform velocity along y-axis.

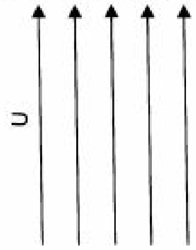


Fig. 5.31

The velocity components u, v along x-axis and y-axis are given by

$$u = 0 \text{ and } v = U \qquad \ldots(5.33)$$

These velocity components in terms of stream function (ψ) and velocity potential function (ϕ) are given as

$$u = \frac{\partial \psi}{\partial y} = \frac{\partial \phi}{\partial x} \qquad \ldots(5.34)$$

and

$$v = -\frac{\partial \psi}{\partial x} = \frac{\partial \phi}{\partial y} \qquad \ldots(5.35)$$

But from equation (5.33), $v = U$. Substituting $v = U$ in equation (5.35), we get

$$U = -\frac{\partial \psi}{\partial x} = \frac{\partial \phi}{\partial y} \quad \text{or} \quad U = -\frac{\partial \psi}{\partial x} \text{ and also } U = \frac{\partial \phi}{\partial y}$$

First part gives $d\psi = -U\, dx$ whereas second part gives $d\phi = U\, dy$.

Integration of these parts gives as

$$\psi = -U \,.\, x + C_1 \text{ and } \phi = U.y + C_2 \qquad \ldots(5.36)$$

where C_1 and C_2 are constant of integration. Let us now plot the stream lines and potential lines.

Plotting of Stream lines. For stream lines, the equation is $\psi = U.x + C_1$

Let $\psi = 0$, where $x = 0$. Then $C_1 = 0$.

Hence the equation of stream lines becomes as $\psi = -U.x$ ...(5.37)

The above equation shows that stream lines are straight lines parallel to y-axis and at a distance of x from the y-axis as shown in Fig. 5.32. The –ve sign shows that the stream lines are in the downward direction.

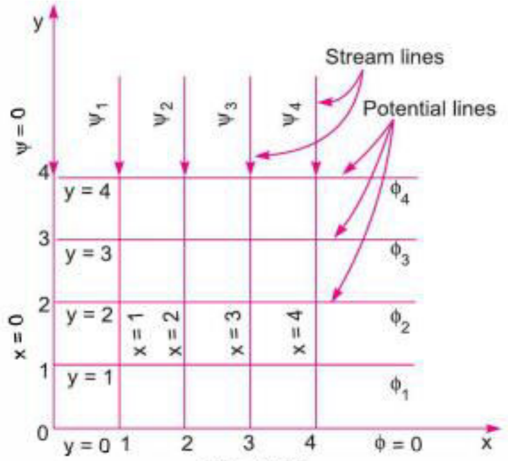


Fig. 5.32

Plotting of Potential lines. For potential lines, the equation is $\phi = U.y + C_2$

Let $\phi = 0$, where $y = 0$. Then $C_2 = 0$.

Hence equation of potential lines becomes as $\phi = U.y$ ...(5.38)

The above equation shows that potential lines are straight lines parallel to x-axis and at a distance of y from the x-axis as shown in Fig. 5.32.

▶ 5.14 SOURCE FLOW

The source flow is the flow coming from a point (source) and moving out radially in all directions of a plane at uniform rate. Fig. 5.33 shows a source flow in which the point O is the source from which the fluid moves radially outward. The strength of a source is defined as the volume flow rate per unit depth. The unit of strength of source is m^2/s. It is represented by q.

Let u_r = radial velocity of flow at a radius r from the source O

q = volume flow rate per unit depth

r = radius

The radial velocity u_r at any radius r is given by,

$$u_r = \frac{q}{2\pi r} \qquad ...(5.39)$$

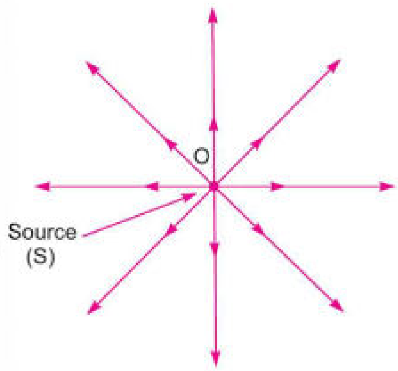


Fig. 5.33 *Source flow (Flow away from source)*

The above equation shows that with the increase of r, the radial velocity decreases. And at a large distance away from the source, the velocity will be approximately equal to zero. The flow is in radial direction, hence the tangential velocity $u_\theta = 0$.

Let us now find the equation of stream function and velocity potential function for the source flow. As in this case, $u_\theta = 0$, the equation of stream function and velocity potential function will be obtained from u_r.

Equation of Stream Function

By definition, the radial velocity and tangential velocity components in terms of stream function are given by

$$u_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \text{ and } u_\theta = -\frac{\partial \psi}{\partial r} \qquad \text{[See equation (5.12A)]}$$

But
$$u_r = \frac{q}{2\pi r} \qquad \text{[See equation (5.39)]}$$

$\therefore$
$$\frac{1}{r}\frac{\partial \psi}{\partial \theta} = \frac{q}{2\pi r}$$

or
$$d\psi = r.\frac{q}{2\pi r}.d\theta = \frac{q}{2\pi}\,d\theta$$

Integrating the above equation w.r.t. θ, we get

$$\psi = \frac{q}{2\pi} \times \theta + C_1, \text{ where } C_1 \text{ is constant of integration.}$$

Let $\psi = 0$, when $\theta = 0$, then $C_1 = 0$.

Hence the equation of stream function becomes as

$$\psi = \frac{q}{2\pi}.\theta \qquad \ldots(5.40)$$

In the above equation, q is constant.

The above equation shows that stream function is a function of θ. For a given value of θ, the stream function ψ will be constant. And this will be a radial line. The stream lines can be plotted by having different values of θ. Here θ is taken in radians.

Plotting of stream lines

When $\theta = 0, \psi = 0$

$$\theta = 45° = \frac{\pi}{4} \text{ radians, } \psi = \frac{q}{2\pi}.\frac{\pi}{4} = \frac{q}{8} \text{ units}$$

$$\theta = 90° = \frac{\pi}{2} \text{ radians, } \psi = \frac{q}{2\pi}.\frac{\pi}{2} = \frac{q}{4} \text{ units}$$

$$\theta = 135° = \frac{3\pi}{4} \text{ radians, } \psi = \frac{q}{2\pi}.\frac{3\pi}{4} = \frac{3q}{8} \text{ units}$$

The stream lines will be radial lines as shown in Fig. 5.34.

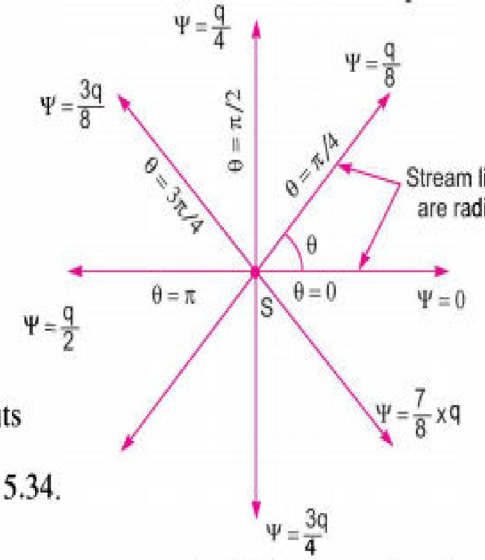


Fig. 5.34 *Stream line for source flow.*

Equation of Potential Function

By definition, the radial and tangential components in terms of velocity function are given by

$$u_r = \frac{\partial \phi}{\partial r} \text{ and } u_\theta = \frac{1}{r} \cdot \frac{\partial \phi}{\partial \theta} \qquad \text{[See equation (5.9A)]}$$

But from equation (5.39), $u_r = \dfrac{q}{2\pi r}$

Equating the two values of u_r, we get

$$\frac{\partial \phi}{\partial r} = \frac{q}{2\pi r} \text{ or } d\phi = \frac{q}{2\pi r}\, dr$$

Integrating the above equation, we get

$$\int d\phi = \int \frac{q}{2\pi r} . dr$$

or

$$\phi = \frac{q}{2\pi} \int \frac{1}{r}\, dr \quad \left[\because \frac{q}{2\pi} \text{ is a constant term}\right]$$

$$= \frac{q}{2\pi} \log_e r \qquad \text{...(5.41)}$$

In the above equation, q is constant.

The above equation shows, that the velocity potential function is a function of r. For a given value of r, the velocity function ϕ will be constant. Hence it will be a circle with origin at the source. The velocity potential lines will be circles with origin at the source as shown in Fig. 5.35.

Let us now find an expression for the pressure in terms of radius.

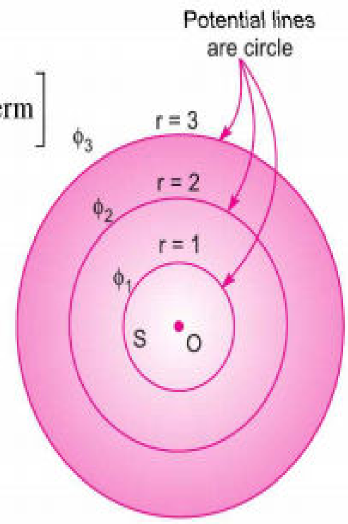


Fig. 5.35 *Potential lines for source.*

Pressure distribution in a plane source flow

The pressure distribution in a plane source flow can be obtained with the help of Bernoulli's equation. Let us assume that the plane of the flow is horizontal. In that case the datum head will be same for two points of flow.

Let p = pressure at a point 1 which is at a radius r from the source at point 1

u_r = velocity at point 1

p_0 = pressure at point 2, which is at a large distance away from the source. The velocity will be zero at point 2. [Refer to equation (5.39)]

Applying Bernoulli's equation, we get

$$\frac{p}{\rho g} + \frac{u_r^2}{2g} = \frac{p_0}{\rho g} + 0 \quad \text{or} \quad \frac{(p - p_0)}{\rho g} = -\frac{u_r^2}{2g}$$

or

$$(p - p_0) = -\frac{\rho . u_r^2}{2}$$

But from equation(5.39), $u_r = \dfrac{q}{2\pi r}$

Substituting the value of u_r in the above equation, we get

$$(p - p_0) = -\left(\frac{\rho}{2}\right) \cdot \left(\frac{q}{2\pi r}\right)^2$$

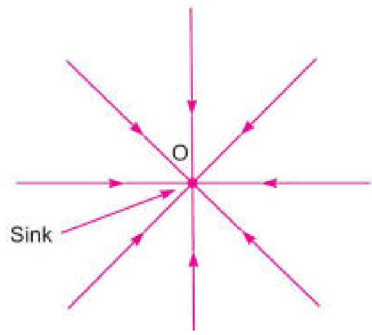
$$= -\frac{\rho q^2}{8\pi^2 r^2} \qquad \ldots(5.42)$$

In the above equation, ρ and q are constants.

