LECTURE NOTES

ON

FLUID DYNAMICS

Mr. Shiva Prasad U
Assistant Professor
Dr. Govardhan D
Professor

INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal – 500043, Hyderabad

MODULE - I

Fluid Mechanics

Mechanics : Deals with action of forces on bodies at rest or in motion.

State of rest and Motion: They are relative and depend on the frame of reference. If the position with reference to frame of reference is fixed with time, then the body is said to be in a state of rest. Otherwise, it is said to be in a state of motion.

Scalar and heater quantities: Quantities which require only magnitude to represent them are called scalar quantities. Quantities which acquire magnitudes and direction to represent them are called vectorquantities.

Eg: Mass, time internal, Distance traveled _ Scalars Weight, Displacement, Velocity _ Vectors

Velocity and Speed: Rate of displacement is called velocity and Rate and distance travelled is called Speed.

Unit: m/s

Acceleration: Rate of change of velocity is called acceleration. Negative acceleration is called retardation.

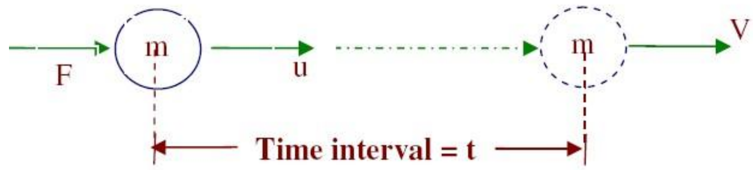
Momentum: The capacity of a body to impart motion to other bodies is called momentum.
The momentum of a moving body is measured by the product of mass and velocity the moving body
Momentum = Mass x Velocity Unit: Kgm/s

Newton's first law of motion: Every body continues to be in its state of rest or uniform motion unless compelled by an external agency.

Inertia: It is the inherent property the body to retain its state of rest or uniform motion.

Force: It is an external agency which overcomes or tends to overcome the inertia of a body. **Newton's second law of motion:** The rate of change of momentum of a body is directly proportional to the magnitudes of the applied force and takes place in the direction of the applied force.

Measurement of force:



Change in momentum in time 't' = mv – mu

$$\text{Rate of change of momentum} = \frac{mv - mu}{t}$$

$$F \propto \frac{mv - mu}{t}$$

$$F \propto m\left(\frac{v-u}{t}\right)$$

$$F \propto ma$$

$$F = K\ ma$$

If F = 1 When m = 1 and u = 1 Then K = 1

F = ma.

Unit: Newton(N)

Mass: Measure of amount of matter contained by the body it is a scalarquantity. Unit:Kg.

Weight: Gravitational force on the body. It is a vector quantity. F = ma

W = mg

Unit:Newton(N) g = 9.81m/s^2

Volume: Measure of space occupied by the body.

Unit: m^3

m^3 = 1000 liters

Work: Work done = Force x Displacement _ Linear motion. Work done = Torque x Angular displacement _ Rotatory motion.

Unit: Nm or J

Energy: Capacity of doing work is called energy. Unit: Nm or J

Potential energy = mgh Kinetic energy = ½ mv^2

Power: Rate of doing work is calledPower.

$$\text{Power:} = \frac{\text{Force x displacement}}{\text{time}}$$

$$= \text{Force x Velocity} \rightarrow \text{Linear Motion.}$$

$$P = \frac{2\Pi N T}{60} \rightarrow \text{Rotatory Motion.}$$

Matter: Anything which possesses mass and requires space to occupy is called matter.

States of matter:

Matter can exist in the following states Solid state.

Solidstate: In case of solids intermolecular force is very large and hencemolecules

are not free to move. Solids exhibit definite shape and volume. Solids undergo certain amount of deformation and then attain state of equilibrium when subjected to tensile, compressive and shear

FluidState: Liquids and gases together are called fluids. Incase ofliquids

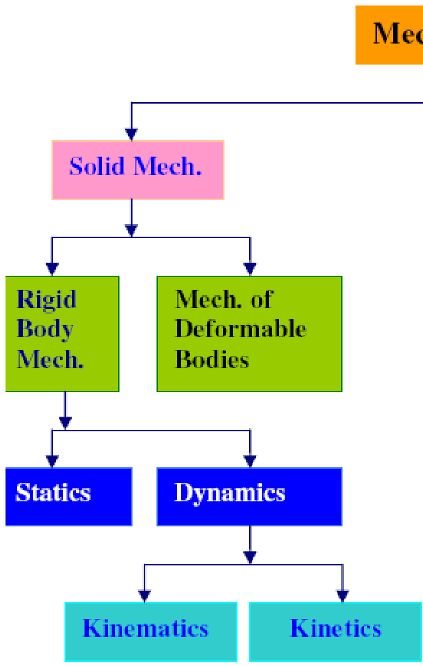
Intermolecular force is comparatively small. Therefore liquids exhibit definite volume. But they assume the shape of thecontainer

Liquids offer very little resistance against tensile force. Liquids offer maximum resistance against compressive forces. Therefore, liquids are also called incompressible fluids. Liquids undergo continuous or prolonged angular deformation or shear strain when subjected to tangential force or shear force. This property of the liquid is called flow of liquid. Any substance which exhibits the property of flow is called fluid. Therefore liquids are considered as fluids.

In case of gases intermolecular force is very small. Therefore the molecules are free to move along any direction. Therefore gases will occupy or assume the shape as well as the volume of the container.

Gases offer little resistance against compressive forces. Therefore gases are called compressible fluids. When subjected to shear force gases undergo continuous or prolonged angular deformation or shear strain. This property of gas is called flow of gases. Any substance which exhibits the property of flow is called fluid. Therefore gases are also considered asfluids.

Branches of Mechanics:



I. Fluid Statics deals with action of forces on fluids at rest or inequilibrium.

II. Fluid Kinematics deals with geometry of motion of fluids without considering the cause of motion

Properties of fluids:

1. Mass density or Specific mass(ρ):

Mass density or specific mass is the mass per unit volume of the fluid.

$$\rho = \frac{Mass}{Volume}$$

$$\rho = \frac{M}{V} \text{ or } \frac{dM}{dV}$$

2. Weight density or Specific weight(ϒ):

Weight density or Specific weight of a fluid is the weight per unit volume. Unit: kg/m3 or kgm3

With the increase in temperature volume of fluid increases and hence mass density decreases. In case of fluids as the pressure increases volume decreases and hence mass density increase

3. Specific gravity or Relative density(S):

It is the ratio of specific weight of the fluid to the specific weight of a standard fluid.

$$S = \frac{\gamma \text{ of fluid}}{\gamma \text{ of standard fluid}}$$

Unit: It is a dimensionless quantity and has no unit.

In case of liquids water at 4^0C is considered as standard liquid.

γ (specific weight) of water at 4°C (standard liquid) is 9.81 $\frac{kN}{m^3}$ or 9.81 x 10^3 $\frac{kN}{m^3}$

Note: We have

1. $S = \frac{\gamma}{\gamma_{standard}}$

   $\therefore \quad \gamma = S \times \gamma_{standard}$

2. $S = \frac{\gamma}{\gamma_{standard}}$

   $S = \frac{\rho \times g}{\rho_{standard} \times g}$

   $S = \frac{\rho}{\rho_{standard}}$

Specific gravity or relative density of a fluid can also be defined as the ratio of mass density of the fluid to mass density of the standard fluid. Mass density of standard water is 1000 kg/m3.

4. Specific volume (∀): It is the volume per unit mass of thefluid.

$$\therefore \quad \forall = \frac{Volume}{mass}$$

$$\forall = \frac{V}{M} \ or \ \frac{dV}{dM}$$

Unit: m3/kg

As the temperature increases volume increases and hence specific volume increases. As the pressure increases volume decreases and hence specific volumedecreases.

Effect of temperature on surface tension ofliquids:

In case of liquids, surface tension decreases with increase in temperature. Pressure has no or very little effect on surface tension of liquids.

Problems:

1. Whatisthepressureinsidethedropletofwater0.05mmindiameterat20^{0}Cifthepressure

outside the droplet is 103 kPa Take σ = 0.0736 N/m at 20^{0}C.

$$p = \frac{4\sigma}{D}$$

$$= \frac{4x0.0736}{0.05x10^{-3}}$$

$$p = 5.888x10^{+3} N/m^2$$

$$p = p_{inside} - p_{outside}$$

$$p_{inside} = (5.888 + 103)10^3$$

$$p_{inside} = 108.88x10^3 Pa$$

$$p_{inside} = ?$$

$$D = 0.05x10^{-3} m$$

$$p_{outside} = 103kPa$$

$$= 103x10^3 N/m^2$$

$$\sigma = 0.0736 N/m$$

2. liquid bubble 2cm in radius has an internal pressure of 13Pa. Calculate the surface tension of liquid film.

$p = \frac{8\sigma}{D}$

$\sigma = \frac{13x4x10^{-2}}{8}$

$\sigma = 0.065 N/m$

$R = 2cm$

$D = 4cm$

$= 4x10^{-2} m$

$p = 13Pa(N/m^2)$

Compressibility:

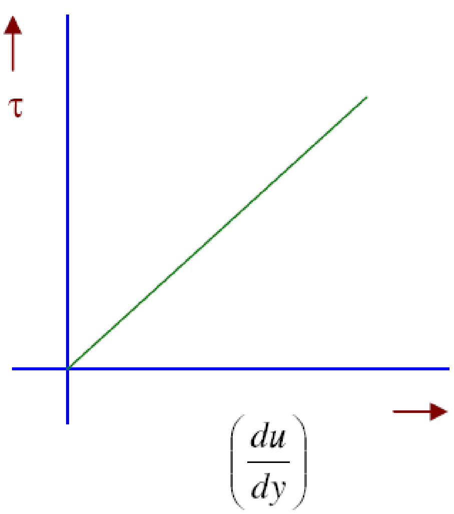
It is the property by virtue of which there will be change in volume of fluid due to change in pressure.

Rheological classification of fluids: (Rheology _ Study of stress – strain behavior).

1. **Newtonian fluids:** A fluid which obeys Newton's law of viscosity i.e., t = μ. *du/dy*is called Newtonian fluid. In such fluids shear stress varies directly as shear strain.

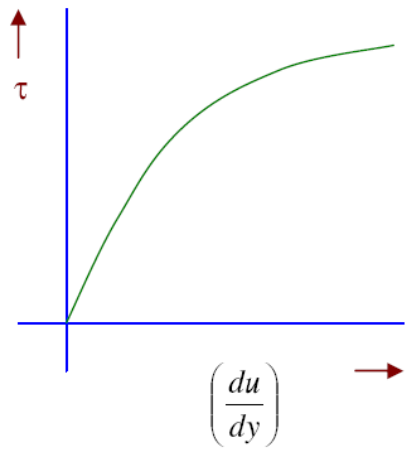
In this case the stress strain curve is a stress line passing through origin the slope of the line gives dynamic viscosity of the fluid.

Eg: Water, Kerosene.



3. **Non- Newtonian fluid:** A fluid which does not obey Newton's law of viscosity is called non-Newton fluid. For suchfluids,

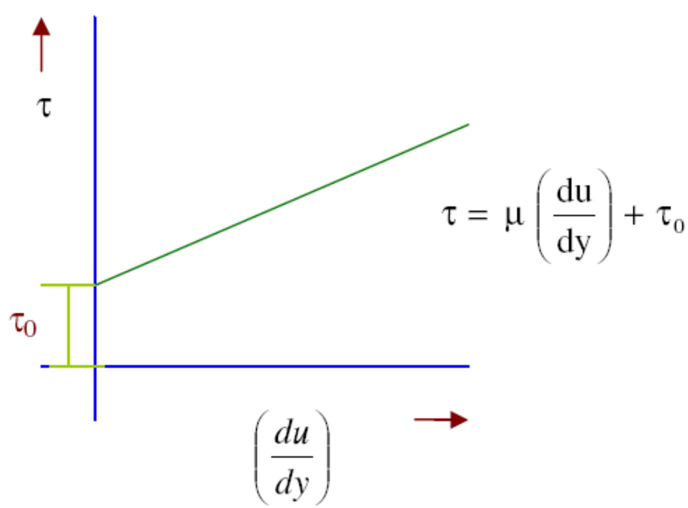
$$\tau = \mu . \left(\frac{du}{dy}\right)^n$$



3. Ideal Plasticfluids:

In this case the strain starts after certain initial stress (t0) and then the stress strain relationship will be linear. t0is called initial yield stress. Sometimes they are also called Bingham's Plastics.

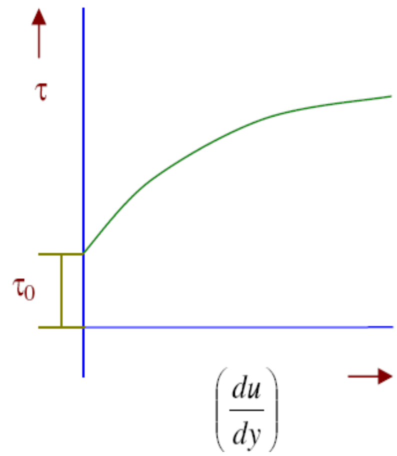
Eg: Industrial sludge.



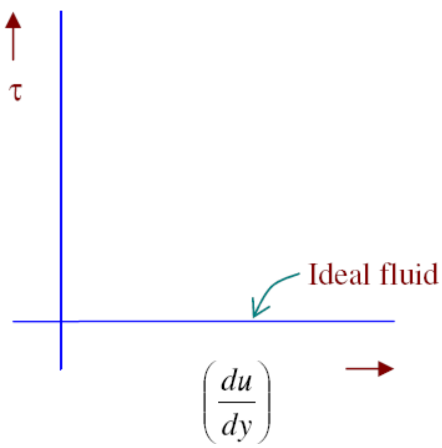
4. Thixotropicfluids:

These require certain amount of yield stress to initiate shear strain. After wards stress-strain relationship will be non – linear.

Eg; Printers ink.

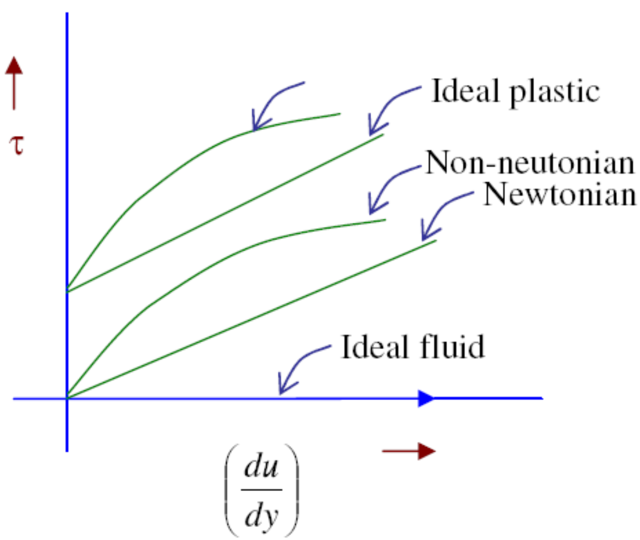


Ideal fluid: Any fluid for which viscosity is assumed to be zero is called Ideal fluid. Forideal fluid t = 0 for allvaluesof du/dy

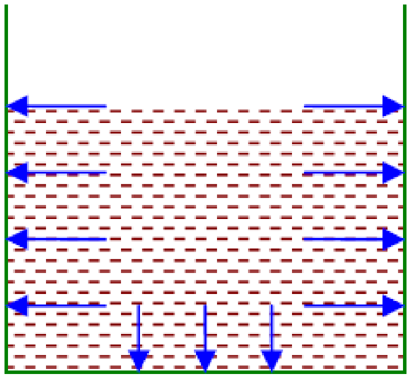


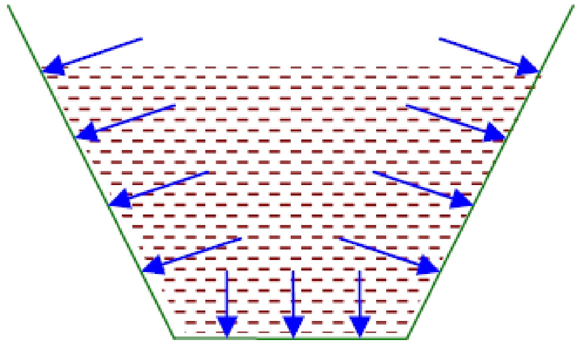
5. Real fluid :

Any fluid which posses certain viscosity is called real fluid. It can be Newtonian or non – Newtonian, thixotropic or ideal plastic.



PRESSURE AND ITS MEASUREMENTS:





Fluid is a state of matter which exhibits the property of flow. When a certain mass of fluids is held in static equilibrium by confining it within solid boundaries, it exerts force along direction perpendicular to the boundary in contact. This force is called fluid pressure.

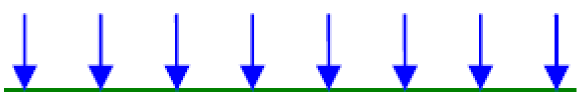
• **Pressuredistribution:**

It is the variation of pressure over the boundary in contact with the fluid. There are two types of pressure distribution.

a) Uniform Pressuredistribution.

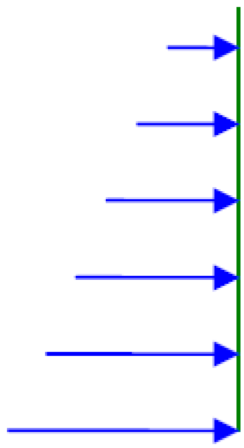
b) Non-Uniform Pressuredistribution.

(a) Uniform Pressuredistribution:



If the force exerted by the fluid is same at all the points of contact boundary then the pressure distribution is said to be uniform.

(b) Non –Uniform Pressuredistribution:



If the force exerted by the fluid is not same at all the points then the pressure distribution is said to be non-uniform.

Intensity of pressure or unit pressure orPressure:

Intensity of pressure at a point is defined as the force exerted over unit area considered around that point. If the pressure distribution is uniform then intensity of pressure will be same at all the points.

Calculation of Intensity of Pressure:

When the pressure distribution is uniform, intensity of pressure at any points is given by the ratio of total force to the total area of the boundary in contact.

Intensity of Pressure 'p' $=F/A$

When the pressure distribution is non- uniform, then intensity of pressure at a point is given by dF/dA.

Unit of Intensity of Pressure: N/m^2 or pascal (Pa).

Note: 1 MPa $=1N/mm^2$

• **Atmosphericpressure**

Air above the surface of liquids exerts pressure on the exposed surface of the liquid and normal to the surface.

This pressure exerted by the atmosphere is called atmospheric pressure.

Atmospheric pressure at a place depends on the elevation of the place and the temperature. Atmospheric pressure is measured using an instrument called 'Barometer' and hence atmospheric pressure is also called Barometric pressure.

