L-01

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – Definitions – Fluid and fluid mechanics – Dimensions and units

Introduction : (Maximum 5 sentences)

Fluid Mechanics is the study of fluids either in motion (fluid dynamics/kinematics) or at rest (fluid statics). Gases and liquids (e.g. air, water) come under the category of fluid. One of the areas of modern fluid mechanics is Computational Fluid Mechanics which deals with numerical solutions using computers.

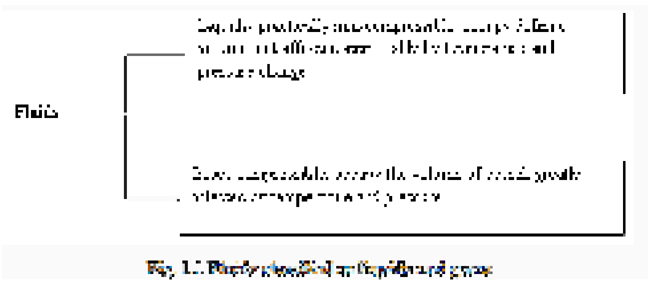
Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

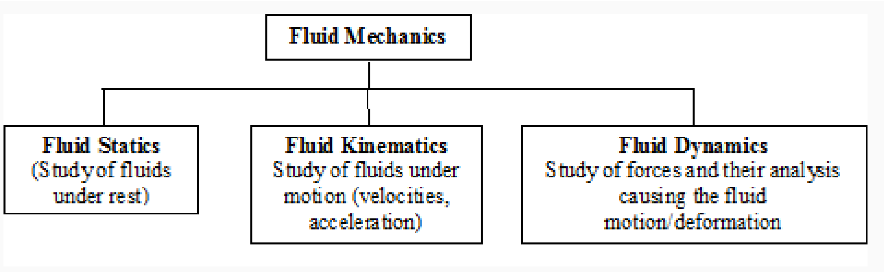
Introduction:

Fluid is a substance which has no definite shape and will continuously deform or flow whenever an external force is applied to it e.g. water, milk, steam, gas, etc. It cannot preserve its shape unless it is restricted into a particular form depending upon the shape of its surroundings.



Fluid Mechanics is the study of fluids either in motion (fluid dynamics/kinematics) or at rest (fluid statics). Gases and liquids (e.g. air, water) come under the category of fluid. One of the areas of modern fluid mechanics is Computational Fluid Mechanics which deals with numerical solutions using computers.

Fluid mechanics comprises of the following subjects:



Fluid mechanics is a branch of Engineering Science, the knowledge of which is needed in the design of:

- •Water supply and treatment system
- •Pumps used for handling of different fluids
- •Ships, submarines, aeroplanes, Automobiles
- •Storage tanks (milk silo, tankers, feed tanks, balance tanks etc.)
- •Piping systems for various utilities, pipefitting & valves, flow meters etc.
- •Measuring instrument
- •Cleaning-In-Place (CIP) systems for optimum performance
- •Heat transfer behaviour in processing equipments (such as HTST pasteurizers, spray dryers etc.)

Units and Dimensions

A unit of measurement is a definite magnitude of a physical quantity. The different systems of unit are:

1. **SI system:** It is the International System of Units (abbreviated SI from the French Le System International Units.
2. **CGS system:** It is a system of physical units based on centimetre as the unit of length, gram as a unit of mass, and second as a unit of time.
3. **MKS system:** It is a metric system of physical units based on meter as the unit of length, kilogram as a unit of mass, and second as a unit of time.
4. **FPS system:** The foot-pound-second system or FPS system is a system of units built on the three fundamental units foot for length, pound for either mass or force and second for time.

Factor	Prefix	Symbol
10^9	Giga	G
10^6	Mega	M
10^3	kilo	k
10^{-2}	centi	c
10^{-3}	milli	m
10^{-6}	micro	μ
10^{-9}	nano	n

Quantity	Dimensions (M LT)	Preferred units (SI)
Length (L)	L	m
Time (T)	T	s
Mass (M)	M	kg
Area (A)	L^2	m^2
Volume (Vol)	L^3	m^3
Velocity (V)	LT^{-1}	m/s
Acceleration (a)	LT^{-2}	m/s^2
Discharge (Q)	$L^3 T^{-1}$	m^3/s
Force (F)	MLT^{-2}	N
Pressure (p)	$ML^{-1}T^{-2}$	Pa
Shear stress (τ)	$ML^{-1}T^{-2}$	N/m^2
Density (ρ)	ML^{-3}	kg/m^3
Specific weight (ω)	$ML^{-2}T^{-2}$	N/m^3
Energy/Work/Heat (E)	ML^2T^{-2}	J
[illegible]	[illegible]	[illegible]

Some important units and conversions

Dyne = g cm/s^2

1 dyne = 10^{-5} N

1 pound = 0.453 kg

Pressure: 1 atm = 101.325 kPa, 1 bar = 10^5 Pa

1 m = 3.28 ft

1 m = 100 cm

1 feet = 30.5 cm

1 feet = 12 inch

1 inch = 2.54 cm

1 km = 0.621 miles

1 ha = 2.47 acre

1 acre = 4 046.85 m^2

1 litre = 0.264 gallon

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

L-02

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : I-FLUID PROPERTIES AND STATICS

Date of Lecture:

Topic of Lecture: – Fluid properties – density-specific weight, specific volume

Introduction : (Maximum 5 sentences)

Kinematics is the branch of science which deals with motion of particles without considering the forces causing the motion.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

Mass density (ρ): Mass of fluid per unit of its volume is called mass density. Density is a measurement that compares the amount of matter an object has to its volume. An object with much matter in a certain volume has high density. An object with little matter in the same amount of volume has a low density. Density is found by dividing the mass of an object by its volume.

$$\rho = \frac{mass}{volume}$$

Unit: kg/m^3

Dimension: ML^{-3}

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

The mass per unit volume of a substance, usually denoted as ρ . Typical values are:

• Water: 1000 kg/m3;

• Mercury: 13546 kg/m3;

• Air: 1.23 kg/m3;

• Paraffin: 800 kg/m3

Weight Density (ω): Weight of fluid per unit of its volume is called weight density. Weight density is weight of a substance per unit volume. Our definitions of mass and weight are made in such a way that weight in Kg is same as mass in Kg on earth surface in standard reference place.Weight density is weight of a substance per unit volume.

$$\frac{Weight\ of\ fluid}{Volume} = \rho . g$$

With increase in temperature volume increase and hence specific weight decreases. With increase in pressure volume decreases and hence specific weight increases.

Therefore specific weight = specific density * acceleration due to gravity

Specific Volume (v): Volume of substance per unit of its mass is called specific volume. Specific volume is defined as the number of cubic meters occupied by one kilogram of matter. It is the ratio of a material's volume to its mass, which is the same as the reciprocal of its density. In other words, specific volume is inversely proportional to density

$$v = \frac{1}{\rho} = \frac{volume}{mass}$$

Unit: m^3/kg

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

Specific volume is a property of materials, defined as the number of cubic meters occupied by one kilogram of a particular substance. The standard **unit** is the meter cubed per kilogram (m^3/kg or $m^3 \cdot kg^{-1}$). In this case, the **unit** is the centimeter cubed per gram (cm^3/g or $cm^3 \cdot g^{-1}$). The specific volume of a substance is defined as the volume per unit mass. Density is defined as mass per unit volume. The reciprocal of density is the specific volume v, which is defined as volume per unit mass. That is, $v = V/m = 1/\rho$.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-03

LECTURE HANDOUTS

CIVIL | **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – Specific gravity, temperature, viscosity

Introduction : (Maximum 5 sentences)

Kinematics is the branch of science which deals with motion of particles without considering the forces causing the motion.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

Specific Gravity: Ratio of density of a substance to the density of pure water at 4^0C is called specific gravity.

$$\text{Specific gravity} = \frac{\text{Density of substance}}{\text{Density of water at } 4°\text{C}} \quad (\text{Dimension } M°L°T°)$$

Viscosity:

Viscosity is the property by virtue of which fluid offers resistance against the flow or shear deformation. In other words, it is the reluctance of the fluid to flow. Viscous force is that force of resistance offered by a layer of fluid for the motion of another layer over it.

In case of liquids, viscosity is due to cohesive force between the molecules of adjacent layers of liquid. In case of gases, molecular activity between adjacent layers is the cause of viscosity.

Newton's Law of Viscosity:

Consider a fluid contained between two parallel plates as shown in the Fig. Plate AD is the stationary plate where as BC is the moving plate and distance between the plates is y units. Initially BC is at rest. The area of the plate is A. Suppose a shear force is applied to top plate at point B. By shear force we mean a force that is applied tangentially and parallel to a surface. It can be seen in figure 1.5. The upper plate starts moving and attains a velocity say u m/s. Now the position changes from ABCD to AB'C'D as shown in figure 1.6.

Then the next layer starts moving and so on. It can be seen in Fig. 1.7.The distribution of fluid velocity from the top plate to the bottom is known as velocity gradient or velocity profile and is given as:

$$Velocity\ gradient = \frac{du}{dy}$$

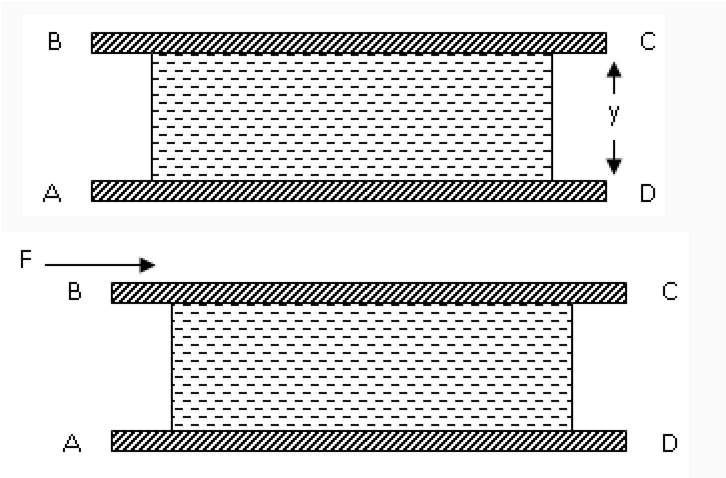


Fig. 1.5 Shear force is applied on the upper plate

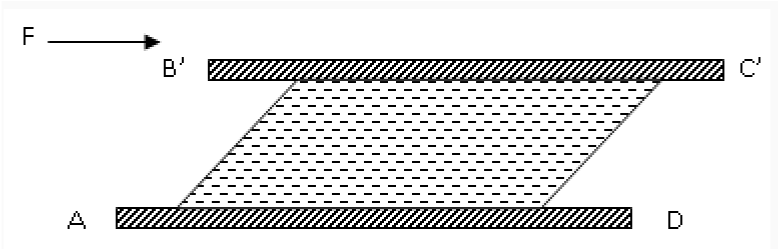


Fig. 1.6 Upper plate starts moving

Shear stress τ = (shear force)/Area

τ is pronounced as Tau and is the symbol of shear stress.

