INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal, Hyderabad -500 043

AERONAUTICAL ENGINEERING

COURSE HANDOUT

Course Name	**FLUID MECHANICS AND HYDRAULICS**
Course Code	**AAE003**
Programme	B.Tech
Semester	III
Course Coordinator	Mr. G Satya Dileep, Assistant Professor, AE
Course Faculty	Mr. R Sabari Vihar, Assistant Professor, AE

UNIT 1
FLUID PROPERTIES AND FLUID STATICS

Definition of a fluid

A fluid is defined as a substance that deforms continuously under the action of a shear stress, however small magnitude present. It means that a fluid deforms under very small shear stress, but a solid may not deform under that magnitude of the shear stress.

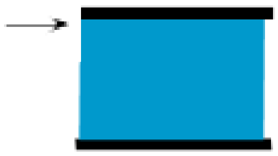


Fig.L-1.1a: Deformation of solid under a constant shear force

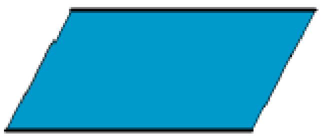


Fig.L-1.1b: Deformation of fluid under a constant shear force

By contrast a solid deforms when a constant shear stress is applied, but its deformation does not continue with increasing time. In Fig.L1.1, deformation pattern of a solid and a fluid under the action of constant shear force is illustrated. We explain in detail here deformation behaviour of a solid and a fluid under the action of a shear force.

In Fig.L1.1, a shear force ***F*** is applied to the upper plate to which the solid has been bonded, a shear stress resulted by the force equals to,

$$\tau = \frac{F}{A}.$$

Where *A* is the contact area of the upper plate. We know that in the case of the solid block the deformation is proportional to the shear stress *t* provided the elastic limit of the solid material is not exceeded.

When a fluid is placed between the plates, the deformation of the fluid element is illustrated in Fig.L1.3. We can observe the fact that the deformation of the fluid element continues to increase as long as the force is applied. The fluid particles in direct contact with the plates move with the same speed of the plates. This can be interpreted that there is no slip at the boundary. This fluid behavior has been verified in numerous experiments with various kinds of fluid and boundary material.

In short, *a fluid continues in motion under the application of a shear stress and can not sustain any shear stress when at rest.*

Fluid as a continuum

In the definition of the fluid the molecular structure of the fluid was not mentioned. As we now the fluids are composed of molecules in constant motions. For a liquid, molecules are closely spaced compared with that of a gas. In most engineering applications the average or macroscopic effects of a large number of molecules is considered. We thus do not concern about the behavior of individual molecules. The fluid is treated as an infinitely divisible substance, a continuum at which the properties of the fluid are considered as a continuous (smooth) function of the space variables and time.

To illustrate the concept of fluid as a continuum consider fluid density as a fluid property at a small region.(Fig.L1.2 (a)). Density is defined as mass of the fluid molecules per unit volume. Thus the mean density within the small region **C** could be equal to mass of fluid molecules per unit volume. When the small region **C** occupies space which is larger than the cube ofmolecular spacing, the number of the molecules will remain constant. This is the limiting volume above which t ect of molecular variations on fluid properties is negligible. A plot of the mean density versus the size of unit volume.

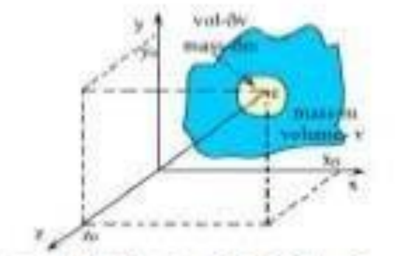


Fig. L-1.2(a):Small region in fluid domain

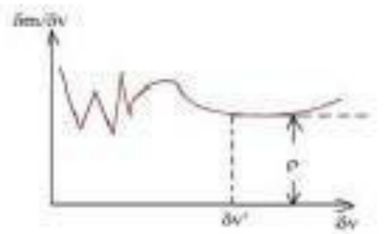


Fig. L-1.2(b): Variation of density with respect to volume of the region

The density of the fluid is defined as

$$\rho = \lim_{\delta v \to \delta v'} \frac{\delta m}{\delta v} \qquad \text{Eq(1.1)}$$

Note that the limiting $\delta v'$ is about $10^{-9}\,mm^3$ for all liquids and for gases at atmospheric temperature. Within the given limiting value, air at the standard condition has approximately 3×10^7 molecules. It justifies in defining a nearly constant density in a region which is larger than the limiting volume.

In conclusion, since most of the engineering problems deal with fluids at a dimension which is larger than the limiting volume, the assumption of fluid as a continuum is valid. For example the fluid density is defined as a function of space (for Cartesian coordinate system, x, y, and z) and time (t) by $\rho = \rho(x, y, z, t)$. This simplification helps to use the differential calculus for solving fluid problems

Properties of fluid

Some of the basic properties of fluids are discussed below-
Density : As we stated earlier the density of a substance is its mass per unit volume. In fluid mechanic it is expressed in three different ways-

1. **Mass density** ρ is the mass of the fluid per unit volume

   Unit- kg/m^3

   Dimension- ML^{-3}

   Typical values: water- *1000 kg/* m^3

   Air- $1.23\ kg/m^3$ at standard pressure and temperature (STP)

2. **Specific weight**, w: - As we express a mass *M* has a weight ***W=Mg*** . The specific weight of the fluid can be defined similarly as its weight per unit volume.

   $$w = \rho g$$

   Unit: N/m^3

   Dimension: $ML^{-2}T^{-2}$

   Typical values; water- $9.810 N/m^3$

   Air- $12.07 N/m^3$ (STP)

3. **Relative density** (Specific gravity), ***S*** :-

Specific gravity is the ratio of fluid density (specific weight) to the fluid density (specific weight) of a standard reference fluid. For liqutids water a is 4^0C considered as standard fluid. Similarly for gases air at specific temperature and pressure is considered as a standard

$$S_{liquid} = \frac{\rho_{liquid}}{\rho_{water\ at\ 4^0C}}$$

reference fluid.

$$S_{gas} = \frac{\rho_{gas}}{\rho_{gas\ at\ STP}}$$

Units: pure number having no units Dimensi $M^0L^0T^0$

Typical ovna.l.es : - Mercury- 13.6
Water-1

Specific volume ν_s : - Specific volume of a fluid is mean volume per unit mass *i.e.* the reciprocal of mass density.

$$\nu_s = \frac{1}{\rho}$$

Units:- m^3/kg

Dimension: $M^{-1}L^3$

Typical values: - Water - $10^{-3} m^3/kg$

Air- $1.23\times10^{-3} m^3/kg$

Viscosity

In section L1 definition of a fluid says that under the action of a shear stress a fluid continuously deforms, and the shear strain results with time due to the deformation. Viscosity is a fluid property, which determines the relationship between the fluid strain rate and the applied shear stress. It can be noted that in fluid flows, shear strain rate is considered, not shear strain as commonly used in solid mechanics. Viscosity can be inferred as a quantative measure of a fluid's resistance to the flow. For example moving an object through air requires very less force compared to water. This means that air has low viscosity than water.

Let us consider a fluid element placed between two infinite plates as shown in fig (Fig-2.1). The upper plate moves at a constant velocity δu under the action of constant shear force δF. The shear stress, *t* is expressed as

$$\tau = \lim_{\delta A \to 0} \frac{\delta F}{\delta A} = \frac{dF}{dA}$$

Basic Properties of Fluid

From the geometry of the figure, we can define

For small $\delta\alpha$, $\tan \delta\alpha = \frac{\delta u \ \delta t}{\delta y}$

Therefore,

$$\frac{\delta\alpha}{\delta t} = \frac{\delta u}{\delta y}$$

The limit of both side of the equality gives $\frac{d\alpha}{dt} = \frac{du}{dy}$

The above expression relates shear strain rate to velocity gradient along the y -axis

Newton 's Viscosity Law

Sir Isaac Newton conducted many experimental studies on various fluids to determine relationship between shear stress and the shear strain rate. The experimental finding showed that a linear relation between them is applicable for common fluids such as water, oil, and air. The relation is

$$\tau \ \alpha \ \frac{d\alpha}{dt}$$

Substituting the relation gives in equation

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

Introducing the constant of proportionality

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

where μ is called absolute or dynamic viscosity. Dimensions and units for μ are $ML^{-1}T^{-1}$ and $N-s/m^2$, respectively. [In the absolute metric system basic unit of co-efficient of viscosity is called poise. 1 poise = $N-s/m^2$]

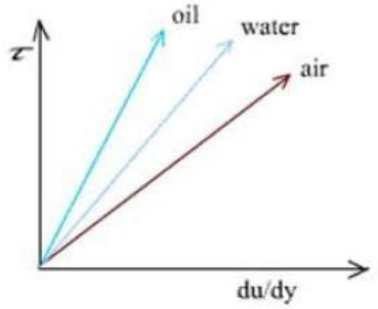


Fig.L-2.2: Relationship between shear stress and velocity gradient of Newtonian fluids

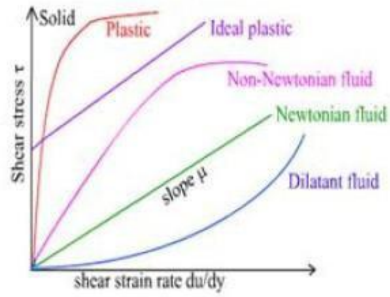


Fig.L-2.3: Relationship between shear stress and shear strain rate of diferent fluids

Typical relationships for common fluids are illustrated in.

The fluids that follow the linear relationship given in equations are called Newtonian fluids.

Kinematic viscosity v

Kinematic viscosity is defined as the ratio of dynamic viscosity to mass density

$$\nu = \frac{\mu}{\rho}$$

Units: m^2/s

Dimension: L^2T^{-1}

Typical values: water $1.14\times10^{-6} m^2s^{-1}$ air $1.46\times10^{-5} m^2/s$

Non - Newtonian fluids

Fluids in which shear stress is not linearly related to the rate of shear strain are non? Newtonian fluids. Examples are paints, blot, polymeric solution, etc. Instead of the dynamic viscosity apparent viscosity, μ_{ap} whichis the slope of shear stress versus shear strain rate

curve, is used for these types of fluid

Based on the behavior of , non-Newtonian fluids are broadly classified into the following groups –

a. *Pseudo plastics* (shear thinning fluids): decreases with increasing shear strain rate. For example polymer solutions, colloidal suspensions, latex paints, pseudo plastic.

b. *Dilatants* (shear thickening fluids) increases with increasing shear strain rate. Examples: Suspension of starch and quick sand (mixture of water and sand).

c. *Plastics* : Fluids that can sustain finite shear stress without any deformation, but once shear stress exceeds the finite stress , they flow like a fluid. The relation between the shear stress and the resulting shear strain is given by

$$\tau = \tau_y + \mu_{ap}\left(\frac{du}{dy}\right)^n$$

Fluids with n = 1 are called Bingham plastic. some examples are clay suspensions, tooth paste and fly ash.

d. *Thixotropic* fluid : μ_{ap} decreases with time under a constant applied shear stress.

Example: some typical liquid-solid suspensions

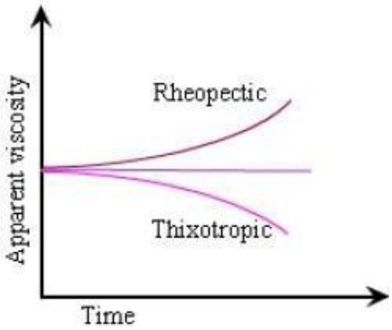


Fig: Thixotropic and Rheopectic fluids

Example 1: Density

If 5 m^3 of certain oil weighs 45 kN calculate the specific weight, specific gravity and mass density of the oil

Solution :

Given data: Volume = 5 m^3

Weight = 45 kN

$$\text{Specific Weight of the oil} = \frac{\text{Weight of the oil}}{\text{Volume of the oil}}$$

$$\text{Specific gravity} = \frac{\text{Specific weight of the oil}}{\text{Specific weight of water}}$$

Example 2: Density

A liquid has a mass density of 1550 kg/m^3. Calculate its specific weight, specific gravity and specific volume

Solution :

Given data: Mass density = 1550 kg/m^3

$$\text{Specific weight} = \text{mass density} \times \text{Acceleration due to gravity}$$

Specific gravity = $\frac{\text{mass density of the liquid}}{\text{mass density of the water}}$

$$\text{Specific volume} = \frac{1}{\text{specific weight}}$$

Answer: $1.52\times10^{4}\,N/m^{3}$; 1.55; $6.57\times10^{-5}m^{3}/N$

Example 3: Viscosity

A plate (2m x 2m), 0.25 mm distant apart from a fixed plate, moves at 40 cm/s and requires a force of 1 N. Determine the dynamic viscosity of the fluid in between the plates

Solution :

Given data: Change of velocity, $dv = 40cm/s$ Distance between the plates, Contact area A = 2x2 = 4 m^2 Force required, F = 1 N $dy = 0.25mm = 0.025cm$

Now,

Shear stress, = F/A = 0.25N/m^2 And, $\tau = \mu \frac{dv}{dy}$

Answer: $1.56\times10^{-4}\,Ns/m^{2}$

Surface tension And Capillarity

Surface tension

In this section we will discuss about a fluid property which occurs at the interfaces of a liquid and gas or at the interface of two immiscible liquids. As shown in Fig (L - 3.1) the liquid molecules- 'A' is under the action of molecular attraction between like molecules (cohesion). However the molecule B' close to the interface is subject to molecular attractions between both like and unlike molecules (adhesion). As a result the cohesive forces cancel for liquid molecule 'A'. But at the interface of molecule 'B' the cohesive forces exceed the adhesive force of the gas. The corresponding net force acts on the interface; the interface is at a state of tension similar to a stretched elastic membrane. As explained, the corresponding net force is referred to as surface

tension, . In short it is apparent tensile stresses which acts at the interface of two immiscible fluids

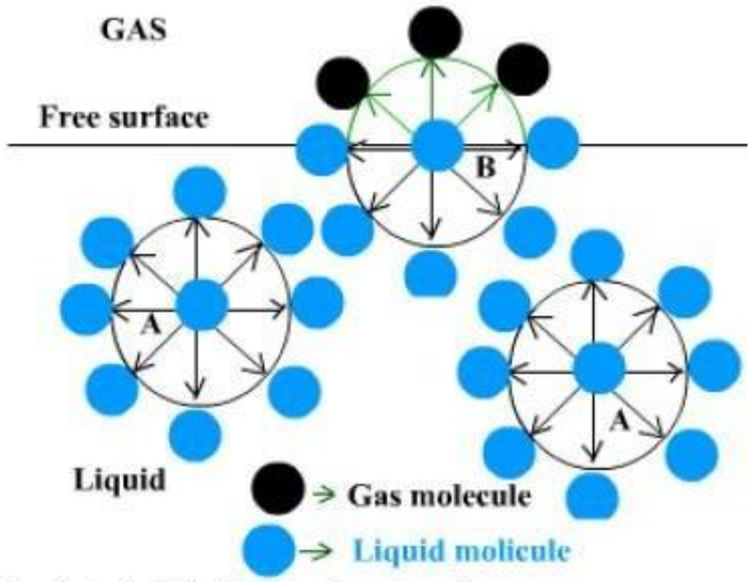


Fig. L-3.1 : Origin of surface tension

Dimension: MT^{-2}
Unit: N/m
Typical values: Water 0.074 N/m at
20° C with air.

Note that surface tension decreases with the liquid temperature because intermolecular cohesive forces decreases. At the critical temperature of a fluid surface tension becomes zero; i.e. the boundary between the fluids vanishes.

Capillarity

If a thin tube, open at the both ends, is inserted vertically in to a liquid, which wets the tube, the liquid will rise in the tube (fig : L -3.4). If the liquid does not wet the tube it will be depressed below the level of free surface outside. Such a phenomenon of rise or fall of the liquid surface

relative to the adjacent level of the fluid is called capillarity. If is the angle of contact between liquid and solid, *d* is the tube diameter, we can determine the capillary rise or depression, *h* by equating force balance in the z-direction (shown in Fig : L-3.5), taking into account surface tension, gravity and pressure. Since the column of fluid is at rest, the sum of all of forces acting on the fluid column is zero.

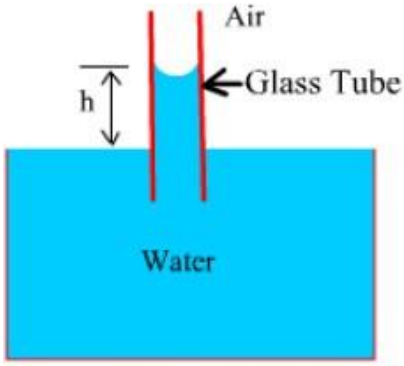


Fig : L - 3.4: Capillarity rise

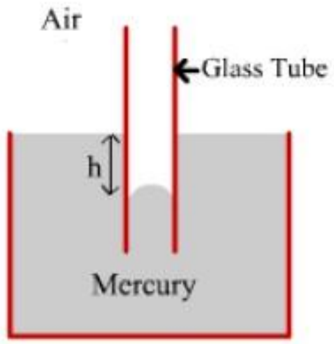


Fig : L - 3.5 : Capillarity depression

The pressure acting on the top curved interface in the tube is atmospheric, the pressure acting on the bottom of the liquid column is at atmospheric pressure because the lines of constant pressure in a liquid at rest are horizontal and the tube is open.

Upward force due to surface tension $= \sigma \cos\theta_c \pi d$

Weight of the liquid column $= \rho g \pi \frac{d^2}{4} h$ Thus equating these two forces we find

$$\sigma \cos\theta_c \pi d = \rho g \pi \frac{d^2}{4} h$$

The expression for *h* becomes

$$h = \frac{4\sigma \cos\theta_c}{\rho g d}$$

Example 1:

Compare the capillary rise of water and mercury in a glass tube of 2 mm diameter at 20^0 C .Given that the surface tension of water and mercury at 20^0 C are 0.0736 N/m and 0.051N/m respectively. Contact angles of water and mercury are 0^0 and 130^0 respectively.

Solution :

Given data: Surface tension of water, s_w = 0.0736 N/m And surface tension mercury, s_m =0.051N/m

Capillary rise in a tube $h = \frac{2\sigma.\cos\theta_c}{\rho g r}$.

For mercury specific weight, $\omega = \rho g = 9810 \times 13.6 N/m^3$ and $\theta_c = 130^0$

Capillary rise for mercury $h_m = -6.68mm$

Note that the negative sign indicates capillary depression. For water specific weight

and $= 9810 N/m^3$. $\theta_c = 0^0$

The capillary rise for water, $h_w = 15mm$

Answer: - 15mm rise and 6.68mm depression

Example 2 :

Find the excess pressure inside a cylindrical jet of water 4 mm diameter than the outside atmosphere? The surface tension of water is 0.0736 N/m at that temperature.

Solution :

Given data:

Surface tension of water s = 0.0736 N/m

Excess pressure in a cylindrical jet $p = \frac{\sigma}{r}$ Answer: - 36.8 Pa

Vapour Pressure Introduction

Since the molecules of a liquid are in constant motion, some of the molecules in the surface layer having sufficient energy will escape from the liquid surface, and then changes from liquid state to gas state. If the space above the liquid is confined and the number of the molecules of the liquid striking the liquid surface and condensing is equal to the number of liquid molecules at any time interval becomes equal, an equilibrium exists. These molecules exerts of partial pressure on the liquid surface known as vapour pressure of the liquid, because degree of molecular activity increases with increasing temperature. The vapour pressure increases with temperature. Boiling occurs when the pressure above a liquid becomes equal to or less then the vapour pressure of the liquid. It means that boiling of water may occur at room temperature if the pressure is reduced sufficiently

For example water will boil at 60 ° C temperature if the pressure is reduced to 0.2 atm

Cavitation

In many fluid problems, areas of low pressure can occur locally. If the pressure in such areas is equal to or less then the vapour pressure, the liquid evaporates and forms a cloud of vapour bubbles. This phenomenon is called cavitation. This cloud of vapour bubbles is swept in to an area of high pressure zone by the flowing liquid. Under the high pressure the bubbles collapses. If this phenomenon occurs in contact with a solid surface, the high pressure developed by collapsing bubbles can erode the material from the solid surface and small cavities may be formed on the surface.

The cavitation affects the performance of hydraulic machines such as pumps, turbines and propellers

Pressure

When a fluid is at rest, the fluid exerts a force normal to a solid boundary or any imaginary plane drawn through the fluid. Since the force may vary within the region of interest, we conveniently define the force in terms of the pressure, P, of the fluid. The pressure is defined as the *force per unit area*

Pascal's Law : Pressure at a point

The Pascal's law states that *the pressure at a point in a fluid at rest is the same in all directions* .

The equilibrium of the fluid element implies that sum of the forces in any direction must be zero. For the x-direction:

Force due to P_x is $P_x \; \delta y \; \delta z$

Component of force due to P_n

$$= -P_n \cdot \delta s \cdot \delta z \cdot \frac{\delta y}{\delta s}$$
$$= -P_n \cdot \delta y \cdot \delta z$$

Summing the forces we get,

$$P_x \ \delta y \ \delta z - P_n \ \delta y \ \delta z = 0$$
$$\text{then } P_x = P_n$$

Similarly in the y-direction, we can equate the forces as given below Force due to P_y =

$$P_y \ \delta x \ \delta z$$

Component of force due to P_n

$$= -P_n \ \delta s \ \delta z \ \frac{\delta x}{\delta s}$$
$$= -P_n \cdot \delta x \cdot \delta z$$

$$\text{Weight of the fluid element} = -\text{Specific weight} \times \text{volume of the element}$$
$$= -\rho \cdot g \cdot \frac{1}{2} \cdot \delta x \cdot \delta y \cdot \delta z$$

The negative sign indicates that weight of the fluid element acts in opposite direction of the z-direction.

Summing the forces yields

$$P_y \ \delta x \ \delta z - P_n \ \delta x \ \delta z - \frac{1}{2} \cdot \rho \cdot g \ \delta x \ \delta y \ \delta z = 0$$

Since the volume of the fluids $\delta x \cdot \delta y \cdot \delta z$ is very small, the weight of the element is negligible in comparison with other force terms.So the above Equation becomes

$P_y = P_n$

Hence, $P_n = P_x = P_y$

Similar relation can be derived for the z-axis direction.

This law is valid for the cases of fluid flow where shear stresses do not exist. The cases are

a. Fluid at rest.
b. No relative motion exists between different fluid layers. For example, fluid at a constant linear acceleration in a container.

c. Ideal fluid flow where viscous force isnegligible

Hydrostatic force on submerged surfaces

Introduction

Designing of any hydraulic structure, which retains a significant amount of liquid, needs to calculate the total force caused by the retaining liquid on the surface of the structure. Other critical components of the force such as the direction and the line of action need to be addressed. In this module the resultant force acting on a submerged surface is derived.

Hydrostatic force on a plane submerged surface

a plane surface of arbitrary shape fully submerged in a uniform liquid. Since there can be no shear force in a static liquid, the hydrostatic force must act normal to the surface.

Consider an element of area $d\bar{A}$ on the upper surface

$$d\bar{F} = -Pd\bar{A}$$

The pressure force acting on the element is

Note that the direction of $d\bar{A}$ is normal to the surface area and the negative sign shows that the pressure force $d\bar{F}$ acts against the surface. The total hydrostatic force on the surface can be computed by integrating the infinitesimal forces over the entire surface area.

$$\bar{F} = \int_A -P \, d\bar{A}$$

If h is the depth of the element, from the horizontal free surface as given in Equation (L2.9) becomes

$$\frac{dP}{dh} = \rho g = w$$

If the fluid density ρ is constant and P 0 is the atmospheric pressure at the free surface, integration of the above equation can be carried out to determine the pressure at the element as given below

$$P = P_0 + \int_0^h w\,dh$$
$$= P_0 + wh$$

Total hydrostatic force acting on the surface is

$$
\begin{aligned}
F &= \int_A P \cdot d\bar{A} \\
&= \int_A (P_0 + wh) \cdot d\bar{A} \\
&= \int_A (P_0 + w \cdot y \sin\theta) \cdot d\bar{A} \\
&= P_0 A + w \cdot \sin\theta \int_A y \; d\bar{A}
\end{aligned}
$$

The integral $\int_A y \; d\bar{A}$ is the first moment of the surface area about the x- axis. If y_c is the y coordinate of the centroid of the area, we can express in which *A* is the total area of the submerged plane. Thus

$$
\begin{aligned}
F &= P_0 \cdot A + w \sin\theta \cdot (y_c A) \\
&= P_c A
\end{aligned}
$$

This equation says that the total hydrostatic force on a submerged plane surface equals to the pressure at the centroid of the area times the submerged area of the surface and acts normal to it.

Centre of Pressure (CP)

The point of action of total hydrostatic force on the submerged surface is called the Centre of Pressure (CP). To find the co-ordinates of CP, we know that the moment of the resultant force about any axis must be equal to the moment of distributed force about the same axis. we can equate the moments about the x-axis.

$$
Y_{cp} F = \int_A y \cdot P \cdot dA
$$

Hydrostatic force on a Curved Submerged surface

The direction of the hydrostatic pressure being normal to the surface varies from point to point.

Consider an elementary area $d\bar{A}$ in the curved submerged surface in a fluid at rest. The pressure force acting on the element is

$$
d\bar{F} = P d\bar{A}
$$

The total hydrostatic force can be computed as

$$
\bar{F} = \int_A -P d\bar{A}
$$

Note that since the direction of the pressure varies along the curved surface, we cannot integrate

the above integral as it was carried out in the previous section. The force vector $\bar{F}$ is expressed in terms of its scalar components as

$$\bar{F} = F_x\hat{i} + F_y\hat{j} + F_z\hat{k}$$

in which $F_x, F_y \text{ and } F_z$ represent the scalar components of F in the x , y and z directions respectively.

For computing the component of the force in the x-direction, the dot product of the force and the unit vector $\bar{i}$

Where dA_x is the area projection of the curved element on a plane perpendicular to the x-axis. This integral means that each component of the force on a curved surface is equal to the force on the plane area formed by projection of the curved surface into a plane normal to the

$$F_x = \int_{A_x} P dA_x$$

component. The magnitude of the force component in the vertical direction (z direction)

Since $P = P_0 + wh$ and neglecting P_0, we can write

$$F_z = \int_{A_z} wh \cdot dA_z$$

$$= \int w dv$$

in which is the weight of liquid above the element surface. This integral shows that the z-component of the force (vertical component) equals to the weight of liquid between the submerged surface and the free surface. The line of action of the component passes through the centre of gravity of the volume of liquid between the free surface and the submerged surface.

Example 1 :

A vertical gate of 5 m height and 3 m wide closes a tunnel running full with water. The pressure at the bottom of the gate is 195 kN/m ^{2} . Determine the total pressure on the gate and position of the centre of the pressure.

Given data: Area of the gate = 5x3 = 15 m ^{2}

The equivalent height of water which gives a pressure intensity of 195 kN/m^2 at the bottom. h

= P/w =19.87m.

Total force $P = wA\bar{x}$

$$\bar{x} = 19.87 - 2.5 = 17.37m$$

And

Centre of Pressure $\bar{h} = \bar{x} + \frac{I_G}{A\bar{x}}$ [$I_G = bd^3/12$]

Answer: 2.56MN and 17.49 m.

Buoyancy

we know that wooden objects float on water, but a small needle of iron sinks into water. This means that a fluid exerts an upward force on a body which is immersed fully or partially in it. F_B upward force that tends to lift the body is called the buoyant force, .

The buoyant force acting on floating and submerged objects can be estimated by employing hydrostatic principle.

Center of Buoyancy

The line of action of the buoyant force on the object is called the center of buoyancy. To find the centre of buoyancy, moments about an axis OO can be taken and equated to the moment of the resultant forces. The equation gives the distance to the centeroid to the object volume.

The centeroid of the displaced volume of fluid is the centre of buoyancy, which, is applicable for both submerged and floating objects. This principle is known as the Archimedes principle which states

A body immersed in a fluid experiences a vertical buoyant force which is equal to the weight of the fluid displaced by the body and the buoyant force acts upward through the centroid of the displaced volume"

Basic equations of fluid statics

An equation representing pressure field *P = P (x, y, z)* within fluid at rest is derived in this section. Since the fluid is at rest, we can define the pressure field in terms of space dimensions (x, y and z) only.

Consider a fluid element of rectangular parellopiped shape(Fig : L - 7.1) within a large fluid region which is at rest. The forces acting on the element are body and surface forces.

Body force :

The body force due to gravity is

L -7.1 $d\bar{F}_B = \rho \cdot g \; \delta x \; \delta y \cdot \delta z$

where $\delta x \cdot \delta y \cdot \delta z$ is the volume of the element.

Surface force : The pressure at the center of the element is assumed to be *P* (x, y, z). Using Taylor series expansion the pressure at point $\left(x, y-\frac{\delta y}{2}, z\right)$ on the surface can be expressed as

$$P\left(x, y-\frac{\delta y}{2}, z\right)=P(x, y, z)+\frac{\delta p}{\delta y}\left(-\frac{\delta y}{2}\right)+\frac{1}{2!}\frac{\partial^2 p}{\partial y^2}\left(-\frac{\delta y}{2}\right)^2+\ \ldots$$

When $\delta y \to 0$, only the first two terms become significant. The above equation becomes

$$P\left(x, y-\frac{\delta y}{2}, z\right)=P(x, y, z)+\frac{\delta p}{\delta y}\left(-\frac{\delta y}{2}\right)$$

Similarly, pressures at the center of all the faces can be derived in terms of *P (x, y, z)* and its gradient.

Note that surface areas of the faces are very small. The center pressure of the face represents the average pressure on that face.

The surface force acting on the element in the y-direction is

$$dF_y=\left\{P+\frac{\delta P}{\delta y}\left\{-\frac{\delta y}{2}\right\}\right\}\delta x\cdot\delta y-\left\{P+\frac{\delta P}{\delta y}\left\{\frac{\delta y}{2}\right\}\right\}\delta x\ \delta z$$
$$=-\frac{\delta P}{\delta y}\cdot\delta x\cdot\delta y\cdot\delta z$$

Similarly the surface forces on the other two directions (x and z) will be

$$dF_x=-\frac{\delta P}{\delta x}\cdot\delta x\cdot\delta y\cdot\delta z$$

$$dF_z=-\frac{\delta P}{\delta z}\ \delta x\ \delta y\ \delta z$$

The surface force which is the vectorical sum of the force scalar components

$$dF_s=-\left(\frac{\delta p}{\delta x}\hat{i}+\frac{\delta p}{\delta y}\hat{j}+\frac{\delta p}{\delta z}\hat{k}\right)(\delta x\cdot\delta y\cdot\delta z)$$
$$=-\nabla p\cdot\delta x\cdot\delta y\cdot\delta z$$

The total force acting on the fluid is

$$d\overline{F} = d\overline{F_s} + d\overline{F_b}$$
$$= \left(-\nabla p + \rho\overline{g}\right)(\delta x \cdot \delta y \cdot \delta z)$$ L - 7.6

The total force per unit volume is

$$\frac{dF}{\delta x \ \delta y \cdot \delta z} = -\nabla p + \rho\vec{g}$$

For a static fluid, *dF=0* .

Then, $\left(-\nabla p + \rho\overline{g}\right) = 0$ L -7.7

$$\begin{bmatrix}\text{Net pressure force} \\ \text{per unit volume} \\ \text{at a point}\end{bmatrix} + \begin{bmatrix}\text{Body force} \\ \text{per unit volume} \\ \text{at a point}\end{bmatrix} = 0$$

If acceleration due to gravity is expressed as , the components of Eq(L- 7.8) in the x, y and z directions are

$$-\frac{\delta p}{\delta z} + \rho g_z = 0$$

$$-\frac{\delta p}{\delta x} + \rho g_x = 0$$

$$-\frac{\delta p}{\delta y} + \rho g_y = 0$$

The above equations are the basic equation for a fluid at rest.

Simplifications of the Basic Equations

If the gravity $\overline{g}$ is aligned with one of the co-ordinate axis, for example *z*- axis, then

$g_x = 0$

$g_y = 0$

$$g_z = -g$$

The component equations are reduced to

$$\frac{\delta p}{\delta x} = 0$$

$$\frac{\delta p}{\delta y} = 0$$

$$\frac{\delta p}{\delta z} = -\rho g$$

Under this assumption, the pressure *P* depends on z only. Therefore, total derivative can be used instead of the partial derivative.

$$\frac{dp}{dz} = -\rho g$$

This simplification is valid under the following restrictions

a. Static fluid
b. Gravity is the only body force.
c. The z-axis is vertical and upward.

Example 1 :

Convert a pressure head of 10 m of water column to kerosene of specific gravity 0.8 and carbon- tetra-chloride of specific gravity of 1.62.

Solution :

Given data:

Height of water column, $h_1 = 10$ m Specific gravity of water $s_1 = 1.0$

Specific gravity of kerosene $s_2 = 0.8$

Specific gravity of carbon-tetra-chloride, $s_3 = 1.62$ For the equivalent water head

Weight of the water column = Weight of the kerosene column. So, $\rho g h_1 s_1 = \rho g h_2 s_2 = \rho g h_3 s_3$

Answer:- 12.5 m and 6.17 m.

Scales of pressure measurement

Fluid pressures can be measured with reference to any arbitrary datum. The common datum are

1. Absolute zero pressure.
2. Local atmospheric pressure

When absolute zero (complete vacuum) is used as a datum, the pressure difference is called an absolute pressure, P_{abs}.

When the pressure difference is measured either above or below local atmospheric pressure, P_{local}, as a datum, it is called the gauge pressure. Local atmospheric pressure can be measured by mercury barometer.