The above equation shows that the pressure is inversely proportional to the square of the radius from the source.

▶ 5.15 SINK FLOW

The sink flow is the flow in which fluid moves radially inwards towards a point where it disappears at a constant rate. This flow is just opposite to the source flow. Fig. 5.36 shows a sink flow in which the fluid moves radially inwards towards point O, where it disappears at a constant rate. The pattern of stream lines and equipotential lines of a sink flow is the same as that of a source flow. All the equations derived for a source flow shall hold to good for sink flow also except that in sink flow equations, q is to be replaced by $(-q)$.



▶ 5.16 FREE-VORTEX FLOW

Free-vortex flow is a circulatory flow of a fluid such that its stream lines are concentric circles.

For a free-vortex flow, $u_\theta \times r$ = constant (say C)

Also, circulation around a stream line of an irrotation vortex is

$$\Gamma = 2\pi r \times u_\theta = 2\pi \times C \qquad (\because r \times u_\theta = C)$$

where u_θ = tangential velocity at any radius r from the centre.

\

$$\therefore \qquad u_\theta = \frac{\Gamma}{2\pi r}$$

The circulation Γ is taken positive if the free vortex is anticlockwise.

For a free-vortex flow, the velocity components are

$$u_\theta = \frac{\Gamma}{2\pi r} \quad \text{and} \quad u_r = 0$$

Equation of Stream Function

By definition, the stream function is given by

$$u_\theta = \frac{-\partial\psi}{\partial r} \quad \text{and} \quad u_r = \frac{1}{r}\frac{\partial\psi}{\partial\theta} \qquad \text{[See equation (5.12}A\text{)]}$$

In case of free-vortex flow, the radial velocity (u_r) is zero. Hence equation of stream function will be obtained from tangential velocity, u_θ. The value of u_θ is given by

$$u_\theta = \frac{\Gamma}{2\pi r}$$

Equating the two values of u_θ, we get

$$-\frac{\partial\psi}{\partial r} = \frac{\Gamma}{2\pi r} \quad \text{or} \quad d\psi = -\frac{\Gamma}{2\pi r}dr$$

Integrating the above equation, we get

$$\int d\psi = \int -\frac{\Gamma}{2\pi r}dr = \left(-\frac{\Gamma}{2\pi}\right)\int \frac{1}{r}dr$$

or

$$\psi = \left(-\frac{\Gamma}{2\pi}\right)\log_e r \qquad \left(\because \frac{\Gamma}{2\pi} \text{ is a constant term}\right) \quad \ldots(5.43)$$

The above equation shows that stream function is a function of radius. For a given value of r, the stream function is constant. Hence the stream lines are concentric circles as shown in Fig. 5.37.

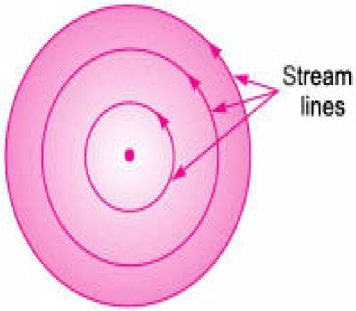


Fig. 5.37

Equation of potential function. By definition, the potential function is given by,

$$u_\theta = \frac{1}{r}\frac{\partial\phi}{\partial\theta} \quad \text{and} \quad u_r = \frac{\partial\phi}{\partial r} \qquad \text{[See equation (5.9}A\text{)]}$$

Here $u_r = 0$ and $u_\theta = \dfrac{\Gamma}{2\pi r}$. Hence, the equation of potential function will be obtained from u_θ.

Equating the two values of u_θ, we get

$$\frac{1}{r}\frac{\partial\phi}{\partial\theta} = \frac{\Gamma}{2\pi r} \quad \text{or} \quad d\phi = r.\frac{\Gamma}{2\pi r}.d\theta = \frac{\Gamma}{2\pi}d\theta$$

Integrating the above equation, we get

$$\int d\phi = \int \frac{\Gamma}{2\pi}d\theta \quad \text{or} \quad \phi = \frac{\Gamma}{2\pi}\int d\theta = \frac{\Gamma}{2\pi}.\theta \qquad \ldots(5.44)$$

The above equation shows that velocity potential function is a function of θ. For a given value of θ, potential function is a constant. Hence equipotential lines are radial as shown in Fig. 5.38.

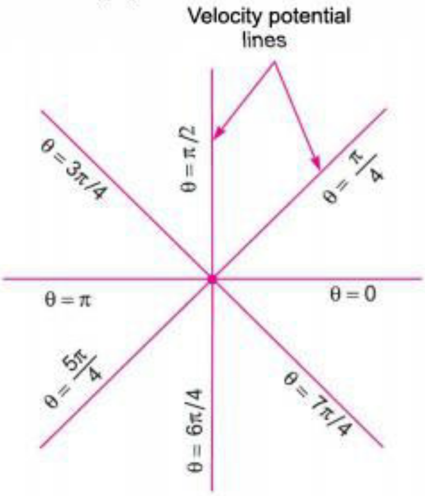


Fig. 5.38 *Potential lines are radial.*

▶ 5.17 SUPER-IMPOSED FLOW

The flow patterns due to uniform flow, a source flow, a sink flow and a free vortex flow can be super-imposed in any linear combination to get a resultant flow which closely resembles the flow around bodies. The resultant flow will still be potential and ideal. The following are the important super-imposed flow :

(*i*) Source and sink pair

(*ii*) Doublet (special case of source and sink combination)

(*iii*) A plane source in a uniform flow (flow past a half body)

(*iv*) A source and sink pair in a uniform flow

(*v*) A doublet in a uniform flow.

5.17.1 Source and Sink Pair. Fig. 5.39 shows a source and a sink of strength q and $(-q)$ placed at A and B respectively at equal distance from the point O on the x-axis. Thus the source and sink are placed symmetrically on the x-axis. The source of strength q is placed at A and sink of strength $(-q)$ is placed at B. The combination of the source and the sink would result in a flownet where stream lines will be circular arcs starting from point A and ending at point B as shown in Fig. 5.40.

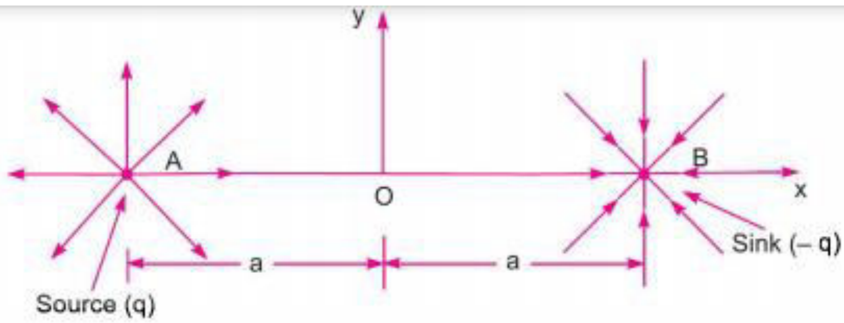


Fig. 5.39 *Source and sink pair.*

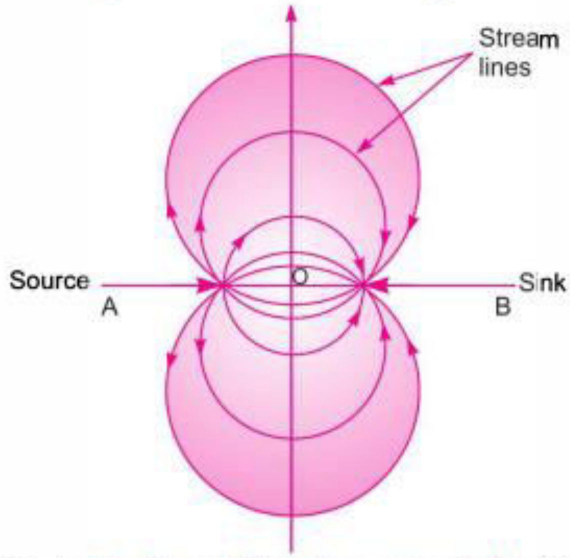


Fig. 5.40 *Stream lines for source-sink pair.*

Equation of stream function and potential function

Let P be any point in the resultant flownet of source and sink as shown in Fig. 5.41.

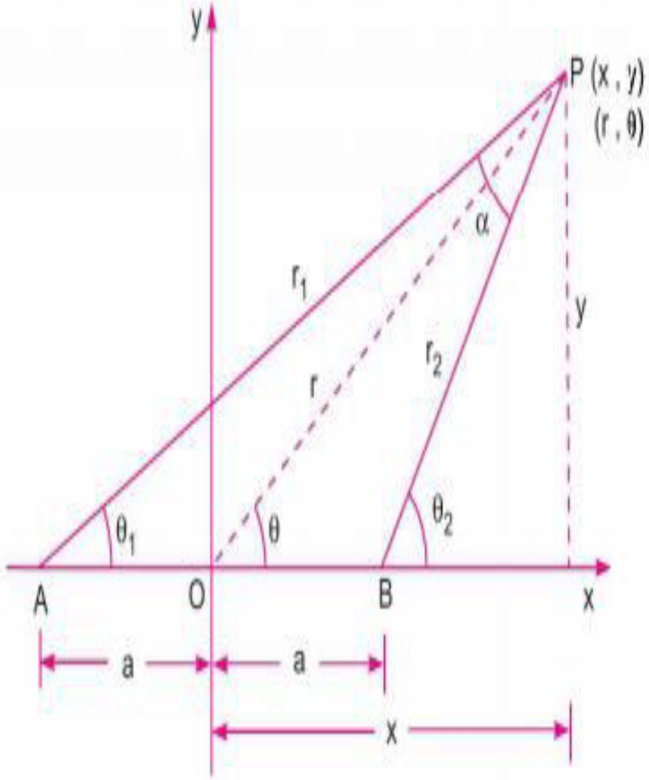


Fig. 5.41

Let r, θ = Cylindrical co-ordinates of point P with respect to origin O

x, y = Corresponding co-ordinates of point P

r_1, θ_1 = Position of point P with respect to source placed at A

r_2, θ_2 = Position of point P with respect to sink placed at B

α = Angle subtended at P by the join of source and sink *i.e.*, angle APB.

