MCE2121

ADVANCED FLUID MECHANICS

LECTURE NOTES

Module-I

Prepared By

Dr. Prakash Chandra Swain

Professor in Civil Engineering

Veer Surendra Sai University of Technology, Burla

Branch - Civil Engineering
Specialization-Water Resources Engineering
Semester – 1st Sem

Department Of Civil Engineering

VSSUT, Burla

Disclaimer

This document does not claim any originality and cannot be used as a substitute for prescribed textbooks. The information presented here is merely a collection by Prof. P.C.Swain with the inputs of Post Graduate students for their respective teaching assignments as an additional tool for the teaching-learning process. Various sources as mentioned at the reference of the document as well as freely available materials from internet were consulted for preparing this document. Further, this document is not intended to be used for commercial purpose and the authors are not accountable for any issues, legal or otherwise, arising out of use of this document. The authors make no representations or warranties with respect to the accuracy or completeness of the contents of this document and specifically disclaim any implied warranties of merchantability or fitness for a particular purpose.

Course Content

Module I

Introduction: Survey of Fluid Mechanics, Structure of Fluid Mechanics Based on Rheological, Temporal Variation, Fluid Type, Motion Characteristic and spatial Dimensionality Consideration, Approaches in Solving Fluid Flow Problems, Fundamental idealizations and Descriptions of Fluid Motion, Quantitative Definition of Fluid and Flow, Reynolds Transport Theorem, Mass, Momentum and Energy Conservation Principles for Fluid Flow.

Potential Flow: Frictionless Irrotational Motions, 2 - Dimensional Stream Function and Velocity Potential Function in Cartesian and Cylindrical Polar Coordinate Systems, Standard Patterns of Flow, Source, Sink, Uniform Flow and irrotational vortex, Combinations of Flow Patterns, method of Images in Solving Groundwater Flow problems, Method of Conformal transformations.

MODULE-I

Lecture Note 1

INTRODUCTION

Fluid is a substance that continually deforms (flows) under an applied shear stress. Fluids are a subset of the phases of matter and include liquids, gases, plasmas and, to some extent, plastic solids. Fluids are substances that have zero shear modulus or, in simpler terms, a fluid is a substance which cannot resist any shear force applied to it.

Fluid mechanics is the branch of science which deals with the behavior of fluids(liquids or gases)at rest as well as in motion. It deals with the static, kinematics and dynamic aspects of fluids.

The study of fluids at rest is called fluid statics. The study of fluid in motion is called fluid kinematics if pressure forces are not considered if pressure force is considered in fluid in motion is called fluid dynamics.

PROPERTIES OF FLUIDS

DENSITY or MASS DENSITY:-

The density, or more precisely, the volumetric mass density, of a substance is its mass per unit volume. The symbol most often used for density is $\boldsymbol{\rho}$(Greek letter rho), Mathematically, density is defined as mass divided by volume

$$\rho=\frac{m}{V}$$

whereρ is the density, m is the mass, and V is the volume. In some cases (for instance, in the United States oil and gas industry), density is loosely defined as its weight per unit volume.

SPECIFIC WEIGHT or WEIGHT DENSITY:-

The specific weight (also known as the unit weight) is the weightper unit volume of a material. It is denoted as the symbol w. Mathematically,

$$w = \frac{\text{weight of fluid}}{\text{Volume of fluid}}$$

$$= \frac{\text{mass of fluid} * \text{acceleration due to gravity}}{\text{Volume of fluid}}$$

$$= \frac{m}{V} * g$$

$$= \rho g$$

SPECIFIC GRAVITY:-

It is defined as the ratio of the weight density of a fluid to weight density of a standard fluid. For liquids the standard fluid is taken as water and for gases the standard fluid is given as air. It is denoted as S.

S(for liquids)= $\frac{\text{Weight density of liquid}}{\text{weight density of water}}$

S(for gases)= $\frac{\text{Weight density of gases}}{\text{weight density of air}}$

COMPRESSIBILITY:-

Compressibility is a measure of the relative volume change of a fluid or solidas a response to a pressure (or mean stress) change.

$$\beta = -\frac{dv}{vdp}$$

where V is volume and p is pressure.

ELASTICITY:-

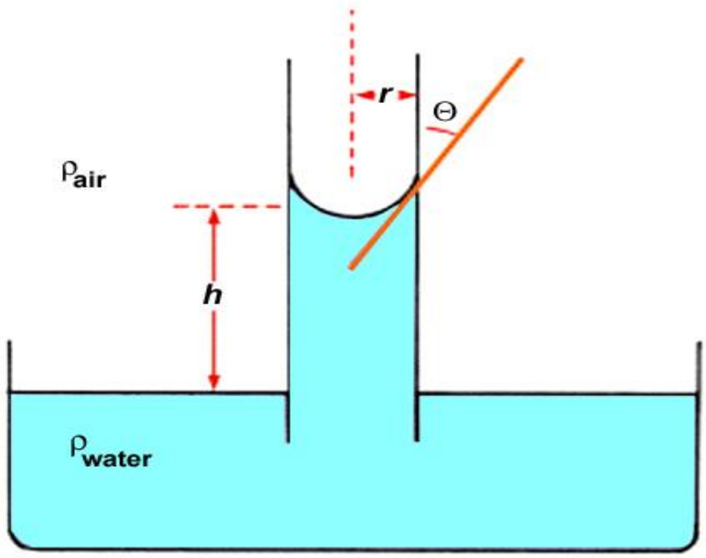
Elasticity is the ability of a body to resist a distorting influence or stress and to return to its original size and shape when the stress is removed. Solid objects will deform when forces are applied on them. If the material is elastic, the object will return to its initial shape and size when these forces are removed.

SURFACE TENSION:-

It is defined as the tensile force acting on the surface of a liquid in contact with a gas or on the surface between two immiscible liquids such that the contact surface behaves like a membrane under tension. The magnitude of this force per unit length of the free surface will have the same value as the surface energy per unit area. It is denoted as σ (called sigma).Unit is N/m.

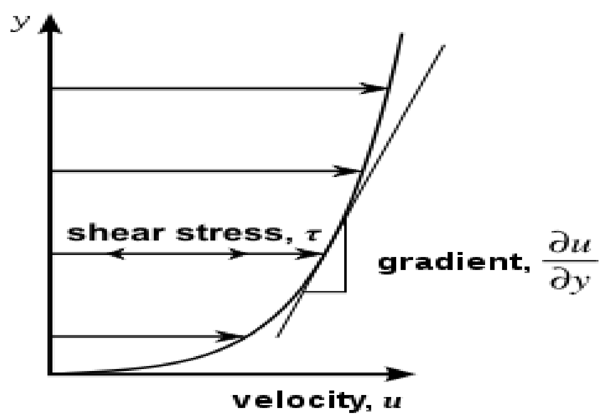
CAPILLARITY:-

Capillarity is defined as a phenomenon of rise or fall of a liquid surface in small tube relative to the adjacent general level of liquid when the tube is held vertically in the liquid. The rise of liquid surface is known as capillary rise while the fall of liquid surface is known as capillary depression. It is expressed in terms of cm or mm of liquid. Its value depends upon the specific weight of the liquid, diameter of the tube and surface tension of the liquid.



VISCOSITY:-

It is defined as the property of a fluid which offers resistance to the movement of one layer of fluid over another adjacent layer of the fluid. When two layers of a fluid, a distance 'dy' apart , move one over the other at different velocities, say u and u+du.

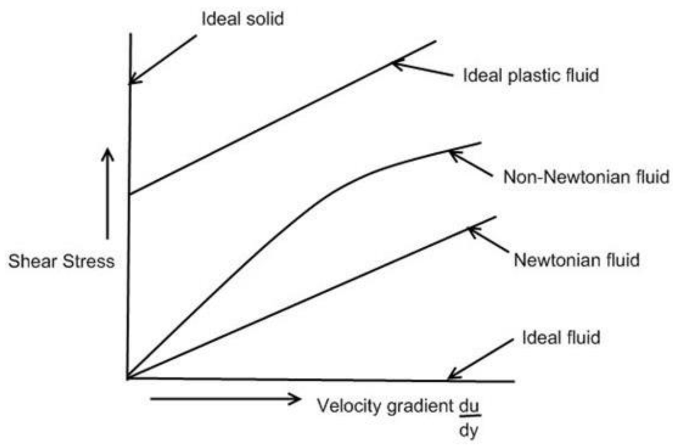


Newton's law of Viscosity:-

It states that the shear stress on a fluid element layer is directly proportional to the rate of shear strain. The constant of proportionality is called the coefficient of viscosity. Mathematically,

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

TYPES OF FLUIDS:-



Types of Fluid

Basically the fluids are classified into 5 types and these are

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

1. Ideal Fluid:

A fluid which can not be compressed and have no viscosity falls in the category of ideal fluid. Ideal fluid is not found in actual practice but it is an imaginary fluid because all the fluid that exist in the environment have some viscosity. therein no ideal fluid in reality.

2. Real Fluid:

A fluid which has at least some viscosity is called real fluid. Actually all the fluids existing or present in the environment are called real fluids. for example water.

3. Newtonian Fluid:

If a real fluid obeys the Newton's law of viscosity (i.e the shear stress is directly proportional to the shear strain) then it is known as the Newtonian fluid.

4. Non-Newtonian Fluid:

If real fluid does not obeys the Newton's law of viscosity then it is called Non-Newtonian fluid.

5. Ideal Plastic Fluid:

A fluid having the value of shear stress more than the yield value and shear stress is proportional to the shear strain (velocity gradient) is known as ideal plastic fluid.

MANOMETER:-

A manometer is an instrument that uses a column of liquid to measure pressure, although the term is currently often used to mean any pressure instrument. Two types of manometer.

1.Simple manometer

2.Differential manometer

The U type manometer, which is considered as a primary pressure standard, derives pressure utilizing the following equation:

$P = P2 - P1 = h\omega\ \rho g$

Where:

P = Differential pressure

$P1$ = Pressure applied to the low pressure connection

$P2$ = Pressure applied to the high pressure connection

$h\omega$ = is the height differential of the liquid columns between the two legs of the manometer

ρ = mass density of the fluid within the columns

g = acceleration of gravity

Simple manometer:-

A simple manometer consists of a glass tube having one of its ends connected to a point where pressure is to be measured and other end remains open to atmosphere. Common types of simple manometers are:

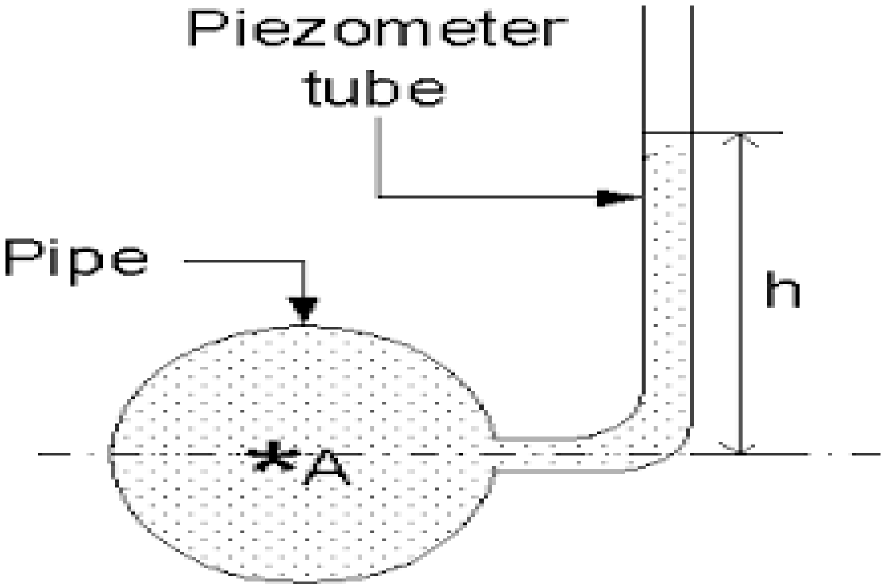
1.Piezometer

2.U tube manometer

3.Single Column manometer

PIEZOMETER

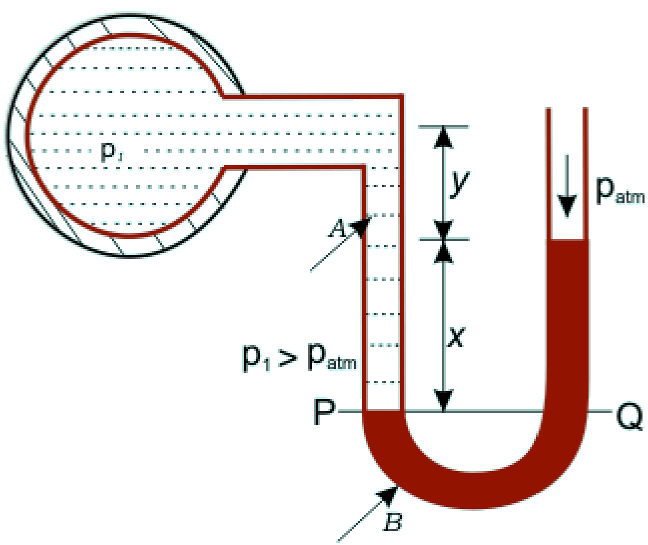
A piezometer is either a device used to measure liquid pressure in a system by measuring the height to which a column of the liquid rises against gravity, or a device which measures the pressure (more precisely, the piezometric head) of groundwaterat a specific point. A piezometer is designed to measure static pressures, and thus differs from a pitot tube by not being pointed into the fluid flow.