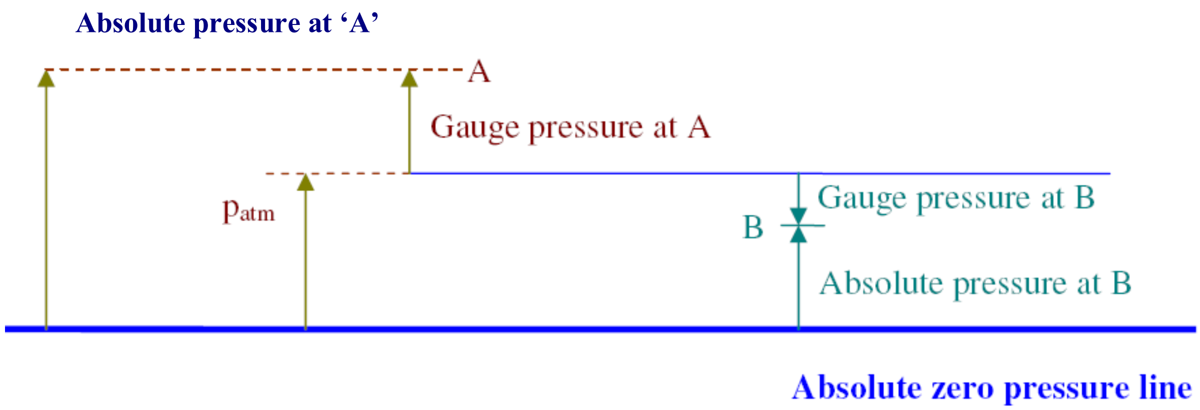
Unit: kPa .

'bar' is also a unit of atmospheric pressure 1bar = 100 kPa.

Absolute pressure and GaugePressure:

Absolute pressure at a point is the intensity of pressure at that point measured with reference to absolute vacuum or absolute zero pressure.

Absolute pressure at a point can never be negative since there can be no pressure less than absolute zero pressure.



Absolute pressure at a point is the intensity of pressure at that point measured with reference to absolute vacuum or absolute zero pressure.

Absolute pressure at a point can never be negative since there can be no pressure less than absolute zero pressure.

If the intensity of pressure at a point is measured with reference to atmospheric pressure, then it is called gauge pressure at that point.

Gauge pressure at a point may be more than the atmospheric pressure or less than the atmospheric pressure. Accordingly gauge pressure at the point may be positive or negative.

Negative gauge pressure is also called vacuum pressure.

From the figure, It is evident that, Absolute pressure at a point = Atmospheric pressure ± Gauge pressure.

NOTE: If we measure absolute pressure at a Point below the free surface of the liquid, then, p = g. Y + patm

If gauge pressure at a point is required, then atmospheric pressure is taken as zero, then, p = g. Y

Pressure Head

It is the depth below the free surface of liquid at which the required pressure intensity is available.

P = gh

h = *P/* g

For a given pressure intensity 'h' will be different for different liquids since, 'g' will be different for different liquids. Whenever pressure head is given, liquid or the property of liquid like specify gravity, specific weight, mass density should be given.

Eg:

(i) 3m ofwater

(ii) 10m of oil of S =0.8.

(iii) 3m of liquid of g = 15kN/m3

(iv) 760mm ofMercury.

(v) 10m _ notcorrect.

NOTE:

1. To convert head of a liquid to head of anotherliquid.

$$S = \frac{\gamma}{\gamma_{Standard}}$$

$$S_1 = \frac{\gamma_1}{\gamma_{Standard}}$$

$$p = \gamma_1 h_1$$

$$\therefore \gamma_1 = S_1 \, \gamma_{Standard}$$

$$p = \gamma_2 h_2$$

$$\gamma_{21} = S_2 \, \gamma_{Standard}$$

$$\boxed{\gamma_1 h_1 = \gamma_2 h_2}$$

$$\therefore S_1 \, \gamma_{Standard} \, h_1 = S_2 \, \gamma_{Standard} \, h_2$$

$$\boxed{S_1 h_1 = S_2 h_2}$$

2. Swater x hwater = Sliquidxhliquid 1x hwater = Sliquidxhliquid

h water = Sliquid x hliquid

Pressure head in meters of water is given by the product of pressure head in meters of liquid and specific gravity of the liquid.

Eg: 10meters of oil of specific gravity 0.8 is equal to 10x0.8 = 8 meters of water. Eg: Atmospheric pressure is 760mm of Mercury.

NOTE:

P = g h

kPa kN/m^3 m

Problem:

1. Calculate intensity of pressure due to a column of 0.3m of (a) water (b)Mercury

(c) Oil of specific gravity-0.8.

a) h = 0.3m of water

$$\gamma = 9.81 \frac{kN}{m^3}$$

$$p = ?$$

$$p = \gamma h$$

$$p = 2.943 \text{ kPa}$$

c) h = 0.3 of Hg

$$\gamma = 13.6 \times 9.81$$

$$\gamma = 133.416 \text{ kN/m}^3$$

$$p = \gamma h$$

$$= 133.416 \times 0.3$$

$$p = 40.025 \text{ kPa}$$

2. Intensity of pressure required at a points is 40kPa. Find corresponding headin
(a) water (b) Mercury (c) oil of specificgravity-0.9.

(a) p = 40 kPa

$$h = \frac{p}{\gamma}$$

h = 4.077 m of water

$$\gamma = 9.81 \frac{kN}{m^3}$$

h = ?

(b) p = 40 kPa

$\gamma = (13.6x9.81\ N/m^3$

$$\gamma = 133.416\ \frac{KN}{m^3}$$

$$h = \frac{p}{\gamma}$$

h = 0.299 m of Mercury

$$h = \frac{p}{\gamma}$$

c) p = 40 kPa

h = 4.53 m of oil S = 0.9

$\gamma = 0.9 \text{ x } 9.81$

$$\gamma = 8.829\ \frac{KN}{m^3}$$

4. Standard atmospheric pressure is 101.3 kPa Find the pressure head in (i) Meters of water (ii) mm of mercury (iii) m of oil of specific gravity0.8.

(i) $p = \gamma h$

101.3 = 9.81 x h

h = 10.3 m of water

(ii) $p = \gamma h$

101.3 = (13.6x9.81) x h

h = 0.76 m of mercury

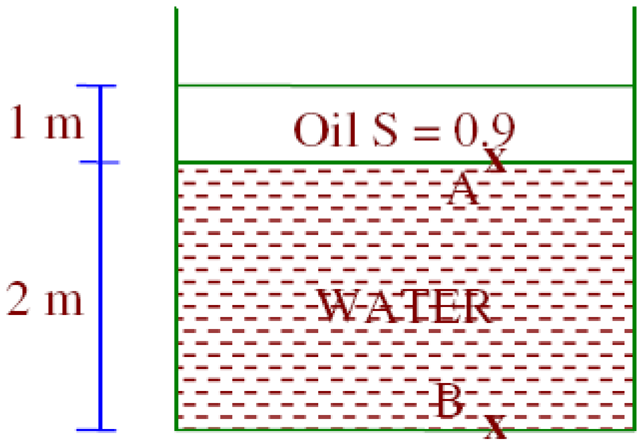
(iii) $p = \gamma h$

101.3 = (0.8 x 9.81 x h

h = 12.9 m of oil of S = 0.8

5. An open container has water to a depth of 2m and above this an oil of S = 0.9 for a depth of 1m.

Find the intensity of pressure at the interface of two liquids and at the bottom of thetank.



$p_A = \gamma_{a1} \; h_{oil}$

$= (0.9 \text{ x } 9.81) \text{ x } 1$

$p_A = 8.829 \text{ kPa}$

$p_B = \gamma_{oil} \; xh_{oil} + \gamma_{water} + h_{water}$

$p_A = 8.829 \text{kPa} + 9.81 \text{ x } 2$

$p_B = 28.45 \text{kPa}$

6. Convert the following absolute pressure to gauge pressure (a) 120kPa (b) 3kPa (c) 15m of H2o (d) 800mm ofHg.

(a) $p_{abs} = p_{atm} + p_{gauge}$

$\therefore\ p_{gauge} = p_{abs} - p_{atm} = 120 - 101.3 = 18.7$ kPa

(b) $p_{gauge} = 3\text{-}101.3 = \text{-}98.3$ kPa

$p_{gauge} = 98.3$ kPa (vacuum)

(c) $h_{abs} = h_{atm} + h_{gauge}$

$15 = 10.3 + h_{gauge}$

$h_{gauge} = 4.7$m of water

(d) $h_{abs} = h_{atm} + h_{gauge}$

$800 = 760 + h_{gauge}$

$h_{gauge} = 40$ mm of mercury

Measurement of Pressure

Various devices used to measure fluid pressure can be classified into,

1. Manometers
2. Mechanicalgauges.

Manometers are the pressure measuring devices which are based on the principal of balancing the column of the liquids whose pressure is to be measured by the same liquid or another liquid. Mechanical gauges consist of an elastic element which deflects under the action of applied pressure and this movement will operate a pointer on a graduated scale.

Classification of Manometers:

Manometers are broadly classified into

a) SimpleManometers

b) DifferentialManometers.

a) SimpleManometers

Simple monometers are used to measure intensity of pressure at a point.

They are connected to the point at which the intensity of pressure is required. Such a point is called gauge point.

b) DifferentialManometers

Differential manometers are used to measure the pressure difference between two points. They are connected to the two points between which the intensity of pressure is required.

Types of Simple Manometers

Common types of simple manometers are

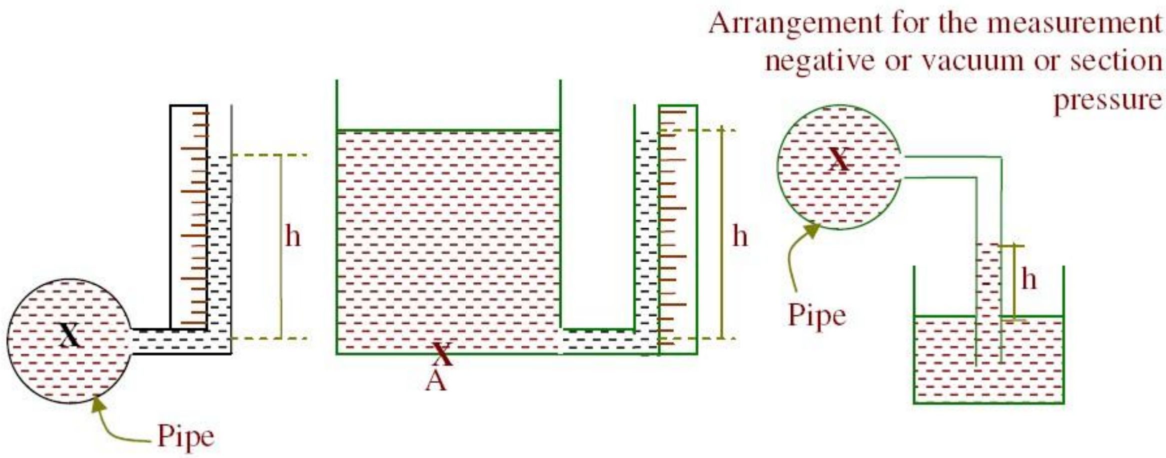
a) Piezometers

b) U-tubemanometers

c) Single tubemanometers

d) Inclined tubemanometers

a) Piezometers:



Piezometer consists of a glass tube inserted in the wall of the vessel or pipe at the level of point at which the intensity of pressure is to be measured. The other end of the piezometer is exposed to air. The height of the liquid in the piezometer gives the pressure head from which the intensity of pressure can be calculated.

To minimize capillary rise effects the diameters of the tube is kept more than 12mm.

Merits

_ Simple in construction

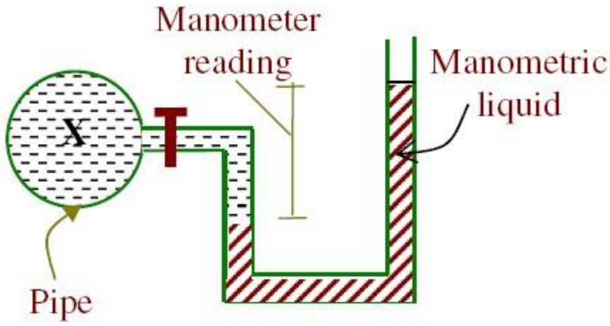
_ Economical

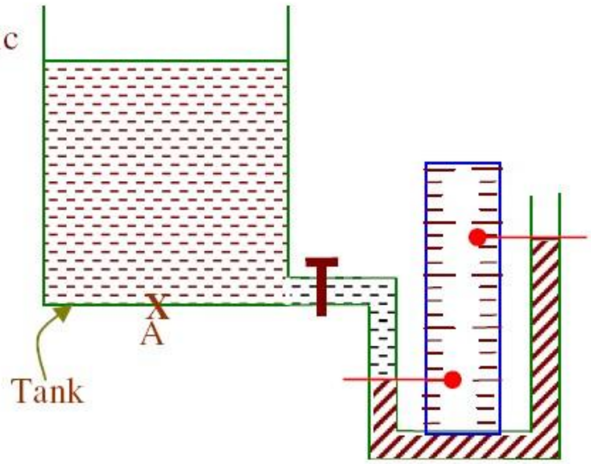
Demerits

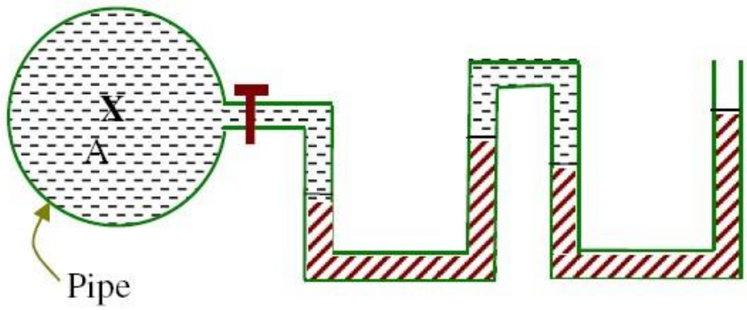
_ Not suitable for high pressure intensity.

_ Pressure of gases cannot be measured.

(b) U-tubeManometers:







A U-tube manometers consists of a glass tube bent in U-Shape, one end of which
is connected to gauge point and the other end is exposed to atmosphere. U-tube consists of a liquid of specific of gravity other than that of fluid whose pressure intensity is to be measured and is called manometricliquid.

• **Manometricliquids**

¨ Manometric liquids should neither mix nor have any chemical reaction with the fluid whose pressure intensity is to be measured.

¨ It should not undergo any thermal variation.

¨ Manometric liquid should have very low vapour pressure.

¨ Manometric liquid should have pressure sensitivity depending upon the magnitude of pressure to be measured and accuracy requirement.

• **To write the gauge equation formanometers**

Gauge equations are written for the system to solve for unknown quantities.

Steps:

1. Convert all given pressure to meters of water and assume unknown pressure in meters ofwaters.

2. Starting from one end move towards the other observing the followingpoints.

¨ Any horizontal movement inside the same liquid will not cause change in pressure.

¨ Vertically downward movement causes increase in pressure and upward motion causes decrease in pressure.

¨ Convert all vertical columns of liquids to meters of water by multiplying them by corresponding specify gravity.

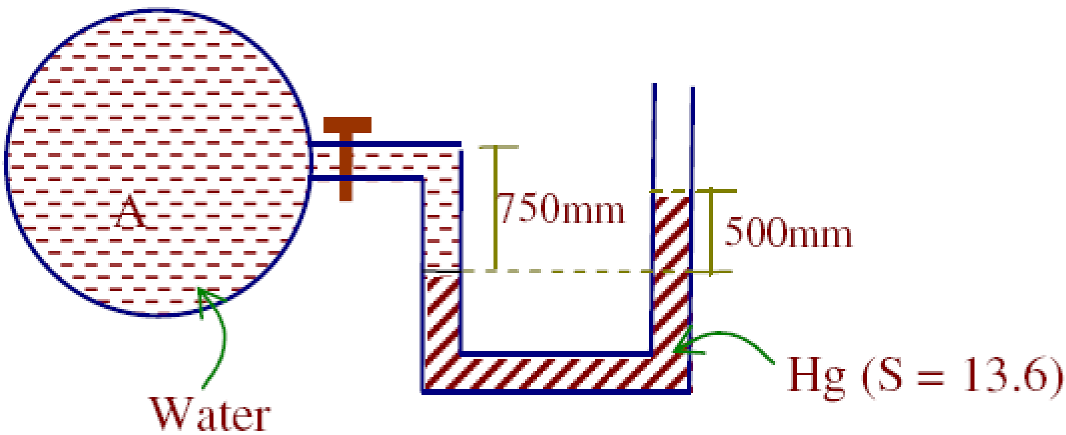
¨ Take atmospheric pressure as zero (gauge pressure computation).

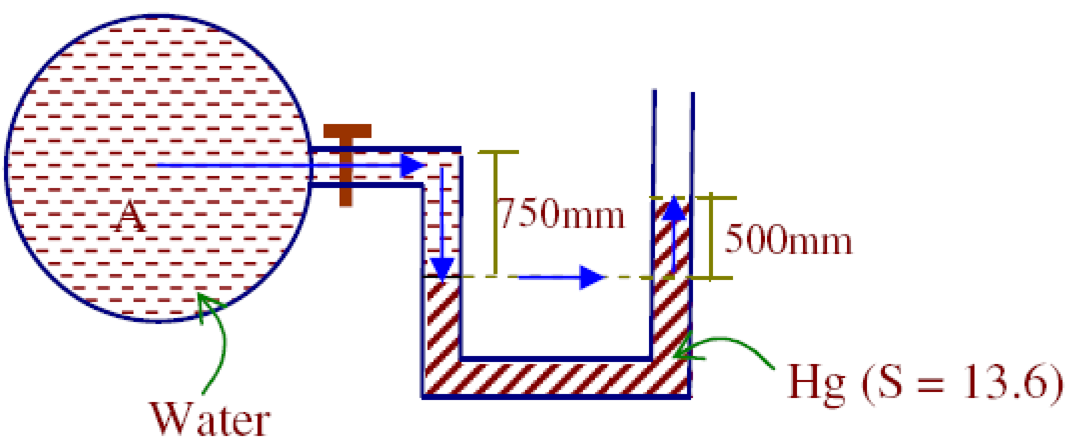
3. Solve for the unknown quantity and convert it into the requiredunit.