Note: shear stress is similar to pressure but here shear force is involved

Here, shear stress is proportional to velocity gradient:

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

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

Here, μ is known as coefficient of viscosity or dynamic viscosity. The SI unit of dynamic viscosity is Ns/m^2.

CGS units of dynamic viscosity is poise:

1 poise = 0.1 Ns/m^2

1 Centi poise (CP) = 0.01 poise

Kinematic Viscosity:

$$Kinematic\ viscosity = \frac{Dynamic\ viscosity}{density\ of\ substance}$$

SI Units: m^2/s

CGS units = Stoke

Temperature:

It is the measure of hotness and coldness of a system. In thermodynamic sense, it is the measure of internal energy of a system. Many a times, the temperature is expressed in centigrade scale (°C) where the freezing and boiling point of water is taken as 0°C and 100°C, respectively. In SI system, the temperature is expressed in terms of absolute value in Kelvin scale (K = °C+ 273).

At an absolute pressure of 1 standard atmosphere (1 atm or 101.325 kPa), for example, the saturation temperature of water is 100°C. Conversely, at a temperature of 100°C, the saturation pressure of water is 1 atm. The energy that is caused by increased temperature makes the molecules move at a faster rate to a level where they overcome the bonds or the binding forces of the molecules. This makes the liquid more fluid decreasing its viscosity.

- **Effect of Pressure on Viscosity of fluids:**

  Pressure has very little or no effect on the viscosity of fluids.

- **Effect of Temperature on Viscosity of fluids:**

  1. *Effect of temperature on viscosity of liquids:* Viscosity of liquids is due to cohesive force between the molecules of adjacent layers. As the temperature increases cohesive force decreases and hence viscosity decreases.
  2. *Effect of temperature on viscosity of gases:* Viscosity of gases is due to molecular activity between adjacent layers. As the temperature increases molecular activity increases and hence viscosity increases.

- **Kinematics Viscosity:** It is the ratio of dynamic viscosity of the fluid to its mass density.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-04

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – compressibility, vapour pressure,

Introduction : (Maximum 5 sentences)

Compressibility is the reciprocal of the bulk modulus of elasticity, K which is defined as the ratio of compressive stress to the volumetric strain.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

Compressibility and bulk modulus:

Compressibility is the reciprocal of the bulk modulus of elasticity, K which is defined as the ratio of compressive stress to the volumetric strain.

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

Bulk modulus (K) = (change in pressure) / (volumetric strain)

Volumetric strain is the change in volume divided by the original volume.

Therefore, (Change in volume) / (original volume) = (change in pressure) / (bulk modulus)

i.e., $-\delta V/V = \delta p/K$

Negative sign for -V indicates the volume decreases as pressure increases.

Bulk Modulus:

The relative change in the volume of a body produced by a unit compressive or tensile stress acting uniformly over its surface. The elasticity is often called the compressibility of the fluid. The bulk modulus of elasticity of water is approximately 2.2 GN/m2, which corresponds to a 0.05% change in volume for a change of 1 MN/m2 in pressure. For most purposes a liquid may be considered as incompressible.

$$\text{Compressibility} = \frac{1}{\text{Bulk Modulus of Elasticity (K)}}$$

$$\text{Bulk Modulus of Elasticity (K)} = \frac{dp}{-dV/V} = \frac{\text{Change in pressure}}{\text{Volumetric Strain}}$$

Vapour pressure:

When vapourization takes place, the molecules escapes from the free surface of the liquid . these vapour molecules get accumulated in the space between 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

Cavitation:

The cavitation 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 higher pressure .

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-05

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – No Prerequisite knowledge required

Introduction : (Maximum 5 sentences)

Kinematics is the branch of science which deals with motion of particles without considering the forces causing the motion.

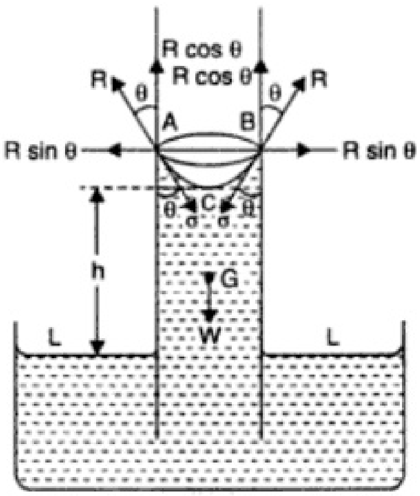
Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ Mechanics of Fluids

Detailed content of the Lecture:

Capillarity:

Capillary action (sometimes capillarity, capillary motion, capillary effect, or wicking) is the ability of a liquid to flow in narrow spaces without the assistance of, or even in opposition to, external forces like gravity. It occurs because of intermolecular forces between the liquid and surrounding solid surfaces.



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

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

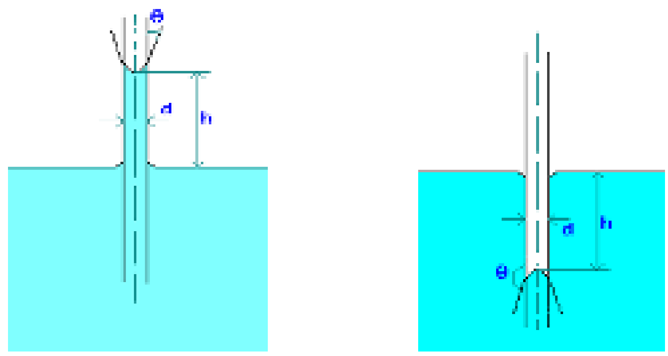
Capillary action is the movement of a liquid through or along another material against an opposing force, such as gravity. Examples of capillary action in water include water moving up a straw or glass tube, moving through a paper or cloth towel, moving through a plant, and tears moving through tear ducts.

Capillary Rise:

A rise in a liquid above the level of zero pressure due to a net upward force produced by the attraction of the water molecules to a solid surface, e.g. glass, soil (for those cases where the adhesion of the liquid to the solid is greater than the cohesion of the liquid to itself).

Capillary Fall:

The rise or fall of a fluid in a capillary tube is governed by the balance of cohesive and adhesive forces. Intermolecular forces are responsible for cohesion and adhesion. The narrower the bore of a glass tube, the greater the extent of raising or lowering of the liquid.



Applications Of Capillarity :

(a) The oil in the wick of a lamp rises due to the capillary action of threads in the wick.

(b) The action of a towel in soaking up moisture from the body is due to the capillary action of cotton in the towel.

(c) Water is retained in a piece of sponge on account of capillarity.

(d) A blotting paper soaks ink by the capillary action of the pores in the blotting paper.

(e) The root-hairs of plants draw water from the soil through capillary action.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=zMzqiAuOSz0
https://www.youtube.com/watch?v=WsksFbFZeeU

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-06

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – No Prerequisite knowledge required

Introduction : (Maximum 5 sentences)

Surface tension is defined as the tensile force acting on the surface of a liquid in contact with a gas or on the surface between two immiscible liquids such that the contact surface behaves like a membrane under tension.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

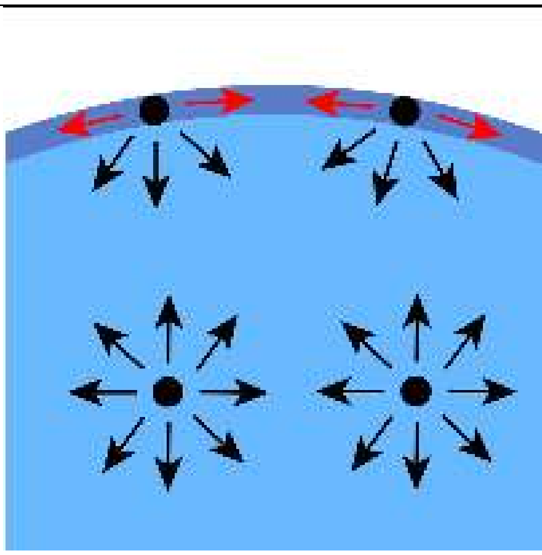
- ✓ Mechanics of Fluids

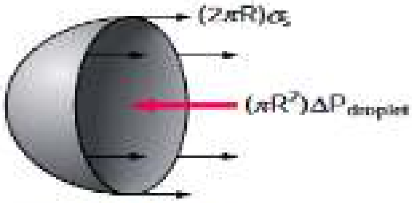
Detailed content of the Lecture:

Surface tension:

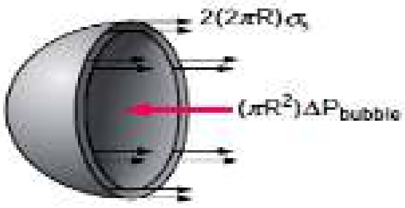
Surface tension is defined as the tensile force acting on the surface of a liquid in contact with a gas or on the surface between two immiscible liquids such that the contact surface behaves like a membrane under tension.

- Surface tension is created due to the unbalanced cohesive forces acting on the liquid molecules at the fluid surface.
- Molecules in the interior of the fluid mass are surrounded by molecules that are attracted to each other equally.
- However, molecules along the surface are subjected to a net force toward the interior.
- The apparent physical consequence of this unbalanced force along the surface is to create the hypothetical skin or membrane.
- A tensile force may be considered to be acting in the plane of the surface along any line in the surface.
- The intensity of the molecular attraction per unit length along any line in the surface is called the surface tension.It is denoted by Greek letter σ(called sigma).
- The SI unit is N/m.





(a) Half a droplet



(b) Half a bubble

Due to the cohesive forces a molecule is pulled equally in every direction by neighboring liquid molecules, resulting in a net force of zero. The molecules at the surface do not have the same molecules on all sides of them and therefore are pulled inward. This creates some internal pressure and forces liquid surfaces to contract to the minimum area.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=zMzqiAuOSz0
https://www.youtube.com/watch?v=WsksFbFZeeU

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
1. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-07

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – Fluid statics: concept of fluid pressure, Absolute and Gauge pressures

Introduction : (Maximum 5 sentences)

Fluid statics or hydrostatics is the branch of fluid mechanics that studies "fluids at rest and the pressure in a fluid or exerted by a fluid on an immersed body".

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

Fluid statics:

Fluid statics or hydrostatics is the branch of fluid mechanics that studies "fluids at rest and the pressure in a fluid or exerted by a fluid on an immersed body". It encompasses the study of the conditions under which fluids are at rest in stable equilibrium as opposed to fluid dynamics

Static fluid pressure:

The pressure exerted by a static fluid depends only upon the depth of the fluid, the density of the fluid, and the acceleration of gravity. The pressure in a static fluid arises from the weight of the fluid and is given by the expression. Pstatic fluid = ρgh where. ρ = m/V = fluid density.

Difference between fluid statics and fluid dynamics:

Fluid Statics and Fluid Dynamics form the two constituents of Fluid Mechanics. Fluid Statics deals with fluids at rest while Fluid Dynamics studies fluids in motion.