At sea level, under normal conditions, the atmospheric pressure is approximately 101.043

kPa. As illustrated

When $P_{abs} < P_{local}$

$P_{gauge} = P_{local} - P_{abs}$

Note that if the absolute pressure is below the local pressure then the pressure difference is known as vacuum suction pressure.

Manometers: Pressure Measuring Devices

Manometers are simple devices that employ liquid columns for measuring pressure difference between two points. some of the commonly used manometers are shown.

In all the cases, a tube is attached to a point where the pressure difference is to be measured and its other end left open to the atmosphere. If the pressure at the point *P* is higher than the local atmospheric pressure the liquid will rise in the tube. Since the column of the liquid in the tube is at rest, the liquid pressure *P* must be balanced by the hydrostatic pressure due to the column of liquid and the superimposed atmospheric pressure, P_{atm}.

$$P = \rho g h + P_{atm}$$

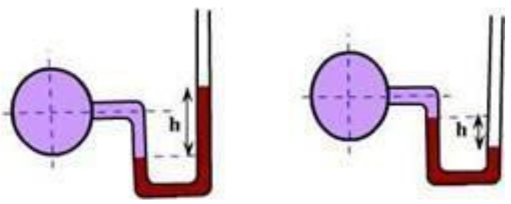
This simplest form of manometer is called a *Piezometer* . It may be inadequate if the pressure difference is either very small or large

U - Tube Manometer

a manometer with two vertical limbs forms a U-shaped measuring tube. A liquid of different density ρ_1 is used as a manometric fluid. We may recall the Pascal's law which states that the pressure on a horizontal plane in a continuous fluid at rest is the same. Applying this equality of pressure at points B and C on the plane gives

$$P + \rho g h = P_{atm} + \rho_1 g h_1$$

$$P - P_{atm} = \rho_1 g h_1 - \rho g h$$



U-tube Manometer

Differential Manometers

Differential Manometers measure difference of pressure between two points in a fluid system and cannot measure the actual pressures at any point in the system

Some of the common types of differential manometers are

a. Upright U-Tube manometer
b. Inverted U-Tube manometer
c. Inclined Differential manometer
d. Micro manometer

UNIT-2

FLUID KINEMATICS AND BASIC EQUATIONS OF FLUID FLOW ANALYSIS

FLUID KINEMATICS

The fluid kinematics deals with description of the motion of the fluids without reference to the force causing the motion.

Thus it is emphasized to know how fluid flows and how to describe fluid motion. This concept helps us to simplify the complex nature of a real fluid flow.

When a fluid is in motion, individual particles in the fluid move at different velocities. Moreover at different instants fluid particles change their positions. In order to analyze the flow behaviour, a function of space and time, we follow one of the following approaches

1. Lagarangian approach
2. Eularian approach

In the Lagarangian approach a fluid particle of fixed mass is selected. We follow the fluid particle during the course of motion with time The fluid particles may change their shape, size and state as they move. As mass of fluid particles remains constant throughout the motion, the basic laws of mechanics can be applied to them at all times. The task of following large number of fluid particles is quite difficult. Therefore this approach is limited to some special applications for example re-entry of a spaceship into the earth's atmosphere and flow measurement system based on particle imagery.

In the Eularian method a finite region through which fluid flows in and out is used. Here we do not keep track position and velocity of fluid particles of definite mass. But, within the region, the field variables which are continuous functions of space dimensions (x , y , z) and time (t), are defined to describe the flow. These field variables may be scalar field variables, vector field variables and tensor quantities. For example, pressure is one of the scalar fields. Sometimes this finite region is referred as control volume or flow domain.

For example the pressure field 'P' is a scalar field variable and defined as

$$P = P(x, y, z, t)$$

Velocity field, a vector field, is defined as $\vec{V} = \vec{V}(x, y, z, t)$ Similarly shear stress is a tensor field variable and defined as

$$\tau = \begin{bmatrix} \tau_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \tau_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \tau_{zz} \end{bmatrix}$$

Note that we have defined the fluid flow as a three dimensional flow in a Cartesian co- ordinates system

Types of Fluid Flow

Uniform and Non-uniform flow : If the velocity at given instant is the same in both magnitude and direction throughout the flow domain, the flow is described as uniform.

When the velocity changes from point to point it is said to be non-uniform flow. Fig.() shows uniform flow in test section of a well designed wind tunnel and ( ) describing non uniform velocity region at the entrance.

Steady and unsteady flows

The flow in which the field variables don't vary with time is said to be steady flow. For steady flow,

$$\frac{\partial v}{\partial t} = 0 \qquad \text{Or} \qquad \vec{v} = \vec{v}(x, y, z)$$

It means that the field variables are independent of time. This assumption simplifies the fluid problem to a great extent. Generally, many engineering flow devices and systems are designed to operate them during a peak steady flow condition.

If the field variables in a fluid region vary with time the flow is said to be unsteady flow.

$$\frac{\partial v}{\partial t} \neq 0 \qquad \vec{v} = \vec{v}(x, y, z, t)$$

One, two and three dimensional flows

Although fluid flow generally occurs in three dimensions in which the velocity field vary with three space co-ordinates and time. But, in some problem we may use one or two space components to describe the velocity field. For example consider a steady flow through a long straight pipe of constant cross-section. The velocity distributions shown in figure are independent of co-ordinate

x and and a function of *r* only. Thus the flow field is one dimensional

Laminar and Turbulent flow

In fluid flows, there are two distinct fluid behaviors experimentally observed. These behaviours were first observed by Sir Osborne Reynolds. He carried out a simple experiment in which water was discharged through a small glass tube from a large tank . A colour dye was injected at the entrance of the tube and the rate of flow could be regulated by a valve at the out let.

When the water flowed at low velocity, it was found that the die moved in a straight line. This clearly showed that the particles of water moved in parallel lines. This type of flow is called laminar flow, in which the particles of fluid moves along smooth paths in layers. There is no exchange of momentum from fluid particles of one layer to the fluid particles of another layer.

This type of flow mainly occurs in high viscous fluid flows at low velocity, for example, oil flows at low velocity.

When the water flowed at high velocity, it was found that the dye colour was diffused over the whole cross section. This could be interpreted that the particles of fluid moved in very irregular paths, causing an exchange of momentum from one fluid particle to another. This type of flow is known as turbulent flow.

Example 1 :

A velocity field is defined by $u = 2y^2$, $v = 3x$, $w = 0$. At point (1,2,0), compute the a) velocity, b) local acceleration and a) convective acceleration

Given velocity field, $u = 2y^2$; $v = 3x$; $w = 0$ so, $V = 2y^2\hat{i} + 3x\hat{j} + 0\hat{k}$

a) Thus, $V_{(1,2,0)} = 8\hat{i} + 3\hat{j} + 0\hat{k}$. And the absolute value $= \sqrt{8^2+3^2+0^2} = 8.54$ units

b) Now from the above equation we can observe that

$\frac{\partial u}{\partial t} = 0;\ \frac{\partial v}{\partial t} = 0;$ and $\frac{\partial w}{\partial t} = 0$ which implies the local acceleration is zero.

c) Also from the above equation we have the acceleration component as follow

$$a_x = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 12xy$$

$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 6y^2$$

$$a_z = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = 0$$

Thus the convective acceleration, $\mathbf{a} = 12xy\hat{i} + 6y^2\hat{j}$

Acceleration at (1,2,0) ; $a_{(1,2,0)} = 24\hat{i} + 24\hat{j}$

Absolute value: $a = \sqrt{24^2 + 24^2} = 24\sqrt{2}$

Velocity Field

The scalar components *u* , *v* and *w* are dependent functions of position and time. Mathematically we can express them as

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

This type of continuous function distribution with position and time for velocity is known as velocity field. It is based on the Eularian description of the flow. We also can represent the Lagrangian description of velocity field.

Let a fluid particle exactly positioned at point *A* moving to another point during time interval Δt . The velocity of the fluid particle is the same as the local velocity at that point as obtained from the Eulerian description

At time *t* , $\vec{V}$ particle at *x* , *y* , *z* $(t) = \vec{V}(x, y, z, t)$

At time $t + \Delta t$ $\vec{V}$, particle at $x', y', z'(t + \Delta t) = \vec{V}(x', y', z', t + \Delta t)$

This means that instead of describing the motion of the fluid flow using the Lagrangian description, the use of Eularian description makes the fluid flow problems quite easier to solve. Besides this difficult, the complete description of a fluid flow using the Lagrangian description requires to keep track over a large number of fluid particles and their movements with time. Thus, more computation is required in the Lagrangian description.

The Acceleration field

At given position A, the acceleration of a fluid particle is the time derivative of the particle's velocity.

Acceleration of a fluid particle: $\vec{a}_{particle} = \frac{d\vec{V}_{particle}}{dt}$

Since the particle velocity is a function of four independent variables (x , y , z and t), we can express the particle velocity in terms of the position of the particle as given below

$$\vec{a}_{particle} = \frac{d\vec{V}_{particle}}{dt} = \frac{d\vec{V}\left(x_{particle}, y_{particle}, z_{particle}\right)}{dt}$$

Applying chain rule, we get

$$\vec{a}_{particle} = \frac{\delta\vec{V}}{\delta t}.\frac{dt}{dt} + \frac{\delta\vec{V}}{\delta x_{particle}}\frac{dx_{particle}}{dt} + \frac{\delta\vec{V}}{\delta y}\frac{dy_{particle}}{dt} + \frac{\delta\vec{V}}{\delta z}\frac{dz_{particle}}{dt}$$

Where δ and d are the partial derivative operator and total derivative operator respectively.

The time rate of change of the particle in the x -direction equals to the x -component of velocity vector, u . Therefore

$$\frac{dx_{particle}}{dt} = u$$

As discussed earlier the position vector of the fluid particle (x particle , y particle , z particle) in the Lagranian description is the same as the position vector (x , y , z) in the Eulerian frame at time t and the acceleration of the fluid particle, which occupied the position (x , y , z) is equal to $a(x,y,z,t)$ in the Eularian description.

Therefore, the acceleration is defined by

$$\vec{a}_{(x,y,z,t)} = \frac{\delta\vec{V}}{\delta t} + u\frac{\delta\vec{V}}{\delta x} + v\frac{\delta\vec{V}}{\delta y} + w\frac{\delta\vec{V}}{\delta z}$$

In vector form

$$\vec{a}_{(x,y,z,t)} = \underbrace{\frac{\delta\vec{V}}{\delta t}}_{\text{(local acceleration)}} + \underbrace{\left(\vec{V}.\nabla\right)\vec{V}}_{\text{(convective acceleration)}}$$

where ∇ is the gradient operator.

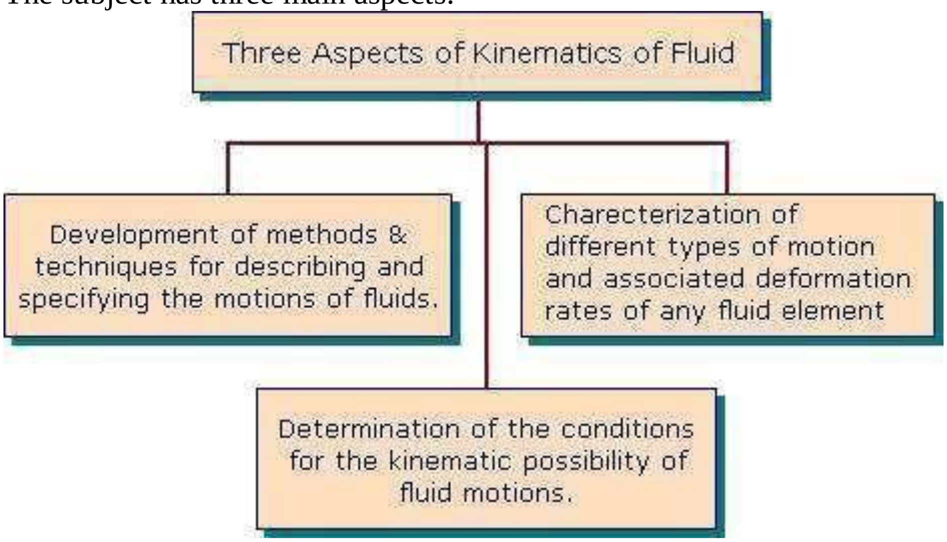
The first term of the right hand side of equation represents the time rate of change of velocity field at the position of the fluid particle at time *t* . This acceleration component is also independent to the change of the particle position and is referred as the local acceleration.\

However the term $(\vec{V}.\nabla)\vec{V}$ accounts for the affect of the change of the velocity at various positions in this field. This rate of change of velocity because of changing position in the field is called the convective acceleration.

Kinematics is the geometry of Motion.

Kinematics of fluid describes the fluid motion and its consequences without consideration of the nature of forces causing the motion.

The subject has three main aspects:



Scalar and Vector Fields

Scalar: Scalar is a quantity which can be expressed by a single number representing its magnitude.

Example: mass, density and temperature.

Scalar Field

If at every point in a region, a scalar function has a defined value, the region is called a scalar field.

Example: Temperature distribution in a rod.

Vector: Vector is a quantity which is specified by both magnitude and direction.

Example: Force, Velocity and Displacement.

Vector Field

If at every point in a region, a vector function has a defined value, the region is called a vector field.

Variation of Flow Parameters in Time and Space

Hydrodynamic parameters like pressure and density along with flow velocity may vary from one point to another and also from one instant to another at a fixed point.

According to type of variations, categorizing the flow: Steady and Unsteady Flow Steady Flow

A steady flow is defined as a flow in which the various hydrodynamic parameters and fluid properties at any point do not change with time.

In Eulerian approach, a steady flow is described as,

$$\vec{V} = V(\vec{S})$$

and

$$\vec{a} = a(\vec{S})$$

Implications:

Velocity and acceleration are functions of space coordinates only.

In a steady flow, the hydrodynamic parameters may vary with location, but the spatial distribution of these parameters remain invariant with time.

In the Lagrangian approach,

Time is inherent in describing the trajectory of any particle.

In steady flow, the velocities of all particles passing through any fixed point at different times will be same.

Describing velocity as a function of time for a given particle will show the velocities at different points through which the particle has passed providing the information of velocity as a function of spatial location as described by Eulerian method. Therefore, the Eulerian and Lagrangian approaches of describing fluid motion become identical under this situation.

Unsteady Flow

An unsteady Flow is defined as a flow in which the hydrodynamic parameters and fluid properties changes with

time.

Uniform and Non-uniform Flows Uniform Flow

The flow is defined as uniform flow when in the flow field the velocity and other hydrodynamic parameters do not change from point to point at any instant of time.

For a uniform flow, the velocity is a function of time only, which can be expressed in Eulerian description as

$\vec{V} = V(t)$

Implication:

For a uniform flow, there will be no spatial distribution of hydrodynamic and other parameters.

Any hydrodynamic parameter will have a unique value in the entire field,

irrespective of whether it changes with time − unsteady uniform flow OR

does not change with time − steady uniform flow.

Thus ,steadiness of flow and uniformity of flow does not necessarily go together.

Non-Uniform Flow

Then the velocity and other hydrodynamic parameters changes from one point to another the flow is

defined as non-uniform. Important points:

1. For a non -uniform flow, the changes with position may be found either in the direction of flow or in directions perpendicular to it.

2. Non-uniformity in a direction perpendicular to the flow is always encountered near solid boundaries past which the fluid flows.

Reason: All fluids possess viscosity which reduces the relative velocity (of the fluid w.r.t. to the wall) to zero at a solid boundary. This is known as no-slip condition.

UNIFORM FLOW

Streamlines

Definition: Streamlines are the Geometrical representation of the of the flow velocity.

Description:

In the Eulerian method, the velocity vector is defined as a function of time and space coordinates.

If for a fixed instant of time, a space curve is drawn so that it

is tangent everywhere to the velocity vector, then this curve is called a Streamline.

Therefore, the Eulerian method gives a series of instantaneous streamlines of the state of motion (Fig. 7.2a).

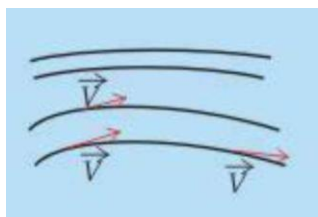


Fig: Streamlines

Alternative Definition:

A streamline at any instant can be defined as an imaginary curve or line in the flow field so that the tangent to the curve at any point represents the direction of theinstantaneous velocity at that point.

Comments:

In an unsteady flow where the velocity vector changes with time, the pattern of streamlines also changes from instant to instant.

In a steady flow, the orientation or the pattern of streamlines will be fixed.

From the above definition of streamline, it can be written as

$\vec{V} \, X \, \overrightarrow{dS}$

Description of the terms:

1. $d\vec{S}$ is the length of an infinitesimal line segment along a streamline at a point .

2. $\vec{V}$ is the instantaneous velocityvector.

The above expression therefore represents the differential equation of a streamline. In a cartesian coordinate-system, representing

$$\vec{S} = \vec{i}dx + \vec{j}dy + \vec{k}dz \qquad \vec{V} = \vec{i}u + \vec{j}v + \vec{k}w$$

the above equation may be simplified as

$$\frac{dx}{u} = \frac{dy}{y} = \frac{dz}{w}$$

Stream tube:

A bundle of neighboring streamlines may be imagined to form a passage through which the fluid flows. This passage is known as a stream-tube.

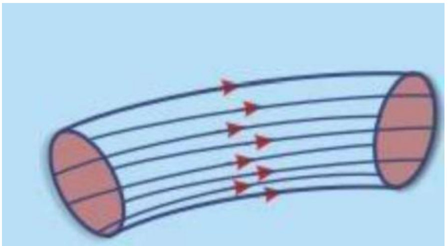


Fig: Stream Tube

Properties of Stream tube:

1. The stream-tube is bounded on all sides by streamlines.

2. Fluid velocity does not exist across a streamline, no fluid may enter or leave a stream- tube except through its ends.

3. The entire flow in a flow field may be imagined to be composed of flows through stream- tubes arranged in some arbitrary positions.

Path Lines

Definition: A path line is the trajectory of a fluid particle of fixed identity as defined by

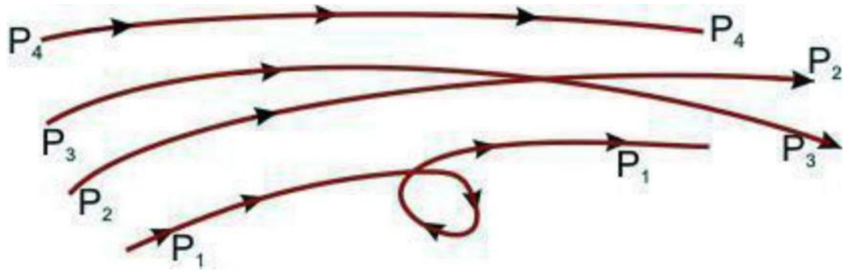


Fig: Path lines

A family of path lines represents the trajectories of different particles, say, P_1, P_2, P_3, etc.

Differences between Path Line and Stream Line

Path Line	Stream Line
This refers to a path followed by a fluid particle over a period of time.	This is an imaginary curve in a flow field for a fixed instant of time, tangent to which gives the instantaneous velocity at that point .
Two path lines can intersect each other as or a single path line can form a loop as different particles or even same particle can arrive at the same point at different instants of time.	Two stream lines can never intersect each other, as the instantaneous velocity vector at any given point is unique.

Note: In a steady flow path lines are identical to streamlines as the Eulerian and Lagrangian versions become the same.

Streak Lines

Definition: A streak line is the locus of the temporary locations of all particles that have passed though a fixed
point in the flow field at any instant of time.

Features of a Streak Line:

while a path line refers to the identity of a fluid particle, a streak line is specified by a fixed point in the flow field.

It is of particular interest in experimental flow visualization.

Example: If dye is injected into a liquid at a fixed point in the flow field, then at a later time t, the dye will indicate the end points of the path lines of particles which have passed through the injection point.

The equation of a streak line at time t can be derived by the Lagrangian method.

If a fluid particle $(\vec{S}_o)$ passes through a fixed point $(\vec{S}_1)$ in course of time t, then the Lagrangian method of description gives the equation

$$\boldsymbol{S(\vec{S}_0, t) = \vec{S}_1}$$

Solving for ,

$$\boldsymbol{\vec{S}_0 = F(\vec{S}_1, t)}$$

If the positions of the particles which have passed through the fixed point are determined, then a streak line can be drawn through these points.

Equation: The equation of the streak line at a time t is given by

$$\vec{S} = f[F\,(\boldsymbol{\vec{S}_1, t}), \boldsymbol{t}]$$

one, Two and Three Dimensional Flows Fluid flow is three-dimensional in nature.

This means that the flow parameters like velocity, pressure and so on vary in all the three coordinate directions. Sometimes simplification is made in the analysis of different fluid flow problems by:

Selecting the appropriate coordinate directions so that appreciable variation of the hydro dynamic parameters take place in only two directions or even in only one.

One-dimensional flow

All the flow parameters may be expressed as functions of time and one space coordinate only.

The single space coordinate is usually the distance measured along the centre-line (not necessarily straight) in which the fluid is flowing.

Example: the flow in a pipe is considered one-dimensional when variations of pressure and velocity occur along the length of the pipe, but any variation over the cross-section is assumed negligible.

In reality, flow is never one-dimensional because viscosity causes the velocity to decrease to zero at the solid boundaries.

If however, the non uniformity of the actual flow is not too great, valuable results may often be obtained from a "one dimensional analysis".

The average values of the flow parameters at any given section (perpendicular to the flow) are assumed to be applied to the entire flow at that section.

Two-dimensional flow

All the flow parameters are functions of time and two space coordinates (say x and y). No variation in z direction.

The same streamline patterns are found in all planes perpendicular to z direction at any instant.

Three dimensional flow

The hydrodynamic parameters are functions of three space coordinates and time.

Translation of a Fluid Element
The movement of a fluid element in space has three distinct features simultaneously.
Translation
Rate of deformation Rotation.

Figure 7.4 shows the picture of a pure translation in absence of rotation and deformation of a fluid element in a two-dimensional flow described by a rectangular cartesian coordinate system.

In absence of deformation and rotation,

a) There will be no change in the length of the sides of the fluid element.

b) There will be no change in the included angles made by the sides of the fluid element.

c) The sides are displaced in parallel direction.

This is possible when the flow velocities u (the x component velocity) and v (the y component velocity) are neither a function of x nor of y, i.e., the flow field is totallyuniform.

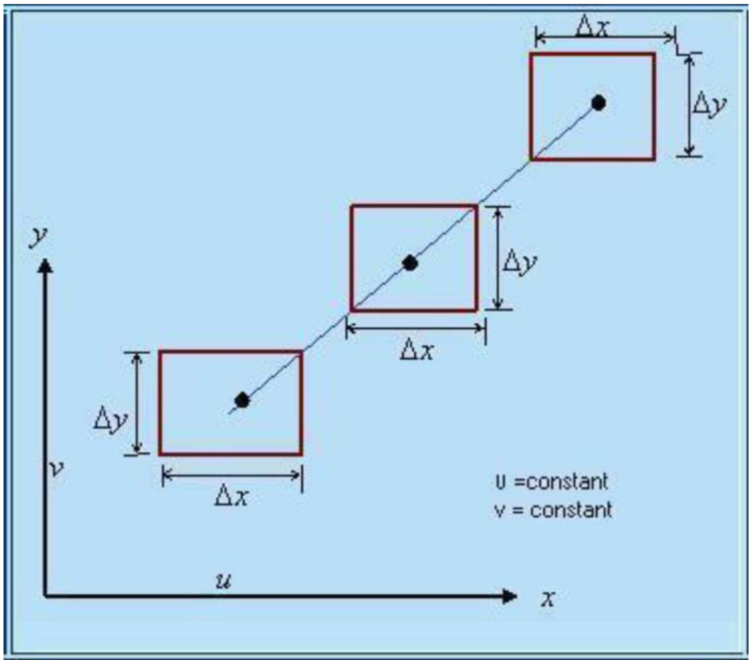


Fig: Fluid Element in pure translation

If a component of flow velocity becomes the function of only one space coordinate along which that velocity component is defined.

For example,

if u = u(:x) and v = v(y), the fluid element ABCD suffers a change in its linear dimensions along with translation

there is no change in the included angle by the sides as shown in Fig

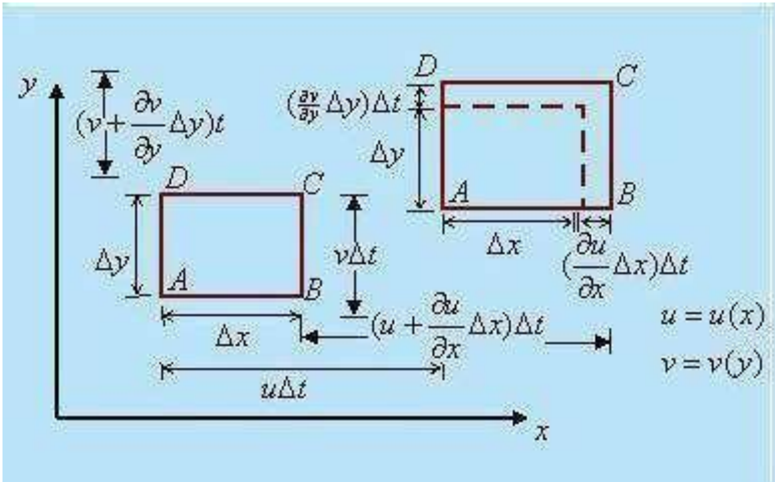


Fig: Fluid Element in Translation with Continuous Linear Deformation

The relative displacement of point B with respect to point A per unit time in x direction is

$$\frac{\partial u}{\partial x}\Delta x$$

$$\frac{\partial v}{\partial y}\Delta y$$

Similarly, the relative displacement of D with respect to A per unit time in y direction is

Translation with Linear Deformations observations from the figure:

Since u is not a function of y and v is not a function of x

All points on the linear element AD move with same velocity in the x direction. All points on the linear element AB move with the same velocity in y direction.

Hence the sides move parallel from their initial position without changing the included angle. This situation is referred to as translation with linear deformation.

Strain rate:

The changes in lengths along the coordinate axes per unit time per unit original lengths are defined as the components of linear deformation or strain rate in the respective directions.

Therefore, linear strain rate component in the x direction

$$\dot{\varepsilon}_{xx} = \frac{\partial u}{\partial x}$$

and, linear strain rate component in y direction

$$\dot{\varepsilon}_{yy} = \frac{\partial v}{\partial y}$$

Rate of Deformation in the Fluid Element

Let us consider both the velocity component u and v are functions of x and y, i.e.,

Point B has a relative displacement in y direction with respect to the point A. Point D has a relative displacement in x direction with respect to point A. The included angle between AB and AD changes.

The fluid element suffers a continuous angular deformation along with the linear deformations in course of its motion.

Rate of Angular deformation:

The rate of angular deformation is defined as the rate of change of angle between the linear segments AB and AD which were initially other perpendicular to each

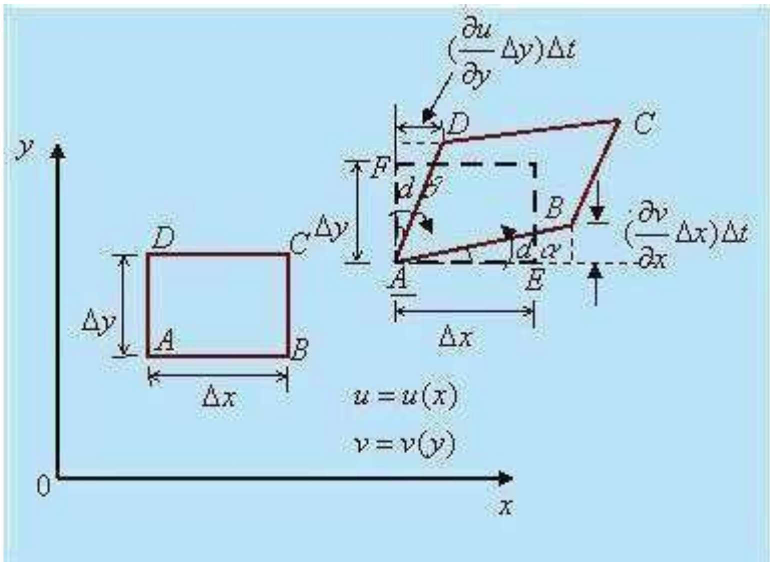


Fig: Fluid element in translation with simultaneous linear and angular deformation rates From the above figure rate of angular deformation,

$$\dot{y}_{xy} = \left(\frac{d\alpha}{dt} + \frac{d\beta}{dt}\right)$$

From the geometry

$$d\alpha = \frac{\partial v}{\partial x} dt$$

$$d\alpha = \lim_{\Delta t \to 0} \left(\frac{\frac{\partial v}{\partial x}\Delta x \Delta t}{\Delta x\left(1 + \frac{\partial u}{\partial x}\Delta t\right)} \right) = \frac{\partial v}{\partial x} dt$$

$$d\beta = \lim_{\Delta t \to 0} \left(\frac{\frac{\partial u}{\partial y}\Delta y \Delta t}{\Delta y\left(1 + \frac{\partial v}{\partial y}\Delta t\right)} \right) = \frac{\partial u}{\partial y} dt$$

Hence,

$$\frac{d\alpha}{dt} + \frac{d\beta}{dt} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)$$

Finally

$$\dot{y}_{xy} = \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)$$

Rotation

The transverse displacement of B with respect to A and the lateral displacement of D with respect to A (Fig. 8.3) can be considered as the rotations of the linear segments AB and AD about A.

This brings the concept of rotation in a flow field.

Definition of rotation at a point:

The rotation at a point is defined as the arithmetic mean of the angular velocities of two perpendicular linear segments meeting at that point.

Example: The angular velocities of AB and AD about A are

$\frac{d\alpha}{dt}$ and $\frac{d\beta}{dt}$ respectively.

Considering the anticlockwise direction as positive, the rotation at A can be written as,

$$\omega_z = \frac{1}{2}\left(\frac{d\alpha}{dt} - \frac{d\beta}{dt}\right)$$

or

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

The suffix z in ω represents the rotation about z-axis.

then u = u (:x, y) and v = v (:x, y) the rotation and angular deformation of a fluid element exist simultaneously.

Special case : Situation of pure Rotation

$\frac{\partial v}{\partial x} = -\frac{\partial u}{\partial y}$, $\dot{\gamma}_{xy} = 0$ and $\omega_z = \frac{\partial v}{\partial x} = -\frac{\partial u}{\partial y}$

observation:

The linear segments AB and AD move with the same angular velocity (both in magnitude and direction).

The included angle between them remains the same and no angular deformation takes place. This situation is
known as pure rotation.

Vorticity

Definition: The vorticity Ω in its simplest form is defined as a vector which is equal to two times the rotation vector

$$\vec{\Omega} = 2\vec{\omega} = \nabla \times \vec{V}$$

For an irrotational flow, vorticity components are zero.

Vortex line:

If tangent to an imaginary line at a point lying on it is in the direction of the Vorticity vector at that point , the line is a vortex line.

The general equation of the vortex line can be written as,

$$\vec{\Omega} \times d\vec{s} = 0$$

In a rectangular cartesian cartesian coordinate system, it becomes

$$\frac{dx}{\Omega_x} = \frac{dy}{\Omega_y} = \frac{dz}{\Omega_z}$$

where,

$\Omega_x = 2\omega_x$

$\Omega_y = 2\omega_y$

$\Omega_z = 2\omega_z$

Vorticity components as vectors:

The vorticity is actually an anti symmetric tensor and its three distinct elements transform like the components of a vector in cartesian coordinates.

This is the reason for which the vorticity components can be treated as vectors.

Existence of Flow

A fluid must obey the law of conservation of mass in course of its flow as it is a material body.

For a Velocity field to exist in a fluid continuum, the velocity components must obey the mass conservation principle.

Velocity components which follow the mass conservation principle are said to constitute a possible fluid flow Velocity components violating this principle, are said to describe an impossible flow.

The existence of a physically possible flow field is verified from the principle of conservation of mass.

The detailed discussion on this is deferred to the next chapter along with the discussion on principles of conservation of momentum and energy.

System Definition

System: A quantity of matter in space which is analyzed during a problem. Surroundings: Everything external to the system.

System Boundary: A separation present between system and surrounding.

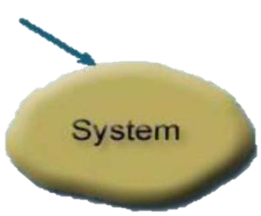
Classification of the system boundary:-

Real solid boundary Imaginary boundary

The system boundary may be further classified as:-

Fixed boundary or Control Mass System Moving boundary or Control Volume System

The choice of boundary depends on the problem being analyzed.



Classification of Systems

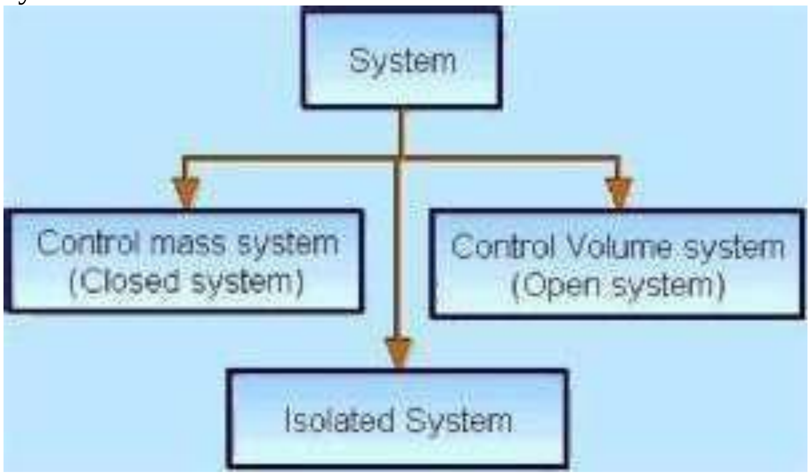


Fig: Types of System

Control Mass System (Closed System)

Its a system of fixed mass with fixed identity.