4. If required calculate absolutepressure.

Problem:

1. Determine the pressure at A for the U- tube manometer shown in fig. Also calculate the absolute pressure at A inkPa.





Let 'h_A' be the pressure head at 'A' in 'meters of water'.

$$h_A + 0.75 - 0.5 x 13.6 = 0$$

$$h_A = 6.05 m\ of\ \ water$$

$$p = \gamma h$$

$$= 9.81 x 6.05$$

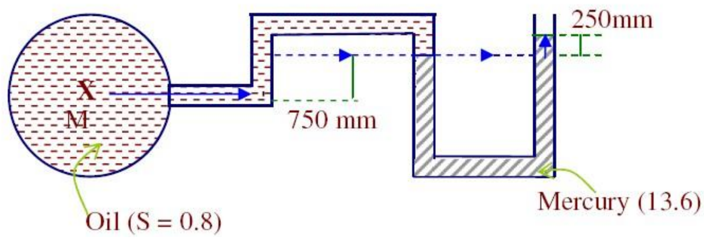
$$p = 59.35\ kPa (gauge\ \ pressure)$$

$$p_{abs} = p_{atm} + p_{gauge}$$

$$= 101.3 + 59.35$$

$$p_{abc} = 160.65\ kPa$$

2. For the arrangement shown in figure, determine gauge and absolute pressure atthe pointM.



Let 'h_M' be the pressure head at the point 'M' in m of water,

h_M - 0.75 x 0.8 – 0.25 x 13.6 = 0

$h_{M =}$ 4 m of water

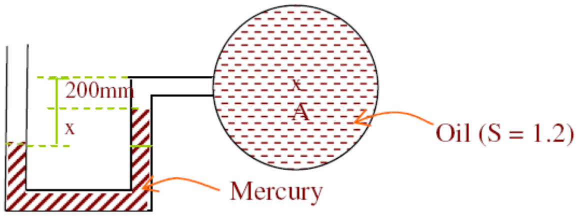
$$p = \gamma h$$

$$p = 39.24\,kPa$$

$$p_{abs} = 101.3 + 39.24$$

$$p_{abs} 140.54\,kPa$$

3. If the pressure at 'At' is 10 kPa (Vacuum) what is the value of'x'?



p_A = 10 kPa (Vacuum)

p_A = - 10 kPa

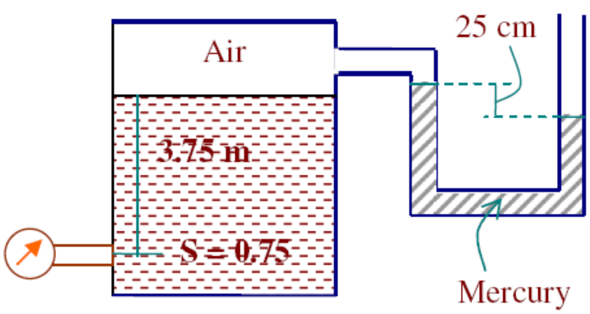
$$\frac{p_A}{\gamma} = \frac{-10}{9.81} = -1.019 \text{ m of water}$$

$$h_A = -1.019 \text{ m of water}$$

$$-1.019 + 0.2 \text{ x } 1.2 + \text{x } (13.6) = 0$$

$$\text{x} = 0.0572 \text{ m}$$

4. The tank in the accompanying figure consists of oil of S = 0.75. Determine the pressure gauge reading in*kN/m*^{2}.



Let the pressure gauge reading be 'h' m of water

h – 3.75 x 0.75 + 0.25 x 13.6 = 0

h = - 0.5875 m of water

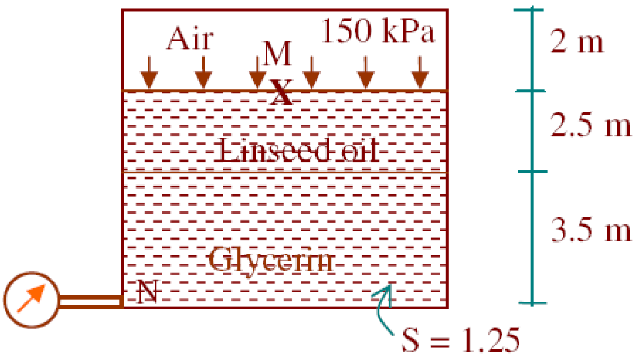
p = γ h

p = -5.763 kPa

p = 5.763 kPa (Vacuum)

5. A closed tank is 8m high. It is filled with Glycerine up to a depth of 3.5m and linseed oil to another 2.5m. The remaining space is filled with air under a pressure of 150 kPa. If a pressure gauge is fixed at the bottom of the tank what will be itsreading.

Also calculate absolute pressure. Take relative density of Glycerine and Linseed oil as 1.25 and 0.93 respectively.



$$P_H = 150\,\text{kPa}$$

$$h_M = \frac{150}{9.81}$$

$$h_M = 15.29\,\text{m of water}$$

Let 'h_N' be the pressure gauge reading in m of water.

h_N -3.5 x 1.25 -2.5 x 0.93 =15.29

h_N = 21.99 m of water

p = 9.81 x 21.99

p = 215.72 kPa (gauge)

p_{abs} = 317.02 kPa

DIFFERENTIAL MANOMETERS

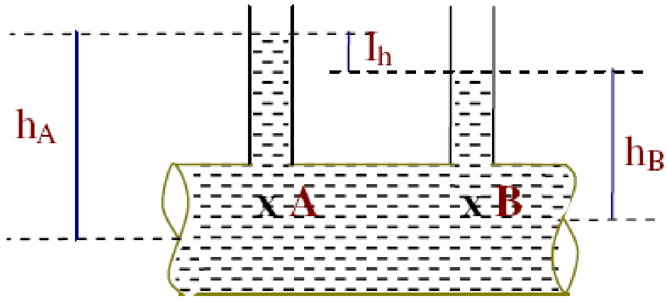
Differential manometers are used to measure pressure difference between any two points. Common varieties of differential manometers are:

(a) Two piezometers.

(b) Inverted U-tubemanometer.

(c) U-tube differentialmanometers.

(d) Micromanometers.

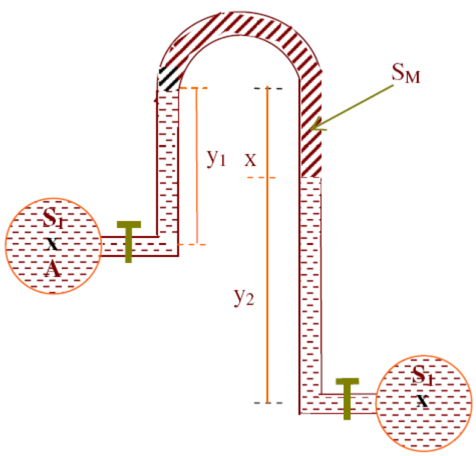


(a) TwoPizometers

The arrangement consists of two pizometers at the two points between which the pressure difference is required. The liquid will rise in both the piezometers. The difference in elevation of liquid levels can be recorded and the pressure difference can becalculated.

It has all the merits and demerits of piezometer.

(b) Inverted U-tube manometers



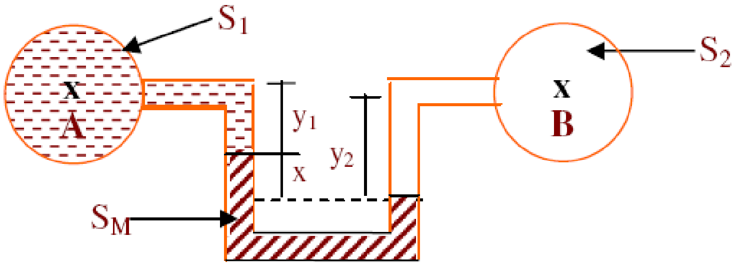
Inverted U-tube manometer is used to measure small difference in pressure between any two points. It consists of an inverted U-tube connecting the two points between which the pressure difference is required. In between there will be a lighter manometric liquid. Pressure difference between the two points can be calculated by writing the gauge equations for the system.

Let 'hA' and 'hB' be the pr head at 'A' and 'B' in meters of water

hA – (Y1 S1) + (x SM) + (y2 S2) = hB. hA – hB = S1 y1 – SM x – S2 y2,

pA – pB = g (hA – hB)

(c) U-tube Differentialmanometers



A differential U-tube manometer is used to measure pressure difference between any two points. It consists of a U-tube containing heavier manometric liquid, the two limbs of which are connected to the gauge points between which the pressure difference is required. U-tube differential manometers can also be used for gases. By writing the gauge equation for the system pressure difference can be determined.

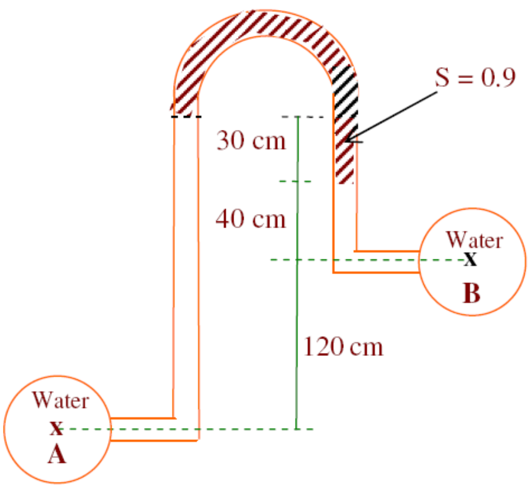
Let 'hA' and 'hB' be the pressure head of 'A' and 'B' in meters of water

hA + S1 Y1 + x SM – Y2 S2 = hBhA – hB = Y2 S2 – Y1 S1– x SM

Problems

(1) An inverted U-tube manometer is shown in figure. Determine the pressure difference between A and B inN/m^2

Let hA and hBbe the pressure heads at A and B in meters of water.



$h_A - (190 \times 10^{-2}) + (0.3 \times 0.9) + (0.4)\,0.9 = h_B$

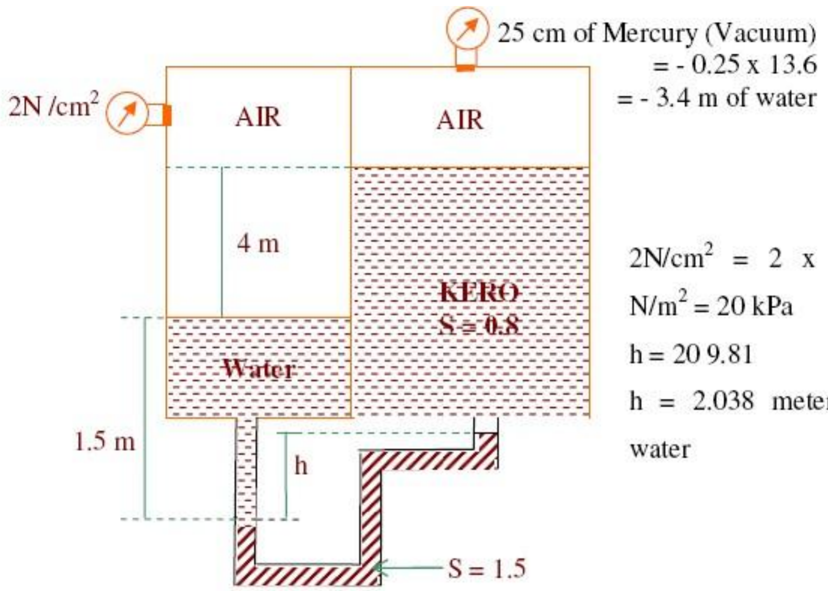
$h_A - h_B = 1.23$ meters of water

$p_A - p_B = \gamma\,(h_A - h_B) = 9.81 \times 1.23$

$p_A - p_B = 12.06$ kPa

$p_A - p_B = 12.06 \times 10^3\ N/m^2$

2. In the arrangements shown in figure. Determine the ho'h'.



$2N/cm^2 = 2 \times 100^2\ N/m^2 = 20$ kPa

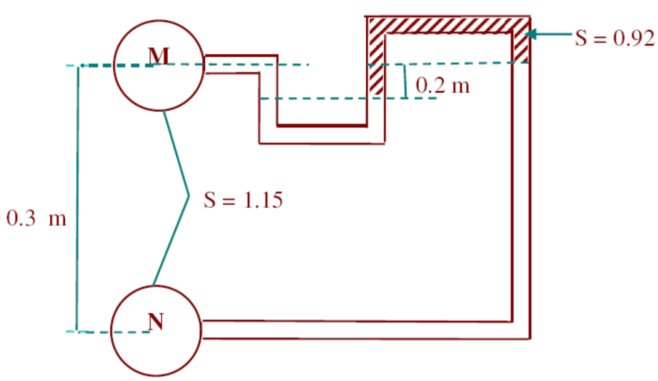
h = 20 9.81

h = 2.038 meters of water

$2.038 + 1.5 - (4 + 1.5 - h)\,0.8 = -3.4$

h = 3.6 m

3. Compute the pressure different between 'M' and 'N' for the system shown infigure.



Let 'h_M' and 'h_N' be the pressure heads at M and N in m of water.

hm + y x 1.15 – 0.2 x 0.92 + (0.3 – y + 0.2) 1.15 = hn

hm + 1.15 y – 0.184 + 0.3 x 1.15 – 1.15 y + 0.2 x 1.15 = hn

hm + 0.391 = hn

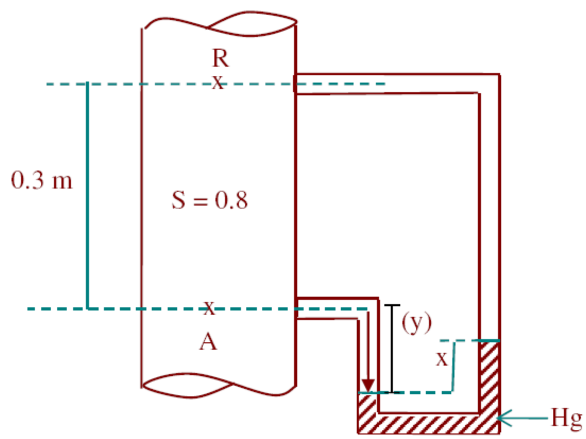
hn – hm = 0.391meters of water

$p_n - p_m = \gamma (h_N - h_m)$

$= 9.81 \times 0.391$

$p_n - p_m = 3.835$ kPa

4. Petrol of specify gravity 0.8 flows up through a vertical pipe. A and B are the two points in the pipe, B being 0.3 m higher than A. Connection are led from A and B to a U–tube containing Mercury. If the pressure difference between A and B is 18 kPa, find the reading ofmanometer.



$p_A - p_B = 18$kPa

$$\frac{P_A - P_B}{\gamma}$$

$$h_A - h_B = \frac{18}{9.81}$$

$$h_A - h_B = 1.835 m\ of\ water$$

h_A + y x 0.8 – x 13.6 – (0.3 + y – x) 0.8 = h_B

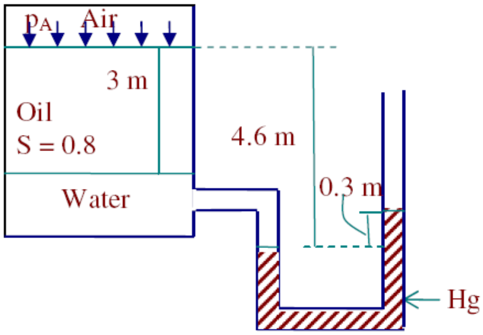
$h_A - h_B$ = – 0.8y + 13.66 x + 0.24 + 0.8 y – 0.8 x

$$h_A - h_B = 12.8x + 0.24$$

1.835 = 12.8x + 0.24

x = 0.1246 m

4. What is the pressure pA in the fig given below? Take specific gravity of oil as0.8.



$h_A + (3 \times 0.8) + (4.6 - 0.3)(13.6) = 0$

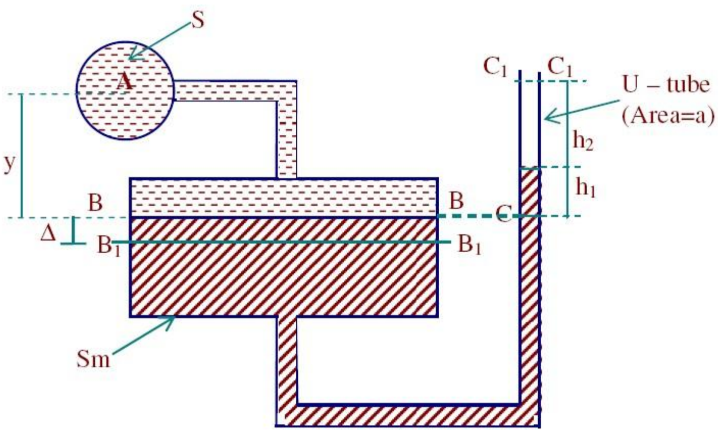
$h_A = 2.24$ m of oil

$p_A = 9.81 \times 2.24$

$p_A = 21.97$ kPa

SINGLE COLUMN MANOMETER:

Single column manometer is used to measure small pressure intensities.



A single column manometer consists of a shallow reservoir having large cross sectional area when compared to cross sectional area of U – tube connected to it. For any change in pressure, change in the level of manometric liquid in the reservoir is small and change in level of manometric liquid in the U-tube is large.

To derive expression for pressure head at A:

BB and CC are the levels of manometric liquid in the reservoir and U-tube before connecting the point A to the manometer.

Let the point A be connected to the manometer. B1B1 and C1 C1 are the levels of manometric liquid. Volume of liquid between BBB1B1 = Volume of liquid between

Let the point A be connected to the manometer. B1B1 and C1 C1 are the levels of manometric liquid. Volume of liquid between BBB1B1 = Volume of liquid between

CCC1C1

$$A\Delta = a\, h_2$$

$$\Delta = \frac{ah_2}{A}$$

Let 'h_A' be the pressure head at A in m of water.

$$h_A + (y + \Delta)\, S - (\Delta + h_1 + h_2)\, Sm = 0$$

$$h_A = (\Delta + h_1 + h_2)\, Sm - (y + \Delta)\, S$$

$$= \Delta\, Sm + \underline{h_1\, Sm} + h_2\, Sm - \underline{yS} - \Delta S$$

$$h_A = \Delta\, (Sm - S) + h_2\, Sm$$

$$h_A = \frac{ah_2}{A}\, (Sm - S) + h_2\, Sm$$

$\therefore$ It is enough if we take one reading to get 'h_2' If '$\frac{a}{A}$' is made very small (by increasing 'A') then the I term on the RHS will be negligible.

$$\text{Then } h_A = h_2\, Sm$$

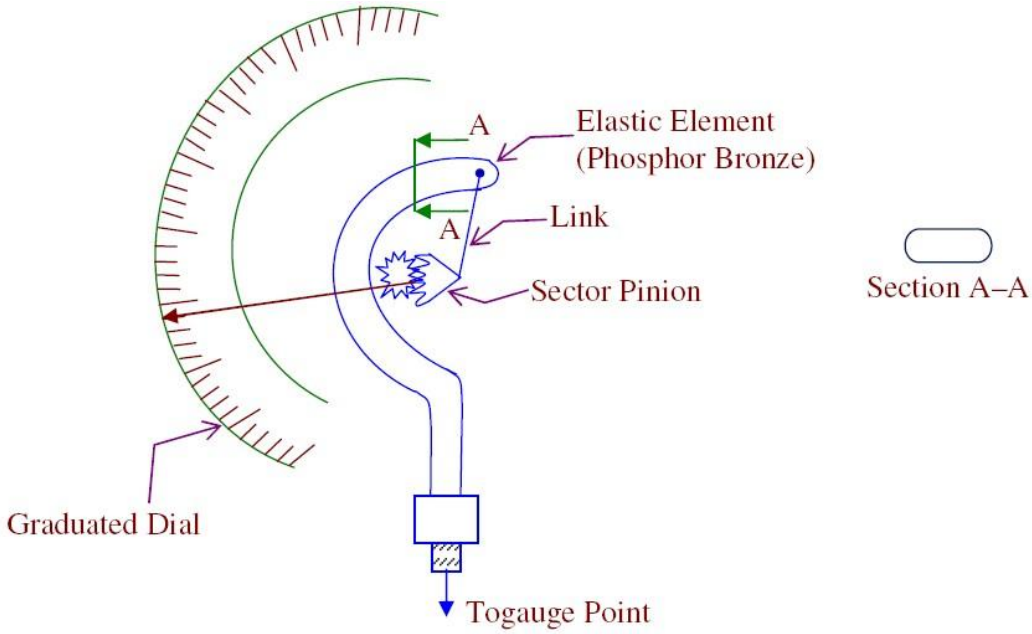
MECHANICAL GAUGES:

Pressure gauges are the devices used to measure pressure at a point.