Importance:

Fluid statics or hydrostatics is the branch of fluid mechanics that studies "fluids at rest and the pressure in a fluid or exerted by a fluid on an immersed body". It encompasses the study of the conditions under which fluids are at rest in stable equilibrium as opposed to fluid dynamics, the study of fluids in motion.

Fluid Elements - Definition:

Fluid element can be defined as an infinitesimal region of the fluid continuum in isolation from its surroundings.

Two types of forces exist on fluid elements

- **Body Force**: distributed over the entire mass or volume of the element. It is usually expressed per unit mass of the element or medium upon which the forces act.
  Example: Gravitational Force, Electromagnetic force fields etc.
- **Surface Force**: Forces exerted on the fluid element by its surroundings through direct contact at the surface.
  Surface force has two components:
  - Normal Force: along the normal to the area
  - Shear Force: along the plane of the area.
- The ratios of these forces and the elemental area in the limit of the area tending to zero are called the normal and shear stresses respectively.
- The shear force is zero for any fluid element at rest and hence the only surface force on a fluid element is the normal component.

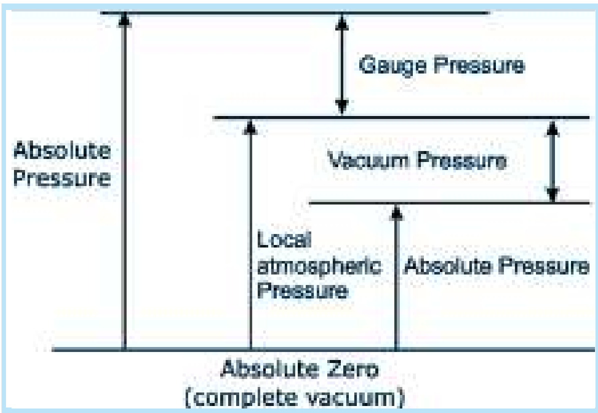


Fig 4.1 The Scale of Pressure

At sea-level, the international standard atmosphere has been chosen as P_{atm} = 101.32 kN/m²

Units and scales of Pressure Measurement

Pressure is usually expressed with reference to either absolute zero pressure (a complete vacuum)or local atmospheric pressure. Pascal (N/m2) is the unit of pressure .

- The absolute pressure: It is the difference between the value of the pressure and the absolute zero pressure. $p_{abs} = p - 0 = p$

- Gauge pressure: It is the diference between the value of the pressure and the local atmospheric pressure(patm)

$$p_{gauge} = p - p_{atm}$$

- Vacuum Pressure: If p<patm then the gauge pressure (p_{gauge}) becomes negative and is called the vacuum pressure.But one should always remember that hydrostatic pressure is always compressive in nature.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=zMzqiAuOSz0
https://www.youtube.com/watch?v=WsksFbFZeeU

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
1. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-08

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – pressure measurements by manometers and pressure gauges.

Introduction : (Maximum 5 sentences)

The most common form of this manometer is the conventional mercury barometer used to measure atmospheric pressure.

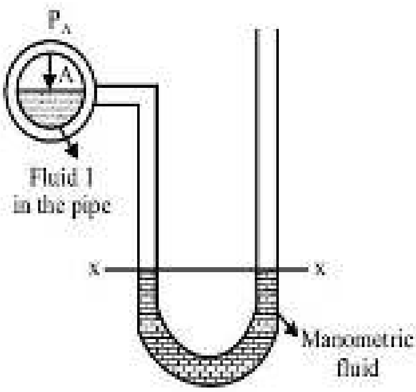
Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

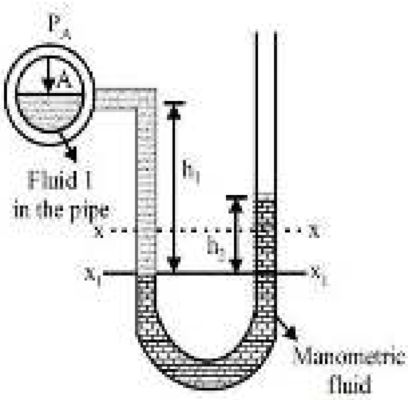
- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

- ➢ Manometer
  - U tube manometer
  - Single column/micro manometer
  - Well type manometer
  - Inclined tube manometer
  - Ring type manometer
- ➢ Pressure gauge
  - Bourdon pressure gauge
  - Bellow pressure gauge
  - Bellow differential pressure gauge

U-tube manometers:





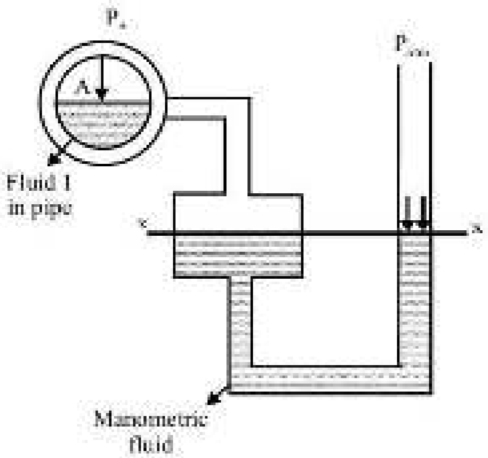
Just before connecting the U tube **Just after connecting the U tube**

- U-tube manometer is a simple manometric device used to measure pressure at a point in a fluid.
- balancing the fluid column by the same or another column of fluid.
- in liquid column used manometric liquids are mercury, water or alcohol.

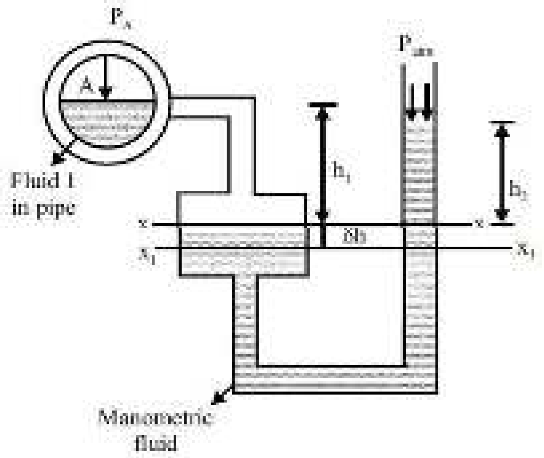
Some of the important and desirable properties of the manometric liquids are:

- High chemical stability
- Low viscosity
- Low coefficient of thermal expansion
- Low volatility
- Low vapour pressure

Single Column/Micro-manometer:



Before connecting micro-manometer

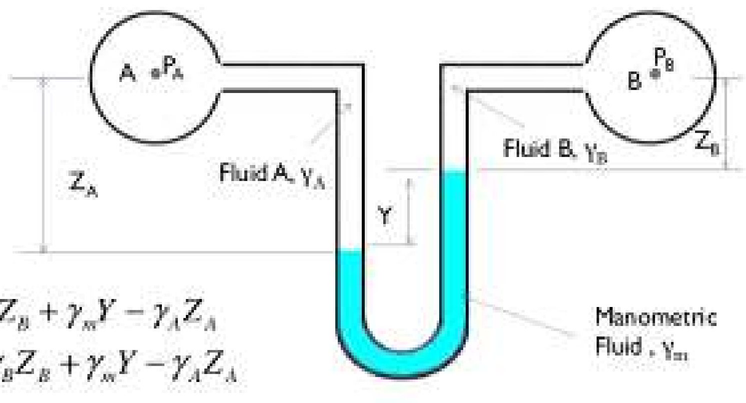


After connecting micro-manometer

- Single column/micro-manometers are just similar to the U-tube manometer, only with a difference of having a small reservoir in the tube.
- Micro-manometer has a glass tube bent in "U" shape or inclined and has a small reservoir in the tube, having some amount of same or other type of fluid called manometric fluid.

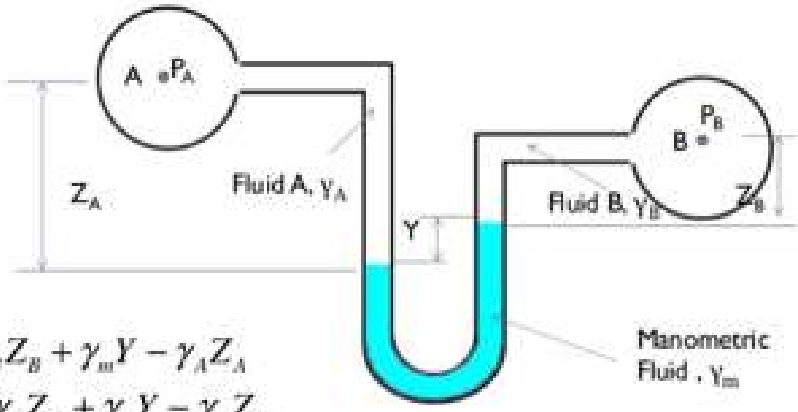
Differential Manometer:

- It is used to measure difference of pressure.
- **Case I: when two vessels/pipes are at same level**



$$P_A = P_B + \gamma_B Z_B + \gamma_m Y - \gamma_A Z_A$$
$$P_A - P_B = +\gamma_B Z_B + \gamma_m Y - \gamma_A Z_A$$

- **Case II: when two vessels/pipes are at different level**



$$P_A = P_B + \gamma_B Z_B + \gamma_m Y - \gamma_A Z_A$$
$$P_A - P_B = +\gamma_B Z_B + \gamma_m Y - \gamma_A Z_A$$

Advantages and Limitation of Manometers

- **Advantages**
- Easy to fabricate
- Less expansive
- Good accuracy
- High sensitivity
- Require little maintenance
- Not affected by vibration
- Specially suitable for low pressure and low differential pressure
- Easy to change sensitivity by changing manometric fluid

- **Limitations**
- Usually bulky and large in size
- Being fragile, get broken easily
- Reading of manometer is get affected by temperature, altitude and gravity
- Capillary action is created due to surface action
- Meniscus has to be measured accurately for better accuracy.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=zMzqiAuOSz0
https://www.youtube.com/watch?v=WsksFbFZeeU

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
1. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-09

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **I-FLUID PROPERTIES AND STATICS**

Date of Lecture:

Topic of Lecture: – pressure measurements by manometers and pressure gauges.

Introduction : (Maximum 5 sentences)

The most common form of this manometer is the conventional mercury barometer used to measure atmospheric pressure.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

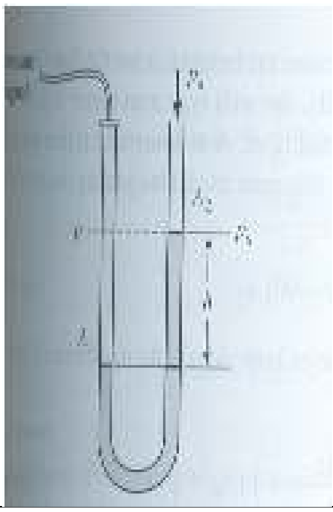
Types of Pressure Transducers:

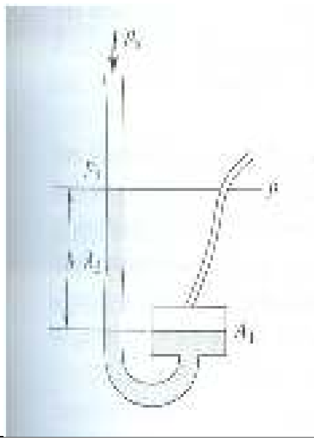
There are many types of pressure transducers Common used :

1) Mechanical Pressure-Measurement Devices (Manometer)
2) Dead-Weight Tester
3) Bourdon-Tube Pressure Gauge
4) Diaphragm and Bellows Gauges

Mechanical Pressure Measurement Devices (Manometer):

Offer simplest means for pressure measurement The fluid manometer is a widely used device for measurement of fluid pressures under steady-state and laboratory conditions.