This type of system is usually referred to as "closed system".

There is no mass transfer across the system boundary. Energy transfer may take place into or out of the system.

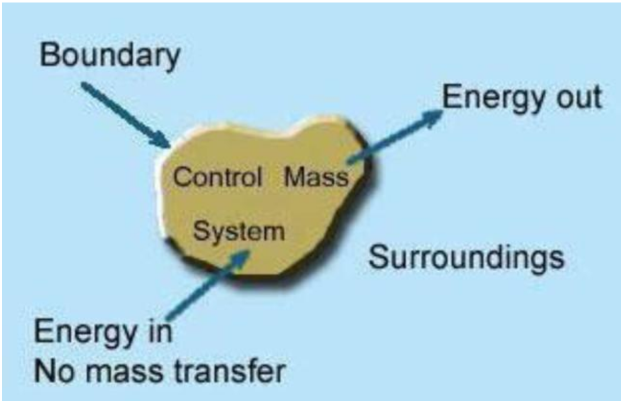


Fig A Control Mass System or Closed System Control Volume System (open System)

Its a system of fixed volume.

Mass transfer can take place across a control volume. Energy transfer may also occur into or out of the system.

A control volume can be seen as a fixed region across which mass and energy transfers are studied.

Its the boundary of a control volume across which the transfer of both mass and energy takes

Control Surface- place.

The mass of a control volume (open system) may or may not be fixed.

then the net influx of mass across the control surface equals zero then the mass of the system is fixed and vice-versa.

The identity of mass in a control volume always changes unlike the case for a control mass system (closed system).

Most of the engineering devices, in general, represent an open system or control volume.

Example:-

Heat exchanger - Fluid enters and leaves the system continuously with the transfer of heat across the system boundary.

Pump - A continuous flow of fluid takes place through the system with a transfer of mechanical energy from the surroundings to the system.

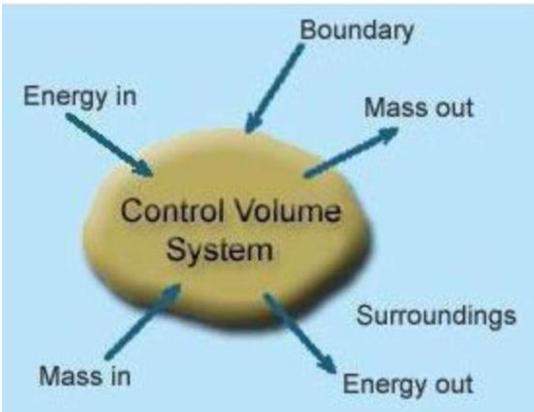


Fig: A Control Volume System or open System Isolated System

Its a system of fixed mass with same identity and fixed energy.

No interaction of mass or energy takes place between the system and the surroundings. In more informal words an isolated system is like a closed shop amidst a busy market.

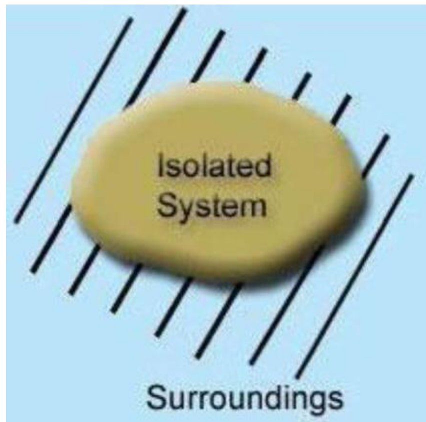


Fig: An Isolated System

Conservation of Mass - The Continuity Equation Law of conservation of mass

The law states that mass can neither be created nor be destroyed. Conservation of mass is inherent to a control mass system (closed system).

The mathematical expression for the above law is stated as:

$\Delta m/\Delta t = 0$,

where m # mass of the system

For a control volume (Fig.9.5), the principle of conservation of mass is stated as

Rate at which mass enters # Rate at which mass leaves the region + Rate of accumulation of mass in the region

or

Rate of accumulation of mass in the control volume = Net rate of mass efflux from the control volume

The above statement expressed analytically in terms of velocity and density field of a flow is known as the equation of continuity.

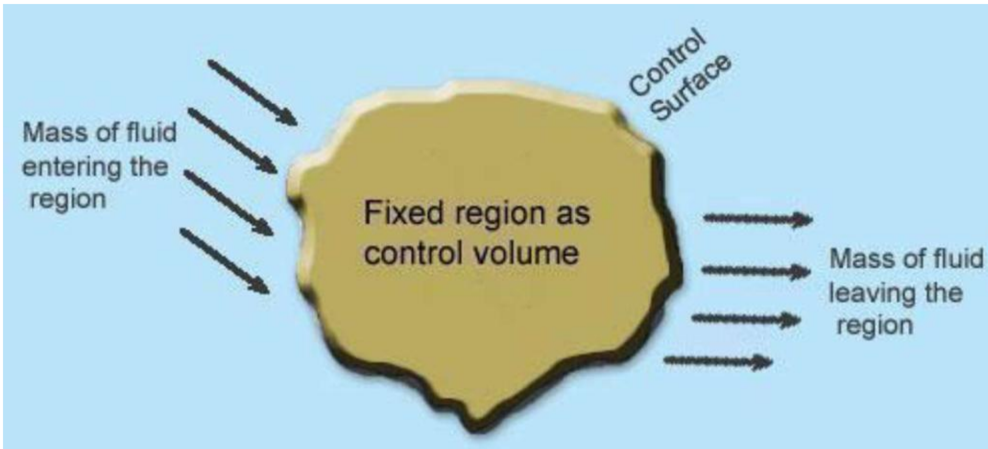


Fig: A Control Volume in a Flow Field Continuity Equation - Differential Form

The point at which the continuity equation has to be derived, is enclosed by an elementary control volume.

The influx, efflux and the rate of accumulation of mass is calculated across each surface within the control volume.

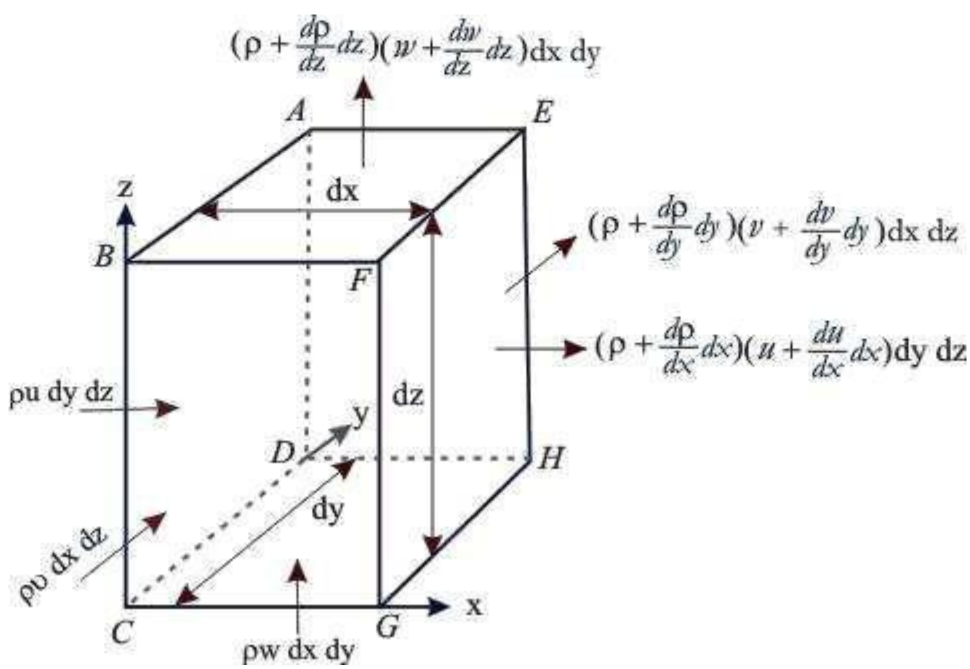


Fig: A Control Volume Appropriate to a Rectangular Cartesian Coordinate System

Consider a rectangular parallelopiped in the above figure as the control volume in a rectangular cartesian frame of coordinate axes.
Net efflux of mass along x -axis must be the excess outflow over inflow across faces normal to x -axis.

EFGH will be $u+\frac{\partial u}{\partial x}dx$ and $\rho+\frac{\partial \rho}{\partial x}dx$ respectively (neglecting the higher order terms in δx

Therefore, the rate of mass entering the control volume through face ABCD # ρu dy dz.

The rate of mass leaving the control volume through face EFGH will be

$$=\left(\rho+\frac{\partial \rho}{\partial x}dx\right)\left(u+\frac{\partial u}{\partial x}dx\right)dydz$$

$$=\left(\rho u+\frac{\partial}{\partial x}(\rho u)dx\right)dydz$$ (neglecting the higher order terms in dx)

Similarly influx and efflux take place in all y and z directions also. Rate of accumulation for a point in a flow field

$$\frac{\partial m}{\partial t}=\frac{\partial}{\partial t}\rho(d\forall)=\frac{\partial \rho}{\partial t}d\forall$$

Using, Rate of influx # Rate of Accumulation + Rate of Efflux

$$\rho u dydz+\rho v dxdz+\rho w dxdy=\frac{\partial \rho}{\partial t}d\forall+(\rho+\frac{\partial \rho}{\partial x}dx)(u+\frac{\partial u}{\partial x}dx)dydz$$
$$+(\rho+\frac{\partial \rho}{\partial y}dy)(v+\frac{\partial v}{\partial y}dy)dxdz+(\rho+\frac{\partial \rho}{\partial z}dz)(w+\frac{\partial w}{\partial z}dz)dxdy$$

Transferring everything to right side

$$0=\left[\left(\rho\frac{\partial u}{\partial x}+u\frac{\partial \rho}{\partial x}\right)+\left(\rho\frac{\partial v}{\partial y}+v\frac{\partial \rho}{\partial y}\right)+\left(\rho\frac{\partial w}{\partial z}+w\frac{\partial \rho}{\partial z}\right)\right]dxdydz+\left(\frac{\partial \rho}{\partial t}\right)d\forall$$

$$\Rightarrow\left[\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)\right]d\forall=0$$

This is the Equation of Continuity for a compressible fluid in a rectangular cartesian coordinate system.

Continuity Equation - Vector Form

The continuity equation can be written in a vector form as

$$\frac{\partial \rho}{\partial t}+\left(\frac{\partial}{\partial x}\hat{i}+\frac{\partial}{\partial y}\hat{j}+\frac{\partial}{\partial z}\hat{k}\right).\left[\rho u\hat{i}+\rho v\hat{j}+\rho w\hat{k}\right]=0$$

or,

$$\frac{\partial \rho}{\partial t}+\nabla.(\rho\vec{V})=0$$

where $\vec{V}=u\hat{i}+v\hat{j}+w\hat{k}$ is the velocity of the point In case of a steady flow,

$$\frac{\partial \rho}{\partial t}=0$$

Hence becomes

$$\nabla.(\rho\vec{V})=0$$

In a rectangular cartesian coordinate system

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

Equation represents the continuity equation for a steady flow. In case of an incompressible flow,

Hence,

$$\frac{\partial \rho}{\partial t}=0$$

Moreover

$$\nabla.(\rho\vec{V})=\rho\nabla.(\vec{V})$$

Therefore, the continuity equation for an incompressible flow becomes

$$\nabla.(\vec{V}) = 0$$

$$\text{or,} \quad \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

In cylindrical polar coordinates eq.9.7 reduces to

$$\frac{1}{R}\frac{\partial}{\partial R}(R^2 V_R) + \frac{1}{\sin\varphi}\frac{\partial V_\theta}{\partial \theta} + \frac{1}{\sin\varphi}\frac{\partial (V_\varphi \sin\varphi)}{\partial \varphi} = 0$$

Continuity Equation - A Closed System Approach

we know that the conservation of mass is inherent to the definition of a closed system as Dm/Dt # 0 (where m is the mass of the closed system).

However, the general form of continuity can be derived from the basic equation of mass conservation of a system.

Derivation :-

Let us consider an elemental closed system of volume V and density ρ.

$$\frac{Dm}{Dt} = 0 \Rightarrow \frac{D}{Dt}(\rho \Delta \forall) = 0$$

$$\Rightarrow \Delta\forall \frac{D\rho}{Dt} + \rho \frac{D(\Delta\forall)}{Dt} = 0$$

$$\Rightarrow \frac{D\rho}{Dt} + \frac{\rho}{\Delta\forall}\frac{D(\Delta\forall)}{Dt} = 0$$

$$\Rightarrow \frac{\partial \rho}{\partial t} + u\frac{\partial \rho}{\partial x} + v\frac{\partial \rho}{\partial y} + w\frac{\partial \rho}{\partial z} + \frac{\rho}{\Delta\forall}\frac{D\Delta\forall}{Dt} = 0$$

Now $\frac{1}{\Delta\forall}\frac{D\Delta\forall}{Dt} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial v} + \frac{\partial w}{\partial z}$

$$\Rightarrow \frac{\partial \rho}{\partial t}+u\frac{\partial \rho}{\partial x}+v\frac{\partial \rho}{\partial y}+w\frac{\partial \rho}{\partial z}+\rho\frac{\partial u}{\partial x}+\rho\frac{\partial v}{\partial y}+\rho\frac{\partial w}{\partial z}=0$$

$$\Rightarrow \frac{\partial \rho}{\partial t}+\left(u\frac{\partial \rho}{\partial x}+\rho\frac{\partial u}{\partial x}\right)+\left(v\frac{\partial \rho}{\partial y}+\rho\frac{\partial v}{\partial y}\right)+\left(w\frac{\partial \rho}{\partial z}+\rho\frac{\partial w}{\partial z}\right)=0$$

$$\Rightarrow \frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0$$

$$\frac{\partial \rho}{\partial t}+\nabla.(\rho\vec{V})=0$$

Stream Function

Let us consider a two-dimensional incompressible flow parallel to the x - y plane in a rectangular cartesian coordinate system.
The flow field in this case is defined by

u = u(:x, y,
t) v = v(:x, y,
t) w = 0

The equation of continuity is

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

If a function $\psi(x, y, t)$ is defined in the manner so that it automatically satisfies the equation of continuity (Eq. (10.1)), then the function is known as stream function.
Note that for a steady flow, ψ is a function of two variables x and y only.

Constancy of ψ on a Streamline

Since ψ is a point function, it has a value at every point in the flow field. Thus a change in the stream function ψ can be written as

$$d\psi=\frac{\partial \psi}{\partial x}dx+\frac{\partial \psi}{\partial y}dy=-vdx+udy$$

The equation of a streamline is given by

$$\frac{u}{dx} = \frac{v}{dy} \quad \text{or} \quad udy - vdx = 0 \text{(since tangent dy/dx equals the velocity v/u)}$$

It follows that $d\psi = 0$ on a streamline.This implies the value of ψ is constant along a streamline. Therefore, the equation of a streamline can be expressed in terms of stream function as

$\psi(x, y)$ = constant

Once the function ψ is known, streamline can be drawn by joining the same values of ψ in the flow field.

Stream function for an irrotational flow

In case of a two-dimensional irrotational flow

$$\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 \qquad \Rightarrow \frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial x}\right) - \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial y}\right) = 0$$

$$\Rightarrow -\frac{\partial^2 \psi}{\partial x^2} - \frac{\partial^2 \psi}{\partial y^2} = 0 \qquad \Rightarrow \frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

$$\Rightarrow \psi_{xx} + \psi_{yy} = 0$$

$$\Rightarrow \nabla^2 \psi = 0$$

Conclusion drawn:For an irrotational flow, stream function satisfies the Laplace's equation

Physical Significance of Stream Funtion ψ

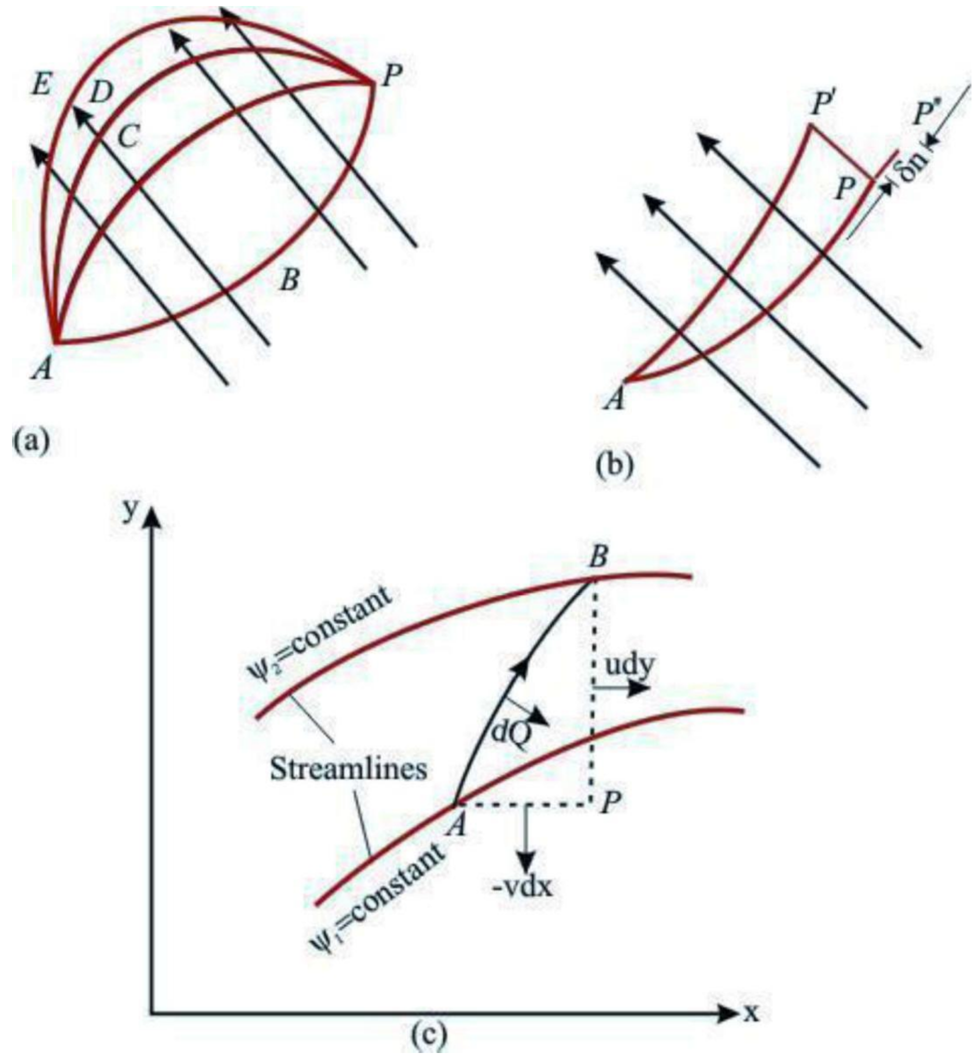


Fig: Physical Interpretation of Stream Function

Let A be a fixed point, whereas P be any point in the plane of the flow. The points A and P are joined by the arbitrary lines ABP and ACP. For an

P) is a function only of the position P. This function is known as thestream function ψ.

The value of ψ at P represents the volume flow rate across any line joining

The value of ψ thus remains same at P' Since P' was taken as any point on the streamline through P, it follows that ψ is constant along a streamline. Thus the flow may be represented by a series of streamlines at equal increments of ψ.

value at A made arbitrarily zero. If a point P' is considered (Fig. 10.1b),PP' being along a then the rate of flow across the curve joining A to P' must be the same as

$$\int dQ = \int_A^B d\psi$$

and P.

$$\therefore Q = \int_A^B d\psi = \psi_2 - \psi_1$$

The stream function, in a polar coordinate system is defined as

$$V_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \quad \text{and} \quad V_\theta = -\frac{\partial \psi}{\partial r}$$

The expressions for Vr and Vθ in terms of the stream function automatically satisfy the equation of continuity given by

$$\frac{\partial}{\partial r}(V_r r) + \frac{\partial}{\partial \theta}(V_\theta) = 0$$

Stream Function in Three Dimensional and Compressible Flow Stream Function in Three Dimensional Flow

In case of a three dimensional flow, it is not possible to draw a streamline with a single stream function.

An axially symmetric three dimensional flow is similar to the two-dimensional case in a sense that the flow field is the same in every plane containing the axis of symmetry.

The equation of continuity in the cylindrical polar coordinate system for an incompressible flow is given by the following equation

$$\frac{\partial V_r}{\partial r} + \frac{V_r}{r} + \frac{1}{r}\frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} = 0$$

$$\frac{1}{r}\frac{\partial V_\theta}{\partial \theta}$$

For an axially symmetric flow (the axis r = 0 being the axis of symmetry), and simplified equation is satisfied by functions defined as

$$rV_r = -\frac{\partial \psi}{\partial z}, \qquad rV_z = \frac{\partial \psi}{\partial r}$$

The function ψ, defined by the Eq.(10.4) in case of a three dimensional flow with an axial symmetry, is called the stokes stream function.

Stream Function in Compressible Flow

For compressible flow, stream function is related to mass flow rate instead of volume flow rate because of the extra density term in the continuity equation (unlike incompressible flow)

The continuity equation for a steady two-dimensional compressible flow is given by

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0$$

Hence a stream function ψ is defined which will satisfy the above equation of continuity as

ρ is used to retain the unit of ψ same as that in the case of an incompressible flow. Physically, the difference in stream function between any two streamlines multiplied by the streamlines.

UNIT-3
FLUID DYNAMICS

Definition

System: A quantity of matter in space which is analyzed during a problem. Surroundings: Everything external to the system.

System Boundary: A separation present between system and surrounding.

Classification of the system boundary:-

Real solid boundary Imaginary boundary

The system boundary may be further classified as:-

Conservation of Momentum$ Momentum Theorem

In Newtonian mechanics, the conservation of momentum is defined by

Newton's second law of motion.

Newton's Second Law of Motion

The rate of change of momentum of a body is proportional to the impressed action and takes place in the direction of the impressed action.

If a force acts on the body ,linear momentum is implied.

If a torque (moment) acts on the body,angular momentum is implied. Reynolds Transport

Theorem

A study of fluid flow by the Eulerian approach requires a mathematical modeling for a control volume either in differential or in integral form. Therefore the physical statements of the principle of conservation of mass, momentum and energy with reference to a control volume become necessary.

This is done by invoking a theorem known as the Reynolds transport theorem which relates the control volume concept with that of a control mass system in terms of a general property of the system.

Statement of Reynolds Transport Theorem

The theorem states that "the time rate of increase of property N within a control mass system is equal to the time rate of increase of property N within the control volume plus the net rate of efflux of the property N across the control surface".

Equation of Reynolds Transport Theorem

After deriving Reynolds Transport Theorem according to the above statement we get

$$\left(\frac{dN}{dt}\right)_{CMS} = \frac{\partial}{\partial t}\iiint_{CV} \eta\rho \, d\forall + \iint_{CS} \eta\rho\vec{V}.\,d\vec{A}$$

In this equation

N - flow property which is transported

η- intensive value of the flow property

Analysis of Finite Control Volumes - the application of momentum theorem we'll see the application of momentum theorem in some practical cases of inertial and non- inertial control volumes.

Inertial Control Volumes

Applications of momentum theorem for an inertial control volume are described with reference to three distinct types of practical problems, namely

Forces acting due to internal flows through expanding or reducing pipe bends. Forces on stationary and moving vanes due to impingement of fluid jets.

Jet propulsion of ship and aircraft moving with uniform velocity. Non-inertial Control Volume

A good example of non-inertial control volume is a rocket engine which works on the principle of jet propulsion.

we shalll discuss each example seperately in the following slides.

Euler's Equation along a Streamline

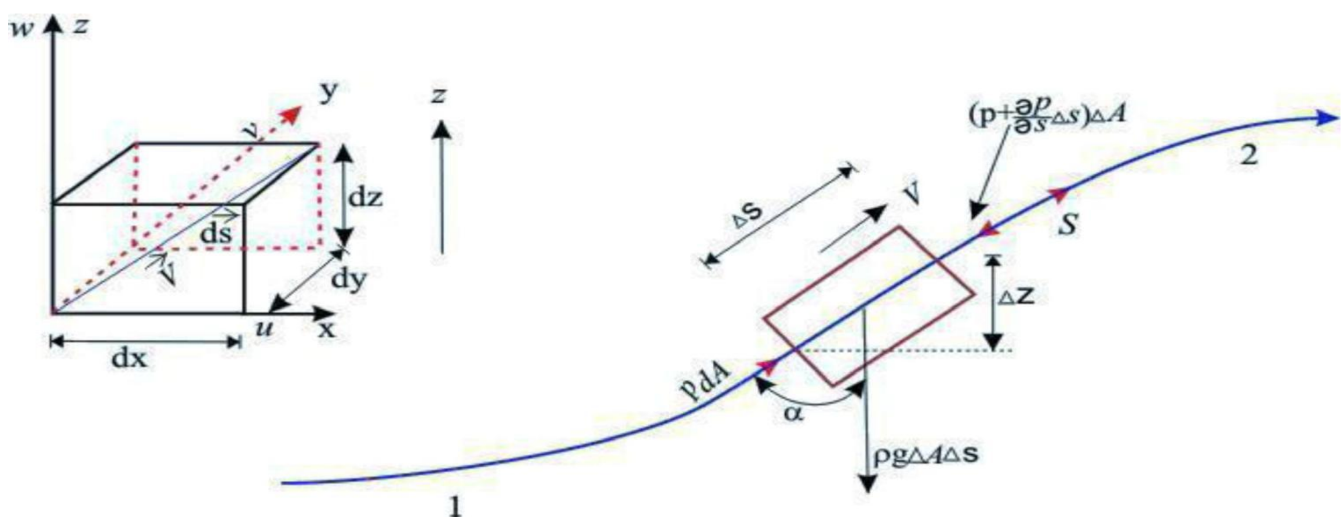


Fig: Force Balance on a Moving Element Along a Streamline Derivation

Euler's equation along a streamline is derived by applying Newton's second law of motion to a fluid element moving along a streamline. Considering gravity as the only body force component acting vertically downward (Fig. 12.3), the net external force acting on the fluid element along the directions can be written as

$$\cos\alpha = \lim_{\Delta s \to 0} \frac{\Delta z}{\Delta s} = \frac{dz}{ds}$$

Hence, the final form of Eq. (12.9) becomes

$$\rho \frac{DV}{Dt} = -\frac{\partial p}{\partial s} - \rho g \frac{dz}{ds}$$

$$\frac{\partial V}{\partial t} + V \frac{\partial V}{\partial s} = -\frac{1}{\rho}\frac{\partial p}{\partial s} - g\frac{dz}{ds}$$

Equation (12.10) is the Euler's equation along a streamline.

Let us consider $d\vec{s}$ along the streamline sothat

$$d\vec{s} = \vec{i}dx + \vec{j}dy + \vec{k}dz$$

Again, we can write from Fig. 12.3

$$\frac{dx}{ds} = \frac{u}{V}, \quad \frac{dy}{ds} = \frac{v}{V} \text{ and } \frac{dz}{ds} = \frac{w}{V}$$

The equation of a streamline is given by

$$\vec{u}dy = \vec{v}dx; \qquad udz = wdx; \qquad vdz = wdy$$

or, $\begin{vmatrix} i & j & k \\ u & v & w \\ dx & dy & dz \end{vmatrix} = 0$ which finally leads to

Multiplying Eqs (12.7a), (12.7b) and (12.7c) by dx, dy and dz respectively and then substituting the above mentioned equalities, we get

$$\rho\left(u\frac{\partial u}{\partial t}\cdot\frac{ds}{V} + u\frac{\partial u}{\partial x}dx + u\frac{\partial u}{\partial y}dy + u\frac{\partial u}{\partial z}dz\right) = -\frac{\partial p}{\partial x}dx + X_x dx$$

$$\rho\left(v\frac{\partial v}{\partial t}\cdot\frac{ds}{V}+v\frac{\partial v}{\partial x}dx+v\frac{\partial v}{\partial y}dy+v\frac{\partial v}{\partial z}dz\right)=-\frac{\partial p}{\partial y}dy+X_y dy$$

$$\rho\left(w\frac{\partial w}{\partial t}\cdot\frac{ds}{V}+w\frac{\partial w}{\partial x}dx+w\frac{\partial w}{\partial y}dy+w\frac{\partial w}{\partial z}dz\right)=-\frac{\partial p}{\partial z}dy+X_z dz$$

Adding these three equations, we can write

$$\rho\left(\frac{ds}{V}\cdot\frac{\partial}{\partial t}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)+\frac{\partial}{\partial x}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)dx+\frac{\partial}{\partial y}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)dy+\frac{\partial}{\partial z}\left(\frac{u^2}{2}+\frac{v^2}{2}+\frac{w^2}{2}\right)dz\right)$$

$$\rho\left(\frac{ds}{V}\cdot\frac{\partial}{\partial t}(\frac{V^2}{2})+\frac{\partial}{\partial x}\left(\frac{V^2}{2}\right)dx+\frac{\partial}{\partial y}\left(\frac{V^2}{2}\right)dy+\frac{\partial}{\partial z}\left(\frac{V^2}{2}\right)dz\right)$$

$$\rho\left[\frac{\partial V}{\partial t}+V(\frac{\partial V}{\partial x}\cdot\frac{dx}{ds}+\frac{\partial V}{\partial y}\cdot\frac{dy}{ds}+\frac{\partial V}{\partial z}\cdot\frac{dz}{ds})\right]=-\left(\frac{\partial p}{\partial x}\cdot\frac{dx}{ds}+\frac{\partial p}{\partial y}\cdot\frac{dy}{ds}+\frac{\partial p}{\partial z}\cdot\frac{dz}{ds}\right)-\rho g\frac{dz}{ds}$$

Hence, $$\boxed{\rho\left[\frac{\partial V}{\partial t}+V\frac{\partial V}{\partial s}\right]=-\frac{\partial p}{\partial s}-\rho g\frac{dz}{ds}}$$

This is the more popular form of Euler's equation because the velocity vector in a flow field is always directed along the streamline.