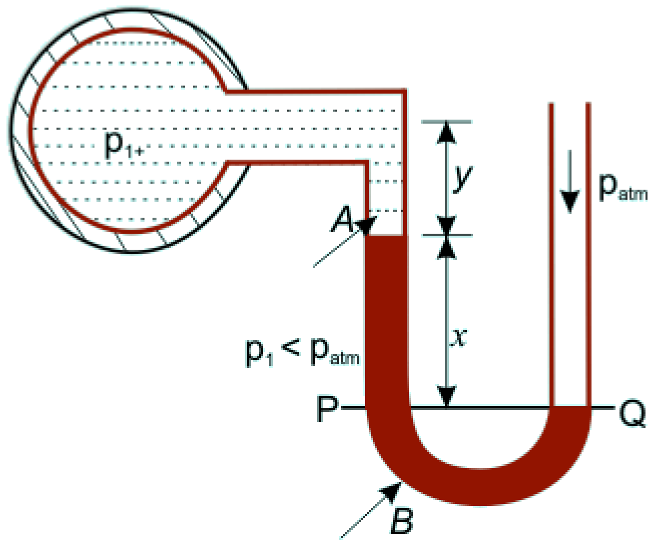
U TUBE MANOMETER-

Manometers are devices in which columns of a suitable liquid are used to measure the difference in pressure between two points or between a certain point and the atmosphere.

Manometer is needed for measuring large gauge pressures. It is basically the modified form of the piezometric tube. A common type manometer is like a transparent "U-tube" as shown in Fig.



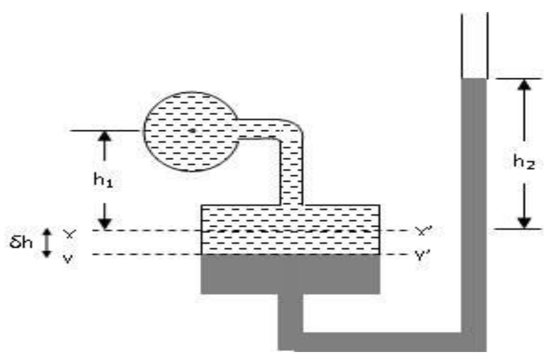
simple manometer to measure gauge pressure



simple manometer to measure vacuum pressure

SINGLE COLUMN MANOMETER:-

It is a modified form of a U tube manometer in which a reservoir having a large cross sectional area.



DIFFERENTIAL MANOMETER:-

Differential Manometers are devices used for measuring the difference of pressure between two points in a pipe or in two different pipes . A differential manometer consists of a U-tube, containing a heavy liquid, whose two ends are connected to the points, which difference of pressure is to be measure.

Most commonly types of differential manometers are:

1- U-tube differential manometer.

2- Inverted U-tube differential manometer

U-tube differential manometer.

For two pipes are at same levels:-

P_A-P_B=h.g(p_g-p_1)

Where:

P_1=density of liquid at A=density of liquid at B.

For two pipes are in different level:-

P_A- P_B= h.g (p_g-p_1) + $p_2$gy- p_1 gx

Where:

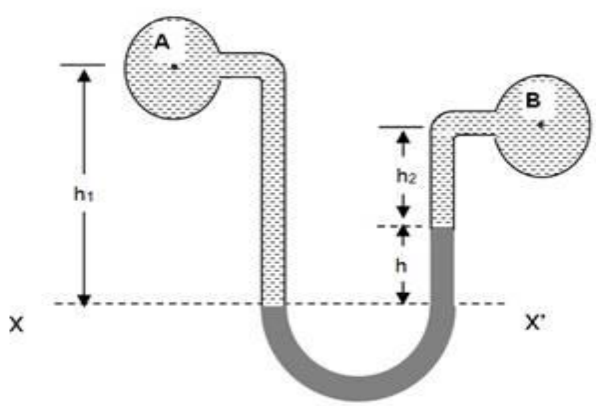
h = difference in mercury level in the U-tube

y = distance of the centre of B, from the mercury level in the right limb.

x = distance of the centre of A, from the mercury level in the left limb. P_1=density of liquid at A.

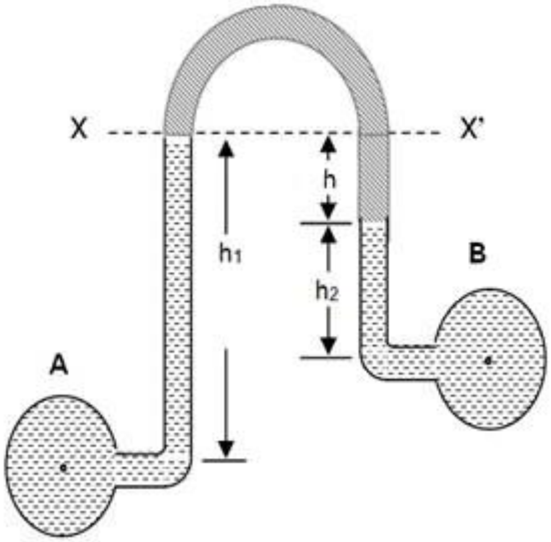
p_2=density of liquid at B.

p_g=density of mercury (heavy liquid)



Inverted U-tube differential manometer

It consists of inverted U-tube, containing a light liquid. The two ends of the tube are connected to the points whose difference of pressure is to be measured. It is used for measuring differences of low pressures.



$P_A - P_B = \rho_1 * g * h_1 - \rho_2 * g * h_2 - \rho_s * g * h$

Where;

h_1 = height of liquid in left limb below the datum line

h_2 = height of liquid in right limb

h=difference of light liquid

ρ_1 =density of liquid at A

ρ_2 =density of liquid at B

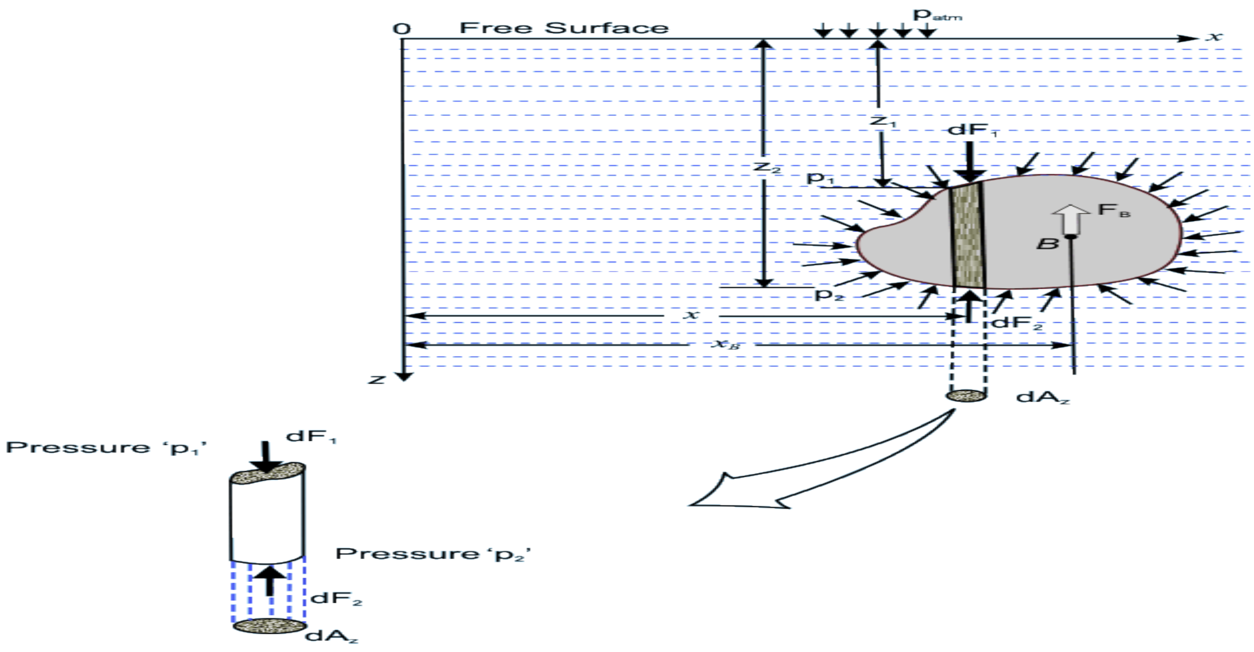
ρ_s =density of light liquid

P_A =pressure at A

P_B=Pressure at B

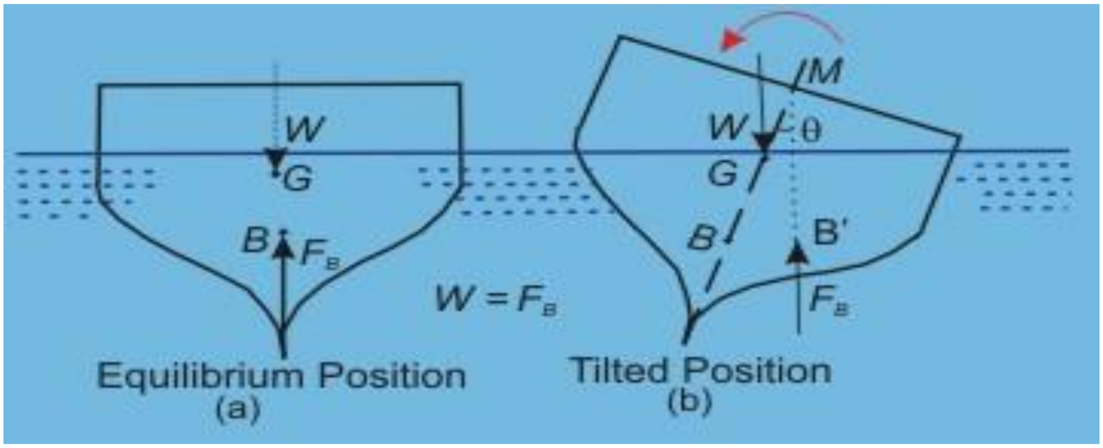
Buoyancy

- When a body is either wholly or partially immersed in a fluid, a lift is generated due to the net vertical component of hydrostatic pressure forces experienced by the body.
- This lift is called the buoyant force and the phenomenon is called buoyancy
- Consider a solid body of arbitrary shape completely submerged in a homogeneous liquid as shown in Fig. Hydrostatic pressure forces act on the entire surface of the body.



Stability of Floating Bodies in Fluid

- When the body undergoes an angular displacement about a horizontal axis, the shape of the immersed volume changes and so the centre of buoyancy moves relative to the body.
- As a result of above observation stable equilibrium can be achieved, under certain condition, even when G is above B. Fig illustrates a floating body -a boat, for example, in its equilibrium position.



Important points to note here are

a. The force of buoyancy F_B is equal to the weight of the body W

b. Centre of gravity G is above the centre of buoyancy in the same vertical line.

c. Figure b shows the situation after the body has undergone a small angular displacement q with respect to the vertical axis.

d. The centre of gravity G remains unchanged relative to the body (This is not always true for ships where some of the cargo may shift during an angular displacement).

e. During the movement, the volume immersed on the right hand side increases while that on the left hand side decreases. Therefore the centre of buoyancy moves towards the right to its new position B'.

Let the new line of action of the buoyant force (which is always vertical) through B' intersects the axis BG (the old vertical line containing the centre of gravity G and the old centre of buoyancy B) at M. For small values of q the point Impractically constant in position and is known as metacentre. For the body shown in Fig. M is above G, and the couple acting on the body in its displaced position is a restoring couple which tends to turn the body to its original position. If M were below G, the couple would be an overturning couple and the original equilibrium would have been unstable. When M coincides with G, the body will assume its new position without any further movement and thus will be in neutral equilibrium. Therefore, for a floating body, the stability is determined not simply by the relative position of B and G,ratherby the relat**ive** position of M and G. The distance of metacentre above G along the line BG is known as metacentric height GM which can be written as

$$\underline{GM = BM - BG}$$

Hence the condition of stable equilibrium for a floating bodycan be expressed in terms ofmetacentric heightas follows:

GM > 0 (M is above G)	**Stable equilibrium**
GM = 0 (M coinciding with G)	**Neutral equilibrium**
GM < 0 (M is below G)	**Unstable equilibrium**

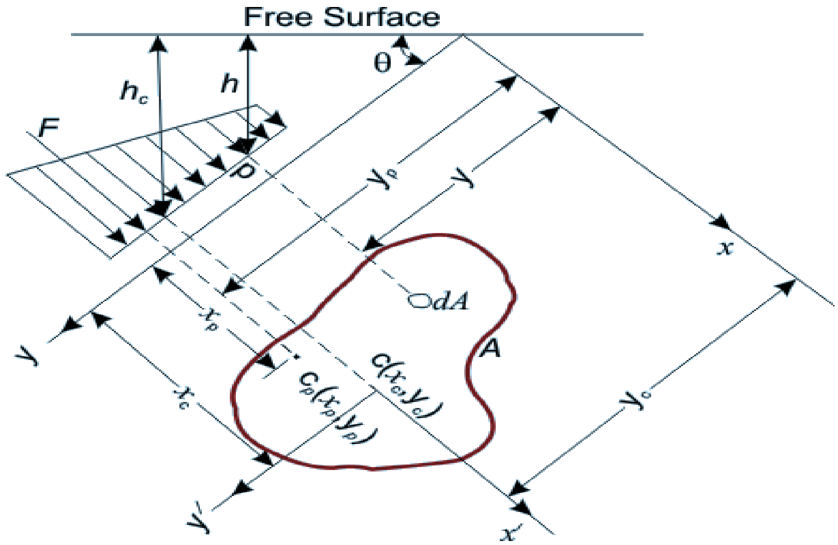
The angular displacement of a boat or ship about its longitudinal axis is known as 'rolling' while that about its transverse axis is known as "pitching".

