FM&HM-COURSE FILE

Department: Mechanical Engineering

Year: II B.Tech II Semester

Subject: **FLUID MECHANICS & HYDRAULIC MACHINES**

CODE: 23HPC0308

Regulation: HM23

FACULTY: Mr.K.HEMADRI M.Tech,(Ph.D)

Fluid Mechanics & Hydraulic Machines

Course Objectives: The students completing this course are expected to

- Understand the properties of fluids, manometry, hydrostatic forces acting on different surfaces
- Understand the kinematic and dynamic behaviour through various laws of fluids like continuity, equations, Euler's and Bernoulli's equation and energy and momentum equations.
- Understand the theory of boundary layer, working and performance characteristics of various hydraulic machines like pumps and turbines.

Course Outcomes:

COs	Statements	Blooms Level
CO1	Understand the basic concepts of fluid properties.	L2
CO2	Estimate the mechanics of fluids in static and dynamic conditions.	L5
CO3	Apply the Boundary layer theory, flow separation and dimensional analysis.	L3
CO4	Estimate the hydrodynamic forces of jet on vanes in different positions.	L5
CO5	Understand the working Principles and performance evaluation of hydraulic pump and turbines.	L2

UNIT -1

Fluid statics: Dimensions and units: physical properties of fluids - specific gravity, viscosity and its significance, surface tension, capillarity, vapor pressure. Atmospheric, gauge and vacuum pressure, Measurement of pressure - Manometers - Piezometer, U-tube, inverted and differential manometers. Pascal's & hydrostatic laws.

Buoyancy and floatation: Meta center, stability of floating body. Submerged bodies. Calculation of meta center height. Stability analysis and applications.

UNIT-II

Fluid kinematics: Introduction, flow types. Equation of continuity for one dimensional flow, circulation and vorticity, Stream line, path line and streak lines and stream tube. Stream function and velocity potential function, differences and relation between them. Condition for irrotational flow, flow net, source and sink, doublet and vortex flow.

Fluid dynamics: surface and body forces -Euler's and Bernoulli's equations for flow along a streamline, momentum equation and its applications, force on pipe bend.

Closed conduit flow: Reynold's experiment- Darcy Weisbach equation- Minor losses in pipes- pipes in series and pipes in parallel-total energy line-hydraulic gradient line.

UNIT-III

Boundary Layer Theory: Introduction, momentum integral equation, displacement, momentum and energy thickness, separation of boundary layer, control of flow separation, Stream lined body, Bluff body and its applications, basic concepts of velocity profiles.

Dimensional Analysis: Dimensions and Units, Dimensional Homogeneity, Non dimensionalization of equations, Method of repeating variables and Buckingham Pi Theorem.

UNIT-IV

Basics of turbo machinery: hydrodynamic force of jets on stationary and moving flat, inclined, and curved vanes, jet striking centrally and at tip, velocity diagrams, work done and

efficiency, flow Over radial vanes.

Hydraulic Turbines: classification of turbines, impulse and reaction turbines, Pelton wheel, Francis turbine and Kaplan turbine-working proportions, work done, efficiencies, hydraulic design-draft tube-theory-functions and efficiency.

UNIT-V

Performance of hydraulic turbines: Geometric similarity, Unit and specific quantities, characteristic curves, governing of turbines, selection of type of turbine, cavitation, surge tank, water hammer. Hydraulic systems- hydraulic ram, hydraulic lift, hydraulic coupling. Fluidics-amplifiers, sensors and oscillators. Advantages, limitations and applications.

Centrifugal pumps: classification, working, work done manometric head- losses and efficiencies-specific speed- pumps in series and parallel-performance characteristic curves, cavitation & NPSH.
Reciprocating pumps: Working, Discharge, slip, indicator diagrams.

Textbooks:

1. Y.A. Cengel, J.M. Cimbala, Fluid Mechanics, Fundamentals and Applications, 6/e, McGraw Hill Publications, 2019.
2. Dixon, Fluid Mechanics and Thermodynamics of Turbo machinery, 7/e, Elsevier Publishers, 2014.

Reference Books:

1. P N Modi and S M Seth, Hydraulics & Fluid Mechanics including Hydraulics Machines, Standard Book House, 2017.
2. RKBansal, Fluid Mechanics and Hydraulic Machines, 10/e, Laxmi Publications (P) Ltd, 2019.
3. Rajput, Fluid Mechanics and Hydraulic Machines, S Chand & Company, 2016.
4. D.S.Kumar, Fluid Mechanics and Fluid Power Engineering, S K Kataria &Sons, 2013.
5. D.Rama Durgaiah, Fluid Mechanics and Machinery,1/e, New Age International, 2002.

Online Learning Resources:

https://archive.nptel.ac.in/courses/112/105/112105206/
https://archive.nptel.ac.in/courses/112/104/112104118/
https://www.edx.org/learn/fluid-mechanics
https://onlinecourses.nptel.ac.in/noc20_ce30/previewnptel.ac.in
www.coursera.org/learn/fluid-powerera

UNIT-I

1. FLUID STATICS

UNITS AND DIMENSIONS:

A dimension is a name which describes the measurable characteristics of an object such as mass, length and temperature etc. a unit is accepted standard for measuring the dimension. The dimensions used are expressed in four fundamental dimensions namely Mass, Length, Time and Temperature. Mass (M) – Kg

Length (L) – m

Time (T) – S

Temperature (t) – ^{0}C or K (Kelvin)

1. **Density:** Mass per unit volume=kg/m^3
2. **Newton:** Unit of force expressed in terms of mass and acceleration, according to Newton's 2nd law motion. Newton is that force which when applied to a mass of 1 kg gives an acceleration 1m/Sec^2. F=Mass x Acceleration = kg – m/sec^2 = N.
3. **Pascal:** A Pascal is the pressure produced by a force of Newton uniformly applied over an area of 1 m^2. Pressure = Force per unit area = N/ m^2 = Pascal or P_a.
4. **Joule:** A joule is the work done when the point of application of force of 1 Newton is displaced Work = Force per unit area = N m = J or Joule.
5. **Watt:** A Watt represents a work equivalent of a Joule done per second. Power = Work done per unit time = J/ Sec = W or Watt.

Density or Mass Density: The density or mass density of a fluid is defined as the ratio of the mass of the fluid to its volume. Thus the mass per unit volume of the fluid is called density. It is denoted by ℓ The unit of mass density is Kg/m^3

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

The value of density of water is 1000Kg/m^3.

Specific weight or Specific density: It is the ratio between the weights of the fluid to its volume. The weight per unit volume of the fluid is called weight density and it is denoted by **w**.

$$w = \frac{\text{Weight of fluid}}{\text{Volume of fluid}} = \frac{\text{Mass of fluid} \times \text{Acceleration due to gravity}}{\text{Volume of fluid}}$$

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

Specific volume: It is defined as the volume of the fluid occupied by a unit mass or volume per unit mass of fluid is called Specific volume.

$$\text{Specific volume} = \frac{\text{Volume of the fluid}}{\text{Mass of fluid}} = \frac{1}{\frac{\text{Mass of fluid}}{\text{Volume of the fluid}}} = \frac{1}{\rho}$$

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

<u>Specific Gravity:</u> It is defined as the ratio of the Weight density (or density) of a fluid to the Weight density (or density) of a standard fluid. For liquids the standard fluid taken is water and for gases the standard liquid taken is air. The Specific gravity is also called relative density. It is a dimension less quantity and it is denoted by **s**.

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

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

Weight density of liquid=S × weight density of water = S×1000 ×9.81 N/ m^3
Density of liquid= S × density of water = S × 1000 Kg/ m^3

If the specific gravity of fluid is known, then the density of fluid will be equal to specific gravity of the fluid multiplied by the density of water

Example: The specific gravity of mercury is 13.6

Hence density of mercury = 13.6 ×1000 = 13600 Kg/ m^3

<u>VISCOSITY</u>: It is defined as the property of a fluid which offers resistance to the movement of one layer of the fluid over another adjacent layer of the fluid. When the two layers of a fluid, at a distance 'dy' apart, move one over the other at different velocities, say u and u+du. The viscosity together with relative velocities causes a shear stress acting between the fluid layers.

The top layer causes a shear stress on the adjacent lower layer while the lower layer causes a shear stress on the adjacent top layer.

This shear stress is proportional to the rate of change of velocity with respect to y. it is denoted by symbol τ (tau)

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

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

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

From the above equation, we have
$$\mu = \frac{\tau}{\left(\frac{du}{dy}\right)}$$

Thus, viscosity is also defined as the shear stress required producing unit rate of shear strain.

The unit of viscosity in CGS is called poise and is equal to dyne-see/ cm^2

KINEMATIC VISCOSITY: It is defined as the ratio between dynamic viscosity and density of fluid. It is denoted by symbol ϑ (nu)

$$\vartheta = \frac{\text{viscosity}}{\text{density}} = \frac{\mu}{\rho}$$

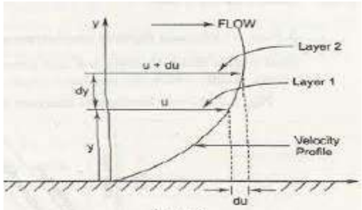


Fig. 1.2

The unit of kinematic viscosity is m^2 /sec

Thus one stoke = cm^2/sec = $\left(\frac{1}{100}\right)^2$ m^2/sec= 10^{-4} m^2/sec

NEWTONS LAW OF VISCOSITY: It states that the shear stress (τ) on a fluid element layer is directly proportional to the rate of shear strain. The constant of proportionality is called the co- efficient of viscosity .It is expressed as: Fluids which obey above relation are known as

NEWTONIAN fluids and fluids which do not obey the above relation are called NON-NEWTONIAN

UNITS OF VISCOSITY

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

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

$$\mu = \frac{\text{Shear stress}}{\frac{\text{Change of velocity}}{\text{change of distance}}}$$

$$= \frac{\text{Force/Area}}{\left(\frac{\text{Length}}{\text{Time}}\right) \times \frac{1}{\text{Length}}} = \frac{\text{Force/(Length)}^2}{\frac{1}{\text{Time}}} = \frac{\text{Force} \times \text{Time}}{(\text{Length})^2}$$

In MKS System Force is represented by (Kg f) and Length by meters (m)

In CGS System Force is represented by dyne and length by cm and

In SI System Force is represented by Newton (N) and Length by meter (m)

MKS unit of Viscosity $= \frac{\text{Kg f} - \text{Sec}}{\text{m}^2}$

CGS Unit of Viscosity $= \frac{\text{dyne} - \text{sec}}{\text{cm}^2}$

S I Unit of Viscosity = $\frac{\text{Newton - sec}}{\text{m}^2} = \frac{\text{N}_\text{S}}{\text{m}^2}$

The unit of Viscosity in CGS is called **Poise,** which is equal to $\frac{(\text{dyne}-\text{sec})}{\text{cm}^2}$.

The numerical conversion of the unit of viscosity from MKS units to CGS unit is as follows:

$$\frac{\text{One Kg f}-\text{sec}}{\text{m}^2} = \frac{9.81\text{N}-\text{sec}}{\text{m}^2}$$ (1Kgf = 9.81 Newton)

But one Newton = One Kg (mass) × one $\left(\frac{\text{m}}{\text{sec}^2}\right)$ (Acceleration)

$= \frac{(1000gms \times 100cm)}{sec^2}$

$\times 100 \frac{gm-cm}{sec^2}$ = 1000

= 1000 × 100 dyne (dyne = gm $\times \frac{cm}{sec^2}$)

$\frac{\text{One Kg f-sec}}{m^2}$ = 9.81 $\times 100000 \frac{dyne-sec}{cm^2}$.

$= 9.81 \times 100000 \frac{dyne-sec}{cm^2} = 9.81 \times 100000 \frac{dyne-sec}{100 \times 100 \times cm^2}$

$= 98.1 \frac{dyne-sec}{cm^2}$

= 98.1 Poise $\frac{(dyne-sec)}{cm^2}$ = Poise

$\frac{\text{One Ns}}{m^2} = \frac{9.81}{9.81}$ Poise = 10 Poise

Or 1 Poise $= \frac{1}{10} \frac{Ns}{m^2}$

i) For **liquids** $\mu = \mu_0 \left(\frac{1}{1+\alpha t+\beta t^2}\right)$

Where, μ = Viscosity of liquid at t c in Poise.

μ_0 = Viscosity of liquid at o c α and β are constants for the Liquid.

For Water, μ_0 = 1.79 x 10^{-3} poise, α =0.03368 and β = 0.000221

The above equation shows that the increase in temp. The Viscosity decreases.

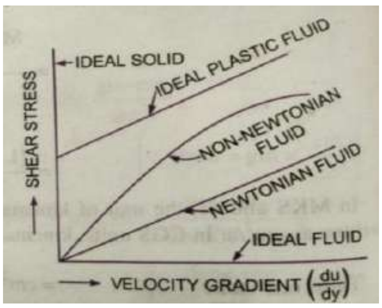
ii) **For gases** $\mu = \mu_0 + \alpha t - \beta t^2$

For air μ_0 =0.000017, α =0.056 x 10^{-6}, β = 0.118 x 10^{-9}

The above equation shows that with increase of temp. The Viscosity increases.

<u>TYPES OF FLUIDS</u>: The fluids may be classified in to the following five types.

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



1. **Ideal fluid**: A fluid which is compressible and is having no viscosity is known as ideal fluid. It is only an imaginary fluid as all fluids have some viscosity.

2. **Real fluid**: A fluid possessing a viscosity is known as real fluid. All fluids in actual practice are real fluids.

3. **Newtonian fluid**: A real fluid, in which the stress is directly proportional to the rate of shear strain, is known as Newtonian fluid.

4. **Non-Newtonian fluid**: A real fluid in which shear stress is not Proportional to the rate of shear strain is known as NonNewtonian fluid.

5. **Ideal plastic fluid:** A fluid, in which shear stress is more than the yield value and shear stress is proportional to the rate of shear strain is known as ideal plastic fluid.

SURFACE TENSION:

Surface tension is defined as the tensile force acting on the surface of a liquid is contact with a gas or on the surface behaves like a membrane under tension. The magnitude of this force per unit length of free surface will have the same value as the surface energy per unit area. It is denoted by σ

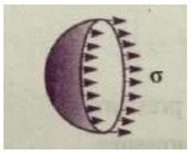
(sigma). In MKS units it is expressed as Kg f/m while in SI units as N/m

Surface Tension on Liquid Droplet:

Consider a small spherical droplet of a liquid of radius 'r' on the entire surface of the droplet, the tensile force due to surface tension will be acting Let σ = surface tension of the liquid p= pressure intensity inside the droplet (In excess of outside pressure intensity) d= Diameter of droplet

Let, the droplet is cut in to two halves. The forces acting on one half (say left half) will be

i) Tensile force due to surface tension acting around the circumference of the cut portion
$= \sigma \times \text{circumference} \quad = \sigma \times \pi d$

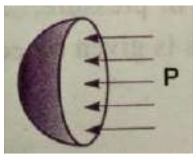


ii) Pressure force on the area $\frac{\pi}{4} d^2 = p \times \frac{\pi}{4} d^2$

These two forces will be equal to and opposite under equilibrium conditions i.e.

$p \times \frac{\pi}{4} d^2 = \sigma \pi d$, $\quad p = \frac{\sigma \pi d}{\frac{\pi}{4} d^2}$ $\quad p = \frac{4\sigma}{d}$

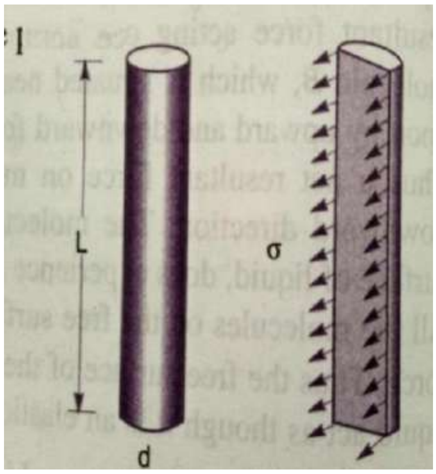
Surface Tension on a Hallow Bubble:



A hallows bubble like soap in air has two surfaces in contact with air, one inside and other outside. Thus, two surfaces are subjected to surface tension.

$P = 8\sigma/d$

SURFACE TENSION ON A LIQUID JET:



Consider a liquid jet of diameter 'd' length 'L'

Let, p = pressure intensity inside the liquid jet above the outside pressure   σ = surface tension of the liquid

Consider the equilibrium of the semi- jet

Force due to pressure = p × area of the semi-jet = p × L × d

Force due to surface tension = σ × 2L

p × L × d = σ× 2L,

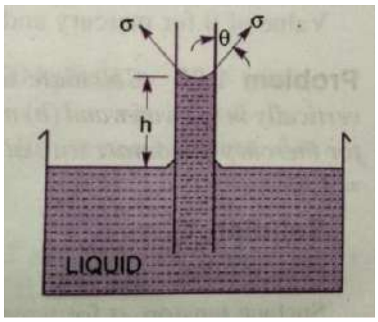
$$\mathbf{p = \frac{2\sigma}{d}}$$

CAPILLARITY

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

EXPRESSION FOR CAPILLARY RISE:

Consider a glass tube of small diameter 'd' opened at both ends and is inserted in a liquid; the liquid will rise in the tube above the level of the liquid outside the tube.



Let 'h' be the height of the liquid in the tube. Under a state of equilibrium, the weight of the liquid of height 'h' is balanced by the force at the surface of the liquid in the tube. But, the force at the surface of the liquid in the tube is due to surface tension.

Let σ = surface tension of liquid

ϴ = Angle of contact between the liquid and glass tube

The weight of the liquid of height 'h' in the tube

= (area of the tube × h) × ρ × g = $\frac{\pi}{4} d^2 \times h \times \rho \times g$

Where 'ρ' is the density of the liquid.

The vertical component of the surface tensile force = (σ× circumference) × cosϴ = σ × πd × cosϴ

For equilibrium, $\frac{\pi}{4}\ d^2 \times h \times \rho \times g = \sigma\ \pi d \cos\Theta$, $\qquad h = \frac{\sigma \pi d \cos\Theta}{\frac{\pi}{4} d^2\ \rho \times g} = \frac{4\sigma \cos\emptyset}{\rho \times g \times d}$.

The value of Θ is equal to '0' between water and clean glass tube, then cosΘ = 1,

$$\boldsymbol{h} = \frac{4\sigma}{\rho \times g \times d}$$

VAPOUR PRESSURE AND CAVITATION

A change from the liquid state to the gaseous state is known as Vaporizations. The vaporization (which depends upon the prevailing pressure and temperature condition) occurs because of continuous escaping of the molecules through the free liquid surface.

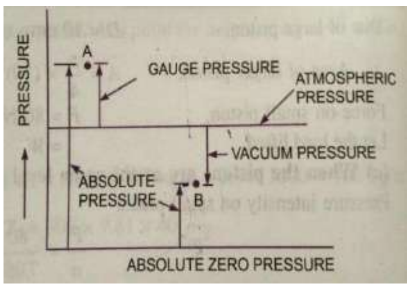
Consider a liquid at a temp. of 20°C and pressure is atmospheric is confined in a closed vessel. This liquid will vaporize at 100°C, the molecules escape from the free surface of the liquid and get accumulated in the space between the free liquid surface and top of the vessel. These accumulated vapours exert a pressure on the liquid surface. This pressure is known as vapour pressure of the liquid or pressure at which the liquid is converted in to vapours.

Consider the same liquid at 20°c at atmospheric pressure in the closed vessel and the pressure above the liquid surface is reduced by some means; the boiling temperature will also reduce. If the pressure is reduced to such an extent that it becomes equal to or less than the vapour pressure, the boiling of the liquid will start, though the temperature of the liquid is 20°C. Thus, the liquid may boil at the ordinary temperature, if the pressure above the liquid surface is reduced so as to be equal or less than the vapour pressure of the liquid at that temperature.

Now, consider a flowing system, if the pressure at any point in this flowing liquid becomes equal to or less than the vapour pressure, the vaporisations of the liquid starts. The bubbles of these vapours are carried by the flowing liquid in to the region of high pressure where they collapse, giving rise to impact pressure. The pressure developed by the collapsing bubbles is so high that the material from the adjoining boundaries gets eroded and cavities are formed on them. This phenomenon is known as **CAVITATION.**

Hence the cavitations is the phenomenon of formation of vapour bubbles of a flowing liquid in a region where the pressure of the liquid falls below the vapour pressure and sudden collapsing of these vapour bubbles in a region of high pressure,. When the vapour bubbles collapse, a very high pressure is created. The metallic surface, above which the liquid is flowing, is subjected to these high pressures, which cause pitting actions on the surface. Thus cavities are formed on the metallic surface and hence the name is **cavitation.**

ABSOLUTE, GAUGE, ATMOSPHERIC and VACCUM PRESSURES



The pressure on a fluid is measured in two different systems. In one system, it is measured above the absolute zero or complete vacuum and it is called the Absolute pressure and in other system, pressure is measured above the atmospheric pressure and is called Gauge pressure.

1. ABSOLUTE PRESSURE: It is defined as the pressure which is measured with reference to absolute vacuum pressure

2. GAUGE PRESSURE: It is defined as the pressure, which is measured with the help of a pressure measuring instrument, in which the atmospheric pressure is taken as datum. The atmospheric on the scale is marked as zero.

3. VACUUM PRESSURE: It is defined as the pressure below the atmospheric pressure

i) Absolute pressure = Atmospheric pressure+ gauge pressure $p_{ab} = p_{atm} + p_{guag}$ ii) Vacuum pressure = Atmospheric pressure - Absolute pressure The atmospheric pressure at sea level at 15^0C is 10.13N/cm^2 or 101.3KN/m^2 in S I Units and 1.033 Kg f/cm^2 in M K S System.

The atmospheric pressure head is 760mm of mercury or 10.33m of water.

MEASUREMENT OF PRESSURE

The pressure of a fluid is measured by the fallowing devices.

1. Manometers 2.Mechanical gauges.

1. Manometers: Manometers are defined as the devices used for measuring the pressure at a point in a fluid by balancing the column of fluid by the same or another column of fluid. They are classified as:

a) Simple Manometers b) Differential Manometers.

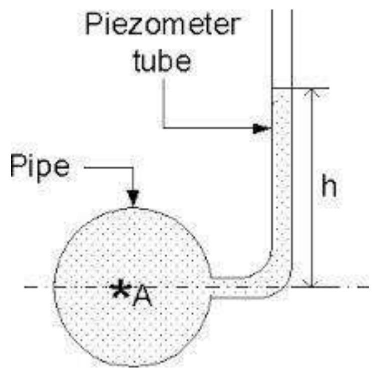
2. Mechanical Gauges: are defined as the devices used for measuring the pressure by balancing the fluid column by the spring or dead weight. The commonly used Mechanical pressure gauges are: a) Diaphragm pressure gauge b) Bourdon tube pressure gauge

c) Dead – Weight pressure gauge d) Bellows pressure gauge.

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

The common types of simple manometers are:

1. Piezo meter.
2. U-tube manometer.
3. Single column manometer.



1. Piezometer: It is a simplest form of manometer used for measuring gauge pressure. One end of this manometer is connected to the point where pressure is to be measured and other end is open to the atmosphere. The rise of liquid in the Piezometer gives pressure head at that point A.

The height of liquid say water is 'h' in piezometer tube, then

Pressure at A = $\rho g h \frac{N}{m^2}$

2. U- tube Manometer:

It consists of a glass tube bent in u-shape, one end of which is connected to a point at which pressure is to be measured and other end remains open to the atmosphere. The tube generally contains mercury or any other liquid whose specific gravity is greater than the specific gravity of the liquid whose pressure is to be measured.

a) For Gauge Pressure: Let B is the point at which pressure is to be measured, whose value is p. The datum line A – A

Let h_1 = height of light liquid above datum line h_2 =height of heavy liquid above datum line S_1 = sp. gravity of light liquid ρ_1= density of light liquid = 1000 S_1

S_2 = sp. gravity of heavy liquid

ρ_2= density of heavy liquid = 1000 S_2

As the pressure is the same for the horizontal surface. Hence the pressure above the horizontal datum line A – A in the left column and the right column of U – tube manometer should be same.