A Control Volume Approach for the Derivation of Euler's Equation

Euler's equations of motion can also be derived by the use of the momentum theorem for a control volume. Derivation

In a fixed x, y, z axes (the rectangular cartesian coordinate system), the parallelopiped which was taken earlier as a control mass system is now considered as a control volume

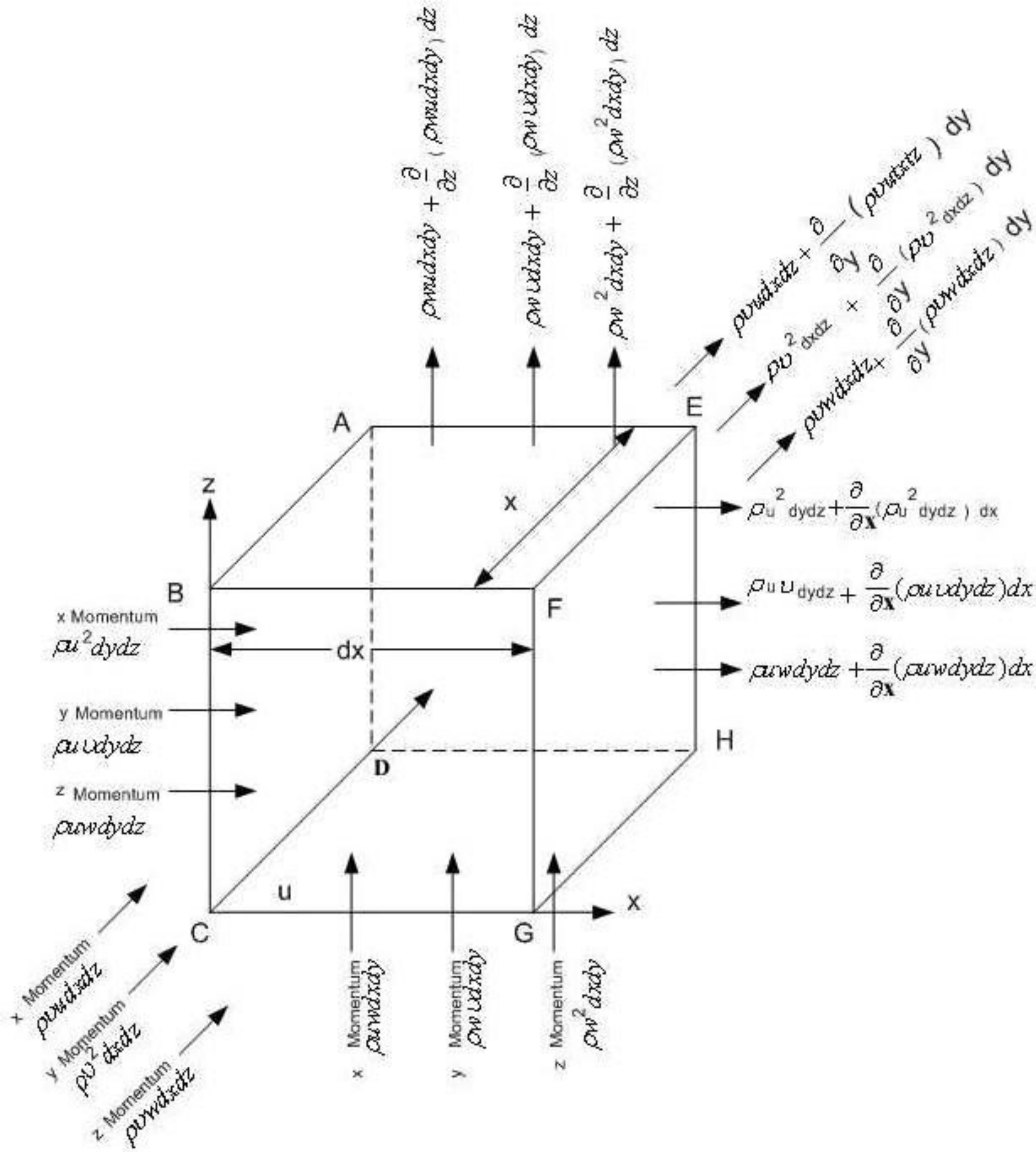


Fig: A Control Volume used for the derivation of Euler's Equation

we can define the velocity vector $\vec{V}$ and the body force per unit volume $\rho\vec{X}$ as

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

$$\rho\vec{X} = \vec{i}\,\rho X_x + \vec{j}\,\rho X_y + \vec{k}\,\rho X_z$$

The rate of x momentum influx to the control volume through the face ABCD is equal to ρu^2 dy dz. The rate of x momentum efflux from the control volume through the face EFGH equals $\rho u^2 dy\, dz + \frac{\partial}{\partial x}\left(\rho u^2 dy\, dz\right)dx$

Therefore the rate of net efflux of x momentum from the control volume due to the faces perpendicular to the x direction (faces ABCD and EFGH)

$\frac{\partial}{\partial x}(\rho u^2)\mathrm{d}\forall$ where, $\mathrm{d}\forall$, the elemental volume # dx dy dz. Similarly,

The rate of net efflux of x momentum due to the faces perpendicular to the y direction (face BCGF and ADHE) # $\frac{\partial}{\partial y}(\rho uv)\mathrm{d}\forall$

The rate of net efflux of x momentum due to the faces perpendicular to the z direction (faces DCGH and ABFE) # $\frac{\partial}{\partial z}(\rho uw)\mathrm{d}\forall$

Hence, the net rate of x momentum efflux from the control volume becomes

$$\left[\frac{\partial}{\partial z}(\rho u^2) + \frac{\partial}{\partial z}(\rho uv) + \frac{\partial}{\partial z}(\rho uw)\right]\mathrm{d}\forall$$

The time rate of increase in x momentum in the control volume can be written as

$$\frac{\partial}{\partial t}(\rho u\,\mathrm{d}\forall) = \frac{\partial}{\partial t}(\rho u)\mathrm{d}\forall$$

(Since, $\mathrm{d}\forall$, by the definition of control volume, is invariant with time)

Applying the principle of momentum conservation to a control volume (Eq. 4.28b), we get

$$\frac{\partial}{\partial t}(\rho u)+\frac{\partial}{\partial x}(\rho u^2)+\frac{\partial}{\partial y}(\rho uv)+\frac{\partial}{\partial z}(\rho uw)=\rho X_x-\frac{\partial p}{\partial x}$$

The equations in other directions y and z can be obtained in a similar way by considering the y momentum and z momentum fluxes through the control volume as

$$\frac{\partial}{\partial t}(\rho v)+\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho v^2)+\frac{\partial}{\partial z}(\rho vw)=\rho X_y-\frac{\partial p}{\partial y}$$

$$\frac{\partial}{\partial t}(\rho w)+\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial y}(\rho vw)+\frac{\partial}{\partial z}(\rho w^2)=\rho X_z-\frac{\partial p}{\partial z}$$

The typical form of Euler's equations given by Eqs (12.11a), (12.11b) and (12.11c) are known as the conservative forms. Bernoulli's Equation Energy Equation of an ideal Flow along a

Streamline

Euler's equation (the equation of motion of an inviscid fluid) flow with gravity as the only body force can be written as

$$V\frac{dV}{ds}=-\frac{1}{\rho}\frac{dp}{ds}-g\frac{dz}{ds}$$

Application of a force through a distance ds along the streamline would physically imply work interaction. Therefore an equation for conservation of energy along a streamline can be obtained by integrating the Eq. (13.6) with respect to ds as

$$\int V\frac{dV}{ds}ds=-\int\frac{1}{\rho}\frac{dp}{ds}ds-\int g\frac{dz}{ds}ds$$

or, $$\frac{V^2}{2}+\int\frac{dp}{\rho}+gz=C$$

there C is a constant along a streamline. In case of an incompressible flow, Eq. can be written as

$$\frac{P}{\rho}+\frac{V^2}{2}+gz=C$$

The Eqs are based on the assumption that no work or heat interaction between a fluid element and the surrounding takes place. The first term of the Eq. (13.8) represents the flow work per unit mass, the second term represents the kinetic energy per unit mass and the third term represents the potential energy per unit mass. Therefore the sum of three terms in the left hand side can be considered as the total mechanical energy per unit mass which remains constant along a streamline for a steady inviscid and incompressible flow of fluid. Hence the Eq. (13.8) is also known as Mechanical energy equation.

This equation was developed first by Daniel Bernoulli in 1738 and is therefore referred to as

$$\frac{P}{\rho g}+\frac{V^2}{2g}+z=C_1(\text{con stant})$$

In a fluid flow, the energy per unit weight is termed as head. Accordingly, equation 13.9 can be interpreted as

Pressure head + Velocity head + Potential head #Total head (total energy per unit weight).

Bernoulli's Equation with Head Loss

The derivation of mechanical energy equation for a real fluid depends much on the information about the frictional work done by a moving fluid element and is excluded from the scope of the book. However, in many practical situations, problems related to real fluids
with the help of a modified form of Bernoulli's equation as
can be analysed

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2+h_f$$

where, h_f represents the frictional work done (the work done against the fluid friction) per unit weight of a fluid element while moving from a station 1 to 2 along a streamline in the direction of flow. The term h_f is usually referred to as head loss between 1 and 2, since it amounts to the loss in total mechanical energy per unit weight between points 1 and 2 on a streamline due to the effect of fluid friction or viscosity. It physically signifies that the difference in the total mechanical energy between stations 1 and 2 is dissipated intointermolecular or thermal energy and is expressed as loss of head h_f in Eq. The term head loss, is conventionally symbolized as h_L instead of h_f in dealing with practical problems. For an inviscid flow h_L # 0, and the total mechanical energy is constant along a streamline.

Bernoulli's Equation In Irrotational Flow

In the previous we have obtained Bernoulli's equation

$$\boxed{\frac{p}{\rho}+\frac{V^2}{2}+gz=C}$$

Thus, the value of C in the above equation is constant only along a streamline and should essentially vary from streamline to streamline.

The equation can be used to define relation between flow variables at point B on the streamline and at point A, along the same streamline. So, in order to apply this equation, one should have knowledge of velocity field beforehand. This is one of the limitations of application of Bernoulli's equation.

Irrotationality of flow field

Under some special condition, the constant C becomes invariant from streamline to field.
streamline and the Bernoulli's equation is applicable with same value of C to the entire flow
The typical condition is the irrotationality of flow field.

Let us consider a steady two dimensional flow of an ideal fluid in a rectangular Cartesian coordinate system. The velocity field is given by

$$\vec{V} = \vec{i}u + \vec{j}v$$

hence the condition of irrotationality is

$$\nabla \times \vec{V} = \left\{ \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right\} = 0$$

$$\frac{\partial u}{\partial y} = \frac{\partial v}{\partial x}$$

The steady state Euler's equation can be written as

$$\rho \left\{ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right\} = -\frac{\partial p}{\partial x}$$

$$\rho \left\{ u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right\} = -\frac{\partial p}{\partial y} \quad -\rho g$$

we consider the y-axis to be vertical and directed positive upward. From the condition of irrotationality given by the Eq. we substitute in $\frac{\partial v}{\partial x}$

place of in $\frac{\partial u}{\partial y}$ the Eq. 14.2a and in $\frac{\partial u}{\partial y}$ place of in the $\frac{\partial v}{\partial x}$ Eq.. This

results in

$$\left\{ u \frac{\partial u}{\partial x} + v \frac{\partial v}{\partial x} \right\} = -\frac{1}{\rho} \frac{\partial p}{\partial x}$$

$$\left\{ u \frac{\partial u}{\partial y} + v \frac{\partial v}{\partial y} \right\} = -\frac{1}{\rho} \frac{\partial p}{\partial y} \quad -g$$

Now multiplying by 'dx' and eqn by 'dy' and then adding these two equations we have

$$u\left\{\frac{\partial u}{\partial x}dx+\frac{\partial u}{\partial y}dy\right\}+v\left\{\frac{\partial v}{\partial x}dx+\frac{\partial v}{\partial y}dy\right\}=-\frac{1}{\rho}\left\{\frac{\partial p}{\partial x}dx+\frac{\partial p}{\partial y}dy\right\}-gdy$$

This can be physically interpreted as the equation of conservation of energy for an arbitrary displacement

$d\vec{r}=\vec{i}dx+\vec{j}dy$. Since, u, v and p are functions of x and y, we can write

$$du=\frac{\partial u}{\partial x}dx+\frac{\partial u}{\partial y}dy$$

$$dv=\frac{\partial v}{\partial x}dx+\frac{\partial v}{\partial y}dy$$

$$dp=\frac{\partial p}{\partial x}dx+\frac{\partial p}{\partial y}dy$$

$$udu+vdv=-\frac{1}{\rho}dp-gdy$$

$$d\left\{\frac{u^2}{2}\right\}+d\left\{\frac{v^2}{2}\right\}=-\frac{1}{\rho}dp-gdy$$

$$d\left\{\frac{u^2+v^2}{2}\right\}=-\frac{1}{\rho}dp-gdy$$

$$d\left\{\frac{V^2}{2}\right\}=-\frac{1}{\rho}dp-gdy$$

The integration of Eq. 14.6 results in

$$\int\frac{dp}{\rho}+\frac{V^2}{2}+gy=C$$

For an incompressible flow,

$$\boxed{\frac{p}{\rho}+\frac{V^2}{2}+gy=C}$$

$$V_\theta = \frac{C}{r}$$

The constant has the same value in the entire flow field, since no restriction was made in the choice of dr which was considered as an arbitrary displacement in evaluating the work.

In deriving the displacement ds was considered along a streamline. Therefore, the total mechanical energy remains constant everywhere in an inviscid and irrotational flow, while it is constant only along a streamline for an inviscid but rotational flow.

The equation of motion for the flow of an inviscid fluid can be written in a vector form as

$$\boxed{\frac{D\vec{V}}{Dt} = -\frac{\nabla p}{\rho} + \vec{X}}$$

where D$\vec{V}$ is the body force vector per unit mass

Plane Circular Vortex Flows

Plane circular vortex flows are defined as flows where streamlines are concentric circles. Therefore, with respect to a polar coordinate system with the centre of the circles as the origin or pole, the velocity field can be described as

where V

The equation of continuity for a two dimensional incompressible flow in a polar coordinate system is

$$\frac{\partial V_r}{\partial r} + \frac{V_r}{r} + \frac{1}{r}\frac{\partial V_\theta}{\partial \theta} = 0$$

$$\frac{\partial H}{\partial r} = \frac{V_\theta}{g}\left(\frac{dV_\theta}{dr} + \frac{V_\theta}{r}\right)$$

Free Vortex Flows

Free vortex flows are the plane circular vortex flows where the total mechanical energy remains constant in the entire flow field. There is neither any addition nor any destruction of energy in the flow field.

Therefore, the total mechanical energy does not vary from streamline to streamline. Hence we have, Integration of equation gives

This describes the velocity field in a free vortex flow, where C is a constant in the entire flow field. The vorticity in a polar coordinate system is defined by -

$$\Omega = \frac{\partial V_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta} + \frac{V_\theta}{r}$$

In case of vortex flows, it can be written as

$$\Omega = \frac{dV_\theta}{dr} + \frac{V_\theta}{r}$$

Therefore we conclude that a free vortex flow is irrotational, and hence, it is also referred to asirrotational vortex.

It has been shown before that the total mechanical energy remains same throughout in an irrotational flow field. Therefore, irrotationality is a direct consequence of the constancy of total mechanical energy in the entire flow field and vice versa.

The interesting feature in a free vortex flow is that as $r \rightarrow 0,\ V_\theta \rightarrow \infty$
It mathematically signifies a point of singularity at r # 0 which, in practice, is impossible. In fact, the definition of a free vortex flow cannot be extended as r # 0 is approached.

In a real fluid, friction becomes dominant as **r→0** and so a fluid in this central region tends to rotate as a solid body. Therefore, the singularity at r # 0 does not render the theory of irrotational vortex useless, since, in practical problems, our concern is with conditions away from the central core.

Pressure Distribution in a Free Vortex Flow

Pressure distribution in a vortex flow is usually found out by integrating the equation of motion in the r direction. The equation of motion in the radial direction for a vortex flow can be written

$$\frac{1}{\rho}\frac{dp}{dr} = \frac{V_\theta^2}{r} - g\cos\theta$$

$$\frac{1}{\rho}\frac{dp}{dr} = \frac{V_\theta^2}{r} - g\frac{dz}{dr}$$

as Integrating with respect to dr, and considering the flow to be incompressible we have,

$$\frac{p}{\rho} = \int \frac{V_\theta^2}{r} dr - gz + A$$

For a free vortex flow,

$$V_\theta = \frac{C}{r}$$

If the pressure at some radius r # r $_a$, is known to be the atmospheric pressure p_{atm} then equation (14.14) can be written as

$$\frac{p - p_{atm}}{\rho} = \frac{C^2}{2}\left(\frac{1}{r_a^2} - \frac{1}{r^2}\right) - g(z - z_a)$$

$$= \frac{\left(V_\theta^2\right)_{r=a}}{2} - \frac{V_\theta^2}{2} - g(z - z_a)$$

where z and z_a are the vertical elevations (measured from any arbitrary datum) at r and r_a.

Equation can also be derived by a straight forward application and r = r.

of Bernoulli's equation between any two points at r = r

In a free vortex flow total mechanical energy remains constant. There is neither any energy interaction between an outside source and the flow, nor is there any dissipation of mechanical energy within the flow. The fluid rotates by virtue of some rotation previously imparted to it or because of some internal action.

Some examples are a whirlpool in a river, the rotatory flow that often arises in a shallow vessel when liquid flows out through a hole in the bottom (as is often seen when water flows out from a bathtub or a wash basin), and flow in a centrifugal pump case just outside the impeller.

Cylindrical Free Vortex

A cylindrical free vortex motion is conceived in a cylindrical coordinate system with axis z directing vertically upwards (Fig. 14.1), where at each horizontal cross-section, there exists a planar free vortex motion with tangential velocity given by Eq. (14.10).

The total energy at any point remains constant and can be written as

$$\frac{p}{\rho} + \frac{C^2}{2r^2} + gz = H\,(Cons.)$$

The pressure distribution along the radius can be found by considering z as constant; again, for any constant pressure p, values of z, determining a surface of equal pressure, can also be found.

If p is measured in gauge pressure, then the value of z, where p # 0 determines the free surface if one exists.

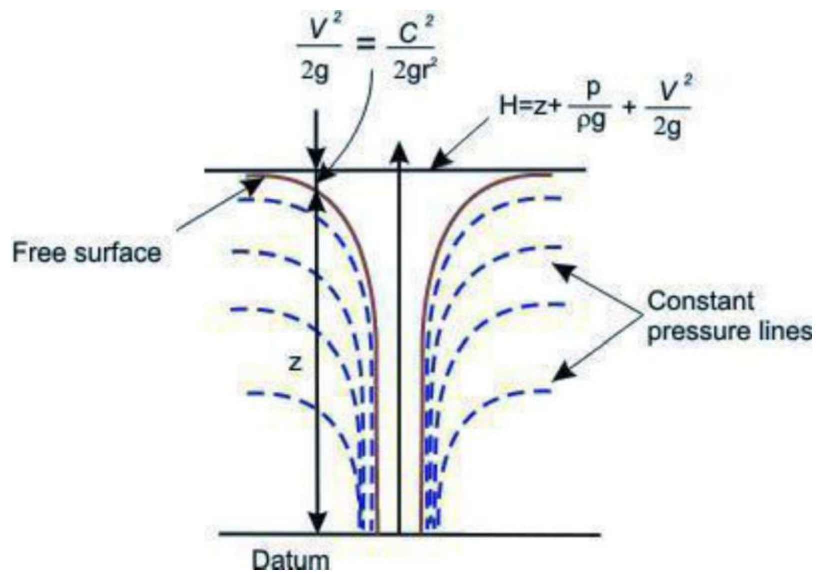


Fig: Cylindrical Free Vortex Forced Vortex Flows

Flows where streamlines are concentric circles and the tangential velocity is directly proportional to the radius of curvature are known as plane circular forced vortex flows.

The flow field is described in a polar coordinate system as,

$$V_\theta = \omega r$$

All fluid particles rotate with the same angular velocity ω like a solid body.
Hence a forced vortex flow is termed as a solid body rotation.

The vorticity Ω for the flow field can be calculated as

$$\Omega = \frac{\partial V_\theta}{\partial r} - \frac{1}{r}\frac{\partial V_r}{\partial \theta} + \frac{V_\theta}{r}$$

$$= \omega - \mathbf{0} + \omega = \mathbf{2}\omega$$

Therefore, a forced vortex motion is not irrotational; rather it is a rotational flow with a constant vorticity 2ω. Equation (14.8) is used to determine the distribution of mechanical energy across the radius as

$$\frac{dH}{dr} = \frac{V_\theta}{g}\left(\frac{dV_\theta}{dr} + \frac{V_\theta}{r}\right) = \frac{2\omega^2 r}{g}$$

Integrating the equation between the two radii on the same horizontal plane, we have,

$$H_2 - H_1 = \frac{\omega^2}{g}(r_2^2 - r_1^2)$$

Thus, we see from that the total head (total energy per unit weight) increases with an increase in radius. The total mechanical energy at any point is the sum of kinetic energy, flow work or pressure energy, and the potential energy.

Therefore the difference in total head between any two points in the same horizontal plane can be written as,

$$H_2 - H_1 = \left[\frac{p_2}{\rho g} - \frac{p_1}{\rho g}\right] + \left[\frac{V_2^2}{2g} - \frac{V_1^2}{2g}\right]$$

$$= \frac{p_2}{\rho g} - \frac{p_1}{\rho g} + \frac{\omega^2}{2g}(r_2^2 - r_1^2)$$

Substituting this expression of H_2-H_1, we get

$$\frac{p_2 - p_1}{\rho} = \frac{\omega^2}{2}(r_2^2 - r_1^2)$$

The same equation can also be obtained by integrating the equation of motion in a radial direction as

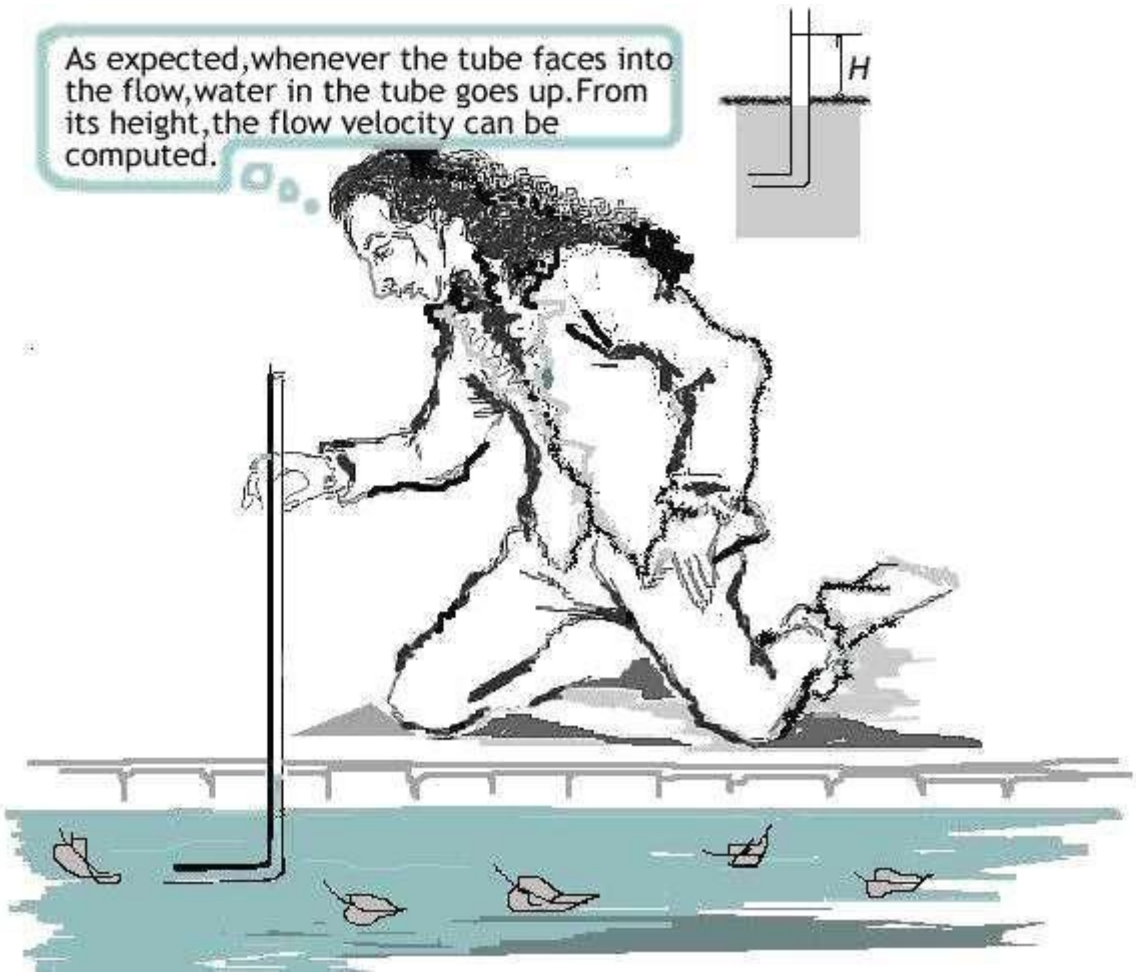
$$\int_1^2 \frac{1}{\rho}\frac{dp}{dr}dr = \int_1^2 \frac{V_\theta^2}{r}dr = \omega^2 \int_1^1 r\,dr$$

$$\frac{p_2 - p_1}{\rho} = \frac{\omega^2}{2}(r_2^2 - r_1^2)$$

Measurement of Flow Rate Through Pipe

Flow rate through a pipe is usually measured by providing a coaxial area contraction within the pipe and by recording the pressure drop across the contraction. Therefore the determination of the flow rate from the measurement of pressure drop depends on the straight forward application of Bernoulli's equation.

Three different flow meters operate on this principle. Venturimeter Orificemeter Flow nozzle.



VENTURIMETER:

Construction: A venturimeter is essentially a short pipe consisting of two conical parts with a short portion of uniform cross-section in between. This short portion has the minimum area and is known as the throat. The two conical portions have the same base diameter, but one is having a shorter length with a larger cone angle while the other is having a larger length with a smaller cone angle.

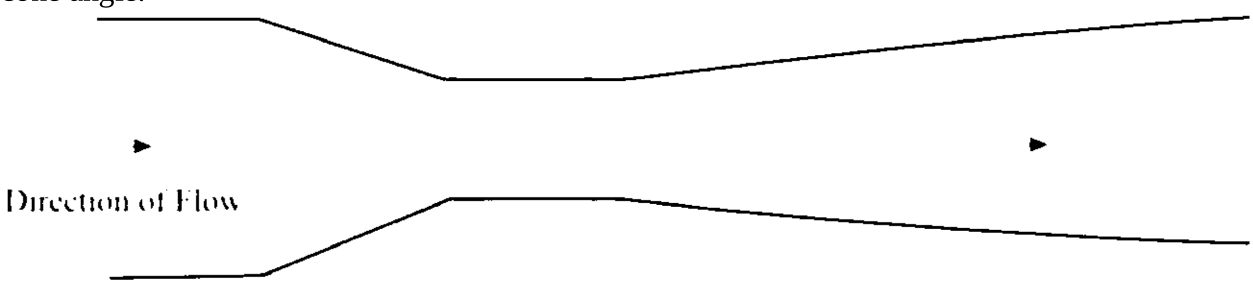


Fig: A Venturimeter

Working:

The venturimeter is always used in a way that the upstream part of the flow takes place through the short conical portion while the downstream part of the flow through the long one.

This ensures a rapid converging passage and a gradual diverging passage in the direction of flow to avoid the loss of energy due to separation. In course of a flow through the converging part, the velocity increases in the direction of flow according to the principle of

continuity, while the pressure decreases according to Bernoulli's theorem.

The velocity reaches its maximum value and pressure reaches its minimum value at the throat. Subsequently, a decrease in the velocity and an increase in the pressure takes place in course of flow through the divergent part. This typical variation of fluid velocity and pressure by allowing it to flow through such a constricted convergent-divergent passage was first demonstrated by an Italian scientist Giovanni Battista Venturi in 1797.

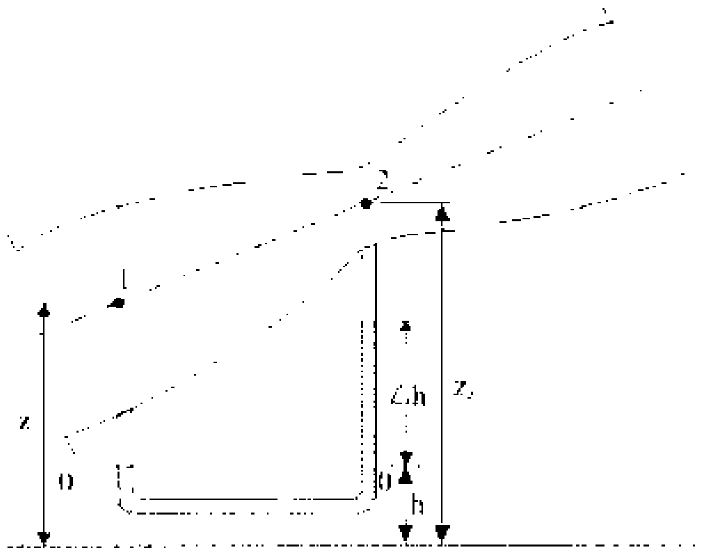


Fig: Measurement of Flow by a Venturimeter

Figure 15.2 shows that a venturimeter is inserted in an inclined pipe line in a vertical plane to measure the flow rate through the pipe. Let us consider a steady, ideal and one dimensional (along the axis of the venturi meter) flow of fluid. Under this situation, the velocity and pressure at any section will be uniform.

Let the velocity and pressure at the inlet (Sec. 1) are V_1 and p_1 respectively, while those at the throat (Sec. 2) are V_2 and p_2. Now, applying

Bernoulli's equation between Secs 1 and 2, we get

$$\frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2$$

$$\frac{V_2^2 - V_1^2}{2g} = \frac{p_1 - p_2}{2g} + z_1 - z_2$$

where ρ is the density of fluid flowing through the venturimeter.

From continuity,

$$V_1 A_1 = V_2 A_2$$

where A_1 and A_2 are the cross-sectional areas of the venturi meter at its throat and inlet respectively.

with the help of Eq. (15.3), Eq. (15.2) can be written as

$$\frac{V_2^2}{2g}\left(1-\frac{A_2^2}{A_1^2}\right)=\left(\frac{p_1}{2g}+z_1\right)-\left(\frac{p_2}{2g}+z_2\right)$$

$$V_2=\frac{1}{\sqrt{1-\frac{A_2^2}{A_1^2}}}\sqrt{2g(h_1^*-h_2^*)} \qquad (15.4)$$

where h_1^* and h_2^* are the piezometric pressure heads at sec. 1 and sec. 2 respectively, and are defined as

$$h_1^*=\frac{p_1}{\rho g}+z_1$$

$$h_2^*=\frac{p_2}{\rho g}+z_2$$

Hence, the volume flow rate through the pipe is given by

$$Q=A_2V_2=\frac{A_2}{\sqrt{1-\frac{A_2^2}{A_1^2}}}\sqrt{2g(h_1^*-h_2^*)}$$

If the pressure difference between Sections 1 and 2 is measured by a manometer as shown in Fig. we can write

$$p_1 + \rho g(z_1 - h_o) = p_2 + \rho g(z_2 - h_o - \Delta h) + \Delta h \rho_m g$$

$$\text{or,} \quad (p_1 + \rho g z_1) - (p_2 + \rho g z_2) = (\rho_m - \rho) g \Delta h$$

$$\left(\frac{p_1}{\rho g} + z_1\right) - \left(\frac{p_2}{\rho g} + z_2\right) = \left(\frac{\rho_m}{\rho} - 1\right)\Delta h$$

$$\text{or,} \quad h_1^* - h_2^* = \left(\frac{\rho_m}{\rho} - 1\right)\Delta h$$

where

ρ is the density of the manometric liquid.

Equation (15.7) shows that a manometer always registers a direct reading of the difference in piezometric pressures. Now, substitution

of $h_1^* - h_2^*$

$$Q = \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}}\sqrt{2g(\rho_m / \rho - 1)\Delta h}$$

If the pipe along with the venturimeter is horizontal, then z1 # z2; and hence $h_1^* - h_2^*$ becomes h1 – h2, where h1 and h2 are the static pressure heads $\left(h_1 = \frac{p_1}{\rho g}, h_2 = \frac{p_2}{\rho g}\right)$

The manometric equation then becomes

$$h_1 - h_2 = \left[\frac{\rho_m}{\rho} - 1\right]\Delta h$$

Measured values of Δh, the difference in piezometric pressures between

Secs I and 2, for a real fluid will always be greater than that assumed in case of an ideal fluid because of frictional losses in addition to the change in momentum.

Therefore, Eq. (15.8) always overestimates the actual flow rate. In order to take this into account, a multiplying factor C_d, called the coefficient of discharge, is incorporated in the as

$$Q_{actual} = C_d \frac{A_1 A_2}{\sqrt{A_1^2 - A_2^2}}\sqrt{2g(\rho_m / \rho - 1)\Delta h}$$

The coefficient of discharge C_d is always less than unity and is defined as

$$C_d = \frac{\text{Actual rate of discharge}}{\text{Theoretical rate of discharge}}$$

where, the theoretical discharge rate is predicted by the with the measured value of Δh, and the actual rate of discharge is the discharge rate measured in practice. Value of C_d for a venturimeter usually lies between 0.95 to 0.98.

ORIFICEMETER:

Construction: An orificemeter provides a simpler and cheaper arrangement for the measurement of fow through a pipe. An orificemeter is essentially a thin circular plate with a sharp edged concentric circular hole in it.

working:

The orifice plate, being fixed at a section of the pipe, creates an obstruction to the flow by providing an opening in the form of an orifice to the flow passage.

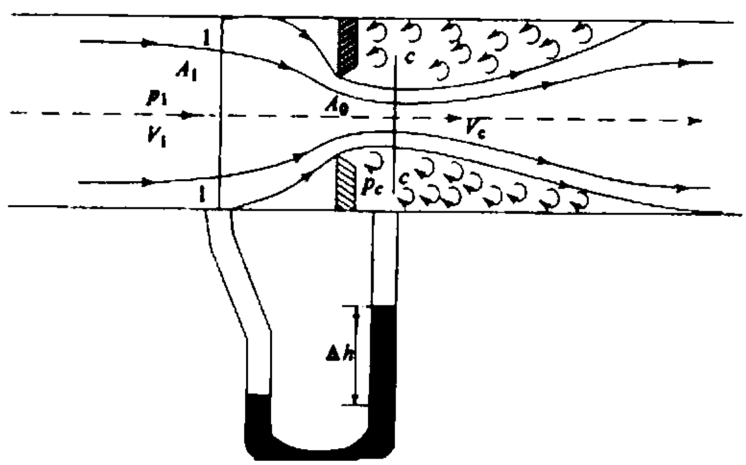


Fig: Flow through an orificemeter

The area A_0 of the orifice is much smaller than the cross-sectional area of the pipe. The flow from an upstream section, where it is uniform, adjusts itself in such a way that it contracts until a section downstream the orifice plate is reached, where the vena contracta is formed, and then expands to fill the passage of the pipe.

one of the pressure tapings is usually provided at a distance of one diameter upstream the orifice plate where the flow is almost uniform (Sec. 1-1) and the other at a distance of half a diameter downstream the orifice plate.

Considering the fluid to be ideal and the downstream pressure taping to be at the vena contracta (Sec. c-c), we can write, by applying

$$\frac{p_1^*}{\rho g}+\frac{V_1^2}{2g}=\frac{p_c^*}{\rho g}+\frac{V_c^2}{2g}$$

where and p_1^* are the p_c^* piezometric pressures at Sec.1-1 and c-c respectively. From the equation of continuity,

$$V_1 A_1 = V_c A_c$$

where A_C is the area of the vena contracta.

with the help of equations can be written as,

$$V_c=\sqrt{\frac{2(p_1^*-p_c^*)}{\rho\left(1-A_c^2/A_1^2\right)}}$$

Correction in Velocity

Recalling the fact that the measured value of the piezometric pressure drop for a real fluid is always more due to friction than that assumed in case of an inviscid flow, a coefficient of velocity

Cv (always less than 1) has to be introduced to determine the actual velocity V_C when the pressure drop $p_1^*-p_c^*$ in is substituted by its measured value in terms of the manometer deflection 'Δh'

Hence,

$$V_c = C_v \sqrt{\frac{2\rho g(\rho_m / \rho - 1)\Delta h}{1 - A_c^2 / A_1^2}}$$

where 'Δh' is the difference in liquid levels in the manometer and ρ_m is the density of the manometric liquid.
Volumetric flow rate

If a coefficient of contraction C_C is defined as, C_C # A_C / A_0 , where A_0 is the area of the orifice, then Eq.(15.14) can be written, with the help of Eq. (15.13),

$$Q = C_c A_0 C_v \sqrt{\frac{2g(\rho_m / \rho - 1)\Delta h}{1 - C_c^2 A_0^2 / A_1^2}}$$

$$= C_c A_0 C_v \sqrt{\frac{2g}{1 - C_c^2 A_0^2 / A_1^2}} \sqrt{(\rho_m / \rho - 1)\Delta h}$$

$$= C \sqrt{(\rho_m / \rho - 1)\Delta h}$$

with, $$C = C_d A_0 \sqrt{\frac{2g}{1 - C_c^2 A_0^2 / A_1^2}}, where (C_d = C_v C_c)$$

The value of C depends upon the ratio of orifice to duct area, and the Reynolds number of flow.