Hydrostatic Thrusts on Submerged Plane Surface

Due to the existence of hydrostatic pressure in a fluid mass, a normal force is exerted on any part of a solid surface which is in contact with a fluid. The individual forces distributed over an area give rise to a resultant force.

Plane Surfaces

Consider a plane surface of arbitrary shape wholly submerged in a liquid so that the plane of the surface makes an angle θ with the free surface of the liquid. We will assume the case where the surface shown in the figure below is subjected to hydrostatic pressure on one side and atmospheric pressure on the other side.



Let p denotes the gauge pressure on an elemental area dA. The resultant force F on the area A is therefore

$$F = \iint_A p\, dA$$

$$F = \rho g \iint h dA = \rho g \sin\theta \iint y dA$$

Where h is the vertical depth of the elemental area dA from the free surface and the distance y is measured from the x-axis, the line of intersection between the extension of the inclined plane and the free surface (Fig.). The ordinate of the centre of area of the plane surface A is defined as

$$y_c = \frac{1}{A}\iint y dA$$

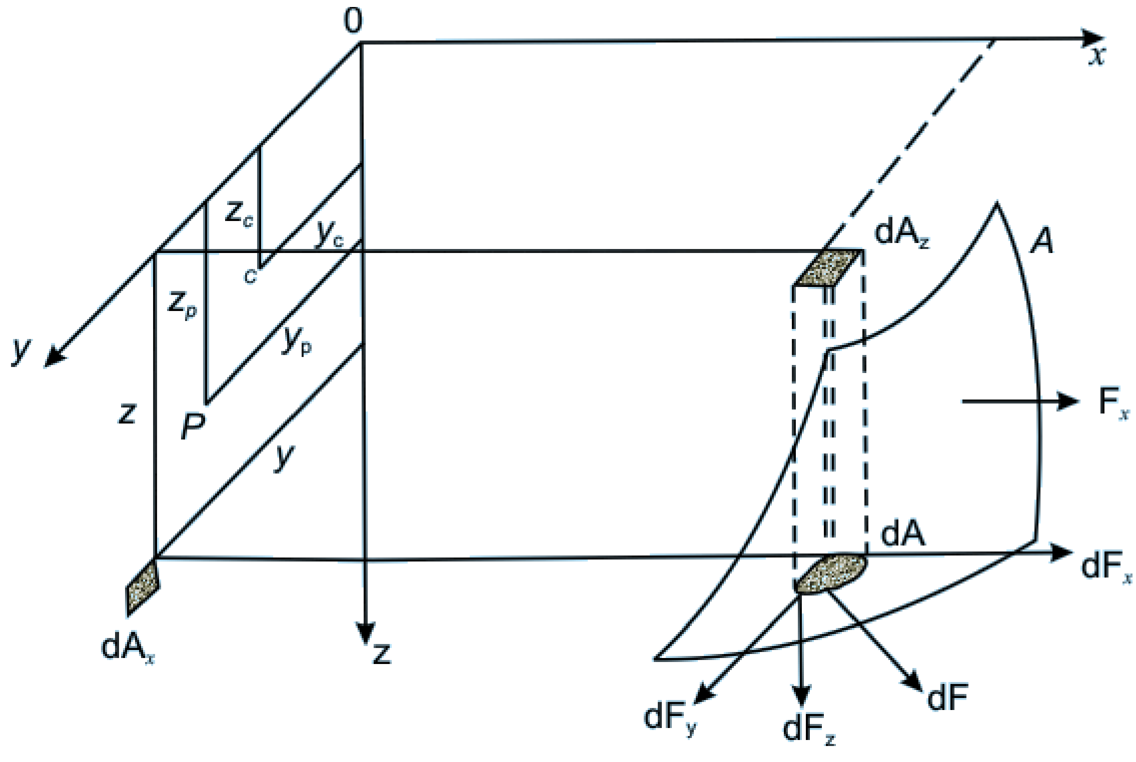
we get

$$F = \rho\, g\, y_c \sin\theta\, A = \rho\, g\, h_c\, A$$

where $h_c (= y_c \sin\theta)$ is the vertical depth (from free surface) of centre c of area .

Hydrostatic Thrusts on Submerged Curved Surfaces

On a curved surface, the direction of the normal changes from point to point, and hence the pressure forces on individual elemental surfaces differ in their directions. Therefore, a scalar summation of them cannot be made. Instead, the resultant thrusts in certain directions are to be determined and these forces may then be combined vectorially. An arbitrary submerged curved surface is shown in Fig. A rectangular Cartesian coordinate system is introduced whose xy plane coincides with the free surface of the liquid and z-axis is directed downward below the x - y plane.



Consider an elemental area dA at a depth z from the surface of the liquid. The hydrostatic force on the elemental area dA is

$$d\,F = \rho\, g\, z\, dA$$

and the force acts in a direction normal to the area dA. The components of the force dF in x, y and z directions are

$$dF_x = ldF = l\rho gzdA$$

$$dF_y = mdF = m\rho gzdA$$

$$dF_z = ndF = n\rho gzdA$$

Where *l*, m and n are the direction cosines of the normal to dA. The components of the surface element dA projected on yz, xz and xy planes are, respectively

$$dA_x = l\,dA$$

$$dA_y = mdA$$

$$dA_z = n\,dA$$

From equations,

$$dF_x = \rho g\, z\, dA_x$$

$$dF_y = \rho g\, z\, dA_y$$

$$dF_z = \rho g\, z\, dA_z$$

Therefore, the components of the total hydrostatic force along the coordinate axes are

$$F_x = \int\int \rho gzdA_x = \rho gz_c A_x$$

$$F_y = \int\int \rho gzdA_y = \rho gz_c A_y$$

$$F_z = \int\int \rho gzdA_z$$

where z_c is the z coordinate of the centroid of area A_x and A_y (the projected areas of curved surface on yz and xz plane respectively). If z_p and y_p are taken to be the coordinates of the point of action of F_x on the projected area A_x on yz plane, , we can write

$$z_p = \frac{1}{A_x z_c} \iint z^2 dA_x = \frac{I_{yy}}{A_x z_c}$$

$$y_p = \frac{1}{A_x z_c} \iint yz dA_x = \frac{I_{yz}}{A_x z_c}$$

where I_{yy} is the moment of inertia of area A_x about y-axis and I_{yz} is the product of inertia of A_x with respect to axes y and z. In the similar fashion, z_p' and x_p' the coordinates of the point of action of the force F_y on area A_y, can be written as

$$z'_p = \frac{1}{A_y z_c} \iint z^2 dA_y = \frac{I_{xx}}{A_y z_c}$$

$$x'_p = \frac{1}{A_y z_c} \iint xz dA_y = \frac{I_{xz}}{A_y z_c}$$

where I_{xx} is the moment of inertia of area A_y about x axis and I_{xz} is the product of inertia of A_y about the axes x and z.

We can conclude from previous Eqs that for a curved surface, the component of hydrostatic force in a horizontal direction is equal to the hydrostatic force on the projected plane surface perpendicular to that direction and acts through the centre of pressure of the projected area. From Eq. the vertical component of the hydrostatic force on the curved surface can be written as

$$F_z = \rho g \iint z \, dA_z = \rho g \forall$$

Where $\forall$ is the volume of the body of liquid within the region extending vertically above the submerged surface to the free surface of the liquid. Therefore, the vertical component of hydrostatic force on a submerged curved surface is equal to the weight of the liquid volume vertically above the solid surface of the liquid and acts through the center of gravity of the liquid in that volume.

Derivation of Reynolds Transport Theorem

To formulate the relation between the equations applied to a control mass system and those applied to a control volume, a general flow situation is considered in Fig. 10.3 where the velocity of a fluid is given relative to coordinate axes *ox*, *oy*, *oz*. At any time t, a control mass system consisting of a certain mass of fluid is considered to have the dotted-line boundaries as indicated. A control volume (stationary relative to the coordinate axes) is considered that exactly coincides with the control mass system at time t (Fig. 10.3a). At time t+δt, the control mass system has moved somewhat, since each particle constituting the control mass system moves with the velocity associated with its location.

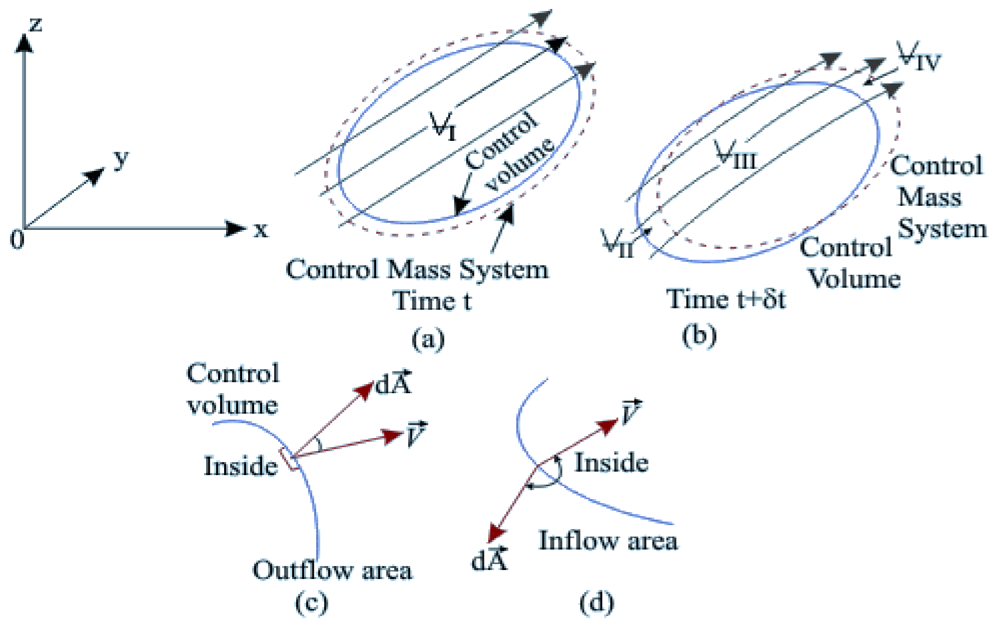


Fig 10.3 Relationship between Control Mass system and control volume concepts in the analysis of a flow field

Consider, N to be the total amount of some property (mass, momentum, energy) within the control mass system at time t, and let η be the amount of this property per unit mass throughout the fluid. The time rate of increase in N for the control mass system is now formulated in terms of the change in N for the control volume. Let the volume of the control mass system and that of the control volume be $\forall_1$at time t with both of them coinciding with each other (Fig. 10.3a). At time t + δt, the volume of the control mass system changes and comprises volumes $\forall_{III}$ and $\forall_{IV}$(Fig.

10.3b). Volumes $\forall_{II}$ and $\forall_{IV}$ are the intercepted regions between the control mass system and control volume at time t+δt. The increase in property N of the control mass system in time δt is given by

$$(N_{t+\delta t} - N_t)_{control\,mass\,system} = \left[\iiint_{\forall_{III}} \eta\rho d\forall + \iiint_{\forall_{IV}} \eta\rho d\forall\right]_{t+\delta t} - \left[\iiint_{\forall_I} \eta\rho d\forall\right]_t$$

where,d∀ represents an element of volume. After adding and subtracting

$$\left[\iiint_{\forall_{II}} \eta\rho d\forall\right]_{t+\delta t}$$

to the right hand side of the equation and then dividing throughout by δt, we have

$$\frac{(N_{t+\delta t} - N_t)_{Control\ mass\ system}}{\delta t} = \frac{\left[\iiint_{\forall_{III}} \eta\rho\, d\forall + \iiint_{\forall_{II}} \eta\rho\, d\forall\right]_{t+\delta t} - \left[\iiint_{\forall_I} \eta\rho\, d\forall\right]_t}{\delta t}$$

$$+ \frac{\left[\iiint_{\forall_{IV}} \eta\rho\, d\forall\right]_{t+\delta t}}{\delta t} - \frac{\left[\iiint_{\forall_{II}} \eta\rho\, d\forall\right]_{t+\delta t}}{\delta t} \qquad (1)$$

The left hand side of Eq.(10.9) is the average time rate of increase in N within the control mass system during the time δt.

In the limit as δt approaches zero, it becomes dN/dt (the rate of change of N within the control mass system at time t).