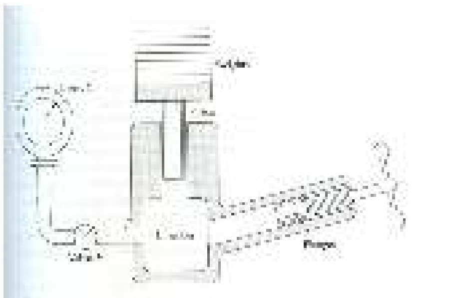


Dead-Weight Tester:

A device used for balancing a fluid pressure with a known weight Typically used for static calibration of pressure gauge and seldom employed for actual measurement.

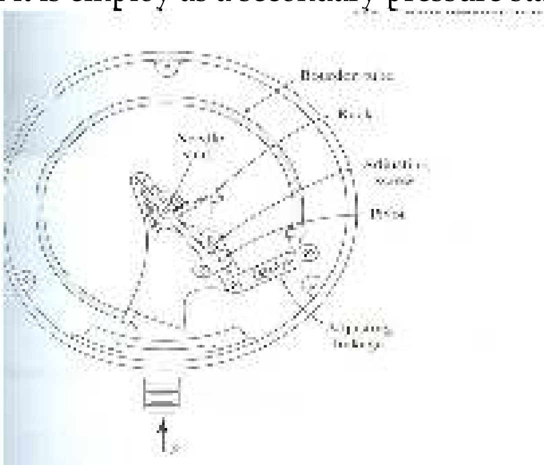
Architecture and operation:

- The apparatus set-up for calibration of the pressure gauge G.
- The chamber and cylinderof the tester are filled with a clean oil by first moving the plunger to its most forward position and then slowly withdrawing it while the oil is poured in through the opening for the piston.



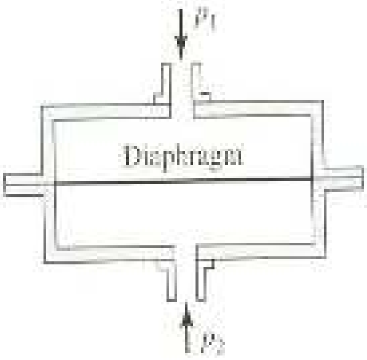
Bourdon-Tube Pressure:

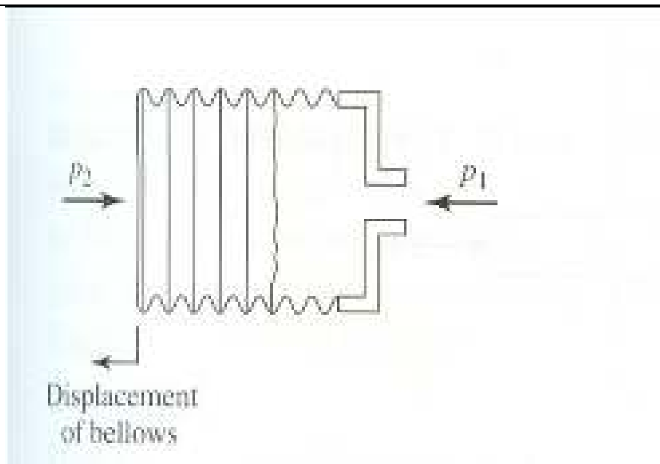
- Used in many applications because of its consistency, And inexpensive measurement of static pressure measurement.
- Commercially available in many sizes (1-to-16 in diameter) and accuracies.
- The heise gauge is an extremely accurate bourdon-tube gauge with an accuracy of 0.1% of full-scale reading and it is employ as a secondary pressure standard in laboratory work.



Diaphragm and Bellows Gauges:

- Represent similar types of elastic deformation devices useful for pressure measurement applications.
- Architecture and operation: Diaphragm gauge:
- Consider first the flat diaphragm subjected to the differential pressure p1-p2.The diaphragm will be deflected in accordance with this pressure differential and the deflection sensed an appropriate displacement transducer.





Diaphragm **Bellows**

Bellows Gauge:

- A differential gauge pressure force causes displacement of the bellows, which may be converted to an electrical signal or undergo a mechanical amplification to Permit display of the output on an indicator dial.
- Various types of bellows gauges.
  - The bellows gauge is generally unsuitable for transient measurements because of the larger relative motion and mass involved.
  - The diaphragm gauge which may be quite stiff involves rather small displacements and is suit for high frequency pressure measurement.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=zMzqiAuOSz0
https://www.youtube.com/watch?v=WsksFbFZeeU

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page- 1-68
1. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No 265-356

Course Teacher

Verified by HOD

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **II-FLUID KINEMATICS AND DYNAMICS**

Date of Lecture:

Topic of Lecture: – Fluid Kinematics Stream, streak and path lines

Introduction : (Maximum 5 sentences)

Kinematics is the branch of science which deals with motion of particles without considering the forces causing the motion..

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

KINEMATICS OF FLOW:

Kinematics is the branch of science which deals with motion of particles without considering the forces causing the motion.

The fluid motion is described by two methods.

(i) Lagrangian Method.

- A single fluid particle is followed during its motion and its velocity, acceleration, density, etc., are described.

(ii) Eulerian Method.

- The velocity, acceleration, density, etc., are described at a point in flow field.

STREAM LINES:

If curves are drawn in a steady flow in such a way that the tangent at any point is in the direction of the velocity vector at that point, such curves are called streamlines. Individual fluid particles must travel on paths whose tangent is always in the direction of the fluid velocity at any point. Thus, path lines are the same as streamlines in steady flows.

A streamline is a path traced out by a massless particle as it moves with the flow. It is easiest to visualize a streamline if we move along with the body (as opposed to moving with the flow).Since the streamline is traced out by a moving particle, at every point along the path the velocity is tangent to the path.

Streamlines, streaklines and pathlines are field **lines** in a fluid flow. They differ only when the flow changes with time, that is, when the flow is not steady. Dye steadily injected into the fluid at a fixed point extends along a **streakline**. Pathlines are the trajectories that individual fluid particles follow.

STREAKLINES:

Streaklines are the loci of points of all the fluid particles that have passed continuously through a particular spatial point in the past. Dye steadily injected into the fluid at a fixed point extends along a streakline.

PATH LINES:

Path lines are the trajectories that individual fluid particles follow. These can be thought of as "recording" the path of a fluid element in the flow over a certain period. The direction the path takes will be determined by the streamlines of the fluid at each moment in time.

Timelines are the lines formed by a set of fluid particles that were marked at a previous instant in time, creating a line or a curve that is displaced in time as the particles move.

The following observations can be made about the fundamental line patterns;

1. Mathematically, it is convenient to calculate a streamline while other three are easier to generate experimentally.
2. The streamlines and timelines are instantaneous lines while pathlines and streakline are generated by passage of time.
3. In a steady flow, all the four basic line patterns are identical. Since, the velocity at each point in the flow field remains constant with time, consequently streamline shapes do not vary. It implies that the particle located on a given streamline will always move along the same streamline. Further, the consecutive particles passing through a fixed point in space will be on the same streamline. Hence, all the lines are identical in a steady flow. They do not coincide for unsteady flows.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

L-11

LECTURE HANDOUTS

CIVIL | **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : II-FLUID KINEMATICS AND DYNAMICS

Date of Lecture:

Topic of Lecture: – Classification of flows

Introduction : (Maximum 5 sentences)

The classification of the fluid flow based on the variation of the fluid flow parameters with time characterizes the flow in two categories, steady and unsteady flow. If the flow parameters, such as velocity, pressure, density and discharge do not vary with time or are independent of time then the flow is steady.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

CLASSIFICATION OF FLOW:

- Steady flow.
- Unsteady Flow.
- Non-Uniform flow.
- Rotational flow.
- Irrotational flow.
- Viscous flow.
- Incompressible Flow.
- Compressible flow

Steady flow:

A steady flow is the one in which the quantity of liquid flowing per second through any section, is constant. This is the definition for the ideal case. True steady flow is present only in Laminar flow. In turbulent flow, there are continual fluctuations in velocity. Pressure also fluctuate at every point.

Unsteady Flow:

Unsteady-State (Transient) Flow. The unsteady-state flow (frequently called transient flow) is defined as the fluid flowing condition at which the rate of change of pressure with respect to time at any position in the reservoir is not zero or constant.

Non-Uniform flow:

Non-uniform flow. Flow is said to be non-uniform, when there is a change in velocity of the flow at different points in a flowing fluid, for a given time. For example, the flow of liquids under pressure through long pipelines of varying diameter is referred to as non-uniform flow.

Rotational flow:

Rotational flow is when the particles of fluids are all rotating about their own axis in addition to their other movement. Irrotational flow is when the individual particles are not rotating around their axis.

Irrotational flow:

Irrotational flow is a flow in which each element of the moving fluid undergoes no net rotation with respect to a chosen coordinate axes from one instant to other.

Viscous flow:

A type of fluid flow in which there is a continuous steady motion of the particles; the motion at a fixed point always remains constant. Also called streamline flow; laminar flow; steady flow.

Incompressible flow:

Incompressible flow is type of flow in which density of fluid remains constant. . Compressible Flow.

Compressible flow:

Compressible flow is flow in which density of fluid changes with respect to distance.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

L-12

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : II-FLUID KINEMATICS AND DYNAMICS

Date of Lecture:

Topic of Lecture: – Continuity equation (one, two and three dimensional forms)

Introduction : (Maximum 5 sentences)

The continuity equation is simply a mathematical expression of the principle of conservation of mass. For a control volume that has a single inlet and a single outlet, the principle of conservation of mass states that, for steady-state flow, the mass flow rate into the volume must equal the mass flow rate out.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

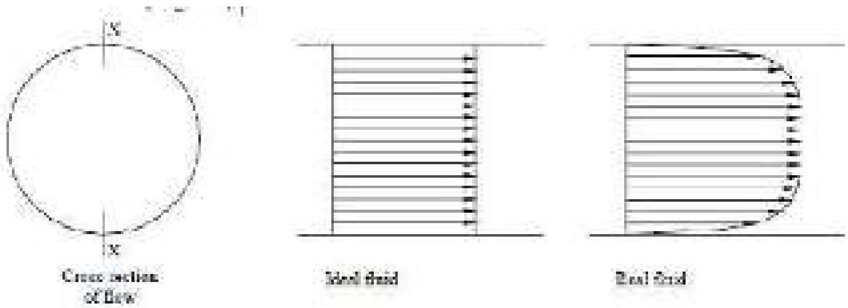
CONTINUITY EQUATION (ONE, TWO AND THREE DIMENSIONAL FORMS):

Derivation of continuity equation is one of the most important derivations in fluid dynamics. The continuity equation is defined as the product of cross sectional area of the pipe and the velocity of the fluid at any given point along the pipe is constant.