They are used to measure high intensity pressures where accuracy requirement is less.

Pressure gauges are separate for positive pressure measurement and negative pressure measurement. Negative pressure gauges are called Vacuumgauges.

BASIC PRINCIPLE:



Mechanical gauge consists of an elastic element which deflects under the action of applied pressure and this deflection will move a pointer on a graduated dial leading to the measurement of pressure. Most popular pressure gauge used is Bordon pressure gauge.

The arrangement consists of a pressure responsive element made up of phosphor bronze or special steel having elliptical cross section. The element is curved into a circular arc, one end of the tube is closed and free to move and the other end is connected to gauge point. The changes in pressure cause change in section leading to the movement. The movement is transferred to a needle using sector pinion mechanism. The needle moves over a graduated dial.

MODULE -II

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 themotion.

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. Lagarangianapproach
2. Eularianapproach

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 particleimagery.

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 definedas

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,

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

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 flowcondition.

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

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

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 ofco-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.

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

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 turbulentflow.

Example 1 :

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

$$V = 2y^2\hat{i} + 3x\hat{j} + 0\hat{k}$$

Given velocity field, $u = 2y\,2$; $v = 3x$; $w = 0$so,

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 observethat

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

c) Alsofromtheaboveequationwehavetheaccelerationcomponentasfollow

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

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

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

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

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.

Letafluidparticleexactlypositionedatpoint*A*movingtoanotherpoint during time interval Δt. The velocity of the fluid particle is the same as the local velocity at that point as obtained from the Euleriandescription

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

Attime $t + \Delta t$ $\vec{V}$, particleat 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.

$$\vec{a}_{particle} = \frac{d\vec{V}_{particle}}{dt}$$

Acceleration of a fluidparticle:

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

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

Where 5 and*d*arethepartialderivativeoperatorandtotalderivativeoperatorrespectively.

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 Eulerianframe 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 Eulariandescription.

Therefore, the acceleration is defined by

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

In vector form

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

where v is the gradientoperator.

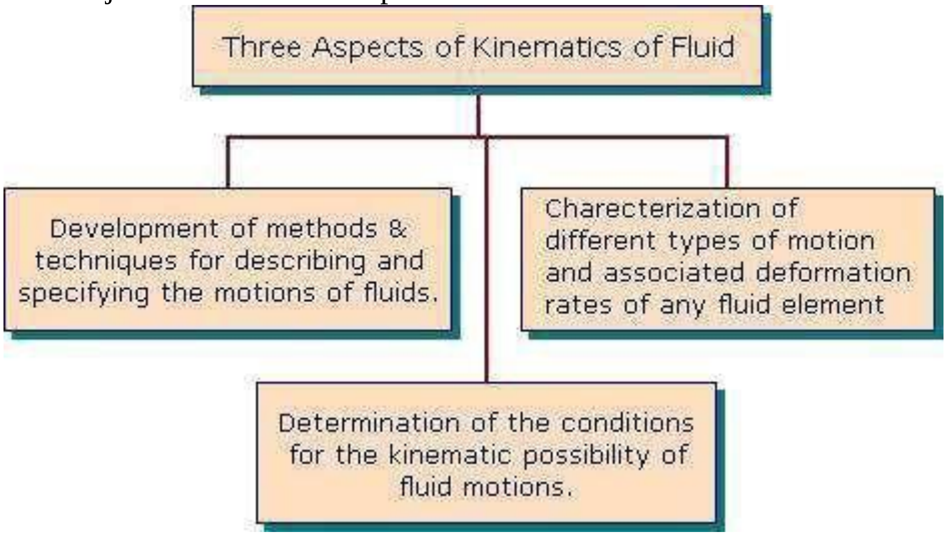
The first term of the right hand side of equation represents the time rate of change of velocity field at the position of the fluid particle at time *t*. This acceleration component is also independent to the change of the particle position and is referred as the local acceleration.\

However the term $\left(\vec{V}.\nabla\right)\vec{V}$ 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 thissituation.

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 inthe direction of flow or in directions perpendicular toit.

2. Non-uniformity in a direction perpendicular to the flow is always encounterednear solid boundaries past which the fluidflows.

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 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

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

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 instantaneousvelocityvector.

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

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

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.

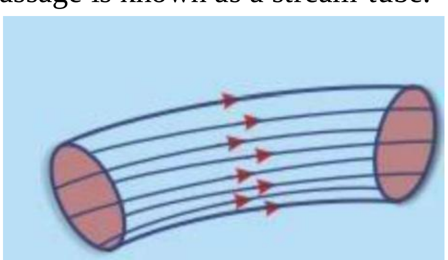


Fig: Stream Tube

Properties of Stream tube:

1. The stream-tube is bounded on all sides bystreamlines.
2. Fluid velocity does not exist across a streamline, no fluid may enter or leave astream-tube except through itsends.
3. The entire flow in a flow field may be imagined to be composed of flows throughstream-tubes arranged in some arbitrarypositions.

Path Lines

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

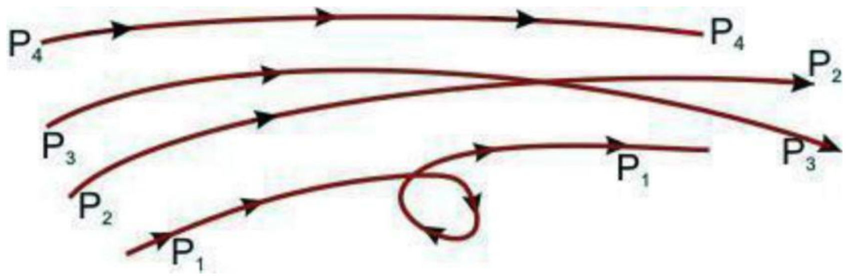


Fig: Path lines

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

PathLine	StreamLine
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 theequation

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

Solving for ,

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

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

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

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 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 fluidelement.

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

c) The sides are displaced in paralleldirection.

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.

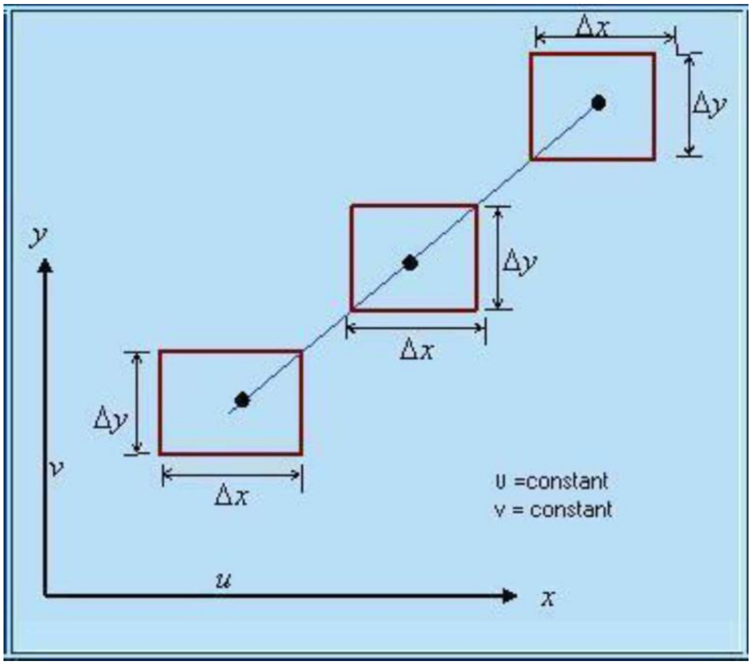


Fig: Fluid Element in pure translation

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

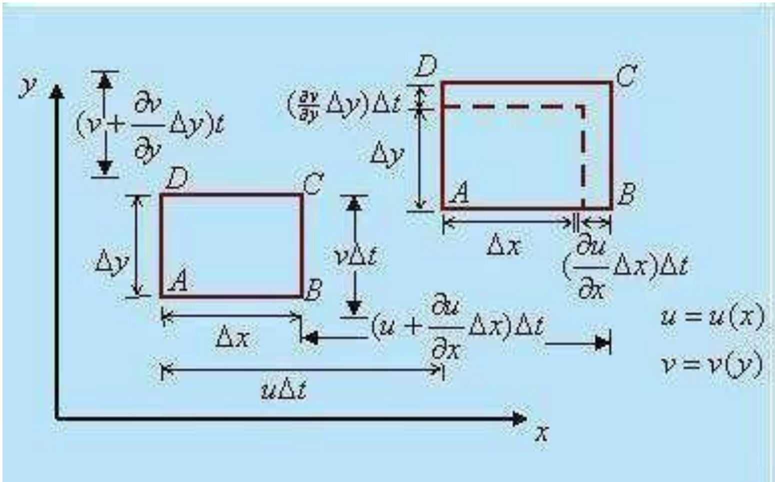


Fig: Fluid Element in Translation with Continuous Linear Deformation

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

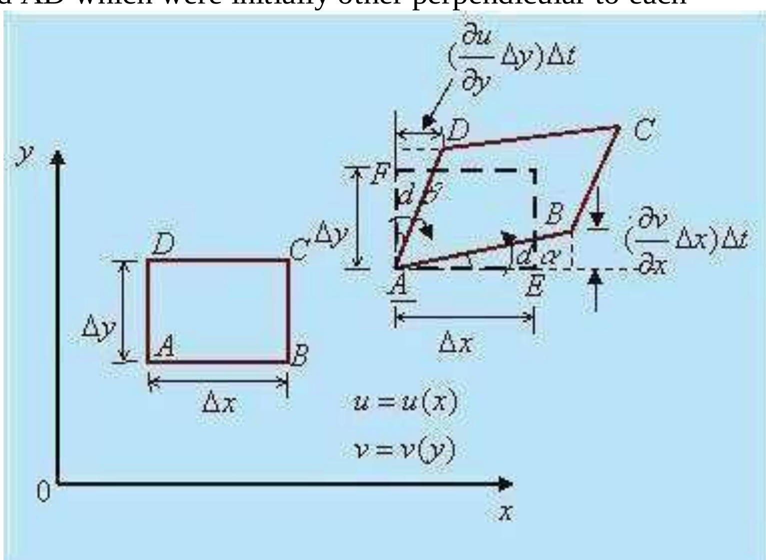


Fig: Fluid element in translation with simultaneous linear and angular deformation rates From the above figure rate of angular deformation,

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

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 suffixzin ω represents the rotation aboutz-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 cartesiancartesian 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.

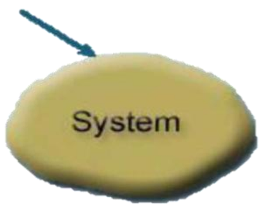
System Boundary: A separation present between system and surrounding.

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

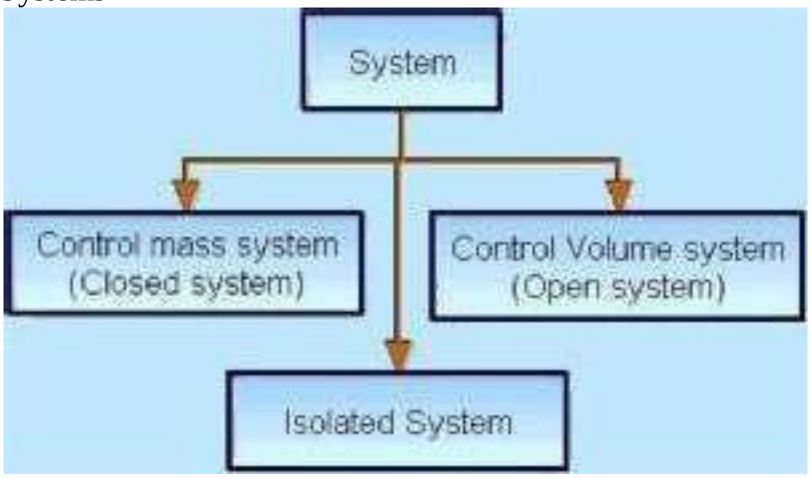
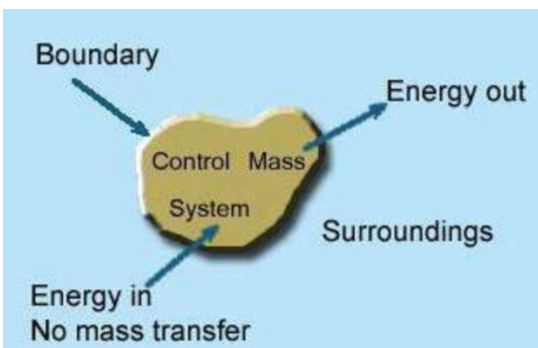


Fig: Types of System

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 thetransferof heat across the systemboundary.

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

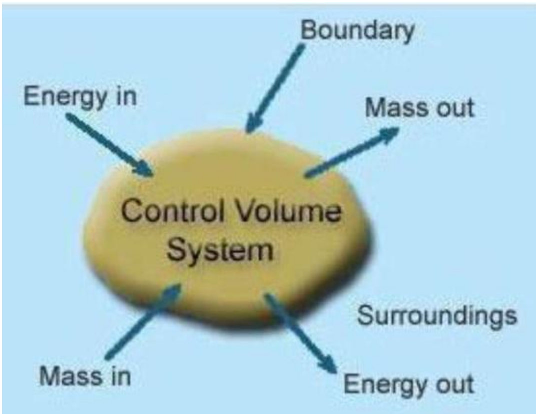


Fig: A Control Volume System or open System Isolated System
Its a system of fixed mass with same identity and fixed energy.
No interaction of mass or energy takes place between the system and the surroundings. In more informal words an isolated system is like a closed shop amidst a busy market.

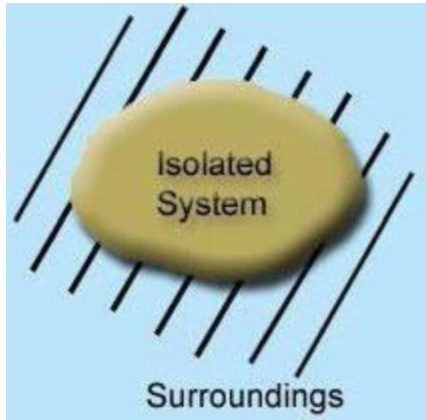


Fig: An Isolated System

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 =

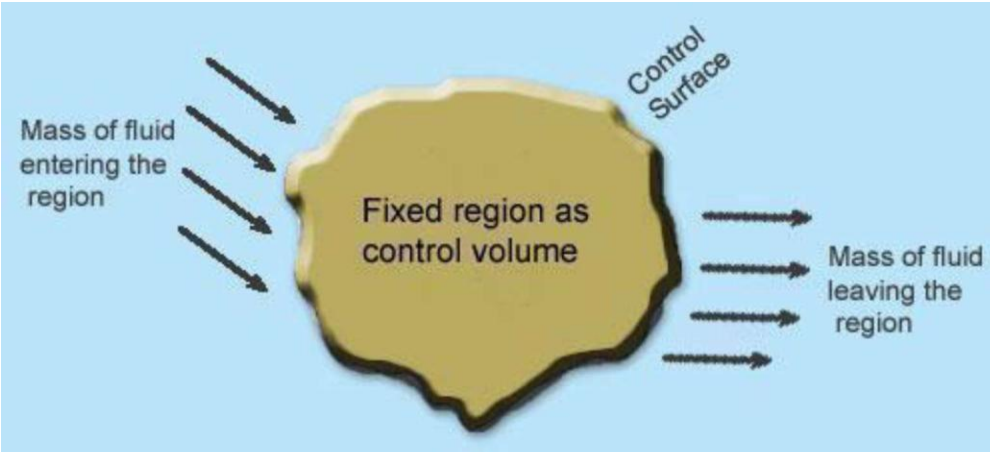


Fig: A Control Volume in a Flow Field Continuity Equation - Differential Form

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.

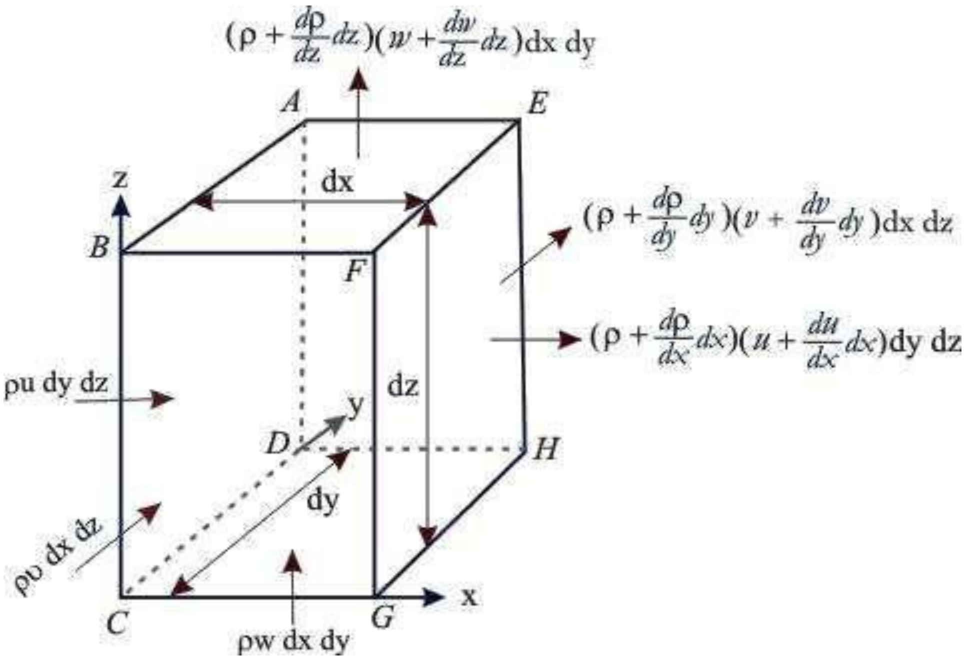


Fig: A Control Volume Appropriate to a Rectangular Cartesian Coordinate System

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 ordertermsin 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

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

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 steadyflow,

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

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

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

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

In cylindrical polar coordinates eq.9.7 reduces to

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

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.