In the first term of the right hand side of the above equation the first two integrals are the amount of N in the control volume at time t+δt, while the third integral is the amount N in the control volume at time t. In the limit, as δt approaches zero, this term represents the time rate of increase of the property N within the control volume and can be written as $\frac{d}{dt}\iiint_{C.V} \eta\rho\, d\forall$

The next term, which is the time rate of flow of N out of the control volume may be written, in the limit $\delta t \longrightarrow 0$

as

$$\lim_{\delta t \to 0} \frac{\left[\iiint\limits_{\forall_{IV}} \eta\rho \, d\forall\right]_{t+\delta t}}{\delta t} = \iint\limits_{outflow\ area} \eta\rho\vec{V}.\, d\vec{A}$$

In which $\vec{V}$ is the velocity vector and $d\vec{A}$ is an elemental area vector on the control surface. The sign of vector $d\vec{A}$ is positive if its direction is outward normal (Fig. 10.3c). Similarly, the last term of the Eq.(10.9) is the rate of flow of N into the control volume is, in the limit $\delta t \to 0$

$$\lim_{\delta t \to 0} \frac{\left[\iiint\limits_{\forall_{II}} \eta\rho \, d\forall\right]_{t+\delta t}}{\delta t} = -\iint\limits_{inflow\ area} \eta\rho\vec{V}.d\vec{A}$$

The minus sign is needed as $\vec{V}.\, d\vec{A}$ is negative for inflow. The last two terms of Eq.(10.9) may be combined into a single one is an integral over the entire surface of the control volume and is written as $\iint\limits_{cs} \eta\rho\vec{V}.\, d\vec{V}$. This term indicates the net rate of outflow N from the control lume. Hence, Eq.(10.9) can be written as

$$\left(\frac{dN}{dt}\right)_{CMS} = \frac{\partial}{\partial t}\iiint\limits_{CV} \eta\rho \, d\forall + \iint\limits_{CS} \eta\rho\vec{V}.\, d\vec{A} \qquad (2)$$

The Eq.(10.10) is known as Reynolds Transport Theorem

Important Note: In the derivation of Reynolds transport theorem (Eq. 10.10), the velocity field was described relative to a reference frame xyz (Fig. 10.3) in which the control volume was kept fixed, and no restriction was placed on the motion of the reference frame xyz. This makes it clear that the fluid velocity in Eq.(10.10) is measured relative to the control volume. To emphasize this point, the Eq. (10.10) can be written as

$$\left(\frac{dN}{dt}\right)_{CMS} = \frac{\partial}{\partial t}\iiint_{CV} \eta\rho \, d\forall + \iint_{CS} \eta\rho(\vec{V}_r \, . \, d\vec{A}) \tag{3}$$

where the fluid velocity $\vec{V}_r$, is defined relative to the control volume as

$$\vec{V}_r = \vec{V} - \vec{V}_c \tag{4}$$

$\vec{V}$ and $\vec{V}_c$ are now the velocities of fluid and the control volume respectively as observed in a fixed frame of reference. The velocity $\vec{V}_c$ of the control volume may be constant or any arbitrary function of time

Lecture Note 2

Fluid Kinematics

Steady flow

A steady flow is one in which all conditions at any point in a stream remain constant with respect to time.

Or

A steady flow is the one in which the quantity of liquid flowing per second through any section, is constant.

This is the definition for the ideal case. True steady flow is present only in Laminar flow. In turbulent flow, there are continual fluctuations in velocity. Pressure also fluctuate at every point. But if this rate of change of pressure and velocity are equal on both sides of a constant average value, the flow is steady flow. The exact term use for this is mean steady flow.
Steady flow may be uniform or non-uniform.

Uniform flow

A truly uniform flow is one in which the velocity is same at a given instant at every point in the fluid.

This definition holds for the ideal case. Whereas in real fluids velocity varies across the section.

But when the size and shape of cross section are constant along the length of channels under consideration, the flow is said to be uniform.

Non-uniform flow

A non-uniform flow is one in which velocity is not constant at a given instant.

Unsteady Flow

A flow in which quantity of liquid flowing per second is not constant, is called unsteady flow.

Unsteady flow is a transient phenomenon. It may be in time become steady or zero flow. For example when a valve is closed at the discharge end of the pipeline.Thus, causing the velocity in

the pipeline to decrease to zero. In the meantime, there will be fluctuations in both velocity and pressure within the pipe.
Unsteady flow may also include periodic motion such as that of waves of beaches. The difference between these cases and mean steady flow is that there is so much deviation from the mean. And the time scale is also much longer.

One, Two and Three Dimensional Flows

Term one, two or three dimensional flow refers to the number of space coordinated required to describe a flow. It appears that any physical flow is generally three-dimensional. But these are difficult to calculate and call for as much simplification as possible. This is achieved by ignoring changes to flow in any of the directions, thus reducing the complexity. It may be possible to reduce a three-dimensional problem to a two-dimensional one, even an one-dimensional one at times.

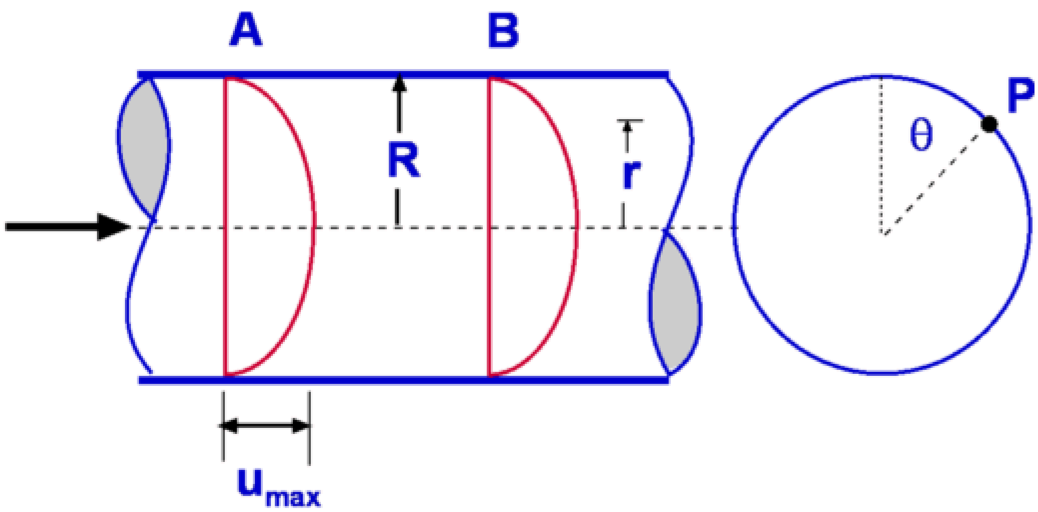


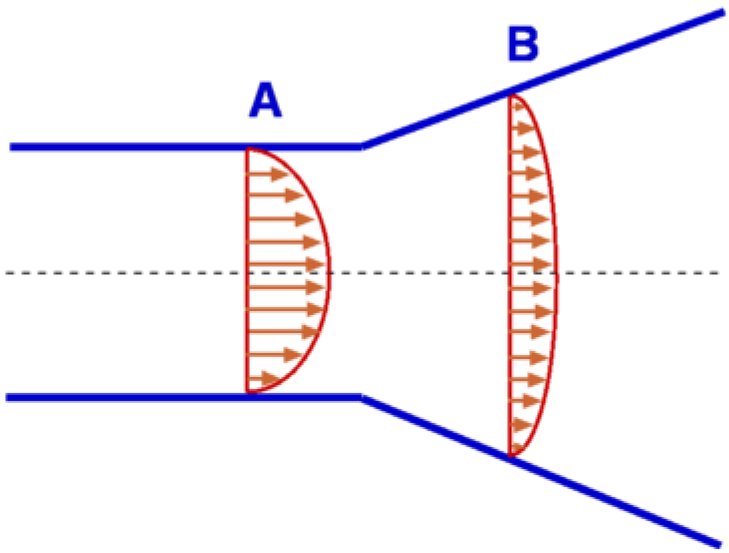
Figure-1: Example of one-dimensional flow

Consider flow through a circular pipe. This flow is complex at the position where the flow enters the pipe. But as we proceed downstream the flow simplifies considerably and attains the state of a fully developed flow. A characteristic of this flow is that the velocity becomes invariant in the flow direction as shown in Fig-1. Velocity for this flow is given by

$$u = u_{max}\left[1 - \left(\frac{r}{R}\right)^2\right] \tag{3.6}$$

It is readily seen that velocity at any location depends just on the radial distance r from the centreline and is independent of distance, x or of the angular position θ. This represents a typical one-dimensional flow.

Now consider a flow through a diverging duct as shown in Fig. 2. Velocity at any location depends not only upon the radial distance r but also on the x-distance. This is therefore a two-dimensional flow.



Example -2: A two-dimensional flow

Concept of a uniform flow is very handy in analyzing fluid flows. A uniform flow is one where the velocity and other properties are constant independent of directions. We usually assume a uniform flow at the entrance to a pipe, far away from a aerofoil or a motor car as shown in Fig.3.

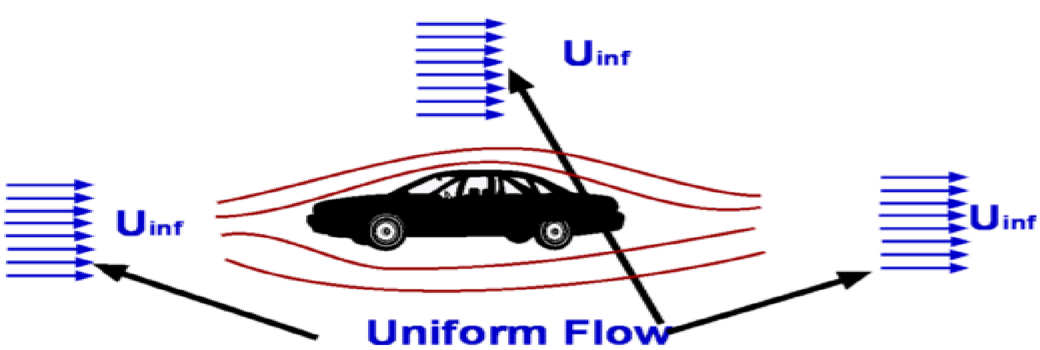


Fig:4 Uniform flow

1. Real fluids

The flow of real fluids exhibits viscous effect that is they tend to "stick" to solid surfaces and have stresses within their body.

You might remember from earlier in the course Newton's law of viscosity:

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

This tells us that the shear stress, τin a fluid is proportional to the velocity gradient - the rate of change of velocity across the fluid path. For a "Newtonian" fluid we can write:

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

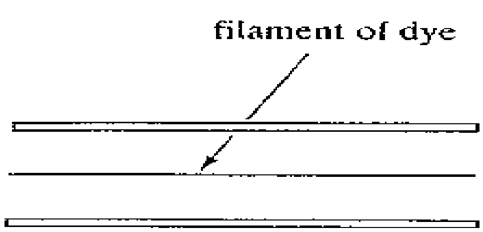
Where the constant of proportionality, μ is known as the coefficient of viscosity (or simply viscosity). We saw that for some fluids - sometimes known as exotic fluids - the value of μ changes with stress or velocity gradient. We shall only deal with Newtonian fluids.

In his lecture we shall look at how the forces due to momentum changes on the fluid and viscous forces compare and what changes take place.

2.Laminar and turbulent flow

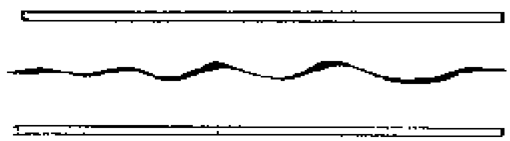
If we were to take a pipe of free flowing water and inject a dye into the middle of the stream, what would we expect to happen?

This



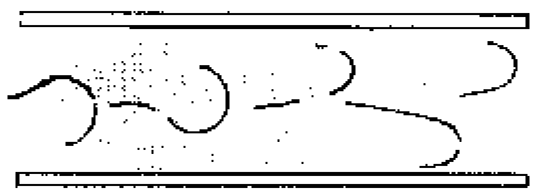
Laminar (viscous)

This



Transitional

Or this



Turbulent

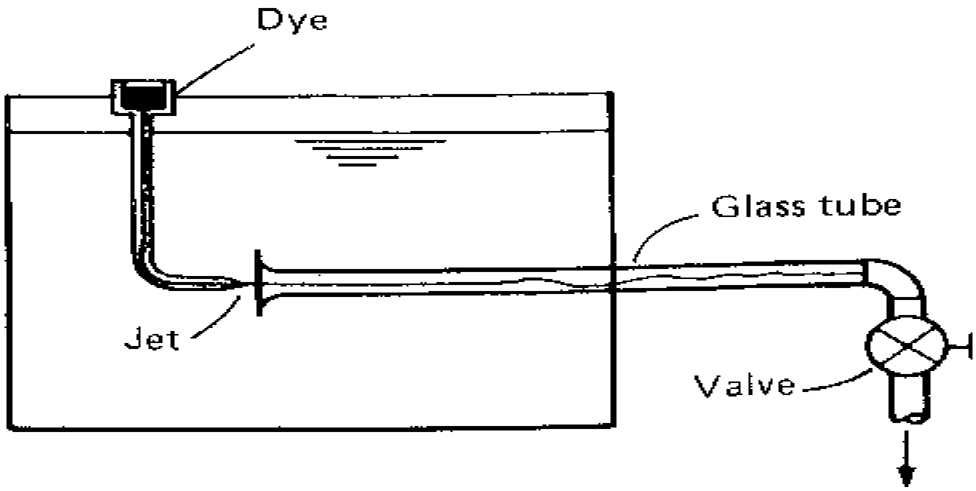
Actually both would happen - but for different flow rates. The top occurs when the fluid is flowing fast and the lower when it is flowing slowly.