The main job in measuring the flow rate with the help of an orificemeter, is to find out accurately the value of C at the operating condition.

The downstream manometer connection should strictly be made to the section where the vena contracta occurs, but this is not feasible as the vena contracta is somewhat variable in position and is difficult to realize.

In practice, various positions are used for the manometer connections and C is thereby affected. Determination of accurate values of C of an orificemeter at different operating conditions is known as calibration of the orifice meter.

Concept and Types of Physical Similarity

The primary and fundamental requirement for the physical

similarity between two problems is that the physics of the problems must be the same.

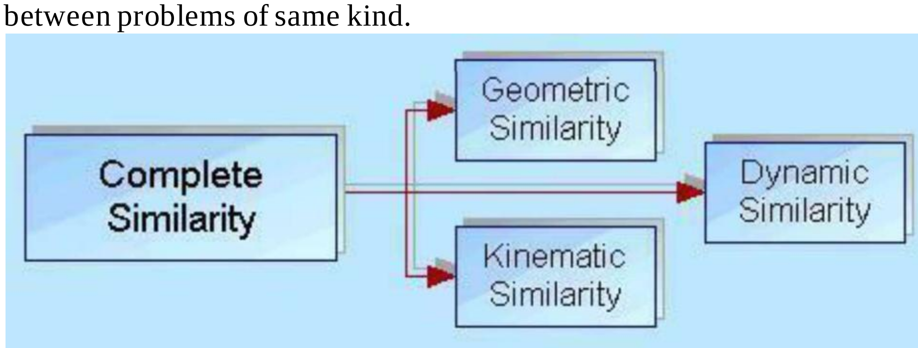
For an example, two flows: one governed by viscous and pressure forces while the other by gravity force cannot be made physically similar. Therefore, the laws of similarity have to be sought between problems described by the same physics.

Definition of physical similarity as a general proposition.

Two systems, described by the same physics, operating under different sets of conditions are said to be physically similar in respect of certain

specified physical quantities; when the ratio of corresponding magnitudes of these quantities between the two systems is the same everywhere.

In the field of mechanics, there are three types of similarities which constitute the complete similarity between problems of same kind.



Geometric Similarity : If the specified physical quantities are geometrical dimensions, the similarity is called Geometric Similarty,

Kinematic Similarity : If the quantities are related to motions, the similarity is called Kinematic Similarity

Dynamic Similarity : If the quantities refer to forces, then the similarity is termed as Dynamic Similarity.

Geometric Similarity

Geometric Similarity implies the similarity of shape such that, the ratio of any length in one system to the corresponding length in other system is the same everywhere.

This ratio is usually known as scale factor.

Therefore, geometrically similar objects are similar in their shapes, i.e., proportionate in their physical dimensions, but differ in size.

In investigations of physical similarity,

The full size or actual scale systems are known as prototypes the laboratory scale systems are referred to as models

use of the same fluid with both the prototype and the model is not necessary

Model need not be necessarily smaller than the prototype. The flow of fluid through an injection nozzle or a carburettor , for example, would be more easily studied by using a model much larger than the prototype.

The model and prototype may be of identical size, although the two may then differ in regard to other factors such as velocity, and properties of the fluid.

If /1 and /2 are the two characteristic physical dimensions of any object, then the requirement of geometrical similarity is

$$\frac{l_{1m}}{l_{1p}} = \frac{l_{2m}}{l_{2p}} = l_r$$

(model ratio)

(The second suffices m and p refer to model and prototype respectively) where lr is the scale factor or sometimes known as the model ratio. shows three pairs of geometrically similar objects, namely, a right circular cylinder, a parallelopiped, and a triangular prism.

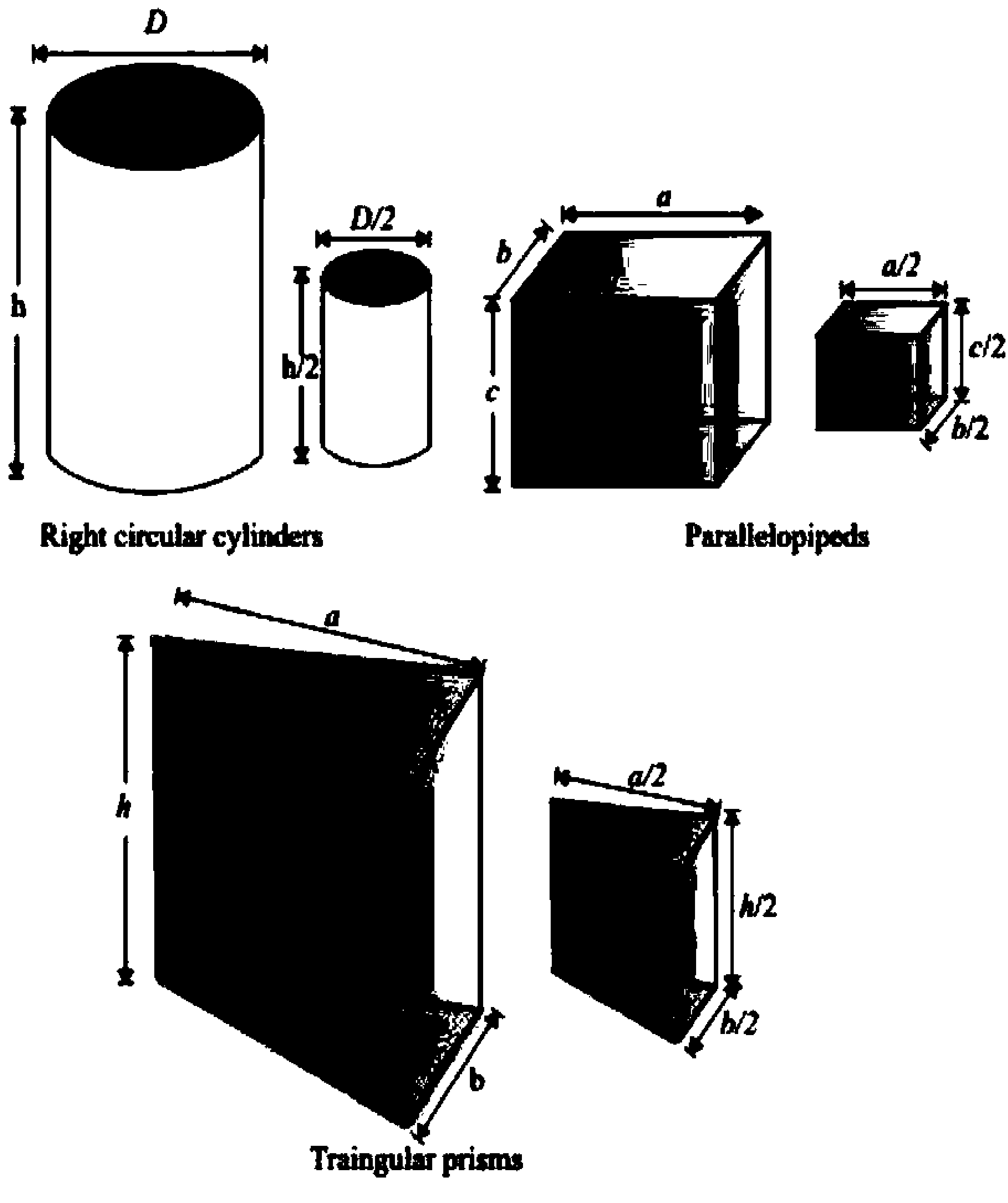


Fig: Geometrically Similar objects In all the above cases model ratio is 1/2

Geometric similarity is perhaps the most obvious requirement in a model system designed to correspond to a given prototype system.

A perfect geometric similarity is not always easy to attain. Problems in achieving perfect geometric similarity are:

For a small model, the surface roughness might not be reduced according to the scale factor (unless the model surfaces can be made very much smoother than those of the prototype). If for any reason the scale factor is not the same throughout, a distorted model results.

Sometimes it may so happen that to have a perfect geometric similarity within the available laboratory space, physics of the problem changes. For example, in case of large prototypes, such as rivers, the size of the model is limited by the available floor space of the laboratory; but if a very low scale factor is used in reducing both the horizontal and vertical lengths, this may result in a stream so shallow that surface tension has a considerable effect and, moreover, the flow may be laminar instead of turbulent. In this situation, a distorted model may be unavoidable (a lower scale factor "for horizontal lengths while a relatively higher scale factor for vertical lengths. The extent to which perfect geometric similarity should be sought therefore depends on the problem being investigated, and the accuracy required from the solution.

Kinematic Similarity

Kinematic similarity refers to similarity of motion.

Since motions are described by distance and time, it implies similarity of lengths (i.e., geometrical similarity) and, in addition, similarity of time intervals.

If the corresponding lengths in the two systems are in a fixed ratio, the velocities of corresponding particles must be in a fixed ratio of magnitude of corresponding time intervals.

If the ratio of corresponding lengths, known as the scale factor, is l_r and the ratio of corresponding time intervals is t_r, then the magnitudes of correspondingvelocities are in the ratio l_r/t_r and the magnitudes of corresponding accelerations are in the ratio l_r/t_r^2.

A well-known example of kinematic similarity is found in a planetarium. Here the galaxies of stars and planets in space are reproduced in accordance with a certain length scale and in simulating the motions of the planets, a fixed ratio of time intervals (and hence velocities and accelerations) is used.

then fluid motions are kinematically similar, the patterns formed by streamlines are geometrically similar at corresponding times.

Since the impermeable boundaries also represent streamlines, kinematically similar flows are possible only past geometrically similar boundaries.

Therefore, geometric similarity is a necessary condition for the kinematic similarity to be achieved, but not the sufficient one.

For example, geometrically similar boundaries may ensure geometrically similar streamlines in the near vicinity of the boundary but not at a distance from the boundary.

Dynamic Similarity

Dynamic similarity is the similarity of forces .

In dynamically similar systems, the magnitudes of forces at correspondingly similar points in each system are in a fixed ratio.

In a system involving flow of fluid, different forces due to different causes may act on a fluid element. These forces are as follows:

Viscous Force (due to viscosity)

Pressure Force (due to different in pressure) Gravity Force (due to gravitational attraction)

Capillary Force (due to surface tension)

Compressibility Force (due to elasticity)

According to Newton 's law, the resultant F_R of all these forces, will cause the acceleration of a fluid element. Hence

Moreover, the inertia force $\vec{F}_i$ is defined as equal and opposite to the resultant accelerating force $\vec{F}_R$

$$\neq \vec{F}_i \qquad \vec{F}_R$$

Therefore can be expressed as

$$\vec{F}_v + \vec{F}_p + \vec{F}_g + \vec{F}_c + \vec{F}_e + \vec{F}_i = 0$$

For dynamic similarity, the magnitude ratios of these forces have to be same for both the prototype and the model. The inertia force $\vec{F}_i$ is usually taken as the common one to describe the ratios as (or putting in other form we equate the the non dimensionalised forces in the two systems)

$$\frac{|\vec{F}_v|}{|\vec{F}_i|}, \frac{|\vec{F}_p|}{|\vec{F}_i|}, \frac{|\vec{F}_g|}{|\vec{F}_i|}, \frac{|\vec{F}_c|}{|\vec{F}_i|}, \frac{|\vec{F}_e|}{|\vec{F}_i|}$$

Magnitudes of Different Forces

A fluid motion, under all such forces is characterised by

Hydrodynamic parameters like pressure, velocity and acceleration due to gravity, Rheological and other physical properties of the fluid involved, and

Geometrical dimensions of the system.

It is important to express the magnitudes of different forces in terms of these parameters, to know the extent of their influences on the different forces acting on a flluid element in the course of its flow.

Inertia Force $\vec{F}_i$

The inertia force acting on a fluid element is equal in magnitude to the mass of the element multiplied by its acceleration.

The mass of a fluid element is proportional to ρl) where, ρ is the density of fluid and l is the characteristic geometrical dimension of the system.

The acceleration of a fluid element in any direction is the rate at which its velocity in that direction changes with time and is therefore proportional in magnitude to some characteristic velocity V divided by some specified interval of time t. The time interval t is proportional to the characteristic length l divided by the characteristic velocity V, so that the acceleration becomes proportional to

The magnitude of inertia force is thus proportional to

$$\rho l^3 V^2 / l = \rho l^2 V^2$$

This can be written as,

$$|\vec{F}_i| \propto \rho l^2 V^2$$

Viscous Force $\vec{F}_v$

The viscous force arises from shear stress in a flow of fluid. Therefore, we can write

Magnitude of viscous force $\vec{F}_v$ # shear stress X surface area over which the shear stress acts Again, shear stress # μ (viscosity) X rate of shear strain

where, rate of shear strain [redacted] velocity gradient $\propto \frac{V}{l}$ and surface $\propto l^2$ Hence

$$\left|\vec{F}_v\right| \propto \mu \frac{V}{l} l^2$$

$$\propto \mu V l$$

Pressure Force $\vec{F}_p$

The pressure force arises due to the difference of pressure in a flow field. Hence it can be written as

$$\left|\vec{F}_p\right| \propto \Delta p . l^2$$

where, p is some characteristic pressure difference in the flow.) Gravity Force $\vec{F}_g$
The gravity force on a fluid element is its weight. Hence,

$$\left|\vec{F}_g\right| \propto \rho l^3 g$$

where g is the acceleration due to gravity or weight per unit mass) Capillary or Surface Tension Force $\vec{F}_c$

The capillary force arises due to the existence of an interface between two fluids. The surface tension force acts tangential to a surface .

It is equal to the coefficient of surface tension ς multiplied by the length of a linear element on the surface perpendicular to which the force acts. Therefore,

$$\left|\vec{F}_c\right| \propto \sigma l$$

Compressibility or Elastic Force $\vec{F}_e$

Elastic force arises due to the compressibility of the fluid in course of its flow.

For a given compression (a decrease in volume), the increase in pressure is proportional to the bulk modulus of elasticity E

This gives rise to a force known as the elastic force.

Hence, for a given compression $\Delta \mathbf{p} \,\alpha\, \mathbf{E}$

(18.1f) $\left|\vec{F}_e\right| \propto El^2$

The flow of a fluid in practice does not involve all the forces simultaneously.

Therefore, the pertinent dimensionless parameters for dynamic similarity are derived from the ratios of significant forces causing the flow.

Dynamic Similarity of Flows governed by Viscous, Pressure and Inertia Forces

The criterion of dynamic similarity for the flows controlled by viscous, pressure and inertia forces are derived from the ratios of the representative magnitudes of these forces with the help of Eq. (18.1a) to (18.1c) as follows:

$$\frac{\text{Viscous force}}{\text{Inertia Force}} = \frac{\left|\vec{F}_v\right|}{\left|\vec{F}_i\right|} \propto \frac{\mu V l}{\rho V^2 l^2} = \frac{\mu}{\rho V l}$$

$$\frac{\text{Pressure force}}{\text{Inertia Force}} = \frac{\left|\vec{F}_p\right|}{\left|\vec{F}_i\right|} \propto \frac{\Delta p l^2}{\rho V^2 l^2} = \frac{\Delta p}{\rho V^2}$$

The term $\rho l V/\mu$ is known as Reynolds number, Re after the name of the scientist who first developed it and is thus proportional to the magnitude ratio of inertia force to viscous force .(Reynolds number plays a vital role in the analysis of fluid flow)

The term $\Delta p/\rho V^2$ is known as Euler number, Eu after the name of the scientist who first derived it. The dimensionless terms Re and Eu represent the critieria of dynamic similarity for the flows which are affected only by viscous, pressure and inertia forces.
Such instances, for example, are

the full flow of fluid in a completely closed conduit, flow of air past a low-speed aircraft and the

flow of water past a submarine deeply submerged to produce no waves on the surface.

Hence, for a complete dynamic similarity to exist between the prototype and the model for this class of flows, the Reynolds number, Re and Euler number, Eu have to be same for the two (prototype and model). Thus

$$\frac{\rho_p\, l_p V_p}{\mu_p} = \frac{\rho_m\, l_m V_m}{\mu_m}$$
(18.2c)

(18.2d) $$\frac{\Delta P_p}{\rho_p V_p^2} = \frac{\Delta P_m}{\rho_m V_m^2}$$

where, the suffix p and suffix m refer to the parameters for prototype and model respectively.

In practice, the pressure drop is the dependent variable, and hence it is compared for the two systems with the help of Eq. (18.2d), while the equality of Reynolds number (Eq. (18.2c)) along with the equalities of other parameters in relation to kinematic and geometric similarities are maintained.

The characteristic geometrical dimension l and the reference velocity V in the expression of the Reynolds number may be any geometrical dimension and any velocity which are significant in determining the pattern of flow.

For internal flows through a closed duct, the hydraulic diameter of the duct D_h and the average flow velocity at a section are invariably used for l and V respectively.

The hydraulic diameter D_h is defined as D_h# 4A/P where A and P are the cross- sectional area and wetted perimeter respectively.

Dynamic Similarity of Flows with Gravity, Pressure and Inertia Forces

A flow of the type in which significant forces are gravity force, pressure force and inertia force, is found when a free surface is present.

Examples can be

the flow of a liquid in an open channel.

the wave motion caused by the passage of a ship through water. the flows over weirs and

spillways.

The condition for dynamic similarity of such flows requires

the equality of the Euler number Eu (the magnitude ratio of pressure to inertia force),

and the equality of the magnitude ratio of gravity to inertia force at corresponding points in the systems being compared.

Thus ,

$$\frac{\text{Gravity force}}{\text{Inertia Force}} = \frac{|\vec{F}_g|}{|\vec{F}_i|} \propto \frac{\rho l^3 g}{\rho V^2 l^2} = \frac{lg}{V^2}$$

In practice, it is often convenient to use the square root of this ratio so to deal with the first power of the velocity.

From a physical point of view, equality of $(lg)^{1/2}/V$ implies equality of lg/V^2 as regard to the concept of dynamic similarity.

The reciprocal of the term $(lg)^{1/2}/V$ is known as Froude number (after william Froude who first suggested the use of this number in the study of naval architecture.)

Hence Froude number, $Fr = V/(lg)^{1/2}$.

Therefore, the primary requirement for dynamic similarity between the prototype and the model involving flow of fluid with gravity as the significant force, is the equality of Froude number, Fr, i.e.,

$$\frac{(l_p g_p)^{1/2}}{V_p} = \frac{(l_m g_m)^{1/2}}{V_m}$$

Dynamic Similarity of Flows with Surface Tension as the Dominant Force

Surface tension forces are important in certain classes of practical problems such as, flows in which capillary waves appear flows of small jets and thin sheets of liquid injected by a nozzle in air flow of a thin sheet of liquid over a solid surface.

Here the significant parameter for dynamic similarity is the magnitude ratio of the surface tension force to the inertia force.

This can be written as $$\frac{|\vec{F}_c|}{|\vec{F}_i|} \propto \frac{\sigma l}{\rho V^2 l^2} = \frac{\sigma}{\rho V^2 l}$$

The term $\sigma/\rho V^2 l$ is usually known as weber number, wb (after the German naval architect Moritz weber who first suggested the use of this term as a relevant parameter.)

Thus for dynamically similar flows $(wb)_m\#(wb)_p$ i.e., $$\frac{\sigma_m}{\rho_m V_m^2 L_m} = \frac{\sigma_p}{\rho_p V_p^2 L_p^2}$$

Dynamic Similarity of Flows with Elastic Force

then the compressibility of fluid in the course of its flow becomes significant, the elastic force along with the pressure and inertia forces has to be considered.

Therefore, the magnitude ratio of inertia to elastic force becomes a relevant parameter for dynamic similarity under this situation.

Thus we can write,

$$\frac{\text{Inertia force}}{\text{Elastic Force}} = \frac{|\vec{F}_i|}{|\vec{F}_e|} \propto \frac{\rho V^2 l^2}{E l^2} = \frac{\rho V^2}{E}$$

The parameter $\rho V^2/E$ is known as Cauchy number ,(after the French mathematician A.L. Cauchy)

If we consider the flow to be isentropic , then it can be written

(18.2i) $$\frac{|\vec{F}_i|}{|\vec{F}_e|} \propto \frac{\rho V^2}{E_s}$$

(where E_S is the isentropic bulk modulus of elasticity)

Thus for dynamically similar flows $(cauchy)_m \# (cauchy)_p$

ie., $$\frac{\rho_m V_m^2}{(E_s)_m} = \frac{\rho_p V_p^2}{(E_s)_p}$$

The velocity with which a sound wave propagates through a fluid medium equals to $\sqrt{E_s / \rho}$.

Hence, the term $\rho V^2 / E_s$ can be written as V^2 / a^2 where a is the acoustic velocity in the fluid medium.

The ratio V/a is known as Mach number, Ma (after an Austrian physicist Earnst Mach)

It has been shown in Chapter 1 that the effects of compressibility become important when the Mach number exceeds 0.33.

The situation arises in the flow of air past high-speed aircraft, missiles, propellers and rotory compressors. In these cases equality of Mach number is a condition for dynamic similarity. Therefore,

(Ma)p/(Ma)m

i.e.

$$\boxed{V_p / a_p = V_m / a_m}$$

Buckingham's Pi Theorem

Assume, a physical phenomenon is described by m number of independent variables like x_1 , x_2 , x_3 , ..., x_m

The phenomenon may be expressed analytically by an implicit functional relationship of the controlling variables as

$$f(x_1, x_2, x_3, \ldots\ldots, x_m) = 0$$

Now if n be the number of fundamental dimensions like mass, length, time, temperature etc ., involved in these m variables, then according to Buckingham's p theorem -

The phenomenon can be described in terms of (m - n) independent

dimensionless groups like π_1 ,π_2 , ..., π_{m-n} , where p terms, represent the

dimensionless parameters and consist of different combinations of a number of dimensional variables out of the m independent variables defining the problem.

Therefore. the analytical version of the phenomenon given by Eq. (19.2) can be reduced to

$$\boxed{F(\pi_1, \pi_2, \ldots\ldots\ldots, \pi_{m-n}) = 0}$$

according to Buckingham's pi theorem

This physically implies that the phenomenon which is basically described by m independent dimensional variables, is ultimately controlled by (m-n) independent dimensionless parameters known as π terms.

Alternative Mathematical Description of (π) Pi Theorem

A physical problem described by m number of variables involving n number of fundamental dimensions (n # m) leads to a system of n linear algebraic equations with m variables of the form

$$a_{11}x_1 + a_{12}x_2 + \ldots\ldots a_{1m}x_m = b_1$$

$$a_{21}x_1 + a_{22}x_2 + \ldots\ldots a_{2m}x_m = b_2$$

$$a_{n1}x_1 + a_{n2}x_2 + \ldots\ldots a_{nm}x_m = b_n$$

$$\boxed{Ax = b}$$

where, $$A = \begin{bmatrix} a_{11} & a_{12} & \cdots & a_{1m} \\ a_{21} & a_{22} & \cdots & a_{2m} \\ a_{n1} & a_{n2} & \cdots & a_{nm} \end{bmatrix}$$

$$X = \begin{bmatrix} x_1 \\ x_2 \\ .. \\ x_m \end{bmatrix} \quad \text{and} \quad b = \begin{bmatrix} b_1 \\ b_2 \\ .. \\ b_m \end{bmatrix}$$

Determination of π terms

A group of n (n # number of fundamental dimensions) variables out of m (m # total number of independent variables defining the problem) variables is first chosen to form a basis so that all n dimensions are represented . These n variables are referred to as repeating variables.

Then the p terms are formed by the product of these repeating variables raised to arbitrary unknown integer exponents and anyone of the excluded (m -n) variables.

For e:xamp/e , if :x_1 :x_2 ...:x_n are taken as the repeating variables. Then

$$\pi_2 = x_1^{a_1} x_2^{a_2} \ldots\ldots x_n^{a_n} x_{n+2}$$

$$\pi_1 = x_1^{a_1} x_2^{a_2} \ldots\ldots x_n^{a_n} x_{n+1}$$

....

$$\pi_{m-n} = x_1^{a_1} x_2^{a_2} \ldots\ldots x_n^{a_n} x_m$$

The sets of integer exponents $a_1, a_2 \ldots a_n$ are different for each p term.

Since p terms are dimensionless, it requires that when all the variables in any p term are expressed in terms of their fundamental dimensions, the exponent of all the fundamental dimensions must be zero.

This leads to a system of n linear equations in a, $a_2 \ldots a_n$ which gives a unique solution for the exponents. This gives the values of $a_1\ a_2 \ldots a_n$ for each p term and hence the p terms are uniquely defined.

In selecting the repeating variables, the following points have to be considered:

The repeating variables must include among them all the n fundamental dimensions, not necessarily in each one but collectively.

The dependent variable or the output parameter of the physical phenomenon should not be included in the repeating variables.

No physical phenomena is represented when - m # n because there is no solution and

m = n because there is a unique solution of the variables involved and hence all the parameters have fixed values.

. Therefore all feasible phenomena are defined with m # n .

then m = n + 1, then, according to the Pi theorem, the number of pi term is one and the phenomenon can be expressed as

$$f(\pi_1) = 0$$

where, the non-dimensional term π_1 is some specific combination of n + 1 variables involved in the problem.

then m = n+ 1 , the number of

Again, it is true that if one of the repeating variables is changed, it results in a different set of π terms. Therefore the interesting question is which set of repeating variables is to be chosen , to arrive at the correct set of π terms to describe the problem. The answer to this question lies in the fact that different sets of π terms resulting from the use of different sets of repeating variables are not independent. Thus, anyone of such interdependent sets is meaningful in describing the same physical phenomenon.

sets from some combination

From any set of such π terms, one can obtain the other meaningful
of the π terms of the existing set without
theorem.

Navier-Strokes Equation

Generalized equations of motion of a real flow named after the inventors CLMH Navier and GG Stokes are derived from the Newton's second law

Newton's second law states that the product of mass and acceleration is equal to sum of the external forces acting on a body.

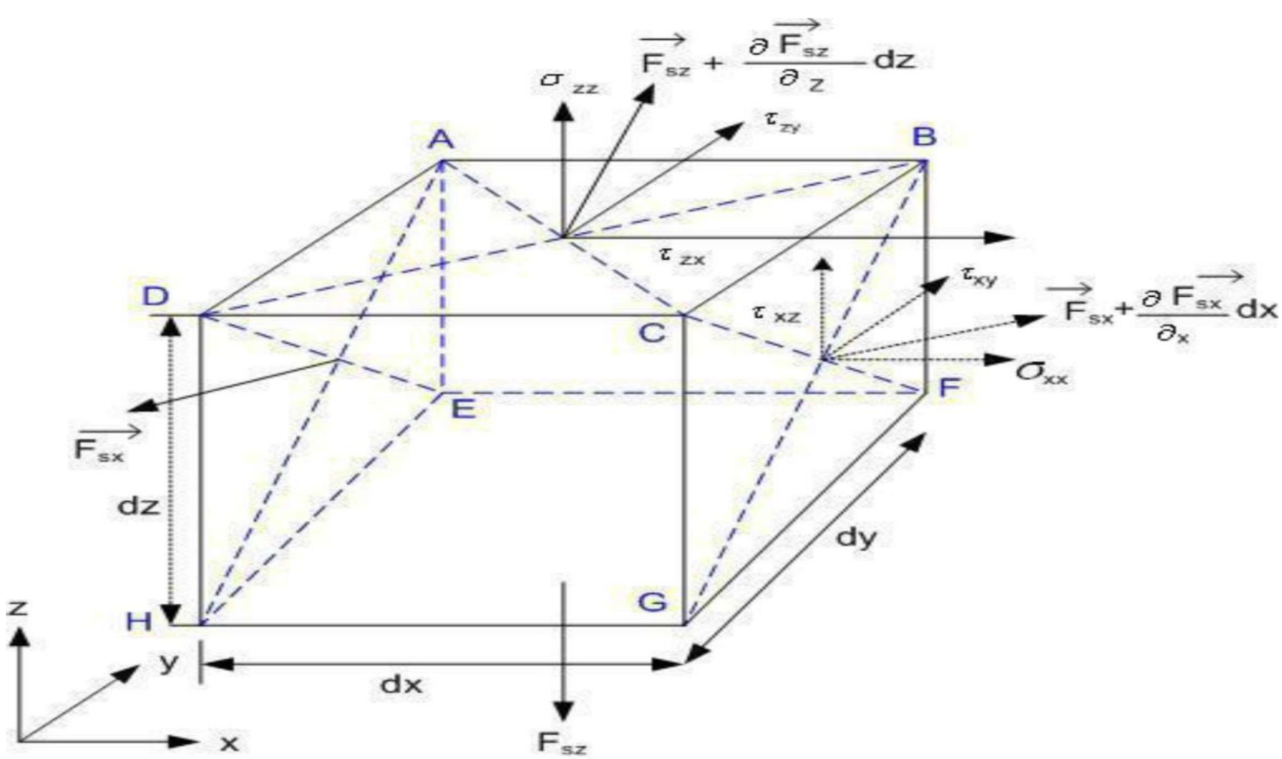
External forces are of two kinds-

one acts throughout the mass of the body ----- body force(gravitational force, electromagnetic force)

another acts on the boundary--------------------- surface force (pressure and frictional force).

objective - we shall consider a differential fluid element in the flow field. Evaluate the surface forces acting on the boundary of the rectangular parallelepiped shown below.

Definition of the components of stress and their /ocations in a differentia/ f/uid e/ement Let the



body force per unit mass be

$$\vec{f}_b = \hat{i}\ f_x + \hat{j}\ f_y + \hat{k}\ f_z$$

and surface force per unit volume be

$$\vec{F} = \hat{i}\ F_x + \hat{j}\ F_y + \hat{k}\ F_z$$

Consider surface force on the surface AEHD, per unit area,

$$\vec{F}_{sx} = \hat{i}\ \sigma_{xx} + \hat{j}\ \tau_{xy} + \hat{k}\ \tau_{xz}$$

[Here second subscript x denotes that the surface force is evaluated for the surface whose outward normal is the x axis]

Surface force on the surface BFGC per unit area is

$$\vec{F}_{sx} + \frac{\partial \vec{F}_{sx}}{\partial x} dx$$

Net force on the body due to imbalance of surface forces on the above two surfaces

Total force on the body due to net surface forces on all six surfaces is

$$\left(\frac{\partial \vec{F}_{sx}}{\partial x} + \frac{\partial \vec{F}_{sy}}{\partial y} + \frac{\partial \vec{F}_{sz}}{\partial z} \right) dx\ dy\ dz$$

And hence, the resultant surface force dF, per unit volume, is

$$d\vec{F} = \frac{\partial \vec{F}_{sx}}{\partial x} + \frac{\partial \vec{F}_{sy}}{\partial y} + \frac{\partial \vec{F}_{sz}}{\partial z}$$ (since Volume=dx dy dz)

$\vec{F}_{sx}$ $\vec{F}_{sy}$ $\vec{F}_{sz}$

$$\vec{F}_{sx} = \hat{i}\ \sigma_{xx} + \hat{j}\ \tau_{xy} + \hat{k}\ \tau_{xz}$$

$$\vec{F}_{sy} = \hat{i}\,\tau_{yx} + \hat{j}\,\sigma_{yy} + \hat{k}\,\tau_{yz}$$

$$\vec{F}_{sz} = \hat{i}\,\tau_{zx} + \hat{j}\,\tau_{zy} + \hat{k}\,\sigma_{zz}$$

The stress system has nine scalar quantities. These nine quantities form a stress tensor.

A general way of deriving the Navier-Stokes equations from the basic laws of physics. Consider a general flow field as represented in Fig. 25.1.

Imagine a closed control volume, $\forall_0$ within the flow field. The control volume is fixed in space and the fluid is moving through it. The control volume occupies reasonably large finite region of the flow field.

A control surface , AO is defined as the surface which bounds the volume $\forall_0$. According to

Reynolds transport theorem, "The rate of change of momentum for a system equals the sum of the rate of change of momentum inside thecontrol volume and the rate of efflux of momentum across the control surface".

The rate of change of momentum for a system (in our case, the control volume boundary and the system boundary are same) is equal to the net external force acting on it.