Pressure above A—A ion the left column = $p + \rho_1 gh_1$

Pressure above A – A in the left column = $\rho_2 gh_2$

Hence equating the two pressures $p + \rho_1 gh_1 = \rho_2 gh_2$ **p =**

$\boldsymbol{\rho_2 gh_2 - \rho_1 gh_1}$

(b) For Vacuum Pressure:

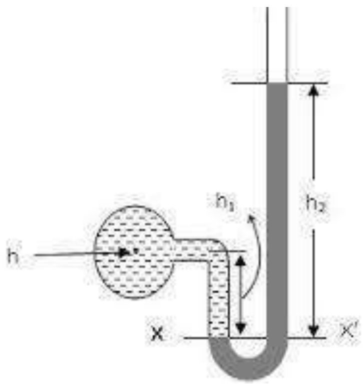
For measuring vacuum pressure, the level of heavy fluid in the manometer will be as shown in fig.

Pressure above A A in the left column = $\rho_2 g\, h_2 + \rho_1 g\, h_1 + P$

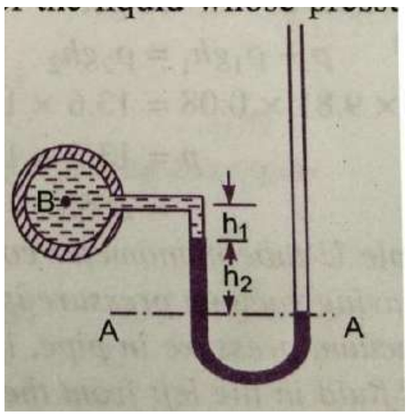
Pressure head in the right column above A A = O

$\rho_2 g\, h_2 + \rho_1 g\, h_1 + P = O$

$\mathbf{p = -(\rho_2 g\, h_2 + \rho_1 g\, h_1)}$



(a) For Gauge Pressure



(b) For Vacuum Pressure

SINGLE COLUMN MANOMETER:

Single column manometer is a modified form of a U- tube manometer in which a reservoir, having a large cross sectional area (about. 100 times) as compared to the area of tube is connected to one of the limbs (say left limb) of the manometer. Due to large cross sectional area of the reservoir for any variation in pressure, the change in the liquid level in the reservoir will be very small which may be neglected and hence the pressure is given by the height of the liquid in the other limb. The other limb may be vertical or inclined. Thus, there are two types of single column manometer

1. Vertical single column manometer. 2. Inclined single column manometer.

VERTICAL SINGLE COLUMN MANOMETER:

Let X – X be the datum line in the reservoir and in the right limb of the manometer, when it is connected to the pipe, when the Manometer is connected to the pipe, due to high pressure at A The heavy in the reservoir will be pushed downwards and will rise in the right limb.

Let, Δh= fall of heavy liquid in the reservoir

h_2= rise of heavy liquid in the right limb h_1= height of the

centre of the pipe above X – X p_A= Pressure at A, which is

to be measured. A = Cross- sectional area of the reservoir a

= cross sectional area of the right limb

S_1= Specific. Gravity of liquid in pipe

S_2= sp. Gravity of heavy liquid in the reservoir and right limb

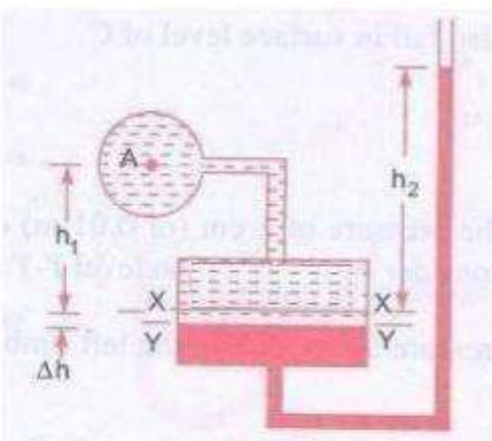
ρ_1 = density of liquid in pipe

ρ_2= density of liquid in reservoir

Fall of heavy liquid reservoir will cause a rise of heavy liquid level in the right limb

$$A \times \Delta h = a \times h_2$$

$$\Delta h = \frac{a \times h_2}{A} \quad \text{-------- (1)}$$



Now consider the datum line Y –Y

The pressure in the right limb above Y – Y

$$= \rho_2 \times g \times (\Delta h + h_2)$$

Pressure in the left limb above Y—Y

$$= \rho_1 \times g \times (\Delta h + h_1) + P_A$$

Equating the pressures, we have

$$\rho_2 \, g \times (\Delta h + h_2) = \rho_1 \times g \times (\Delta h + h_1) + p_A$$

$$p_A = \rho_2 \times g \times (\Delta h + h_2) - \rho_1 \times g \times (\Delta h + h_1)$$

$$= \Delta h \, (\rho_2 g \; \rho_1 g) + h_2 \rho_2 g - h_1 \rho_1 g$$

But, from eq (1) $\Delta h = \frac{a \times h_2}{A}$

$p_A = \frac{a \times h_2}{A}(\rho_2 g - \rho_1 g) + h_2\rho_2 g - h_1\rho_1 g$

As the area A is very large as compared to a, hence the ratio $\frac{a}{A}$ becomes very small and can be neglected

Then, **$p_A = h_2\rho_2 g - h_1\rho_1 g$ ---- (2)**

INCLINED SINGLE COLUMN MANOMETER:

The manometer is more sensitive. Due to inclination the distance moved by heavy liquid in the right limb will be more.

Let L= length of heavy liquid moved in the rite limb Θ =

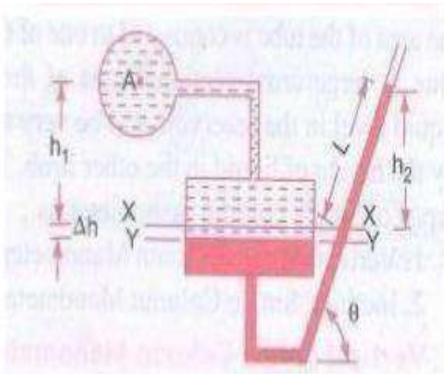
inclination of right. Limb with horizontal.

H_2 = vertical rise of heavy liquid in the right limb above X – X

= L sinΘ

From above eq (2), the pressure at A is $p_A = h_2\rho_2 g - h_1\rho_1 g$

Substituting the value of h_2 $p_A = L \sin\Theta\, \rho_2 g - h_1\rho_1 g$



DIFFERENTIAL MANOMETERS:

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

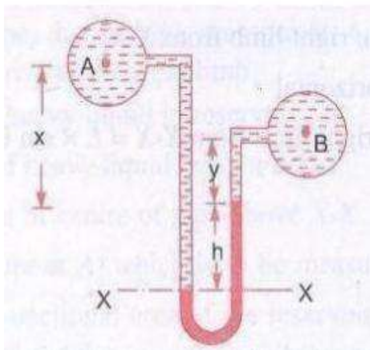
1. U- Tube differential manometer 2. Inverted U- tube differential manometer.

1. U- Tube differential manometer:

a) Let the two points A and B are at different levels and also contains liquids of different sp.gr.

b) These points are connected to the U – Tube differential manometer. Let the pressure at A and B are p_A and p_B.

Let h = Difference of mercury levels in the u – tube y = Distance of centre of B from the mercury level in the right limb x = Distance of centre of A from the mercury level in the left limb



ρ_1 = Density of liquid A

ρ_2 = Density of liquid B

ρ_g = Density of heavy liquid or mercury

Taking datum line at X – X

Pressure above X – X in the left limb $=\rho_1 g(h + x) + p_A$ (where p_A = Pressure at A)

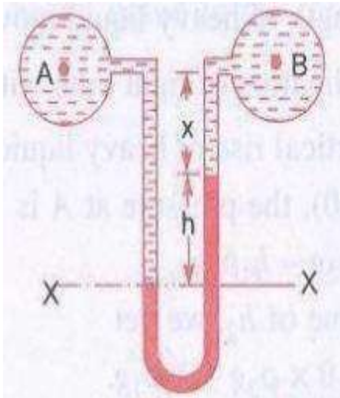
Pressure above X – X in the right limb = ρ_g g h + ρ_2g y + P_B (where p_B = Pressure at B) Equating the above two pressures, we have

$$\rho_1 g(h + x) + p_A = \rho_g g h + \rho_2 g y + p_B$$

$$p_A - p_B = \rho_g g h + \rho_2 g y - \rho_1 g (h + x)$$

$$= h g (\rho_g - \rho_1) + \rho_2 g y - \rho_1 g x$$

∴ Difference of Pressures at A and B = h g (ρ_g - ℓ_1) + ρ_2 g y - ρ_1g x



Let the two points A and B arte at the same level and contains the same liquid of density ℓ_1

Then pressure above X – X in the right limb = ρ_g g h + ρ_1g x + P_B

Pressure above X—X in the left limb = ρ_1g (h + x) + P_A Equating the two pressures

$$\rho_g g h + \rho_1 g x + p_B = \rho_1 g (h + x) + p_A$$

$$P_A - P_B = \rho_g g h + \rho_1 g x - \rho_1 g(h + x)$$

$$= g h (\rho_g - \rho_1)$$

Difference of pressure at A and B = g h (ρ_g - ρ_1)

Inverted U – Tube differential manometer

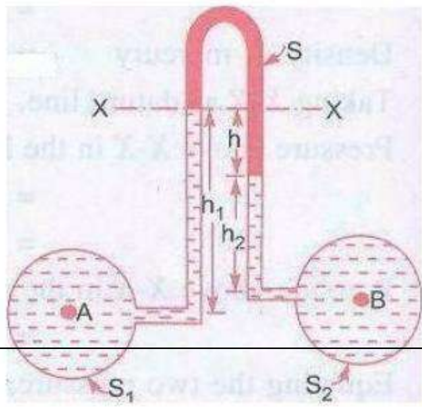
It consists of a inverted U – tube, containing a light liquid. The two ends of the tube are connected to the points whose difference of pressure is to be measured. It is used for measuring difference of low pressures. Let an inverted U – tube differential manometer connected to the two points A and B. Let pressure at A is more than pressure at B.

Let h_1 = Height of the liquid in the left limb below the datum line X-X h_2 = Height

of the liquid in the right limb.

h = Difference of height of liquid

ρ_1 = Density of liquid A



ρ_2 = Density of liquid B

ρ_s = Density of light liquid p_A = Pressure at A p_B= Pressure at B

Taking x – x as datum line

The pressure in the left limb below $x - x$ = $p_A - \rho_1\ g\ h_1$

Pressure in the right limb below $x - x$ = $p_B - \rho_2\ g\ h_2 - \rho_s\ g\ h$ Equating

the above two pressures $p_A - \rho_1\ g\ h_1 = p_B - \rho_2\ g\ h_2 - \rho_s\ g\ h$ p_A

$- p_B = \rho_1\ g\ h_1 - \rho_2\ g\ h_2 - \rho_s\ g\ h$

Difference of pressure at A and B = $\rho_1\ g\ h_1 - \rho_2\ g\ h_2 - \rho_s\ g\ h$

PROBLEMS

1. Calculate the density, specific weight and weight of one liter of petrol of specific gravity = 0.7

Sol: i) Density of a liquid = S × Density of water = S x 1000 kg/ m^3

ρ= 0.7 × 1000

ρ = 700Kg/m^3

ii) Specific weight w = $\rho \times g = 700 \times 9.81 = 6867$ N/ m^3

iii) Weight (w) Volume = 1 liter = 1 × 1000 cm^3 = $\frac{1000}{10^6} m^3$ = **<u>0.001m³</u>**

We know that, specific weight w = $\frac{\text{weight of fluid}}{\text{volume of the fluid}}$

Weight of petrol = w× volume of petrol

= w × 0.001

= 6867 × 0.001 **= <u>6.867 N</u>**

2. Two horizontal plates are placed 1.25 cm apart, the space between them being filled with oil of viscosity 14 poises. Calculate shear stress in oil, if the upper plate is moved velocity of 2.5 m/sec.

Sol: Given distance between the plates dy= 1.25 cm = 0.0125m

Viscosity μ = 14 poise = $\frac{14}{10}$ N s/m^2

Velocity of upper plate u = 2.5 m/sec

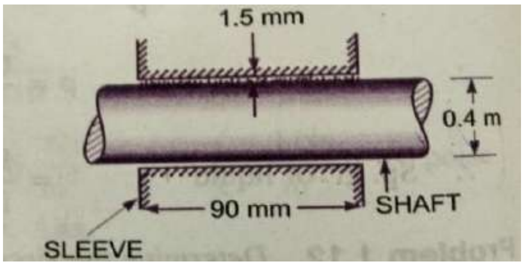
Shear stress $\tau = \mu \frac{du}{dy}$

Where du = change of velocity between plates = u – 0 = u = 2.5 m/sec

$\tau = \frac{14}{10} \times \frac{2.5}{0.0125}$

Shear stress τ = 280 N/m^2

3. The dynamic viscosity of oil used for lubrication between a shaft and sleeve is 6 poise. The shaft dia. is 0.4m and rotates at 190 rpm. Calculate the power lost in the bearing for a sleeve length of 90mm. The thickness foil film is 1.5mm



Sol: Given, Viscosity μ = 6 poise $= \frac{6}{10}\frac{N\,s}{m^2} = 0.6 \frac{N\,s}{m^2}$

Dia. of shaft D = 0.4M

Speed of shaft N = 190 rpm

Sleeve length L = 90mm = 90×10^{-3} m

Thickness of a film t= 1.5mm = 1.5mm = 1.5×10^{-3} m

Tangential velocity of shaft = u = $\frac{\pi DN}{60} = \frac{\pi \times 0.4 \times 190}{60}$ = 3.98 m/sec

Using the relation $\tau = \mu \frac{du}{dy}$

Where du = change of velocity = u – 0 = u = 3.98 m/sec dy = change of distance = t = 1.5×10^{-3}m

$\tau = 0.6 \times \frac{3.98}{1.5 \times 10^{-3}}$ = 1592 N/m^2

This is the shear stress on the shaft

Shear force on the shaft F = shear stress $\times$ area =1592 $\times$ π DL = 1592 $\times$ π $\times$ 0.4 x 90 $\times$ 10^{-3} = 180.05 N

Torque on the shaft T = Force $\times \frac{D}{2}$ = 180.05 $\times \frac{0.4}{2}$ = 36.01 Nm

Power lost = $\frac{2\pi NT}{60}$ = 36.01 $\times \frac{2\pi \times 190}{60} \times$ 36.01

Power lost = 716.48 W

4. A cylinder 0.12m radius rotates concentrically inside a fixed cylinder of 0.13 m radius. Both cylinders are 0.3m long. Determine the viscosity of liquid which fills the space between the cylinders, if a torque of 0.88 Nm is required to maintain an angular velocity of 2 π rad / sec.

Sol : Diameter of inner cylinder = 0.24m

Diameter of outer cylinder = 0.26 m

Length of cylinder L = 0.3 m

Torque T = 0.88 NM w
= 2 π N/60 = 2 π

N = speed = 60rpm

Let the viscosity = μ

Tangential velocity of cylinder u = $\frac{\pi DN}{60} = \frac{\pi \times 0.24 \times 60}{60}$ = 0.7536 m/sec

Surface area of cylinder A = π DL = π × 0.24 x 0.3 = 0.226 m^2

Now using the relation $\tau = \mu \frac{du}{dy}$

Where du = u – 0 = 0.7536 m/sec

dy = $\frac{26-0.24}{2}$ 0. = 0.02
= 0.01 m $\tau = \mu \times \frac{0.7536}{0.01}$

Shear force, F = shear stress x area

= μ × 75.36 × 0.226 = 17.03 μ

Torque T = F ×D/2 = 17.03 μ $\times \frac{0.24}{2}$

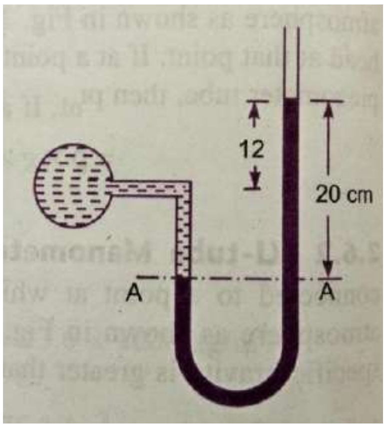
0.88 = μ × 2.0436

$\mu = \frac{0.88}{2.0436}$

= 0.4306 Ns/ m^2

= 0.4306 × 10 poise

Viscosity of liquid = 4.306 poise



5. The right limb of a simple U – tube manometer containing mercury is open to the atmosphere, while the left limb is connected to a pipe in which a fluid of sp.gr.0.9 is flowing. The centre of pipe is 12cm below the level of mercury in the right limb. Find the pressure of fluid in the pipe, if the difference of mercury level in the two limbs is 20 cm.

Given, Sp.gr. of liquid S_1= 0.9

Density of fluid $\rho_1 = S_1 \times 1000 = 0.9 \times 1000$ =900 kg/ m^3

Sp.gr. of mercury S_2 = 13.6

Density of mercury ρ_2 =13.6× 1000 = 13600 kg/m^3

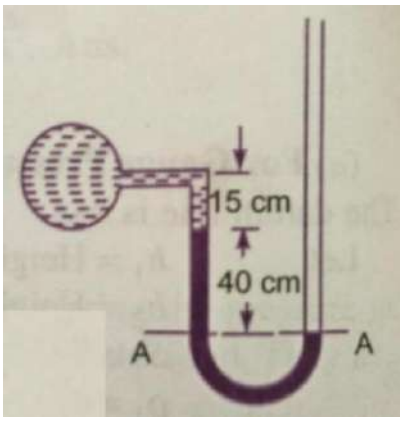
Difference of mercury level h_2 = 20cm = 0.2m

Height of the fluid from A – A h_1 = 20 – 12 = 8cm = 0.08 m

Let 'P' be the pressure of fluid in pipe Equating pressure at A – A, we get $p + \rho_1 g h_1 = \rho_2 g h_2$ $p + 900 \times 9.81 \times 0.08 = 13.6 \times 1000 \times 9.81 \times 0.2$ $p = 13.6 \times 1000 \times 9.81 \times 0.2 - 900 \times 9.81 \times 0.08$ $p = 26683 - 706$ $p = 25977$ N/m^2 $p = 2.597$ N/cm^2

Pressure of fluid = 2.597 N/ cm^2

6. A simple U – tube manometer containing mercury is connected to a pipe in which a fluid of sp.gr. 0.8 And having vacuum pressure is flowing. The other end of the manometer is open to atmosphere. Find the vacuum pressure in pipe, if the difference of mercury level in the two limbs is 40cm. and the height of the fluid in the left tube from the centre of pipe is 15cm below.



Given,

Sp.gr of fluid $S_1 = 0.8$

Sp.gr. of mercury $S_2 = 13.6$

Density of the fluid = $S_1 \times 1000 = 0.8 \times 1000 = 800$

Density of mercury = 13.6×1000

Difference of mercury level h_2 = 40cm = 0.4m

Height of the liquid in the left limb = 15cm =0.15m

Let the pressure in the pipe = p

Equating pressures above datum line A-- A

$\rho_2 g h_2 + \rho_1 g h_1 + P = 0$

$P = - [\rho_2 g h_2 + \rho_1 g h_1]$

$= - [13.6 \times 1000 \times 9.81 \times 0.4 + 800 \times 9.81 \times 0.15]$

$= 53366.4 + 1177.2$

= -54543.6 N/m^2

P = - 5.454 N/cm^2

7. What are the gauge pressure and absolute pressure at a point 3m below the surface of a liquid having a density of 1.53 x 10^3 kg/m^3? If the atmospheric pressure is equivalent to 750mm of mercury. The specific gravity of mercury is 13.6 and density of water 1000 kg/m^3 Given:

Depth of the liquid, z_1 = 3m

Density of liquid $\rho_1 = 1.53 \times 10^3$ kg/ m^3

Atmospheric pressure head z_0 = 750 mm of mercury = $\frac{750}{1000}$ = 0.75m of Hg Atmospheric pressure $p_{atm} = \rho_0 \times g \times z_0$

Where ρ_0 = density of Hg = sp.gr. of mercury x density of water

$= 13.6 \times 1000$ kg/ m^3

And z_0 = pressure head in terms of mercury = 0.75m of Hg

$P_{atm} = (13.6 \times 1000) \times 9.81 \times 0.75$ N/m^2 = 100062 N/ m^2

Pressure at a point, which is at a depth of 3m from the free surface of the liquid is

$P = \rho_1 \times g \times z_1 = 1.53 \times 10^3 \times 9.81 \times 3$

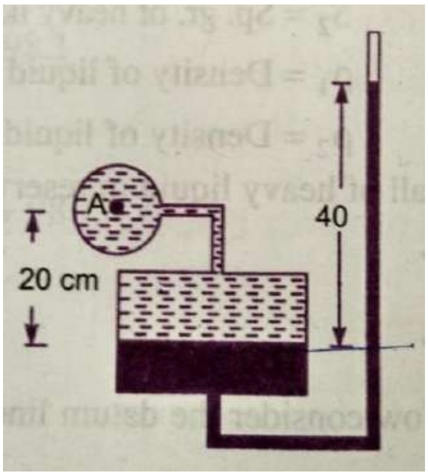
Gauge pressure P = 45028 N/ m^2

Absolute Pressure = Gauge pressure + Atmospheric pressure

= 45028 + 100062

Absolute Pressure = 145090 N/m^2

8. A single column manometer is connected to the pipe containing liquid of sp.gr.0.9. Find the pressure in the pipe if the area of the reservoir is 100 times the area of the tube of manometer. sp.gr. of mercury is 13.6. Height of the liquid from the centre of pipe is 20cm and difference in level of mercury is 40cm. Given,



Sp.gr. of liquid in pipe $S_1 = 0.9$

Density ρ_1 = 900 kg/ m^3

$\frac{\text{Area of reservoir}}{\text{Area of right limb}} = \frac{A}{a} = 100$

Height of the liquid h_1 = 20cm = 0.2m

Rise of mercury in the right limb h_2 = 40cm = 0.4m Pressure in pipe A

$p_A = \frac{A}{a} \times h_2[\rho_2 g - \rho_1 g] + h_2 \rho_2 g - h_1 \rho_1 g$

$= \frac{1}{100} \times 0.4\ [13600 \times 9.81 - 900 \times 9.81] + 0.4 \times 13600 \times 9.81 - 0.2 \times 900 \times 9.81$

$= \frac{0.4}{100} [133416 - 8829] + 53366.4 - 1765.8$

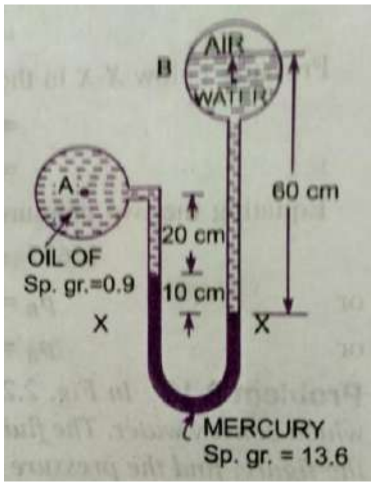
= 533.664 + 53366.4 – 1765.8

= 52134 N/m^2

Pressure in pipe A= 5.21 N/ cm^2

9. A pipe contains an oil of sp.gr.0.9. A differential manometer is connected at the two points A and B shows a difference in mercury level at 15cm. find the difference of pressure at the two points.

Given: Sp.gr. of oil S_1 = 0.9: density ρ_1 = 0.9 x 1000 = 900 kg/ m^3



Difference of level in the mercury h = 15cm = 0.15 m

Sp.gr. of mercury = 13.6, Density = 13.6 × 1000 = 13600 kg/m^3

The difference of pressure $p_A - p_B = g \times h \times (\rho_g - \rho_1)$

= 9.81 x 0.15 (13600 – 900)

$p_A - p_B$ = 18688 N/ m^2

10. A differential manometer is connected at two points A and B .At B air pressure is 9.81 N/cm^2. Find absolute pressure at A.