The top situation is known as turbulent flow and the lower as laminar flow.

In laminar flow the motion of the particles of fluid is very orderly with all particles moving in straight lines parallel to the pipe walls.

But what is fast or slow? And at what speed does the flow pattern change? And why might we want to know this?

The phenomenon was first investigated in the 1880s by Osbourne Reynolds in an experiment which has become a classic in fluid mechanics.



He used a tank arranged as above with a pipe taking water from the centre into which he injected a dye through a needle. After many experiments he saw that this expression

$$\frac{\rho u d}{\mu}$$

where ρ = density, u = mean velocity, d = diameter and ν = viscosity

would help predict the change in flow type. If the value is less than about 2000 then flow is laminar, if greater than 4000 then turbulent and in between these then in the transition zone.

This value is known as the Reynolds number, Re:

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$

Laminar flow: Re < 2000

Transitional flow: 2000 < Re < 4000

Turbulent flow: Re > 4000

What are the units of this Reynolds number? We can fill in the equation with SI units:

$$\rho = kg/m^3, \quad u = m/s, \qquad d = m$$
$$\mu = Ns/m^2 = kg/ms$$

$$\mathrm{Re} = \frac{\rho u d}{\mu} = \frac{kg}{m^3}\frac{m}{s}\frac{m}{1}\frac{m}{kg} = 1$$

i.e. it has no units. A quantity that has no units is known as a non-dimensional (or dimensionless) quantity. Thus the Reynolds number, Re, is a non-dimensional number.

We can go through an example to discover at what velocity the flow in a pipe stops being laminar.

If the pipe and the fluid have the following properties:

water density ρ = 1000 kg/m^3

pipe diameter d = 0.5m

(dynamic) viscosity,ν = 0.55x10^{-3} Ns/m^2

We want to know the maximum velocity when the Re is 2000.

$$\mathrm{Re} = \frac{\rho u d}{\mu} = 2000$$
$$u = \frac{2000\mu}{\rho d} = \frac{2000 \times 0.55 \times 10^{-3}}{1000 \times 0.5}$$
$$u = 0.0022\, m/s$$

If this were a pipe in a house central heating system, where the pipe diameter is typically 0.015m, the limiting velocity for laminar flow would be, 0.0733 m/s.

Both of these are very slow. In practice it very rarely occurs in a piped water system - the velocities of flow are much greater. Laminar flow does occur in situations with fluids of greater viscosity - e.g. in bearing with oil as the lubricant.

At small values of Re above 2000 the flow exhibits small instabilities. At values of about 4000 we can say that the flow is truly turbulent. Over the past 100 years since this experiment, numerous more experiments have shown this phenomenon of limits of Re for many different Newtonian fluids - including gasses.

What does this abstract number mean?

We can say that the number has a physical meaning, by doing so it helps to understand some of the reasons for the changes from laminar to turbulent flow.

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$
$$= \frac{\text{inertial forces}}{\text{viscous forces}}$$

It can be interpreted that when the inertial forces dominate over the viscous forces (when the fluid is flowing faster and Re is larger) then the flow is turbulent. When the viscous forces are dominant (slow flow, low Re) they are sufficient enough to keep all the fluid particles in line, then the flow is laminar.

Laminar flow

- Re < 2000
- 'low' velocity
- Dye does not mix with water
- Fluid particles move in straight lines
- Simple mathematical analysis possible
- Rare in practice in water systems.

Transitional flow

- 2000 > Re < 4000
- 'medium' velocity
- Dye stream wavers in water - mixes slightly.

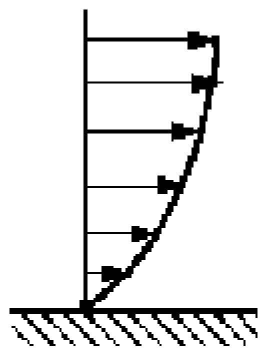
Turbulent flow

- Re > 4000
- 'high' velocity
- Dye mixes rapidly and completely
- Particle paths completely irregular
- Average motion is in the direction of the flow
- Cannot be seen by the naked eye
- Changes/fluctuations are very difficult to detect. Must use laser.
- Mathematical analysis very difficult - so experimental measures are used

3. Pressure loss due to friction in a pipeline

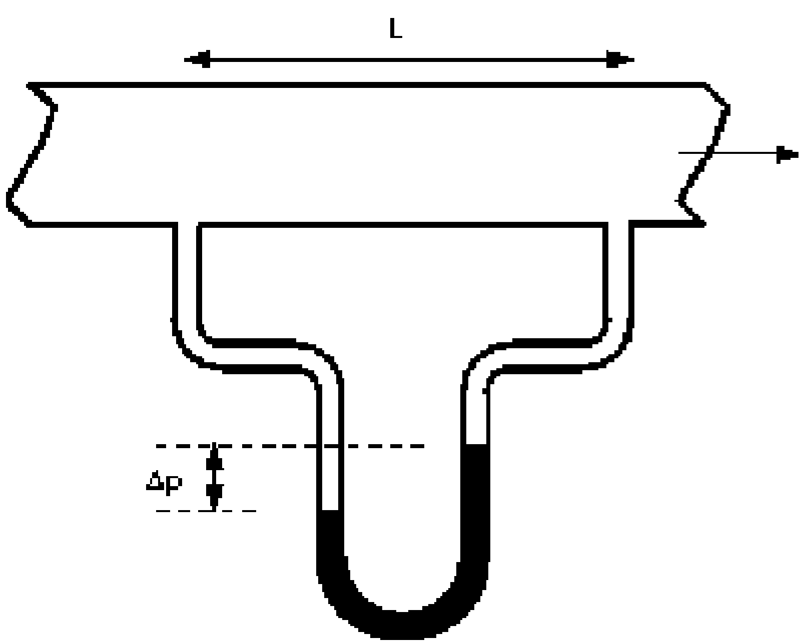
Up to this point on the course we have considered ideal fluids where there have been no losses due to friction or any other factors. In reality, because fluids are viscous, energy is lost by flowing fluids due to friction which must be taken into account. The effect of the friction shows itself as a pressure (or head) loss.

In a pipe with a real fluid flowing, at the wall there is a shearing stress retarding the flow, as shown below.



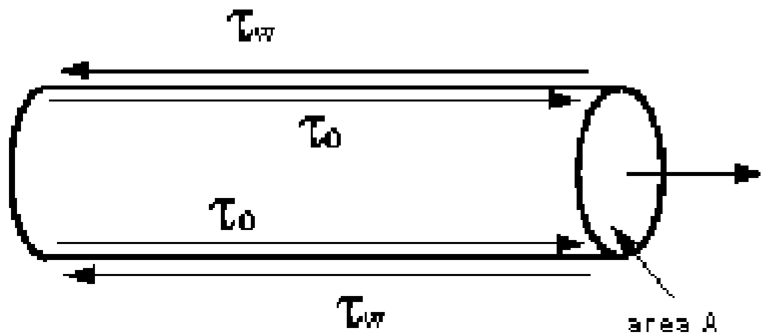
If a manometer is attached as the pressure (head) difference due to the energy lost by the fluid overcoming the shear stress can be easily seen.

The pressure at 1 (upstream) is higher than the pressure at 2.



We can do some analysis to express this loss in pressure in terms of the forces acting on the fluid.

Consider a cylindrical element of incompressible fluid flowing in the pipe, as shown



The pressure at the upstream end is *p*, and at the downstream end the pressure has fallen by Δp to (p-Δp).

The driving force due to pressure (F = Pressure x Area) can then be written

driving force = Pressure force at 1 - pressure force at 2

$$pA-(p-\Delta p)A=\Delta p\,A=\Delta p\frac{\pi d^2}{4}$$

The retarding force is that due to the shear stress by the walls

$$\begin{aligned}&= \text{shear stress} \times \text{area over which it acts}\\ &= \tau_w \times \text{area of pipe wall}\\ &= \tau_w \pi d L\end{aligned}$$

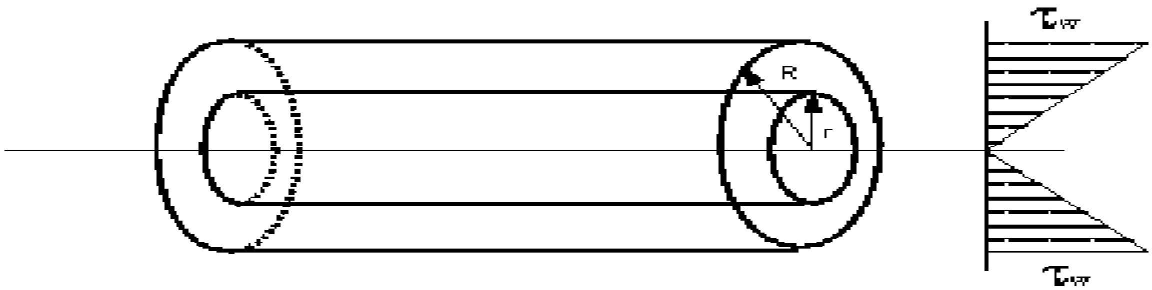
As the flow is in equilibrium,

driving force = retarding force

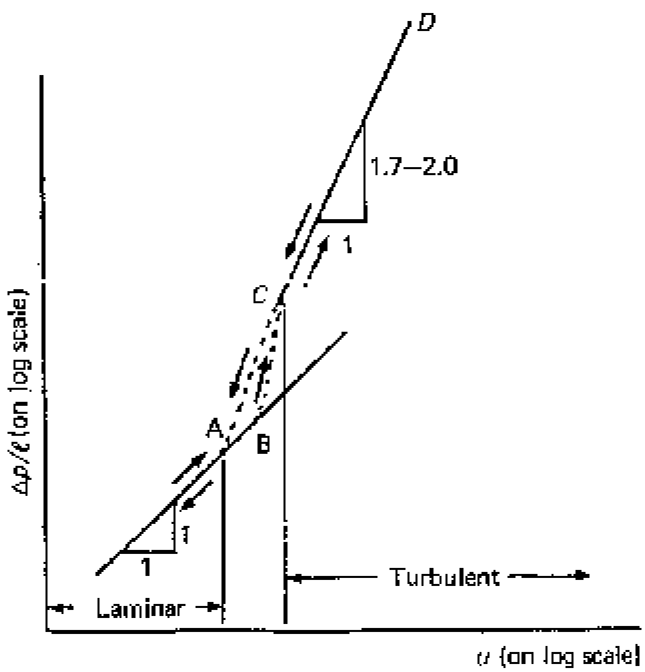
$$\Delta p\frac{\pi d^2}{4}=\tau_w \pi d L$$

$$\Delta p=\frac{\tau_w 4L}{d}$$

Giving an expression for pressure loss in a pipe in terms of the pipe diameter and the shear stress at the wall on the pipe.



The shear stress will vary with velocity of flow and hence with Re. Many experiments have been done with various fluids measuring the pressure loss at various Reynolds numbers. These results plotted to show a graph of the relationship between pressure loss and Re look similar to the figure below:



This graph shows that the relationship between pressure loss and Re can be expressed as

laminar $\Delta p \propto u$

turbulent $\Delta p \propto u^{1.7 \text{ (or } 2.0)}$

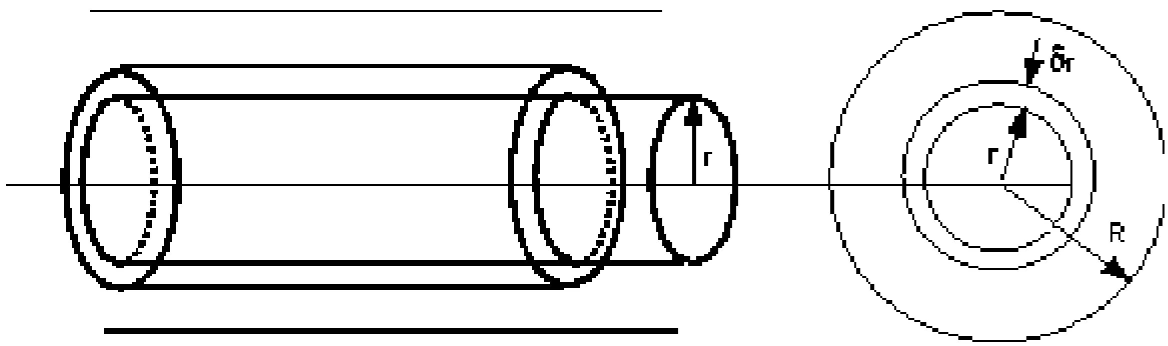
As these are empirical relationships, they help in determining the pressure loss but not in finding the magnitude of the shear stress at the wall on a particular fluid. We could then use it to give a general equation to predict the pressure loss.

4. Pressure loss during laminar flow in a pipe

In general the shear stress τ_w. is almost impossible to measure. But for laminar flow it is possible to calculate a theoretical value for a given velocity, fluid and pipe dimension.