Let us find the equation for the resultant stream function and velocity potential function. The equation for stream function due to source is given by equation (5.40) as $\psi_1 = \frac{q \cdot \theta_1}{2\pi}$ whereas due to sink it is given by $\psi_2 = \frac{(-q\theta_2)}{2\pi}$. The equation for resultant stream function (ψ) will be the sum of these two stream function.

$$\therefore \quad \psi = \psi_1 + \psi_2$$

$$= \frac{q\theta_1}{2\pi} + \left(\frac{-q\theta_2}{2\pi}\right) = \frac{-q}{2\pi}(\theta_2 - \theta_1)$$

$$= \frac{-q}{2\pi} \cdot \alpha \quad [\because \quad \alpha = \theta_2 - \theta_1. \text{ In triangle } ABP,\ \theta_1 + \alpha + (180^\circ - \theta_2) = 180^\circ \quad \therefore \quad \alpha = \theta_2 - \theta_1]$$

$$= \frac{-q \cdot \alpha}{2\pi} \qquad \ldots(5.45)$$

The equation for potential function due to source is given by equation (5.41) as $\phi_1 = \frac{q}{2\pi} \log_e r_1$ and due to sink it is given as $\phi_2 = \frac{-q}{2\pi} \log_e r_2$. The equation for resultant potential function (ϕ) will be the sum of these two potential function.

$$\therefore \quad \phi = \phi_1 + \phi_2$$

$$= \frac{q}{2\pi} \log_e r_1 + \left(\frac{-q}{2\pi}\right) \log_e r_2$$

Now $$r = \sqrt{x^2 + y^2} = \sqrt{0.5^2 + 1^2} = \sqrt{1.25}$$

$$\therefore \quad \sin\theta = \frac{y}{r} = \frac{1}{\sqrt{1.25}} = 0.894 \text{ and } \cos\theta = \frac{x}{r} = \frac{0.5}{\sqrt{1.25}} = 0.447$$

Substituting the values of r, $\sin\theta$ and $\cos\theta$ in above equations (*i*), (*ii*) and (*iii*), we get

$$\psi = -\frac{\mu}{2\pi}\frac{\sin\theta}{r} = -\frac{5}{2\pi} \times \frac{0.894}{\sqrt{1.25}} = \mathbf{-0.636\ m^2/s.\ Ans.}$$

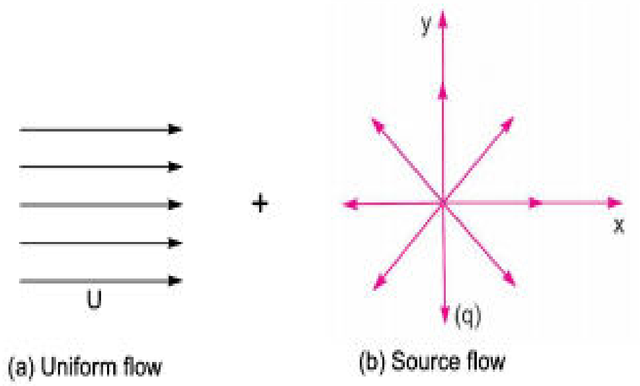
$$u_r = -\frac{\mu}{2\pi} \times \frac{1}{r^2} \times \cos\theta = -\frac{5}{2\pi} \times \frac{1}{(1.25)} \times 0.447 = -0.2845 \text{ m/s}$$

and $$u_\theta = -\frac{\mu}{2\pi} \times \frac{\sin\theta}{r^2} = -\frac{5}{2\pi} \times \frac{0.894}{1.25} = -0.569 \text{ m/s}$$

$\therefore$ Resultant velocity, $$V = \sqrt{u_r^2 + u_\theta^2}$$

$$= \sqrt{(-0.2845)^2 + (-0.569)^2} = \mathbf{0.636\ m/s.\ Ans.}$$

5.17.3 A Plane Source in a Uniform Flow (Flow Past a Half-Body). Fig. 5.46 (*a*) shows a uniform flow of velocity U and Fig. 5.46 (*b*) shows a source flow of strength q. When this uniform flow is flowing over the source flow, a resultant flow will be obtained as shown in Fig. 5.46. This resultant flow is also known as the flow past a half-body. Let the source is placed on the origin O. Consider a point $P(x, y)$ lying in the resultant flow field with polar co-ordinates r and θ as shown in Fig. 5.46.



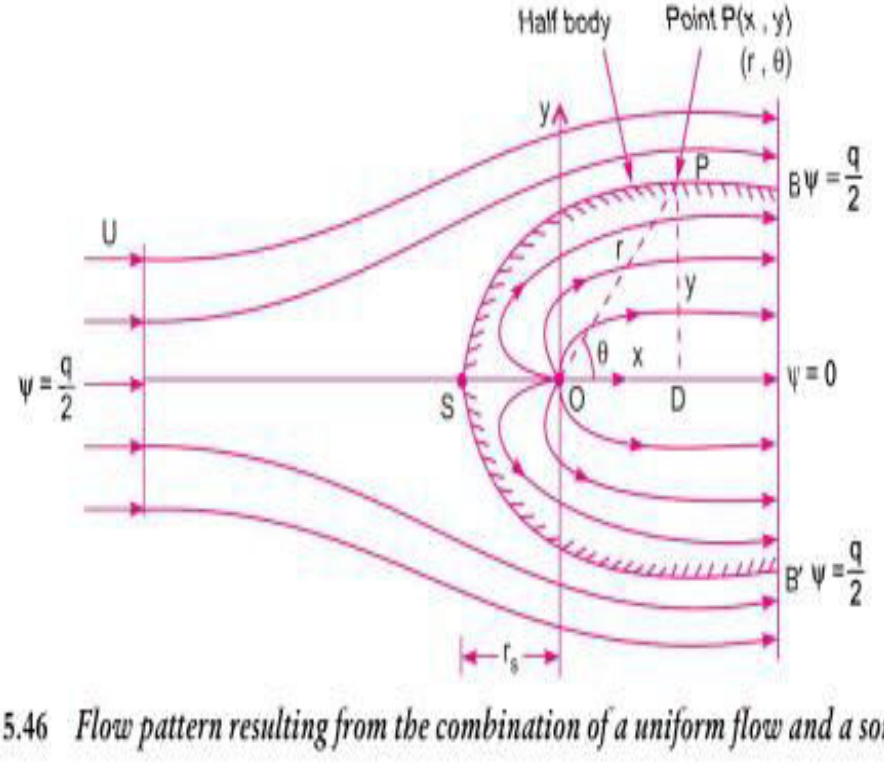


Fig. 5.46 *Flow pattern resulting from the combination of a uniform flow and a source.*

The stream function (ψ) and potential function (ϕ) for the resultant flow are obtained as given below :

ψ = Stream function due to uniform flow + stream function due to source

$$= U \cdot y + \frac{q}{2\pi}\theta \quad \text{...(5.54)}$$

$$= U \cdot r\sin\theta + \frac{q}{2\pi}\theta \quad (\because \; y = r\sin\theta) \text{ ...(5.54A)}$$

and ϕ = Velocity potential function due to uniform flow + Velocity potential function due to source

$$= U \cdot x + \frac{q}{2\pi}\log_e r = U \cdot r\cos\theta + \frac{q}{2\pi}\log_e r \quad \text{...(5.54B)}$$

The following are the important points for the resultant flow pattern :

(*i*) *Stagnation point.* On the left side of the source, at the point S lying on the x-axis, the velocity of uniform flow and that due to source are equal and opposite to each other. Hence the net velocity of the combined flow field is zero. This point is known as stagnation point and is denoted by S. The polar co-ordinates of the stagnation point S are r_S and π, where r_S is radial distance of point S from O.

The net velocity (or resultant velocity) is zero at the stagnation point S.

$$u_r = \frac{1}{r}\frac{\partial\psi}{\partial\theta} = \frac{1}{r}\frac{\partial}{\partial\theta}\left(U.r\sin\theta + \frac{q}{2\pi}\theta\right) \quad \left[\because \; \psi = U.r\sin\theta + \frac{q}{2\pi}\theta\right]$$

$$\therefore \quad = \frac{1}{r}\left[U.r\cos\theta + \frac{q}{2\pi}\right] = U \cdot \cos\theta + \frac{q}{2\pi r}$$

At the stagnation point, $\theta = \pi$ radians (180°) and $r = r_S$ and net velocity is zero. This means $u_r = 0$ and $v_\theta = 0$. Substituting these values in the above equation, we get

$$0 = U \cdot \cos 180° + \frac{q}{2\pi r_S} \quad [\because \; u_r = 0, \theta = 180° \text{ and } r = r_S]$$

$$= -U + \frac{q}{2\pi r_S} \quad \text{or} \quad U = \frac{q}{2\pi r_S}$$

or

$$r_S = \frac{q}{2\pi U} \quad \text{...(5.55)}$$

From the above equation it is clear that position of stagnation point depends upon the free stream velocity U and source strength q. At the stagnation point, the value of stream function is obtained from equation (5.54A) as

$$\psi = U \cdot r\sin\theta + \frac{q}{2\pi}\cdot\theta$$

For the stagnation point, the above equation becomes as

$$\therefore \quad \psi_s = U \cdot r_s \sin 180° + \frac{q}{2\pi}\times\theta$$

[$\because$ At stagnation point, $\theta = \pi$ radians = 180° and $r = r_S$]

$$= 0 + \frac{q}{2} = \frac{q}{2} \quad \text{...(5.56)}$$

The above relation gives the equation of stream line passing through stagnation point. We know that no fluid mass crosses a stream line. Hence a stream line is a *virtual solid surface.*

(*ii*) *Shape of resultant flow.* At the stagnation point S, the net velocity is zero. The fluid particles that issue from the source cannot proceed further to the left of stagnation point. They are carried along the contour BSB' that separates the source flow from uniform flow. The curve BSB' can be regarded as the **solid boundary** of a round nosed body such as a bridge pier around which the uniform flow is forced to pass. The contour BSB' is called the half body, because it has only the leading point, it trails to infinity at down stream end.

The value of stream function of the stream line passing through stagnation point S and passing over the solid boundary (*i.e.*, curve BSB') is $\psi_S = \frac{q}{2}$.

Thus the composite flow consists of :

(1) flow over a plane half-body (*i.e.*, flow over curve BSB') outside $\psi = \frac{q}{2}$ and

(2) source flow within the plane half-body.

The plane half-body is described by the dividing stream line, $\psi = \frac{q}{2}$.

But the stream function at any point in the combined flow field is given by equation (5.54) as

$$\psi = U \,.\, y + \frac{q}{2\pi}\,\theta$$

If we take $\psi = \frac{q}{2}$ in the above equation, we will get the equation of the dividing stream line.

$\therefore$ Equation of the dividing stream line (*i.e.*, equation of curve BSB') will be

$$\frac{q}{2} = U \,.\, y + \frac{q}{2\pi} \,.\, \theta \text{ or } U \,.\, y = \frac{q}{2} - \frac{q}{2\pi}\,\theta = \frac{q}{2}\left(1 - \frac{\theta}{\pi}\right)$$

or $$y = \frac{q}{2U}\left(1 - \frac{\theta}{\pi}\right) \qquad \ldots(5.57)$$

From the above equation, the main dimensions of the plane half-body may be obtained. From this equation, it is clear that y is maximum, when $\theta = 0$.