Density of air = 0.9 × 1000 = 900 kg/m^3 Density of mercury = 13.6 × 10^3 kg/ m^3

Let pressure at A is p_A

Taking datum as X – X

Pressure above X – X in the right limb

= 1000 × 9.81 × 0.6 + p_B = 5886 + 98100 = 103986

Pressure above X – X in the left limb

= 13.6 × 10^3 × 9.81 × 0.1 +0900 × 9.81 × 0.2 + p_A

= 13341.6 +1765.8 +p_A

Equating the two pressures heads

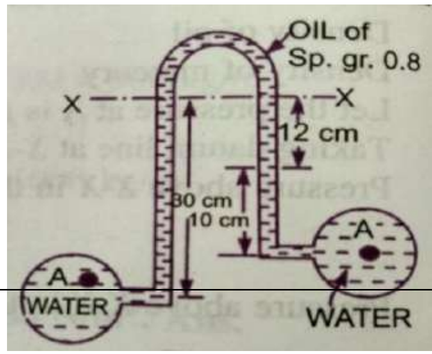
103986 = 13341.6 + 1765.8 + p_A

= 15107.4 + p_A

p_A = 103986 – 15107.4

= 88878.6 N/m^2 **p_A = 8.887 N/cm^2**

11. Water is flowing through two different pipes to which an inverted differential manometer having an oil of sp.gr. 0.8 is connected. The pressure head in the pipe A is 2m of water. Find the pressure in the pipe B for the manometer readings shown in fig. Given: Pressure head at A $= \frac{p_A}{\rho g} = 2$m of water $p_A = \rho \times g \times 2$ = 1000 × 9.81 × 2 = 19620 N/m^2



Pressure below X – X in the left limb

$= p_A - \rho_1 g h_1$

$= 19620 - 1000 \times 9.81 \times 0.3 = 16677$ N/m^2

Pressure below X – X in the right limb

$= p_B - 1000 \times 9.81 \times 0.1 - 800 \times 9.81 \times 0.12$

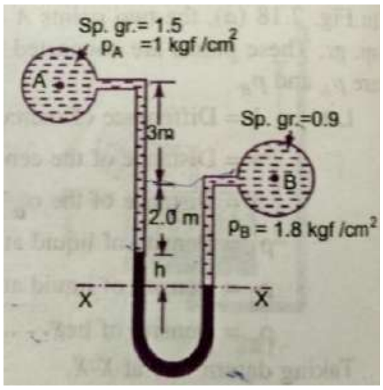
$= p_B - 981 - 941.76 = p_B - 1922.76$

Equating the two pressures, we get,

$16677 = p_B - 1922.76$ $p_B = 16677 + 1922.76$

p_B = 18599.76 N/m^2

12. A different manometer is connected at two points A and B of two pipes. The pipe A contains liquid of sp.gr. = 1.5 while pipe B contains liquid of sp.gr. = 0.9. The pressures at A and B are 1 kgf/cm^2 and 1.80 Kg f/cm^2 respectively. Find the difference in mercury level in the differential manometer.



Sp.gr. of liquid at A $S_1 = 1.5$

Sp.gr. of liquid at B $S_2 = 0.9$

Pressure at A p_A= 1 kgf/c m^2 = $1 \times 10^4 \times$ kg/m^2 = $1 \times 10^4 \times 9.81$N/m^2

Pressure at B p_B = 1.8 kgf/cm^2 = $1.8 \times 10^4 \times 9.81$ N/m^2 [1kgf = 9.81 N]

Density of mercury = 13.6×1000 kg/m^3

Taking X – X as datum line

Pressure above X – X in left limb

$= 13.6 \times 1000 \times 9.81 \times h + 1500 \times 9.81(2+3) + (9.81 \times 10^4)$

Pressure above X – X in the right limb = $900 \times 9.81(h + 2) + 1.8 \times 9.81 \times 10^4$

Equating the two pressures, we get

$13.6 \times 1000 \times 9.81h + 1500 \times 9.81 \times 5 + 9.81 \times 10^4 = 900 \times 9.81(h + 2) + 1.8 \times 9.81 \times 10^4$

Dividing both sides by 1000×9.81

$13.6 h + 7.5 + 10 = 0.9(h+2) + 18$

$(13.6 - 0.9)h = 1.8 + 18 - 17.5 = 19.8 - 17.5 = 2.3$ $h = \frac{2.3}{12.7} = 0.181$m h = 18.1 cm

PASCALS LAW :-

The pressure at a point in a fluid at rest is the same in all directions; the pressure would be the same on all planes passing through a specific point. This fact is also known as Pascal's principle, or Pascal's law.

$$F = PA$$

Let us understand the working principle of Pascal's law with an example.

A pressure of 2000 Pa is transmitted throughout a liquid column due to a force being applied on a piston. If the piston has an area of 0.1 m^2, what force is applied?

This can be calculated using Pascal's Law formula.

F = PA

Here,

P = 2000 Pa = N/m^2

A = 0.1 m^2

Substituting values, we arrive at F = 200 N

Applications of Pascal's Law

- Hydraulic Lift: The image you saw at the beginning of this article is a simple line diagram of a hydraulic lift. This is the principle of the working of hydraulic lift. It works based on the principle of equal pressure transmission throughout a fluid (Pascal's Law).
- The construction is such that a narrow cylinder (in this case A) is connected to a wider cylinder (in this case B). They are fitted with airtight pistons on either end. The inside of the cylinders is filled with fluid that cannot be compressed.
- Pressure applied at piston A is transmitted equally to piston B without diminishing the use of the fluid that cannot be compressed. Thus, piston B effectively serves as a platform to lift heavy objects like big machines or vehicles. A few more applications include a hydraulic jack and hydraulic press, and forced amplification is used in the braking system of most cars.

HYDROSTATIC LAW :-

According to Hydrostatic Law, the rate of increase of pressure in a vertical direction is equal to the weight density of the fluid at that point when the fluid is stationary.

The pressure at any point in a fluid at rest is obtained by the Hydrostatic Law which states that the rate of increase of pressure in a vertically downward direction must be equal to the specific weight of the fluid at that point.

BOUYANCY AND FLOTATION

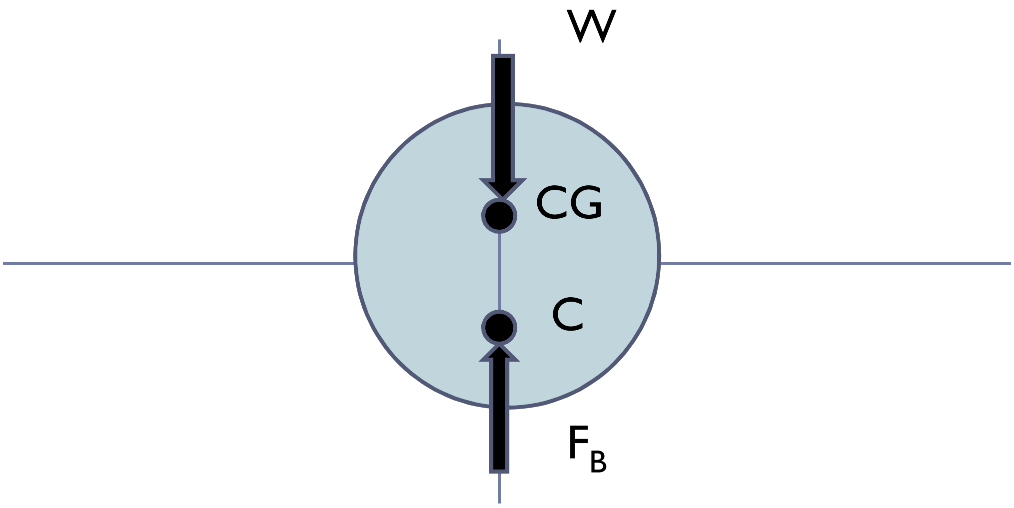
The tendency of immersed body to be lifted up in the fluid due to an upward force opposite to action of gravity is known as buoyancy.

The force tending to lift up the body under such conditions is known as buoyant force or force of buoyancy or up-thrust.

The magnitude of the buoyant force can be determined by **Archimedes' principle** which states

"When a body is immersed in a fluid either wholly or partially, it is buoyed or lifted up by a force which is equal to the weight of fluid

Displaced by the body"



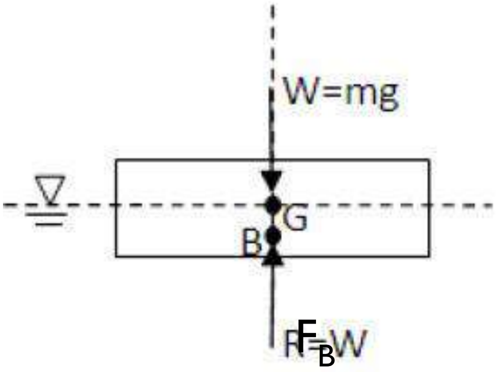
center of buoyancy.

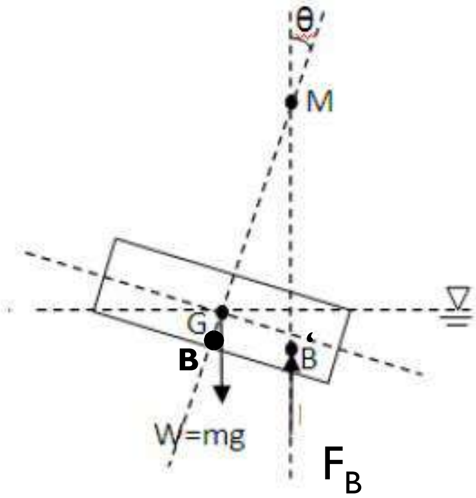
Metacenter and Metacentric Height

Center of Buoyancy (B) The point of application of the force of buoyancy on the body is known as the center of buoyancy.

Metacenter (M): The point about which a body in stable equilibrium start to oscillate when given a small angular displacement is called metacenter.

It may also be defined as point of intersection of the axis of body passing through center of gravity (CG or G) and original center of buoyancy (B) and a vertical line passing through the center of buoyancy (B') of tilted position of body.

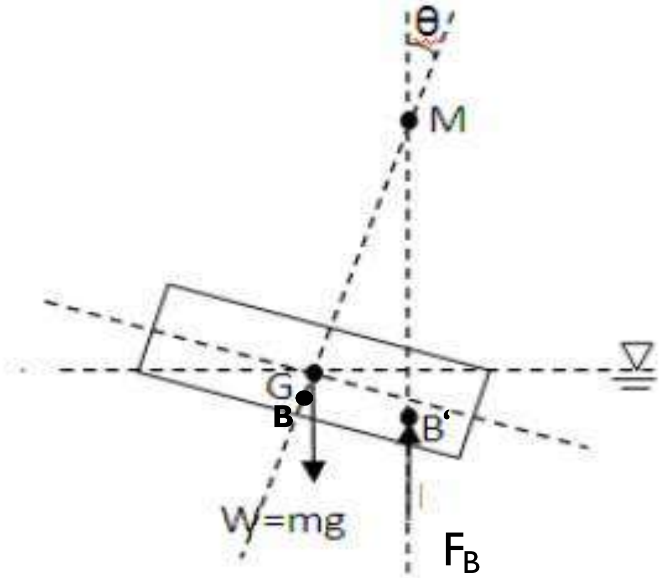




Metacenter and Metacentric Height

Metacentric height (GM): The distance between the center of gravity (G) of floating body and the metacenter (M) is called metacentric height. (i.e., distance GM shown in fig)

$$GM=BM-BG$$



Condition of Stability

For Stable Equilibrium

Position of metacenter (M) is above than center of gravity (G) **For Unstable Equilibrium**

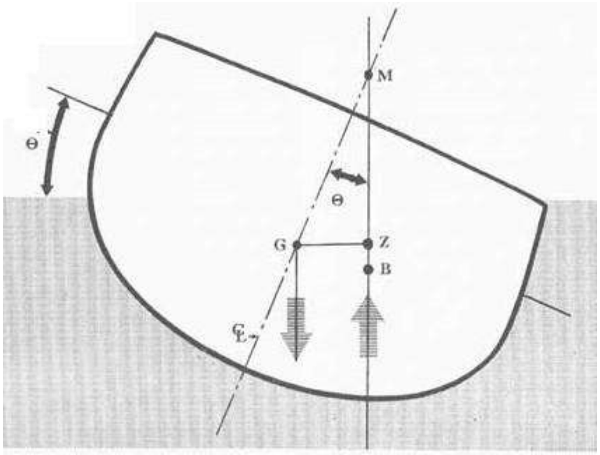
Position of metacenter (M) is below than center of gravity (G)

For Neutral Equilibrium

Position of metacenter (M) coincides center of gravity (G)

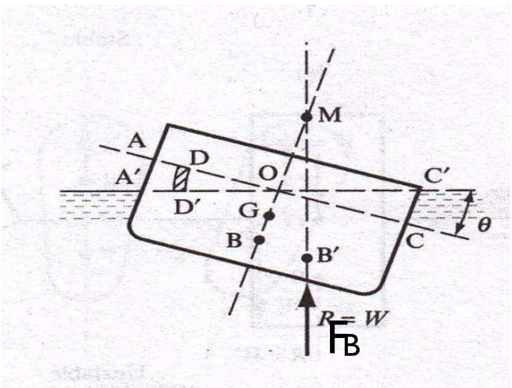
The metacentric height may be determined by the following two methods

1. **Analytical method**
2. Experimental method

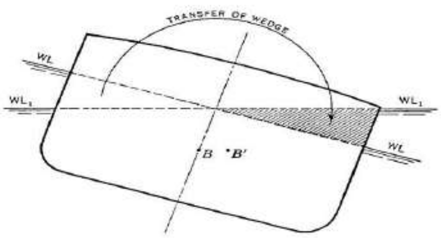


In Figure shown AC is the original waterline plane and B the

center of buoyancy in the equilibrium position.
When the vessel is tilted through small angle θ, the center of buoyancy will move to B' as a result of the



Cross-section

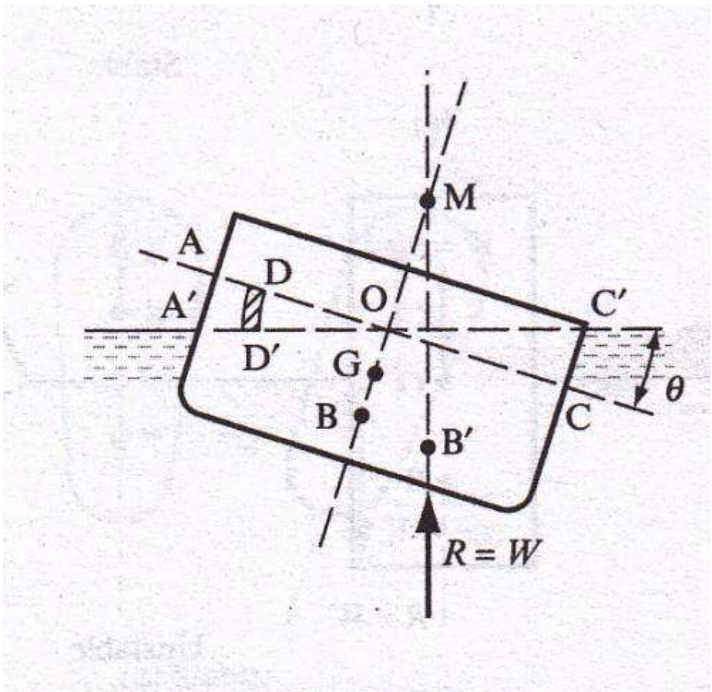


alteration in the shape of displaced fluid.
A'C' is the waterline plane in the displaced position.

To find the metacentric height **GM** ,consider
distance x from O. The height of elementary area is given by xθ. Therefore, volume of the elementary area becomes $dV = (x\theta)dA$
The upward force of buoyancy on this elementary area is then $dF_B = \gamma(x\theta)dA$
Moment of dF_B (moment due to movement of wedge) about O is given by;
$\int x.dF_B = \int x\gamma(x\theta)dA = (\gamma\theta)\int x^2 dA$
$\int x.dF_B = \gamma\theta I$
The change in the moment of the buoyancy Force, FB is
$F_B = F_B BB' = \gamma V(BM\theta)$
For equilibrium, the moment due to movement of wedge=change in moment of buoyancy force
$\gamma\theta I = \gamma V(BM\theta)$

$$BM = \frac{I}{V}$$

$GM = BM - BG$

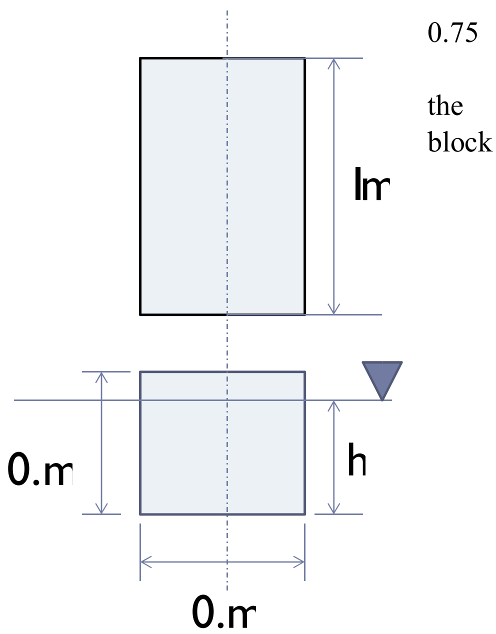


Cross-section

F_B

PROBLEMS:

Q. 1) A wooden block of specific gravity 0.75 floats in water. If the size of the block is 1mx0.5mx0.4m, find its meta centric height



Solution:

Given Data:
Size of wooden block= 1mx0.5mx0.4m,

Specific gravity of wood=0.75

Specific weight of wood=0.75(9.81)=7.36kN/m2 Weight of wooden block=(specific weight)x(volume)

Weight of wooden block=7.36(1x0.5x0.4)=1.47kN Let h is depth of immersion=?

For equilibrium

Weight of water displaced = weight of wooden block

9.81(1x0.5xh)=1.47 >> h=0.3m

Distance of center of buoyancy =OB = 0.3/2 = 0.15m

Distance of center of gravity=OG=0.4/2=0.2m

Now; BG=OG-OM=0.2-0.15=0.05m

Also; BM=I/V

I=moment of inertia of rectangular section

$I=(1)x(0.5)^3/12=0.0104m$

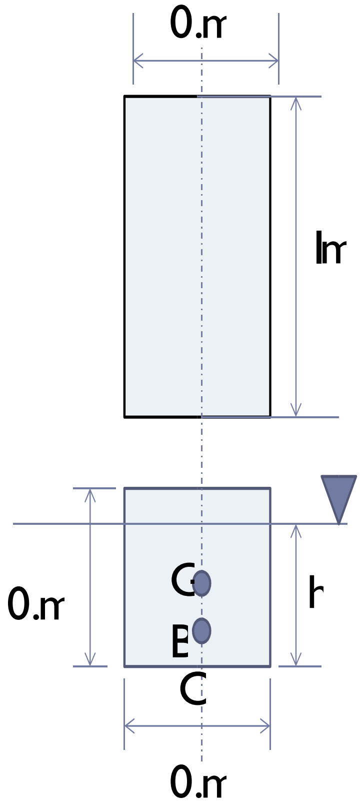
V=volume of water displaced by wooden block

V=(1)x(0.5)x(0.3)=0.15m3

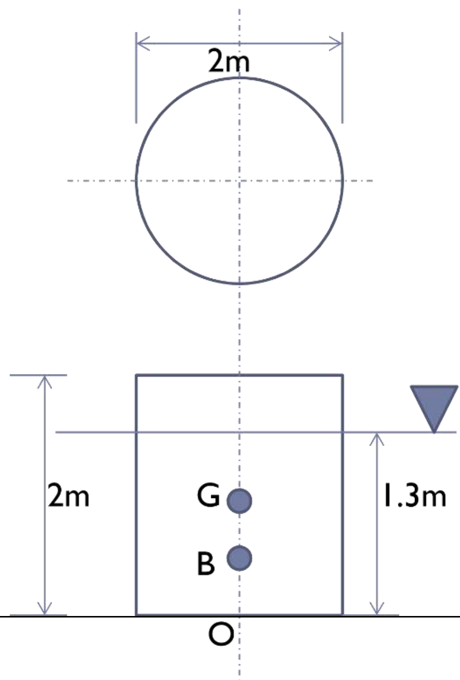
BM=I/V=0.0104/0.15=0.069m

Therefore, meta centric height=GM=BM-BG

GM=0.069-0.05=0.019m



2) Q 2. A solid cylinder 2m in diameter and 2m high is floating in water with its axis vertical. If the specific gravity of the material of cylinder is 0.65, find its meta-centric height. State also whether the equilibrium is stable of unstable. **Solution: Given Data:**



Size of solid cylinder= 2m dia, & 2m height

Specific gravity solid cylinder=0.65 Let h is depth of immersion=?

For equilibrium

Weight of water displaced = weight of wooden block

$9.81(\pi/4(2)^2(h))=9.81(0.65).(\pi/4(2)^2(2))$ h=0.65(2)=1.3m

Center of buoyancy from O=OB=1.3/2=0.65m

Center of gravity from O=OG=2/2=1m

BG=1-0.65=0.35m

Also; BM=I/V

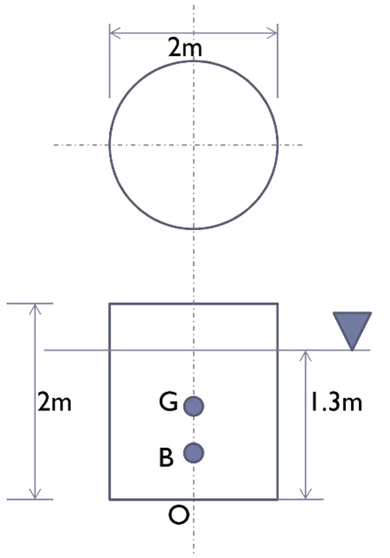
Moment of inertia=$I=(\pi/64)(2)^4=0.785m^4$

Volume displaced=$V=(\pi/4)(2)^4(1.3)=4.084m^3$

BM=I/V=0.192m

GM=BM-BG=0.192-0.35=-0.158m

-ve sign indicate that the metacenter (M) is below the center of gravity (G), therefore, the cylinder is in **unstable equilibrium**



UNIT-II

Fluid Kinematics

Introduction

Fluid Kinematics refers to the description of the motion of fluid particles, considering only their velocities and positions at different times, but without focusing on the forces or energy involved.

Key Concepts:

- **Flow Field**: A region of space where fluid flow is happening, and each point in the field has a velocity vector associated with it.
- **Velocity Field**: Describes the velocity of the fluid at any given point in space and time.
- **Particle Trajectory**: The path traced by a fluid particle as it moves through the flow field.

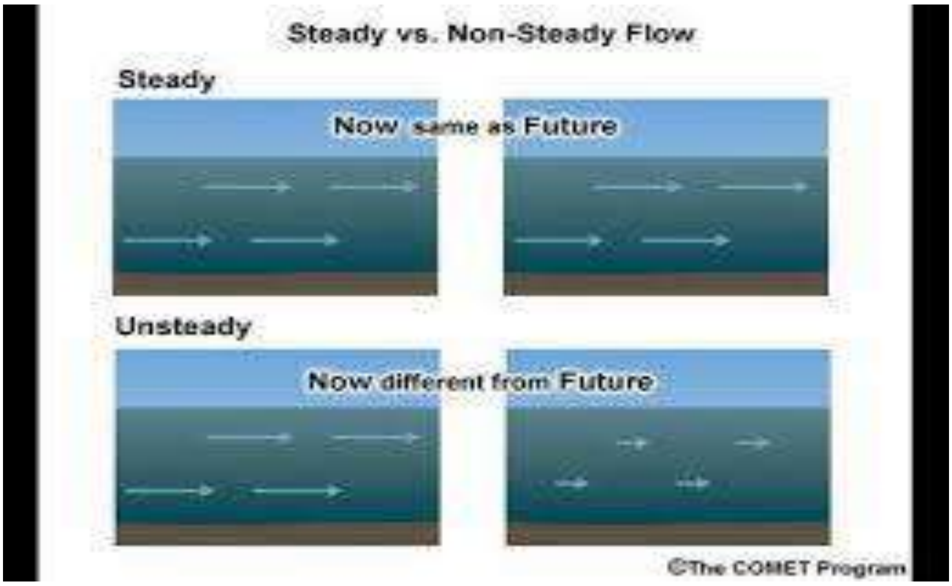
Example:

Imagine a river where water flows downstream. If we focus on the motion of a single water molecule (without considering the forces driving the water), we are analyzing fluid kinematics.