Although, in general, all fluid flow occurs in three dimensions, so that, velocity, pressure and other factors vary with reference to three orthogonal axes, in some problems the major changes occur in two directions or even in only one direction. Changes along the other axis or axes can, in such cases, be ignored without introducing major errors, thus simplifying the analysis.

Flow is described as one-dimensional if the factors, or parameters, such as velocity,pressure and elevation, describing the flow at a given instant, vary only along the direction of flow and not across the cross-section at any point.

If the flow is unsteady, these parameters may vary with time. The one dimension is taken as the distance along the streamline of the flow, even though this may be a curve in space, and the values of velocity, pressure and elevation at each point along this streamline will be the average values across a section normal to the streamline (Fig.1).



In two-dimensional flow it is assumed that the flow parameters may vary in the direction of flow and in one direction at right angles, so that the streamlines are curves lying in a plane and identical in all planes parallel to this plane.

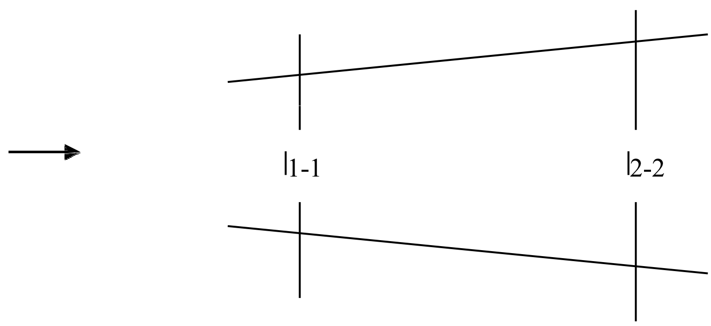
Thus, the flow over a weir of constant cross-section (Fig.2) and infinite width perpendicular to the plane of the diagram can be treated as two-dimensional. In three-dimensional flow it is assumed that the flow parameters may vary in space, x in the direction of motion, y and z in the plane of the cross-section.

RATE OF FLOW OR DISCHARGE (Q):

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

CONTINUITY EQUATION:

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



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

ρ_1 =Density at section 1-1

A_1=Area of pipe at section 1-1

And V_2, ρ_2, A_2 are corresponding values at section 2-2

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

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

According to law of conservation of mass

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

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 \text{ ....(1)}$$

The above equation is applicable to the compressible as well as incompressible fluids is called Continuity Equation. If the fluid is incompressible, then $\rho_1 = \rho_2$ and continuity equation (1) reduces to

$$A_1 V_1 = A_2 V_2$$

The diameters of a pipe at the sections 1 and 2 are 10cm and 15cm respectively. Find the discharge through the pipe if the velocity of water flowing through the pipe at section 1 is 5m/s. Determine the velocity at section 2.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

L-13

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : II-FLUID KINEMATICS AND DYNAMICS

Date of Lecture:

Topic of Lecture: – Stream and potential functions

Introduction : (Maximum 5 sentences)

If stream function (ψ) exists, it is possible case of fluid flow which may be rotational or irrotational.A scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

STREAM FUNCTION:

It is defined as the scalar function of space and time, such that its partial derivative with respect to any direction gives the velocity component at right angles to that direction. It is denoted by ψ (Psi) and only for two dimensional flow.Mathematically. For steady flow is defined as ψ = f(x,y) such that ,

$$(\partial \psi / \partial x) = v$$

$$(\partial \psi / \partial y) = - u.$$

Properties of Stream Function:

The properties of stream function (ψ) are:

1. If stream function (ψ) exists, it is possible case of fluid flow which may be rotational or irrotational.
2. If stream function (ψ) satisfies the Laplace equation, it is a possible case of irrotational flow

VELOCITY POTENTIAL FUNCTION:

It is defined as a scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction. It is defined by Φ (Phi). Mathematically, the velocity, potential is defined as Φ = f (x,y,z) for steady flow

Such that.

$u = -(\partial \Phi/\partial x)$

$v = -(\partial \Phi/\partial y)$

$w = -(\partial \Phi/\partial z)$

where, u,v and w are the components of velocity in x.y and z directions respectively.

Local acceleration:

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

Convective acceleration:

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

Equipotential line :

A line along which the velocity potential $\emptyset$ is constant, is called equipotential line.

For equipotential line., $d\emptyset = 0$

$udx + vdy = 0$

$dy/dx = -u/v$

dy/dx = slope of equipotential line.

For a line of constant stream function., $d\psi = 0$

$dy/dx = u/v$

Relationship between stream function & velocity potential function :

$$\partial\phi/\partial x = \partial\psi/\partial y$$

$$\partial\phi/\partial y = -\partial\psi/\partial x$$

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-14

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **II-FLUID KINEMATICS AND DYNAMICS**

Date of Lecture:

Topic of Lecture: – Flow nets,Fluid dynamics

Introduction : (Maximum 5 sentences)

A grid obtained by drawing a series of equipotential lines and stream lines is called a flow net.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

FLOW NET:

A grid obtained by drawing a series of equipotential lines and stream lines is called a flow net.

<u>Types of Motion :</u>

A fluid particle while moving may undergo anyone or combination of following four types of displacements :

i) Linear translation: - movement of fluid elements in such a way that it moves bodily from one position to another position .

ii) Linear deformation: - deformation of a fluid element in linear direction when the element moves.

iii) Angular deformation: - the average change in the angle contained by two adjacent sides.

iv) Rotation: - movement of fluid element in such away that both of its axes rotates in the same direction .

Vorticity :

It is defined as the value twice of the rotation .

Vortex Flow:

It is defined as the flow of a fluid along a curved path or the flow of a rotating mass of fluid.

Types :

i) Forced vortex flow .- It is a type of vortex flow, in which some external torque is required to rotate the fluid mass.

ii) Free vortex flow.- when external torque is required to rotate the fluid mass, then the type of flow is called free vortex flow.

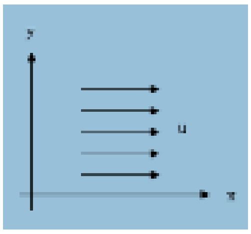
Potential / Ideal Flow:

Ideal fluid is a fluid which is incompressible and inviscid. Incompressible fluid is a fluid for which the density(ρ) remains constant. Inviscid fluid is a fluid for which viscosity(μ) is zero. Hence a fluid for which the density is constant and viscosity is zero, is known as an ideal fluid.

Important Cases Of Potential Flow

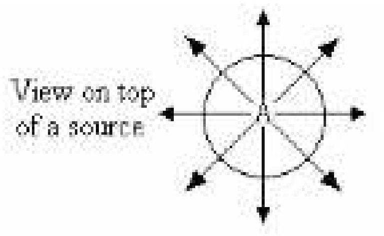
i) Uniform flow: The uniform flow may be.,

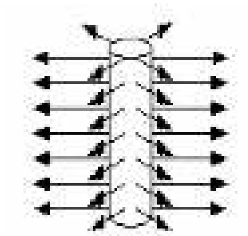
a) Parallel to x_axis .- in uniform flow, the velocity remains constant. All fluid particles are moving with the same velocity.



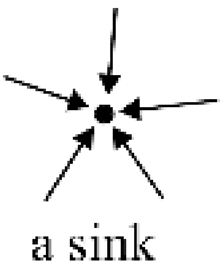
b) Parallel to y_axis.- the uniform potential flow parallel to y_axis in which the uniform velocity is along y_axis.

ii) Source flow. - The flow coming from a point and moving out radially in all directions of a plane at a uniform rate.

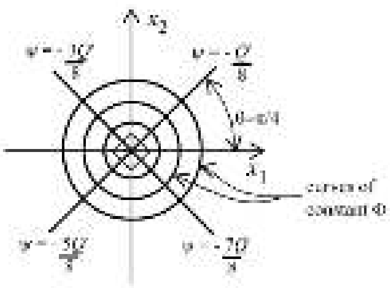




iii) Sink flow.- the flow in which fluid moves radially inwards towards a point where it disappears at a constant rate.



iv) Free-vortex flow.- It is a circulatory flow of a fluid such that its stream lines are concenteric circles.



v) Superimposed flow: the flow patterns due to uniform flow,a source flow, a sink flow and a free vortex flow can be super_imposed in any linear combination to get a resultant flow which closely resembles the flow around bodies. The resultant flow will still be potential and ideal. There are some important super_imposed flow below.

Fluid Dynamics:

Dynamic of fluid flow is the study of fluid motion with the forces causing flow. The dynamic behavior of the fluid is analysed by the Newton's second law of motion, which relates the accelaration with the forces.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

LECTURE HANDOUTS | **L-15**

CIVIL | **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **II-FLUID KINEMATICS AND DYNAMICS**

Date of Lecture:

Topic of Lecture: – Equation of motion-Euler's equation along a streamline

Introduction : (Maximum 5 sentences)

The Euler's equation for a steady flow of an ideal fluid along a streamline is a relation between the velocity, pressure, and density of a moving fluid. It is based on Newton's Second Law of Motion.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

EQUATIONS OF MOTION:

The Euler's equation for a steady flow of an ideal fluid along a streamline is a relation between the velocity, pressure, and density of a moving fluid. It is based on Newton's Second Law of Motion.The flow is continuous, steady and along the streamline. The velocity of the flow is uniform over the section.

According to Newton's second law of motion, the net force acting on a fluid element is equal to mass of the fluid element multiplied by the acceleration .

$$Fx = m.ax.$$

In the fluid flow the following forces are present.,

1) Fx,*gravity force.*
2) Fp,*pressure force.*
3) Fv,*force due to viscosity.*
4) Ft,*force due to turbulent.*
5) Fc ,*force due to compressibility.*

$$Fx=(Fg)x+(Fp)x+(Fv)x+(Ft)x+(Fc)x$$

Reynold's Equation of Motion:

If the force is compressibility, Fc is negligible, the resulting net force

$$Fx=(Fg)x+(Fp)x+(Fv)x+(Ft)x$$

Navier's_ Stokes Equation :

For flow, where Ft is negligible,

T he resulting net force $Fx=(Fg)x+(Fp)x+(Fv)x+(Fc)x$

Euler's Equation of Motion :

If the flow is assumed to be ideal, viscous force(Fv) is zero and the resulting net force

$$Fx=(Fg)x+(Fp)x+(Ft)x+(Fc)x$$

$$dp\ \rho+gdz+vdv=0$$

which is the Euler's equation of motion .

Euler's equation use:

Euler's formula, Either of two important mathematical theorems of Leonhard Euler. The first is a topological invariance (see topology) relating the number of faces, vertices, and edges of any polyhedron. It is written F + V = E + 2, where F is the number of faces, V the number of vertices, and E the number of edges.