In laminar flow the paths of individual particles of fluid do not cross, so the flow may be considered as a series of concentric cylinders sliding over each other - rather like the cylinders of a collapsible pocket telescope.

As before, consider a cylinder of fluid, length L, radius **r**, flowing steadily in the center of a pipe.



We are in equilibrium, so the shearing forces on the cylinder equal the pressure forces.

$$\tau\, 2\pi r\, L = \Delta p\, A = \Delta p \pi r^2$$
$$\tau = \frac{\Delta p}{L}\frac{r}{2}$$

By Newtons law of viscosity we have $\tau = \mu \frac{du}{dy}$, where y is the distance from the wall. As we are measuring from the pipe centre then we change the sign and replace y with r distance from the centre, giving

$$\tau = -\mu \frac{du}{dr}$$

Which can be combined with the equation above to give

$$\frac{\Delta p}{L}\frac{r}{2} = -\mu \frac{du}{dr}$$
$$\frac{du}{dr} = -\frac{\Delta p}{L}\frac{r}{2\mu}$$

In an integral form this gives an expression for velocity,

$$u = -\frac{\Delta p}{L}\frac{1}{2\mu}\int r\,dr$$

Integrating gives the value of velocity at a point distance r from the centre

$$u_r = -\frac{\Delta p}{L}\frac{r^2}{4\mu} + C$$

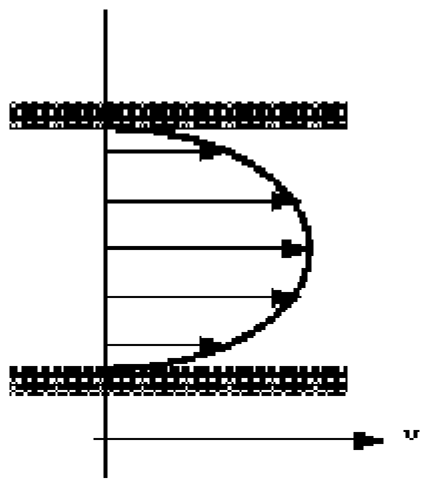
At *r = 0*, (the centre of the pipe), $u = u_{max}$, at *r = R* (the pipe wall) *u = 0*, giving

$$C = \frac{\Delta p}{L}\frac{R^2}{4\mu}$$

so, an expression for velocity at a point r from the pipe centre when the flow is laminar is

$$u_r = \frac{\Delta p}{L}\frac{1}{4\mu}\left(R^2 - r^2\right)$$

Note how this is a parabolic profile (of the form $y = ax^2 + b$) so the velocity profile in the pipe looks similar to the figure below



What is the discharge in the pipe?

$$Q = u_m A$$
$$u_m = \int_0^R u_r \, dr$$
$$= \frac{\Delta p}{L}\frac{1}{4\mu}\int_0^R \left(R^2 - r^2\right) dr$$
$$= \frac{\Delta p}{L}\frac{R^2}{8\mu} = \frac{\Delta p \, d^2}{32\mu L}$$

So the discharge can be written

$$Q = \frac{\Delta p \, d^2}{32\mu L}\frac{\pi d^2}{4}$$
$$= \frac{\Delta p}{L}\frac{\pi d^2}{128\mu}$$

This is the Hagen-Poiseuille equation for laminar flow in a pipe. It expresses the discharge Q in terms of the pressure gradient ($\frac{\partial p}{\partial x} = \frac{\Delta p}{L}$), diameter of the pipe and the viscosity of the fluid.

We are interested in the pressure loss (head loss) and want to relate this to the velocity of the flow. Writing pressure loss in terms of head loss h_f, i.e. $p = \rho g h_f$

$$u = \frac{\rho g h_f d^2}{32\mu L}$$
$$h_f = \frac{32\mu L u}{\rho g d^2}$$

This shows that pressure loss is directly proportional to the velocity when flow is laminar.

It has been validated many time by experiment.

It justifies two assumptions:

1. fluid does not slip past a solid boundary
2. Newton's hypothesis.

Streamline :

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 stream lines can never intersect each other, as the instantaneous velocity vector at any given point is unique.
The differential equation of streamline may be written as

$$\frac{du}{u} = \frac{dv}{v} = \frac{dw}{w}$$

whereu,v, and w are the velocity components in x, y and z directions respectively as sketched.

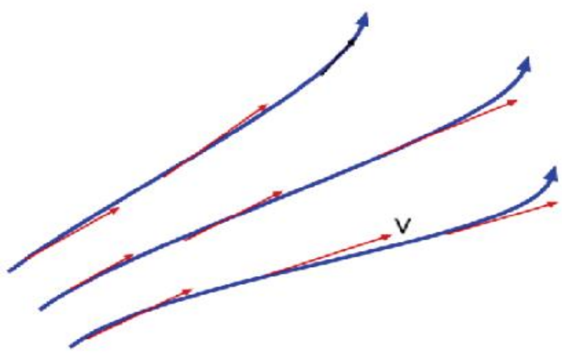


Fig. Streamlines

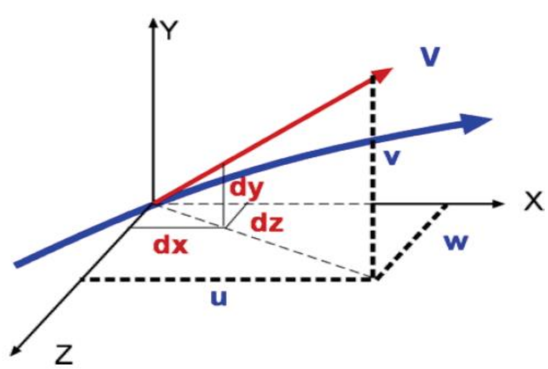


Fig. Streamline function

Stream tube :

If streamlines are drawn through a closed curve, they form a boundary surface across which fluid cannot penetrate. Such a surface bounded by streamlines is a sort of tube, and is known as a streamtube.

From the definitionof streamline, it is evident that no fluid can cross the bounding surface of the streamtube. This implies that the quantity(mass) of fluid entering the streamtube at one end must be the same as the quantity leaving it at the other. The streamtubeis generally assumed to be a small cross-sectional area so that the velocity over it could be considered uniform.

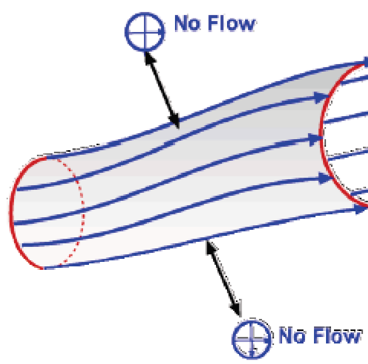


Fig.Streamtube

Pathline :

A pathline is the locus of a fluid particle as it moves along. In others word, a pathline is a curve traced by a single fluid particle during its motion.

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.

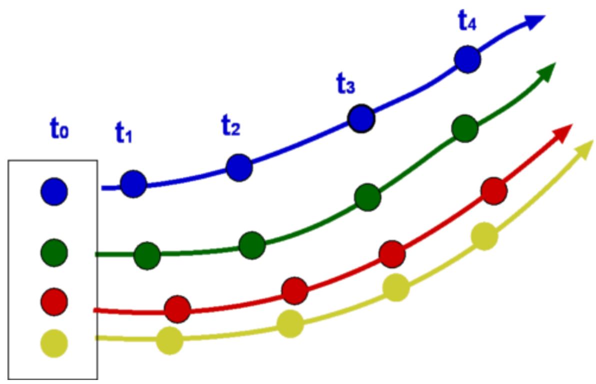


Fig.Pathline

Streak line :

Streakline concentrates on fluid particles that have gone through a fixed station orpoint. At some instant of time the position of all these particles are marked and aline is drawn through them. Such a line is called a streakline. Thus, a streakline connects all particles passing through a given point.

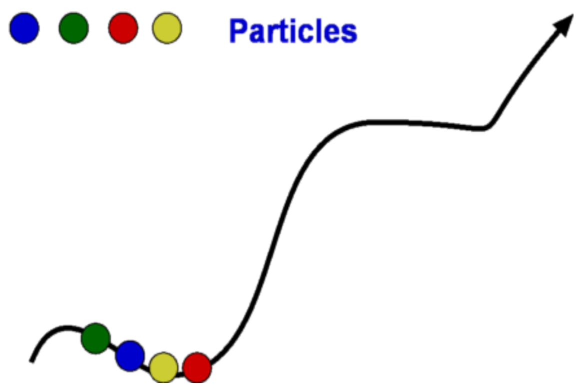


Fig.Streaklines

In a steady flow the streamline, pathline and streakline all coincide. In an unsteady flow they can be different. Streamlines are easily generated mathematically while pathline and streaklines are obtained through experiments.

Stream function :

The idea of introducing stream function works only if the continuity equation is reduced to two terms. There are 4-terms in the continuity equation that one can get by expanding the vector equation i.e.,

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

For a steady, incompressible, plane, two-dimensional flow, this equation reduces to,

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

Here, the striking idea of stream function works that will eliminate two velocity components u and v into a single variable. So, the *stream function* $\psi(x, y)$ relates to the velocity components in such a way that continuity equation is satisfied.

$$u = \frac{\eth\psi}{\eth y};\ v = -\frac{\eth\psi}{\eth x}$$

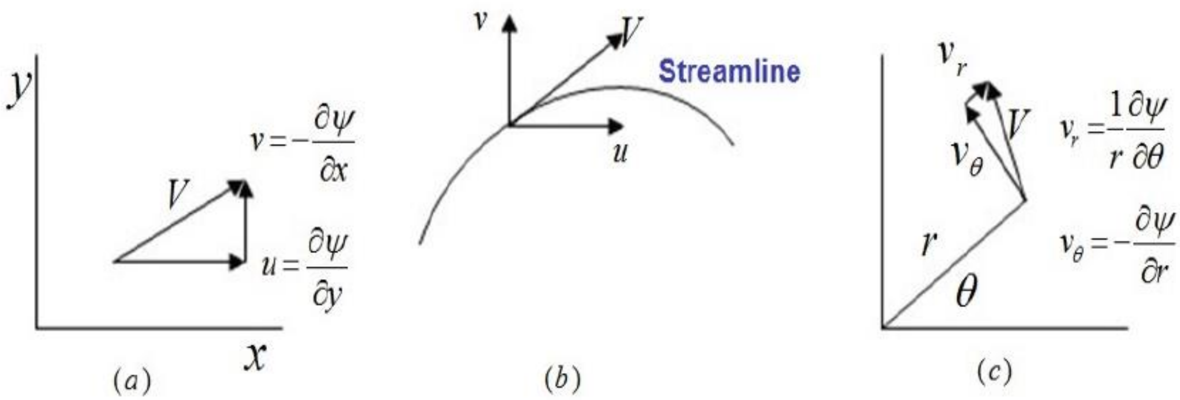


Fig. Velocity components along a streamline

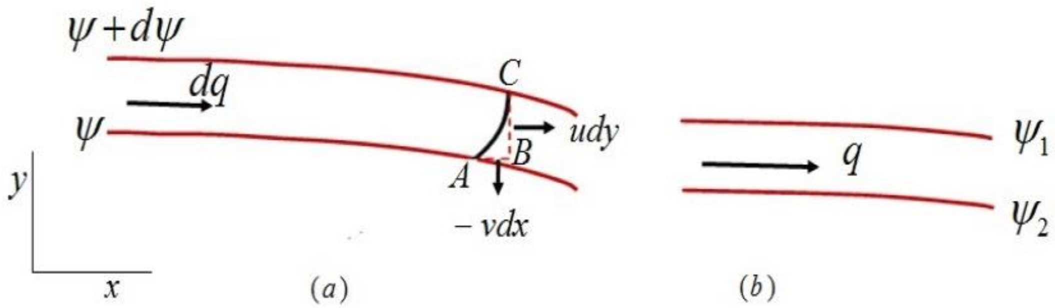


Fig. Flow between two streamlines

Velocity potential :

An irrotational flow is defined as the flow where the vorticity is zero at every point. It gives rise to a scalar function Φ which is similar and complementary to the stream function ψ. Let us consider the equations of irrortional flow and scalar function Φ. In an irrotational flow, there is no vorticity ξ.

The velocity potential is represented by Φ and is defined by the following expression :

$$-\Phi = \int V_s ds$$

in which V_s is the velocity along a small length element ds. So we get

$$d\Phi = -V_s\, ds$$

$$\text{or } V_s = -(d\Phi / ds)$$

The velocity potential is a scalar quantity dependent upon space and time. Its negative derivative with respect to any direction gives the velocity in that direction, that is

$$u = -\frac{\partial \Phi}{\partial x}\ , v = -\frac{\partial \Phi}{\partial y}\ , w = -\frac{\partial \Phi}{\partial z}$$

In polar co-ordinates (r, θ, z), the velocity components are

$$v_r = -\ \frac{\partial \Phi}{\partial r}\ ,\ v_\theta = \frac{\partial \Phi}{r\,\partial r}\ ,\ v_z = -\ \frac{\partial \Phi}{\partial z}$$

The velocity potential Φ thus provides an alternative means of expressing velocity components. The minus sign in equation appears because of the convention that the velocity potential decreases in the direction of flow just as the electrical potential decreases in the direction in which the current flows. The velocity potential is not a physical quantity which could be directly measured and, therefore, its zero position may be arbitrarily chosen.