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

Stream Function

Let u 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

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

Ifafunction $\psi(x, y, t)$ is defined in themanner 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

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

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

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

$$\psi(x, y) = \text{constant}$$

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

PhysicalSignificanceof Stream Funtion ψ

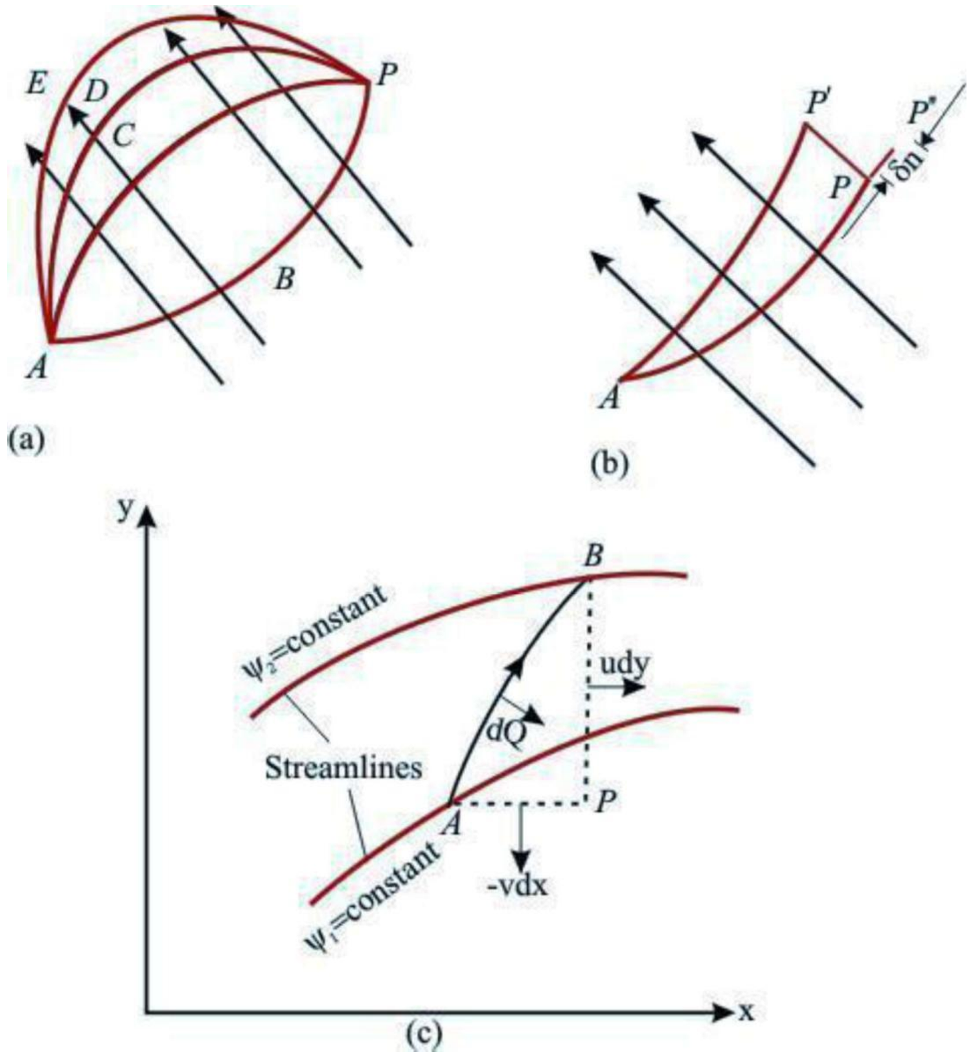


Fig: Physical Interpretation of Stream Function

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 the stream 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

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.

MODULE - III
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 thesystem.

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.

weshalll discuss each example seperately in the following slides.

Euler's Equation along aStreamline

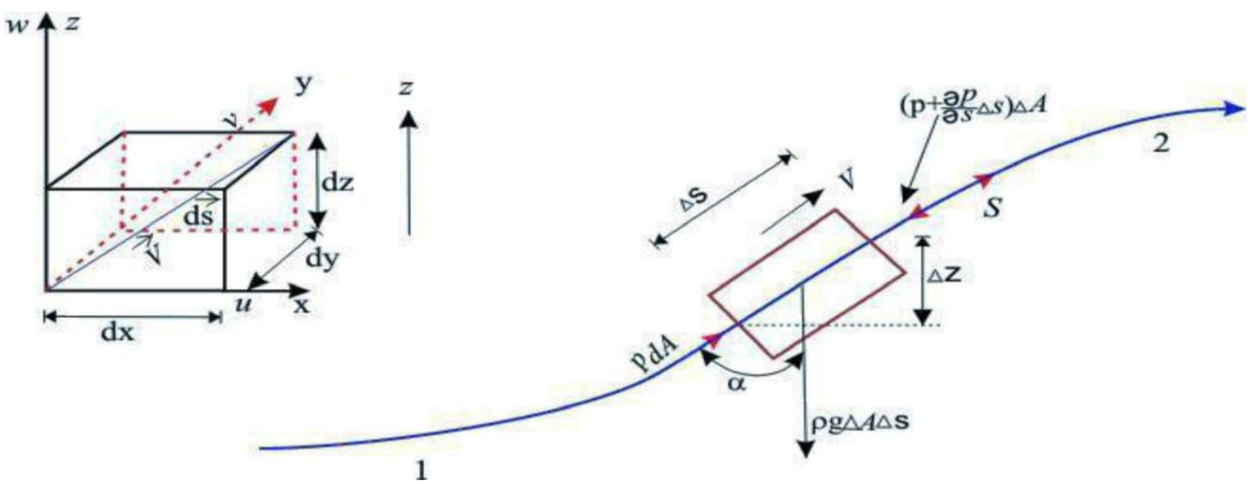


Fig: Force Balance on Moving Element Along a Streamline Derivation

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 streamlinesothat

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

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

$$\vec{V} \times d\vec{S} = 0$$

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

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

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

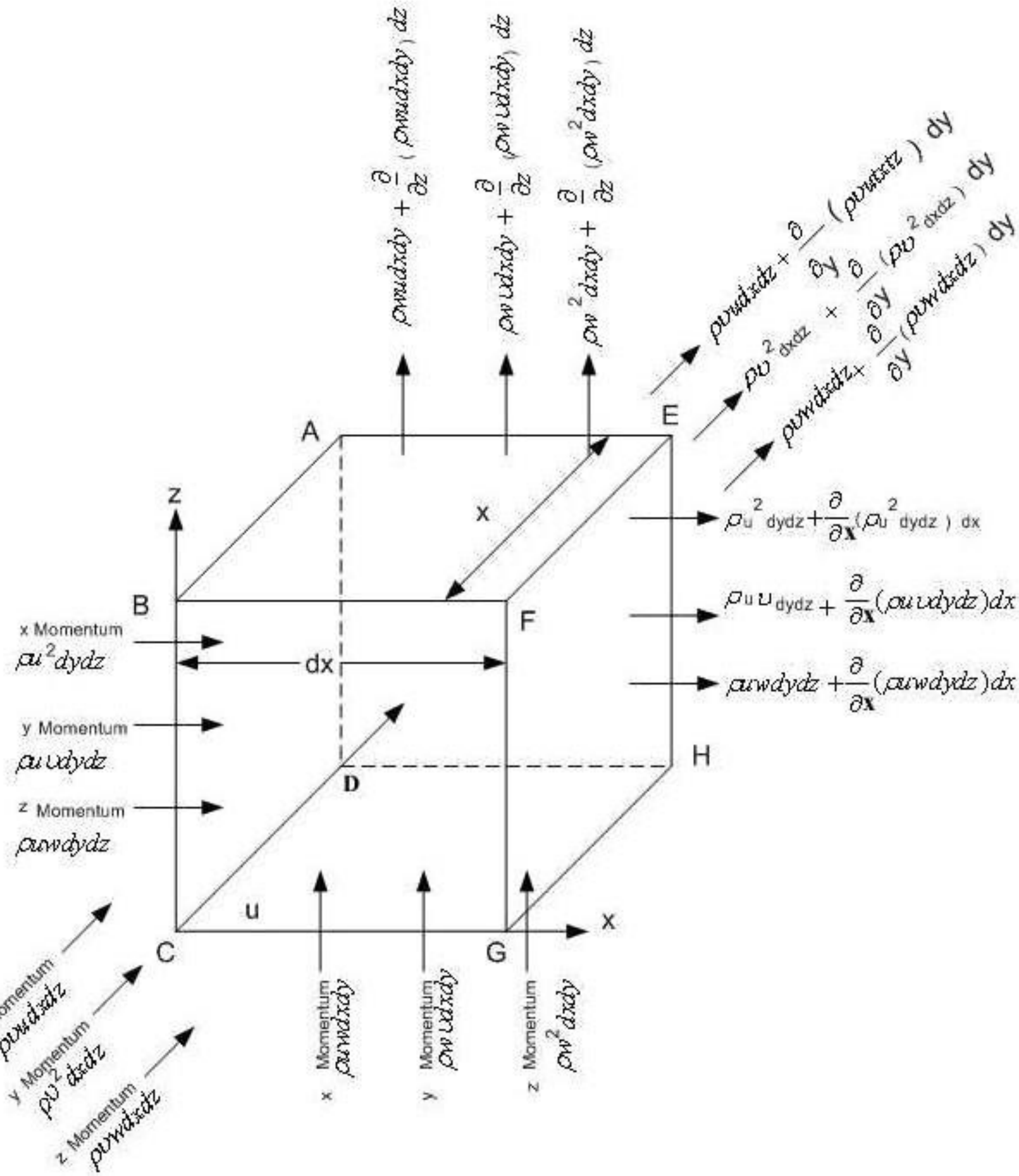


Fig: A Control Volume used for the derivation of Euler's Equation

Wecandefinethevelocityvector $\vec{V}$ andthebodyforceperunitvolume $\rho\vec{X}$ as

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

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

Therate of x momentum influx to the control volume through the face ABCD is equal to dy dz. The rate of x momentum efflux from the

control volume through the face EFGHequals $\rho u^2 dy\, dz + \frac{\partial}{\partial x}\left(\rho u^2 dy\, dz\right)dx$

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

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

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

typical form of Euler's equations given by Eqs (12.11a), (12.11b) and (12.11c) are

The 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 thescopeofthebook.However,inmanypracticalsituations,problemsrelatedtorealfluids

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 InIrrotational 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 substitutein $\frac{\partial v}{\partial x}$ 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

resultsin

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

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

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

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

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 canwrite

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

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

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

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

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

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

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

The integration of Eq. results in

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

For an incompressibleflow,

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

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 thework.

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 unitmass

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

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

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
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 $r \to 0,\ V_\theta \to \infty$
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 dominants **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 centralcore.

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

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

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

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

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.

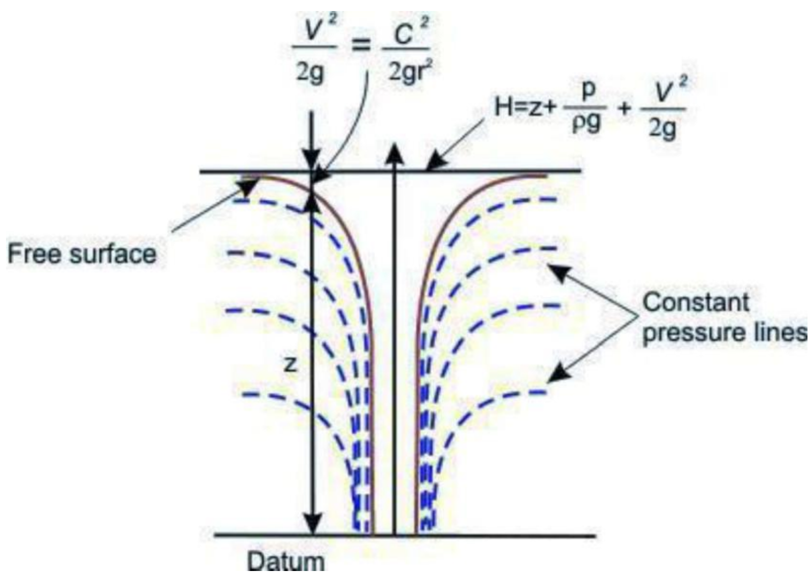


Fig: Cylindrical Free Vortex Forced Vortex Flows

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

The vorticity Ω for the flow field can be calculated as

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

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

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 potentialenergy.

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

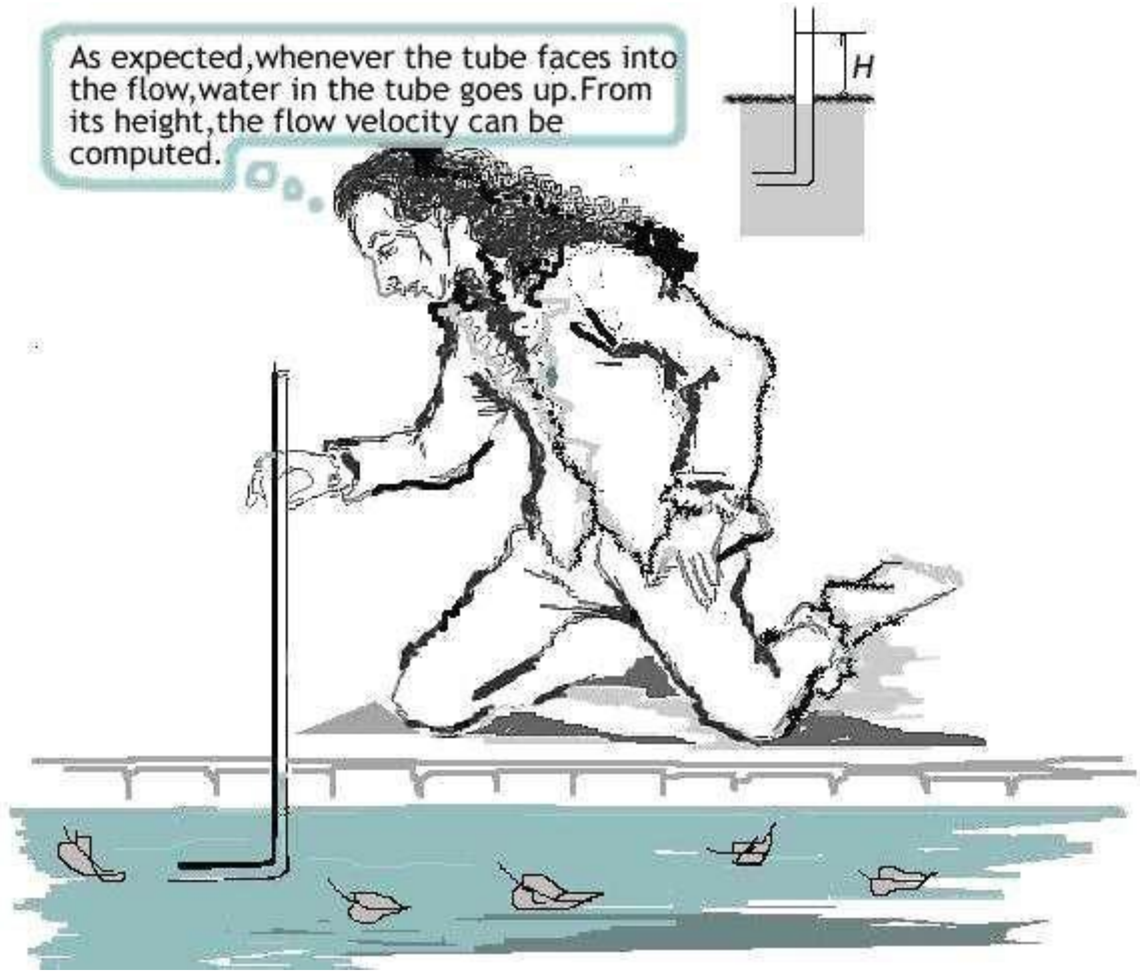
$$\int_1^2 \frac{1}{\rho}\frac{dp}{dr}dr = \int_1^2 \frac{V_\theta^2}{r}dr = \omega^2 \int_1^1 r dr$$

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

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. VenturimeterOrificemeter 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.

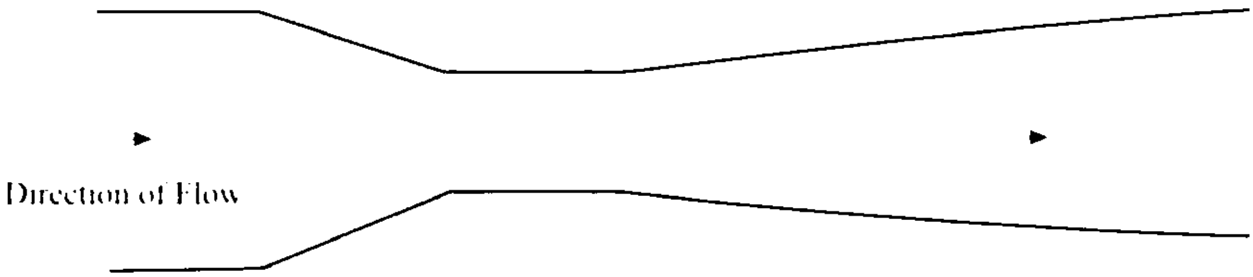


Fig: A Venturimeter

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.

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

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; andhence

$h_1^* - h_2^*$ becomes h1 − h2, whereh1and 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, thedifferencein 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 valueof Δh,and theactual rateof dischargeis thedischarge 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.

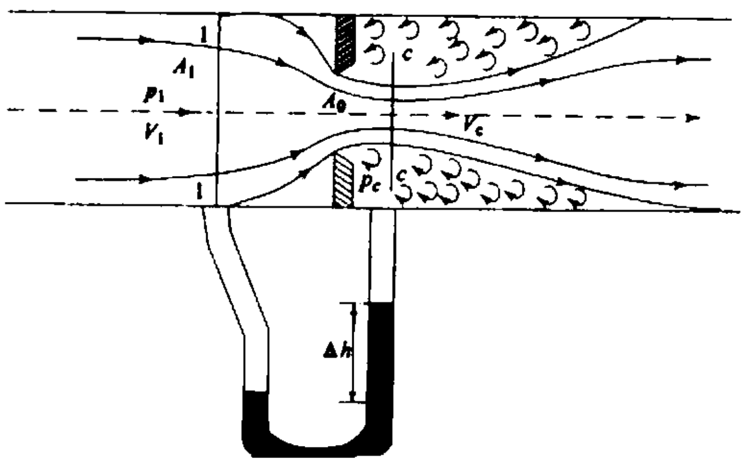


Fig: Flow through an orificemeter

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

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

p_1^* p_c^*

where and are the 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,

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

Concept and Types of Physical Similarity

The primary and fundamental requirement for the physicalsimilarity 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

Therefore,geometricallysimilarobjectsaresimilarintheirshapes,i.e.,proportionateintheir physicaldimensions,butdifferinsize.

In investigations of physical similarity,Thefullsizeoractualscalesystemsareknownasprototypesthelaboratoryscalesystemsare referred to asmodelsuse of the same fluid with both the prototype and the model is not necessaryModel 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.