2. Flow Types

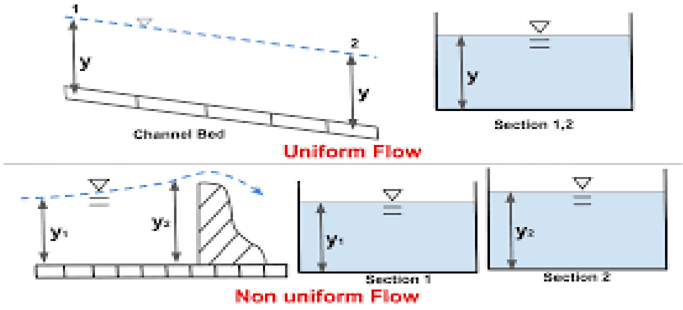
Fluid flow can be categorized based on different characteristics such as time dependency, velocity profile, and behavior.

a. Steady vs. Unsteady Flow



b.

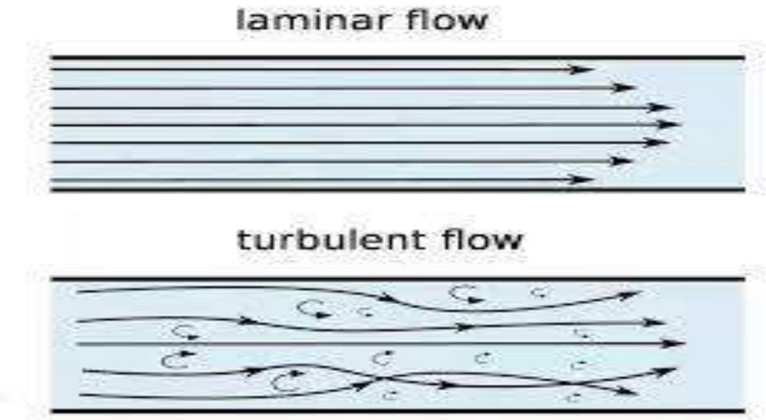
- **Steady Flow**: The velocity of fluid at each point does not change with time.
  - Example: Water flowing at a constant speed through a straight pipe.
- **Unsteady Flow**: The velocity at a given point varies over time.
  - Example: Water in a river where the flow speed changes due to varying environmental conditions.



c. **Uniform vs. Non-uniform Flow**

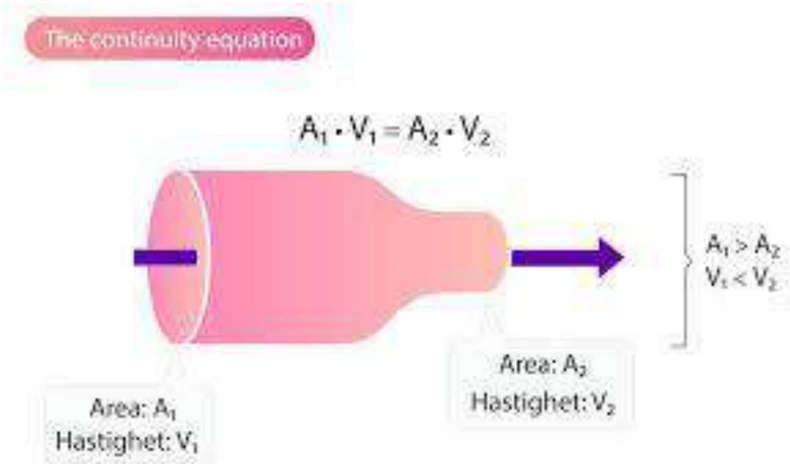
- **Uniform Flow**: The velocity is constant along a streamline at every point in space.
    - Example: Flow of water at a constant speed through a uniform pipe.
- **Non-uniform Flow**: The velocity varies from one point to another.
    - Example: Flow of water in a river, where the speed differs from one place to another.

c. Laminar vs. Turbulent Flow



- **Laminar Flow**: Fluid particles move in smooth, parallel layers without mixing. It occurs at low velocities.
    - Example: Oil flowing through a small pipe at low speed.
- **Turbulent Flow**: The flow is chaotic, with eddies and swirls. This occurs at high velocities.
    - Example: Water flowing rapidly in a storm drain.

3. **Equation of Continuity for One-Dimensional Flow**



4.

The **Equation of Continuity** expresses the conservation of mass for an incompressible fluid. It states that the mass flow rate of fluid remains constant along a streamline.

A1v1=A2v2A_1 v_1 = A_2 v_2A1v1=A2v2

Where:

- A1A_1A1 and A2A_2A2 are the cross-sectional areas at points 1 and 2.
- v1v_1v1 and v2v_2v2 are the velocities at points 1 and 2.

Example:

In a pipe, if the area decreases (e.g., in a narrower section), the velocity must increase to maintain the same flow rate.

4. Circulation and Vorticity

Circulation

Circulation is the total rotation or "spin" of fluid particles around a closed curve. It is defined as the line integral of the velocity field around a closed path:

Γ=∮v·dl\Gamma = \oint \mathbf{v} \cdot d\mathbf{l}Γ=∮v·dl

Where:

- Γ\GammaΓ is the circulation.
- v\mathbf{v}v is the velocity field.
- dld\mathbf{l}dl is the differential length element along the closed path.

Vorticity

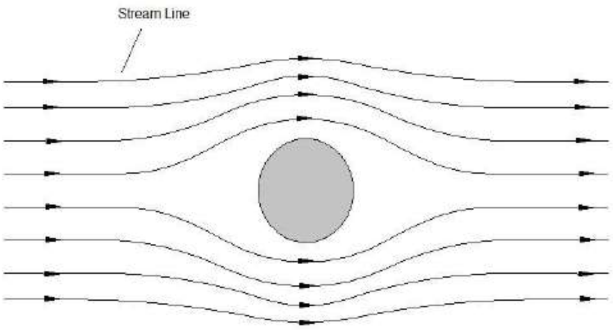
Vorticity is a vector field that measures the local rotation of fluid at a point. It is the curl of the velocity vector:

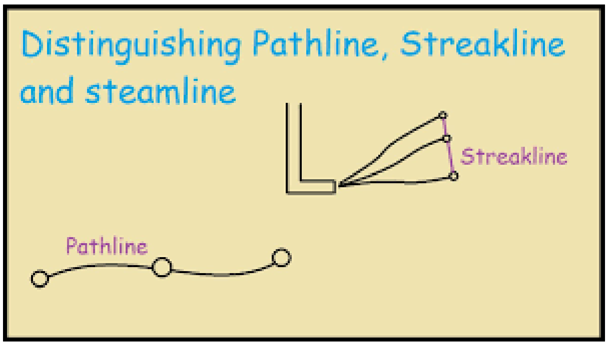
ω=∇×v\mathbf{\omega} = \nabla \times \mathbf{v}ω=∇×v

Example:

- **Circulation**: In a whirlpool, the fluid particles exhibit circulation around the center.
- **Vorticity**: The water near the center of a vortex has a non-zero vorticity.

Streamline, Pathline, and Streakline





a. Streamline

A **streamline** is a line that is tangent to the velocity vector at every point. It represents the direction of flow for a given flow field. In steady flow, streamlines do not change over time.

- **Key Property**: In steady flow, streamlines never cross each other.

b. Pathline

A **pathline** is the actual path followed by a single fluid particle over time.

c. Streakline

A **streakline** is the locus of points that have passed through a particular point at different times. It is formed by all particles that have passed through a specific point in space.

Example:

- **Streamline**: Water flowing steadily in a straight pipe has streamlines that follow the direction of the flow.
- **Pathline**: A specific water molecule's path as it moves from the surface of a river to the sea.
- **Streakline**: The line traced by particles passing through a point where a dye is injected into the flow.

Streamline, Pathline, and Streakline Example:

In a river, streamlines would represent the water flow at each point, pathlines would trace a specific water molecule's journey, and streaklines would show the collection of water particles passing through the same point where dye is released.

6. Stream Tube

A **stream tube** is a three-dimensional volume of fluid enclosed by streamlines. It helps to visualize the flow of fluid within a confined boundary, where fluid particles are bound to stay inside.

- **Key Property**: The flow rate inside a stream tube remains constant, and the velocity along the streamlines within the tube adjusts accordingly to conserve mass.

Example:

In a tapered pipe, the fluid can be imagined as flowing through a stream tube. The streamlines forming the tube adjust to the changes in cross-sectional area.

7. Stream Function and Velocity Potential Function

Stream Function (ψ)

The **stream function** is used for incompressible, two-dimensional flow. It is a scalar function such that the velocity components in the x and y directions can be derived as the partial derivatives of the stream function.

- **Mathematical Expression**: u=∂ψ∂y,v=−∂ψ∂xu = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}u=∂y∂ψ,v=−∂x∂ψ Where uuu and vvv are the velocity components in the x and y directions.

Velocity Potential Function (ϕ)

The **velocity potential function** is a scalar function used in irrotational flow. The velocity components can be derived as the gradient of the potential function.

- **Mathematical Expression**: v=∇ϕ\mathbf{v} = \nabla \phiv=∇ϕ Where v\mathbf{v}v is the velocity vector, and ϕ\phiϕ is the velocity potential.

Differences:

- **Stream Function** applies to incompressible, two-dimensional flow and describes streamlines.
- **Velocity Potential Function** applies to irrotational flow and describes equipotential lines.

Relation:

For irrotational and incompressible flow, there is a connection between the two functions. The stream function represents the flow's pattern, while the potential function describes the flow's velocity.

8. Condition for Irrotational Flow

For a flow to be **irrotational**, the vorticity at every point in the flow must be zero:

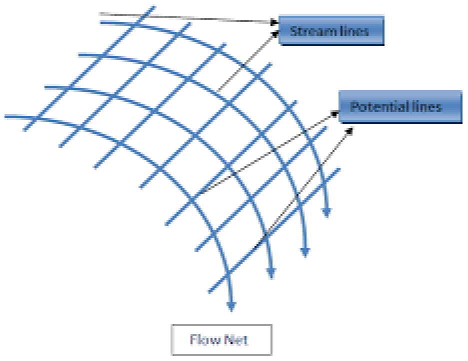
∇×v=0\nabla \times \mathbf{v} = 0∇×v=0

This implies that there is no rotation or swirling of the fluid particles at any point in the flow.

Example:

If water flows smoothly from a tap in a straight line without any swirling, it can be considered irrotational.

9. Flow Net



A **flow net** is a graphical representation of the flow field, showing both streamlines and equipotential lines. The flow net is used in the study of groundwater flow and in fluid flow through porous media.

- **Streamlines**: Represent the paths followed by fluid particles.
- **Equipotential Lines**: Lines where the fluid has the same potential.

Flow nets are essential for visualizing how fluid moves through porous media and analyzing flow in hydraulic structures.

10. Source and Sink, Doublet, and Vortex Flow

a. Source

A **source** is a point in the flow field where fluid is emanating outward in all directions. The velocity increases radially as the distance from the source decreases.

- **Example**: A small hole in a container from which water is emerging.

b. Sink

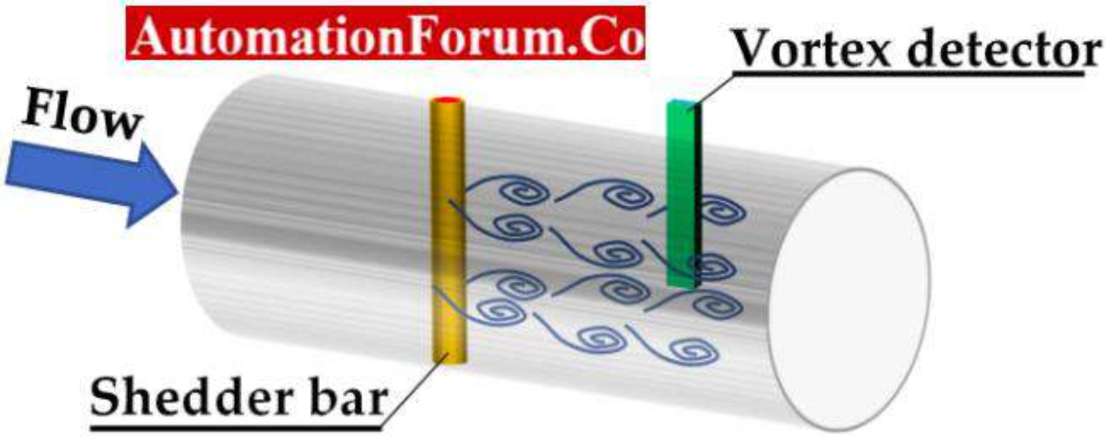
A **sink** is a point where fluid is converging from all directions. The velocity decreases radially as the distance from the sink increases.

- **Example**: A drain where water is being pulled inward.

c. Doublet

A **doublet** is a combination of a source and a sink placed infinitesimally close to each other. It is used to model certain flow fields.

d. Vortex Flow

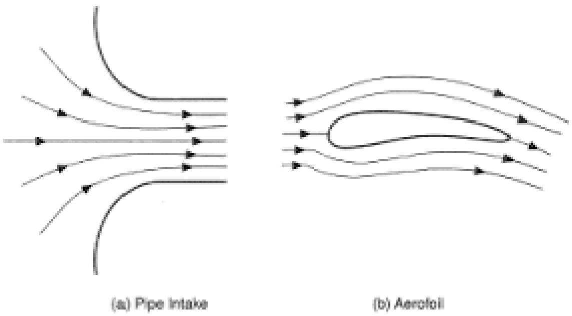


A **vortex flow** is characterized by a circular motion around a central point, with streamlines forming concentric circles.

- **Example**: Water swirling in a bathtub drain.

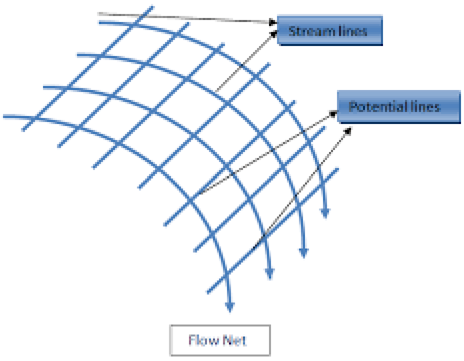
Illustrations

Streamline Example:



- Streamlines represent the flow direction of fluid particles in steady flow.

Flow Net Example:



- Flow nets represent streamlines (curved lines) and equipotential lines (straight lines).

Fluid Dynamics:

Surface and Body Forces

- **Surface Forces**: These are forces that act on the surface of a fluid, such as pressure and shear stress.
    - **Pressure Force**: This force is normal to the surface and results from the fluid's internal pressure.
    - **Shear Force**: This force is parallel to the surface and results from the fluid's viscosity and internal friction.
- **Body Forces**: These are forces that act throughout the entire volume of the fluid, such as gravitational and electromagnetic forces.
    - **Gravitational Force**: Acts vertically downward and is proportional to the fluid's density and mass.
    - **Electromagnetic Forces**: These forces can influence charged particles within a fluid, although they're typically considered less significant in most classical fluid dynamics problems.

2. Euler's and Bernoulli's Equations for Flow Along a Streamline

- **Euler's Equation**: This is a form of the momentum principle for fluid flow. It's used to describe the relationship between velocity, pressure, and external forces in a fluid along a streamline. For steady, incompressible flow, Euler's equation in vector form is:

DvDt+1ρ∇p=g\frac{D\mathbf{v}}{Dt} + \frac{1}{\rho} \nabla p = \mathbf{g}DtDv+ρ1∇p=g

Where:

- v\mathbf{v}v is the velocity vector
- ρ\rhoρ is the density of the fluid
- ppp is the pressure
- g\mathbf{g}g is the body force per unit mass (e.g., gravitational force)
- DDt\frac{D}{Dt}DtD represents the material derivative (rate of change following the flow)

- **Bernoulli's Equation**: A simplified form of Euler's equation for steady, incompressible, non-viscous flow along a streamline. It expresses the conservation of mechanical energy:

p+12ρv2+ρgh=constantp + \frac{1}{2} \rho v^2 + \rho gh = \text{constant}p+21ρv2+ρgh=constant

Where:

- ppp is the pressure
- ρ\rhoρ is the density

- v is the flow velocity
- g is the acceleration due to gravity
- h is the height (potential energy per unit mass)

Bernoulli's equation is applicable to streamline flow where no energy is lost due to viscosity or turbulence.

3. Momentum Equation and Its Applications

- **Momentum Equation**: This equation is derived from the conservation of momentum principle and is used to analyze the forces acting on a fluid flow. It is written as:

$$\sum F = \frac{d}{dt} \int_V \rho \mathbf{v} \, dV$$

Where:

- F represents the sum of external forces (body forces and surface forces)
- $\mathbf{v}$ is the velocity vector
- V is the control volume

Applications:

- **Jet Propulsion**: The momentum equation can be used to calculate the thrust of a jet engine by considering the mass flow rate and velocity of the exhaust gases.
- **Pipe Flow**: It helps determine the force exerted by the fluid on pipe bends, valve mechanisms, and flow obstructions.

4. Force on Pipe Bend

- When a fluid flows through a pipe bend, there is a change in direction of the flow, which leads to a centripetal force. The force exerted by the fluid on the bend can be determined using the momentum equation.
  - For a bend with a curved radius R, and flow rate Q, the force on the pipe bend is given by:

$$F = \rho Q (v_1 - v_2) \cdot \hat{i}$$

Where:

- ρ is the fluid density
- Q is the flow rate
- v_1, v_2 are the velocities before and after the bend
- $\hat{i}$ is the unit vector in the direction of the applied force.

Closed Conduit Flow:

- **Reynolds Experiment**: This is an experiment conducted by Osborne Reynolds that demonstrated the transition from laminar to turbulent flow in a pipe based on the Reynolds number (ReReRe):

Re=ρvDμRe = \frac{\rho v D}{\mu}Re=μρvD

Where:

  - DDD is the diameter of the pipe
  - vvv is the average velocity
  - ρ\rhoρ is the fluid density
  - μ\muμ is the dynamic viscosity

For Re<2000Re < 2000Re<2000, the flow is laminar. For Re>4000Re > 4000Re>4000, the flow is turbulent, and for 2000<Re<40002000 < Re < 40002000<Re<4000, the flow is transitional.

Darcy-Weisbach Equation

- The Darcy-Weisbach equation is used to calculate the pressure drop or head loss due to friction in a pipe:

$$hf = 4flv^2/2gD$$

Where:

  - Δhf is the head loss due to friction
  - f is the Darcy friction factor (depends on ReReRe and pipe roughness)
  - L is the length of the pipe
  - D is the diameter of the pipe
  - v is the velocity of the fluid
  - g is the acceleration due to gravity

Minor Losses in Pipes

- **Minor Losses**: These losses are due to factors such as
    1) bends,
    2) fittings,
    3) valves
    4) sudden expansions
    5) contractions in the pipe.

8. Pipes in Series and Pipes in Parallel

- **Pipes in Series**: When multiple pipes are connected end-to-end, the total head loss is the sum of individual head losses across each pipe:

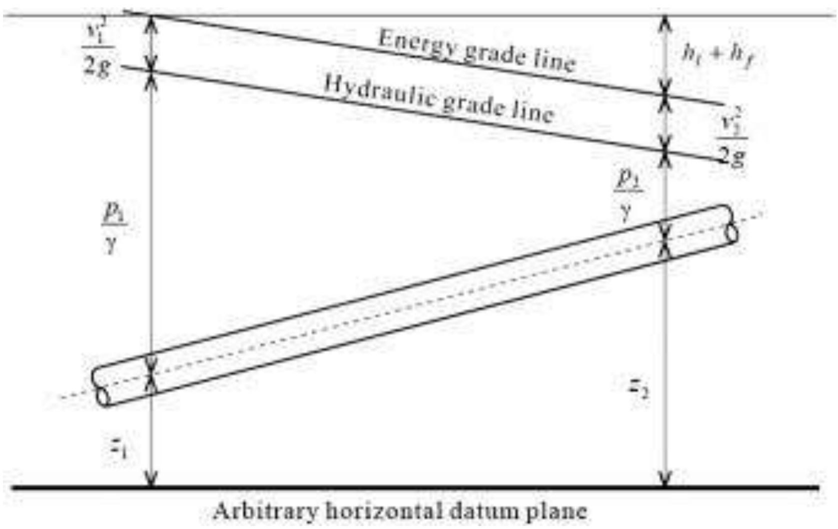
Δhtotal=∑Δhi$\Delta h_{\text{total}} = \sum \Delta h_i$Δhtotal=∑Δhi

Where ΔhiΔh_iΔhi is the head loss in each individual pipe.

- **Pipes in Parallel**: For pipes in parallel, the total flow rate is the sum of the flow rates in the individual pipes, but the head loss is the same for all pipes.

Qtotal=∑Qi$Q_{\text{total}} = \sum Q_i$Qtotal=∑Qi

9. Total Energy Line (TEL) and Hydraulic Gradient Line (HGL)



- **Total Energy Line (TEL)**: The total energy line represents the sum of the pressure head, velocity head, and elevation head at any point along a pipe.

E=h+v22g+z$E = h + \frac{v^2}{2g} + z$E=h+2gv2+z

Where:

  - hhh is the pressure head
  - v22g$\frac{v^2}{2g}$2gv2 is the velocity head
  - zzz is the elevation head

- **Hydraulic Gradient Line (HGL)**: The HGL represents the elevation of the fluid's pressure at any point along the pipe. It's a curve that shows how the pressure head varies along the pipe.

Note: The HGL is always below the TEL by an amount equal to the velocity head.

Problems

Sure! Let's go through some problems related to the **continuity equation**, **velocity potential**, and **stream potential lines**.

Problems on Continuity Equation

Problem: A pipe carrying water has a cross-sectional area A1=0.05 m2A_1 = 0.05 \, \text{m}^2 at point 1 and A2=0.02 m2A_2 = 0.02 \, \text{m}^2 at point 2. If the velocity at point 1 is v1=3 m/sv_1 = 3 \, \text{m/s}, calculate the velocity at point 2, v2v_2, assuming steady, incompressible flow.

Solution: We can use the **continuity equation**, which states that for an incompressible fluid, the mass flow rate must remain constant along the pipe. The equation is:

A1v1=A2v2A_1 v_1 = A_2 v_2

Where:

- A1A_1 and A2A_2 are the cross-sectional areas at points 1 and 2, respectively.
- v1v_1 and v2v_2 are the velocities at points 1 and 2, respectively.

Substituting the given values:

0.05 m2×3 m/s=0.02 m2×v20.05 \, \text{m}^2 \times 3 \, \text{m/s} = 0.02 \, \text{m}^2 \times v_2

Solving for v2v_2:

v2=0.05×30.02=0.150.02=7.5 m/sv_2 = \frac{0.05 \times 3}{0.02} = \frac{0.15}{0.02} = 7.5 \, \text{m/s}

Thus, the velocity at point 2 is **7.5 m/s**.

Problem 1: Water is flowing through a tapered pipe. At point 1, the diameter is D1=0.4 mD_1 = 0.4 \, \text{m}D1=0.4m and the flow velocity is v1=5 m/sv_1 = 5 \, \text{m/s}v1=5m/s. At point 2, the diameter is D2=0.2 mD_2 = 0.2 \, \text{m}D2=0.2m. Find the velocity at point 2, v2v_2v2, assuming steady, incompressible flow.

Solution: Use the **continuity equation**:

A1v1=A2v2A_1 v_1 = A_2 v_2A1v1=A2v2

Where:

- A1=πD124A_1 = \frac{\pi D_1^2}{4}A1=4πD12 and A2=πD224A_2 = \frac{\pi D_2^2}{4}A2 =4πD22 are the cross-sectional areas at points 1 and 2.
- v1=5 m/sv_1 = 5 \, \text{m/s}v1=5m/s and D1=0.4 mD_1 = 0.4 \, \text{m}D1=0.4m, D2=0.2 mD_2 = 0.2 \, \text{m}D2=0.2m.