Now, we shall transform these statements into equation by accounting for each term,

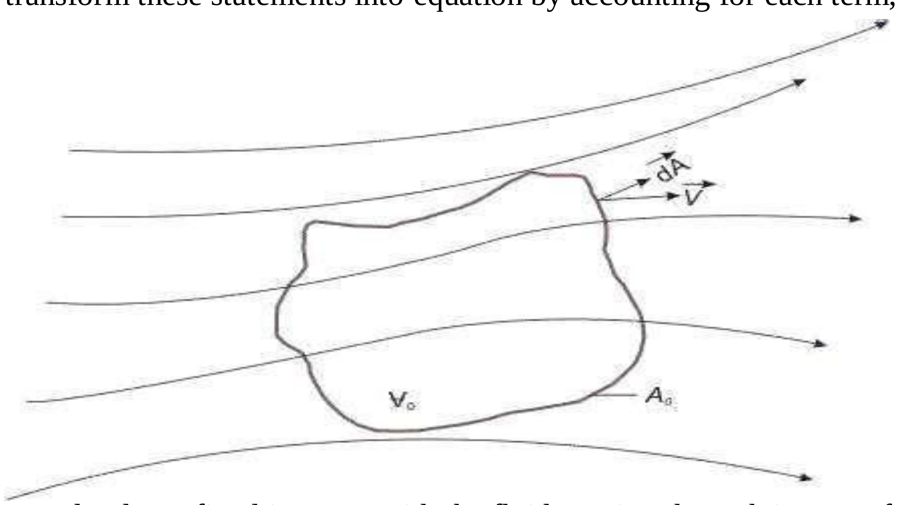


Fig: Finite control volume fixed in space with the fluid moving through it Rate of change of

momentum inside the control volume

$$= \frac{\partial}{\partial t} \int \iint_{\forall_0} \rho \vec{V} d\forall$$

Rate of efflux of momentum through control surface

$$\iint_{A_0} \rho \vec{V} (\vec{V} . d\vec{A}) = \iint_{A_0} \rho \vec{V}\vec{V} . \vec{n} dA$$
$$= \iiint_{\forall_0} \{\vec{V}(\nabla . \rho \vec{V}) + \rho \vec{V} . \nabla \vec{V}\} d\forall$$

Surface force acting on the control volume

$$= \iiint_{\forall_0} (\nabla . \sigma) d\forall$$

3ody force acting on the control volume

$$\iiint_{\forall_0} \rho \vec{f}_b d\forall$$

$\vec{f}_b$ in Eq. (25.4) is the body force per unit mass. Finally, we get,

or

$$\iiint_{\forall_0} \left(\frac{\partial}{\partial t} (\rho \vec{V}) + \{\vec{V}(\nabla . \rho \vec{V}) + \rho \vec{V} . \nabla \vec{V}\} \right) d\forall$$
$$= \iiint_{\forall_0} (\nabla . \sigma + \rho \vec{f}_b) d\forall$$

or, $$\rho \frac{\partial \vec{V}}{\partial t} + \vec{V} \frac{\partial \rho}{\partial t} + \rho \vec{V} . \nabla \vec{V} + \vec{V}(\nabla . \rho \vec{V}) = \nabla . \sigma + \rho \vec{f}_b$$

or $$\rho \left(\frac{\partial \vec{V}}{\partial t} + \vec{V} . \nabla \vec{V} \right) + \vec{V} \left(\frac{\partial \rho}{\partial t} + \nabla . \rho \vec{V} \right) = \nabla . \sigma + \rho \vec{f}_b$$

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \vec{V} = 0$$

we know that is the general form of mass conservation equation (popularly known as the continuity equation), valid for both compressible and incompressible flows.

Invoking this relationship in we obtain

Equation is referred to as Cauchy's equation of motion . In this equation, is the stress tensor,

Invoking above two relationships into we get

$$\rho \frac{D\vec{V}}{Dt} = -\nabla p + \mu \nabla^2 \vec{V} + \frac{1}{3}\mu \nabla\left(\nabla . \vec{V}\right) + \rho \vec{f}_b$$

This is the most general form of Navier-Stokes equation.

Exact Solutions of Navier-Stokes Equations

Consider a class of flow termed as parallel flow in which only one velocity term is nontrivial and all the fluid particles move in one direction only.

we choose x to be the direction along which all fluid particles travel , i.e. $u \neq 0,\ v = w = 0$. Invoking this in continuity equation, we get

$$\frac{\partial u}{\partial x} + \cancelto{0}{\frac{\partial v}{\partial y}} + \cancelto{0}{\frac{\partial w}{\partial z}} = 0$$

which means $u = u(y, z, t)$

Now. Navier-Stokes equations for incompressible flow become

$$\frac{\partial u}{\partial t} + u\cancelto{0}{\frac{\partial u}{\partial x}} + v\cancelto{0}{\frac{\partial u}{\partial y}} + w\cancelto{0}{\frac{\partial u}{\partial z}} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left[\cancelto{0}{\frac{\partial^2 u}{\partial x^2}} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

$$\cancelto{0}{\frac{\partial v}{\partial t}} + u\cancelto{0}{\frac{\partial v}{\partial x}} + v\cancelto{0}{\frac{\partial v}{\partial y}} + w\cancelto{0}{\frac{\partial v}{\partial z}} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left[\cancelto{0}{\frac{\partial^2 v}{\partial x^2}} + \cancelto{0}{\frac{\partial^2 v}{\partial y^2}} + \cancelto{0}{\frac{\partial^2 v}{\partial z^2}}\right]$$

$$\cancelto{0}{\frac{\partial w}{\partial t}} + u\cancelto{0}{\frac{\partial w}{\partial x}} + v\cancelto{0}{\frac{\partial w}{\partial y}} + w\cancelto{0}{\frac{\partial w}{\partial z}} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left[\cancelto{0}{\frac{\partial^2 w}{\partial x^2}} + \cancelto{0}{\frac{\partial^2 w}{\partial y^2}} + \cancelto{0}{\frac{\partial^2 w}{\partial z^2}}\right]$$

So, we obtain

$\frac{\partial p}{\partial t} = \frac{\partial p}{\partial z} = 0$ which means $p = p(x)\ alone$

and $$\frac{\partial u}{\partial t} = -\frac{1}{\rho}.\frac{\partial p}{\partial x} + \nu\left[\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

UNIT-4
BOUNDARY LAYER THEORY AND PIPE FLOW

Introduction

The boundary layer of a flowing fluid is the thin layer close to the wall In a flow field, viscous stresses are very prominent within this layer.

Although the layer is thin, it is very important to know the details of flow within it.

The main-flow velocity within this layer tends to zero while approaching the wall (no-slip condition).

Also the gradient of this velocity component in a direction normal to the surface is large as compared to the gradient in the streamwise direction.

Boundary Layer Equations

In 1904, Ludwig Prandtl, the well known German scientist, introduced the concept of boundary layer and derived the equations for boundary layer flow by correct reduction of Navier-Stokes equations.

He hypothesized that for fluids having relatively small viscosity, the effect of internal friction in the fluid is significant only in a narrow region surrounding solid boundaries or bodies over which the fluid flows.

Thus, close to the body is the boundary layer where shear stresses exert an increasingly larger effect on the fluid as one moves from free stream towards the solid boundary.

However, outside the boundary layer where the effect of the shear stresses on the flow is small compared to values inside the boundary layer (since the velocity gradient $\partial u / \partial y$ is negligible), the fluid particles experience no vorticity and therefore, the flow is similar to a potential flow.

Hence, the surface at the boundary layer interface is a rather fictitious one, that divides rotational and irrotational flow. Fig 28.1 shows Prandtl's model regarding boundary layer flow.

Hence with the exception of the immediate vicinity of the surface, the flow is frictionless (inviscid) and the velocity is 1 (the potential velocity).

In the region, very near to the surface (in the thin layer), there is friction in the flow which signifies that the fluid is retarded until it adheres to the surface (no-slip condition).

The transition of the mainstream velocity from zero at the surface (with respect to the surface) to full magnitude takes place across the boundary layer.

About the boundary layer

Boundary layer thickness is δ which is a function of the coordinate direction :x .

The thickness is considered to be very small compared to the characteristic lengthl of the domain.

In the normal direction, within this thin layer, the gradient $\partial u / \partial y$ is very large compared to the gradient in the flow direction $\partial u / \partial x$.

Considering the Navier-Stokes equations together with the equation of continuity, the following dimensional form is obtained.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

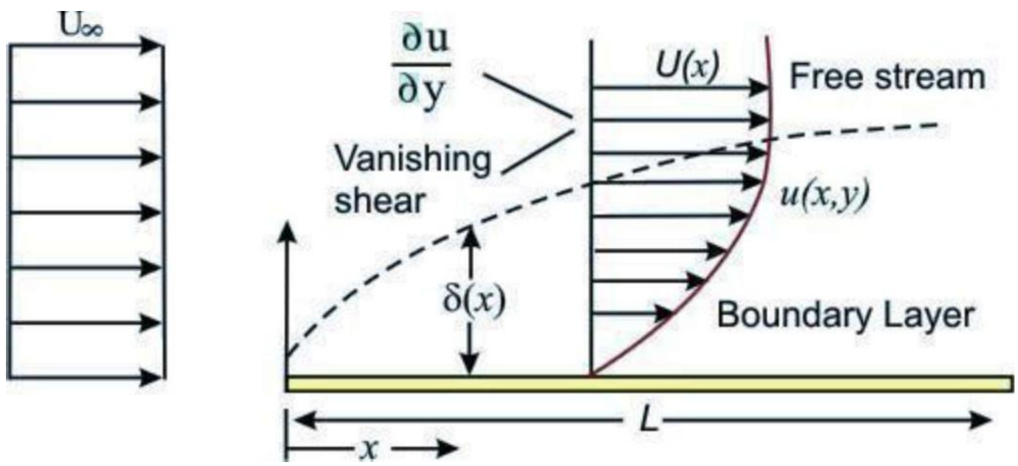


Fig: Boundary layer and Free Stream for Flow over a flat plate

u - velocity component along x direction. v - velocity component along y direction p - static pressure

- ρ density.

- μ dynamic viscosity of the fluid

The equations are now non-dimensionalised.

The length and the velocity scales are chosen as l and U_∞ respectively. The non- dimensional variables are:

$$u^* = \frac{u}{U_\infty}, v^* = \frac{v}{U_\infty}, p^* = \frac{p}{\rho U_\infty^2}$$

$$x^* = \frac{x}{L}, y^* = \frac{y}{L}$$

where U_∞ is the dimensional free stream velocity and the pressure is non-dimensionalised by twice the dynamic pressure $p_d = (1/2)\rho U_\infty^2$.

Using these non-dimensional variables,

$$u^*\frac{\partial u^*}{\partial x^*} + v^*\frac{\partial u^*}{\partial y^*} = -\frac{\partial p^*}{\partial x^*} + \frac{1}{\mathrm{Re}}\left[\frac{\partial^2 u^*}{\partial x^{*2}} + \frac{\partial^2 u^*}{\partial y^{*2}}\right]$$

$$u^*\frac{\partial v^*}{\partial x^*} + v^*\frac{\partial v^*}{\partial y^*} = -\frac{\partial p^*}{\partial x^*} + \frac{1}{\mathrm{Re}}\left[\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{\partial^2 v^*}{\partial y^{*2}}\right]$$

$$\frac{\partial u^*}{\partial x^*} + \frac{\partial v^*}{\partial y^*} = 0$$

where the Reynolds number,

$$\mathrm{Re} = \frac{\rho U_\infty L}{\mu}$$

beyond the boundary layer is 1.
For the case of external flow over a flat plate, this 1 is equal to .

Based on the above, we can identify the following scales for the boundary layer variables:

The symbol describes a value much smaller than 1.

Now we analyse, and look at the order of magnitude of each individual term

The continuity equation

One general rule of incompressible fluid mechanics is that we are not allowed to drop any term from the continuity equation.

As a consequence of the order of magnitude analysis, can be dropped from the :x direction momentum equation, because on multiplication with it assumes the smallest order of magnitude.

Eq 28.5 - y direction momentum equation.

All the terms of this equation are of a smaller magnitude than those of Eq. (28.4).

This equation can only be balanced if is of the same order of magnitude as other terms.

Thus they momentum equation reduces to

This means that the pressure across the boundary layer does not change. The pressure is impressed on the boundary layer, and its value is determined by hydrodynamic considerations.

This also implies that the pressure p is only a function of :x. The pressure forces on a body are solely determined by the inviscid flow outside the boundary layer.

The application of at the outer edge of boundary layer gives on integrating the well known Bernoulli's equation is obtained

a constant

The unknown pressure p in the x-momentum equation can be determined from Bernoulli's Eq. (28.9), if the inviscid velocity distribution 1(:x) is also known.

we solve the Prandtl boundary layer equations

for and with 1 obtained from the outer inviscid flow analysis. The equations are solved by commencing at the leading edge of the body and moving downstream to the desired location

it allows the no-slip boundary condition to be satisfied which constitutes a significant improvement over the potential flow analysis while solving real fluid flow problems.

$$at\ y^* = (\varepsilon) = \frac{\delta}{L},\ u^* = 1$$

The Prandtl boundary layer equations are thus a simplification of the Navier-Stokes equations.

$$at\ y = \delta, u = U(x)$$

Boundary Layer Coordinates

The boundary layer equations derived are in Cartesian coordinates.

The Velocity components u and v represent x and y direction velocities respectively.

$$u^*(x,y) \qquad v^*(x,y)$$

For objects with small curvature, these equations can be used with - x coordinate : streamwise direction

y coordinate : normal component

They are called Boundary Layer Coordinates.

Application of Boundary Layer Theory

The Boundary-Layer Theory is not valid beyond the point of separation.

At the point of separation, boundary layer thickness becomes quite large for the thin layer approximation to be valid.

It is important to note that boundary layer theory can be used to locate the point of seperation itself.

In applying the boundary layer theory although U is the free-stream velocity at the outer edge of the boundary layer, it is interpreted as the fluid velocity at the wall calculated from inviscid flow considerations (known as Potential wall Velocity)

Mathematically, application of the boundary - layer theory converts the character of governing Navier-Stroke equations from elliptic to parabolic

This allows the marching in flow direction, as the solution at any location is independent of the conditions farther downstream velocity U_∞.

The fluid extends to infinity in all directions from the plate. The physical problem is already illustrated in Fig. 28.1 Blasius wanted to determine

(a) the velocity field solely within the boundary layer,
(b) the boundary layer thickness (δ),
(c) the shear stress distribution on the plate, and
(d) the drag force on the plate.

The Prandtl boundary layer equations in the case under consideration are

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = \nu\frac{\partial^2 u}{\partial y^2}$$

$$\nu = \mu / \rho$$

The boundary conditions are

$$at\ y = 0,\ u = v = 0$$

$$at\quad y = \infty,\quad u = U_\infty$$

Note that the substitution of the term $-\frac{1}{\rho}\frac{dp}{dx}$ in the original boundary layer momentum equation in terms of the free stream velocity produces $U_\infty \frac{dU_\infty}{dx}$ which is equal to zero.

Hence the governing Eq. (28.15) does not contain any
pressure-gradient term.

U_∞, ν, x, y $\quad u = u(U_\infty, \nu, x, y)$

[illegible]

This relation has five variables U_∞, ν, x, y.

It involves two dimensions, length and time.

Thus it can be reduced to a dimensionless relation in terms of (5-2) #3 quantities (Buckingham Pi Theorem)

Thus a similarity variables can be used to find the solution

Such flow fields are called self-similar flow field .

Law of Similarity for Boundary Layer Flows

It states that the u component of velocity with two velocity profiles of u(:x,y) at different :x locations differ only by scale factors in u and y .

Therefore, the velocity profiles u(:x,y) at all values of :x can be made congruent if they are plotted in coordinates which have been made dimensionless with reference to the scale factors.

The local free stream velocity 1(:x) at section :x is an obvious scale factor for u, because the dimensionless u(:x) varies between zero and unity with y at all sections.

The scale factor for y , denoted by g(:x) , is proportional to the local boundary layer thickness so that y itself varies between zero and unity.

Velocity at two arbitrary :x locations, namely x_1 and x_2 should satisfy the equation

$$\frac{u[x_1,\{y/g(x_1)\}]}{U(x_1)} = \frac{u[x_2,\{y/g(x_2)\}]}{U(x_2)}$$

Now, for Blasius flow, it is possible to identify g(x) with the boundary layers thickness δ we know

$$\varepsilon = \frac{\delta}{L} \sim \frac{1}{\sqrt{\mathrm{Re}_L}}$$

Thus in terms of x we get

$$\frac{\delta}{x} \sim \frac{1}{\sqrt{\frac{U_\infty x}{\nu}}}$$

$$\delta \sim \sqrt{\frac{\nu x}{U_\infty}}$$

i.e.,

$$\frac{u}{U_\infty} = F\left(\frac{y}{\sqrt{\frac{\nu x}{U_\infty}}}\right) = F(\eta)$$

where or more precisely, $\eta \sim \frac{y}{\delta}$ and $\delta \sim \sqrt{\frac{\nu x}{U_\infty}}$

$$\eta = \frac{y}{\sqrt{\frac{\nu x}{U_\infty}}}$$

$$y = \eta\sqrt{\frac{\nu x}{U_\infty}}$$

$$dy = \sqrt{\frac{\nu x}{U_\infty}}d\eta$$

The stream function can now be obtained in terms of the velocity components as

$$\psi = \int u dy = \int U_\infty F(\eta)\sqrt{\frac{\nu x}{U_\infty}}d\eta = \sqrt{U_\infty \nu x}\int F(\eta)d\eta$$

or

$$\psi = \sqrt{U_\infty \nu x} f(\eta) + D$$

where D is a constant. Also $\int F(\eta)d\eta = f(\eta)$ and the constant of integration is zero if the stream function at the solid surface is set equal to zero.

Now, the velocity components and their derivatives are:

$$u = \frac{\partial \psi}{\partial y} = \frac{\partial \psi}{\partial \eta}\frac{\partial \eta}{\partial y} = U_\infty f'(\eta)$$

$$v = -\frac{\partial \psi}{\partial x} = -\sqrt{U_\infty \nu}\left[\frac{1}{2}\frac{1}{\sqrt{x}}f(\eta) + \sqrt{x}f'(\eta)\left\{-\frac{1}{2}\frac{y}{\sqrt{\nu x/U_\infty}}\frac{1}{x}\right\}\right]$$

or

$$v = \frac{1}{2}\sqrt{\frac{\nu U_\infty}{x}}\left[\eta f'(\eta) - f(\eta)\right]$$

$$\frac{\partial u}{\partial x} = U_\infty f''(\eta)\frac{\partial \eta}{\partial x} = U_\infty f''(\eta)\left[-\frac{1}{2}\ \frac{y}{\sqrt{\nu x / U_\infty}}\ \frac{1}{x}\right]$$

$$\frac{\partial u}{\partial x} = -\frac{U_\infty}{2}\ \frac{\eta}{x}\ f''(\eta)$$

$$\frac{\partial u}{\partial y} = U_\infty f''(\eta)\frac{\partial \eta}{\partial y} = U_\infty f''(\eta)\left[\frac{1}{\sqrt{\nu x / U_\infty}}\right]$$

$$\frac{\partial u}{\partial y} = U_\infty \sqrt{\frac{U_\infty}{\nu x}} f''(\eta)$$

$$\frac{\partial^2 u}{\partial y^2} = U_\infty \sqrt{\frac{U_\infty}{\nu x}} f'''(\eta)\left\{\frac{1}{\sqrt{\nu x / U_\infty}}\right\}$$

$$\frac{\partial^2 u}{\partial y^2} = \frac{U_\infty^2}{\nu x} f'''(\eta)$$

Substituting

$$-\frac{U_\infty^2}{2}\frac{\eta}{x} f'(\eta) f''(\eta) + \frac{U_\infty^2}{2x}\left[\eta f'(\eta) - f(\eta)\right] f''(\eta) = \frac{U_\infty^2}{x} f'''(\eta)$$

$$-\frac{1}{2}\frac{U_\infty^2}{x} f(\eta) f''(\eta) = \frac{U_\infty^2}{x} f'''(\eta)$$

or,

where

$$f(\eta) = \int F(\eta) d\eta + C$$
$$= \int \frac{u}{U_\infty} d\eta + C$$

and

$$\eta = \frac{y}{\sqrt{\frac{vx}{U_\infty}}}$$

This is known as Blasius Equation .

Contd. from Previous Slide

The boundary conditions as in combination with Eqns become

at $y = 0, u = 0$, therefore $\eta = 0 : f(\eta) = 0, f'(\eta) = 0$

at therefore (28.23)

Blasius obtained the solution of this equation in the form of series expansion through analytical techniques

we shall not discuss this technique. However, we shall discuss a numerical technique to solve the aforesaid equation which can be understood rather easily.

Note that the equation for f does not contain .

Boundary conditions at $x = 0$ and $y = \infty$ merge into the condition $\eta \rightarrow \infty, u/U_\infty = f' = 1$. This is the key feature of similarity solution.

we can rewrite Eq. (28.22) as three first order differential equations in the following way

$f' = G$

Let us next consider the boundary conditions. The condition remains valid. The condition means that $G' = H$.

The condition $H' = -\frac{1}{2} fH$ gives us .

Note that the equations for f and G have initial values. However, the value for H(0) is not known. Hence, we do not have a usual initial-value problem.

Shooting Technique $f(0) = 0$

$f'(0) = 0$ $G(0) = 0$

$f'(\infty) = 1$ $G(\infty) = 1$

$$l_1 = hF_2(f_x, G_x, H_x, \eta_x)$$

$$m_1 = hF_3(f_x, G_x, H_x, \eta_x)$$

$$k_2 = hF_1\left\{\left((f_x + \frac{1}{2}k_1), (G_x + \frac{1}{2}l_1), (H_x + \frac{1}{2}m_1), (\eta_x + \frac{h}{2})\right)\right\}$$

$$l_2 = hF_2\left\{\left((f_x + \frac{1}{2}k_1), (G_x + \frac{1}{2}l_1), (H_x + \frac{1}{2}m_1), (\eta_x + \frac{h}{2})\right)\right\}$$

$$m_2 = hF_3\left\{\left((f_x + \frac{1}{2}k_1), (G_x + \frac{1}{2}l_1), (H_x + \frac{1}{2}m_1), (\eta_x + \frac{h}{2})\right)\right\}$$

In a similar way K3, /3, m3 and k4, /4, m4 mare calculated following standard formulae for the Runge-Kutta integration.
For example, K3 is given by

$$k_3 = hF_1\left\{\left((f_x + \frac{1}{2}k_2), (G_x + \frac{1}{2}l_2), (H_x + \frac{1}{2}m_2), (\eta_x + \frac{h}{2})\right)\right\}$$

The functions F1,
F2and F3 are G, H , - f H I J respectively. Then at a distance $\Delta\eta$ from the wall, we have

$$f(\Delta\eta) = f(0) + G(0)\Delta\eta$$

$$G(\Delta\eta) = G(0) + H(0)\Delta\eta$$

As it has been mentioned earlier $H(\Delta\eta) = H(0) + H'(0)\Delta\eta$ is unknown. It must be determined such that the condition is satisfied.

$$H'(\Delta\eta) = -\frac{1}{2}f(\Delta\eta)H(\Delta\eta)$$

$$f''(0) = H(0) = A$$

$$f'(\infty) = G(\infty) = 1$$

explain the solution procedure in greater detail. The program uses Runge Kutta integration together with the Newton Raphson method

Download the program

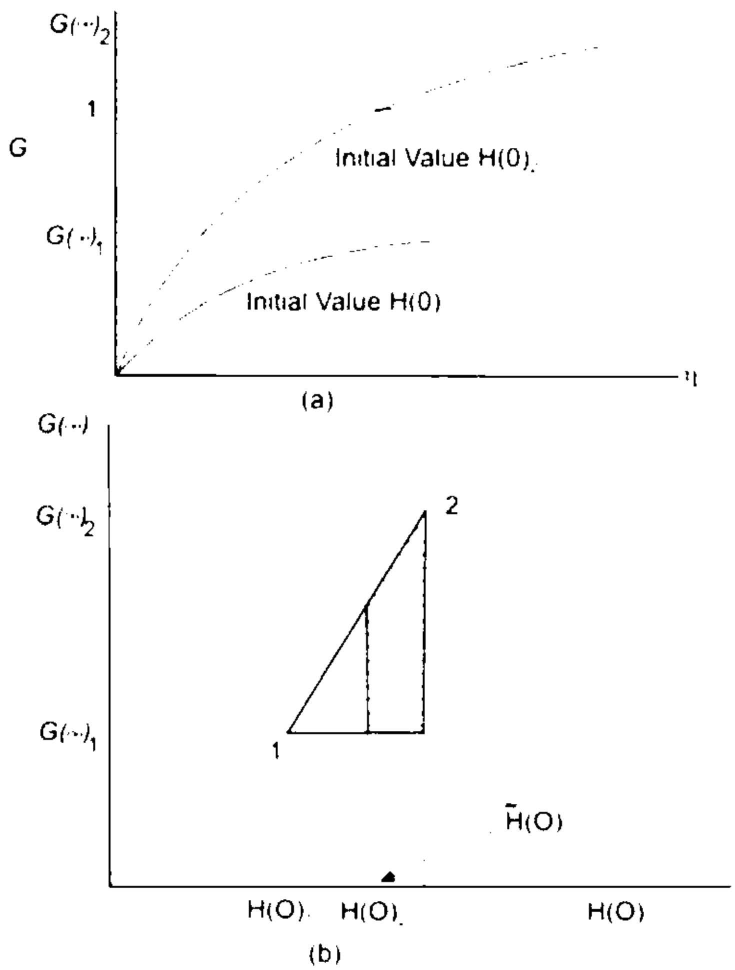


Fig: Correcting the initial guess for H(o)

Measurements to test the accuracy of theoretical results were carried out by many scientists. In his experiments, J. Nikuradse, found excellent agreement with the theoretical results with respect to velocity distribution within the boundary layer of a stream of air on a flat plate.

In the next slide we'll see some values of the velocity profile shape and in tabular format.

Values of the velocity profile shape

0	0	0	0.33206
0.2	0.00664	0.006641	0.33199
0.4	0.02656	0.13277	0.33147
0.8	0.10611	0.26471	0.32739
1.2	0.23795	0.39378	0.31659
1.6	0.42032	0.51676	0.29667
2.0	0.65003	0.62977	0.26675
2.4	0.92230	0.72899	0.22809
2.8	1.23099	0.81152	0.18401
3.2	1.56911	0.87609	0.13913
3.6	1.92954	0.92333	0.09809
4.0	2.30576	0.95552	0.06424
4.4	2.69238	0.97587	0.03897
4.8	3.08534	0.98779	0.02187
5.0	3.28329	0.99155	0.01591
8.8	7.07923	1.00000	0.00000

End of Lecture 28!

To start next lecture click next button or select from wall Shear Stress

with the profile known, wall shear can be evaluated as

$$\tau_w = \mu \frac{\partial u}{\partial y}\bigg|_{y=0}$$

Now, $$\frac{\partial u}{\partial y} = U_\infty f'(\eta)\frac{\partial \eta}{\partial y}$$

or $$\tau_\omega = \mu U_\infty f''(\eta)\frac{\partial \eta}{\partial y}\bigg|_{\eta=0}$$

$$= \mu U_\infty H \frac{\partial \eta}{\partial y}\bigg|_{\eta=0}$$

or $$\tau_\omega = \mu U_\infty \times 0.3326 \times \frac{1}{\sqrt{(\nu x)/U_\infty}}$$

$$[f''(0) = 0.3326]$$

$$\tau_\omega = \frac{0.332\rho U_\infty^2}{\sqrt{\mathrm{Re}_x}}$$

$$C_{fx} = \frac{\tau_\omega}{1/2\rho U_\infty^2}$$

Substituting we get

$$C_{f_x} = \frac{0.664}{\sqrt{\text{Re}_x}}$$

(Skin Friction Coefficient)

In 1951, Liepmann and Dhawan , measured the shearing stress on a flat plate directly. Their results showed a striking confirmation of Eq. (29.1).

Total frictional force per unit width for the plate of length L is

$$F = \int_0^L \tau_w dx$$

$$F = \int_0^L \frac{0.332 \rho U_\infty^2}{\sqrt{U_\infty / \nu}} \frac{dx}{\sqrt{x}}$$

or

$$F = \left[\frac{0.332 \rho U_\infty^2}{\sqrt{U_\infty / \nu}} \times \frac{x^{1/2}}{1/2} \right]_0^L$$

or

$$F = 0.664 \times \rho U_\infty^2 \sqrt{\frac{\nu L}{U_\infty}}$$

and the average skin friction coefficient is

$$\overline{C_f} = \frac{F}{1/2(\rho U_\infty^2 L)} = \frac{1.328}{\sqrt{\text{Re}_L}}$$

where, $\text{Re}_L = U_\infty L / \nu$.

For a flat plate of length L in the streamwise direction and width w perpendicular to the flow, the Drag D would be

$$D = F(2wL) = 0.664(2wL)\rho U_\infty^2\left(\frac{\nu L}{U_\infty}\right)^{1/2} = 1.328wL\left(\frac{\rho\mu U_\infty^3}{L}\right)^{1/2} \quad (29.4)$$

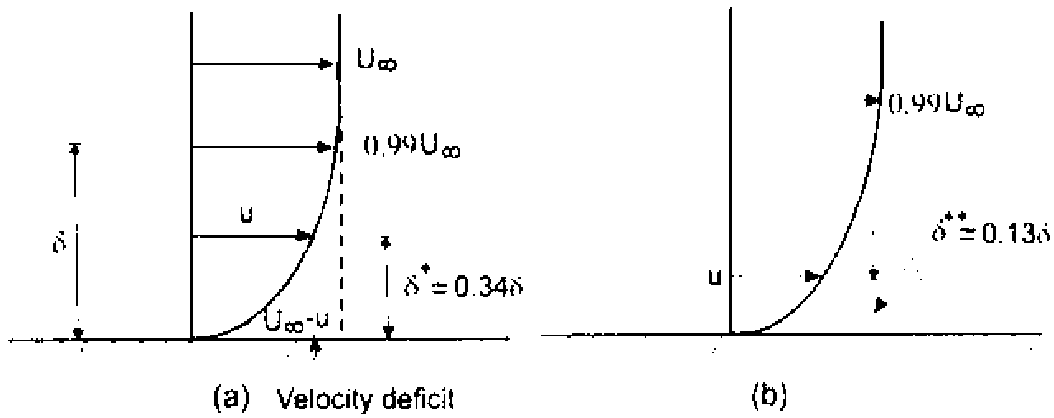


Fig. 29.1 (Displacement thickness) (b) Momentum thickness (δ^*)

Displacement thickness : It is defined as the distance by which the external potential flow is displaced outwards due to the decrease in velocity in the boundary layer.

$$U_\infty \delta^* = \int_0^\infty (U_\infty - u)dy$$

$$\delta^* = \int_0^\infty \left(1 - \frac{u}{U_\infty}\right)dy$$

Therefore, $dy = \delta d\eta = \sqrt{\frac{\nu x}{U_\infty}}d\eta$

u / U_∞

Substituting the values of and from Eqs (28.21a) and (28.19) into Eq.(29.6), we obtain

$$\delta^* = \sqrt{\frac{\nu x}{U_\infty}}\int_0^\infty (1 - f')d\eta = \sqrt{\frac{\nu x}{U_\infty}}\lim_{\eta\to\infty}[\eta - f(\eta)]$$

or, $\delta^* = 1.7208\sqrt{\frac{\nu x}{U_\infty}} = \frac{1.7208x}{\sqrt{Re_x}}$ (29.7)

Following the analogy of the displacement thickness, a momentum thickness may be defined.

Momentum thickness (δ^{**}): It is defined as the loss of momentum in the boundary layer as compared with that of potential flow. Thus

$$\rho U_\infty^2 \delta^{**} = \int_0^\infty \rho u (U_\infty - u) dy$$

$$\delta^{**} = \int_0^\infty \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty}\right) dy$$

with the substitution (u/U_∞) and

can evaluate numerically the value of δ^{**} for a flat plate as

$$\delta^{**} = \sqrt{\frac{\nu x}{U_\infty}} \int_0^\infty f'(1 - f') d\eta$$

$$or \quad \delta^{**} = 0.664\sqrt{\frac{\nu x}{U_\infty}} = \frac{0.664x}{\sqrt{Re_x}}$$

The relationships between $\delta, \delta^* and \delta^{**}$ have been shown in Fig. 29.1.

Momentum-Integral Equations For The Boundary Layer

To employ boundary layer concepts in real engineering designs, we need approximate methods that would quickly lead to an answer even if the accuracy is somewhat less.

Karman and Pohlhausen devised a simplified method by satisfying only the boundary conditions of the boundary layer flow rather than satisfying Prandtl's differential equations for each and every particle within the boundary layer. we shall discuss this method herein.