Hence At $\theta = 0$, y is maximum and $y_{\max} = \frac{q}{2U}$ ... the maximum ordinate

At $\theta = \frac{\pi}{2}$, $y = \frac{q}{2U}\left(1 - \frac{\pi}{2}\cdot\frac{1}{\pi}\right) = \frac{q}{4U}$ ... the ordinate above the origin

At $\theta = \pi$, $y = \frac{q}{2U}\left(1 - \frac{\pi}{\pi}\right) = 0$ ... the leading point of the half-body

At $\theta = \frac{3\pi}{2}$, $y = \frac{q}{2U}\left(1 - \frac{3\pi}{2\pi}\right) = -\frac{q}{4U}$ ... the ordinate below the origin.

The main dimensions are shown in Fig. 5.47.

(*iii*) *Resultant velocity at any point*

The velocity components at any point in the flow field are given by

$$u_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = \frac{1}{r}\frac{d}{d\theta}\left[U.r\sin\theta + \frac{q}{2\pi}\theta\right]$$

$$= \frac{1}{r}\left[U.r\cos\theta + \frac{q}{2\pi}\right] = U.\cos\theta + \frac{q}{2\pi r}$$

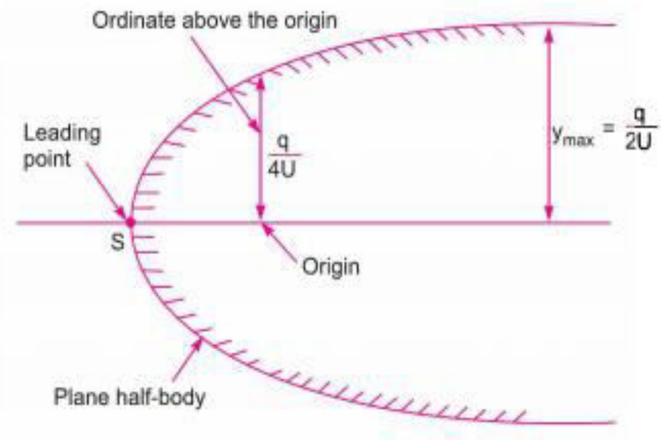


Fig. 5.47

The above equation gives the radial velocity at any point in the flow field. This radial velocity is due to uniform flow and due to source. Due to source the radial velocity is $\frac{q}{2\pi r}$. Hence the velocity due to source diminishes with increase in radial distance from the source. At large distance from the source the contribution of source is negligible and hence free stream uniform flow is not influenced by the presence of source.

$$u_\theta = -\frac{\partial\psi}{\partial r} = -\frac{\partial}{\partial r}\left[U.r\sin\theta + \frac{q}{2\pi}\theta\right]$$

$$= -[U.\sin\theta + 0] = -U\sin\theta \qquad \left[\because \frac{q}{2\pi}\theta \text{ is constant w.r.t. } r\right]$$

$\therefore$ Resultant velocity, $\quad V = \sqrt{u_r^2 + u_\theta^2}$

(*iv*) *Location of stagnation point*

At the stagnation point, the velocity components are zero. Hence equating the radial and tangential velocity components to zero, we get

$$u_r = 0 \quad \text{or} \quad U\cos\theta + \frac{q}{2\pi r} = 0 \quad \text{or} \quad U\cos\theta = -\frac{q}{2\pi r}$$

or $\quad r\cos\theta = -\frac{q}{2\pi U} \quad$ But $\quad r\cos\theta = x$

$\therefore \quad x = -\frac{q}{2\pi U}$

When $\quad u_\theta = 0 \quad$ or $\quad -U\sin\theta = 0 \quad$ or $\quad \sin\theta = 0 \quad$ as U cannot be zero

or $\quad \theta = 0 \quad$ or $\quad \pi$ But $y = r\sin\theta \quad \therefore\ y = 0$

Hence stagnation point is at $\left(-\frac{q}{2\pi U}, 0\right)$, the leading point of the half-body.

(*v*) *Pressure at any point in flow field*

Let p_0 = pressure at infinity where velocity is U

p = pressure at any point P in the flow field, where velocity is V

Now applying the Bernoulli's equation at a point at infinity and at a point P in the flow field, we get

$$\frac{p_0}{\rho g} + \frac{U^2}{2g} = \frac{p}{\rho g} + \frac{V^2}{2g} \quad \text{or} \quad \frac{U^2}{2g} - \frac{V^2}{2g} = \frac{p}{\rho g} - \frac{p_0}{\rho g} = \frac{p - p_0}{\rho g}$$

The pressure co-efficient is defined as

$$C_p = \frac{p - p_0}{\frac{1}{2}\rho U^2}$$

$$= \frac{\rho g \left[\frac{U^2}{2g} - \frac{V^2}{2g}\right]}{\frac{1}{2}\rho U^2} \qquad \left[\because \; p - p_0 = \rho g\left(\frac{U^2}{2g} - \frac{V^2}{2g}\right)\right]$$

$$= \frac{U^2 - V^2}{U^2} = 1 - \left(\frac{V}{U}\right)^2 \qquad \text{...(5.58)}$$

5.17.4 A Source and Sink Pair in a Uniform Flow (Flow Past a Rankine Oval Body). Fig. 5.51 (*a*) shows a uniform flow of velocity U and Fig. 5.51 (*b*) shows a source sink pair of equal strength. When this uniform flow is flowing over the source sink pair, a resultant flow will be obtained as shown in Fig. 5.51 (*c*). This resultant flow is also known as the flow past a Rankine oval body.

Let U = Velocity of uniform flow along x-axis

q = Strength of source

$(-q)$ = Strength of sink

$2a$ = Distance between source and sink which is along x-axis.

The origin O of the x-y co-ordinates is mid-way between source and sink. Consider a point $P(x, y)$ lying in the resultant flow field. The stream function (ψ) and velocity potential function (ϕ) for the resultant flow field are obtained as given below :

ψ = Stream function due to uniform flow + stream function due to source + stream function due to sink

$$= \psi_{\text{uniform flow}} + \psi_{\text{source}} + \psi_{\text{sink}}$$

$$= U \times y + \frac{q}{2\pi}\theta_1 + \frac{(-q)}{2\pi} \times \theta_2$$

(where θ_1 is the angle made by P with source along x-axis and θ_2 with sink)

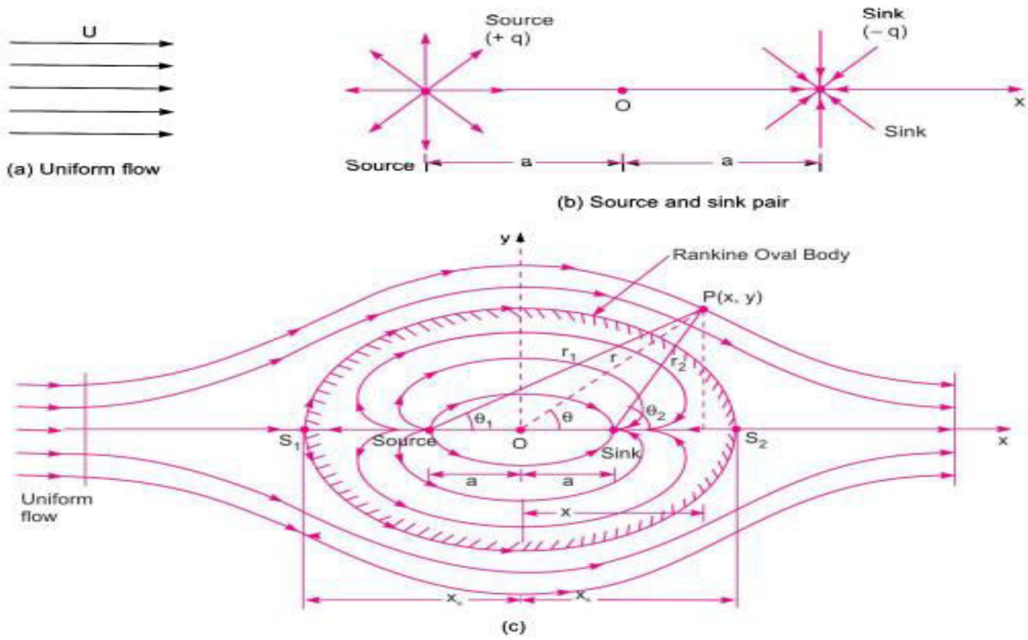


Fig. 5.51

$$= U \times y + \frac{q\theta_1}{2\pi} - \frac{q\theta_2}{2\pi} = U \times y + \frac{q}{2\pi}(\theta_1 - \theta_2)$$

$$= U \times r \sin\theta + \frac{q}{2\pi}(\theta_1 - \theta_2) \qquad (\because \quad y = r\sin\theta) \quad ...(5.59)$$

and ϕ = potential function due to uniform flow + potential function due to source + potential function due to sink

$$= \phi_{\text{uniform flow}} + \phi_{\text{source}} + \phi_{\text{sink}}$$

$$= U \times x + \frac{q}{2\pi}\log_e r_1 + \frac{(-q)}{2\pi}\log_e r_2$$

$$= U \times r\cos\theta + \frac{q}{2\pi}[\log_e r_1 - \log_e r_2] \qquad (\because \quad x = r\cos\theta)$$

$$= U \times r\cos\theta + \frac{q}{2\pi}\left[\log_e \frac{r_1}{r_2}\right] \quad ...(5.60)$$

The following are the important points for the resultant flow pattern :

(*a*) There will be two stagnation points S_1 and S_2, one to the left of the source and other to the right of the sink. At the stagnation points, the resultant velocity (*i.e.*, velocity due to uniform flow, velocity due to source and velocity due to sink) will be zero. The stagnation point S_1 is to the left of the source and stagnation point S_2 will be to the right of the sink on the x-axis.

Let x_S = Distance of the stagnation points from origin O along x-axis.

Let us calculate this distance x_S.

For the stagnation point S_1,

(*i*) Velocity due to uniform flow = U

(*ii*) Velocity due to source = $\dfrac{q}{2\pi(x_S - a)}$ $\left[\begin{array}{l}\because \text{ The velocity at any radius due to source} = \dfrac{q}{2\pi r} \\ \text{For } S_1\text{, the radius from source} = (x_S - a)\end{array}\right]$

(*iii*) Velocity due to sink $= \dfrac{-q}{2\pi(x_S + a)}$ [$\because$ At S_1, the radius from sink $= (x_S + a)$]

At point S_1, the velocity due to uniform flow is in the positive x-direction whereas due to source and sink are in the –ve x-direction.

$\therefore$ The resultant velocity at $S_1 = U - \dfrac{q}{2\pi(x_S - a)} - \dfrac{(-q)}{2\pi(x_S + a)}$

But the resultant velocity at stagnation point S_1 should be zero.