First, calculate the areas:

A1=π(0.4)24=0.1256 m2A_1 = \frac{\pi (0.4)^2}{4} = 0.1256 \, \text{m}^2A1=4π(0.4)2=0.1256m2

A2=π(0.2)24=0.0314 m2A_2 = \frac{\pi (0.2)^2}{4} = 0.0314 \, \text{m}^2A2=4π(0.2)2=0.0314m2

Now use the continuity equation:

0.1256×5=0.0314×v20.1256 \times 5 = 0.0314 \times v_20.1256×5=0.0314×v2

Solving for v2v_2v2:

v2=0.1256×50.0314=20 m/sv_2 = \frac{0.1256 \times 5}{0.0314} = 20 \, \text{m/s}v2 =0.03140.1256×5=20m/s

Thus, the velocity at point 2 is **20 m/s**.

2. Problems on Velocity Potential

Problem: The velocity potential ϕ\phi for a two-dimensional flow is given by the equation:

ϕ(x,y)=3x+2y\phi(x, y) = 3x + 2y

Find the velocity components uu and vv in the xx- and yy-directions.

Solution: The velocity components can be found by taking the spatial derivatives of the velocity potential:

u=∂ϕ∂x,v=∂ϕ∂yu = \frac{\partial \phi}{\partial x}, \quad v = \frac{\partial \phi}{\partial y}

Given:

ϕ(x,y)=3x+2y\phi(x, y) = 3x + 2y

- In the xx-direction:

u=∂ϕ∂x=∂∂x(3x+2y)=3u = \frac{\partial \phi}{\partial x} = \frac{\partial}{\partial x} (3x + 2y) = 3

- In the yy-direction:

v=∂ϕ∂y=∂∂y(3x+2y)=2v = \frac{\partial \phi}{\partial y} = \frac{\partial}{\partial y} (3x + 2y) = 2

Thus, the velocity components are:

- u=3 m/su = 3 \, \text{m/s}
- v=2 m/sv = 2 \, \text{m/s}

Problem 2: The velocity potential ϕ\phiϕ for a two-dimensional flow is given by:

ϕ(x,y)=2x+3y\phi(x, y) = 2x + 3yϕ(x,y)=2x+3y

Find the velocity components uuu and vvv in the xxx- and yyy-directions.

Solution: The velocity components are given by the partial derivatives of the velocity potential:

u=∂ϕ∂x,v=∂ϕ∂yu = \frac{\partial \phi}{\partial x}, \quad v = \frac{\partial \phi}{\partial y}u=∂x∂ϕ ,v=∂y∂ϕ

Given:

ϕ(x,y)=2x+3y\phi(x, y) = 2x + 3yϕ(x,y)=2x+3y

- In the xxx-direction:

u=∂ϕ∂x=∂∂x(2x+3y)=2u = \frac{\partial \phi}{\partial x} = \frac{\partial}{\partial x} (2x + 3y) = 2u=∂x∂ϕ=∂x∂(2x+3y)=2

- In the yyy-direction:

v=∂ϕ∂y=∂∂y(2x+3y)=3v = \frac{\partial \phi}{\partial y} = \frac{\partial}{\partial y} (2x + 3y) = 3v=∂y∂ϕ=∂y∂(2x+3y)=3

Thus, the velocity components are:

- u=2 m/su = 2 \, \text{m/s}u=2m/s
- v=3 m/sv = 3 \, \text{m/s}v=3m/s

3. Problem on Stream Function and Streamlines

Problem: For a steady, incompressible, two-dimensional flow, the stream function ψ\psi is given by:

ψ(x,y)=x2−y2\psi(x, y) = x^2 - y^2

(a) Find the streamlines of the flow.

(b) If the velocity potential is given by ϕ=x2+y2\phi = x^2 + y^2, find the velocity components.

Solution:

Part (a): Streamlines:

Streamlines are given by constant values of the stream function ψ(x,y)\psi(x, y). Thus, the equation for the streamlines is:

ψ(x,y)=C\psi(x, y) = C

Where CC is a constant. From the given stream function:

x2−y2=Cx^2 - y^2 = C

Thus, the streamlines are defined by the equation:

x2−y2=Cx^2 - y^2 = C

This represents a family of hyperbolas.

Part (b): Velocity Components:

The velocity components in the xx- and yy-directions can be found from the stream function as:

u=∂ψ∂y,v=−∂ψ∂xu = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}

- In the xx-direction:

u=∂ψ∂y=∂∂y(x2−y2)=−2yu = \frac{\partial \psi}{\partial y} = \frac{\partial}{\partial y} (x^2 - y^2) = -2y

- In the yy-direction:

v=−∂ψ∂x=−∂∂x(x2−y2)=−2xv = -\frac{\partial \psi}{\partial x} = -\frac{\partial}{\partial x} (x^2 - y^2) = -2x

Thus, the velocity components are:

- u=−2y m/su = -2y \, \text{m/s}
- v=−2x m/sv = -2x \, \text{m/s}

---

4. Problem on Relation Between Stream Function and Velocity Potential

Problem: In a two-dimensional incompressible flow, the stream function and velocity potential are related. Given the stream function:

ψ(x,y)=4xy\psi(x, y) = 4xy

and the velocity potential:

ϕ(x,y)=x2−y2\phi(x, y) = x^2 - y^2

(a) Check if the flow is irrotational.

(b) Find the velocity components.

Solution:

Part (a): Checking if the Flow is Irrotational:

A flow is irrotational if the vorticity ω\omega is zero. The vorticity is given by:

ω=∂v∂x−∂u∂y\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}

Where uu and vv are the velocity components. First, find uu and vv from the stream function and velocity potential.

- From the stream function ψ(x,y)=4xy\psi(x, y) = 4xy:

u=∂ψ∂y=4x,v=−∂ψ∂x=−4yu = \frac{\partial \psi}{\partial y} = 4x, \quad v = -\frac{\partial \psi}{\partial x} = -4y

Now calculate the vorticity:

ω=∂(−4y)∂x−∂(4x)∂y=0−0=0\omega = \frac{\partial (-4y)}{\partial x} - \frac{\partial (4x)}{\partial y} = 0 - 0 = 0

Since the vorticity is zero, the flow is **irrotational**.

Part (b): Velocity Components:

From the given stream function ψ(x,y)=4xy\psi(x, y) = 4xy, we have already determined that the velocity components are:

- u=4xu = 4x
- v=−4yv = -4y

Thus, the velocity components are:

- u=4x m/su = 4x \, \text{m/s}
- v=−4y m/sv = -4y \, \text{m/s}

Problem 3: The stream function for a two-dimensional incompressible flow is given by:

ψ(x,y)=x2−y2\psi(x, y) = x^2 - y^2ψ(x,y)=x2−y2

(a) Find the streamlines of the flow.

(b) If the velocity potential is given by ϕ(x,y)=2x+y\phi(x, y) = 2x + yϕ(x,y)=2x+y, find the velocity components.

Solution:

Part (a): Streamlines

Streamlines are given by constant values of the stream function:

ψ(x,y)=C\psi(x, y) = Cψ(x,y)=C

Substitute the given stream function:

x2−y2=Cx^2 - y^2 = Cx2−y2=C

Thus, the streamlines are represented by the equation:

x2−y2=Cx^2 - y^2 = Cx2−y2=C

This is a family of hyperbolas.

Part (b): Velocity Components

To find the velocity components, we use the relationships:

u=∂ψ∂y,v=−∂ψ∂xu = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}u=∂y∂ψ ,v=−∂x∂ψ

- In the xxx-direction:

u=∂ψ∂y=∂∂y(x2−y2)=−2yu = \frac{\partial \psi}{\partial y} = \frac{\partial}{\partial y} (x^2 - y^2) = -2yu=∂y∂ψ=∂y∂(x2−y2)=−2y

- In the yyy-direction:

v=−∂ψ∂x=−∂∂x(x2−y2)=−2xv = -\frac{\partial \psi}{\partial x} = -\frac{\partial}{\partial x} (x^2 - y^2) = -2xv=−∂x∂ψ=−∂x∂(x2−y2)=−2x

Thus, the velocity components are:

- u=−2y m/su = -2y \, \text{m/s}u=−2ym/s
- v=−2x m/sv = -2x \, \text{m/s}v=−2xm/s

4. Problem on Stream Function and Velocity Potential

Problem 4: The velocity potential ϕ\phiϕ and the stream function ψ\psiψ for a flow are given by:

ϕ(x,y)=x2+y2\phi(x, y) = x^2 + y^2ϕ(x,y)=x2+y2 ψ(x,y)=2xy\psi(x, y) = 2xyψ(x,y)=2xy

(a) Check if the flow is incompressible.

(b) Find the velocity components uuu and vvv.

Solution:

Part (a): Checking if the Flow is Incompressible

A flow is incompressible if the **divergence of velocity** is zero, i.e.,

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

First, we find the velocity components from the velocity potential ϕ(x,y)\phi(x, y)ϕ(x,y) and stream function ψ(x,y)\psi(x, y)ψ(x,y):

- From the velocity potential ϕ(x,y)\phi(x, y)ϕ(x,y):

u=∂ϕ∂x=2x,v=∂ϕ∂y=2yu = \frac{\partial \phi}{\partial x} = 2x, \quad v = \frac{\partial \phi}{\partial y} = 2yu=∂x∂ϕ=2x,v=∂y∂ϕ=2y

- From the stream function ψ(x,y)\psi(x, y)ψ(x,y):

u=∂ψ∂y=2x,v=−∂ψ∂x=−2yu = \frac{\partial \psi}{\partial y} = 2x, \quad v = -\frac{\partial \psi}{\partial x} = -2yu=∂y∂ψ=2x,v=−∂x∂ψ=−2y

Now, check the divergence:

∂u∂x+∂v∂y=∂∂x(2x)+∂∂y(−2y)=2−2=0\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = \frac{\partial}{\partial x} (2x) + \frac{\partial}{\partial y} (-2y) = 2 - 2 = 0∂x∂u+∂y∂v=∂x∂(2x)+∂y∂(−2y)=2−2=0

Since the divergence is zero, the flow is **incompressible**.

Part (b): Velocity Components

From the given stream function and velocity potential:

- u=2x m/su = 2x \, \text{m/s}u=2xm/s
- v=2y m/sv = 2y \, \text{m/s}v=2ym/s

Thus, the velocity components are:

- u=2x m/su = 2x \, \text{m/s}u=2xm/s
- v=2y m/sv = 2y \, \text{m/s}v=2ym/s

Problem 5: A fluid is flowing through a pipe. The stream function is given by ψ(x,y)=3x−2y\psi(x, y) = 3x - 2yψ(x,y)=3x−2y. The velocity at point (x, y) is given by v=(u,v)\mathbf{v} = (u, v)v=(u,v), where uuu and vvv are the velocity components in the xxx- and yyy-directions.

(a) Find the velocity components uuu and vvv.

(b) Using the **continuity equation**, calculate the relationship between the velocity components for incompressible flow.

Solution:

Part (a): Velocity Components

To find the velocity components, we use the relationships:

u=∂ψ∂y,v=−∂ψ∂xu = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}u=∂y∂ψ,v=−∂x∂ψ

Given the stream function:

ψ(x,y)=3x−2y\psi(x, y) = 3x - 2yψ(x,y)=3x−2y

- In the xxx-direction:

u=∂ψ∂y=∂∂y(3x−2y)=−2u = \frac{\partial \psi}{\partial y} = \frac{\partial}{\partial y} (3x - 2y) = -2u=∂y∂ψ=∂y∂(3x−2y)=−2

- In the yyy-direction:

v=−∂ψ∂x=−∂∂x(3x−2y)=−3v = -\frac{\partial \psi}{\partial x} = -\frac{\partial}{\partial x} (3x - 2y) = -3v=−∂x∂ψ=−∂x∂(3x−2y)=−3

Thus, the velocity components are:

- u=−2 m/su = -2 \, \text{m/s}u=−2m/s
- v=−3 m/sv = -3 \, \text{m/s}v=−3m/s

Part (b): Continuity Equation

For incompressible flow, the **continuity equation** is:

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

Substitute the velocity components:

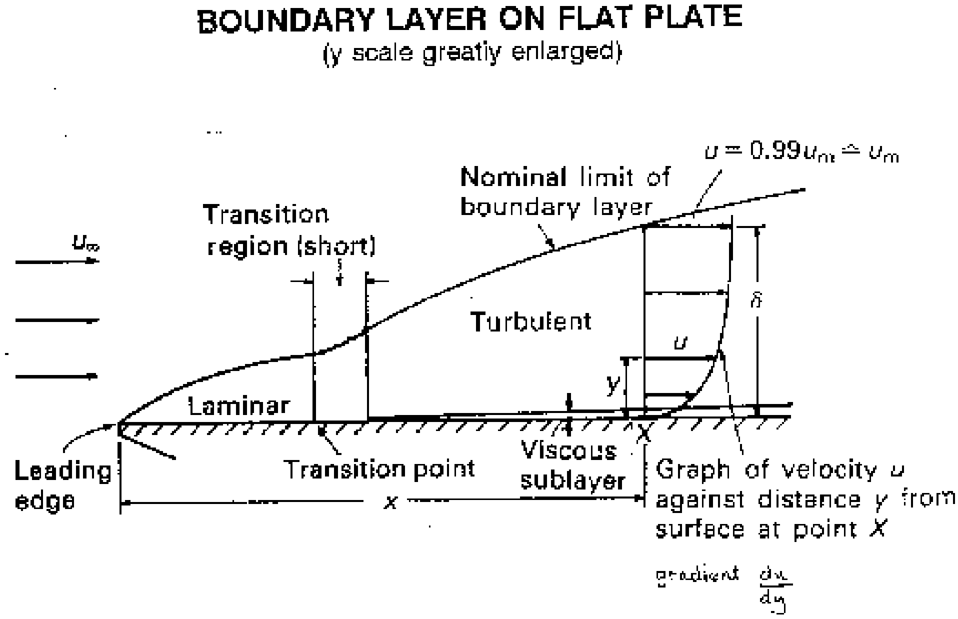
∂(−2)∂x+∂(−3)∂y=0\frac{\partial (-2)}{\partial x} + \frac{\partial (-3)}{\partial y} = 0∂x∂(−2)+∂y∂(−3)=0

Since both partial derivatives are zero, the continuity equation holds true, and the flow is incompressible.

!

UNIT-III

Boundary Layer Theory



Introduction

- The boundary layer is a thin region of fluid flow near a solid surface where viscous effects are significant.
- Introduced by Ludwig Prandtl in 1904 to simplify the Navier-Stokes equations for high Reynolds number flows.
- Outside the boundary layer, fluid can be assumed to be inviscid, simplifying fluid dynamics analysis.

Momentum Integral Equation

- Derived from the Navier-Stokes equations by integrating over the boundary layer thickness.
- Expressed as: where:
  - is the free-stream velocity,
  - is velocity within the boundary layer,
  - is the boundary layer thickness,
  - is the wall shear stress.
- Provides an approximate method to estimate boundary layer growth.

Displacement, Momentum, and Energy Thickness

Detailed Notes on Displacement, Momentum, and Energy Thickness in Fluid Mechanics

In fluid mechanics, when a fluid flows over a solid surface, a thin region known as the boundary layer forms due to viscosity. The velocity of the fluid changes from zero at the surface (due to the no-slip condition) to the free-stream velocity. Various thickness parameters are used to quantify different effects within this boundary layer.

Types of Thicknesses in Boundary Layers:

1. Displacement Thickness ()
2. Momentum Thickness ()
3. Energy Thickness ()

Each of these thicknesses provides insights into the flow characteristics such as mass flow displacement, momentum deficit, and energy loss.

1)Displacement Thickness ():

Definition: Displacement thickness represents the height by which the solid surface would have to be displaced in an inviscid flow to account for the reduction in mass flow caused by the presence of the boundary layer.

Mathematical Expression:

where:

- = local velocity at distance from the wall
- = free-stream velocity
- = differential thickness of the boundary layer

Significance:

- Represents the effective reduction in flow rate due to the boundary layer.
- Important in aerodynamics for determining lift and drag on surfaces.
- Used in calculating boundary layer effects in compressible flows.

2)Momentum Thickness ():

Definition: Momentum thickness quantifies the loss of momentum due to the presence of the boundary layer. It represents the thickness of an equivalent layer of inviscid fluid that has the same momentum flux as the deficit caused by the boundary layer.

Mathematical Expression:

Significance:

- Important in predicting drag on bodies.
- Used in integral boundary layer equations to estimate shear stress.
- Helps in analyzing turbulent boundary layers and skin friction.

3)Energy Thickness ():

Definition: Energy thickness represents the energy deficit within the boundary layer compared to the free-stream velocity profile. It measures how much the kinetic energy flux is reduced due to the boundary layer effects.

Mathematical Expression:

Significance:

- Useful in determining the heat transfer characteristics of the boundary layer.
- Essential in analyzing the energy balance in high-speed aerodynamic flows.
- Important in calculating losses in turbomachinery and flow separation effects.

5. Comparison of Thicknesses

Thickness Type	Represents	Importance
Displacement Thickness ()	Mass flux deficit	Flow rate and lift effects
Momentum Thickness ()	Momentum loss	Drag and shear stress analysis
Energy Thickness ()	Kinetic energy deficit	Energy balance and heat transfer

- **Displacement Thickness ()**:
  - Represents the distance by which the solid boundary would need to be displaced to maintain the same mass flow rate as in an inviscid flow.
  -
- **Momentum Thickness ()**:
  - Measures the loss of momentum due to the presence of the boundary layer.
- **Energy Thickness ()**:
  - Represents the energy deficit within the boundary layer.
  -

Separation of Boundary Layer

- Occurs when the flow reverses direction due to an adverse pressure gradient.
- Key indicators:
  - Vanishing wall shear stress ().
  - Flow reversal in the boundary layer.

Control of Flow Separation

- **Suction and Blowing**: Removes low-energy fluid or injects high-energy fluid.
- **Streamlining the body**: Reducing abrupt changes in geometry.
- **Vortex Generators**: Enhance mixing within the boundary layer.

- **Boundary Layer Suction**: Used in aircraft to delay separation.
- **Surface Roughness and Riblets**: Modify turbulence to control separation.

Streamlined Body and Bluff Body

- **Streamlined Body**:
    - A shape designed to reduce drag by minimizing flow separation.
    - Example: Aircraft wings, streamlined cars.
- **Bluff Body**:
    - A body that experiences large pressure drag due to flow separation.
    - Example: Buildings, vehicles like buses and trucks.

Basic Concepts of Velocity Profiles

- **Laminar Boundary Layer**: Parabolic profile.
- **Turbulent Boundary Layer**: Fuller velocity profile due to increased mixing.
- **Velocity Distribution**: Typically follows a power-law or logarithmic relation.

Dimensional Analysis

Dimensional analysis is a powerful tool used in physics and engineering to analyze the relationships between different physical quantities by studying their dimensions. In this context, "dimension" refers to the fundamental units that make up a physical quantity. Here’s an in-depth look at the dimensions and units involved in dimensional analysis:

1. Basic Dimensions

The fundamental physical quantities, called *base quantities*, are typically represented in terms of the following dimensions:

- **Length (L)**: Represents distance or size, measured in meters (m) in SI units.
- **Mass (M)**: Represents the quantity of matter in an object, measured in kilograms (kg) in SI units.
- **Time (T)**: Represents duration, measured in seconds (s) in SI units.
- **Electric Current (I)**: Represents the flow of electric charge, measured in amperes (A) in SI units.
- **Temperature (θ)**: Represents the degree of hotness or coldness, measured in kelvin (K) in SI units.
- **Amount of Substance (N)**: Represents the quantity of particles in a substance, measured in moles (mol) in SI units.
- **Luminous Intensity (J)**: Represents the perceived power of light, measured in candela (cd) in SI units.

2. Derived Quantities

Derived quantities are combinations of the fundamental dimensions. For example:

- **Speed (v)**: Has the dimensions of length per time (L/T). Its unit is meters per second (m/s).
- **Force (F)**: Has the dimensions of mass times length per time squared ($M * L/T^2$). Its unit is the newton (N), where 1N=1 kg·m/s21 N = 1 \, \text{kg} \cdot \text{m/s}^21N=1kg·m/s2.
- **Energy (E)**: Has the dimensions of mass times length squared per time squared ($M * L^2/T^2$). Its unit is the joule (J), where 1J=1 kg·m2/s21 J = 1 \, \text{kg} \cdot \text{m}^2/\text{s}^21J=1kg·m2/s2.
- **Pressure (P)**: Has the dimensions of force per area ($M * L^{-1} * T^{-2}$). Its unit is the pascal (Pa), where 1 Pa=1 N/m2=1 kg/m·s21 \, \text{Pa} = 1 \, \text{N/m}^2 = 1 \, \text{kg}/\text{m} \cdot \text{s}^21Pa=1N/m2=1kg/m·s2.

3. Dimensional Formula

The dimensional formula represents a physical quantity in terms of the basic dimensions. For example:

- **Speed**: $[L\ T^{-1}]$
- **Force**: $[M\ L\ T^{-2}]$
- **Energy**: $[M\ L^2\ T^{-2}]$

These formulas allow us to express complex physical laws in terms of simpler dimensions.

4. Units

Units are the standard quantities used to measure physical quantities. For example, the unit of length in the SI system is the meter (m), and the unit of mass is the kilogram (kg). Units can also be derived from the base units:

- **Speed**: m/s (meters per second)
- **Acceleration**: m/s^2 (meters per second squared)
- **Force**: N (newton), which is equivalent to $kg \cdot m/s^2$
- **Energy**: J (joule), which is equivalent to $kg \cdot m^2/s^2$

5. Dimensional Consistency and Conversion

One key aspect of dimensional analysis is ensuring that equations are dimensionally consistent, meaning that both sides of an equation must have the same dimensions. This helps check the validity of formulas and ensures proper unit conversions.

Example: In the equation for Newton's second law of motion, F=maF = maF=ma, both sides of the equation must have the same dimensions:

- The left-hand side (Force, F) has the dimensions $[M\ L\ T^{-2}]$.
- The right-hand side (mass, m times acceleration, a) has the dimensions $[M] * [L\ T^{-2}]$, which simplifies to $[M\ L\ T^{-2}]$.

Since the dimensions match, the equation is dimensionally consistent.

6. Dimensional Analysis in Problem Solving

Dimensional analysis is often used for:

- **Checking units**: Ensuring equations are correct by checking the dimensional consistency.
- **Converting units**: Converting between different units of measurement (e.g., from inches to centimeters or pounds to kilograms).
- **Deriving relationships**: Deriving formulas or understanding the relationship between different physical quantities.

7. Scaling and Similarity

Dimensional analysis is crucial in fluid dynamics, heat transfer, and other fields for scaling models and understanding similarity between different systems, like when studying the behavior of a model and scaling it up to real-world conditions.

Summary

Dimensional analysis relies on:

- Base dimensions (L, M, T, etc.) and derived quantities formed by combinations of them.
- The importance of dimensional formulas in representing physical quantities.
- The necessity for dimensional consistency in equations.
- The application of dimensional analysis in unit conversions, problem-solving, and scaling.

- Dimensions are the fundamental physical quantities (e.g., Length , Mass , Time , etc.).
- Units are standardized measurements (SI units: meters, kilograms, seconds, etc.).

Dimensional Homogeneity

The Principle of Dimensional Homogeneity

- In any valid physical equation, the dimensions of each term must be consistent. If an equation contains terms with different dimensions, it is considered dimensionally inconsistent and is likely invalid.
- Example: If we have an equation like F=maF = maF=ma (Force equals mass times acceleration), the dimensional analysis would show: [F]=[m][a]=[M][L/T2]=[ML/T2][F] = [m][a] = [M][L/T^2] = [M L/T^2][F]=[m][a]=[M][L/T2]=[ML/T2] Since both sides of the equation have the same dimensions of ML/T2M L/T^2ML/T2, the equation is dimensionally homogeneous.

- All terms in an equation must have the same dimensions.
- Helps in checking the correctness of equations and deriving new relationships.

Non-dimensionalization of Equations

- Converts equations into dimensionless form to identify dominant parameters.

- Useful for simplifying and generalizing problems.