Euler's formula, Either of two important mathematical theorems of Leonhard Euler. The first is a topological invariance (see topology) relating the number of faces, vertices, and edges of any polyhedron. It is written F + V = E + 2, where F is the number of faces, V the number of vertices, and E the number of edges.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-16

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : II-FLUID KINEMATICS AND DYNAMICS

Date of Lecture:

Topic of Lecture: – Bernoulli's equation

Introduction : (Maximum 5 sentences)

Pressure head + Velocity head + Potential head =Total head (total energy per unit weight). where, hf represents the frictional work done (the work done against the fluid friction) per unit weight of a fluid element while moving from a station 1 to 2 along a streamline in the direction of flow.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

Bernoulli's Equation-Energy Equation of an ideal Flow along a Streamline

"Bernoulli's equation states that for an incompressible and inviscid fluid, the total mechanical energy of the fluid is constant"

Euler's equation (the equation of motion of an inviscid fluid) along a stream line for a steady flow with gravity as the only body force can be written as

$$V\frac{dV}{ds} = -\frac{1}{\rho}\frac{dp}{ds} - g\frac{dz}{ds} \qquad (13.6)$$

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

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

$$\text{or,} \quad \frac{V^2}{2} + \int \frac{dp}{\rho} + gz = C \qquad (13.7)$$

Where C is a constant along a streamline. In case of an incompressible flow, Eq. (13.7) can be written as

$$\frac{P}{\rho} + \frac{V^2}{2} + gz = C \qquad (13.8)$$

The Eqs (13.7) and (13.8) are based on the assumption that no work or heat interaction between a fluid element and the surrounding takes place. The first term of the Eq. (13.8) represents the flow work per unit mass, the second term represents the kinetic energy per unit mass and the third term represents the potential energy per unit mass.

Therefore the sum of three terms in the left hand side of Eq. (13.8) can be considered as the total mechanical energy per unit mass which remains constant along a streamline for a steady inviscid and incompressible flow of fluid. Hence the Eq. (13.8) is also known as **Mechanical energy equation**.

This equation was developed first by Daniel Bernoulli in 1738 and is therefore referred to as Bernoulli's equation. Each term in the Eq. (13.8) has the dimension of energy per unit mass. The equation can also be expressed in terms of energy per unit weight as

$$\frac{P}{\rho g} + \frac{V^2}{2g} + z = C_1 (\text{constant}) \qquad (13.9)$$

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

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

Bernoulli's Equation with Head Loss:

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

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 + h_f \qquad (13.10)$$

where, h_f represents the frictional work done (the work done against the fluid friction) per unit weight of a fluid element while moving from a station 1 to 2 along a streamline in the direction of flow. The term h_f is usually referred to as head loss between 1 and 2, since it amounts to the loss in total mechanical energy per unit weight between points 1 and 2 on a streamline due to the effect of fluid friction or viscosity.

Assumptions :

The following assumptions are made in the derivation of Bernoulli's equation.,

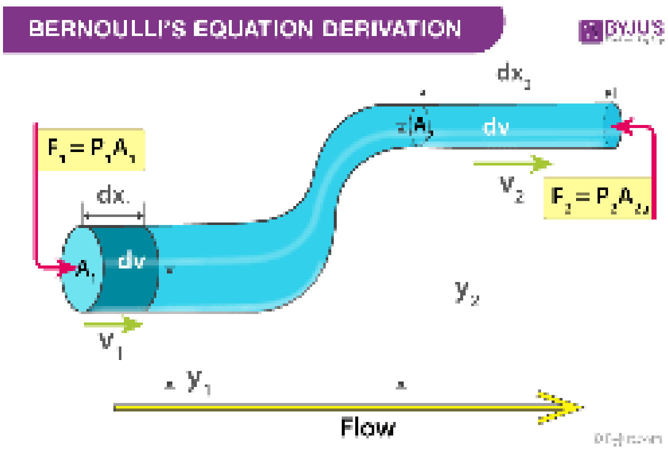
i) The fluid is ideal (viscosity is zero).

ii) The flow is steady.

iii) The flow is incompressible.

iv) The flow is irrotational.

It physically signifies that the difference in the total mechanical energy between stations 1 and 2 is dissipated into intermolecular or thermal energy and is expressed as loss of head h_f in Eq. (13.10).



The term head loss, is conventionally symbolized as h_L instead of h_f in dealing with practical problems. For an inviscid flow $h_L = 0$, and the total mechanical energy is constant along a streamline.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

L-17

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **II-FLUID KINEMATICS AND DYNAMICS**

Date of Lecture:

Topic of Lecture: – Bernoulli's equation-Applications

Introduction : (Maximum 5 sentences)
"Bernoulli's equation states that for an incompressible and inviscous fluid, the total mechanical energy of the fluid is constant"

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)
- ✓ No Prerequisite knowledge required

Detailed content of the Lecture:

Practial applications of Bernoulli's equation :

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

1. Venturimeter.
2. Orifice meter.
3. Pitot_tube.

VENTURIMETER

It is used for measuring the rate of a flow of a fluid flowing through the pipe. The basic principle is that by reducing the cross-sectional area of the flow passage, a pressure difference is created and the measurement of pressure difference enables the determination of the discharge through pipes. It consists of three parts.,

(i) A short converging part.

(ii) Throat.

(iii) Diverging part.

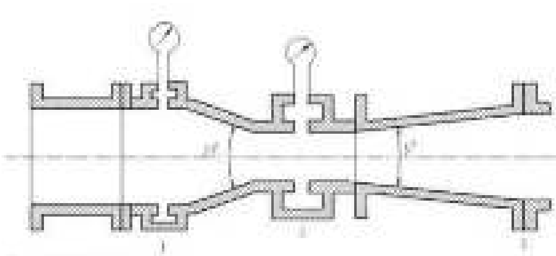


Fig. 1: Venturimeter

$$Q_{act} = C_d \times \frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}} \times \sqrt{2gh}$$

The value of ‘h’ is given by differential u-tube manometer:

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

Let $S_h = sp.gravity\ of\ the\ heavier\ liquid$

$So = sp.gravity\ of\ liquid\ flowing\ the\ pipe$

$x = difference\ of\ the\ heavier\ liquid\ column\ in\ u{-}tube$

$$h = x\left[\frac{S_h}{S_o} - 1\right]$$

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

Let $Sl = sp.gravity\ of\ the\ lighter\ liquid$

$So = sp.gravity\ of\ liquid\ flowing\ the\ pipe$

$x = difference\ of\ the\ heavier\ liquid\ column\ in\ u{-}tube$

$$h = x\left[1 - \frac{S_l}{S_o}\right]$$

Case III: This case related to the inclined venturimeter having differential u-tube manometer. Let the differential manometer contains heavier liquid, then ‘h’ is given by

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

Case IV: This case related to the inclined venturimeter having differential u-tube manometer. Let the differential manometer contains lighter liquid, then 'h' is given by

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

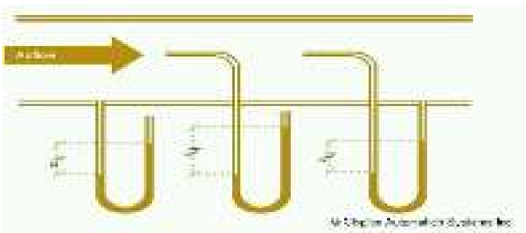
ORFICE METER

It is a device used for measuring the rate of a flow of a fluid flowing through the pipe. It is cheaper than venturimeter. It also works on the same principle as that of venturimeter.

$$Q = C_d \times \frac{a_0 a_1}{\sqrt{a_1^2 - a_0^2}} \times \sqrt{2gh}$$

PITOT TUBE

It is a device used for measuring the velocity of flow at any point in a pipe. It is based on the principle that if the velocity of flow at a point becomes zero, the pressure there is increased due to the conversion of energy into pressure energy.



$$h = x\left[\frac{S_g}{S_o} - 1\right]$$

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

L-18

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : II-FLUID KINEMATICS AND DYNAMICS

Date of Lecture:

Topic of Lecture: Flow measurements

Introduction : (Maximum 5 sentences)

Flow measurement is the quantification of bulk fluid movement. Flow can be measured in a variety of ways.Positive-displacement flowmeters, which accumulate a fixed volume of fluid and then count the number of times the volume is filled to measure flow.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

✓ No Prerequisite knowledge required

Detailed content of the Lecture:

FLOW MEASUREMENTS:

Flow measurement is the quantification of bulk fluid movement. Flow can be measured in a variety of ways. The common types of flowmeters with industrial applications are listed below:

- a) Obstruction type (differential pressure or variable area)
- b) Inferential (turbine type)
- c) Electromagnetic
- d) Positive-displacement flow meters, which accumulate a fixed volume of fluid and then count the number of times the volume is filled to measure flow.
- e) Fluid dynamic (vortex shedding)
- f) Anemometer
- g) Ultrasonic

- h) Mass flowmeter (Coriolis force).

Flow measurement methods other than positive-displacement flowmeters rely on forces produced by the flowing stream as it overcomes a known constriction, to indirectly calculate flow. Flow may be measured by measuring the velocity of fluid over a known area. For very large flows, tracer methods may be used to deduce the flow rate from the change in concentration of a dye or radioisotope.

VARIABLE-AREA FLOWMETERS:

A "variable area meter" measures fluid flow by allowing the cross sectional area of the device to vary in response to the flow, causing some measurable effect that indicates the rate. A rotameter is an example of a variable area meter, where a weighted "float" rises in a tapered tube as the flow rate increases; the float stops rising when area between float and tube is large enough that the weight of the float is balanced by the drag of fluid flow.

A kind of rotameter used for medical gases is the Thorpe tube flowmeter. Floats are made in many different shapes, with spheres and spherical ellipses being the most common. Some are designed to spin visibly in the fluid stream to aid the user in determining whether the float is stuck or not.

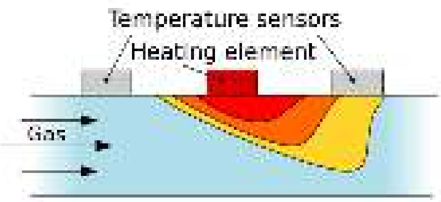
Rotameters are available for a wide range of liquids but are most commonly used with water or air. They can be made to reliably measure flow down to 1% accuracy. Another type is a variable area orifice, where a spring-loaded tapered plunger is deflected by flow through an orifice.

THERMAL MASS FLOWMETERS:

Thermal mass flowmeters generally use combinations of heated elements and temperature sensors to measure the difference between static and flowing heat transfer to a fluid and infer its flow with a knowledge of the fluid's specific heat and density.

The fluid temperature is also measured and compensated for. If the density and specific heat characteristics of the fluid are constant, the meter can provide a direct mass flow readout, and does not need any additional pressure temperature compensation over their specified range.