$\therefore$
$$U - \frac{q}{2\pi(x_S - a)} + \frac{q}{2\pi(x_S + a)} = 0$$

or
$$U = \frac{q}{2\pi(x_S - a)} - \frac{q}{2\pi(x_S + a)}$$

$$= \frac{q}{2\pi}\left[\frac{1}{(x_S - a)} - \frac{1}{(x_S + a)}\right] = \frac{q}{2\pi}\left[\frac{(x_S + a) - (x_S - a)}{(x_S - a)(x_S + a)}\right] = \frac{q}{2\pi}\frac{2a}{\left(x_S^2 - a^2\right)}$$

or
$$x_S^2 - a^2 = \frac{q\,.\,a}{\pi U}$$

or
$$x_S^2 = a^2 + \frac{qa}{\pi U} = a^2\left[1 + \frac{q}{\pi a U}\right]$$

$\therefore$
$$x_S = a\sqrt{\left(1 + \frac{q}{\pi a U}\right)} \qquad \text{...(5.61)}$$

The above equation gives the location of the stagnation point on the x-axis.

(*b*) The stream line passing through the stagnation points is having zero velocity and hence can be replaced by a solid body. This solid body is having a shape of oval as shown in Fig. 5.51. There will be two flow fields, one within the oval contour and the other outside the solid body. The flow field within the oval contour will be due to source and sink whereas the flow field outside the body will be due to uniform flow only.

The shape of solid body is obtained from the stream line having stream function equal to zero. But the stream function is given by equation as

$$\psi = U \times r \sin\theta + \frac{q}{2\pi}(\theta_1 - \theta_2)$$

For the shape of solid body, $\psi = 0$

$\therefore$
$$0 = U \times r \sin\theta + \frac{q}{2\pi}(\theta_1 - \theta_2)$$

or
$$U \times r \sin\theta = -\frac{q}{2\pi}(\theta_1 - \theta_2) = \frac{q}{2\pi}(\theta_2 - \theta_1)$$

$\therefore$
$$r = \frac{q}{2\pi}\frac{(\theta_2 - \theta_1)}{U \sin\theta} \qquad \text{...(5.62)}$$

From the above equation, the distances of the surface of the solid body from the origin can be obtained or the shape of the solid body can be obtained. The maximum width of the body (y_{max}) will be equal to OM as shown in Fig. 5.52.

From triangle AOM, we have

$$\tan \theta_1 = \frac{OM}{AO}$$

or $$OM = AO \tan \theta_1 = a \tan \theta_1$$

or $$y_{max} = a \tan \theta_1 \qquad (\because\ OM = y_{max}) \quad ...(5.63)$$

Let us find the value of θ_1.

When the point P lies on M, then $r = OM$, $\theta = 90° = \frac{\pi}{2}$

and $$\theta_2 = 180° - \theta_1 = \pi - \theta_1 \qquad \text{[Refer to Fig. 5.52]}$$

$$[\because\ AM = BM \quad \therefore \quad \text{Angle } ABM = \text{Angle } BAM = \theta_1]$$

Substituting these values in equation (5.62), we get

$$OM = \frac{q}{2\pi} \frac{((\pi - \theta_1) - \theta_1)}{U \sin \frac{\pi}{2}} = \frac{q}{2\pi} \frac{(\pi - 2\theta_1)}{U}$$

or $$y_{max} = \frac{q(\pi - 2\theta_1)}{2\pi U} \qquad \text{[where } OM = y_{max}\text{]}$$

or $$2\pi U y_{max} = q(\pi - 2\theta_1) \quad \text{or} \quad \frac{2\pi U y_{max}}{q} = \pi - 2\theta_1$$

or $$2\theta_1 = \pi - \frac{2\pi U y_{max}}{q} \quad \text{or} \quad \theta_1 = \frac{\pi}{2} - \frac{\pi U y_{max}}{q}$$

Substituting this value of θ_1 in equation (5.63), we get

$$y_{max} = a \tan \left[\frac{\pi}{2} - \frac{\pi U y_{max}}{q}\right] = a \cot \left[\frac{\pi U y_{max}}{q}\right] \quad ...(5.64)$$

From the above equation, the value of y_{max} is obtained by hit and trial method till L.H.S. = R.H.S. In this equation $\left(\frac{\pi U y_{max}}{q}\right)$ is in radians.

The length and width of the Rankine oval is obtained as :

Length, $$L = 2 \times x_S$$

$$= 2 \times a \sqrt{\left(1 + \frac{q}{\pi a U}\right)} \qquad \left[\because\ x_S = a\sqrt{\left(1 + \frac{q}{\pi a U}\right)}\right] \quad ...(5.65)$$

and Width, $$B = 2 \times y_{max}$$

$$= 2a \cot \left(\frac{\pi U y_{max}}{q}\right). \quad ...(5.66)$$

PATHLINES, STREAMLINES, AND STREAKLINES OF A FLOWE

Consider an unsteady flow with a velocity field given by V = V(x, y, z, t). Also, consider an infinitesimal fluid element moving through the flow field, say, element A as shown in Figure 2.27a.

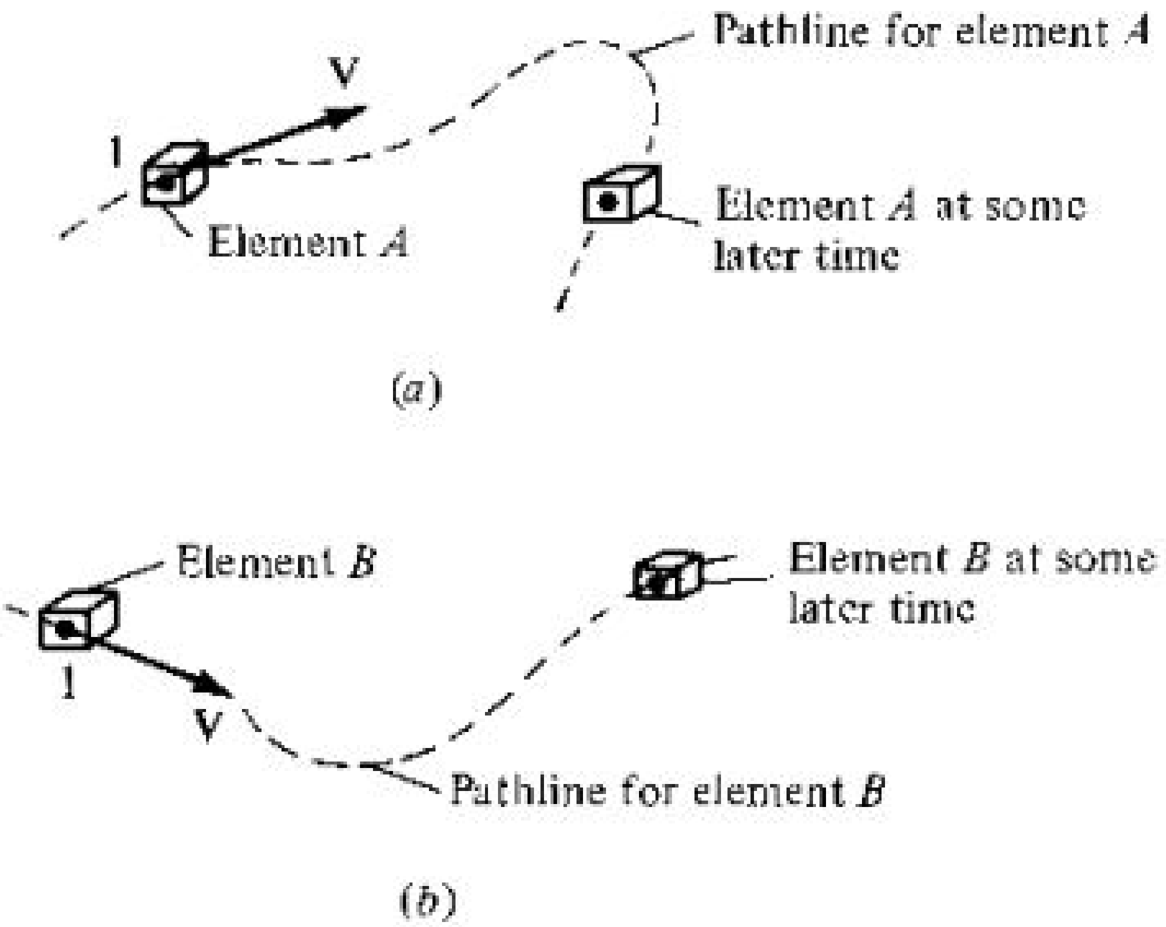


Figure 6. Pathlines for two different fluid elements passing through the same point in space: unsteady flow

Element A passes through point 1. Let ustrace the path of element A as it moves downstream from point 1, as given by the dashed line in Figure 6 a.

Such a path is defined as the pathline for element A. Now, trace the path of another fluid element, say, element B as shown in Figure 6 b. Assume that element B also passes through point 1, but at some different time from element A.

The pathline of element B is given by the dashed line in Figure 6 b. Because the flow is unsteady, the velocity at point 1 (and at all other points of the flow) changes with time.

Hence, the pathlines of elements A and B are different curves in Figure 6 a and b. In general, for unsteady flow, the pathlines for different fluid elements passing through the same point are not the same.

The concept of a streamline was introduced in a somewhat heuristic manner. Let us be more precise here. By definition, a streamline is a curve whose tangent at any point is in the direction of the velocity vector at that point. Streamlines are illustrated in Figure 7.

The streamlines are drawn such that their tangents at every point along the streamline are in the same direction as the velocity vectors at those points. If the flow is unsteady, the streamline pattern is different at different times because the velocity vectors are fluctuating with time in both magnitude and direction.

In general, streamlines are different from pathlines. You can visualize a pathline as a time-exposure photograph of a given fluid element, whereas a streamline pattern is like a single frame of a motion picture of the flow. In an unsteady flow, the streamline pattern changes; hence, each "frame" of the motion picture is different.

However, for the case ofsteady flow(which applies to most of the applications in this book), the magnitude and direction of the velocity vectors at all points are fixed, invariant with time.

Hence, the pathlines for different fluid elements going through the same point are the same. Moreover, the pathlines and streamlines are identical. Therefore, in steady flow, there is no distinction between pathlines and streamlines; they are the same curves in space. This fact is reinforced in Figure 8, which illustrates the fixed, time-invariant streamline (pathline) through point 1.

In Figure 8, a given fluid element passing through point 1 traces a pathline downstream. All subsequent fluid elements passing through point 1 at later times trace the same pathline. Since the velocity vector is tangent to the pathline at all points on the pathline for all times, the pathline is also a streamline.

For the remainder of this book, we deal mainly with the concept of streamlines rather than pathlines; however, always keep in mind the distinction described above.