Method of Repeating Variables and Buckingham Pi Theorem

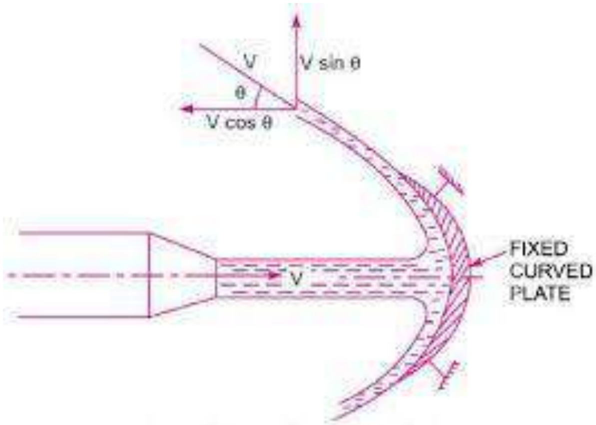
- **Buckingham Pi Theorem**:
  - If a physical problem involves variables and fundamental dimensions, then it can be described using dimensionless groups (groups).
  - Steps:
    1. List all variables.
    2. Determine the fundamental dimensions.
    3. Identify repeating variables.
    4. Form dimensionless groups.
- **Common Dimensionless Numbers**:
  - Reynolds Number (): (Ratio of inertial to viscous forces).
  - Froude Number (): (Flow gravity effects).
  - Mach Number (): (Ratio of velocity to speed of sound).

UNIT-IV

Basics of Turbo machinery

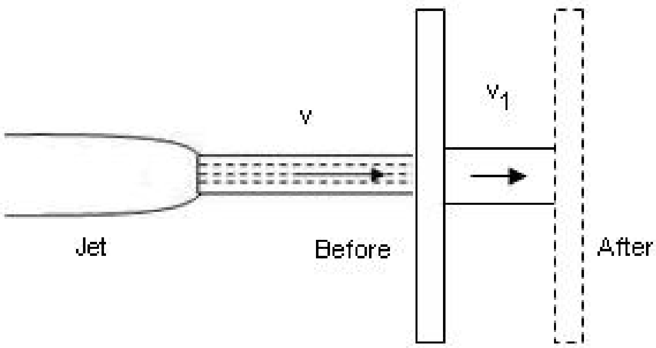
Hydrodynamic Force of Jets on Stationary and Moving Flat, Inclined, and Curved Vanes

- **Jet Impact on Stationary Vanes:**



  - When a high-velocity jet strikes a stationary vane, the force exerted by the jet can be determined by the rate of change of momentum.
  - The hydrodynamic force on the vane is equal to the change in momentum of the fluid as it moves from the jet to the vane.
  - **Equation**:
    F=m˙(V1−V2)F = \dot{m} (V_1 - V_2)F=m˙(V1−V2)
    where m˙\dot{m}m˙ is the mass flow rate, V1V_1V1 is the velocity of the jet before impact, and V2V_2V2 is the velocity after impact.

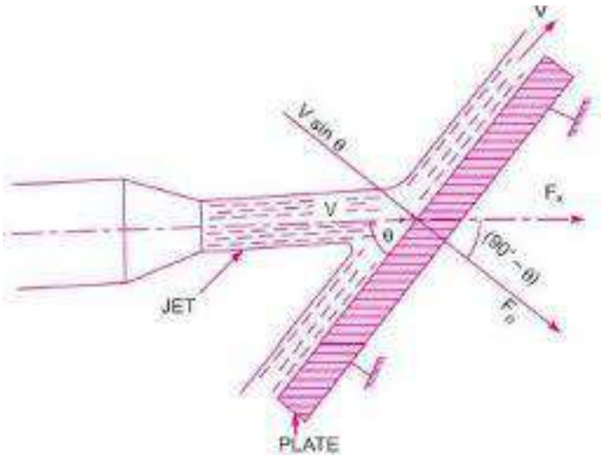
- **Jet Impact on Moving Vanes:**



  - If the vanes are moving, the relative velocity between the jet and the vane changes, which in turn affects the force exerted on the vane.

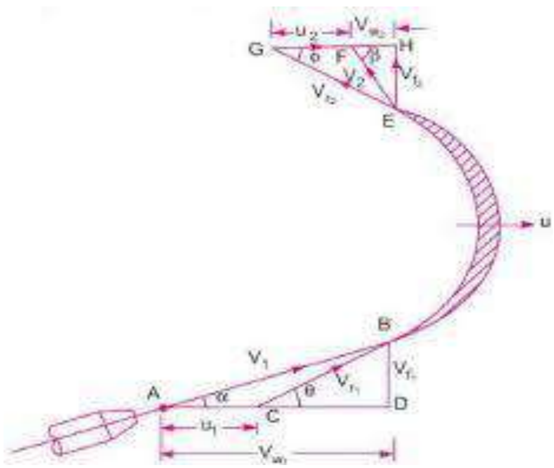
- The velocity of the jet relative to the vane is the difference between the velocity of the jet and the vane.
- For moving vanes, the hydrodynamic force is greater as the velocity of the vane adds to the velocity of the jet.

- **Jet Impact on Inclined and Curved Vanes:**



- For inclined and curved vanes, the angle of impact changes the direction and magnitude of the resulting hydrodynamic force.
- The force can be broken into components parallel and perpendicular to the surface of the vane.

Jet Striking Centrally and at Tip



- **Central Impact:**
  - If the jet strikes the vane centrally (normal to the surface), the force is maximized, and the velocity diagrams are symmetrical.

- **Tip Impact:**
    - If the jet strikes the vane at the tip, the angle of attack is more critical, and the force exerted can be less than that from a central impact.
    - The velocity diagrams become asymmetrical, and the work done by the jet on the vane varies with the tip velocity.

Velocity Diagrams

- **Velocity Triangle:**
    - The velocity diagram (or velocity triangle) is a graphical representation of the flow velocities at various stages in a turbomachine.
    - It helps in determining the components of the velocity and understanding the energy conversion process.
    - The key velocities in the diagram are:
        - **Absolute velocity** (VVV) - velocity of the fluid relative to the ground.
        - **Relative velocity** (WWW) - velocity of the fluid relative to the moving vane.
        - **Tangential velocity** (UUU) - velocity of the vane.

Work Done and Efficiency

- **Work Done**:
    - The work done by the jet on the vane is calculated based on the change in kinetic energy of the fluid.
    - **Formula**:
      W=ṁ×(V1U1−V2U2)W = \dot{m} \times (V_1 U_1 - V_2 U_2)W=ṁ×(V1U1−V2U2) where V1,V2V_1, V_2V1,V2 are the velocities before and after impact, and U1,U2U_1, U_2U1,U2 are the tangential velocities at inlet and outlet.
- **Efficiency**:
    - Efficiency is the ratio of useful work output to the energy input.
    - **Formula**:
      η=Output/ input

Flow Over Radial Vanes

- **Radial Flow**: In a radial vane machine, the flow of fluid is perpendicular to the axis of rotation, and the velocity components change in the radial direction.
- The velocity triangles for radial flow are also important for determining the work done, power output, and efficiency.
- The radial vane's design is aimed at controlling the pressure and velocity distribution across the flow path.

PROBLEMS

Problem 1: Velocity and Pressure Distribution for a Jet Hitting a Surface

Description: A water jet with a velocity V0=10 m/sV_0 = 10 \, \text{m/s}V0=10m/s exits a nozzle with a cross-sectional area A0=0.005 m2A_0 = 0.005 \, \text{m}^2A0=0.005m2. The jet strikes a flat surface at an angle of θ=30∘\theta = 30^\circθ=30∘. The density of water is ρ=1000 kg/m3\rho = 1000 \, \text{kg/m}^3ρ=1000kg/m3, and viscosity is negligible. Calculate the following:

1. The velocity of the jet immediately after striking the surface.
2. The pressure distribution along the surface beneath the impact region.
3. The total force exerted on the surface by the jet impact.

Solution:

1. **Velocity after impact:**

Since the jet impacts the surface, it will decelerate perpendicular to the surface due to stagnation, while maintaining its tangential velocity (parallel to the surface). The velocity normal to the surface becomes zero immediately after impact, while the velocity parallel to the surface remains the same as the initial tangential velocity:

- Normal velocity component Vnormal=V0cos(θ)V_{\text{normal}} = V_0 \cos(\theta)Vnormal=V0cos(θ)
- Tangential velocity component Vtangent=V0sin(θ)V_{\text{tangent}} = V_0 \sin(\theta)Vtangent=V0sin(θ)

Thus, the velocity of the jet after striking the surface will primarily be tangential, and the velocity parallel to the surface is:

Vtangent=10×sin(30∘)=10×0.5=5 m/sV_{\text{tangent}} = 10 \times \sin(30^\circ) = 10 \times 0.5 = 5 \, \text{m/s}Vtangent=10×sin(30∘)=10×0.5=5m/s

So, the jet will continue to move at 5 m/s5 \, \text{m/s}5m/s after the impact.

2. **Pressure Distribution:**

For an incompressible jet, the stagnation pressure on the surface (where the velocity normal to the surface is zero) can be calculated using Bernoulli's equation:

Pstag=P∞+12ρV02P_{\text{stag}} = P_{\infty} + \frac{1}{2} \rho V_0^2Pstag=P∞+21ρV02

Assuming atmospheric pressure P∞=0P_{\infty} = 0P∞=0 (gauge pressure) and using ρ=1000 kg/m3\rho = 1000 \, \text{kg/m}^3ρ=1000kg/m3, we get:

Pstag=12×1000×102=50000 PaP_{\text{stag}} = \frac{1}{2} \times 1000 \times 10^2 = 50000 \, \text{Pa}Pstag=21×1000×102=50000Pa

So, the stagnation pressure at the point of impact is 50000 Pa50000 \, \text{Pa}50000Pa. The pressure distribution will vary downstream from the impact point as the jet decelerates and spreads out.

3. **Force exerted on the surface:**

The total force exerted on the surface can be calculated using the momentum flux. The mass flow rate m˙\dot{m}m˙ is:

m˙=ρA0V0=1000×0.005×10=50 kg/s\dot{m} = \rho A_0 V_0 = 1000 \times 0.005 \times 10 = 50 \, \text{kg/s}m˙=ρA0V0=1000×0.005×10=50kg/s

The force on the surface due to the jet impact is given by the change in momentum flux:

F=m˙V0cos(θ)=50×10×cos(30∘)=50×10×0.866=433 NF = \dot{m} V_0 \cos(\theta) = 50 \times 10 \times \cos(30^\circ) = 50 \times 10 \times 0.866 = 433 \, \text{N}F=m˙V0 cos(θ)=50×10×cos(30∘)=50×10×0.866=433N

So, the total force exerted on the surface by the jet impact is 433 N433 \, \text{N}433N.

Problem 2: Energy Loss Due to Jet Impact

Description: An air jet with an initial velocity of V0=50 m/sV_0 = 50 \, \text{m/s}V0=50m/s and a mass flow rate of m˙=0.1 kg/s\dot{m} = 0.1 \, \text{kg/s}m˙=0.1kg/s strikes a surface. Due to turbulence, 40% of the kinetic energy is lost during the impact. Calculate the following:

1. The initial kinetic energy of the jet.
2. The energy lost due to turbulence.
3. The remaining kinetic energy after impact.

Solution:

1. **Initial Kinetic Energy:**

The kinetic energy of the jet is calculated using the equation:

Ek=12m˙V02E_k = \frac{1}{2} \dot{m} V_0^2Ek=21m˙V02

Substituting the given values:

Ek=12×0.1×502=12×0.1×2500=125 J/s=125 WE_k = \frac{1}{2} \times 0.1 \times 50^2 = \frac{1}{2} \times 0.1 \times 2500 = 125 \, \text{J/s} = 125 \, \text{W}Ek=21×0.1×502=21 ×0.1×2500=125J/s=125W

So, the initial kinetic energy of the jet is 125 W125 \, \text{W}125W.

2. **Energy Lost Due to Turbulence:**

Given that 40% of the energy is dissipated due to turbulence, the energy lost is:

Elost=0.4×125=50 WE_{\text{lost}} = 0.4 \times 125 = 50 \, \text{W}Elost=0.4×125=50W

So, the energy lost due to turbulence is 50 W50 \, \text{W}50W.

3. **Remaining Kinetic Energy:**

The remaining kinetic energy is the difference between the initial energy and the lost energy:
Eremaining=125−50=75 WE_{\text{remaining}} = 125 - 50 = 75 \, \text{W}Eremaining=125−50=75W
So, the remaining kinetic energy after the impact is 75 W75 \, \text{W}75W.

Problem 3: Jet Impingement Heat Transfer

Description: A jet of hot gas with a temperature of T0=600 KT_0 = 600 \, \text{K}T0=600K, mass flow rate m˙=0.2 kg/s\dot{m} = 0.2 \, \text{kg/s}m˙=0.2kg/s, and specific heat capacity cp=1000 J/kg\cdotpKc_p = 1000 \, \text{J/kg·K}cp=1000J/kg\cdotpK impinges on a cold surface at Ts=300 KT_s = 300 \, \text{K}Ts=300K. Calculate the following:

1. The heat transfer rate from the jet to the surface if the temperature of the jet decreases by 150 K upon impact.
2. How would the heat transfer rate change if the velocity of the jet were doubled?

Solution:

1. **Heat Transfer Rate:**

The heat transfer rate is calculated using the equation:
Q=m˙cpΔTQ = \dot{m} c_p \Delta TQ=m˙cpΔT
where ΔT\Delta TΔT is the temperature change of the jet.
Substituting the values:
Q=0.2×1000×150=30000 W=30 kWQ = 0.2 \times 1000 \times 150 = 30000 \, \text{W} = 30 \, \text{kW}Q=0.2×1000×150=30000W=30kW
So, the heat transfer rate from the jet to the surface is 30 kW30 \, \text{kW}30kW.

2. **Effect of Doubling the Velocity:**

If the velocity of the jet is doubled, the mass flow rate will also double (since the mass flow rate is proportional to the velocity). Thus, the new mass flow rate will be:
m˙new=2×0.2=0.4 kg/s\dot{m}_{\text{new}} = 2 \times 0.2 = 0.4 \, \text{kg/s}m˙new=2×0.2=0.4kg/s
The heat transfer rate will be:
Qnew=0.4×1000×150=60000 W=60 kWQ_{\text{new}} = 0.4 \times 1000 \times 150 = 60000 \, \text{W} = 60 \, \text{kW}Qnew=0.4×1000×150=60000W=60kW
So, doubling the velocity of the jet doubles the heat transfer rate, increasing it to 60 kW60 \, \text{kW}60kW.

Problem 4: Shockwave Formation Due to Supersonic Jet Impact

Description: A supersonic jet of air exits a nozzle with a Mach number M=2M = 2M=2. The diameter of the nozzle is D=0.05 mD = 0.05 \, \text{m}D=0.05m. The jet strikes a flat surface, and a shockwave forms. Assume the speed of sound in air is a=343 m/sa = 343 \, \text{m/s}a=343m/s. Calculate the following:

1. The exit velocity of the jet.
2. The pressure change due to the formation of the shockwave.

Solution:

1. **Exit Velocity:**

The velocity of the jet is given by:

V=M·a=2×343=686 m/sV = M \cdot a = 2 \times 343 = 686 \, \text{m/s}V=M·a=2×343=686m/s

So, the exit velocity of the jet is 686 m/s686 \, \text{m/s}686m/s.

2. **Pressure Change Due to Shockwave:**

The pressure change across a normal shock can be calculated using the normal shock relations:

P2P1=(γ+1)M2(γ−1)M2+2\frac{P_2}{P_1} = \frac{(\gamma + 1) M^2}{(\gamma - 1) M^2 + 2}P1 P2=(γ−1)M2+2(γ+1)M2

For air, γ=1.4\gamma = 1.4γ=1.4. Substituting M=2M = 2M=2:

P2P1=(1.4+1)×22(1.4−1)×22+2=2.4×40.4×4+2=9.63.6=2.67\frac{P_2}{P_1} = \frac{(1.4 + 1) \times 2^2}{(1.4 - 1) \times 2^2 + 2} = \frac{2.4 \times 4}{0.4 \times 4 + 2} = \frac{9.6}{3.6} = 2.67P1P2 =(1.4−1)×22+2(1.4+1)×22=0.4×4+22.4×4=3.69.6=2.67

So, the pressure after the shock is 2.67 times the initial pressure P1P_1P1.

Problem 5: Two-Dimensional Jet Impingement on a Surface

Description: A water jet with a velocity of V0=15 m/sV_0 = 15 \, \text{m/s}V0=15m/s and a diameter of D0=0.01 mD_0 = 0.01 \, \text{m}D0=0.01m strikes a flat surface perpendicularly. The mass flow rate of the jet is m˙=1.5 kg/s\dot{m} = 1.5 \, \text{kg/s}m˙=1.5kg/s, and the density of water is ρ=1000 kg/m3\rho = 1000 \, \text{kg/m}^3ρ=1000kg/m3. Calculate the following:

1. The velocity of the jet at the nozzle.
2. The pressure drop at the surface under the impact region due to the jet's stagnation.
3. The total force exerted on the surface by the jet.

Solution:

1. **Velocity of the Jet at the Nozzle:**

The mass flow rate is given by:

m˙=ρA0V0\dot{m} = \rho A_0 V_0m˙=ρA0V0

The area of the nozzle A0A_0A0 is:

A0=πD024=π(0.01)24=7.85×10−5 m2A_0 = \frac{\pi D_0^2}{4} = \frac{\pi (0.01)^2}{4} = 7.85 \times 10^{-5} \, \text{m}^2A0=4πD02=4π(0.01)2=7.85×10−5m2

Solving for the velocity V0V_0V0:

V0=m˙ρA0=1.51000×7.85×10−5=19.1 m/sV_0 = \frac{\dot{m}}{\rho A_0} = \frac{1.5}{1000 \times 7.85 \times 10^{-5}} = 19.1 \, \text{m/s}V0=ρA0m˙=1000×7.85×10−51.5=19.1m/s

So, the velocity at the nozzle is 19.1 m/s19.1 \, \text{m/s}19.1m/s.

2. **Pressure Drop at the Surface:**

The stagnation pressure PstagP_{\text{stag}}Pstag is given by:

Pstag=12ρV02P_{\text{stag}} = \frac{1}{2} \rho V_0^2Pstag=21ρV02

Substituting the known values:

Pstag=12×1000×(15)2=112500 PaP_{\text{stag}} = \frac{1}{2} \times 1000 \times (15)^2 = 112500 \, \text{Pa}Pstag=21×1000×(15)2=112500Pa

The pressure drop at the surface due to the impact of the jet is 112500 Pa112500 \, \text{Pa}112500Pa.

3. **Total Force Exerted on the Surface:**

The total force can be calculated using the momentum flux:

F=m˙V0=1.5×15=22.5 NF = \dot{m} V_0 = 1.5 \times 15 = 22.5 \, \text{N}F=m˙V0=1.5×15=22.5N

So, the total force exerted on the surface by the jet is 22.5 N22.5 \, \text{N}22.5N.

Problem 6: Energy Dissipation in a High-Speed Jet

Description: A jet of air with an initial velocity of V0=300 m/sV_0 = 300 \, \text{m/s}V0=300m/s and a mass flow rate of m˙=0.5 kg/s\dot{m} = 0.5 \, \text{kg/s}m˙=0.5kg/s strikes a flat surface at a 90-degree angle. 25% of the jet's kinetic energy is lost due to turbulence and heat transfer. Calculate the following:

1. The initial kinetic energy of the jet.
2. The energy lost due to turbulence.
3. The remaining kinetic energy after the impact.

Solution:

1. **Initial Kinetic Energy:**

The kinetic energy of the jet is given by:

Ek=12m˙V02E_k = \frac{1}{2} \dot{m} V_0^2Ek=21m˙V02

Substituting the given values:

Ek=12×0.5×3002=0.25×90000=22500 J/s=22.5 kWE_k = \frac{1}{2} \times 0.5 \times 300^2 = 0.25 \times 90000 = 22500 \, \text{J/s} = 22.5 \, \text{kW}Ek=21×0.5×3002=0.25×90000=22500J/s=22.5kW

So, the initial kinetic energy of the jet is 22.5 kW22.5 \, \text{kW}22.5kW.

2. **Energy Lost Due to Turbulence:**

Given that 25% of the energy is dissipated:
Elost=0.25×22500=5625 J/s=5.625 kWE_{\text{lost}} = 0.25 \times 22500 = 5625 \, \text{J/s} = 5.625 \, \text{kW}Elost=0.25×22500=5625J/s=5.625kW
So, the energy lost due to turbulence is 5.625 kW5.625 \, \text{kW}5.625kW.

3. **Remaining Kinetic Energy:**

The remaining kinetic energy is the difference between the initial energy and the lost energy:
Eremaining=22500−5625=16875 J/s=16.875 kWE_{\text{remaining}} = 22500 - 5625 = 16875 \, \text{J/s} = 16.875 \, \text{kW}Eremaining=22500−5625=16875J/s=16.875kW
So, the remaining kinetic energy after the impact is 16.875 kW16.875 \, \text{kW}16.875kW.

Problem 7: Supersonic Jet Impact and Shockwave Formation

Description: A supersonic jet of air with a Mach number M=3M = 3M=3 strikes a flat surface. The jet has a diameter of D=0.05 mD = 0.05 \, \text{m}D=0.05m, and the speed of sound in air is a=343 m/sa = 343 \, \text{m/s}a=343m/s. Calculate the following:

1. The velocity of the jet at the nozzle.
2. The shockwave pressure increase due to the jet impact.

Solution:

1. **Velocity of the Jet:**

The velocity of the jet is given by:
V=M·a=3×343=1029 m/sV = M \cdot a = 3 \times 343 = 1029 \, \text{m/s}V=M·a=3×343=1029m/s
So, the velocity of the jet at the nozzle is 1029 m/s1029 \, \text{m/s}1029m/s.

2. **Shockwave Pressure Increase:**

The pressure increase across a normal shock can be calculated using the shock relations. For air (γ=1.4\gamma = 1.4γ=1.4) and M=3M = 3M=3, the pressure ratio is:
P2P1=(γ+1)M2(γ−1)M2+2\frac{P_2}{P_1} = \frac{(\gamma + 1) M^2}{(\gamma - 1) M^2 + 2}P1P2=(γ−1)M2+2(γ+1)M2
Substituting the values:
P2P1=(1.4+1)×32(1.4−1)×32+2=2.4×90.4×9+2=21.65.6=3.857\frac{P_2}{P_1} = \frac{(1.4 + 1) \times 3^2}{(1.4 - 1) \times 3^2 + 2} = \frac{2.4 \times 9}{0.4 \times 9 + 2} = \frac{21.6}{5.6} = 3.857P1P2=(1.4−1)×32+2(1.4+1)×32=0.4×9+22.4×9=5.621.6=3.857
So, the pressure after the shock is 3.8573.8573.857 times the initial pressure.

Problem 8: Jet and Crossflow Interaction

Description: A jet of water is injected into a crossflow of air. The velocity of the jet is Vjet=10 m/sV_{\text{jet}} = 10 \, \text{m/s}Vjet=10m/s, and the velocity of the crossflow is Vcf=15 m/sV_{\text{cf}} = 15 \, \text{m/s}Vcf=15m/s. The diameter of the jet is D=0.02 mD = 0.02 \, \text{m}D=0.02m. The density of water is ρw=1000 kg/m3\rho_{\text{w}} = 1000 \, \text{kg/m}^3ρw =1000kg/m3, and the density of air is ρa=1.2 kg/m3\rho_{\text{a}} = 1.2 \, \text{kg/m}^3ρa=1.2kg/m3.

1. Calculate the momentum flux of the water jet.
2. Determine the total momentum flux in the direction of the crossflow.
3. Discuss how the angle of the jet affects the mixing and spread of the jet.

Solution:

1. **Momentum Flux of the Water Jet:**

The mass flow rate of the water jet is:

m˙=ρwAjetVjet=1000×πD24×10=1000×π(0.02)24×10=31.42 kg/s\dot{m} = \rho_{\text{w}} A_{\text{jet}} V_{\text{jet}} = 1000 \times \frac{\pi D^2}{4} \times 10 = 1000 \times \frac{\pi (0.02)^2}{4} \times 10 = 31.42 \, \text{kg/s}m˙=ρwAjetVjet=1000×4πD2×10=1000×4π(0.02)2 ×10=31.42kg/s

The momentum flux is:

Momentum flux=m˙Vjet=31.42×10=314.2 N\text{Momentum flux} = \dot{m} V_{\text{jet}} = 31.42 \times 10 = 314.2 \, \text{N}Momentum flux=m˙Vjet=31.42×10=314.2N

So, the momentum flux of the water jet is 314.2 N314.2 \, \text{N}314.2N.