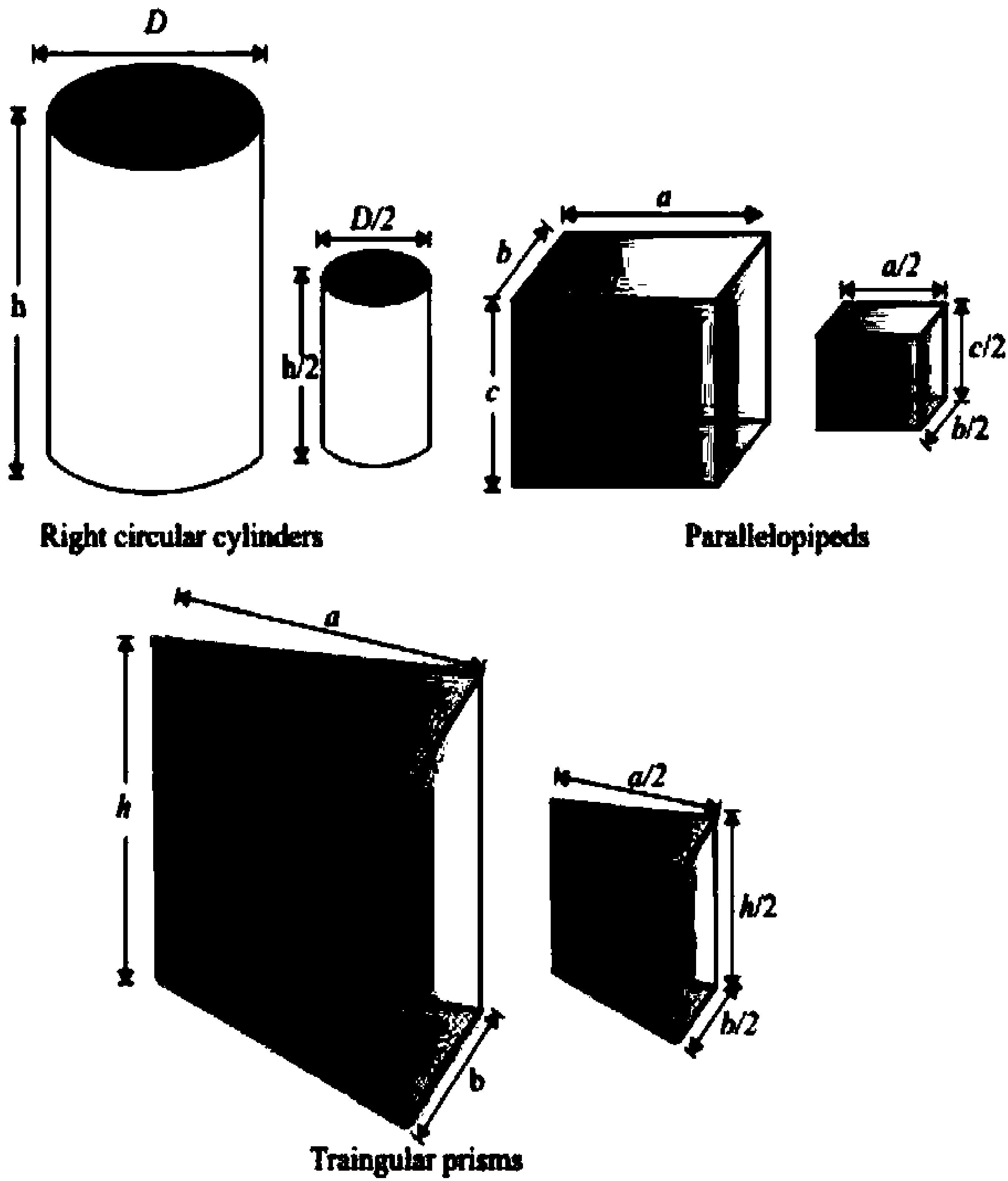


Fig: Geometrically Similar objects In all the above cases model ratio is 1/2

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 maybe 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 resultantaccelerating force $\vec{F_R}$

$$\vec{F_i} \quad \vec{F_R}$$

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 non dimensionalisedforces in the twosystems)

$$\frac{|\vec{F_v}|}{|\vec{F_i}|}, \frac{|\vec{F_p}|}{|\vec{F_i}|}, \frac{|\vec{F_g}|}{|\vec{F_i}|}, \frac{|\vec{F_c}|}{|\vec{F_i}|}, \frac{|\vec{F_e}|}{|\vec{F_i}|}$$

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 aflluid element in the course of its flow.

InertiaForce $\vec{F}_i$

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 isproportionalto ρl where, is ρ the density of fluid and l isthe characteristic geometrical dimension of thesystem.

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,

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

ViscousForce $\vec{F}_v$

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 shearstrain

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 intheflow.) GravityForce $\vec{F}_g$
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 $\vec{F}_c$ orSurfaceTensionForce

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,

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

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

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 agivencompression $\Delta \mathbf{p} \; \alpha \; \mathbf{E}$

(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:
Theterm $\rho l V/\mu$ is known as Reynolds number, Re after the name of the scientist whofirst developed it and is thus proportional to the magnitude ratio of inertia force to viscousforce
.(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 similarityfor the flows which are affected only by viscous, pressure and inertiaforces.
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

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

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

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. (d), while the equality of Reynolds number (Eq. (c)) 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 offlow.

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 ,

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

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 dynamicsimilarity.

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

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

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.,

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

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 flowsofsmalljetsandthinsheetsofliquidinjectedbyanozzleinairflowofathinsheetof liquid over a solidsurface.

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

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

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,

$$\frac{\text{Inertia force}}{\text{Elastic Force}} = \frac{|\vec{F}_i|}{|\vec{F}_e|} \propto \frac{\rho V^2 l^2}{E l^2} = \frac{\rho V^2}{E}$$

Theparameter $\rho V^2/E$ is known as Cauchy number ,(after the French mathematicianA.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 inthe fluidmedium.

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

A physical problem described by m number of variables involving n number offundamental dimensions (n # m) leads to a system of n linear algebraic equations with m variables of the form

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

The sets of integer exponents a_1, a_2 . . . 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 . . . a_n which gives a unique solution for the exponents. This gives the values of a_1 a_2 . . . $a_n$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,thenon- dimensional term π_1 is some specific combination of n + 1 variablesinvolved in theproblem.

then m = n+ 1 , the number of

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.

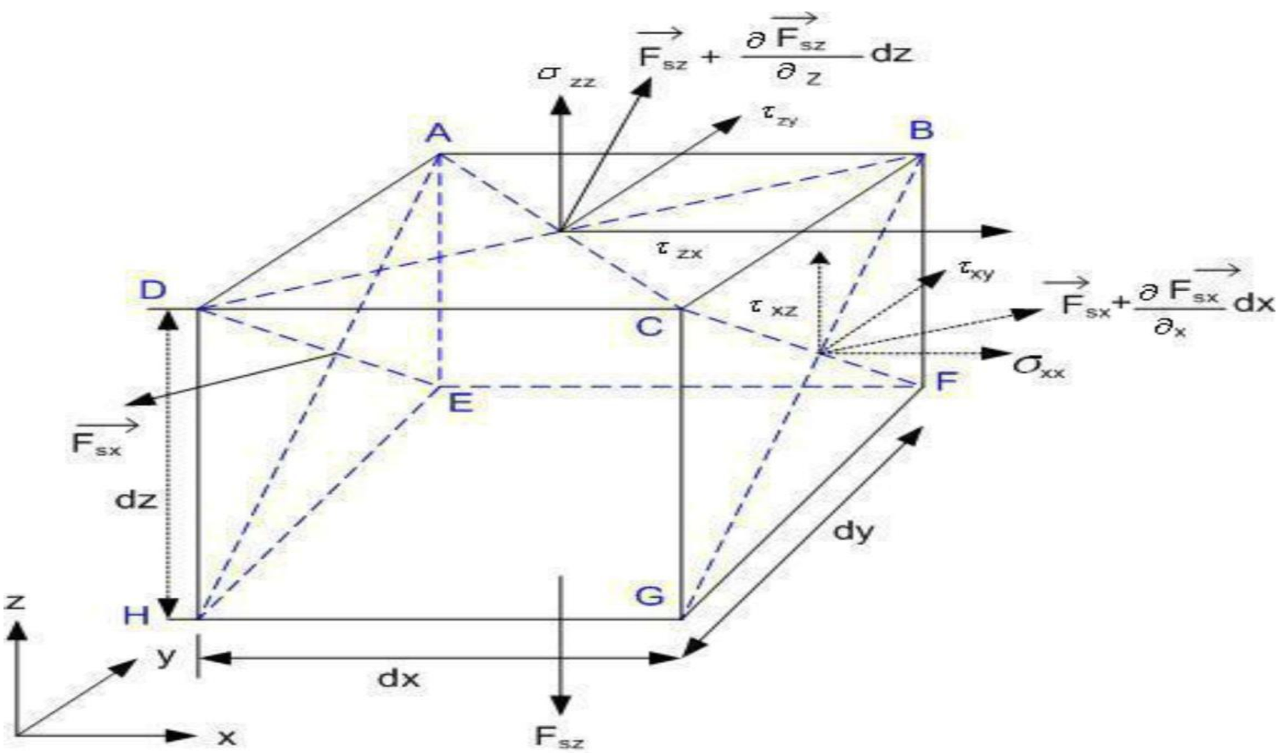
External forces are of two kinds-

one acts throughout the mass of thebody----- body force(gravitational force, electromagneticforce)

another acts ontheboundary--------------------- surface force (pressureand frictionalforce).

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 locations in a differentia/ fluid element 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

$$d\vec{F} = \frac{\partial \vec{F}_{sx}}{\partial x} + \frac{\partial \vec{F}_{sy}}{\partial y} + \frac{\partial \vec{F}_{sz}}{\partial z}$$ (sinceVolume=dxdydz)

$\vec{F}_{sx}$ $\vec{F}_{sy}$ $\vec{F}_{sz}$

$$\vec{F}_{sx} = \hat{i}\ \sigma_{xx} + \hat{j}\ \tau_{xy} + \hat{k}\ \tau_{xz}$$

$$\vec{F}_{sy} = \hat{i}\ \tau_{yx} + \hat{j}\ \sigma_{yy} + \hat{k}\ \tau_{yz}$$

$$\vec{F}_{sz} = \hat{i}\ \tau_{zx} + \hat{j}\ \tau_{zy} + \hat{k}\ \sigma_{zz}$$

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 flowfield.

A control surface, AO is defined as the surface which bounds the volume $\forall_0$. According to

Reynoldstransporttheorem,"Therateofchangeofmomentumfora
system equals the sum of the rate of change of momentum inside thecontrol volume and the rate of efflux of momentum across the controlsurface".

Therateofchangeofmomentumforasystem(inourcase,thecontrolvolumeboundary andthesystemboundaryaresame)isequaltothenetexternalforceactingonit.

Now, we shall transform these statements into equation by accounting for each term,

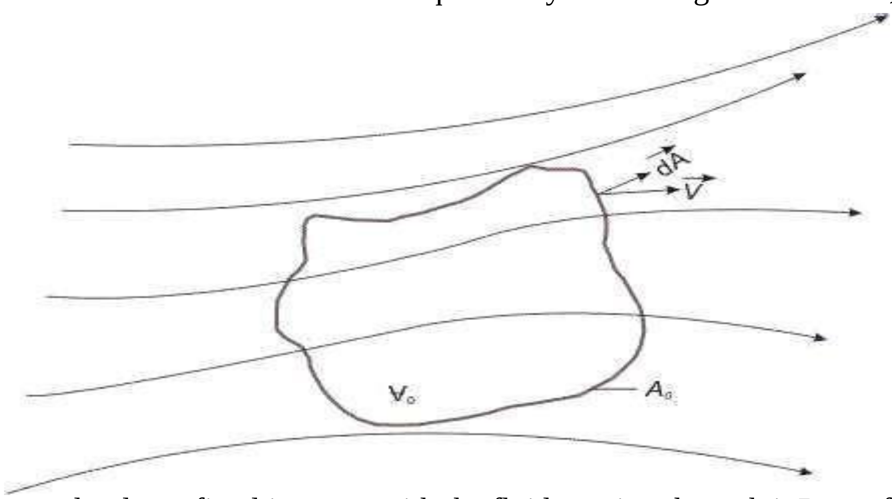


Fig: Finite control volume fixed in space with the fluid moving through it Rate of change of

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 unitmass. Finally, weget,

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 $$\rho\left(\frac{\partial\vec{V}}{\partial t} + \vec{V}.\nabla\vec{V}\right) + \vec{V}\left(\frac{\partial\rho}{\partial t} + \nabla.\rho\vec{V}\right) = \nabla.\sigma + \rho\vec{f}_b$$

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

weknowthatisthegeneralformofmassconservationequation(popularlyknownasthecontinuityequation),validforbothcompressible and incompressibleflows.
Invoking this relationship in we obtain

Equation is referred to as Cauchy's equation of motion. In this equation, is thestress 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.

wechoosetobe x thedirectionalongwhichallfluidparticlestravel,i.e. $u \neq 0, v = w = 0$
. Invoking this in continuity equation, we get

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

Whichmeans $u = u(y, z, t)$

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

$\frac{\partial p}{\partial t} = \frac{\partial p}{\partial z} = 0$ whichmeans $p = p(x)\ alone$

and $$\frac{\partial u}{\partial t} = -\frac{1}{\rho}.\frac{\partial p}{\partial x} + \nu\left[\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]$$

MODULE -IV
BOUNDARY LAYER THEORY

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 $\partial u / \partial y$ isnegligible), 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

Boundarylayerthicknessis δ whichisafunctionofthecoordinatedirection:x.

Thethicknessisconsideredtobeverysmallcomparedtothecharacteristiclengthlofthe 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.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]$$

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

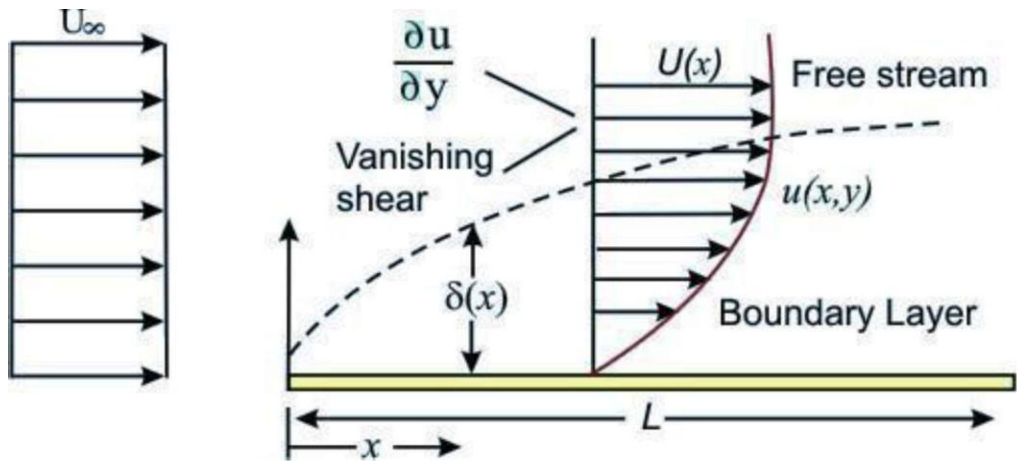


Fig: Boundary layer and Free Stream for Flow over a flat plate

u - velocity component along x direction. v - velocity component along y direction p - static pressure

- ρ density.

- μ dynamic viscosity of thefluid

The equations are now non-dimensionalised.

Thelengthandthevelocityscalesarechosenas1and U_∞ respectively.The non- dimensional variablesare:

$$u^*=\frac{u}{U_\infty}, v^*=\frac{v}{U_\infty}, p^*=\frac{p}{\rho U_\infty^2}$$

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 isequal to .

Based on the above, we can identify the following scales for the boundary layervariables:

Thesymbol describes a value much smaller than1.

Now we analyse, and look at theorderof magnitude ofeach individualterm

The continuity equation

One general rule of incompressible fluid mechanics is that we are not allowed to drop any term from the continuity equation.

Asaconsequenceoftheorderofmagnitudeanalysis,canbedroppedfromthe:xdirection momentum equation, because onmultiplicationwith it assumes thesmallestorder of magnitude.

Eq. - 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.

Thisalsoimpliesthatthepressurepisonlyafunctionof:x.The pressure forcesonabodyare solely determined by the inviscid flow outside the boundarylayer.

The application of at the outer edge of boundary layer gives on integrating the well known Bernoulli's equation is obtaineda 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.

The Prandtl boundary layer equations are thus a simplification of the Navier-Stokes equations.

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.

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.

Substituting
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.
we can rewrite Eq. (28.22) as three first order differential equations in the following way

Let us next consider the boundary conditions.Thecondition remainsvalid. Thecondition meansthat .

In a similar way K_3, $/_3$, m_3 and k_4, $/_4$, m_4 mare calculated following standard formulae for the Runge-Kutta integration.
For example, K_3 is given by

The functions F_1,
$F_2$and F_3 are G, H , - f H I J respectively. Then at a distance from the wall, we have
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 tovelocitydistribution within the boundary layer of a stream of air on a flatplate.

In the next slide we'll see some values of the velocity profile shape and in tabular format.

Values of the velocity profile shape

or, $$\delta^* = 1.7208\sqrt{\frac{\nu x}{U_\infty}} = \frac{1.7208x}{\sqrt{Re_x}} \qquad (29.7)$$

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

$$\int_0^\delta \left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} \right) dy = \int_0^\delta \left(-\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial y^2} \right) dy$$

or, $$\int_0^\delta u\frac{\partial u}{\partial x}dy + \int_0^\delta v\frac{\partial u}{\partial y}dy = \int_0^\delta -\frac{1}{\rho}\frac{\partial p}{\partial x}dy + \int_0^\delta \nu\frac{\partial^2 u}{\partial y^2}dy$$

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 continuityequation

or, $$\int_0^\delta v\frac{\partial u}{\partial y}dy = -U_\infty \int_0^\delta \frac{\partial u}{\partial x}dy + \int_0^\delta u\frac{\partial u}{\partial x}dy$$

$$\int_0^\delta 2u\frac{\partial u}{\partial x}dy - U_\infty \int_0^\delta \frac{\partial u}{\partial x}dy = -\int_0^\delta \frac{1}{\rho}\frac{\partial p}{\partial x}dy - \nu\left.\frac{\partial u}{\partial y}\right|_{y=0}$$

Substituting the relation between $\frac{\partial p}{\partial x}$ and the freestreamvelocity U_∞ for the inviscidzone in Eq. (29.12) weget

$$\int_0^\delta 2u\frac{\partial u}{\partial x}dy - U_\infty \int_0^\delta \frac{\partial u}{\partial x}dy - \int_0^\delta U_\infty \frac{dU_\infty}{dx}dy = -\left(\frac{\mu \left.\frac{\partial u}{\partial y}\right|_{y=0}}{\rho} \right)$$

$$\int_0^\delta \left(2u\frac{\partial u}{\partial x} - U_\infty \frac{\partial u}{\partial x} - U_\infty \frac{dU_\infty}{dx} \right) dy = -\frac{\tau_w}{\rho}$$

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 increasethe 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 $\delta^* = \int_0^\delta \left(1 - \frac{u}{U_\infty}\right) dy$ isthe

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 turbulentflows.