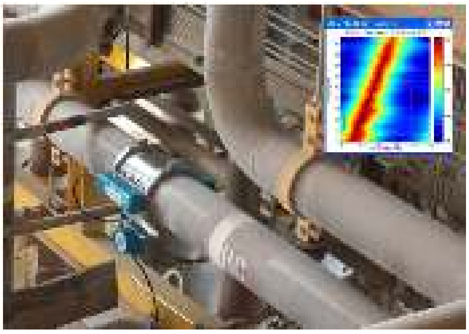
Technological progress has allowed the manufacture of thermal mass flowmeters on a microscopic scale as MEMS sensors; these flow devices can be used to measure flow rates in the range of nanoliters or microliters per minute.



SONAR FLOW MEASUREMENT:

Sonar flowmeters are non-intrusive clamp-on devices that measure flow in pipes conveying slurries, corrosive fluids, multiphase fluids and flows where insertion type flowmeters are not desired.

Sonar flowmeters have been widely adopted in mining, metals processing, and upstream oil and gas industries where traditional technologies have certain limitations due to their tolerance to various flow regimes and turn down ratios.



Sonar flow meters have the capacity of measuring the velocity of liquids or gases non-intrusively within the pipe and then leverage this velocity measurement into a flow rate by using the cross-sectional area of the pipe and the line pressure and temperature.

The principle behind this flow measurement is the use of underwater acoustics.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.165-258
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.296-348

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-19

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:– Shear stress, pressure gradient relationship

Introduction : (Maximum 5 sentences)
Flow through pipes is an important engineering problem in fluid mechanics. Almost in allour daily operations, we come across pipe flow. For example, the household water supply,sewage flows etc. The pipe flow is also used for the transportation of chemicals and petroleumproducts in different chemical and oil industries.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Shear stress:

Shear stress is defined as a force per unit area, acting parallel to an infinitesimal surface element. Shear stress is primarily caused by friction between fluid particles, due to fluid viscosity.

The form of the relation between shear stress and rate of strain depends on a fluid, and most common fluids obey Newton's law of viscosity, which states that the shear stress is proportional to the strain rate:

$\tau = \mu \, d\gamma \, dt \, . \, \nu = \mu \, \rho \, .$

$\gamma = \Delta x \, \delta y = (u(y + \delta y) - u(y))\Delta t \, \delta y \, .$

Maximum shear stress:

The maximum shear stress is the maximum concentrated shear force in a small area. ... The neutral axis of a cross section is the axis at which the value of the normal stress and strain are equal to zero.

Shear stresses in a Newtonian fluid:

A fluid at rest can not resist shearing forces. Under the action of such forces it deforms continuously, however small they are. The resistance to the action of shearing forces in a fluid appears only when the fluid is in motion. This implies the principal difference between fluids and solids. For solids the resistance to a shear deformation depends on the deformation itself, that is the shear stress τ is a function of the shear strain γ. For fluids the shear stress τ is a function of the rate of strain dγ/dt. The property of a fluid to resist the growth of shear deformation is called viscosity.

Pressuregradient relationship:

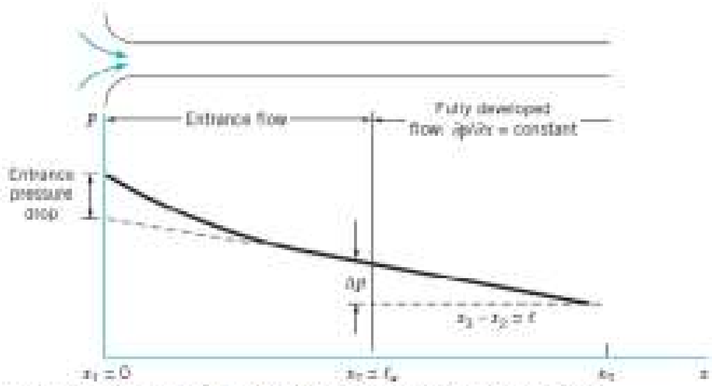


Figure 5.7. Pressure distribution along a horizontal pipe.

Fully developed steady flow in a constant diameter pipe may be driven by gravity and/or pressure forces. For horizontal pipe flow, gravity has no effect except for a hydrostatic pressure variation across the pipe, γ, that is usually negligible.

It is the pressure difference, $\Delta P = P1 - P2$ between one section of the horizontal pipe and another which forces the fluid through the pipe. Viscous effects provide the restraining force that exactly balances the pressure force, thereby allowing the fluid to flow through the pipe with no acceleration.

If viscous effects were absent in such flows, the pressure would be constant throughout the pipe, except for the hydrostatic variation. In non-fully developed flow regions, such as the entrance region of a pipe, the fluid accelerates or decelerates as it flows (the velocity profile changes from a uniform profile at the entrance of the pipe to its fully developed profile at the end of the entrance region). The result is a pressure distribution along the horizontal pipe as shown in Fig.5.7. The magnitude of the pressure gradient, $\partial P \partial x$, is larger in the entrance region than in the fully developed region, where it is a constant, $\partial P \partial x = -\Delta P L < 0$

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-20

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:– Laminar flows through pipes and between plates

Introduction : (Maximum 5 sentences)

Laminar flow is defined as the parallel flow of fluid in one direction at a constant speed between two layers with no disturbance. In a straight pipe, laminar flow is considered as the relative movement of an arrangement of concentric chambers of liquid, the outside one settled against the wall of a pipe and the others moving at expanding speeds as the focal point of the pipe is drawn nearer.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Fully Developed Laminar Flow:

We mentioned that flow in pipes is laminar for $Re \leq 2100$, and that the flow is fully developed if the pipe is sufficiently long (relative to the entry length) so that the entrance effects are negligible. In this section we consider the steady laminar flow of an incompressible fluid with constant properties in the fully developed region of a straight circular pipe.

We obtain the momentum equation by applying a momentum balance to a differential volume element, and obtain the velocity profile by solving it. Then we use it to obtain a relation for the friction factor. An important aspect of the analysis here is that it is one of the few available for viscous flow.

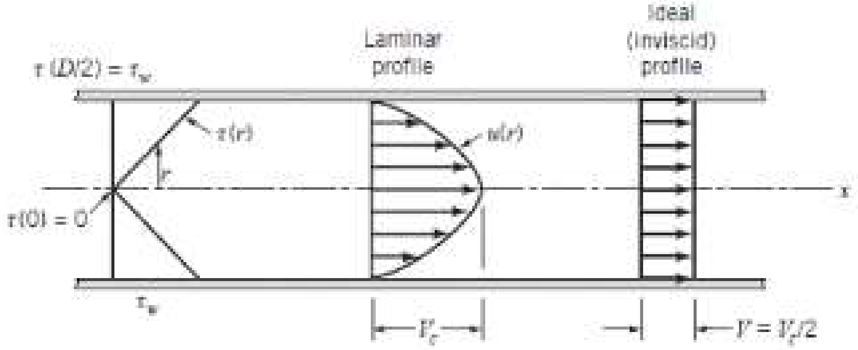


Figure 5.8. Shear stress distribution within the fluid in a pipe (laminar or turbulent flow) and typical velocity profiles

For laminar flow of a Newtonian fluid, the shear stress is simply proportional to the velocity gradient. In the notation associated with our pipe flow, this becomes

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

The negative sign is included to give $\tau > 0$ with du/dr<0 (the velocity decreases from the pipe centerline to the pipe wall).

The velocity profile can be written as

$$u(r) = \left(\frac{\Delta p D^2}{16\mu \ell}\right)\left[1 - \left(\frac{2r}{D}\right)^2\right] = V_c\left[1 - \left(\frac{2r}{D}\right)^2\right]$$

$$u(r) = \mathbf{V_c}\left(1 - \frac{r^2}{R^2}\right)$$

It is usually advantageous to describe a process in terms of dimensionless quantities. To this end we rewrite the pressure drop equation for laminar horizontal pipe flow, as $\Delta P = 32\mu l V/D$ 2 and divide both sides by the dynamic pressure, ρV 2 /2 obtain the dimensionless form as

$$\Delta p = f \frac{\ell}{D} \frac{\rho V^2}{2} \qquad f = \Delta p(D/\ell)/(\rho V^2/2)$$

is termed the friction factor, or sometimes the Darcy friction factor (This parameter should not be confused with the less-used Fanning friction. which is defined to be f/4. In this text we will use only the Darcy fiction factor.)

Thus the friction factor for laminar fully developed pipe flow is simply f= 64/Re. By substituting the pressure drop in terms of the wall shear stress, we obtain an alternate expression for the friction factor as a dimensionless wall shear stress

Laminar Flow in Noncircular Pipes:

The friction factor f relations are given in the below Table 5.1. for fully developed laminar flow in pipes of various cross sections. The Reynolds number for flow in these pipes is based on the hydraulic diameter Dh=4Ac /p, where Ac is the crosssectional area of the pipe and p is its wetted perimeter.

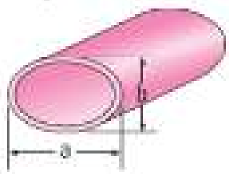
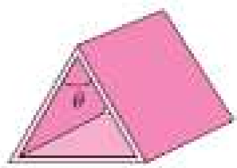
The nature of the flow (laminar vs. turbulent) not only depends on its velocity but also itsdensity, viscosity and length scale. For flow between parallel plates, the flow is laminar when Re < 1,400; whereas for pipe flow, the flow is laminar when Re < 2,100.

Laminar flow is defined as the parallel flow of fluid in one direction at a constant speed between two layers with no disturbance.In a straight pipe, laminar flow is considered as the relative movement of an arrangement of concentric chambers of liquid, the outside one settled against the wall of a pipe and the others moving at expanding speeds as the focal point of the pipe is drawn nearer.

In laminar flow, thin layers (sub-layer) of water are formed in a definite direction at different speeds without mixing in between the layers

Table 5.1. Friction factor for fully developed laminar flow

Friction factor for fully developed *laminar flow* in pipes of various cross sections ($D_h = 4A_c/p$ and $Re = V_{avg} D_h/\nu$)

Tube Geometry	a/b or $\theta°$	Friction Factor f
Circle 	—	64.00/Re
Rectangle 	a/b	
	1	56.92/Re
	2	62.20/Re
	3	68.36/Re
	4	72.92/Re
	6	78.80/Re
	8	82.32/Re
	∞	96.00/Re
Ellipse 	a/b	
	1	64.00/Re
	2	67.28/Re
	4	72.96/Re
	8	76.60/Re
	16	78.16/Re
Isosceles triangle 	θ	
	10°	50.80/Re
	30°	52.28/Re
	60°	53.32/Re
	90°	52.60/Re
	120°	50.96/Re

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-21

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:– Laminar flows through pipes and between plates

Introduction : (Maximum 5 sentences)
In Non-ideal fluid dynamics, the Hagen–Poiseuille equation, also known as the Hagen–Poiseuille law, Poiseuille law or Poiseuille equation, is a physical law that gives the pressure drop in an incompressible and Newtonian fluid in laminar flow flowing through a long cylindrical pipe of constant cross section.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

The derivation of the Hagen-Poiseuille equation for laminar flow in straight, circular pipes is based on the following two assumptions; a) The viscous property of fluid follows Newton's law of viscosity, that is, $\tau=\mu(du/dy)$, b) There is no relative motion between fluid particles and solid boundaries, that is, no slip of fluid particles at the solid boundary. Fig. 7.4 illustrates the laminar motion of fluid in a horizontal circular pipe located at a sufficiently great distance from the entrance section when a steady laminar flow occurs in a straight stretch of horizontal pipe, a pressure gradient must be maintained in the direction of flow to overcome the frictional forces on the concentric cylindrical surfaces, as shown in Fig. 7.4.