2. **Total Momentum Flux in the Direction of the Crossflow:**

To account for the crossflow, we will assume that the momentum of the jet will be partially "redirected" due to the interaction. Assuming the jet is directed at a 45-degree angle to the crossflow, the momentum flux in the direction of the crossflow is:

Momentum flux (crossflow)=Momentum flux×cos(45∘)=314.2×cos(45∘)=314.2×0.707=222.8 N\text{Momentum flux (crossflow)} = \text{Momentum flux} \times \cos(45^\circ) = 314.2 \times \cos(45^\circ) = 314.2 \times 0.707 = 222.8 \, \text{N}Momentum flux (crossflow)=Momentum flux×cos(45∘)=314.2×cos(45∘)=314.2×0.707=222.8N

3. **Effect of the Jet Angle on Mixing:**

The angle of the jet relative to the crossflow affects the trajectory and the interaction region. A smaller angle (closer to perpendicular) would cause the jet to penetrate deeper into the crossflow, promoting better mixing. A larger angle would result in the jet dispersing more quickly along the surface, leading to less effective mixing and a larger spread of the jet.

Hydraulic Turbines:

Turbines can be classified based on various criteria, such as the fluid used to drive them, their energy conversion process, and the direction of fluid flow. Here's a detailed overview of different classifications:

1. Classification Based on the Type of Fluid

- Steam Turbine:
  - Operates using steam as the working fluid.
  - Commonly used in power plants (thermal power plants) to generate electricity.
  - The steam expands through the turbine blades, causing the rotor to spin.
- Gas Turbine:
  - Uses hot gases (usually from combustion) to generate power.
  - Common in both power generation and aviation (jet engines).
  - Works on the principle of the Brayton cycle (compression, combustion, and expansion of gases).
- Hydraulic (Water) Turbine:
  - Uses water to drive the turbine.
  - Widely used in hydropower plants.
  - Water flow turns the blades, generating mechanical energy.
- Wind Turbine:
  - Uses the kinetic energy of the wind to rotate the blades.
  - Used for generating electricity in wind farms.
- Air Turbine:
  - Similar to a gas turbine but uses compressed air to generate mechanical power.

2.Classification Based on Energy Conversion Process

- Impulse Turbine:
  - Converts kinetic energy of the fluid directly into mechanical energy.
  - The working fluid (e.g., steam or water) strikes the turbine blades at high speed, causing them to move.
  - Examples: Pelton wheel (water turbine), De Laval turbine (steam turbine).
- Reaction Turbine:
  - Converts the energy of the fluid in two steps: first expanding the fluid and then reacting with the blades to produce motion.
  - The fluid experiences both pressure and velocity changes as it passes through the blades.

    - Examples: Francis turbine (water turbine), Kaplan turbine (water turbine), Parsons turbine (steam turbine).

3. Classification Based on the Direction of Fluid Flow

- Axial Flow Turbine:
    - The fluid moves parallel to the axis of rotation.
    - Common in both steam and gas turbines.
    - Example: Jet engines (gas turbines), steam turbines used in power plants.
- Radial Flow Turbine:
    - The fluid flows radially from the center of the rotor to the edges (or vice versa).
    - Found in some water turbines and certain small gas turbines.
    - Example: Radial inflow turbines used in some small engines.
- Mixed Flow Turbine:
    - A combination of both axial and radial flow.
    - The fluid moves at an angle to the axis but is not purely axial or radial.
    - Example: Francis turbine (in hydropower plants), used in some gas turbines.

4. Classification Based on Function

- Power Turbine:
    - Designed to generate mechanical power.
    - Found in power plants and engines.
    - Examples: Steam, gas, and water turbines used for generating electricity.
- Compressor Turbine:
    - Used to drive compressors in engines, such as in jet engines.
    - The turbine helps compress air for combustion in the engine.
- Propulsion Turbine:
    - Used in the propulsion of ships, aircraft, and rockets.
    - The energy generated is directly converted into thrust or movement.

5. Classification Based on Size and Application

- Large-Scale Turbines:
    - Typically used in industrial applications and power generation.
    - Examples: Steam turbines for electricity generation in large plants, gas turbines for power plants.
- Small-Scale Turbines:
    - Often used in specialized or small-scale applications, such as small hydropower plants or in small aircraft engines.

- Examples: Microturbines for distributed energy generation.

6. Classification Based on Construction

- Single-Stage Turbine:
  - Has only one row of blades.
  - Suitable for smaller applications or low power generation.
- Multi-Stage Turbine:
  - Contains multiple rows of blades, with each stage extracting more energy from the fluid.
  - Common in large-scale power generation systems.
- Impulse Reaction Turbine:
  - Combines both impulse and reaction processes in different stages.
  - Often used in specific types of steam turbines or hydropower turbines.

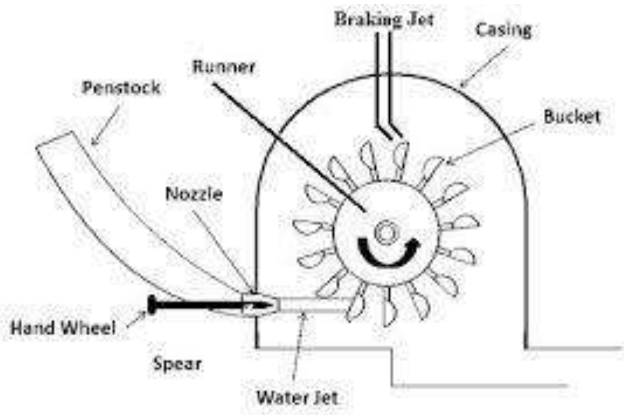
7. Other Classifications

- Turbine Blades:
  - Fixed Blades: Blades that do not move.
  - Rotating Blades: Blades that rotate with the fluid flow.
  - Variable Blades: Blades that can adjust their angle to optimize performance.

Examples of Specific Turbines:

- Pelton Turbine: A type of impulse turbine used in high-head hydropower plants.
- Francis Turbine: A reaction turbine used in medium- to low-head hydropower plants.
- Kaplan Turbine: A type of reaction turbine that is adjustable and used in low-head hydropower plants.
- De Laval Turbine: A high-speed steam turbine that is an example of an impulse turbine.

1. Impulse Turbine:



Impulse turbines are a type of water turbine where the force of the water jet is used to produce rotational energy. The key feature of impulse turbines is that the water pressure is converted to kinetic energy (velocity) before it strikes the turbine blades.

Key Characteristics:

- **Working Principle:** In impulse turbines, the water flows through a nozzle, and as the water exits the nozzle at high velocity, it strikes the blades of the turbine. The kinetic energy of the high-speed water is transferred to the turbine blades, causing them to rotate.
- **Energy Transfer:** The water jet does not lose its pressure before hitting the blades, only its velocity. The blades are designed to redirect the water flow efficiently, thereby transferring momentum to the turbine wheel.
- **No Pressure Change on Blades:** Since the pressure of the water doesn't significantly change when it hits the blades, the turbine is called an impulse turbine.
- **Application:** Commonly used in high-head conditions where there is a large height difference (elevation) between the water source and the turbine.

Advantages:

- Simple in design.
- Can operate under a wide range of flow rates.
- High efficiency at high heads.

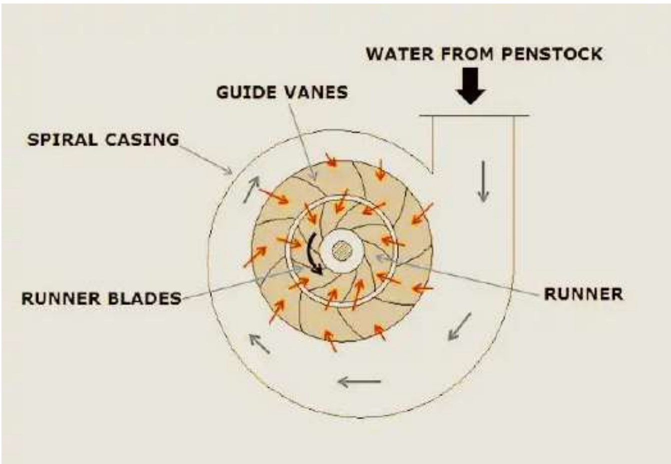
Disadvantages:

- Not efficient in low-head conditions (low height difference).

Examples:

- **Pelton Wheel** (described below).

2. Reaction Turbine:



Reaction turbines operate under the principle of both pressure and velocity changes. In reaction turbines, water flows over the blades, and both the velocity and pressure of the water change as it moves through the turbine.

Key Characteristics:

- **Working Principle:** Water enters the turbine under high pressure and flows through the blades. As the water passes through the turbine blades, its velocity increases, and its pressure decreases. This pressure difference across the blades causes them to move.
- **Energy Transfer:** The reaction force is generated due to the change in both pressure and velocity of water. Both sides of the blades experience a change in pressure and velocity, leading to the turbine's rotation.
- **Type of Flow:** The flow through a reaction turbine is continuous, and the blades are completely immersed in water, unlike impulse turbines.

Advantages:

- More efficient in low-head conditions.
- Can operate with a steady flow of water.
- Suitable for large-scale power generation.

Disadvantages:

- Complex design with a need for sealed components.

Examples:

- **Francis Turbine**
- **Kaplan Turbine**

3. Pelton Wheel:

The **Pelton Wheel** is an example of an **Impulse Turbine** designed for high-head applications. It is specifically used to extract energy from the kinetic energy of a jet of water.

Key Characteristics:

- **Design:** The Pelton wheel consists of a wheel with buckets (or cups) attached around its circumference. A high-velocity water jet is directed at these cups.
- **Working Principle:** Water is ejected from a nozzle and strikes the buckets, causing them to rotate. The force of the water jet is converted into mechanical energy. After striking a bucket, the water is redirected (splashed away), so the water's kinetic energy is absorbed, and the wheel rotates.

- **Energy Transfer:** The change in the water's velocity (from the nozzle to the bucket) results in the rotational movement of the wheel. Since the pressure is constant before and after the nozzle, the Pelton wheel works based purely on the momentum transfer of the water jet.

Design Features:

- **Buckets:** The Pelton wheel uses double-cupped buckets that allow water to be directed in two ways, ensuring maximum energy transfer.
- **Nozzle:** A nozzle is used to direct a focused jet of water toward the buckets, and it may include a mechanism to regulate the flow of water to match the turbine's load.

Advantages:

- Efficient for high-head conditions (water falling from a significant height).
- Simple construction and design.
- High efficiency in converting high-velocity water jets into mechanical energy.

Disadvantages:

- Not suitable for low-head applications.
- Needs a high-velocity water jet for operation.

Applications:

- Hydroelectric power plants in mountainous regions where water falls from a significant height.

Comparison:

Feature	**Impulse Turbine**	**Reaction Turbine**
Energy Transfer	Kinetic energy (velocity)	Pressure and velocity change
Water Pressure	Remains constant at the blades	Changes across the blades
Example	Pelton Wheel	Francis, Kaplan Turbines
Application	High-head conditions	Low to medium-head conditions
Design	Simpler	More complex, sealed system

Both types of turbines are crucial for harnessing hydroelectric power, but they operate under different conditions and with different efficiencies based on the head (height) of the water. The Pelton wheel is a classic impulse turbine used in high-head scenarios, while reaction turbines like the Francis and Kaplan turbines are more common for large-scale, medium to low-head hydroelectric systems.

.

Hydraulic Design and Draft Tube

- **Draft Tube**: The draft tube is used to reduce the velocity of water exiting a turbine while recovering kinetic energy.

- **Theory**: The draft tube operates based on the principle of a pressure recovery, where water expands to a larger diameter, converting kinetic energy into pressure energy.
- **Function**: It reduces the speed of the water as it exits the turbine, allowing for a more efficient recovery of energy.
- **Efficiency**: A well-designed draft tube enhances the overall efficiency of the turbine system.

Problems on Impulse Turbine

Problem 1: An impulse turbine receives steam at a pressure of 6 bar and a temperature of 250°C. The steam is expanded through a nozzle which creates a velocity of 180 m/s. The turbine blades are designed to be tangent to the direction of steam flow, ensuring no axial velocity. The mass flow rate of the steam is 12 kg/s.

Calculate the power developed by the turbine.

Given:

- Pressure of steam = 6 bar
- Temperature of steam = 250°C
- Velocity of steam through nozzle V=180 m/sV = 180 \, \text{m/s}V=180m/s
- Mass flow rate m˙=12 kg/s\dot{m} = 12 \, \text{kg/s}m˙=12kg/s

Solution:

1. **Calculate the velocity of steam through the nozzle:**
   The velocity of steam from the nozzle is already given as V=180 m/sV = 180 \, \text{m/s}V=180m/s, so no need for further calculation here.
2. **Calculate the power developed by the turbine:**
   The power developed by the turbine is given by the equation:
   P=m˙×V×(velocity change)P = \dot{m} \times V \times \left(\text{velocity change} \right)P=m˙×V×(velocity change)
   Since the blades are tangent to the steam flow direction, there is no axial velocity, and the change in velocity is simply VVV, the velocity through the nozzle.
   P=m˙×V2P = \dot{m} \times V^2P=m˙×V2
3. **Substitute the known values:**
   P=12×(180)2=12×32400=388800 W=388.8 kWP = 12 \times (180)^2 = 12 \times 32400 = 388800 \, \text{W} = 388.8 \, \text{kW}P=12×(180)2=12×32400=388800W=388.8kW
   Answer: The power developed by the turbine is 388.8 kW.

Problem 2:

Problem: An impulse turbine operates at a speed of 2000 rpm. The steam enters the blades of the turbine with a velocity of 250 m/s and exits with a velocity of 150 m/s. The blade speed is 80 m/s, and the mass flow rate of the steam is 10 kg/s.

Calculate the power developed by the turbine.

Given:

- Blade speed Vb=80 m/sV_b = 80 \, \text{m/s}Vb=80m/s
- Inlet velocity V1=250 m/sV_1 = 250 \, \text{m/s}V1=250m/s
- Exit velocity V2=150 m/sV_2 = 150 \, \text{m/s}V2=150m/s
- Mass flow rate m˙=10 kg/s\dot{m} = 10 \, \text{kg/s}m˙=10kg/s
- Blade speed Vb=80 m/sV_b = 80 \, \text{m/s}Vb=80m/s

Solution:

1. **Velocity components:**

   In an impulse turbine, the power depends on the change in the tangential velocity of the steam as it interacts with the blades. The equation for power is:

   P=m˙×(V1−Vb)×(V1−Vb)P = \dot{m} \times \left(V_1 - V_b \right) \times \left(V_1 - V_b \right)P=m˙×(V1−Vb)×(V1−Vb)

   Where:

   - VbV_bVb is the blade velocity.
   - V1V_1V1 is the initial velocity of steam.

   We can simplify by finding the relative velocity difference:

2. **Substitute values into the formula:**

Problem: An impulse turbine is supplied with steam at a pressure of 8 bar and a temperature of 300°C. The steam is expanded through a nozzle, which accelerates it to a velocity of 200 m/s. The steam enters the turbine blades with no axial velocity. The mass flow rate of the steam is 10 kg/s.

Calculate the power developed by the turbine.

Given:

- Inlet velocity of steam V=200 m/sV = 200 \, \text{m/s}V=200m/s
- Mass flow rate m˙=10 kg/s\dot{m} = 10 \, \text{kg/s}m˙=10kg/s
- The steam enters with no axial velocity.

Solution:

1. **Calculate the power developed by the turbine:**

The power developed by the turbine is given by the equation:

P=m˙×V2P = \dot{m} \times V^2P=m˙×V2

Where:

- m˙\dot{m}m˙ is the mass flow rate,
- VVV is the velocity of steam through the nozzle.

2. **Substitute the known values:**

P=10×(200)2=10×40000=400000 W=400 kWP = 10 \times (200)^2 = 10 \times 40000 = 400000 \, \text{W} = 400 \, \text{kW}P=10×(200)2=10×40000=400000W=400kW

Answer: The power developed by the turbine is **400 kW**.

Problem 3: An impulse turbine has steam entering at a velocity of 250 m/s and leaving at 150 m/s. The blades of the turbine rotate with a tangential speed of 100 m/s. The mass flow rate of the steam is 15 kg/s.

Calculate the work done per second (power) by the turbine.

Given:

- Inlet velocity V1=250 m/sV_1 = 250 \, \text{m/s}V1=250m/s
- Exit velocity V2=150 m/sV_2 = 150 \, \text{m/s}V2=150m/s
- Blade velocity Vb=100 m/sV_b = 100 \, \text{m/s}Vb=100m/s
- Mass flow rate m˙=15 kg/s\dot{m} = 15 \, \text{kg/s}m˙=15kg/s

Solution:

1. **Calculate the change in velocity relative to the blades:**

In impulse turbines, the power depends on the change in tangential velocity. The velocity relative to the blades is the difference between the steam velocity and blade velocity.

For inlet steam:

Vrel1=V1−Vb=250−100=150 m/sV_{rel1} = V_1 - V_b = 250 - 100 = 150 \, \text{m/s}Vrel1=V1−Vb=250−100=150m/s

For exit steam:

Vrel2=V2−Vb=150−100=50 m/sV_{rel2} = V_2 - V_b = 150 - 100 = 50 \, \text{m/s}Vrel2=V2−Vb=150−100=50m/s

2. **Calculate the power developed:**

The power developed is given by:

P=m˙×(Vrel12−Vrel22)P = \dot{m} \times \left(V_{rel1}^2 - V_{rel2}^2 \right)P=m˙×(Vrel12−Vrel22)

1. **Substitute the known values:**

P=15×(1502−502)=15×(22500−2500)=15×20000=300000 W=300 kWP = 15 \times \left(150^2 - 50^2 \right) = 15 \times (22500 - 2500) = 15 \times 20000 = 300000 \, \text{W} = 300 \, \text{kW}P=15×(1502−502)=15×(22500−2500)=15×20000=300000W=300kW

Answer: The power developed by the turbine is **300 kW**.

Reaction Turbine Problem 1:

Problem: In a reaction turbine, the steam enters the blades at a pressure of 12 bar and a temperature of 350°C with a velocity of 180 m/s. It exits the blades at 1 bar with a velocity of 120 m/s. The mass flow rate of steam is 8 kg/s.

Calculate the power developed by the turbine.

Given:

- Inlet velocity V1=180 m/sV_1 = 180 \, \text{m/s}V1=180m/s
- Exit velocity V2=120 m/sV_2 = 120 \, \text{m/s}V2=120m/s
- Mass flow rate m˙=8 kg/s\dot{m} = 8 \, \text{kg/s}m˙=8kg/s

Solution:

1. **Calculate the power developed by the turbine:**

In a reaction turbine, the power developed is given by the equation:

P=m˙×(V12−V22)P = \dot{m} \times \left(V_1^2 - V_2^2 \right)P=m˙×(V12−V22)

2. **Substitute the known values:**

P=8×(1802−1202)P = 8 \times \left(180^2 - 120^2 \right)P=8×(1802−1202)

P=8×(32400−14400)=8×18000=144000 W=144 kWP = 8 \times (32400 - 14400) = 8 \times 18000 = 144000 \, \text{W} = 144 \, \text{kW}P=8×(32400−14400)=8×18000=144000W=144kW

Answer: The power developed by the turbine is **144 kW**.

Reaction Turbine Problem 2:

Problem: A reaction turbine is supplied with steam at a velocity of 150 m/s. The steam exits the turbine blades at a velocity of 100 m/s. The blade speed is 80 m/s. The mass flow rate is 10 kg/s.

Calculate the power developed by the turbine.

Given:

- Inlet velocity V1=150 m/sV_1 = 150 \, \text{m/s}V1=150m/s
- Exit velocity V2=100 m/sV_2 = 100 \, \text{m/s}V2=100m/s
- Blade velocity Vb=80 m/sV_b = 80 \, \text{m/s}Vb=80m/s
- Mass flow rate m˙=10 kg/s\dot{m} = 10 \, \text{kg/s}m˙=10kg/s

Solution:

1. **Calculate the relative velocities:**

The relative velocity is the difference between the steam velocity and blade velocity.

For inlet steam:

Vrel1=V1−Vb=150−80=70 m/sV_{rel1} = V_1 - V_b = 150 - 80 = 70 \, \text{m/s}Vrel1=V1−Vb =150−80=70m/s

For exit steam:

Vrel2=V2−Vb=100−80=20 m/sV_{rel2} = V_2 - V_b = 100 - 80 = 20 \, \text{m/s}Vrel2=V2−Vb =100−80=20m/s

2. **Calculate the power developed:**

The power developed by the turbine is given by:

P=m˙×(Vrel12−Vrel22)P = \dot{m} \times \left(V_{rel1}^2 - V_{rel2}^2 \right)P=m˙×(Vrel12 −Vrel22)

3. **Substitute the known values:**

P=10×(702−202)=10×(4900−400)=10×4500=45000 W=45 kWP = 10 \times \left(70^2 - 20^2 \right) = 10 \times (4900 - 400) = 10 \times 4500 = 45000 \, \text{W} = 45 \, \text{kW}P=10×(702−202)=10×(4900−400)=10×4500=45000W=45kW

Answer: The power developed by the turbine is **45 kW**.

Bonus Problem (Combined Impulse and Reaction Turbine):

Problem: An impulse reaction turbine operates with steam at an inlet velocity of 200 m/s and an outlet velocity of 120 m/s. The turbine blades rotate at a tangential velocity of 80 m/s. The mass flow rate of steam is 10 kg/s.

Calculate the power developed by the turbine.

Given:

- Inlet velocity V1=200 m/sV_1 = 200 \, \text{m/s}V1=200m/s
- Exit velocity V2=120 m/sV_2 = 120 \, \text{m/s}V2=120m/s
- Blade velocity Vb=80 m/sV_b = 80 \, \text{m/s}Vb=80m/s
- Mass flow rate m˙=10 kg/s\dot{m} = 10 \, \text{kg/s}m˙=10kg/s

Solution:

1. **Calculate the relative velocities:**

For inlet steam:

Vrel1=V1−Vb=200−80=120 m/sV_{rel1} = V_1 - V_b = 200 - 80 = 120 \, \text{m/s}Vrel1=V1−Vb =200−80=120m/s

For exit steam:

Vrel2=V2−Vb=120−80=40 m/sV_{rel2} = V_2 - V_b = 120 - 80 = 40 \, \text{m/s}Vrel2=V2−Vb =120−80=40m/s

2. **Calculate the power developed:**

The power developed by the turbine is:

P=m˙×(Vrel12−Vrel22)P = \dot{m} \times \left(V_{rel1}^2 - V_{rel2}^2 \right)P=m˙×(Vrel12 −Vrel22)

3. **Substitute the known values:**

P=10×(1202−402)=10×(14400−1600)=10×12800=128000 W=128 kWP = 10 \times \left(120^2 - 40^2 \right) = 10 \times (14400 - 1600) = 10 \times 12800 = 128000 \, \text{W} = 128 \, \text{kW}P=10×(1202−402)=10×(14400−1600)=10×12800=128000W=128kW

Answer: The power developed by the turbine is **128 kW**.

UNIT-V

Performance of Hydraulic Turbines

1. **Geometric Similarity:**
    - **Geometric similarity** refers to the proportional relationship between the physical dimensions of two hydraulic turbines.
    - It ensures that the turbines are similar in shape, but their size varies. When scaling up or down a turbine, key dimensions (like the diameter of blades, the height of the runner, etc.) follow a geometric ratio to maintain similar performance characteristics.
2. **Unit and Specific Quantities:**
    - **Unit quantities** are used to represent the performance of hydraulic turbines in a standardized way.
    - **Unit discharge (Q_0):** the discharge per unit of turbine's capacity.
    - **Unit power (P_0):** the power developed per unit of turbine capacity.
    - **Specific speed (N_s):** a dimensionless number used to characterize the turbine's geometry and performance. Ns=NQH3/4N_s = \frac{N\sqrt{Q}}{H^{3/4}}Ns=H3/4NQ

where:

- N = rotational speed in RPM
- Q = discharge (flow rate)
- H = head (height of water)

3. **Characteristic Curves:**
    - These curves show the relationship between various operational parameters of turbines such as power, efficiency, and flow rate under different operating conditions.
    - Common curves:
        - **Power vs. Flow**: Represents the power generated by the turbine at various flow rates.
        - **Efficiency vs. Flow**: Represents the efficiency of the turbine as a function of flow rate.
        - **Head vs. Flow**: Displays the available head at various flow rates.
4. **Governing of Turbines:**
    - Governing refers to controlling the speed and output of the turbine. It is done by varying the flow of water to the turbine blades, thus adjusting power output.
    - Types of governing mechanisms:
        - **Mechanical governing:** Uses mechanical devices like valves to adjust flow.
        - **Electronic governing:** Uses sensors and control systems for precise regulation.