Flownet :

The flownet is a graphical representation of two-dimensional irrotational flow and consists of a family of streamlines intersecting orthogonally a family of equipotential lines (they intersect at right angles) and in the process forming small curvilinear squares.

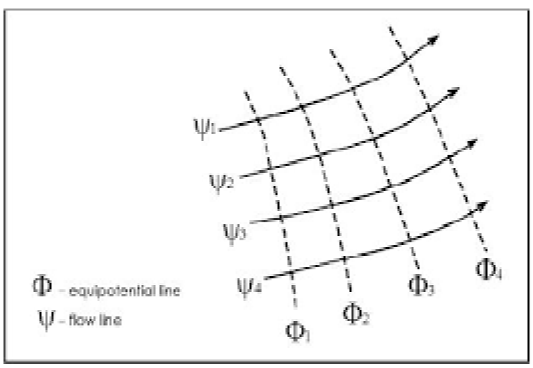


Fig.Flownet

Uses of flownet :

- For given boundaries of flow, the velocity and pressure distribution can be determined, if the velocity distribution and pressure at any reference section are known
- Loss of flow due to seepage in earth dams and unlined canals can be evaluated
- Uplift pressures on the undesirable (bottom) of the dam can be worked out

Relation between stream function & velocity potential

$\emptyset$exists only in irrotational flow where as φexists in both rotational as well as irrotational flow

$$u= -\frac{\partial \emptyset}{\partial x} = \frac{\partial \varphi}{\partial y} \;\&\; v=\frac{\partial \varphi}{\partial x} = -\frac{\partial \emptyset}{\partial y}$$

therefore, $\frac{\partial \varphi}{\partial x} = \frac{\partial \emptyset}{\partial y} \;\&\; \frac{\partial \emptyset}{\partial x} = -\frac{\partial \varphi}{\partial y}$

Lecture Note 3

Potential Flow

Introduction

In a plane irrotaional flow, one can use either velocity potential or stream function to define the flow field and both must satisfy *Laplace equation* . Moreover, the analysis of this equation is much easier than direct approach of fully viscous Navier-Stokes equations. Since the *Laplace equation* is linear, various solutions can be added to obtain other solutions. Thus, if we have certain basic solutions, then they can be combined to obtain complicated and interesting solutions. The analysis of such flow field solutions of *Laplace equation* is termed as *potential theory* . The potential theory has a lot of practical implications defining complicated flows. Here, we shall discuss the stream function and velocity potential for few elementary flow fields such as uniform flow, source/sink flow and vortex flow. Subsequently, they can be superimposed to obtain complicated flow fields of practical relevance.

Governing equations for irrotational and incompressible flow

The analysis of potential flow is dealt with combination of potential lines and streamlines. In a planner flow, the velocities of the flow field can be defined in terms of stream functions $\{\psi(x,y)\}$ and potential functions $\{\phi(x,y)\}$ as,

$$u=\frac{\partial\{\psi(x,y)\}}{\partial y};\quad v=-\frac{\partial\{\psi(x,y)\}}{\partial x} \qquad (3.4.1)$$

$$u=\frac{\partial\{\phi(x,y)\}}{\partial x};\quad v=\frac{\partial\{\phi(x,y)\}}{\partial y}$$

The stream function ψ is defined such that continuity equation is satisfied whereas, for low speed irrotational flows $\left(\nabla\times\vec{V}=0\right)$, if the viscous effects are neglected, the continuity equation $\left(\nabla\cdot\vec{V}=0\right)$, reduces to Laplace equation for ϕ. Both the functions satisfy the Laplace equations i.e.

$$\frac{\partial^2\phi}{\partial x^2}+\frac{\partial^2\phi}{\partial y^2}=0;\quad \frac{\partial^2\psi}{\partial x^2}+\frac{\partial^2\psi}{\partial y^2}=0 \qquad (3.4.3)$$

Thus, the following obvious and important conclusions can be drawn from Eq. (3.4.2);

- • Any irrotational, incompressible and planner flow (two-dimensional) has a velocity potential and stream function and both the functions satisfy Laplace equation.

  • Conversely, any solution of Laplace equation represents the velocity potential or stream function for an irrotational, incompressible and two-dimensional flow.

  Note that Eq. (3.4.2) is a second-order linear partial differential equation. If there are n separate solutions such as, $\phi_1(x,y), \phi_2(x,y), \ldots\ldots\ldots, \phi_n(x,y)$ then the sum (Eq. 3.4.3) is also a solution of Eq. (3.4.2).

$$\phi(x,y) = \phi_1(x,y) + \phi_2(x,y) + \ldots\ldots\ldots + \phi_n(x,y) \quad (3.4.3)$$

It leads to an important conclusion that a complicated flow pattern for an irrotational, incompressible flow can be synthesized by adding together a number of elementary flows which are also irrotational and incompressible. However, different values of ϕ or ψ represent the different streamline patterns of the body and at the same time they satisfy the *Laplace equation*. In order to differentiate the streamline patterns of different bodies, it is necessary to impose suitable boundary conditions as shown in Fig. 3.4.1. The most common boundary conditions include far-field and wall boundary conditions. on the surface of the body (i.e. wall).

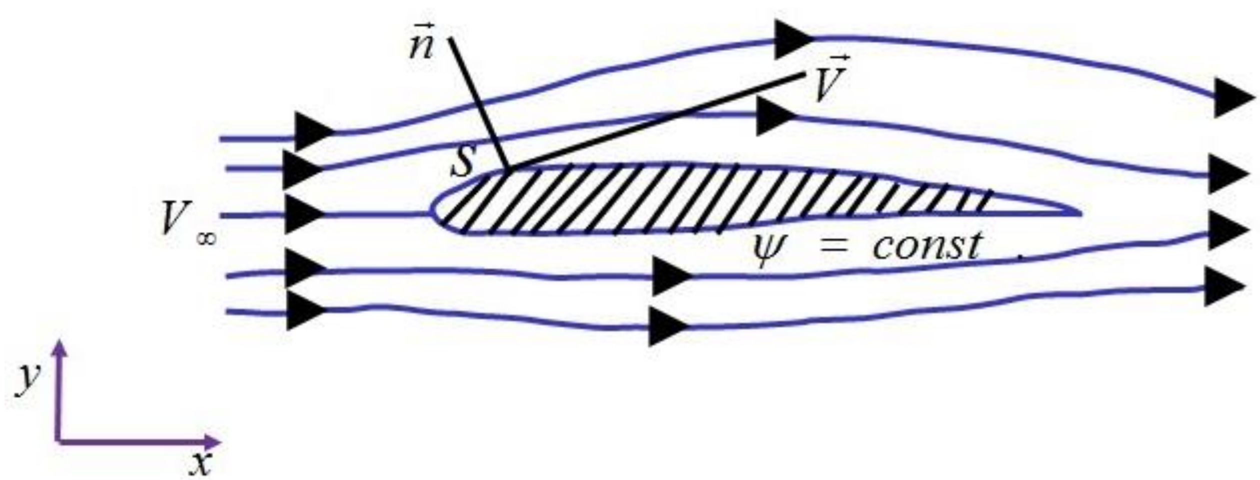


Fig. 3.4.1: Boundary conditions of a streamline body.

Far away from the body, the flow approaches uniform free stream conditions in all directions. The velocity field is then specified in terms of stream function and potential function as,

$$u = \frac{\partial \psi}{\partial y} = \frac{\partial \phi}{\partial x} = V_\infty \qquad (3.4.4)$$

$$v = -\frac{\partial \psi}{\partial x} = \frac{\partial \phi}{\partial y} = 0$$

On the solid surface, there is no velocity normal to the body surface while the tangent at any point on the surface defines the surface velocity. So, the boundary conditions can be written in terms of stream and potential functions as,

$$\frac{\partial \phi}{\partial n} = 0; \ \frac{\partial \psi}{\partial s} = 0 \qquad (3.4.5)$$

Here, s is the distance measured along the body surface and n is perpendicular to the body. Thus, any line of constant ψ in the flow may be interpreted as body shape for which there is no velocity normal to the surface. If the shape of the body is given by $y_b = f(x)$, then $\psi_{body} = \psi_{y=y_b} = \text{constant}$ is alternate boundary condition of Eq. (3.4.5). If we deal with wall boundary conditions in terms of u and v, then the equation of streamline evaluated at body surface is given by,

$$\left(\frac{dy_b}{dx}\right)_{\psi=\text{constant}} = -\frac{1}{\left(dy_b/dx\right)_{\phi=\text{constant}}} = \left(\frac{v}{u}\right)_{surface} \qquad (3.4.6)$$

It is seen that lines of constant ϕ (equi-potential lines) are orthogonal to lines of constant ψ (streamlines) at all points where they intersect. Thus, for a potential flow field, a *flow-net* consisting of family of streamlines and potential lines can be drawn, which are orthogonal to each other. Both the set of lines are laplacian and they are useful tools to visualize the flow field graphically.

Referring to the above discussion, the general approach to the solution of irrotational, incompressible flows can be summarized as follows;

- • Solve the Laplace equation for ϕ and ψ along with proper boundary conditions.

• Obtain the flow velocity from Eq. (3.4.1)

• Obtain the pressure on the surface of the body using Bernoulli's equation.

$$p + \frac{1}{2}\rho V^2 = p_\infty + \frac{1}{2}\rho V_\infty^2 \qquad (3.4.7)$$

In the subsequent section, the above solution procedure will be applied to some basic elementary incompressible flows and later they will be superimposed to synthesize more complex flow problems.

Uniform Flow

The simplest type of *elementary flow* for which the streamlines are straight, parallel with constant velocity, is known as *uniform flow*. Consider a *uniform flow* in positive x -direction as shown in Fig. 3.4.2. This flow can be represented as,

$$u = V_\infty;\ \ v = 0 \tag{3.4.8}$$

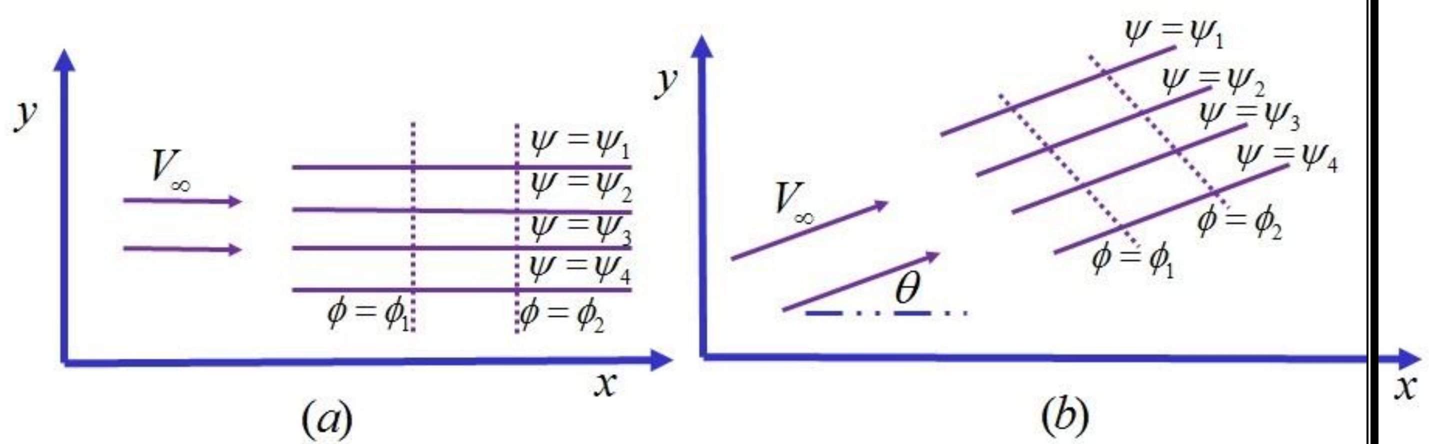


Fig. 3.4.3: Schematic representation of a uniform flow.