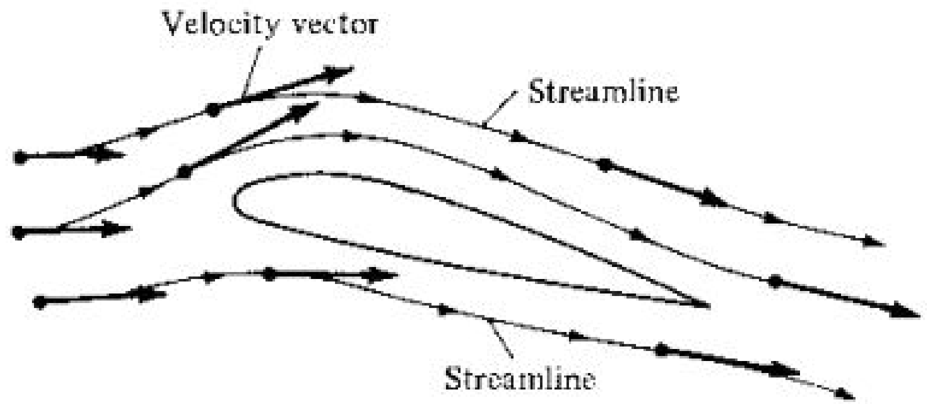


Figure 7 Streamlines.

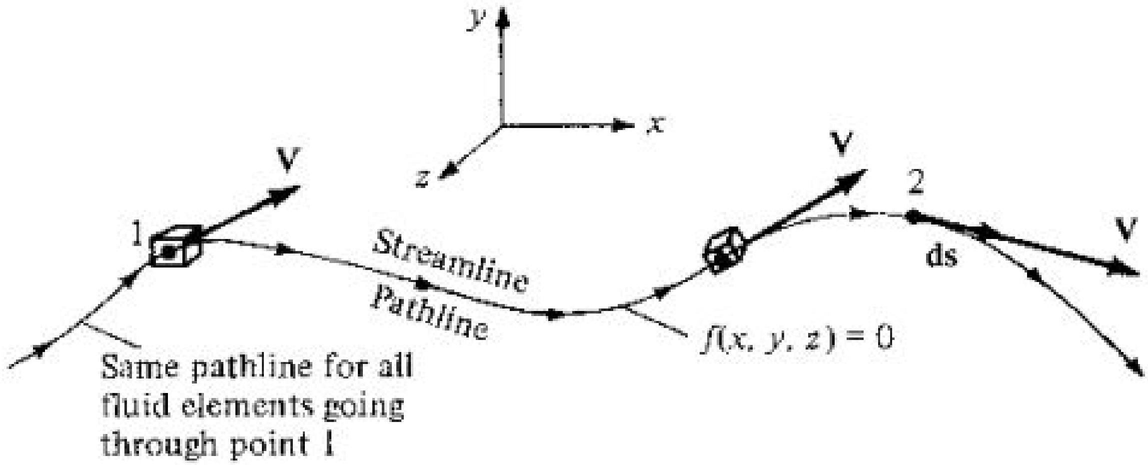


Figure 8. For steady flow, streamlines and pathlines are the same

Question: Given the velocity field of a flow, how can we obtain the mathematical equation for a streamline? Obviously, the streamline illustrated in Figure 8 is a curve in space, and hence it can be described by the equation f (x, y, z) = 0. How can we obtain this equation?

To answer this question, let ds be a directed element of the streamline, such as shown at point 2 in Figure 8.

The velocity at point 2 is V, and by definition of a streamline, V is parallel to ds. Hence, from the definition of the vector cross product [see Equation (2.4)],

$$ds \times V = 0 \qquad \text{----- } 2.115$$

Equation (2.115) is a valid equation for a streamline. To put it in a more recognizable form, expand Equation (2.115) in cartesian coordinates:

$$ds = dxi + dyj + dzk$$

V = ui + vj + wk

$$\mathbf{ds} \times \mathbf{V} = \begin{vmatrix} \mathbf{i} & \mathbf{j} & \mathbf{k} \\ dx & dy & dz \\ u & v & w \end{vmatrix}$$

$$= \mathbf{i}(w\,dy - v\,dz) + \mathbf{j}(u\,dz - w\,dx) + \mathbf{k}(v\,dx - u\,dy) = 0 \qquad (2.116)$$

Since the vector given by Equation (2.116) is zero, its components must each be zero:

$$\boxed{\begin{aligned} w\,dy - v\,dz &= 0 \\ u\,dz - w\,dx &= 0 \\ v\,dx - u\,dy &= 0 \end{aligned}} \qquad \begin{aligned} &(2.117a) \\ &(2.117b) \\ &(2.117c) \end{aligned}$$

Equations (2.117a to c) are differential equations for the streamline. Knowing u, v, and w as functions of x, y, and z, Equations (2.117a to c) can be integrated to yield the equation for the streamline: f (x, y, z) = 0. To reinforce the physical meaning of Equations (2.117a to c), consider a streamline in two dimensions, as sketched in Figure 9.

The equation of this streamline is y = f (x). Hence, at point 1 on the streamline, the slope is dy/dx. However, V with x and y components u and v, respectively, is tangent to the streamline at point 1.Thus, the slope of the streamline is also given by v/u, as shown in Figure 9. Therefore,

$$\boxed{\frac{dy}{dx} = \frac{v}{u}} \qquad (2.118)$$

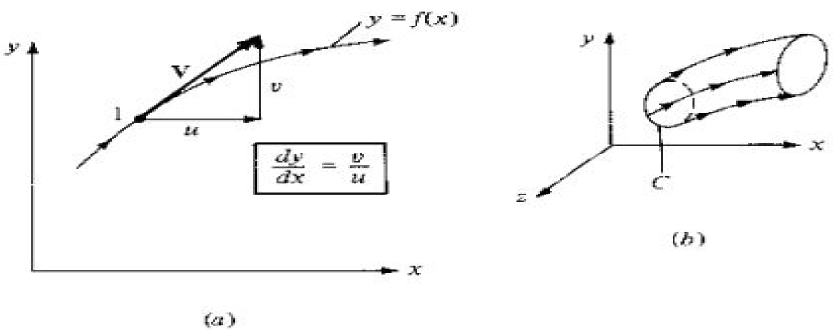


Figure 9. (a) Equation of a stream in two-dimensional Cartesian space. (b) Sketch of a streamtube in three-dimensional space.

Equation (2.118) is a differential equation for a streamline in two dimensions. From Equation (2.118), v

$$v\,dx - u\,dy = 0$$

which is precisely Equation (2.117c). Therefore, Equations (2.117a to c) and (2.118) simply state mathematically that the velocity vector is tangent to the streamline.

A concept related to streamlines is that of a streamtube. Consider an arbitrary closed curve C in three-dimensional space, as shown in Figure 9b.

Consider the streamlines which pass through all points on C. These streamlines form a tube in space as sketched in Figure 9b; such a tube is called a streamtube.

For example, the walls of an ordinary garden hose form a streamtube for the water flowing through the hose. For a steady flow, a direct application of the integral form of the continuity equation [Equation (2.53)] proves that the mass flow across all cross sections of a streamtube is constant.

ANGULAR VELOCITY, VORTICITY, AND STRAIN

Consider an infinitesimal fluid element moving in a flow field. As it translates along a streamline, it may also rotate, and in addition its shape may become distorted as sketched in Figure 10. The amount of rotation and distortion depends on the velocity field; the purpose of this section is to quantify this dependency.

Consider a two-dimensional flow in the xy plane. Also, consider an infinitesimal fluid element in this flow. Assume that at time t the shape of this fluid element is rectangular, as shown at the left of Figure 10.

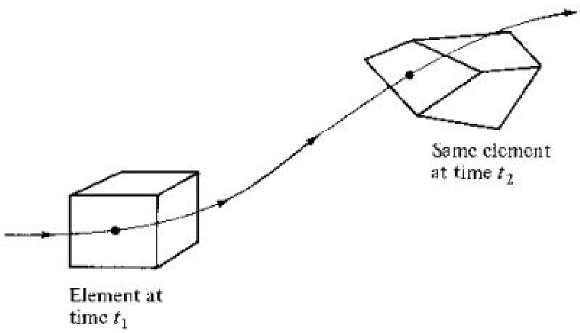


Figure 10. The motion of a fluid element along a streamline is a combination of translation and rotation; in addition, the shape of the element can become distorted.

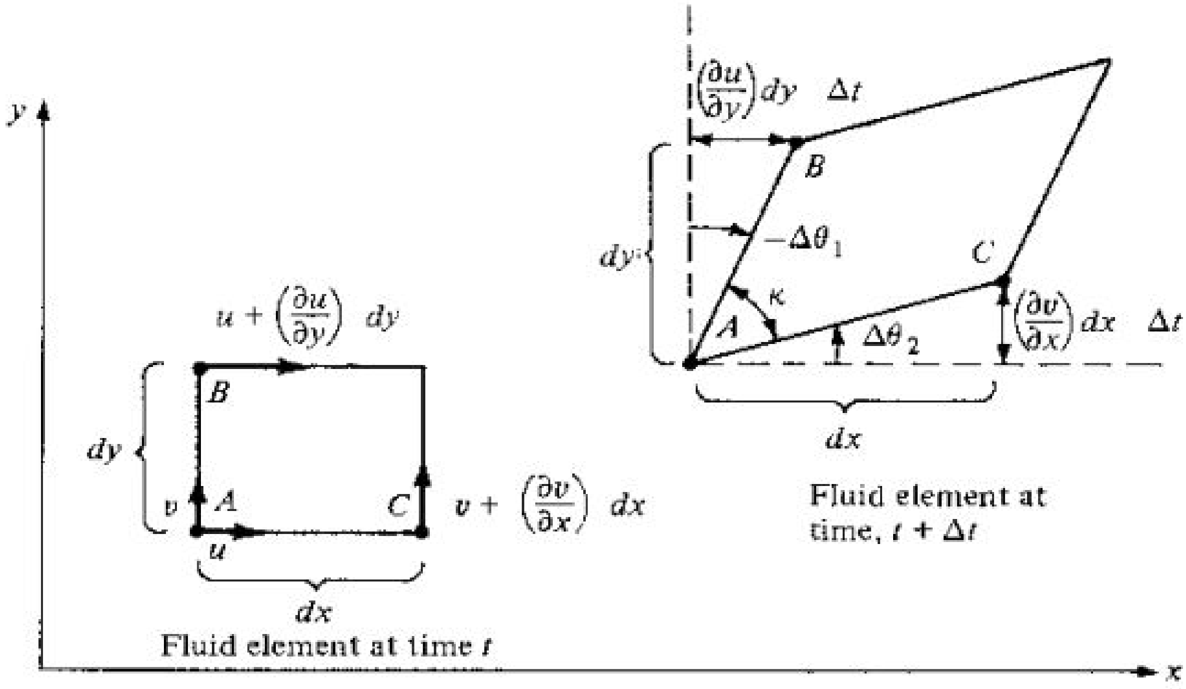


Figure 11. Rotation and distortion of a fluid element.