5. **Selection of Type of Turbine:**
    - The selection depends on the specific application, head, and discharge.
        - **Pelton Turbines** are used for high-head, low-flow applications.
        - **Francis Turbines** are suited for medium-head, medium-flow conditions.
        - **Kaplan Turbines** are used for low-head, high-flow applications.
6. **Cavitation:**
    - Cavitation occurs when local pressure drops below the vapor pressure of water, leading to the formation of bubbles that implode when pressure increases.
    - Cavitation causes damage to turbine blades and reduces efficiency.
    - Prevention: Proper design, maintaining sufficient net positive suction head (NPSH), and avoiding excessive flow rates.
7. **Surge Tank:**
    - A surge tank is used to absorb sudden changes in water flow (such as during turbine shutdown or startup) to prevent water hammer.
    - It helps maintain constant pressure in the pipeline, protecting the system from mechanical damage.
8. **Water Hammer:**
    - Water hammer refers to the sudden increase in pressure when there is a rapid change in water flow (such as shutting a valve quickly).
    - This can cause severe damage to pipes and machinery, including turbines. Surge tanks are commonly used to mitigate this effect.

Hydraulic Systems

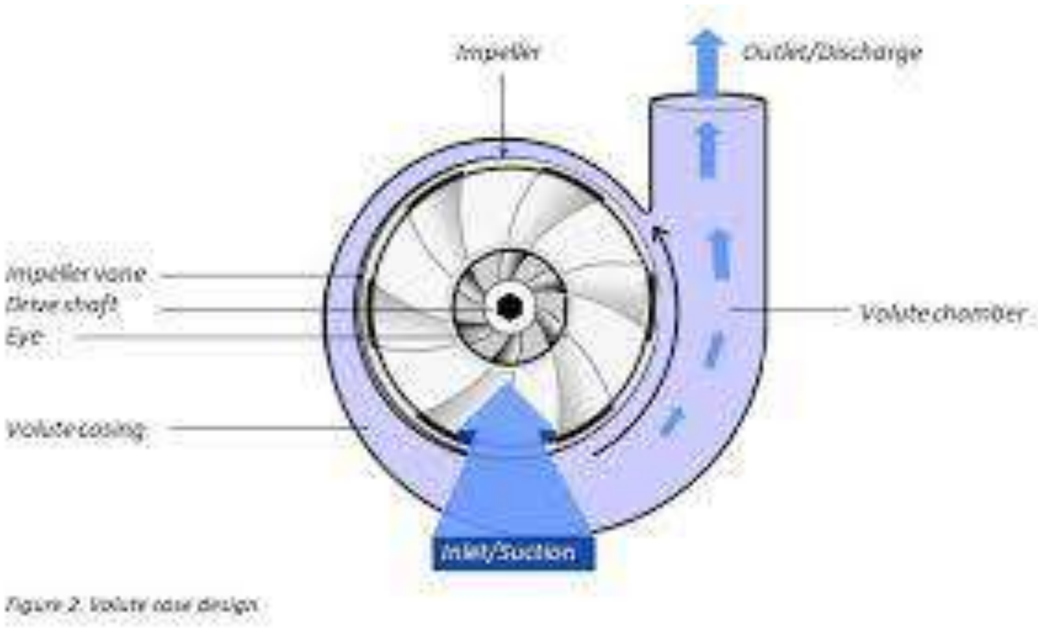
1. **Hydraulic Ram:**
    - A hydraulic ram uses the kinetic energy of flowing water to pump a smaller volume of water to a higher elevation.
    - It is typically used in locations where there is a low-flow, high-head source of water.
2. **Hydraulic Lift:**
    - A hydraulic lift uses the hydraulic principle of Pascal's law to lift heavy loads with minimal effort.
    - In these systems, pressure is applied to a liquid, and this pressure is transferred equally to a piston, lifting the load.
3. **Hydraulic Coupling:**
    - A hydraulic coupling is a device used to transmit rotary motion through hydraulic fluid.

- It is commonly used to connect a motor to machinery and can smoothly transmit torque, even if there is a change in speed.

Fluidics

1. **Fluidic Amplifiers:**
    - Fluidic amplifiers are devices that control fluid flow and pressure to amplify small input signals into larger output signals.
    - They typically use the principles of fluid dynamics (rather than mechanical parts) to increase flow.
2. **Fluidic Sensors:**
    - Fluidic sensors are used to detect changes in the properties of fluids such as pressure, flow, or temperature.
    - These sensors convert fluid property changes into electrical signals, which can be used for control systems.
3. **Fluidic Oscillators:**
    - Fluidic oscillators use the flow of a fluid to generate periodic oscillations or oscillatory motion.
    - These are used in various applications like generating sound waves, controlling systems, and in precision instruments.
4. **Advantages, Limitations, and Applications:**
    - **Advantages:**
        - Fluidic devices have no moving parts, reducing wear and tear.
        - They offer fast response times and high precision.
        - They are useful in environments where electrical or mechanical solutions might fail.
    - **Limitations:**
        - Fluidic systems can be sensitive to environmental conditions (e.g., temperature).
        - They may require complex fluid circuits and control mechanisms.
    - **Applications:**
        - Fluidic systems are used in aerospace (e.g., control of aircraft), robotics, and medical devices.

Centrifugal Pumps



1. **Classification:**
    - Centrifugal pumps are classified based on:
        - **Single-stage vs. Multi-stage:** Single-stage pumps have one impeller; multi-stage pumps have multiple impellers for higher head.
        - **Radial, Axial, and Mixed Flow:** Depending on the direction of fluid flow.
2. **Working:**
    - A centrifugal pump works by imparting kinetic energy to the fluid through a rotating impeller, which is then converted to pressure energy as the fluid exits the pump.
3. **Work Done and Manometric Head:**
    - **Work Done** is the energy transferred to the fluid by the pump, usually measured in terms of power (watts).
    - **Manometric Head (H_m):** The total head developed by the pump, which is the height to which the pump can raise water.
4. **Losses and Efficiencies:**
    - Losses in centrifugal pumps arise from:
        - **Friction losses** in the pump casing, pipes, and impeller.
        - **Mechanical losses** from bearing and shaft friction.
        - **Hydraulic losses** due to changes in flow velocity and direction.
    - **Efficiency** is the ratio of useful work done to the energy input to the pump.
5. **Specific Speed:**

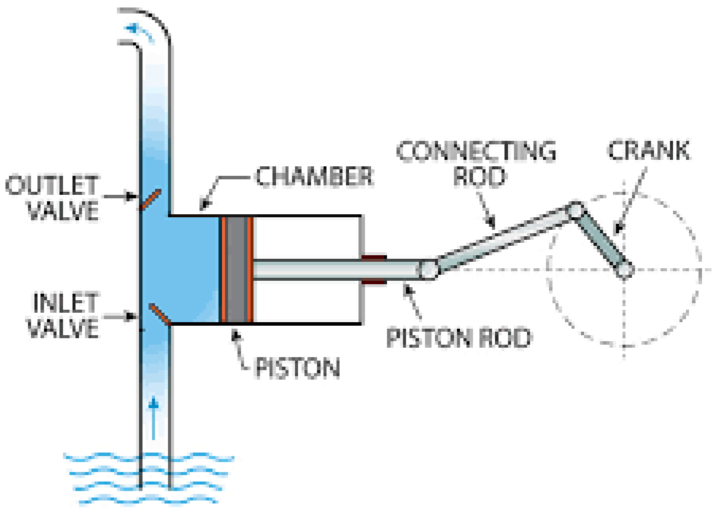
- **Specific speed (N_s)** is a dimensionless number used to determine the type of centrifugal pump and its performance. Ns=NQH3/4N_s = \frac{N \sqrt{Q}}{H^{3/4}}Ns=H3/4NQ

where:

- N = pump speed in RPM
- Q = flow rate (m^3/s)
- H = head (meters)

6. **Pumps in Series and Parallel:**
   - **Pumps in Series:** Used to increase the total head (pressure) without changing the flow rate.
   - **Pumps in Parallel:** Used to increase the flow rate without affecting the head.
7. **Performance Characteristic Curves:**
   - Performance curves describe the relationship between flow rate (Q) and head (H), power (P), and efficiency (η).
   - These curves are essential for evaluating and selecting the right pump for a specific application.
8. **Cavitation & NPSH:**
   - **Cavitation** occurs when the pressure at the pump inlet falls below the vapor pressure of the liquid, causing bubbles to form and damage the pump.
   - **Net Positive Suction Head (NPSH):** The pressure available at the pump inlet to prevent cavitation.
     - **NPSH required (NPSH_R):** The minimum pressure required by the pump to avoid cavitation.
     - **NPSH available (NPSH_A):** The pressure available from the system to prevent cavitation.

Reciprocating Pumps



1. **Working:**
    - Reciprocating pumps use a piston or diaphragm to move fluid in and out of the pump chamber. The piston moves back and forth, creating a suction on the inlet and pressure on the outlet.
2. **Discharge:**
    - The discharge of a reciprocating pump depends on the stroke length, stroke frequency, and area of the piston.
3. **Slip:**
    - Slip is the difference between the theoretical and actual discharge of the pump, typically caused by leakage.
4. **Indicator Diagrams:**
    - These diagrams plot the pressure in the pump cylinder during a complete pumping cycle.
    - The diagram helps in analyzing the performance of the pump, including suction, compression, and discharge phases. It is typically used for troubleshooting and performance evaluation.

Problems :

A centrifugal pump is pumping water at a flow rate of 150 liters per second (L/s) with a head of 25 meters. The efficiency of the pump is 75%. The density of water is 1000 kg/m^3, and gravitational acceleration is 9.81 m/s^2. Calculate the shaft power required by the pump.

Solution:

First, let’s calculate the power required by the pump using the formula for hydraulic power:

Phydraulic=ρ·g·Q·H1000P_{\text{hydraulic}} = \frac{\rho \cdot g \cdot Q \cdot H}{1000}Phydraulic =1000ρ·g·Q·H

Where:

- ρ\rhoρ = density of water = 1000 kg/m^3
- ggg = gravitational acceleration = 9.81 m/s^2
- QQQ = flow rate = 150 L/s = 150 × 10^{-3} m^3/s
- HHH = head = 25 m

Substitute the values into the equation:

Phydraulic=1000·9.81·150×10−3·251000P_{\text{hydraulic}} = \frac{1000 \cdot 9.81 \cdot 150 \times 10^{-3} \cdot 25}{1000}Phydraulic=10001000·9.81·150×10−3·25

Phydraulic=36.7875 kWP_{\text{hydraulic}} = 36.7875 \text{ kW}Phydraulic=36.7875 kW

Now, considering the efficiency of the pump, the shaft power required can be found using:

Pshaft=PhydraulicηP_{\text{shaft}} = \frac{P_{\text{hydraulic}}}{\eta}Pshaft=ηPhydraulic

Where:

- η\etaη = efficiency = 75% = 0.75

Substitute the values:

Pshaft=36.78750.75P_{\text{shaft}} = \frac{36.7875}{0.75}Pshaft=0.7536.7875

Pshaft=49.05 kWP_{\text{shaft}} = 49.05 \text{ kW}Pshaft=49.05 kW

Thus, the shaft power required by the pump is **49.05 kW**.

Problem 2: Net Positive Suction Head (NPSH) Calculation

Question: A centrifugal pump is designed to operate with a flow rate of 250 liters per second (L/s) at a total head of 30 meters. The suction pipe is 10 meters in length, and the friction loss in the suction line is 2 meters. The atmospheric pressure is 101.3 kPa, and the vapour pressure of water at the pumping temperature is 3.5 kPa. The specific gravity of water is 1.

Calculate the Net Positive Suction Head Available (NPSHA) for this pump.

Solution:

The formula for NPSHA is:

NPSHA=Patmρg−Hvaporρg−Hf−Hsuction\text{NPSHA} = \frac{P_{\text{atm}}}{\rho g} - \frac{H_{\text{vapor}}}{\rho g} - H_{\text{f}} - H_{\text{suction}}NPSHA=ρgPatm−ρgHvapor−Hf −Hsuction

Where:

- PatmP_{\text{atm}}Patm = atmospheric pressure = 101.3 kPa = 101,300 Pa
- ρ\rhoρ = density of water = 1000 kg/m³
- ggg = gravitational acceleration = 9.81 m/s²
- HvaporH_{\text{vapor}}Hvapor = vapor pressure = 3.5 kPa = 3500 Pa
- HfH_{\text{f}}Hf = friction head loss = 2 m
- HsuctionH_{\text{suction}}Hsuction = suction head = 10 m (length of suction pipe)

Now calculate the pressure head contribution from atmospheric pressure and vapor pressure:

1. Atmospheric pressure head:

Patmρg=101,3001000×9.81=10.32 m\frac{P_{\text{atm}}}{\rho g} = \frac{101,300}{1000 \times 9.81} = 10.32 \text{ m}ρgPatm=1000×9.81101,300=10.32 m

2. Vapor pressure head:

Hvaporρg=35001000×9.81=0.356 m\frac{H_{\text{vapor}}}{\rho g} = \frac{3500}{1000 \times 9.81} = 0.356 \text{ m}ρgHvapor=1000×9.813500=0.356 m

Now substitute the values into the NPSHA formula:

NPSHA=10.32−0.356−2−10\text{NPSHA} = 10.32 - 0.356 - 2 - 10NPSHA=10.32−0.356−2−10
NPSHA=10.32−12.356\text{NPSHA} = 10.32 - 12.356NPSHA=10.32−12.356
NPSHA=−2.036 m\text{NPSHA} = -2.036 \text{ m}NPSHA=−2.036 m
Since the NPSHA is negative, this means that cavitation is likely to occur under these operating conditions, and the system may require adjustments, such as reducing friction losses or increasing the suction head.

Problem 3: Efficiency and Flow Rate Calculation

Question: A centrifugal pump operates at a flow rate of 100 L/s with a total head of 20 meters. The brake horsepower (BHP) of the pump is measured to be 20 kW, and the efficiency is given as 70%. Calculate the actual flow rate when the pump's efficiency is increased to 80%, assuming the same total head and brake horsepower.

Solution:

The formula for the pump efficiency is:
η=PhydraulicPshaft\eta = \frac{P_{\text{hydraulic}}}{P_{\text{shaft}}}η=PshaftPhydraulic
Where:

- PhydraulicP_{\text{hydraulic}}Phydraulic is the hydraulic power (use formula in previous problems)
- PshaftP_{\text{shaft}}Pshaft is the shaft power, which is related to the brake horsepower

Step 1: Calculate the original hydraulic power at 70% efficiency.

η=PhydraulicBHP\eta = \frac{P_{\text{hydraulic}}}{BHP}η=BHPPhydraulic
Substitute the given values:
0.7=Phydraulic20 kW0.7 = \frac{P_{\text{hydraulic}}}{20 \text{ kW}}0.7=20 kWPhydraulic
Thus,
Phydraulic=0.7×20=14 kWP_{\text{hydraulic}} = 0.7 \times 20 = 14 \text{ kW}Phydraulic =0.7×20=14 kW

Step 2: Calculate the new hydraulic power required for 80% efficiency.

Now that we know the hydraulic power required (14 kW), let's calculate the new required flow rate for 80% efficiency.
ηnew=0.8\eta_{\text{new}} = 0.8ηnew=0.8
Since the hydraulic power is directly proportional to the flow rate, we can use the ratio of efficiencies and power:
Phydraulic,newPhydraulic,old=ηnewηold\frac{P_{\text{hydraulic,new}}}{P_{\text{hydraulic,old}}} = \frac{\eta_{\text{new}}}{\eta_{\text{old}}}Phydraulic,oldPhydraulic,new=ηoldηnew
Phydraulic,new14=0.80.7\frac{P_{\text{hydraulic,new}}}{14} = \frac{0.8}{0.7}14Phydraulic,new

=0.70.8 Phydraulic,new=0.80.7×14=16 kWP_{\text{hydraulic,new}} = \frac{0.8}{0.7} \times 14 = 16 \text{ kW}Phydraulic,new=0.70.8×14=16 kW

Thus, the new hydraulic power required is 16 kW.

Step 3: Calculate the new flow rate.

Since hydraulic power is proportional to the flow rate, the new flow rate can be calculated as:

Qnew=Qold×Phydraulic,newPhydraulic,oldQ_{\text{new}} = Q_{\text{old}} \times \frac{P_{\text{hydraulic,new}}}{P_{\text{hydraulic,old}}}Qnew=Qold×Phydraulic,oldPhydraulic,new

Substitute the values:

Qnew=100 L/s×1614Q_{\text{new}} = 100 \, \text{L/s} \times \frac{16}{14}Qnew=100L/s×1416

Qnew=114.29 L/sQ_{\text{new}} = 114.29 \, \text{L/s}Qnew=114.29L/s

Thus, the new flow rate when the efficiency is increased to 80% is approximately **114.29 L/s**.

Problem 1

Determining the Discharge of a Reciprocating Pump

Question:

A single-acting reciprocating pump has a cylinder diameter of 150 mm and a stroke length of 400 mm. The pump operates at a speed of 40 strokes per minute. Calculate the discharge of the pump in liters per minute (LPM).

Solution:

To calculate the discharge, we use the following formula:

Q=A×L×NQ = A \times L \times NQ=A×L×N

Where:

- QQQ is the discharge (in cubic meters per minute),
- AAA is the cross-sectional area of the cylinder (in square meters),
- LLL is the stroke length (in meters),
- NNN is the number of strokes per minute.

Step 1: Calculate the cross-sectional area of the pump cylinder.

The area of a circle is given by:

A=π×r2A = \pi \times r^2A=π×r2

The radius rrr is half the diameter, so:

r=1502=75 mm=0.075 mr = \frac{150}{2} = 75 \, \text{mm} = 0.075 \, \text{m}r=2150 =75mm=0.075m

Now calculate the area:

A=π×(0.075)2=3.1416×0.005625=0.01767 m2A = \pi \times (0.075)^2 = 3.1416 \times 0.005625 = 0.01767 \, \text{m}^2A=π×(0.075)2=3.1416×0.005625=0.01767m2

Step 2: Use the stroke length and speed to find the discharge.

Convert the stroke length to meters:

L=400 mm=0.4 mL = 400 \, \text{mm} = 0.4 \, \text{m}L=400mm=0.4m

Now substitute the values into the discharge formula:

Q=0.01767×0.4×40=0.2827 m3/minQ = 0.01767 \times 0.4 \times 40 = 0.2827 \, \text{m}^3/\text{min}Q=0.01767×0.4×40=0.2827m3/min

Step 3: Convert the discharge to liters per minute (LPM).

Since 1 cubic meter = 1000 liters:

Q=0.2827 m3/min×1000=282.7 LPMQ = 0.2827 \, \text{m}^3/\text{min} \times 1000 = 282.7 \, \text{LPM}Q=0.2827m3/min×1000=282.7LPM

Answer: The discharge of the pump is 282.7 LPM.

Problem 2: Calculating the Power Required for a Reciprocating Pump

Question:

A double-acting reciprocating pump is pumping water from a well to a tank. The pump has a bore diameter of 250 mm and a stroke length of 500 mm. The pump operates at 30 strokes per minute. The total head is 20 meters, and the efficiency of the pump is 70%. Calculate the power required to drive the pump.

Solution:

To calculate the power required, we use the following formula:

P=ρ×g×H×QηP = \frac{\rho \times g \times H \times Q}{\eta}P=ηρ×g×H×Q

Where:

- PPP is the power (in watts),
- ρ\rhoρ is the density of water (1000 kg/m31000 \, \text{kg/m}^31000kg/m3),
- ggg is the acceleration due to gravity (9.81 m/s29.81 \, \text{m/s}^29.81m/s2),
- HHH is the total head (in meters),
- QQQ is the discharge (in cubic meters per second),
- η\etaη is the efficiency of the pump.

Step 1: Calculate the discharge.

We can use the formula for discharge for a double-acting pump:

Q=A×L×N×260Q = \frac{A \times L \times N \times 2}{60}Q=60A×L×N×2

Here, AAA is the area of the cylinder, LLL is the stroke length, and NNN is the number of strokes per minute.

First, calculate the area of the cylinder:

A=π×r2=3.1416×(0.125)2=0.049087 m2A = \pi \times r^2 = 3.1416 \times (0.125)^2 = 0.049087 \, \text{m}^2A=π×r2=3.1416×(0.125)2=0.049087m2

Now, calculate the discharge:

Q=0.049087×0.5×30×260=0.049087 m3/sQ = \frac{0.049087 \times 0.5 \times 30 \times 2}{60} = 0.049087 \, \text{m}^3/\text{s}Q=600.049087×0.5×30×2=0.049087m3/s

Step 2: Substitute the values into the power equation.

Now, substitute the known values into the power formula:

P=1000×9.81×20×0.0490870.7P = \frac{1000 \times 9.81 \times 20 \times 0.049087}{0.7}P=0.71000×9.81×20×0.049087 P=1000×9.81×20×0.0490870.7=1373.87 wattsP = \frac{1000 \times 9.81 \times 20 \times 0.049087}{0.7} = 1373.87 \, \text{watts}P=0.71000×9.81×20×0.049087=1373.87watts

Answer: The power required to drive the pump is approximately 1.37 kW.

Problem 3: Suction and Delivery Heads in a Reciprocating Pump

Question:

A reciprocating pump has a suction pipe diameter of 100 mm and a delivery pipe diameter of 75 mm. The pump is running at 25 strokes per minute. The suction head is 4 meters, and the delivery head is 10 meters. The pump efficiency is 80%. Calculate the net positive suction head (NPSH) available to the pump.

Solution:

The net positive suction head (NPSH) can be calculated using the following formula:

NPSH=Atmospheric Pressure Head−Vapour Pressure Head−Frictional Head Loss−Suction Head\text{NPSH} = \text{Atmospheric Pressure Head} - \text{Vapour Pressure Head} - \text{Frictional Head Loss} - \text{Suction Head}NPSH=Atmospheric Pressure Head−Vapour Pressure Head−Frictional Head Loss−Suction Head

However, since we are not given detailed information about atmospheric pressure, vapour pressure, and frictional head losses, we simplify it to the suction head available at the inlet to the pump. The NPSH available is essentially the suction head, which is 4 meters.

Answer: The NPSH available to the pump is 4 meters.

For reference:

Refer fluid mechanics and hydraulic machines by author " Er. K. RAJPUT "

From S Chand & company Ltd

Textbooks:

1. Y.A.Cengel, J.M.Cimbala, Fluid and Applications, 6/e, McGraw Hill Publications, 2019.

Mechanics, Fundamentals

2. Dixon, Fluid Mechanics and Thermodynamics of Turbomachinery, 7/e, Elsevier Publishers, 2014.

Reference Books:

1. PN Modi and S M Seth, Hydraulics & Fluid Mechanics including Hydraulics Machines, Standard Book House, 2017.
2. RKBansal, Fluid Mechanics and Hydraulic Machines, 10/e, Laxmi Publications(P) Ltd, 2019.
3. Rajput, Fluid Mechanics and Hydraulic Machines, S Chand & Company, 2016.
4. D.S.Kumar, Fluid Mechanics and Fluid Power Engineering, SK Kataria &Sons, 2013.

. D.Rama Durgaiah, Fluid Mechanics and Machinery, 1/e, New Age International, 2002. 5

Online Learning Resources:

https://archive.nptel.ac.in/courses/112/105/112105206/

https://archive.nptel.ac.in/courses/112/104/112104118/

https://www.edx.org/learn/fluid-mechanics

https://onlinecourses.nptel.ac.in/noc20 ce30/previewnptel.ac.in

www.coursera.org/learn/fluid-powerera