Consider the case of steady, two-dimensional and incompressible flow, i.e. we shall refer to Eqs (28.10) to (28.14). Upon integrating the dimensional form of Eq. (28.10) with respect to y = 0 (wall) to y = δ (where δ signifies the interface of the free stream and the boundary layer), we obtain

$$\int_0^\delta \left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} \right) dy = \int_0^\delta \left(-\frac{1}{\rho}\frac{\partial p}{\partial x} + v\frac{\partial^2 u}{\partial y^2} \right) dy$$

or, $$\int_0^\delta u\frac{\partial u}{\partial x}dy + \int_0^\delta v\frac{\partial u}{\partial y}dy = \int_0^\delta -\frac{1}{\rho}\frac{\partial p}{\partial x}dy + \int_0^\delta v\frac{\partial^2 u}{\partial y^2}dy$$

The second term of the left hand side can be expanded as

$$\int_0^\delta v\frac{\partial u}{\partial y}dy = [vu]_0^\delta - \int_0^\delta u\frac{\partial v}{\partial y}dy$$

or, $$\int_0^\delta v\frac{\partial u}{\partial y}dy = U_\infty v_\delta + \int_0^\delta u\frac{\partial u}{\partial x}dy \left(\sin ce\ \frac{\partial u}{\partial x} = -\frac{\partial v}{\partial y} \right)$$ by continuity equation

or, $$\int_0^\delta v\frac{\partial u}{\partial y}dy = -U_\infty \int_0^\delta \frac{\partial u}{\partial x}dy + \int_0^\delta u\frac{\partial u}{\partial x}dy$$

$$\int_0^\delta 2u\frac{\partial u}{\partial x}dy - U_\infty \int_0^\delta \frac{\partial u}{\partial x}dy = -\int_0^\delta \frac{1}{\rho}\frac{\partial p}{\partial x}dy - v\left.\frac{\partial u}{\partial y}\right|_{y=0}$$

Substituting the relation between $\frac{\partial p}{\partial x}$ and the free stream velocity U_∞ for the inviscid zone in Eq. (29.12) we get

$$\int_0^\delta 2u\frac{\partial u}{\partial x}dy - U_\infty \int_0^\delta \frac{\partial u}{\partial x}dy - \int_0^\delta U_\infty \frac{dU_\infty}{dx}dy = -\left(\frac{\mu \left.\frac{\partial u}{\partial y}\right|_{y=0}}{\rho} \right)$$

$$\int_0^\delta \left(2u\frac{\partial u}{\partial x} - U_\infty \frac{\partial u}{\partial x} - U_\infty \frac{dU_\infty}{dx} \right) dy = -\frac{\tau_w}{\rho}$$

which is reduced to

$$\int_0^\delta \frac{\partial}{\partial x}\left[u(U_\infty - u) \right] dy + \frac{dU_\infty}{dx}\int_0^\delta (U_\infty - u) dy = \frac{\tau_w}{\rho}$$

Since the integrals vanish outside the boundary layer, we are allowed to increase the integration limit to infinity (i.e $\delta = \infty$.)

$$\int_0^\delta \frac{\partial}{\partial x}\left[u(U_\infty - u) \right] dy + \frac{dU_\infty}{dx}\int_0^\delta (U_\infty - u) dy = \frac{\tau_w}{\rho}$$

or, $$\frac{d}{dx}\int_0^\delta \left[u(U_\infty - u) \right] dy + \frac{dU_\infty}{dx}\int_0^\delta (U_\infty - u) dy = \frac{\tau_w}{\rho}$$

Substituting Eq. (29.6) and (29.7) in Eq. (29.13) we obtain

$$\frac{d}{dx}\left[U_\infty^2 \delta^{**} \right] + \delta^* U_\infty \frac{dU_\infty}{dx} = \frac{\tau_w}{\rho}$$

where $\delta^* = \int_0^\delta \left(1 - \frac{u}{U_\infty} \right) dy$ is the displacement thickness

is momentum thickness $\delta^{**} = \int_0^\delta \frac{u}{U_\infty}\left(1 - \frac{u}{U_\infty} \right) dy$

is known as momentum integral equation for two dimensional incompressible laminar boundary layer. The same remains valid for turbulent boundary layers as well.

Needless to say, the wall shear stress (τ_w) will be different for laminar and turbulent flows.

The term $U_\infty \frac{dU_\infty}{dx}$ signifies space-wise acceleration of the free stream. Existence of this term means that free stream pressure gradient is present in the flow direction.

For example, we get finite value of $U_\infty \frac{dU_\infty}{dx}$ outside the boundary layer in the entrance region of a pipe or a channel. For external flows, the

existence of $U_\infty \frac{dU_\infty}{dx}$ depends on the shape of the body.

During the flow over a flat plate, $U_\infty \frac{dU_\infty}{dx} = 0$ and the momentum integral equation is reduced to

$$\frac{d}{dx}\left[U_\infty^2 \delta^{**}\right] = \frac{\tau_w}{\rho}$$

Seperation of Boundary Layer

It has been observed that the flow is reversed at the vicinity of the wall under certain conditions.

The phenomenon is termed as separation of boundary layer.

Separation takes place due to excessive momentum loss near the wall in a boundary layer trying to move downstream against increasing pressure,

i.e., $\frac{dp}{dx} > 0$, which is called adverse pressure gradient.

Figure 29.2 shows the flow past a circular cylinder, in an infinite medium.

Up to $\theta = 90^\circ$, the flow area is like a constricted passage and the flow behaviour is like that of a nozzle.

Beyond $\theta = 90^\circ$ the flow area is diverged, therefore, the flow behaviour is much similar to a diffuser

This dictates the inviscid pressure distribution on the cylinder which is shown by a firm line in Fig. 29.2.

Here

p_∞ : pressure in the free stream

U_∞ : velocity in the free stream and

p is the local pressure on the cylinder.

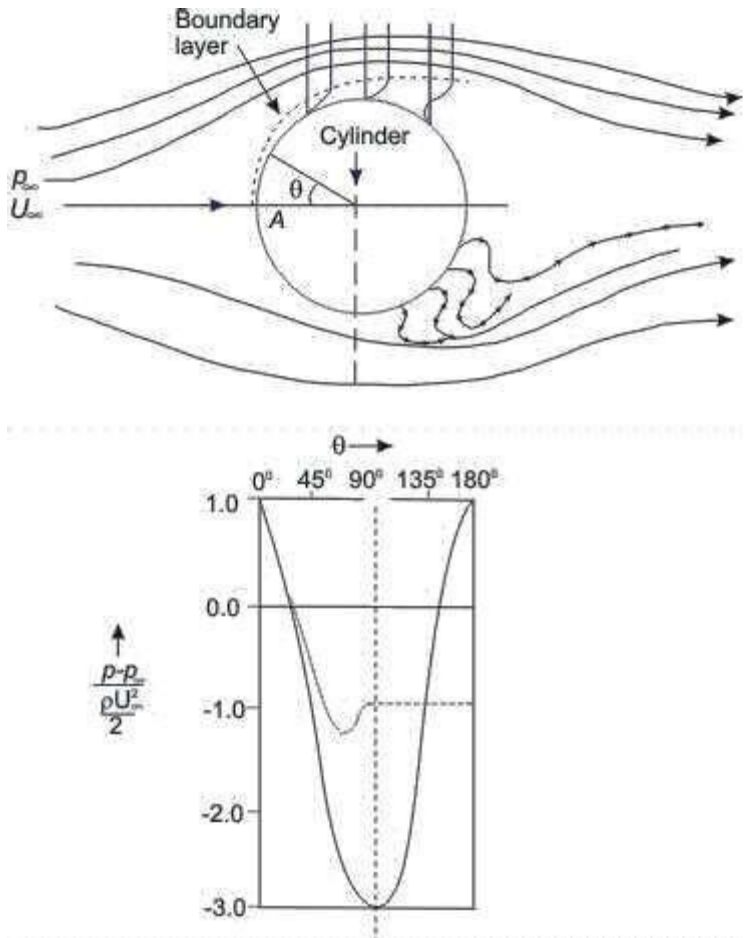


Fig.Flow separation and formation of wake behind a circular cylinder Consider the forces in the flow field. In the inviscid region,

Until $\theta = 90°$ the pressure force and the force due to streamwise acceleration i.e. inertia forces are acting in the same direction (pressure gradient being negative/favourable)

Beyond $\theta = 90°$, the pressure gradient is positive or adverse. Due to the adverse pressure gradient the pressure force and the force due to acceleration will be opposing each other in the in viscid zone of this part.

So long as no viscous effect is considered, the situation does not cause any sensation. In the viscid region (near the solid boundary),

Up to $\theta = 90^\circ$, the viscous force opposes the combined pressure force and the force due to acceleration. Fluid particles overcome this viscous resistance due to continuous conversion of pressure force into kinetic energy.

Beyond $\theta = 90^\circ$, within the viscous zone, the flow structure becomes different. It is seen that the force due to acceleration is opposed by both the viscous force and pressure force.

Depending upon the magnitude of adverse pressure gradient, somewhere around $\theta = 90^\circ$, the fluid particles, in the boundary layer are separated from the wall and driven in the upstream direction. However, the far field external stream pushes back these separated layers together with it and develops a broad pulsating wake behind the cylinder.

The mathematical explanation of flow-separation : The point of separation may be defined as the limit between forward and reverse flow in the layer very close to the wall, i.e., at the point of separation

$$\left(\frac{\partial u}{\partial y}\right)_{y=0} = 0$$

This means that the shear stress at the wall, $\tau_w = 0$. But at this point, the adverse pressure continues to exist and at the downstream of this point the flow acts in a reverse direction resulting in a back flow.

we can also explain flow separation using the argument about the second derivative of velocity u at the wall. From the dimensional form of the momentum at the wall, where u = v = 0, we can write

(29.17) $$\left(\frac{\partial^2 u}{\partial y^2}\right)_{y=0} = \frac{1}{\mu}\frac{dp}{dx}$$

Consider the situation due to a favourable pressure gradient where $\frac{dp}{dx} < 0$ we have,

. (From Eq. (29.17))

As we proceed towards the free stream, the

velocity u approaches U_∞ asymptotically, so $\partial u / \partial y$ decreases at a continuously lesser rate in ydirection.

This means that $\partial^2 u / \partial y^2$ remains less than zero near the edge of the boundary layer. The curvature of a velocity profile $\partial^2 u / \partial y^2$ is always negative as shown in (Fig. 29.3a)

Consider the case of adverse pressure gradient, $\partial p / \partial x > 0$

At the boundary, the curvature of the profile must be positive (since $\partial p / \partial x > 0$).

Near the interface of boundary layer and free stream the previous argument regarding $\partial u / \partial y$ and $\partial^2 u / \partial y^2$ still holds good and the curvature is negative.

Thus we observe that for an adverse pressure gradient, there must exist a point for which $\partial^2 u / \partial y^2 = 0$. This point is known as point of inf/ection of the velocity profile in the boundary layer as shown in Fig. 29.3b

However, point of separation means $\partial u / \partial y = 0$ at the wall.

$\partial^2 u / \partial y^2 > 0$ at the wall since separation can only occur due to adverse pressure gradient.

But we have already seen that at the edge of the boundary layer, $\partial^2 u / \partial y^2 < 0$. It is therefore, clear that if there is a point of separation, there must exist a point of inflection in the velocity profile.

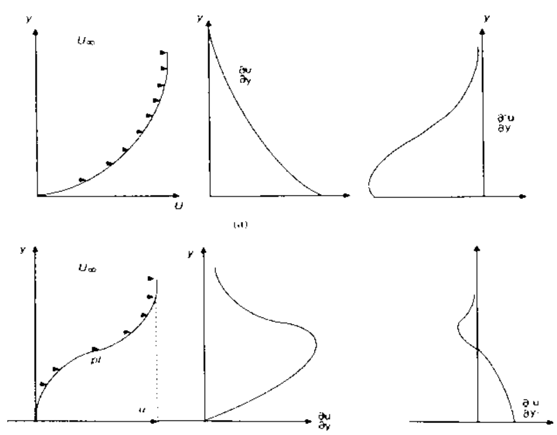


Fig: Velocity distribution within a boundary layer

(a) Favourable pressure gradient, $\frac{dp}{dx} < 0$

(b) adverse pressure gradient, $\frac{dp}{dx} > 0$

Let us reconsider the flow past a circular cylinder and continue

our discussion on the wake behind a cylinder. The pressure distribution which was shown by the firm line is obtained from the potential flow theory. However. somewhere near $\theta = 90^\circ$ (in experiments it has been observed to be at $\theta = 81^\circ$) . the boundary layer detaches itself from the wall.

Meanwhile, pressure in the wake remains close to separation-point-pressure since the eddies (formed as a consequence of the retarded layers being carried together with the upper layer through the action of shear) cannot convert rotational kinetic energy into pressure head. The actual pressure distribution is shown by the dotted line.

Since the wake zone pressure is less than that of the forward stagnation point the cylinder experiences a drag force which is basically attributed to the pressure difference.

The drag force, brought about by the pressure difference is known as form drag whereas the shear stress at the wall gives rise to skin friction drag. Generally, these two drag forces together are responsible for resultant drag on a body

Karman-Pohlhausen Approximate Method For Solution of Momentum Integral Equation over A Flat Plate

The basic equation for this method is obtained by integrating the :x direction momentum equation (boundary layer momentum equation) with respect to y from the wall (at y # 0) to a distance $\delta(x)$ which is assumed to be outside the boundary layer. Using this notation, we can rewrite the Karman momentum integral equation as

The effect of pressure gradient is described by the second term on the left hand side. For pressure gradient surfaces in external flow or for the developing sections in internal flow, this term contributes to the pressure gradient.

we assume a velocity profile which is a polynomial

of $\eta = y/\delta$. being a form of similarity variable , implies that with the growth of boundary layer as distance :x varies from the leading

edge, the velocity profile (u/U_∞) remains geometrically similar. we choose a

velocity profile in the form

$$\frac{u}{U_\infty} = a_0 + a_1\eta + a_2\eta^2 + a_3\eta^3$$

In order to determine the constants a_0, a_1, a_2 and a_3 we shall prescribe the following boundary conditions

$$at\ \ y = 0, u = 0 \quad or \quad at\ \ \eta = 0, \frac{u}{U_\infty} = 0$$

at

$$at\ \ y = 0, \frac{\partial^2 u}{\partial y^2} = 0 \quad or \quad at\ \ \eta = 0, \frac{\partial^2}{\partial \eta^2}(u/U_\infty) = 0$$

The wall shear stress is given by

$$\tau_w = \mu \frac{\partial u}{\partial y}\bigg|_{y=0}$$

$$or\ \tau_w = \mu\left[\frac{\partial}{\delta\partial\eta}\left\{U_\infty\left(\frac{3}{2}\eta - \frac{1}{2}\eta^3\right)\right\}\right]_{\eta=0}$$

$$or\ \tau_w = \frac{3\mu U_\infty}{2\delta}$$

Substituting the values of δ^{**} and τ_w in Eq. (30.5) we get,

$$\frac{39}{280}\frac{d\delta}{dx} = \frac{3\mu U_\infty}{2\delta\rho U_\infty^2}$$

$$or\ \int \delta d\delta = \int \frac{140}{13}\frac{\mu}{\rho U_\infty}dx + C_1$$

$$or\ \frac{\delta^2}{2} = \frac{140}{13}\frac{\nu x}{U_\infty} + C_1$$

where C_1 is any arbitrary unknown constant.

The condition at the leading edge ( ) yields Finally we obtain, $x = 0,\ \delta = 0$ $C_1 = 0$

$$\delta^2 = \frac{280}{13}\frac{\nu x}{U_\infty}$$

$$or\ \delta = 4.64\sqrt{\frac{\nu x}{U_\infty}}$$

$$or\ \delta = \frac{4.64x}{\sqrt{Re}}$$

This is the value of boundary layer thickness on a flat plate. Although, the method is an approximate one, the result is found to be reasonably accurate. The value is slightly lower than the exact solution of laminar flow over a flat plate . As such, the accuracy depends on the order of the velocity profile. we could have have used a fourth order polynomial instead --

$$\frac{u}{U_\infty} = a_0 + a_1\eta + a_2\eta^2 + a_3\eta^3 + a_4\eta^4$$

In addition to the boundary conditions in Eq. (30.3), we shall require another boundary condition at

$$y = \delta, \frac{\partial^2 u}{\partial y^2} = 0 \;\; or\; at\;\; \eta = 1, \frac{\partial^2 (u / U_\infty)}{\partial \eta^2} = 0$$

This yields the constants as . Finally the velocity profile will be $a_4 = 1$

$$\frac{u}{U_\infty} = 2\eta - 2\eta^3 + \eta^4$$

Subsequently, for a fourth order profile the growth of boundary layer is given by

Integ $\delta = \frac{5.83x}{\sqrt{\mathrm{Re}_x}}$

A wide variety of "integral methods" in this category have been discussed by Rosenhead . The Thwaites method is found to be a very elegant method, which is an extension of the method due to Holstein and Bohlen

. we shall discuss the Holstein-Bohlen method in this section.

This is an approximate method for solving boundary layer equations for two-dimensional generalized flow. The integrated Eq. (29.14) for laminar flow with pressure gradient can be written as

$$\frac{d}{dx}\left[U^2 \delta^{**}\right] + \delta^* U \frac{dU}{dx} = \frac{\tau_\omega}{\rho}$$

The velocity profile at the boundary layer is considered to be a fourth-order polynomial in terms of the dimensionless distance $\eta = y/\delta$, and is expressed as The boundary conditions are

$$u/U = a\eta + b\eta^2 + c\eta^3 + d\eta^4$$

$$\eta = 0 : u = 0, v = 0 \qquad \frac{\nu}{\delta^2}\frac{\partial^2 u}{\partial\eta^2} = \frac{1}{\rho}\frac{dp}{dx} = -U\frac{dU}{dx}$$

$$\eta = 1 : u = U \qquad \frac{\partial u}{\partial \eta} = 0,\ \frac{\partial^2 u}{\partial \eta^2} = 0$$

A dimensionless quantity, known as shape factor is introduced as

$$\lambda = \frac{\delta^2}{\nu}\frac{dU}{dx}$$

The following relations are obtained

$$a = 2 + \frac{\lambda}{6},\ b = -\frac{\lambda}{2},\ c = -2 + \frac{\lambda}{2},\ d = 1 - \frac{\lambda}{6}$$

Now, the velocity profile can be expressed as

$$u/U = F(\eta) + \lambda G(\eta),$$

where

$$F(\eta) = 2\eta + 2\eta^3 + \eta^4,\ G(\eta) = \frac{1}{6}\eta(1-\eta)^3$$

The shear stress $\tau_\omega = \mu(\partial u/\partial y)_{y=0}$ is given by

$$\frac{\tau_\omega \delta}{\mu U} = 2 + \frac{\lambda}{6}$$

we use the following dimensionless parameters,

$$L = \frac{\tau_\omega \delta^{**}}{\mu U} = \frac{\delta^{**}}{\delta}\left(2 + \frac{\lambda}{6}\right)$$

$$K = \frac{(\delta^{**})^2}{\nu}\frac{dU}{dx} = \left(\frac{\delta^{**}}{\delta}\right)^2 \lambda$$

$$H = \delta^* / \delta^{**}$$

The integrated momentum reduces to

$$U\frac{d\delta^{**}}{dx}+\delta^{**}(2+H)\frac{dU}{dx}=\frac{\nu L}{\delta^{**}}$$

$$U\frac{d}{dx}\left[\frac{(\delta^{**})^2}{\nu}\right]=2[L-K(H+2)]$$

The parameter l is related to the skin friction The parameter K is linked to the pressure gradient.

If we take K as the independent variable . l and H can be shown to be the functions of K since

$$\frac{\delta^*}{\delta}=\int_0^1 (F(\eta)+\lambda G(\eta))(1-F(\eta)-\lambda G(\eta))d\eta$$

$$=\frac{37}{315}-\frac{\lambda}{945}-\frac{\lambda^2}{9072}$$

$$K=\frac{[\delta^{**}]^2}{\delta^2}\lambda=\lambda\left(\frac{37}{315}-\frac{\lambda}{945}-\frac{\lambda^2}{9072}\right)^2$$

Therefore,

$$L=\left(2+\frac{\lambda}{6}\right)\frac{\delta^{**}}{\delta}=\left(2+\frac{\lambda}{6}\right)\left(\frac{37}{315}-\frac{\lambda}{945}-\frac{\lambda^2}{9072}\right)=f_1(k)$$

$$H=\frac{\delta^*}{\delta^{**}}=\frac{(3/10)-(\lambda/120)}{(37/315)-(\lambda/945)-(\lambda^2/9072)}=f_2(k)$$

The right-hand side of Eq. (30.18) is thus a function of K alone.

walz pointed out that this function can be approximated with a good degree of accuracy by a linear function of K so that

$$2[L-K(H-2)]=a-bK \quad [Walz\text{'s approximation}]$$ [walz' approximation]

Equation can now be written as

$$\frac{d}{dx}\left(\frac{U[\delta^{**}]^2}{\nu}\right) = a - (b-1)\frac{U[\delta^{**}]^2}{\nu}\frac{1}{U}\frac{dU}{dx}$$

Solution of this differential equation for the dependent

variable $U[\delta^{**}]^2/\nu$ subject to the boundary condition 1 = 0 when :x = 0 , gives

$$\frac{U[\delta^{**}]^2}{\nu} = \frac{a}{U^{b-1}}\int_0^x U^{b-1}dx$$

with a = 0.47 and b # 6. the approximation is particularly close between the stagnation point and the point of maximum velocity.

Finally the value of the dependent variable is

$$\left[\delta^{**}\right]^2 = \frac{0.47\nu}{U^6}\int_0^x U^5\,dx$$

By taking the limit of Eq. (30.22), according to L'Hopital's rule, it can be shown that

$$[\delta^{**}]^2\,|_{x=0} = 0.47\,\nu/6U'(0)$$

This corresponds to K # 0.0783.

Note that $\left|\delta^{**}\right|$ is not equal to zero at the stagnation point. If $\left[[\delta^{**}]^2/\nu\right]$ is determined from Eq. (30.22), K(:x) can be obtained from Eq. (30.16).

Table 30.1 gives the necessary parameters for obtaining results, such as velocity profile and shear stress τ_ω The approximate method can be applied successfully to a wide range of problems.

Table 30.1 Auxiliary functions after Holstein and Bohlen

	K		
12	0.0948	2.250	0.356

10	0.0919	2.260	0.351
8	0.0831	2.289	0.340
7.6	0.0807	2.297	0.337
7.2	0.0781	2.305	0.333

1.2	0.23795	0.39378	0.39378
1.6	0.42032	0.51676	0.51676
2.0	0.65003	0.62977	0.62977
2.4	0.92230	0.72899	0.72899
2.8	1.23099	0.81152	0.81152
3.2	1.56911	0.87609	0.87609
3.6	1.92954	0.92333	0.92333
4.0	2.30576	0.95552	0.95552
4.4	2.69238	0.97587	0.97587
4.8	3.08534	0.98779	0.98779
5.0	3.28329	0.99155	0.99155
8.8	7.07923	1.00000	1.00000

As mentioned earlier, K is related to the pressure gradient and the shape factor.

Introduction of K in the integral analysis enables extension of Karman-Pohlhausen method for solving flows over curved geometry. However, theanalysis is not valid for the geometries, where $\lambda < -12$ and $\lambda > +12$

Point of Seperation

$$\tau_{\omega} = 0$$

For point of seperation,

$$\Rightarrow \frac{\mu U}{\delta}\left(2+\frac{\lambda}{6}\right)$$

or, $$2 + \frac{\lambda}{6} = 0$$

or, $$\lambda = -12$$

Turbulent Flow:

Introduction

The turbulent motion is an irregular motion.

Turbulent fluid motion can be considered as an irregular condition of flow in which various quantities (such as velocity components and pressure) show a random variation with time and space in such a way that the statistical average of those quantities can be quantitatively expressed.

It is postulated that the fluctuations inherently come

from disturbances (such as roughness of a solid surface) and they may be either dampened out due to viscous damping or may grow by drawing energy from the free stream.

At a Reynolds number less than the critical, the kinetic energy of flow is not enough to sustain the random fluctuations against the viscous damping and in such cases laminar flow continues to exist. At somewhat higher Reynolds number than the critical Reynolds number, the kinetic energy of flow supports the growth of fluctuations and transition to turbulence takes place.

Characteristics of Turbulent Flow

The most important characteristic of turbulent motion is the fact that velocity and pressure at a point fluctuate with time in a random manner.

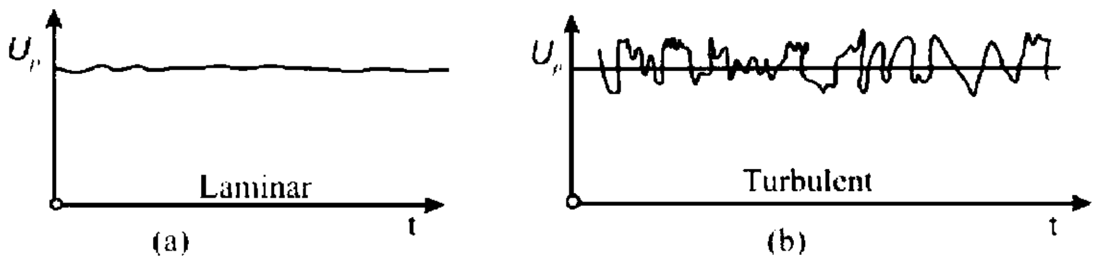


Fig. Variation of horizontal components of velocity for laminar and turbulent flows at a point P

The mixing in turbulent flow is more due to these fluctuations. As a result we can see more uniform velocity distributions in turbulent pipe flows as compared to the laminar flows .

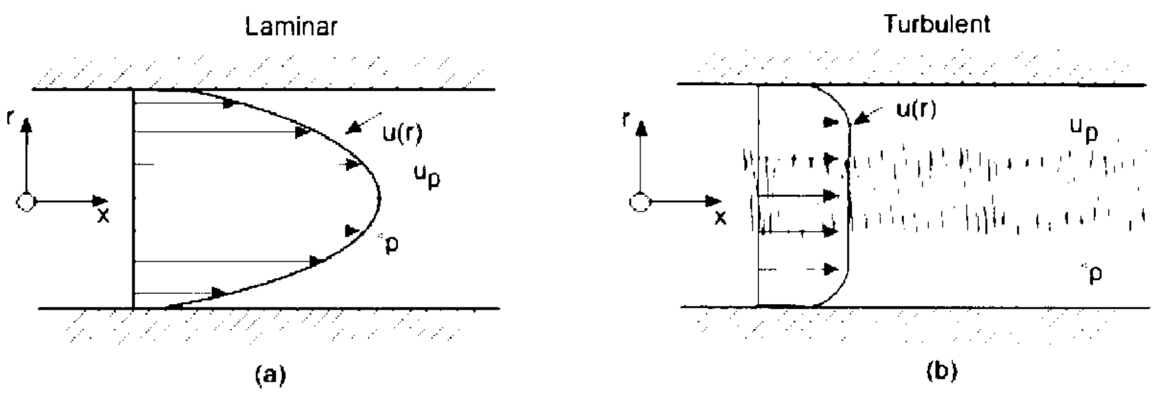


Fig. Comparison of velocity profiles in a pipe for (a) laminar and (b) turbulent flows

Turbulence can be generated by - frictional forces at the confining solid walls the flow of layers of fluids with different velocities over one another

The turbulence generated in these two ways are considered to be different.

Turbulence generated and continuously affected by fixed walls is designated as wall turbulence , and turbulence generated by two adjacent layers of fluid in absence of walls is termed as free turbulence . one of the effects of viscosity on turbulence is to make the flow more homogeneous and less dependent on direction.

Turbulence can be categorised as below -

Homogeneous Turbulence: Turbulence has the same structure quantitatively in all parts of the flow field.

Isotropic Turbulence: The statistical features have no directional preference and perfect disorder persists.

Anisotropic Turbulence: The statistical features have directional preference and the mean velocity has a gradient.

Homogeneous Turbulence : The term homogeneous turbulence implies that the velocity fluctuations in the system are random but the average turbulent characteristics are independent of the position in the fluid, i.e., invariant to axis translation.

Consider the root mean square velocity fluctuations

, ,

In homogeneous turbulence, the rms values of u', v' and w' can all be different, but each value must be constant over the entire turbulent field. Note that even if the rms fluctuation of any component, say u' s are constant over the entire field the instantaneous values of u necessarily differ from point to point at any instant.

Isotropic Turbulence: The velocity fluctuations are independent of the axis of reference,
i.e. invariant to axis rotation and reflection. Isotropic turbulence is by its definition always homogeneous . In such a situation, the gradient of the mean velocity does not exist, the mean velocity is either zero or constant throughout.

In isotropic turbulence fluctuations are independent of the direction of reference and

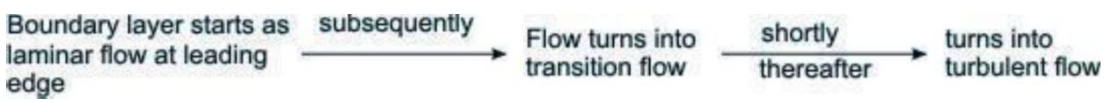
It is re-emphasised that even if the rms fluctuations at any point are same, their instantaneous values necessarily differ from each other at any instant.

Turbulent flow is diffusive and dissipative . In general, turbulence brings about better mixing of a fluid and produces an additional diffusive effect. Such a diffusion is termed as "Eddy-diffusion ".(Note that this is different from molecular diffusion)

At a large Reynolds number there exists a continuous transport of energy from the free stream to the large eddies. Then, from the large eddies smaller eddies are continuously formed. Near the wall smallest eddies destroy themselves in dissipating energy, i.e., converting kinetic energy of the eddies into intermolecular energy.

Laminar-Turbulent Transition

For a turbulent flow over a flat plate,



The turbulent boundary layer continues to grow in thickness, with a small region below it called a viscous sublayer. In this sub layer, the flow is well behaved,just as the laminar boundary layer

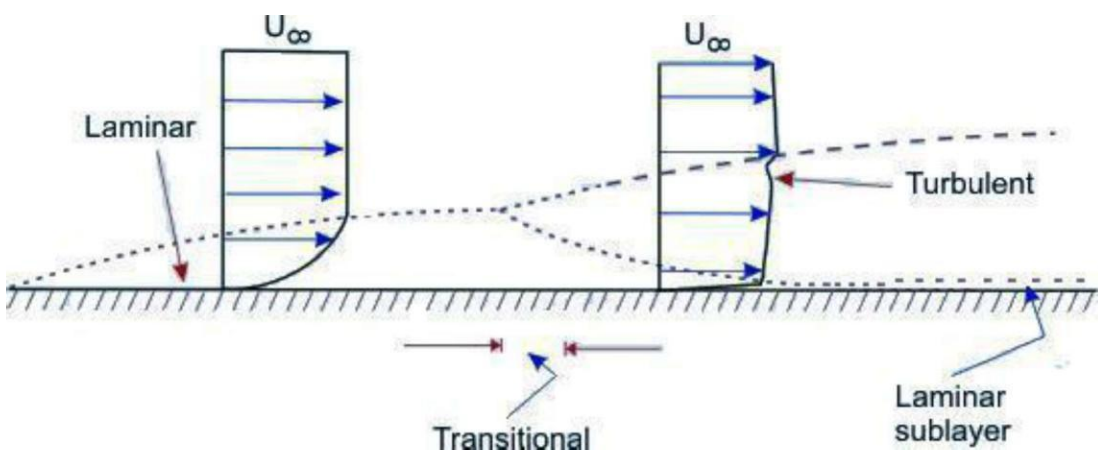


Fig. Laminar - turbulent transition illustration

Observe that at a certain axial location, the laminar boundary layer tends to become unstable. Physically this means that the disturbances in the flow grow in amplitude at this location.

Free stream turbulence, wall roughness and acoustic signals may be among the sources of such disturbances. Transition to turbulent flow is thus initiated with the instability in laminar flow

The possibility of instability in boundary layer was felt by Prandtl as early as 1912.The theoretical analysis of Tollmien and Schlichting showed that unstable waves could exist if the Reynolds number was 575.

The Reynolds number was defined as

$$Re = U_\infty \delta^* / \nu$$

where U_∞ is the free stream velocity , δ^* is the displacement thickness and ■ is the kinematic viscosity .

Taylor developed an alternate theory, which assumed that the transition is caused by a momentary separation at the boundary layer associated with the free stream turbulence.

In a pipe flow the initiation of turbulence is usually observed at Reynolds numbers($U_\infty D/\nu$)in the range of 2000 to 2700.

The development starts with a laminar profile, undergoes a transition, changes over to turbulent profile and then stays turbulent

thereafter (Fig. 32.4). The length of development is of the order of 25 to 40 diameters of the pipe.

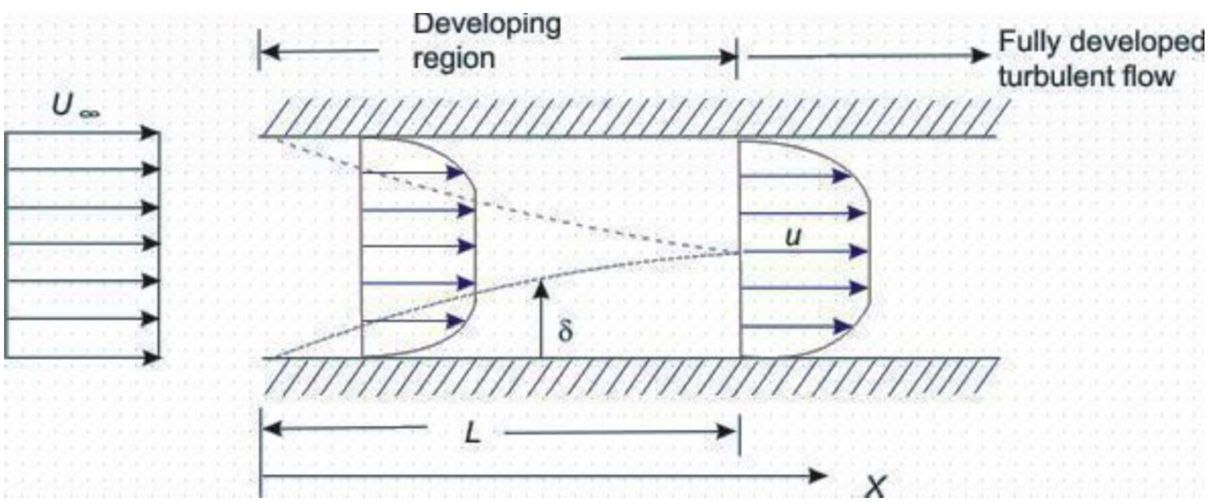


Fig. Development of turbulent flow in a circular duct

Fully Developed Turbulent Flow In A Pipe For Moderate Reynolds Numbers

The entry length of a turbulent flow is much shorter than that of a laminar flow, J. Nikuradse determined that a fully developed profile for turbulent flow can be observed after an entry length of 25 to 40 diameters. we shall focus to fully developed turbulent flow in this section.