Assume that the fluid element is moving upward and to the right; its position and shape at time t + Δt are shown at the right in Figure 11.

Note that during the time increment Δ t, the sides AB and AC have rotated through the angular displacements − Δθ1 and Δθ2, respectively. (Counterclockwise rotations by convention are considered positive; since line AB is shown with a clockwise rotation in Figure 11, the angular displacement is negative, −θ1.) At present, consider just the line AC.

It has rotated because during the time increment t, point C has moved differently from point A. Consider the velocity in the y direction.

At point A at time t, this velocity is v, as shown in Figure 11. Point C is a distance dx from point A; hence, at time t the vertical component of velocity of point C is given by v +(∂v/∂x) dx. Hence,

$$\begin{aligned}
\text{Distance in } y \text{ direction that } A \text{ moves} \\ \text{during time increment } \Delta t &= v\Delta t \\
\text{Distance in } y \text{ direction that } C \text{ moves} \\ \text{during time increment } \Delta t &= \left(v + \frac{\partial v}{\partial x}dx\right)\Delta t \\
\text{Net displacement in } y \text{ direction} \\ \text{of } C \text{ relative to } A &= \left(v + \frac{\partial v}{\partial x}dx\right)\Delta t - v\Delta t \\
&= \left(\frac{\partial v}{\partial x}dx\right)\Delta t
\end{aligned}$$

This net displacement is shown at the right of Figure 2.33. From the geometry of Figure 2.33,

$$\tan \Delta\theta_2 = \frac{[(\partial v/\partial x)\,dx]\,\Delta t}{dx} = \frac{\partial v}{\partial x}\Delta t \tag{2.119}$$

Since $\Delta\theta_2$ is a small angle, $\tan \Delta\theta_2 \approx \Delta\theta_2$. Hence, Equation (2.119) reduces to

$$\Delta\theta_2 = \frac{\partial v}{\partial x}\Delta t \tag{2.120}$$

Now consider line AB. The x component of the velocity at point A at time t is u, as shown in Figure 11. Because point B is a distance dy from point A, the horizontal component of velocity of point B at time t is $u + (\partial u/\partial y)\, dy$.

By reasoning similar to that above, the net displacement in the x direction of B relative to A over the time increment t is $[(\partial u/\partial y)\, dy]\, t$, as shown in Figure 11. Hence,

$$\tan(-\Delta\theta_1) = \frac{[(\partial u/\partial y)\,dy]\,\Delta t}{dy} = \frac{\partial u}{\partial y}\Delta t \tag{2.121}$$

Since $-\Delta\theta_1$ is small, Equation (2.121) reduces to

$$\Delta\theta_1 = -\frac{\partial u}{\partial y}\Delta t \tag{2.122}$$

Consider the angular velocities of lines AB and AC, defined as $d\theta_1/dt$ and $d\theta_2/dt$, respectively. From Equation (2.122), we have

$$\frac{d\theta_1}{dt} = \lim_{\Delta t \to 0} \frac{\Delta\theta_1}{\Delta t} = -\frac{\partial u}{\partial y} \tag{2.123}$$

From Equation (2.120), we have

$$\frac{d\theta_2}{dt} = \lim_{\Delta t \to 0} \frac{\Delta\theta_2}{\Delta t} = \frac{\partial v}{\partial x} \tag{2.124}$$

By definition, the angular velocity of the fluid element as seen in the xy plane is the average of the angular velocities of lines AB and AC. Let ω_z denote this angular velocity. Therefore, by definition,

$$\omega_z = \frac{1}{2}\left(\frac{d\theta_1}{dt} + \frac{d\theta_2}{dt}\right) \tag{2.125}$$

Combining Equations (2.123) to (2.125) yields

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \tag{2.126}$$

In the above discussion, we have considered motion in the xy plane only. However, the fluid element is generally moving in three-dimensional space, and its angular velocity is a vector ω that is oriented in some general direction, as shown in Figure 12. In Equation (2.126),

we have obtained only the component of ω in the z direction; this explains the subscript z in Equations (2.125) and (2.126). The x and y components of ω can be obtained in a similar fashion.

The resulting angular velocity of the fluid element in three-dimensional space is

$\omega = \omega x_i + \omega y_j + \omega z_k$

$$\boxed{\boldsymbol{\omega} = \frac{1}{2}\left[\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\mathbf{i} + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\mathbf{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\mathbf{k}\right]} \tag{2.127}$$

- Equation (2.127) is the desired result; it expresses the angular velocity of the fluid element in terms of the velocity field, or more precisely, in terms of derivatives of the velocity field.
- The angular velocity of a fluid element plays an important role in theoretical aerodynamics, as we shall soon see.

- However, the expression 2ω appears frequently, and therefore we define a new quantity, vorticity, which is simply twice the angular velocity. Denote vorticity by the vector ξ :

$\xi \equiv 2\omega$

- Hence, from Equation (2.127),

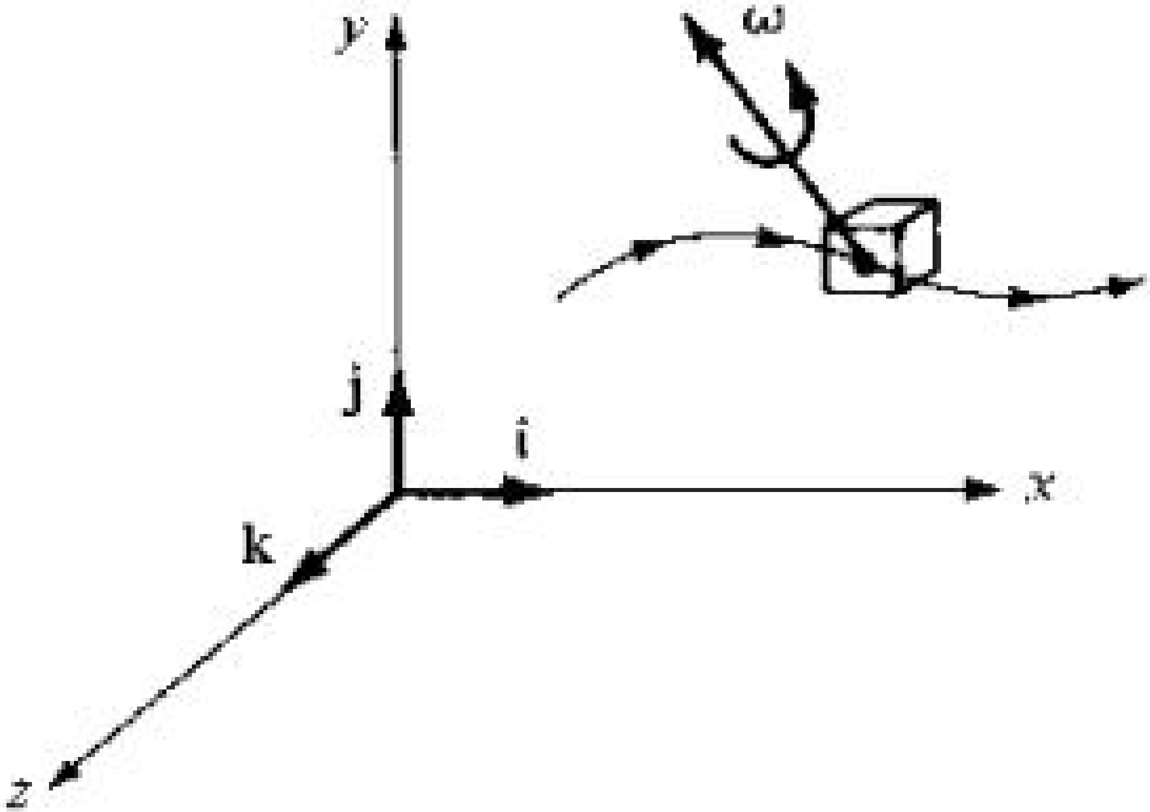


Figure 12. Angular velocity of a fluid element in three-dimensional space.

$$\boxed{\boldsymbol{\xi} = \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\mathbf{i} + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\mathbf{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\mathbf{k}} \qquad (2.128)$$

Recall Equation (2.22) for ∇ × V in cartesian coordinates. Since u, v, and w denote the x, y, and z components of velocity, respectively, note that the right sides of Equations (2.22) and (2.128) are identical. Hence, we have the important result that

$$\boxed{\xi = \nabla \times \mathbf{V}} \qquad (2.129)$$

In a velocity field, the curl of the velocity is equal to the vorticity. The above leads to two important definitions: 1. If $\nabla \times V = 0$ at every point in a flow, the flow is called rotational. This implies that the fluid elements have a finite angular velocity. 2. If $\nabla \times V = 0$ at every point in a flow, the flow is called irrotational. This implies that the fluid elements have no angular velocity; rather, their motion through space is a pure translation.

The case of rotational flow is illustrated in Figure 13. Here, fluid elements moving along two different streamlines are shown in various modes of rotation.

In contrast, the case of irrotational flow is illustrated in Figure 14. Here, the upper streamline shows a fluid element where the angular velocities of its sides are zero.

The lower streamline shows a fluid element where the angular velocities of two intersecting sides are finite but equal and opposite to each other, and so their sum is identically zero.

In both cases, the angular velocity of the fluid element is zero (i.e., the flow is irrotational).