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 flowdirection.

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 thebody.

During the flow over a flat plate, $U_\infty \frac{dU_\infty}{dx} = 0$ and the momentum integral equation is reducedto

$$\frac{d}{dx}\left[U_\infty^2 \delta^{**}\right] = \frac{\tau_w}{\rho}$$

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 pressuregradient.

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 behaviouris like thatof anozzle.

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 freestream

U_∞ : velocity in the free streamand

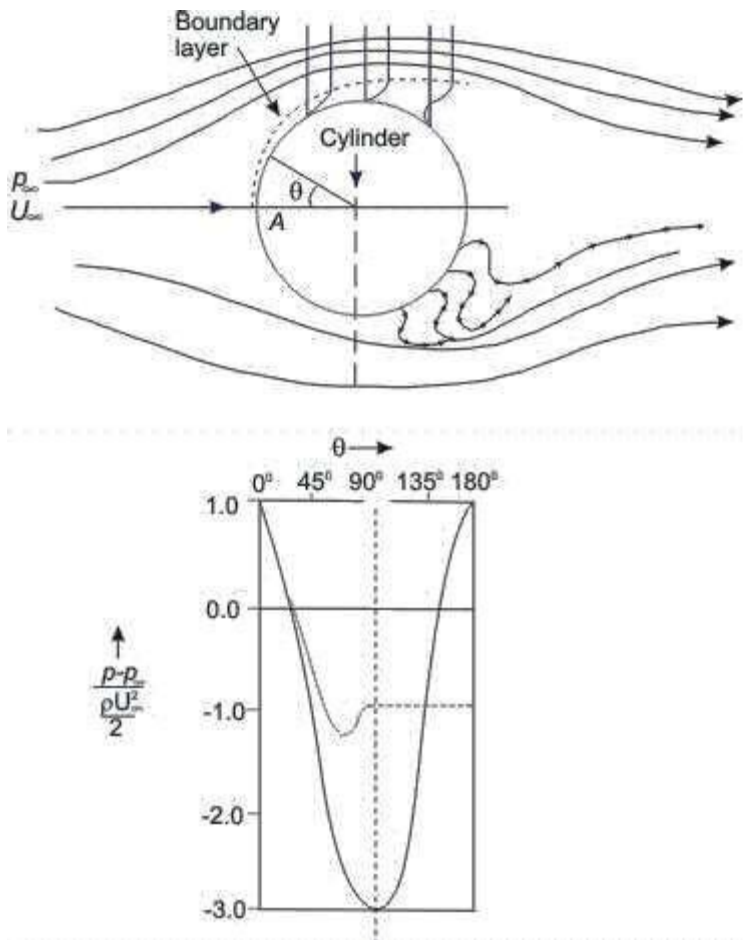


Fig.Flow separation and formation of wake behind a circular cylinder Consider the forces in the flow field. In the inviscid region,

Until $\theta = 90^\circ$ the pressure force and the force due to streamwise acceleration i.e. inertia forces are acting in the same direction (pressure gradient beingnegative/favourable)

Beyond $\theta = 90^\circ$, 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 thispart.

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 kineticenergy.

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 pressureforce.

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 separatedlayers together with it and develops a broad pulsating wake behind thecylinder.

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 thepoint ofseparation

$$\left(\frac{\partial u}{\partial y}\right)_{y=0} = 0$$

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 backflow.

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 $\frac{dp}{dx} < 0$ wehave,

$\left(\partial^2 u / \partial y^2\right)_{wall} < 0$. (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 lesserrate inydirection.

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 pressuregradient, $\partial p / \partial x > 0$

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 previousargument regarding $\partial u / \partial y$ and $\partial^2 u / \partial y^2$ still holds good and the curvature isnegative.

Thus we observe that for an adverse pressure gradient, there must exist a pointforwhich $\partial^2 u / \partial y^2 = 0$. 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 thewall.

$\partial^2 u / \partial y^2 > 0$ at the wall since separation can only occur due to adverse pressuregradient.

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 inflectionin the velocityprofile.

Favourable pressuregradient, $\frac{dp}{dx} < 0$

(a) adverse pressuregradient, $\frac{dp}{dx} > 0$

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.somewherenear $\theta = 90^\circ$(in experiments it has been observed to be at $\theta = 81^\circ$) . theboundary 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 pressuredifference.

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 which is $\delta(x)$ 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 growthof boundarylayerasdistance:xvariesfromtheleading

edge, thevelocityprofile (u/U_∞) remainsgeometricallysimilar. we choose a velocity profile in theform

$$\frac{u}{U_\infty} = a_0 + a_1\eta + a_2\eta^2 + a_3\eta^3$$

In order to determinetheconstants a_0, a_1, a_2 and a_3 we shall prescribethe following boundaryconditions

$$at\ y = 0, u = 0 \quad or \quad at\ \eta = 0, \frac{u}{U_\infty} = 0$$

$$at\ y = 0, \frac{\partial^2 u}{\partial y^2} = 0 \quad or \quad at\ \eta = 0, \frac{\partial^2}{\partial \eta^2}(u/U_\infty) = 0$$

at

The wall shear stress is given by

$$\tau_w = \mu \frac{\partial u}{\partial y}\Big|_{y=0}$$

$$or\ \ \tau_w = \mu\left[\frac{\partial}{\delta\partial\eta}\left\{U_\infty\left(\frac{3}{2}\eta - \frac{1}{2}\eta^3\right)\right\}\right]_{\eta=0}$$

$$or\ \ \tau_w = \frac{3\mu U_\infty}{2\delta}$$

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 (at $x = 0, \delta = 0$) yields $C_1 = 0$. Finally we obtain,

$$\delta^2 = \frac{280}{13}\frac{\nu x}{U_\infty}$$

$$or\ \ \delta = 4.64\sqrt{\frac{\nu x}{U_\infty}}$$

$$or\ \ \delta = \frac{4.64x}{\sqrt{Re}}$$

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

$$y = \delta, \frac{\partial^2 u}{\partial y^2} = 0 \quad or \; at \; \eta = 1, \frac{\partial^2 (u/U_\infty)}{\partial \eta^2} = 0$$

This yields the constants as . Finally the velocity profile will be $a_0 = 0, a_1 = 2, a_2 = 0, a_3 = -2$ and $a_4 = 1$

$$\frac{u}{U_\infty} = 2\eta - 2\eta^3 + \eta^4$$

Subsequently, for a fourth order profile the growth of boundary layer is given by

$$\delta = \frac{5.83x}{\sqrt{\mathrm{Re}_x}}$$

Integ

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 conditionsare

$$u/U = a\eta + b\eta^2 + c\eta^3 + d\eta^4$$

$$\eta = 0 : u = 0, v = 0 \qquad \frac{\nu}{\delta^2}\frac{\partial^2 u}{\partial \eta^2} = \frac{1}{\rho}\frac{dp}{dx} = -U\frac{dU}{dx}$$

$$\eta = 1 : u = U \qquad \frac{\partial u}{\partial \eta} = 0,\ \frac{\partial^2 u}{\partial \eta^2} = 0$$

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

$$F(\eta) = 2\eta + 2\eta^3 + \eta^4,\ G(\eta) = \frac{1}{6}\eta(1-\eta)^3$$

Theshearstress $\tau_\omega = \mu(\partial u/\partial y)_{y=0}$ is givenby

$$\frac{\tau_\omega \delta}{\mu U} = 2 + \frac{\lambda}{6}$$

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 pressuregradient.

IfwetakeKastheindependentvariable.landHcanbeshowntobethefunctionsof Ksince

$$\frac{\delta^*}{\delta}=\int_0^1(F(\eta)+\lambda G(\eta))(1-F(\eta)-\lambda G(\eta))d\eta$$

$$=\frac{37}{315}-\frac{\lambda}{945}-\frac{\lambda^2}{9072}$$

$$K=\frac{[\delta^{**}]^2}{\delta^2}\lambda=\lambda\left(\frac{37}{315}-\frac{\lambda}{945}-\frac{\lambda^2}{9072}\right)^2$$

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

$$\frac{U[\delta^{**}]^2}{\nu}=\frac{a}{U^{b-1}}\int_0^x U^{b-1}dx$$

witha=0.47andb#6.theapproximationisparticularlyclosebetweenthestagnation pointandthepointtofmaximumvelocity.

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)canbeobtainedfromEq.(30.16).

Table 30.1 gives the necessary parameters for obtaining results, such as velocity profileand shearstress τ_{ω} The approximate method can be applied successfully to a widerange of problems.

Table .1 Auxiliary functions after Holstein and Bohlen

	K λ	$f_1(K)$	$f_2(K)$
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 shapefactor.

Introduction of K in the integral analysis enables extension of Karman-Pohlhausenmethod for solving flows over curved geometry. However, theanalysis is not valid for the geometries, where $\lambda < -12$ and $\lambda > +12$
to turbulence takes place.

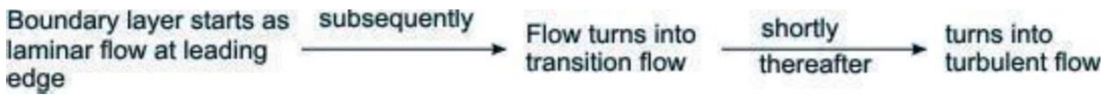
Turbulence can be generated by - frictional forces at the confining solid walls the flow of layers of fluids with different velocities over one another

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

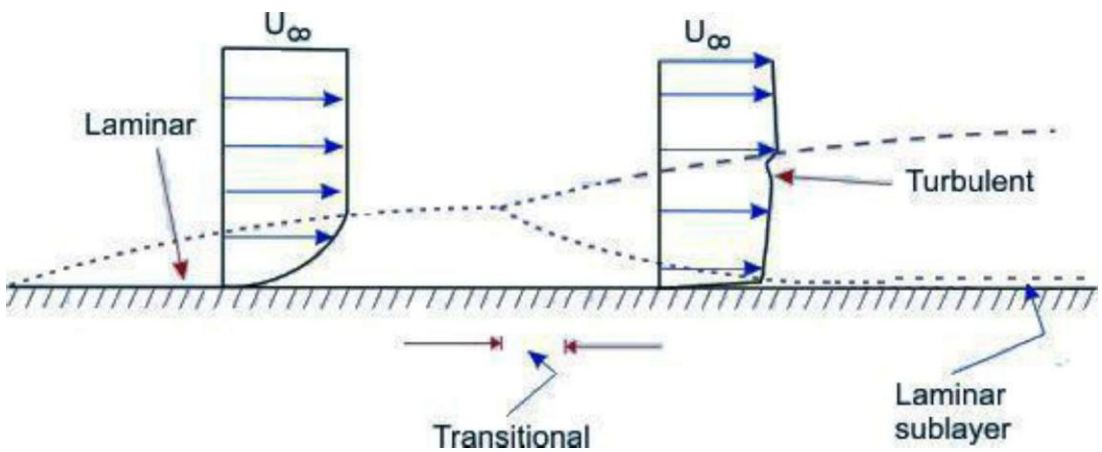


Fig. Laminar - turbulent transition illustration

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 anis 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.

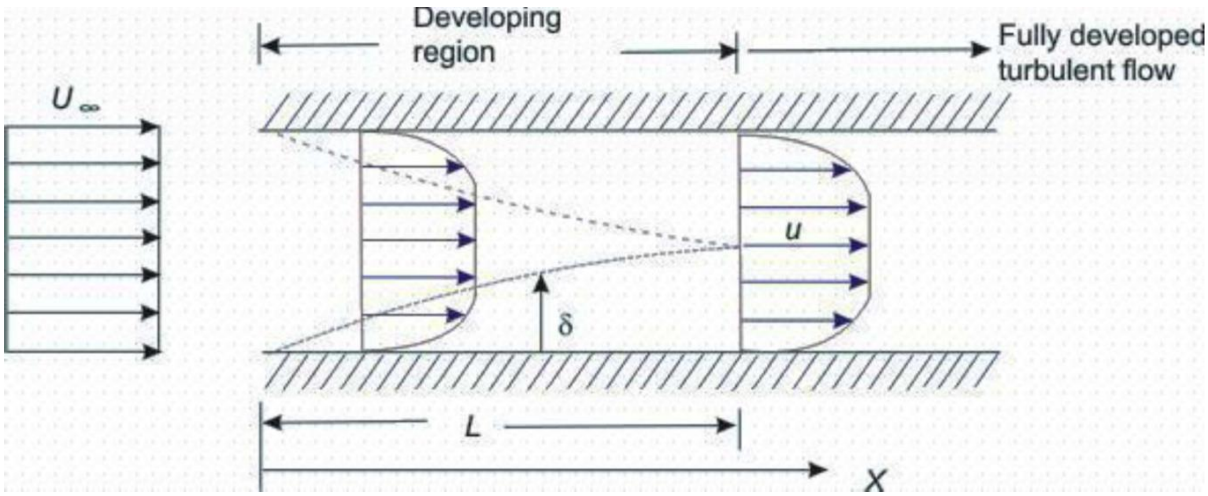


Fig. Development of turbulent flow in a circular duct

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

$$\left(-\frac{dp}{dx} = \frac{2\tau_w}{R}\right)$$

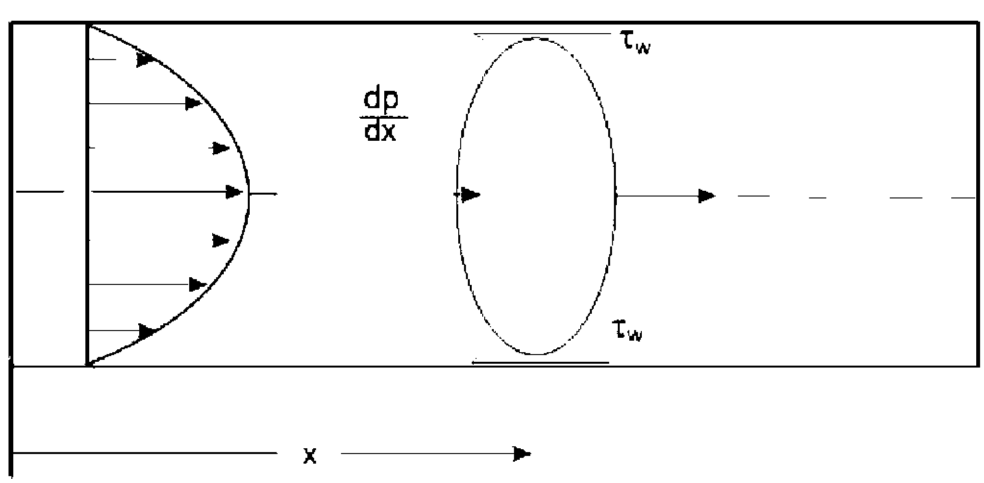


Fig: Fully developed turbulent pipe flow

It can be said that in a fully developed flow, the pressure gradient balances the wall shearstress only and has a constant value at any x. However, the friction factor (Darcy friction factor) is defined in a fully developed flowas

$$-\left(\frac{dp}{dx}\right) = \frac{\rho f U_{av}^2}{2D}$$

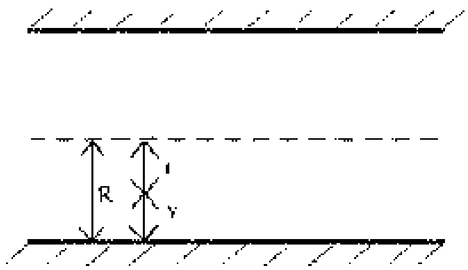
Comparing, we can write

$$\tau_w = \frac{f}{8}\rho U_{av}^2$$

H. Blasius conducted a critical survey of available experimental results and established the empirical correlation for the above equation as

$$f = 0.3164\,\mathrm{Re}^{-0.25} \quad \text{where} \quad \mathrm{Re} = \rho U_{av} D / \mu$$

Itisfound that the Blasius's formula is valid in the range of Reynolds number of $\mathrm{Re} \leq 10^5$. 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:

Thefrictionfactorinthecaseofapipeflowwasalreadymentionedinlecture26. we will elaborate

further on friction factor or friction coefficient in thissection.

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 ductrespectively.

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

$$\tau_w = \frac{\Delta p^* A}{SL}$$

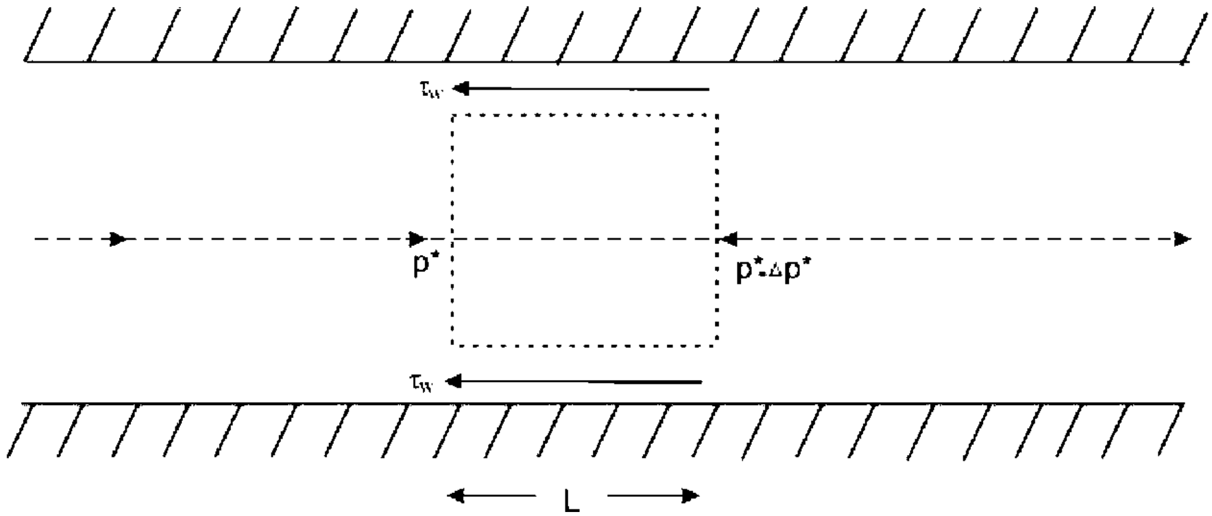


Fig: Force Balance of a fluid element in the course of flow through a duct

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)

$$h_f = \frac{\Delta p^*}{\rho g} = \frac{f L V^2}{2 g D_h}$$

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

$$f = \frac{64}{\mathrm{Re}}$$

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

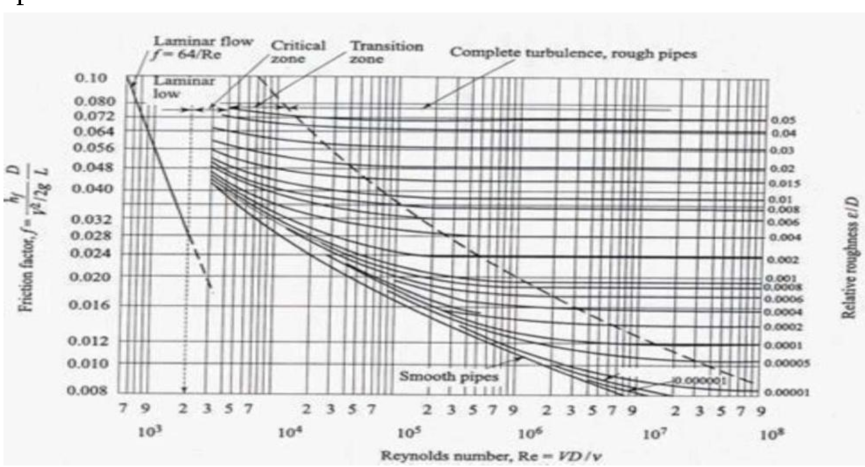


Fig: Friction Factors for pipes (adapted from Trans. ASME, 66,672, 1944)

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.