The uniform flow is a physically possible incompressible flow that satisfies continuity equation $\left(\nabla \cdot \vec{V} = 0\right)$ and the flow is irrotational $\left(\nabla \times \vec{V} = 0\right)$. Hence, the velocity potential can be written as,

$$\frac{\partial \phi}{\partial x} = u = V_\infty;\ \ \frac{\partial \phi}{\partial y} = v = 0$$
$$\frac{\partial \psi}{\partial y} = u = V_\infty;\ \ \frac{\partial \psi}{\partial y} = -v = 0 \tag{3.4.9}$$

Integrating Eq.(3.4.9) with respect to x ,

$$\phi = V_\infty x + f_1(y);\ \ \phi = g_1(x) + C_1$$
$$\Rightarrow g_1(x) = V_\infty x \text{ and } f_1(y) = C_1$$
$$\Rightarrow \phi = V_\infty x + C_1 \qquad (3.4.10)$$
$$\psi = V_\infty y + f_2(x);\ \ \psi = g_2(y) + C_2$$
$$\Rightarrow g_2(y) = V_\infty y \text{ and } f_2(x) = C_2$$
$$\Rightarrow \psi = V_\infty y + C_2$$

In practical flow problems, the actual values of ϕ and ψ are not important, rather it is always used as differentiation to obtain the velocity vector. Hence, the constant appearing in Eq. (3.4.10) can be set to zero. Thus, for a uniform flow, the stream functions and potential function can be written as,

$$\phi = V_\infty x; \quad \psi = V_\infty y \qquad (3.4.11)$$

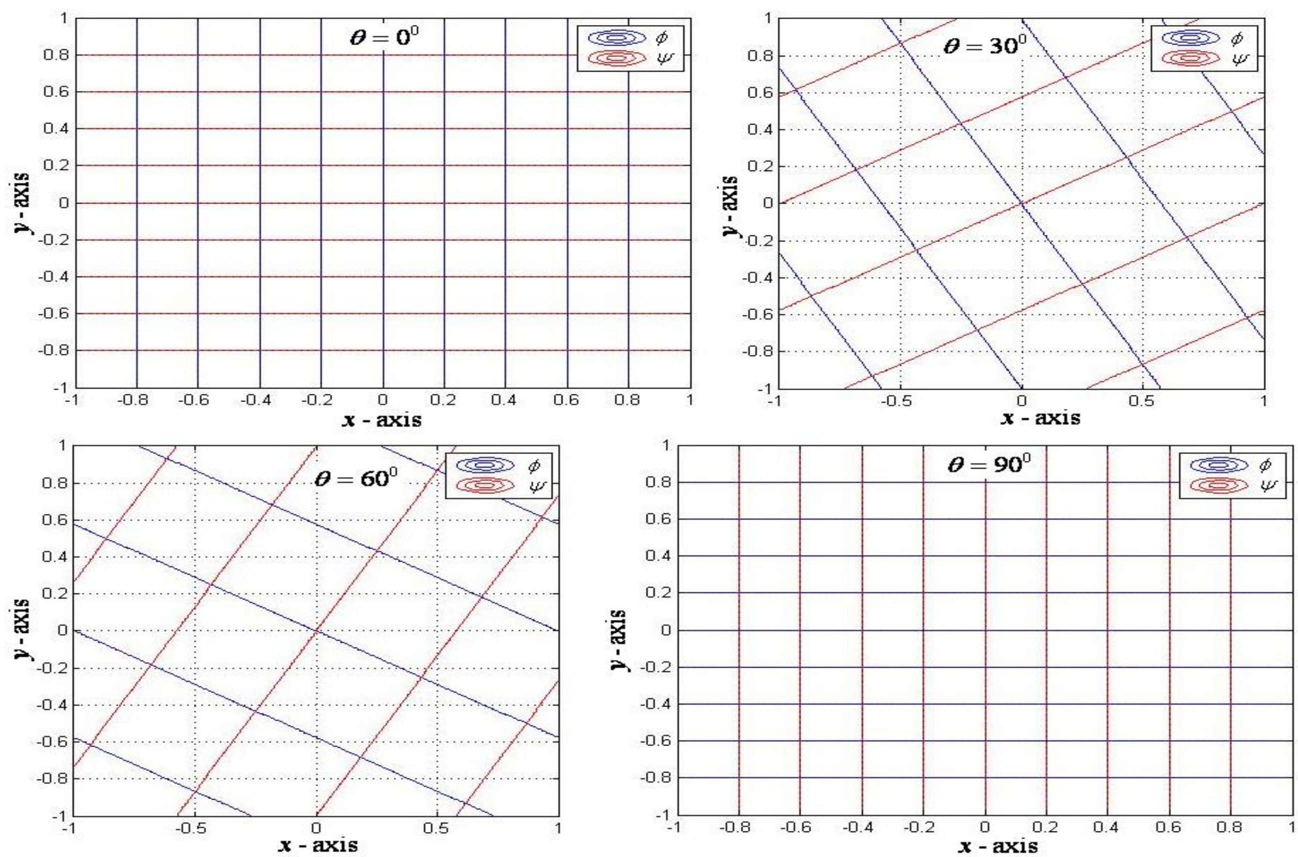


Fig. 3.4.3: Flow nets drawn for uniform flow.

When Eq. (3.4.11), is substituted in Eq. (3.4.3), Laplace equation is satisfied. Further, if the uniform flow is at an angle θ with respect to x-axis as shown in Fig. 3.4.3, then the generalized form of stream function and potential function is represented as follows;

$$\phi = V_\infty \left(x\cos\theta + y\sin\theta \right); \quad \psi = V_\infty \left(y\cos\theta - x\sin\theta \right) \tag{3.4.13}$$

The flow nets can be constructed by assuming different values of constants in Eq. (3.4.11) and with different angle θ as shown in Fig. 3.4.3. The circulation in a uniform flow along a closed curve is zero which gives the justification that the uniform flow is irrotational in nature.

$$\Gamma = \oint_C \vec{V} \cdot d\vec{s} = \vec{V}_\infty \oint_C d\vec{s} = 0 \tag{3.4.13}$$

Source/Sink Flow

Consider a two-dimensional incompressible flow where the streamlines are radially outward from a central point 'O' (Fig. 3.4.4). The velocity of each streamlines varies inversely with the distance from point 'O'. Such a flow is known as *source flow* and its opposite case is the *sink flow*, where the streamlines are directed towards origin. Both the *source and sink* flow are purely radial. Referring to the Fig. 3.4.4, if v_r and v_θ are the components of velocities along radial and tangential direction respectively, then the equations of the streamlines that satisfy the continuity equation $\left(\nabla \cdot \vec{V} = 0 \right)$ are,

$$v_r = \frac{c}{r}; \quad v_\theta = 0 \tag{3.4.14}$$

Here, the constant c can be related to the volume flow rate of the source. If we define Λ as the volume flow rate per unit length perpendicular to the plane, then,

$$\Lambda = (2\pi r) v_r$$
$$\text{or, } v_r = \frac{\Lambda}{2\pi r} \quad \text{and} \quad c = \frac{\Lambda}{2\pi} \tag{3.4.15}$$

The potential function can be obtained by writing the velocity field in terms of cylindrical coordinates. They may be written as,

$$\frac{\partial \phi}{\partial r} = v_r = \frac{\Lambda}{2\pi r};\ \frac{1}{r}\frac{\partial \phi}{\partial \theta} = v_\theta = 0$$
$$\frac{1}{r}\frac{\partial \psi}{\partial \theta} = v_r = \frac{\Lambda}{2\pi r};\ -\frac{\partial \psi}{\partial y} = v_\theta = 0 \qquad (3.4.16)$$

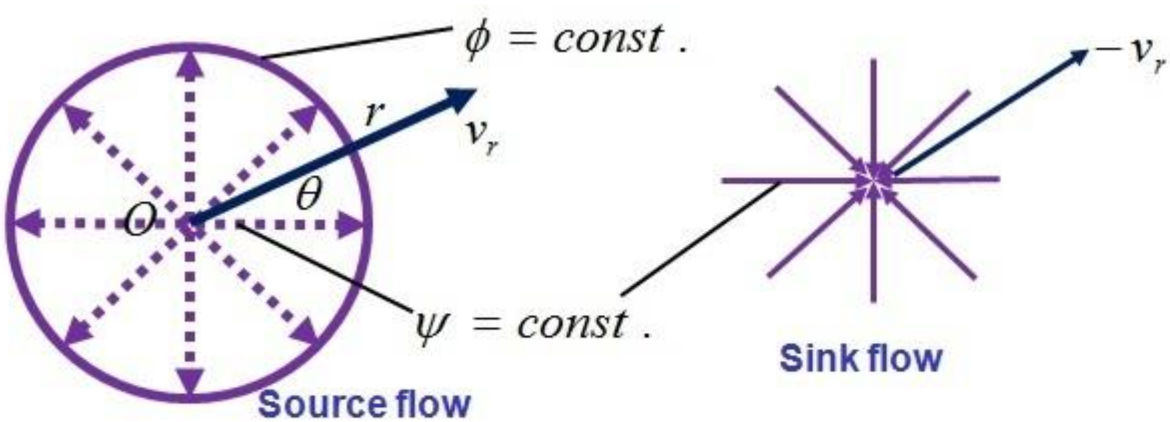


Fig. 3.4.4: Schematic representation of a source and sink flow.

Integrating Eq. (3.4.16) with respect to r and θ, we can get the equation for potential function and stream function for a source and sink flow.

$$\phi = \frac{\Lambda}{2\pi}\ln r + f_3(\theta);\ \phi = C_3 + g_3(r)$$
$$\Rightarrow f_3(\theta) = C_3 \text{ and } g_3(r) = \frac{\Lambda}{2\pi}\ln r$$
$$\Rightarrow \phi = \frac{\Lambda}{2\pi}\ln r + C_3 \qquad (3.4.17)$$
$$\psi = \frac{\Lambda}{2\pi}\theta + f_4(r);\ \psi = C_4 + g_4(\theta)$$
$$\Rightarrow f_4(r) = C_4 \text{ and } g_4(\theta) = \frac{\Lambda}{2\pi}$$
$$\Rightarrow \psi = \frac{\Lambda}{2\pi}\theta + C_4$$

The constant appearing in Eqs (3.4.17) can be dropped to obtain the stream function and potential function.

$$\phi = \frac{\Lambda}{2\pi}\ln r; \quad \psi = \frac{\Lambda}{2\pi}\theta \tag{3.4.18}$$

This equation will also satisfy the Laplace equation in the polar coordinates. Also, it represents the streamlines to be straight and radially outward/inward depending on the source or sink flow while the potential lines are concentric circles shown as flow nets in Fig. 3.4.5. Both the streamlines and equi-potential lines are mutually perpendicular.

It is to be noted from Eq. (3.4.15) that, the velocity becomes infinite at origin $(r=0)$ which is physically impossible. It represents a mathematical singularity where the continuity equation $\left(\nabla \cdot \vec{V}=0\right)$ is not satisfied. We can interpret this point as discrete source/sink of given strength with a corresponding induced flow field about this point. Although the *source and sink* flows do not exist, but many real flows can be approximated at points, away from the origin, using the concept of *source and sink* flow.

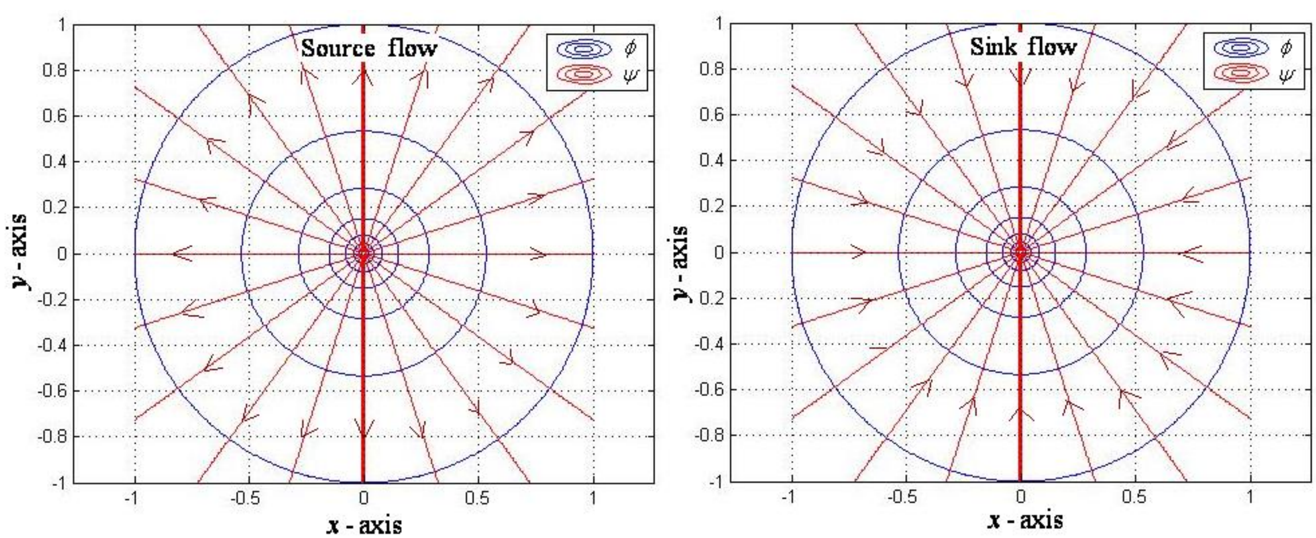


Fig. 3.4.5: Flow nets drawn for of a source and sink flow.

References:

1. Wand D.J., and Harleman D.R. (91964) “Fluid Dynamics”, Addison Wesley.

2. Schlichting, H.: (1976) “Boundary Layer theory”, International Text – Butterworth

3. Lamb, H. (1945) “Hydrodynamics”, International Text – Butterworth

4. Lamb, H.R. (1945) “Hydrodynamics”, Rover Publications

5. Rouse, H. (1957), “Advanced Fluid Mechanics”, John Wiley & Sons, N York

6. White, F.M. (1980) “Viscous Fluid Flow”, McGraw Hill Pub. Co, N York

7. Yalin, M.S.(1971), “Theory of Hydraulic Models”, McMillan Co., 1971.

8. Mohanty A.K. (1994), “Fluid Mechanics”, Prentice Hall of India, N Delhi