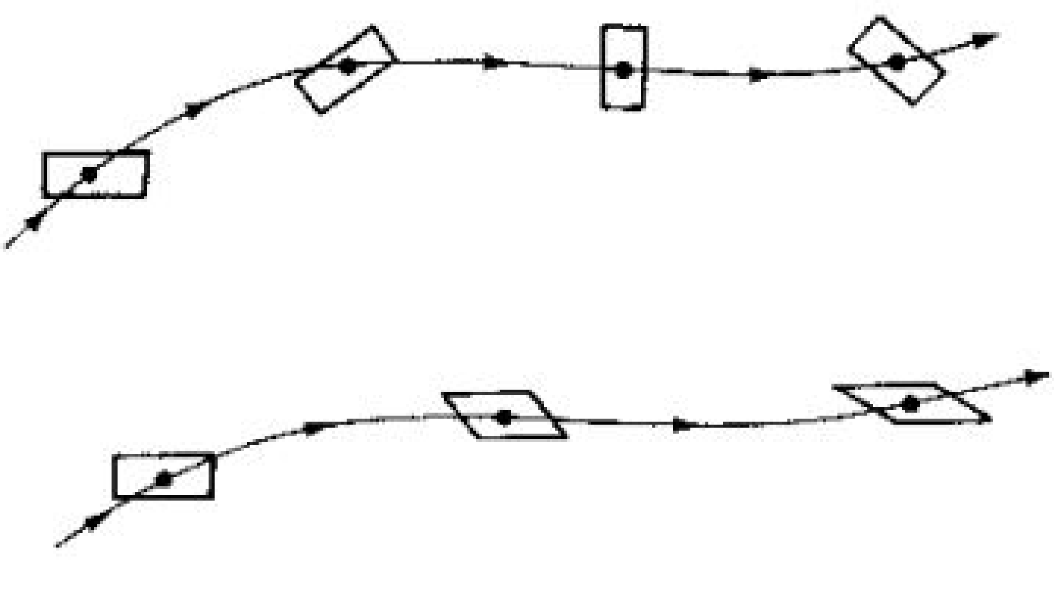


Figure 13. Fluid elements in a rotational flow

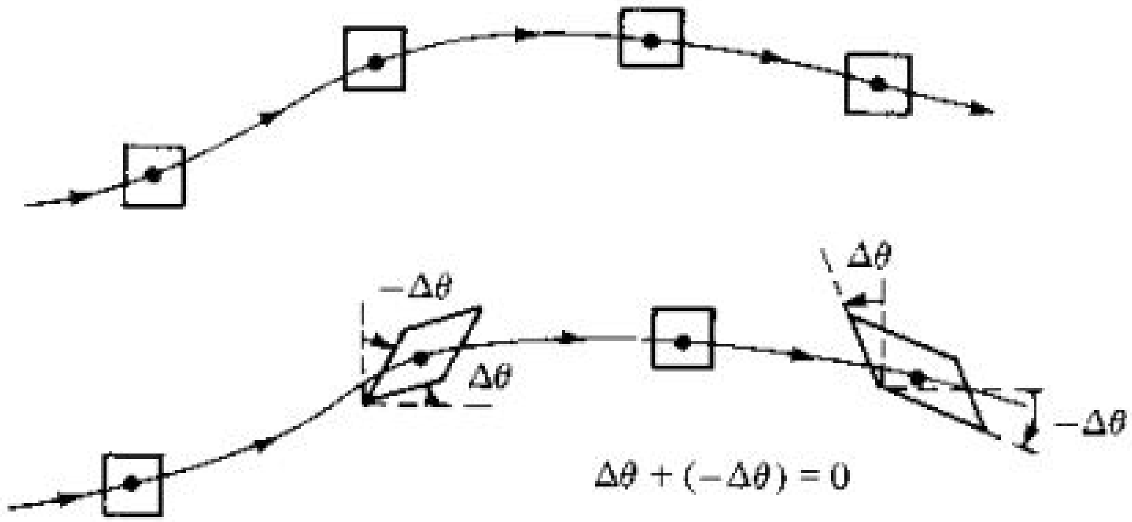


Figure 14. Fluid elements in an irrotational flow.

If the flow is two-dimensional (say, in the xy plane), then from Equation (2.128),

$$\boldsymbol{\xi} = \xi_z \mathbf{k} = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) \mathbf{k} \tag{2.130}$$

Also, if the flow is irrotational, $\boldsymbol{\xi} = 0$. Hence, from Equation (2.130),

$$\boxed{\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0} \tag{2.131}$$

- Equation (2.131) is the condition of irrotationality for two-dimensional flow. We will have frequent occasion to use Equation (2.131).
- Why is it so important to make a distinction between rotational and irrotational flows? The answer becomes blatantly obvious as we progress in our study of aerodynamics; we find that irrotational flows are much easier to analyze than rotational flows.
- However, irrotational flow may at first glance appear to be so special that its applications are limited. Amazingly enough, such is not the case. There are a large number of practical aerodynamic problems where the flow field is essentially irrotational, for example, the subsonic flow over

airfoils, the supersonic flow over slender bodies at small angle of attack, and the subsonic-supersonic flow through nozzles.

- For such cases, there is generally a thin boundary layer of viscous flow immediately adjacent to the surface; in this viscous region the flow is highly rotational.
- However, outside this boundary layer, the flow is frequently irrotational. As a result, the study of irrotational flow is an important aspect of aerodynamics.
- Return to the fluid element shown in Figure 2.33. Let the angle between sides AB and AC be denoted by κ.
- As the fluid element moves through the flow field, κ will change. In Figure 2.33, at time t, κ is initially 90°. At time $t + \Delta t$, κ has changed by the amount $\Delta\kappa$, where
- By definition, the strain of the fluid element as seen in the xy plane is the change in κ, where positive strain corresponds to a decreasing κ. Hence, from Equation (2.132),

$$\text{Strain} = -\Delta\kappa = \Delta\theta_2 - \Delta\theta_1 \qquad (2.133)$$

- In viscous flows (to be discussed in Chapters 15 to 20), the time rate of strain is an important quantity. Denote the time rate of strain by ε_{xy}, where in conjunction with Equation (2.133)

$$\varepsilon_{xy} \equiv -\frac{d\kappa}{dt} = \frac{d\theta_2}{dt} - \frac{d\theta_1}{dt} \tag{2.134}$$

Substituting Equations (2.123) and (2.124) into (2.134), we have

$$\varepsilon_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \tag{2.135a}$$

In the yz and zx planes, by a similar derivation the strain is, respectively,

$$\varepsilon_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \tag{2.135b}$$

and

$$\varepsilon_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \tag{2.135c}$$

Note that angular velocity (hence, vorticity) and time rate of strain depend solely on the velocity derivatives of the flow field. These derivatives can be displayed in a matrix as follows:

$$\begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial z} \\ \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial x} & \frac{\partial w}{\partial y} & \frac{\partial w}{\partial z} \end{bmatrix}$$

The sum of the diagonal terms is simply equal to $\nabla \cdot V$, which from Section 2.3 is equal to the time rate of change of volume of a fluid element; hence, the diagonal terms represent the dilatation of a fluid element.

The off-diagonal terms are cross derivatives which appear in Equations (2.127), (2.128), and (2.135a to c).

Hence, the off-diagonal terms are associated with rotation and strain of a fluid element. In summary, in this section, we have examined the rotation and deformation of a fluid element moving in a flow field.

The angular velocity of a fluid element and the corresponding vorticity at a point in the flow are concepts that are useful in the analysis of both inviscid and viscous flows; in particular, the absence of vorticity—irrotational flow—greatly simplifies the analysis of the flow, as we will see. We take advantage of this simplification in much of our treatment of inviscid flows in subsequent chapters.

CIRCULATION

You are reminded again that this is a tool-building chapter. Taken individually, each aerodynamic tool we have developed so far may not be particularly exciting. However, taken collectively, these tools allow us to obtain solutions for some very practical and exciting aerodynamic problems.

In this section, we introduce a tool that is fundamental to the calculation of aerodynamic lift, namely, circulation. This tool was used independently by Frederick Lanchester (1878–1946) in England, Wilhelm Kutta (1867–1944) in Germany, and Nikolai Joukowski (1847–1921) in Russia to create a breakthrough in the theory of aerodynamic lift at the turn of the twentieth century.

The relationship between circulation and lift and the historical circumstances surrounding this breakthrough are discussed in Chapters 3 and 4.

The purpose of this section is only to define circulation and relate it to vorticity. Consider a closed curve C in a flow field, as sketched in Figure 2.38. Let V and ds be the velocity and directed line segment, respectively, at a point on C. The circulation, denoted by Ŵ, is defined as

$$\boxed{\Gamma \equiv -\oint_C \mathbf{V} \cdot \mathbf{ds}} \qquad (2.136)$$

The circulation is simply the negative of the line integral of velocity around a closed curve in the flow; it is a kinematic property depending only on the velocity field and the choice of the curve C.

Line Integrals, by mathematical convention the positive sense of the line integral is counterclockwise. However, in aerodynamics, it is convenient to consider a positive circulation as being clockwise.

Hence, a minus sign appears in the definition given by Equation (2.136) to account for the positive-counterclockwise sense of the integral and the positive-clockwise sense of circulation

The use of the word circulation to label the integral in Equation (2.136) may be somewhat misleading because it leaves a general impression of something moving around in a loop. Indeed, according to the American Heritage Dictionary of the English Language, the first definition given to the word “circulation” is “movement in a circle or circuit.”

However, in aerodynamics, circulation has a very precise technical meaning, namely, Equation (2.136). It does not necessarily mean that the fluid elements are moving around in circles within this flow field—a common early misconception of new students of aerodynamics.

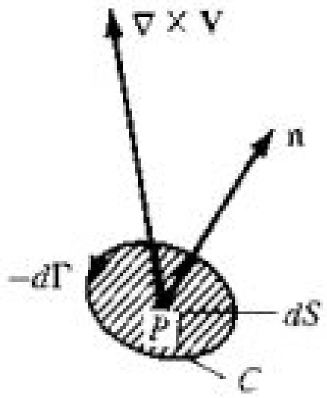
Rather, when circulation exists in a flow, it simply means that the line integral in Equation (2.136) is finite.

For example, if the airfoil in Figure 2.28 is generating lift, the circulation taken around a closed curve enclosing the airfoil will be finite, although the fluid elements are by no means executing circles around the airfoil (as clearly seen from the streamlines sketched in Figure 2.28).

Circulation is also related to vorticity as follows. Refer back to Figure 2.11, which shows an open surface bounded by the closed curve C. Assume that the surface is in a flow field and the velocity at point P is V, where P is any point on the surface (including any point on curve C). From Stokes’ theorem [Equation (2.25)],

$$\boxed{\Gamma \equiv -\oint_C \mathbf{V} \cdot \mathbf{ds} = -\iint_S (\nabla \times \mathbf{V}) \cdot \mathbf{dS}} \qquad (2.137)$$

Hence, the circulation about a curve C is equal to the vorticity integrated over any open surface bounded by C. This leads to the immediate result that if the flow is irrotational everywhere within the contour of integration (i.e., if $\nabla \times V = 0$ over any surface bounded by C), then $\hat{W} = 0$. A related result is obtained by letting the curve C shrink to an infinitesimal size, and denoting the circulation around this infinitesimally small curve by $d\hat{W}$. Then, in the limit as C becomes infinitesimally small, Equation (2.137) yields



$$\boxed{(\nabla \times \mathbf{V}) \cdot \mathbf{n} = -\frac{d\Gamma}{dS}}$$

Figure 2.39 Relation between vorticity and circulation.

$$d\Gamma = -(\nabla \times \mathbf{V}) \cdot \mathbf{dS} = -(\nabla \times \mathbf{V}) \cdot \mathbf{n}\, dS$$

or

$$\boxed{(\nabla \times \mathbf{V}) \cdot \mathbf{n} = -\frac{d\Gamma}{dS}} \qquad (2.138)$$

where d S is the infinitesimal area enclosed by the infinitesimal curve C. Referring to Figure 2.39, Equation (2.138) states that at a point P in a flow, the component of vorticity normal to d S is equal to the negative of the "circulation per unit area," where the circulation is taken around the boundary of d S.