Considering a fully developed turbulent pipe flow we can write

$$2\pi R\tau_w = -\left(\frac{dp}{dx}\right)\pi R^2$$

or

$$\left(-\frac{dp}{dx} = \frac{2\tau_w}{R}\right)$$

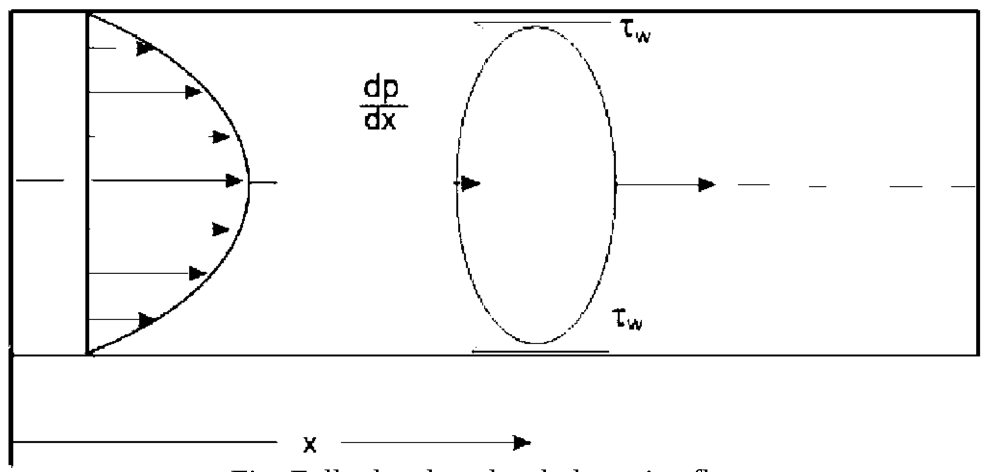


Fig: Fully developed turbulent pipe flow

It can be said that in a fully developed flow, the pressure gradient balances the wall shear stress only and has a constant value at any x. However, the friction factor (Darcy friction factor) is defined in a fully developed flow as

$$-\left(\frac{dp}{dx}\right) = \frac{\rho f U_{av}^2}{2D}$$

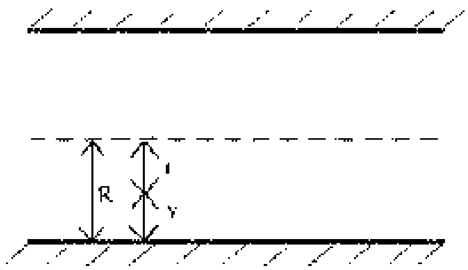
Comparing, we can write

$$\tau_w = \frac{f}{8}\rho U_{av}^2$$

H. Blasius conducted a critical survey of available experimental results and established the empirical correlation for the above equation as

$$f = 0.3164\,\mathrm{Re}^{-0.25} \quad \text{where} \quad \mathrm{Re} = \rho U_{av} D/\mu$$

It is found that the Blasius's formula is valid in the range of Reynolds number of $\mathrm{Re} \le 10^5$. At the time when Blasius compiled the experimental data, results for higher Reynolds



From equation

$$\pi R^2 U_{av} = 2\pi \bar{u} \int_0^R (R-y)(y/R)^{1/n}(-dy)$$

or

$$\pi R^2 U_{av} = 2\pi \bar{u}\left[\frac{n}{n+1}\left(R^{\frac{n-1}{n}} y^{\frac{n+1}{n}}\right) - \frac{n}{2n+1}\left(y^{\frac{2n+1}{n}} R^{-\frac{1}{n}}\right)\right]_0^R$$

or

$$\pi R^2 U_{av} = 2\pi \bar{u}\left[R^2 \frac{n}{n+1} - \frac{n}{2n+1} R^2\right]$$

or

$$\pi R^2 U_{av} = 2\pi R^2 \bar{u}\left[\frac{n^2}{(n+1)(2n+1)}\right]$$

or

$$\frac{U_{av}}{\bar{u}} = \frac{2n^2}{(n+1)(2n+1)}$$

Now, for different values of n (for different Reynolds numbers) we

shall obtain different values. on substitution $U_{av}/\bar{u}$ of Blasius resistance formula, the following expression for the shear stress at the wall can be obtained.

$$\left(\frac{\bar{u}}{u_\tau}\right)^{7/4} = 44.44\left(\frac{u_\tau R}{\nu}\right)^{1/4}$$

or

$$\frac{\bar{u}}{u_\tau} = 8.74\left(\frac{u_\tau R}{\nu}\right)^{1/7}$$

Now we can assume that the above equation is not only valid at the pipe axis (y # R) but also at any distance from the wall y and a general form is proposed as

$$\frac{\bar{u}}{u_\tau} = 8.74\left(\frac{y u_\tau}{\nu}\right)^{1/7}$$

Concluding Remarks :

It can be said that (1/7)th power velocity distribution law can be derived from Blasius's resistance formula .

$Re \le 10^5$

Equation (34.24b) gives the shear stress relationship in pipe flow at a moderate Reynolds number, i.e . Unlike very high Reynolds number flow, here laminar effect cannot be neglected and the laminar sub layer brings about remarkable influence on the outer zones.

The friction factor for pipe flows, is valid for a specific range of Reynolds number and for a particular surface condition. f

Concept of Friction Factor in a pipe flow:

The friction factor in the case of a pipe flow was already mentioned in lecture 26. we will elaborate further on friction factor or friction coefficient in this section.

Skin friction coefficient for a fully developed flow through a closed duct is defined as

$$C_f = \frac{\tau_w}{(1/2)\rho V^2}$$

where, V is the average velocity of flow given by $V = Q/A$, Q and A are the volumeflow rate through the duct and the cross-sectional area of the duct respectively.

From a force balance of a typical fluid element (Fig. 35.1) in course of its flow through a duct of constant cross-sectional area, we can write

$$\tau_w = \frac{\Delta p^* A}{SL}$$

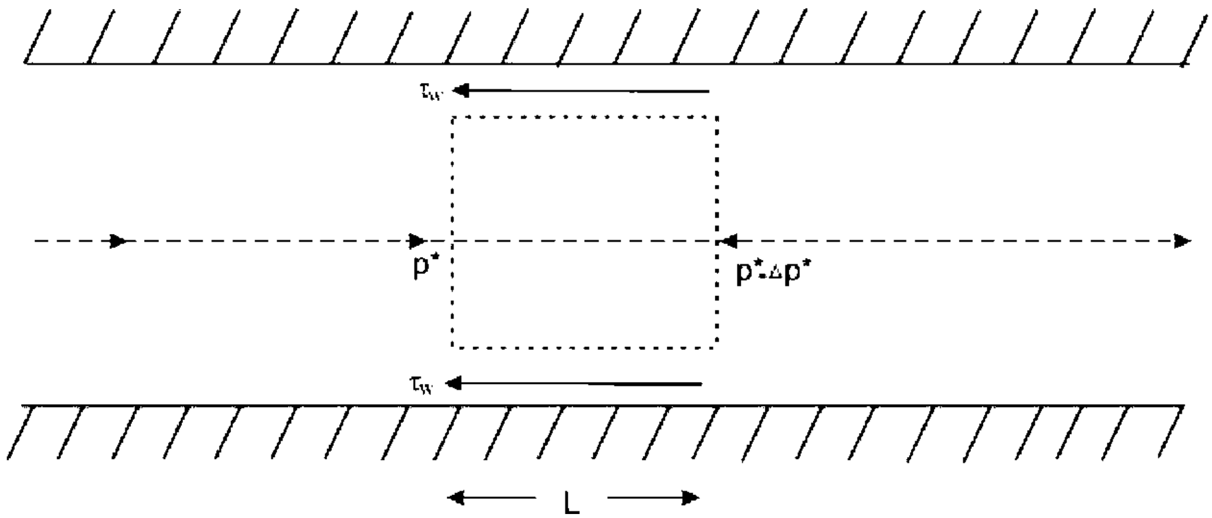


Fig: Force Balance of a fluid element in the course of flow through a duct

where, τ_w is the shear stress at the wall and Δp^* is the piezometric pressure drop over a length of L . A and S are respectively the cross-sectional area and wetted perimeter of the duct. Substituting we have,

$$C_f = \frac{\Delta p^* A}{SL(1/2)\rho V^2} = \frac{1}{4}\frac{D_h}{L}\frac{\Delta p^*}{(1/2)\rho V^2}$$

where, $D_h = 4A/S$ and is known as the hydraulic diameter .

In case of a circular pipe, D_h#D, the diameter of the pipe. The coefficient C_f defined by Eqs (35.1) or (35.3) is known as Fanning's friction factor .

To do away with the factor 1/4 in the Eq. (35.3), Darcy defined a friction factor f (Darcy's friction factor) as

$$f = \frac{D_h}{L}\frac{\Delta p^*}{(1/2)\rho V^2}$$

Comparison .

$$f = 4C_f$$

$$f = \frac{D_h}{L}\frac{\Delta p^*}{(1/2)\rho V^2}$$

written in a different fashion for its use in the solution of pipe flow problems in practice as

$$\Delta p^* = f \cdot \frac{L}{D_h} \cdot \frac{\rho}{2} V^2$$

or in terms of head loss (energy loss per unit weight)

$$h_f = \frac{\Delta p^*}{\rho g} = \frac{fLV^2}{2gD_h}$$

where, h_f represents the loss of head due to friction over the length L of the pipe.

In order to evaluate h_f, we require to know the value of f. The value of f can be determined from Moody's Chart.

Variation of Friction Factor

In case of a laminar fully developed flow through pipes, the friction factor, f is found from the exact solution of the Navier-Stokes equation as discussed in lecture 26. It is given by

$$f = \frac{64}{\text{Re}}$$

In the case of a turbulent flow, friction factor depends on both the Reynolds number and the roughness of pipe surface.

Sir Thomas E. Stanton (1865-1931) first started conducting experiments on a number of pipes of various diameters and materials and with various fluids.Afterwards, a German engineer Nikuradse carried out experiments on flows through pipes in a very wide range of Reynolds number.

A comprehensive documentation of the experimental and theoretical investigations on the laws of friction in pipe flows has been presented in the form of a diagram, as shown in Fig. 35.2, by L.F. Moody to show the variation of friction factor, f with the pertinent governing parameters, namely, the Reynolds number of flow and the relative roughness ε/D of the pipe. This diagram is known as Moody's Chart which is employed till today as the best means for predicting the values of f.depicts that

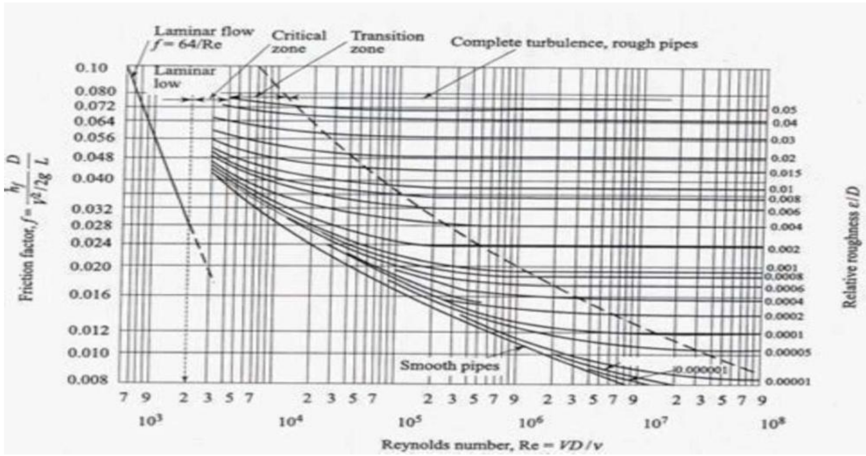


Fig: Friction Factors for pipes (adapted from Trans. ASME, 66,672, 1944)

The friction factor f at a given Reynolds number, in the turbulent region, depends on the relative roughness, defined as the ratio of average roughness to the diameter of the pipe, rather than the absolute roughness.

For moderate degree of roughness, a pipe acts as a smooth pipe up to a value of Re where the curve of f vs Re for the pipe coincides with that of a smooth pipe. This zone is known as the smooth zone of flow .

The region where f vs Re curves become horizontal showing that f is independent of Re, is known as the rough zone and the intermediate region between the smooth and rough zone is known as the transition zone.

The position and extent of all these zones depend on the relative roughness of the pipe. In the smooth zone of flow, the laminar sublayer becomes thick, and hence, it covers appreciably the irregular surface protrusions. Therefore all the curves for smooth flow coincide.

with increasing Reynolds number, the thickness of sublayer decreases and hence the surface bumps protrude through it. The higher is the roughness of the pipe, the lower is the value of Re at which the curve of f vs Re branches off from smooth pipe curve

In the rough zone of flow, the flow resistance is mainly due to the form drag of those protrusions. The pressure drop in this region is approximately proportional to the square of the average velocity of flow. Thus f becomes independent of Re in this region.

In practice, there are three distinct classes of problems relating to flow through a single pipe line as follows:

The flow rate and pipe diameter are given. one has to determine the loss of head over a given length of pipe and the corresponding power required to maintain the flow over that length.

The loss of head over a given length of a pipe of known diameter is given. one has to find out the flow rate and the transmission of power accordingly.

The flow rate through a pipe and the corresponding loss of head over a part of its length are given. one has to find out the diameter of the pipe.

In the first category of problems, the friction factor f is found out explicitly from the given values of flow rate and pipe diameter. Therefore,

the loss of head h_f and the power required, P can be calculated by the straightforward application of.

UNIT V
Turbo Machinery

Introduction and Working principle of hydraulic turbines

Hydraulic turbines: are the machines which converts the hydraulic energy of water into mechanical energy. Therefore, these may be considered as hydraulic motors or prime movers.

Pump: it converts mechanical energy into hydraulic energy. The mechanical energy developed by the turbine is used in running an electric generator which is directly coupled to the shaft of the turbine. The electric generator thus generates electric power which is known as hydroelectric power.

Electric Motor: Electric motor coverts electrical energy to mechanical energy.

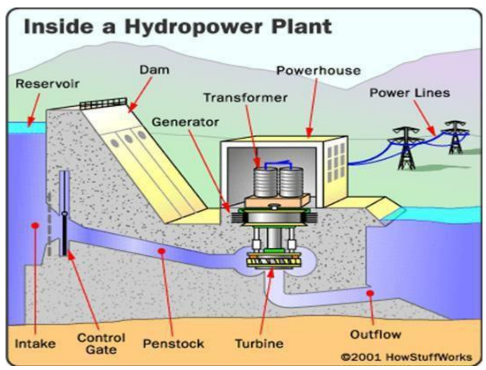
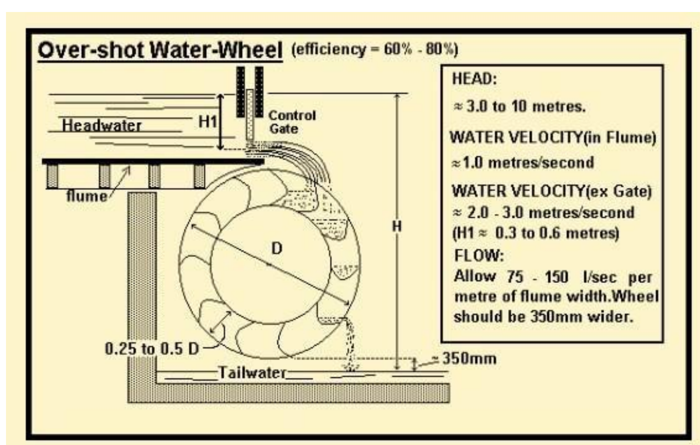
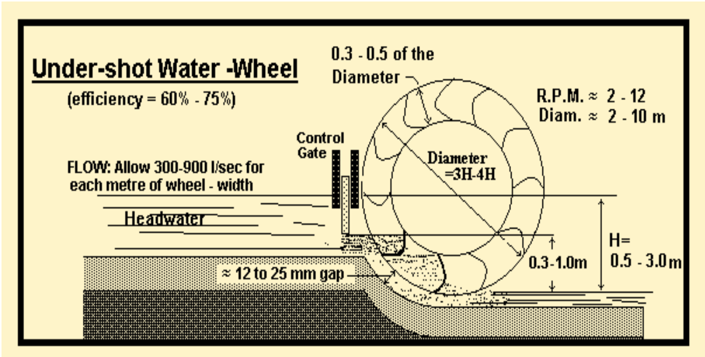


Fig: Electric Motor

DEVELOPMENT OF TURBINES





In the early days of water, pump development water wheels made of wood are widely used which uses either (falling water) potential energy or kinetic energy of the flowing stream of water. The wheel consists of series of straight vanes on its periphery, water was permitted to enter at the top and imbalance created by the weight of the water causes wheel to rotate (over shot wheel uses potential energy, under short wheel uses kinetic energy). Since, the low efficiency and low power generation and these could not be directly coupled to modern fast electric generators for the purpose of power generation. Therefore, the water wheels are completely replaced by modern hydraulic turbines, which will run at any head and desired speed enabling the generator to be coupled directly.

In general turbine consists of wheel called runner or rotor having a number of specially developed vanes or blades or buckets. The water possessing large amount of hydro energy when strikes the runner, it does the work on runner and causes it to rotate.

Classification of Hydraulic Turbines

1. According to the type of energy at the inlet
2. According to the direction of flow through runner
3. According to head at inlet
4. According to specific speed of turbine
5. According to Position of the shaft

1. According to the type of energy at the inlet

a) *Impulse turbine:*

All the available energy of the water is converted into kinetic energy by passing it through a contracting nozzle provided at the end of penstock

Ex: Pelton wheel turbine, Turgo-impulse turbine, Girard turbine, Bank turbine, Jonval turbine etc.

b) Reaction Turbine:

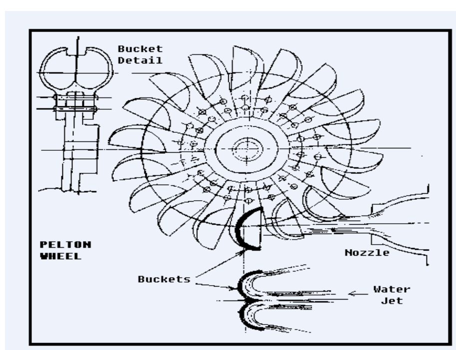
- At the entrance of the runner, only a part of the available energy of water is converted into kinetic energy and a substantial part remains in the form of pressure energy.
- As the water flow through the turbine pressure energy converts into kinetic energy gradually. Therefore the pressure at inlet of runner is higher than the pressure at outlet and it varies through out the passage of the turbine.
- For this gradual change of pressure to the possible the runner must be completely enclosed in a air-tight casing and the passage is entirely full of water throughout the operation of turbine
- The difference of pressure between the inlet and outlet of the runner is called reaction
- pressure and hence the turbines are known as reaction turbines.
- Ex: Francis turbine, Kaplan turbine, Thomson Turbine, Fourneyron turbine, Propeller turbine, etc

2. <u>According to the direction of flow through runner:</u>

a) Tangential flow turbine b) Radial flow turbine
c) Axial flow turbine d) Mixed flow turbine

Tangential flow turbine:

The water flows along the tangent to the path of rotation of the runner Ex: Pelton wheel turbine



Radial flow Turbine

- The water flows in the radial direction through the runner.
- **Inward radial flow turbine:** The water enters the outer circumference and flows radially inwards towards the centre of the runner.
- Ex: Old Francis turbine, Thomson turbine, Girard turbine etc

- **Outward radial flow turbine**: The water enters at the centre and flows radially outwards towards the outer periphery of the runner.
- Ex: Fourneyron turbine.

a) Axial flow turbine**:**

The water flow through runner wholly and mainly along the direction parallel to the axis of rotation of the runner.

Ex: Kaplan turbine, Jonval, Girard axial flow turbine, Propeller turbine, etc

b) Mixed flow turbines

The water enters the runner at the outer periphery in the radial direction and leaves it at the centre of the axial direction parallel to the rotation of the runner.

Ex: Modern Francis turbine.

a) High head turbines: These turbines work under very high heads 255m - 1770m and above. Requires relatively less quantity of water.
Ex: Pelton wheel turbine or impulse turbine.
b) Medium head turbines: These turbines are capable of working under medium heads ranging from 60m - 250m These turbines requires large quantity of water.
Ex: Francis Turbine
c) Low head turbines: these turbines are capable of working under the heads less than 60mts. These turbines requires large quantity of water.
Ex: Kaplan turbine, propeller turbine.

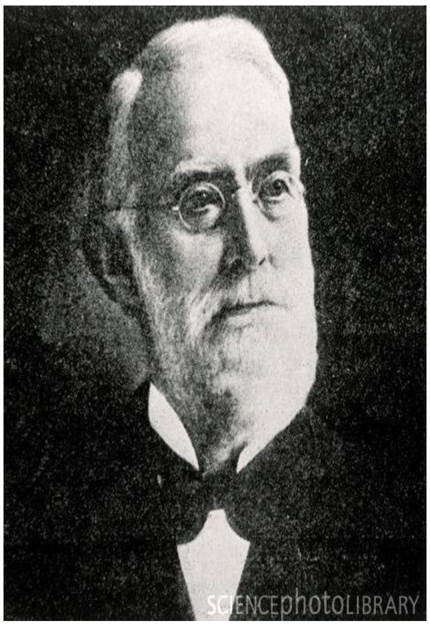
a) Low specific speed turbines: specific speed turbine varies from 8.5 to 30.
Ex: Pelton wheel turbine
b) Medium specific speed turbines: specific speed varies from 50 to 340 Ex: Francis turbine.
c) High specific speed turbines: specific speed varies from 255-860. Ex: Kaplan and propeller turbine.

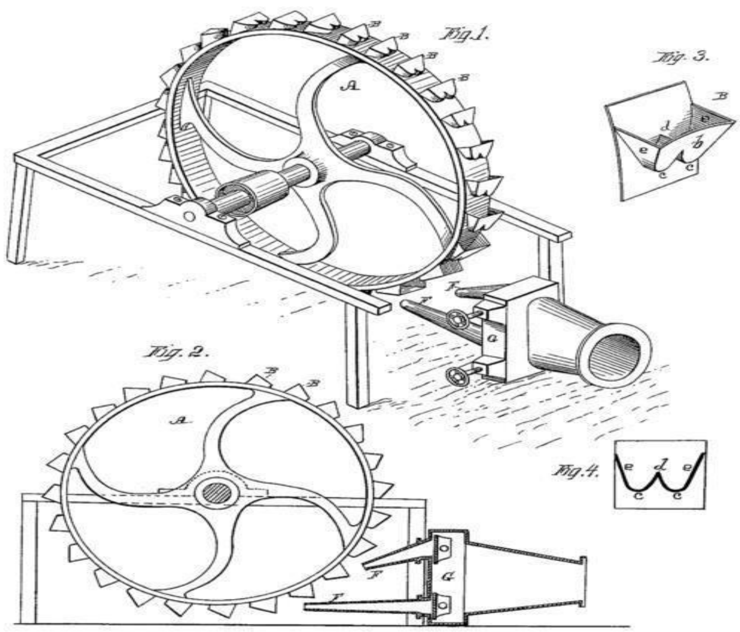
According to the position of the shaft:

a) Horizontal disposition of shaft

b) Vertical disposition of shaft. Turbine Shaft

PELTON WHEEL TURBINE





- This is named after Lester A. Pelton, American engineer who contributed much to its development in about 1880. It is well suited for operating under high heads.
- It’s an impulse, high head, low specific speed and tangential flow turbine.
- The runner consists of a circular disc with a number of buckets evenly spaced around its periphery.

- The buckets have a shape of double semi-ellipsoidal cups. Each bucket is divided into 2 symmetrical parts by sharp edged ridge known as splitter.
- One or more nozzles are mounted so that each directs a jet along a tangential to the pitch circle of runner or axis of blades.
- The jet of water impinges on the splitter, which divides jet into equal halves, each of which after flowing around the smooth inner surface of the bucket leaves at its outer edge.
- The buckets are so shaped that the angle at the outlet lip varies from 10 to 20 degrees. So that the jet of outer deflects through 160 to 170. The advantage of having double cup-shaped bucket is that

the axial thrust neutralizes each other being equal and opposite and having bearing supporting the wheel shaft are not supported to any axial thrust or end thrust.

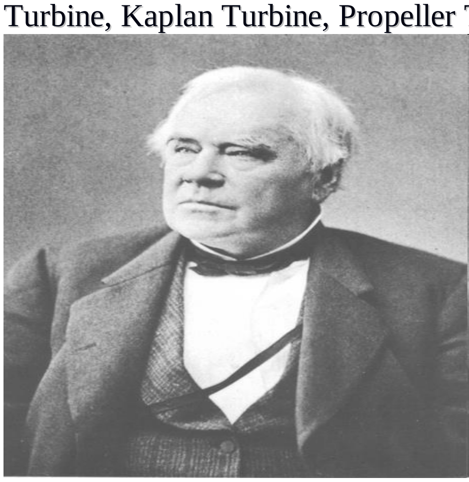
- The back of the bucket is shaped that as it swings downward into the jet no water is wasted by splashing.
- At the lower tips of the bucket a notch is cut which prevents the jet striking the preceding bucket and also avoids the deflection of water towards the centre of the wheel.
- For low heads buckets are made of C.I, for high heads buckets are made of Cast Steel ,bronze, stainless steel.

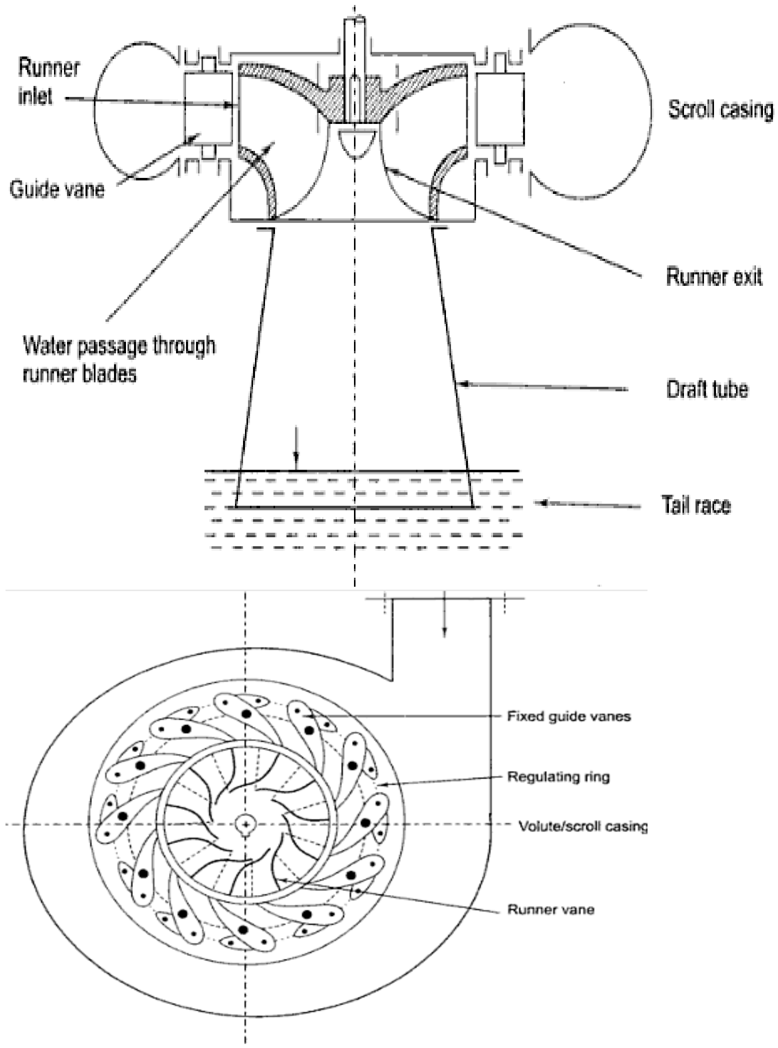
- In order to control the quantity of water striking the runner, the nozzle is fitted at the end of the penstock is provided with a spear valve having streamlined head which is fixed at the end of the rod.
- When the shaft of pelton wheel is horizontal, not more than to two jets are used if the shaft vertical six number of jets are possible.

- A casing is made of C.I or fabricated steel plates is usually provided for a pelton wheel to prevent splashing of water, to lead water to the tail race and also act as safeguard against accidents.
- Large pelton wheels are usually equipped with a small break nozzle which when opened directs a jet of water on the back of the buckets, therby bringing the wheel quickly to rest after it is shut down, otherwise it takes considerable time to come to rest.

Reaction Turbines:

- In reaction turbines, the available energy of water at inlet of the turbine is sum of pressure energy and kinetic energy and during the flow of water through the runner a part of pressure energy is converted into kinetic energy, such type of turbine is reaction turbine. Ex: Francis Turbine, Kaplan Turbine, Propeller Turbine, etc





Sectional view of Francis Turbine

The main components of Francis Turbine: Scroll Casing:

The water from the penstock enters the scroll casing or spiral casing which completely surrounds the runner. The purpose of casing is to provide even distribution of water around the circumference of the runner and to maintain constant velocity of water so distributed.

In order to maintain constant velocity of water through out its path around the runner, the cross-sectional area of casing is gradually decreased. The casing is made of cast steel or plate steel.

2. Stay Ring:

From the scroll casing the water passes through a speed ring or stray ring. Stay ring consists of outer and lower ring held together by series of fixed vanes called stay vanes.

Number of stay vanes usually half of the number of guide vanes. Stay vane performs two functions, one is to direct the water from the scroll casing to the guide vanes and other is to rest the load imposed upon it by the internal pressure of water and the weight of the turbine and electrical generator and transmits the same to the foundation. Speed ring is made of C.I or C.S.

2. Guide Vanes:

From the stay ring water passes through a series of guide vanes provided around the periphery of the runner. The function of guide vanes is to regulate the quantity of water supplied to the runner and to direct the water on to the runner with design angle.

The guide vanes are airfoil shaped and made of C.S or S.S or P.S. Each guide vane is provided with two stems; the upper stem passes through head cover and lower stem seats in bottom ring. By a system of levers and links all the guide vanes may be turned about their stems, so as to alter the width of the passage between the adjacent guide vanes, thereby allowing a variable quantity of water to strike the runner. The guide vanes are operated either by means of a wheel or automatically by a governor.

2. RUNNER:

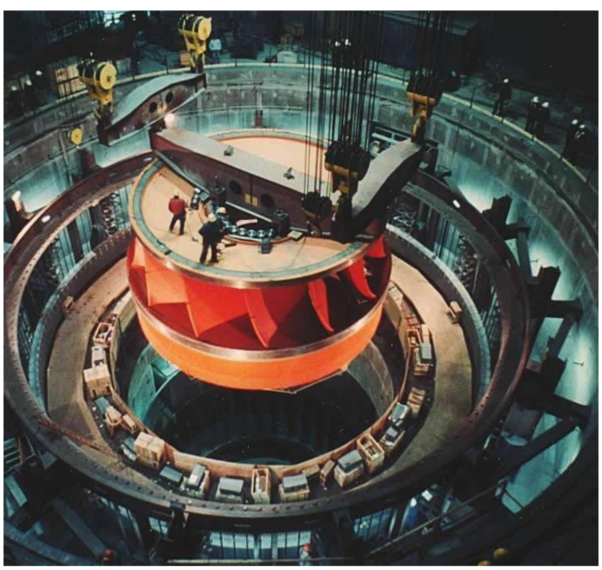
The runner of a Francis turbine consists of a series of a curved vanes (from 16 to 24) evenly arranged around the circumference in the annular space between two plates.

The vanes are so shaped that water enters the runner radially at the outer periphery and leaves it axially at the inner periphery.

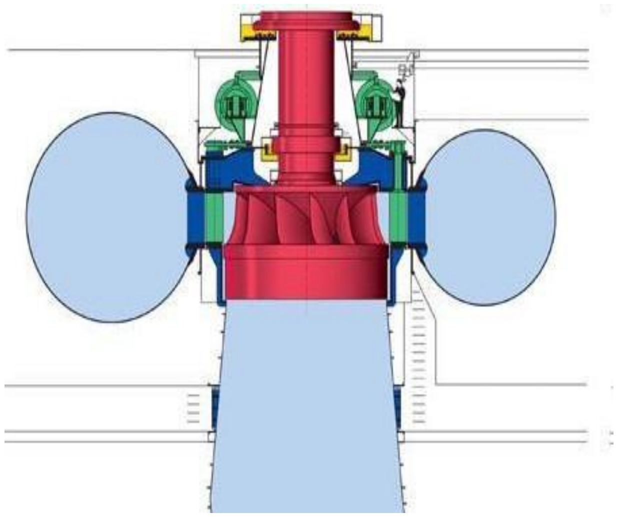
The change in the direction of flow of water from radial to axial, as it passes through the runner, produces a circumferential force on the runner which makes the runner to rotate and thus contributes to the useful output of the runner.

Runner vanes are made of SS and other parts are made of CI or CSThe runner is keyed to a shaft which is usually of forged steel. The torque produced by the runner is transmitted to the generator through the shaft which is usually connected to the generator shaft by a bolted flange connection.

Francis turbine installation:



KAPLAN TURBINE



- **Kaplan** turbine is developed by the Austrian Engineer Viktor Kaplan, it is suitable for low heads and requires large quantity of water to develop large amount of power. Since it is a reaction turbine, it operates in an entirely closed conduit from head race to tail race.

The main components of a Kaplan turbine

Scroll Casing

Guide vanes Mechanism

Hub with vanes or runner of turbine, and

Draft Tube

The function of above components is same as that of Francis turbine

The water from the penstock enters the scroll casing and then moves to the guide vanes. From the guide vanes, the water turns through 900 and flows axially through the runner.

The runner of a Kaplan turbine has four or six blades (eight in exceptional cases). The blades attached to a hub are so shaped that water flows axially through the runner.

The adjustment of the runner blades is usually carried out automatically by means of a servomotor operating inside the hollow coupling of turbine and generator shaft.

When both guide vane angle and runner blade angle may varied, a high efficiency can be maintained. Even at part load, when a lower discharge is flowing through the runner, a high efficiency can be attained in case of Kaplan turbine.

Simultaneously the guide vane and runner vane angles are adjusted the water under all the working conditions flows through the runner blades without shock. as such the eddy losses which inevitable in Francis turbine and propeller turbines are almost completely eliminated in a Kaplan turbine.