MODULE-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 primemovers.

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 hydroelectricpower.

Electric Motor: Electric motor coverts electrical energy to mechanicalenergy.

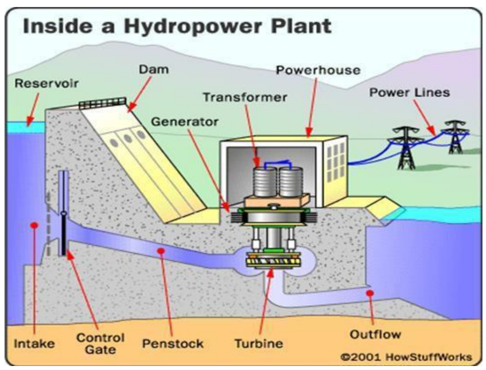
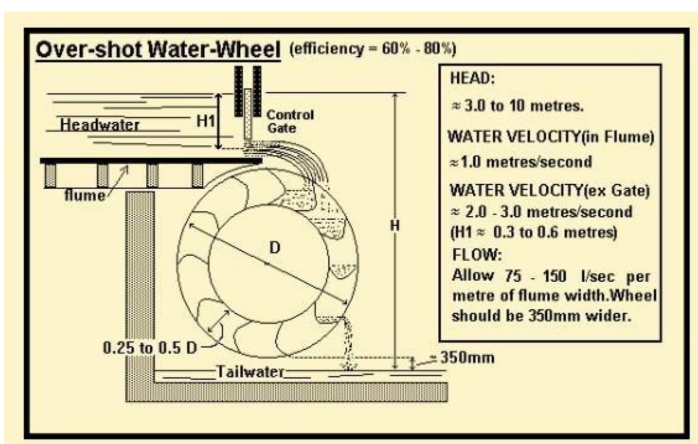
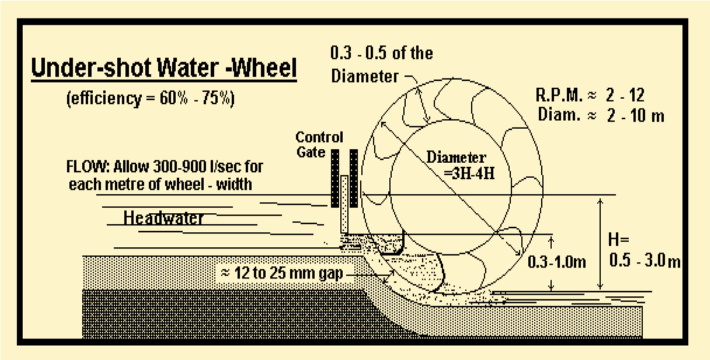


Fig: Electric Motor

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 coupleddirectly.

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 torotate.

<u>Classification of Hydraulic Turbines</u>

1. According to the type of energy at theinlet
2. According to the direction of flow throughrunner
3. According to head atinlet
4. According to specific speed ofturbine
5. According to Position of theshaft

1. <u>According to the type of energy at theinlet</u>

a) Impulseturbine:

All the available energy of the water is converted into kinetic energy by passing it througha contracting nozzle provided at the end ofpenstock

Ex: Pelton wheel turbine, Turbo-impulse turbine, Girard turbine, Bank turbine, Jonval turbine etc.

b) ReactionTurbine:

- At the entrance of the runner, only a part of the available energy of water is convertedinto kinetic energy and a substantial part remains in the form of pressureenergy.
- 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 theturbine.

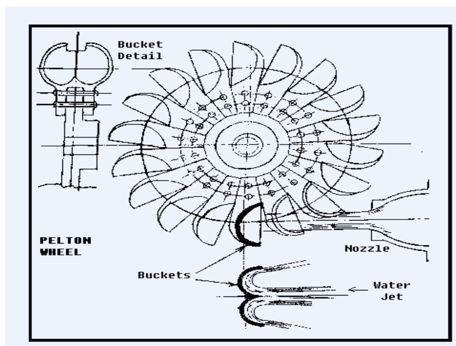
- For this gradual change of pressure to the possible the runner must be completely enclosed in a air-tight casing and the passage is entirely full of water throughout the operation ofturbine
- The difference of pressure between the inlet and outlet of the runner is calledreaction
- pressure and hence the turbines are known as reactionturbines.
- Ex: Francis turbine, Kaplan turbine, Thomson Turbine, Fourneyron turbine,Propeller turbine,etc

2. <u>According to the direction of flow throughrunner:</u>

a) Tangentialflowturbine b) Radial flowturbine
c) Axialflowturbine d) Mixed flowturbine

Tangential flowturbine:

The water flows along the tangent to the path of rotation of the runner Ex: Pelton wheel turbine



Radial flowTurbine

- The water flows in the radial direction through therunner.
- **Inward radial flow turbine:** The water enters the outer circumference and flowsradially inwards towards the centre of therunner.
- Ex: Old Francis turbine, Thomson turbine, Girard turbineetc

- **Outward radial flow turbine**: The water enters at the centre and flows radiallyoutwards towards the outer periphery of therunner.
- Ex: Fourneyronturbine.

a) Axialflowturbine**:**

Thewaterflowthroughrunnerwhollyandmainlyalongthedirectionparalleltotheaxisof rotationoftherunner.

Ex:Kaplanturbine,Jonval,Girardaxialflowturbine,Propellerturbine,etc

b) Mixedflowturbines

Thewaterentersthe runnerattheouterperipheryintheradialdirectionandleavesitatthecentreoftheaxial directionparalleltotherotationoftherunner.

Ex:ModernFrancisturbine.

a) Highheadturbines:Theseturbinesworkunderveryhighheads255m-1770mandabove. Requiresrelativelylessquantityofwater.

Ex:Peltonwheelturbineorimpulseturbine.

b) Mediumheadturbines:Theseturbinesarecapableofworkingundermediumheadsranging from60m-250mTheseturbinesrequireslargequantityofwater.

Ex:FrancisTurbine

c) Lowheadturbines:theseturbinesarecapableofworkingundertheheadslessthan60mts. Theseturbinesrequireslargequantityofwater.

Ex:Kaplanturbine,propellerturbine.

a) Lowspecificspeedturbines:specificspeedturbinevariesfrom8.5to30.

Ex:Peltonwheelturbine

b) Mediumspecificspeedturbines:specificspeedvariesfrom50to340 Ex:Francisturbine.

c) Highspecificspeedturbines:specificspeedvariesfrom255-860.

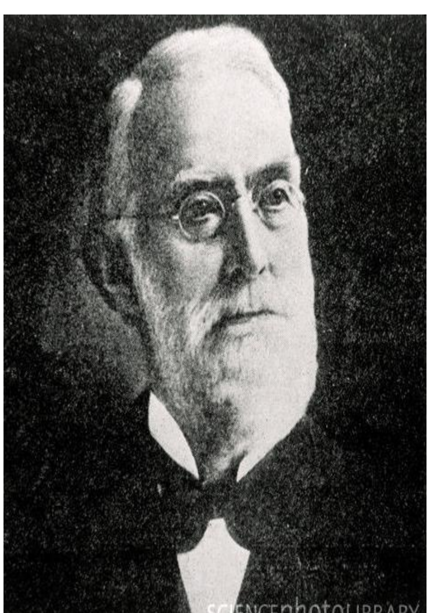
Ex:Kaplanandpropellerturbine.

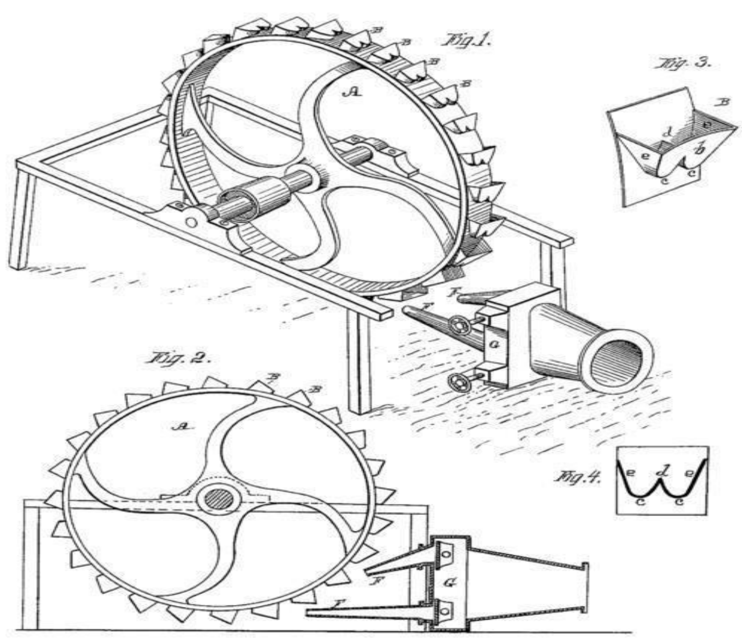
Accordingtothepositionoftheshaft:

a) Horizontaldispositionofshaft

b) Verticaldispositionofshaft. TurbineShaft

<u>PELTON WHEEL TURBINE</u>





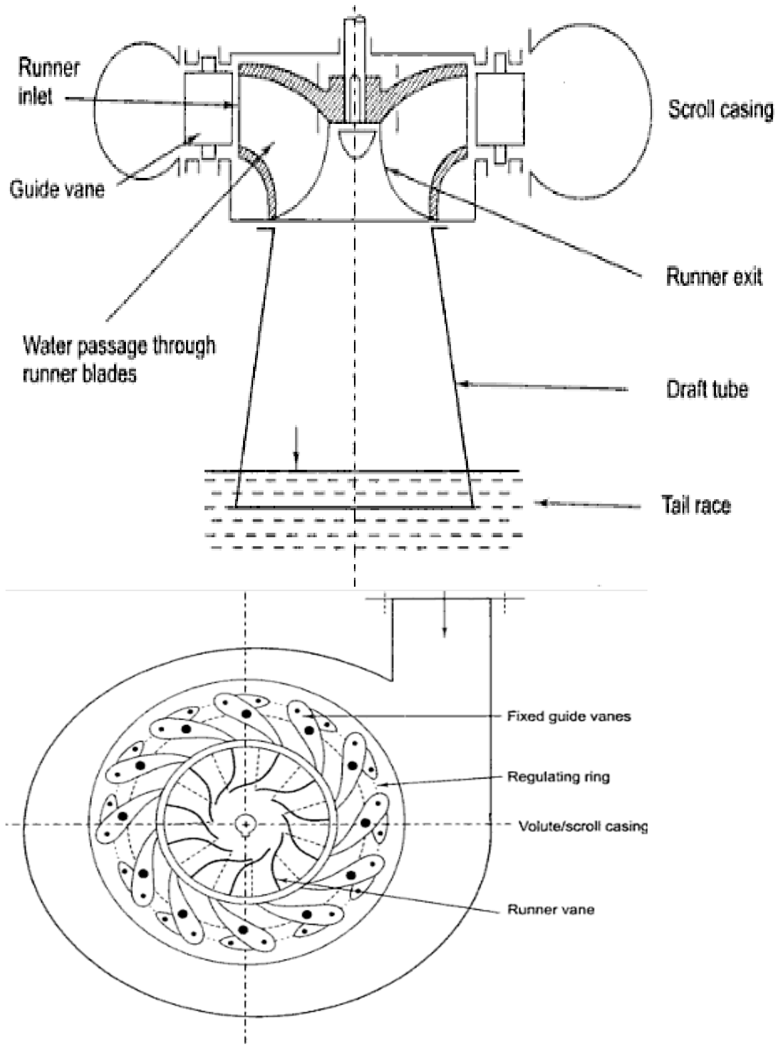
- This is named after Lester A. Pelton, American engineer who contributed much to its development in about 1880. It is well suited for operating under high heads.
- It's an impulse, high head, low specific speed and tangential flow turbine.
- The runner consists of a circular disc with a number of buckets 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 isthat

Theaxialthrustneutralizeseachotherbeingequalandoppositeandhavingbearingsupportingthewheels haftarenotsupportedtoanyaxialthrustorendthrust.

- 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 jetofwaterontthebackofthebuckets,therbybringingthewheelquicklytorestafteritisshut down,otherwiseittakesconsiderabletimetocometorest.

Reaction Turbines:

Reactionturbines,theavailableenergyofwateratinletoftheturbineissumofpressureenergyandkineticenergyandduringtheflowofwaterthroughtherunnerapartofpressureenergyisconvertedintokineticenergy,suchtypeofturbineisreactionturbine.Ex:FrancisTurbine,Kaplan Turbine,PropellerTurbine,etc



Sectional view of Francis Turbine

The main components of Francis Turbine: Scroll Casing:

The water from thepenstockentersthescrollcasingorspiralcasingwhichcompletely surroundstherunner.Thepurposeofcasingistoprovideevendistributionofwateraroundthe circumferenceoftherunnerandtomaintainconstantvelocityofwatersodistributed.

Inordertomaintainconstantvelocityofwaterthroughoutitspatharoundtherunner,the cross-sectionalareaofcasingisgraduallydecreased.Thecasingismadeofcaststeelorplatesteel.

2. Stay Ring:

Fromthescrollcasingthewaterpassesthroughaspeedringorstrayring.Stayringconsistsofouterandlowerringheldtogetherbyseriesoffixedvanescalledstayvanes.

Numberofstayvanesusuallyhalfofthenumberofguidevanes.Stayvaneperformstwo functions,oneistodirectthewaterfromthescrollcasingtotheguidevanesandotheristorestthe loadimposeduponitbytheinternalpressureofwaterandtheweightoftheturbineandelectrical generatorandtransmitsthesametothefoundation.SpeedringismadeofC.IorC.S.

2. Guide Vanes:

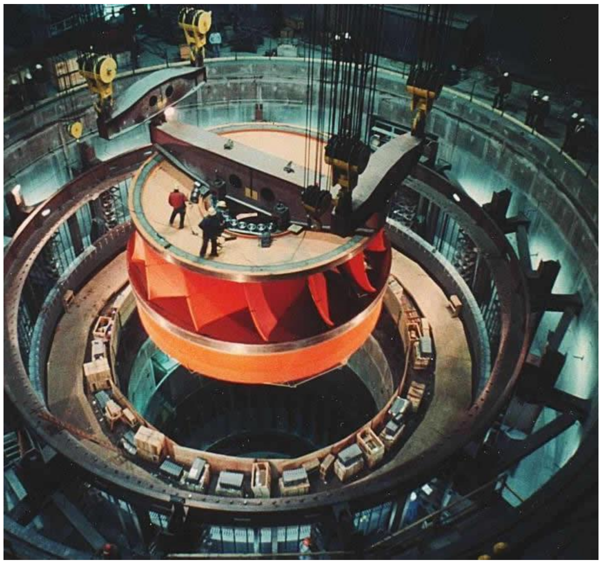
Fromthestayringwaterpassesthroughaseriesofguidevanesprovidedaroundtheperipheryoftherunner. Thefunctionofguidevanesistoregulatethequantityofwatersuppliedtotherunnerandtodirectthewateron totherunnerwithdesignangle.

Theguidevanesareairfoilshapedandmadeof C.SorS.SorP.S.Eachguidevaneis providedwithtwostems;theupperstempassesthroughheadcoverandlowerstemseatsinbottom ring.Byasystemofleversandlinksalltheguidevanesmaybeturnedabouttheirstems,soasto alterthewidthofthepassagebetweentheadjacentguidevanes,therebyallowingavariable quantityofwatertostriketherunner.Theguidevanesareoperatedeitherbymeansofawheelor automaticallybyagovernor.

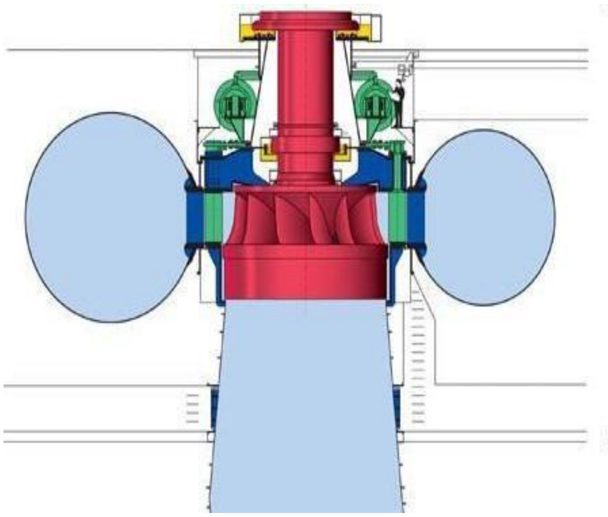
2. RUNNER:

TherunnerofaFrancisturbineconsistsofaseriesofacurvedvanes(from16to24)evenlyarrangedaroundthecircumferenceintheannularspacebetweentwoplates. Thevanesaresoshapedthatwaterenterstherunnerradiallyattheouterperipheryandleavesitaxiallyattheinnerperiphery. Thechangeinthedirectionofflowofwaterfromradialtoaxial,asitpassesthroughtherunner,producesacircumferentialforceontherunnerwhichmakestherunnertorotateandthuscontributestotheusefuloutputoftherunner. RunnervanesaremadeofSSandotherpartsaremadeofCIorCSTherunneriskeyedtoashaftwhichisusuallyofforgedsteel.Thetorqueproducedbythe runneristransmittedtothegeneratorthroughtheshaftwhichisusuallyconnectedtothegenerator shaftbyaboltedflangeconnection.

Francis turbine installation:



KAPLANTURBINE



Kaplan turbineisdevelopedbytheAustrianEngineerViktorKaplan,itissuitableforlowheads andrequireslargequantityofwatertodeveloplargeamountofpower.Sinceitisareactionturbine,itoperatesinanentirelyclosedconduitfromheadracetotailrace.

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 servomotoroperatinginsidethehollowcouplingofturbineandgeneratorshaft.

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.