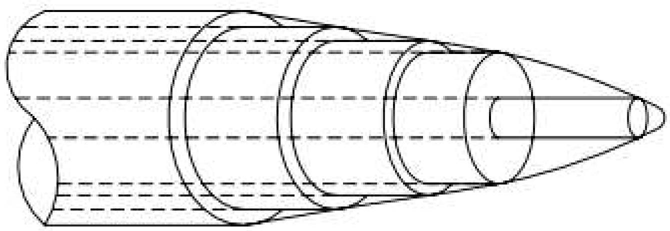


Fig. 7.4

Each concentric cylindrical layer of fluid is assumed to slide over the other in an axialirection. For practical purposes the pressure may be regarded as distributed uniformly overany chosen cross section of the pipe.

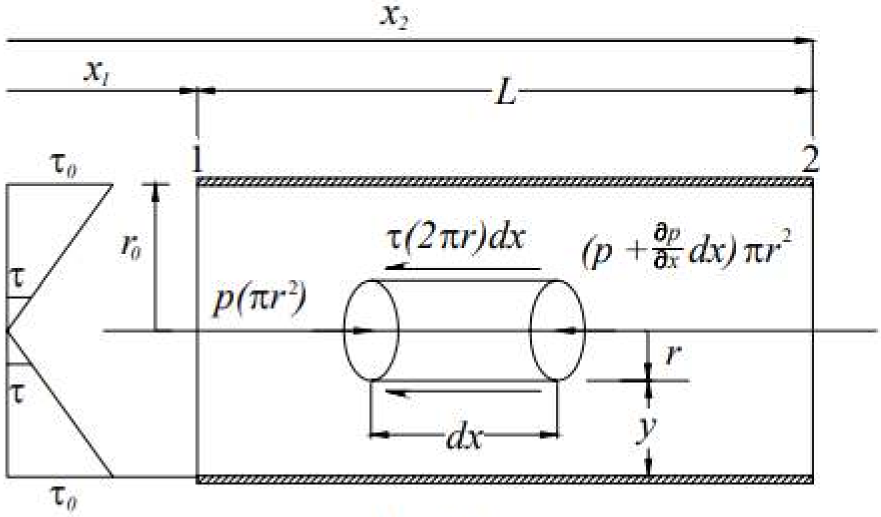


Fig. 7.5

POISEUILLES LAW FORMULA:

The law of Poiseuille states that the flow of liquid depends on the following variables such as the length of the tube(L), radius (r), pressure gradient (ΔP) and the viscosity of the fluid (η) in accordance with their relationship.

The entire relation or the Poiseuille's Law formula is given by

$$Q = \Delta P\pi r4 / 8\eta l$$

Wherein,The Pressure Gradient (ΔP) Shows the pressure differential between the two ends of the tube, defined by the fact that every fluid will always flow from the high pressure (P1) to the low-pressure area (P2) and the flow rate is calculated by the $\Delta P = P1\text{-}P2$.

The radius of the narrow tube:

The flow of liquid direct changes with the radius to the power four.

Viscosity (η):

The flow rate of the fluid is inversely proportional to the viscosity of the fluid.

Length of the arrow tube (L):

The flow rate of the fluid is inversely proportional to the length of the narrow tube.

Resistance(R):

The resistance is calculated by 8Ln / πr4 and hence the Poiseuille's law isQ= (ΔP) R

Poiseuille's Law is a description of the pressure of a fluid as it travels through a cylindrical pipe.In an operating room, Poiseuille's law is extremely important. For procedures requiring general anesthesia, a patient's airway must be kept open for proper exchange of oxygen and carbon dioxide.

FLOW IN PIPES:

- Fluid flow in circular and noncircular pipes is commonly encountered in practice. The hot and cold water that we use in our homes is pumped through pipes. Water in a city is distributed by extensive piping networks. Oil and natural gas are transported hundreds of miles by large pipelines.
- Blood is carried throughout our bodies by arteries and veins.The cooling water in an engine is transported by hoses to the pipes in the radiator where it is cooled as it flows.
- Thermal energy in a hydronic space heating system is transferred to the circulating water in the boiler, and then it is transported to the desired locations through pipes.
- Fluid flow is classified as external and internal, depending on whether the fluid is forced to flow over a surface or in a conduit. Internal and external flows exhibit very different characteristics.
- In this chapter we consider internal flow where the conduit is completely filled with the fluid, and flow is driven primarily by a pressure difference.
- This should not be confused with open-channel flow where the conduit is partially filled by the fluid and thus the flow is partially bounded by solid surfaces, as in an irrigation ditch, and flow is driven by gravity alone.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

L-22

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:– Turbulent flow

Introduction : (Maximum 5 sentences)
Turbulent flow occurs when the Reynolds number exceeds 4000. Eddy currents are present within the flow and the ratio of the internal roughness of the pipe to the internal diameter of the pipe needs to be considered to be able to determine the friction factor.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

While laminar flow is "orderly" turbulent flow is "Random" and "Chaotic". It is also found that a flow in a pipe is laminar if the Reynolds Number (based on diameter of the pipe) is less than 2100 and is turbulent if it is greater than 4000.

Causes of Turbulent Flow:

- ➢ Turbulence is caused by excessive kinetic energy in parts of a fluid flow, which overcomes the damping effect of the fluid's viscosity. For this reason turbulence is commonly realized in low viscosity fluids.This increases the energy needed to pump fluid through a pipe.

Characteristics of Turbulent Flow:

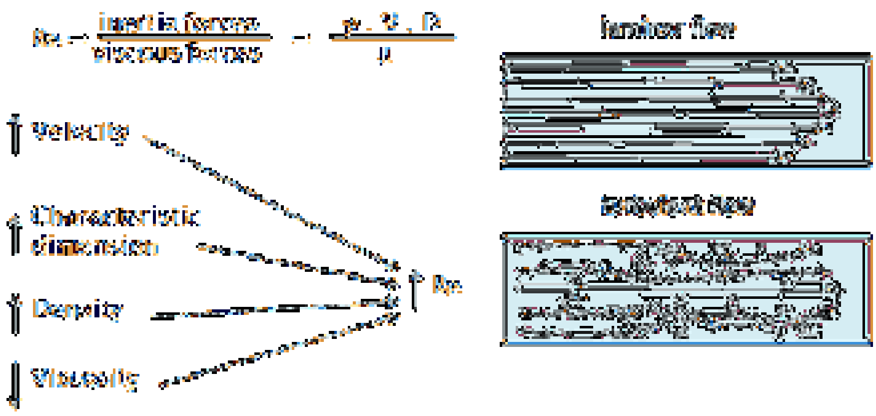
- Turbulent flow tends to occur at higher velocities, low viscosity and at higher characteristic linear dimensions.
- If the Reynolds number is greater than Re > 3500, the flow is turbulent.
- **Irregularity:** The flow is characterized by the irregular movement of particles of the fluid. The movement of fluid particles is chaotic. For this reason, turbulent flow is normally treated statistically rather than deterministically.
- **Diffusivity:** In turbulent flow, a fairly flat velocity distribution exists across the section of pipe,

with the result that the entire fluid flows at a given single value and drops rapidly extremely close to the walls. The characteristic which is responsible for the enhanced mixing and increased rates of mass, momentum and energy transports in a flow is called “diffusivity”.

- **Rotationality:** Turbulent flow is characterized by a strong three-dimensional vortex generation mechanism. This mechanism is known as vortex stretching.
- **Dissipation:** A dissipative process is a process in which the kinetic energy of turbulent flow is transformed into internal energy by viscous shear stress.

<u>Reynolds Number:</u>

- The Reynolds number is the ratio of inertial forces to viscous forces and is a convenient parameter for predicting if a flow condition will be laminar or turbulent.
- It can be interpreted that when the viscous forces are dominant (slow flow, low Re) they are sufficient enough to keep all the fluid particles in line, then the flow is laminar.
- Even very low Re indicates viscous creeping motion, where inertia effects are negligible. When the inertial forces dominate over the viscous forces (when the fluid is flowing faster and Re is larger) then the flow is turbulent.



- It is a dimensionless number comprised of the physical characteristics of the flow. An increasing Reynolds number indicates an increasing turbulence of flow.
- Turbulent flow, type of fluid (gas or liquid) flow in which the fluid undergoes irregular fluctuations, or mixing, in contrast to laminar flow, in which the fluid moves in smooth paths or layers.
- In turbulent flow the speed of the fluid at a point is continuously undergoing changes in both magnitude and direction.
- The flow of wind and rivers is generally turbulent in this sense, even if the currents are gentle. The air or water swirls and eddies while its overall bulk moves along a specific direction.
- Most kinds of fluid flow are turbulent, except for laminar flow at the leading edge of solids moving relative to fluids or extremely close to solid surfaces, such as the inside wall of a pipe, or in cases of fluids of high viscosity (relatively great sluggishness) flowing slowly through small channels.

- Common examples of turbulent flow are blood flow in arteries, oil transport in pipelines, lava flow, atmosphere and ocean currents, the flow through pumps and turbines, and the flow in boat wakes and around aircraft-wing tips.

Advantage Of Turbulent flow:

A turbulent flow can be either an advantage or disadvantage. A turbulent flow increases the amount of air resistance and noise; however, a turbulent flow also accelerates heat conduction and thermal mixing. Therefore, understanding, handling, and controlling turbulent flows can be crucial for successful product design.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

L-23

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : III - FLOW THROUGH PIPES

Date of Lecture:

Topic of Lecture:Darcy- Weisbach formula –Pipe roughness

Introduction : (Maximum 5 sentences)
In fluid dynamics, the Darcy–Weisbach equation is an empirical equation, which relates the head loss, or pressure loss, due to friction along a given length of pipe to the average velocity of the fluid flow for an incompressible fluid.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Darcy- Weisbach formula:

- Flow of fluid through a pipe The flow of liquid through a pipe is resisted by viscous shear stresses within the liquid and the turbulence that occurs along the internal walls of the pipe, created by the roughness of the pipe material.
- This resistance is usually known as pipe friction and is measured is feet or metres head of the fluid, thus the term head loss is also used to express the resistance to flow. Many factors affect the head loss in pipes, the viscosity of the fluid being handled, the size of the pipes, the roughness of the internal surface of the pipes, the changes in elevations within the system and the length of travel of the fluid.
- The resistance through various valves and fittings will also contribute to the overall head loss. A method to model the resistances for valves and fittings is described elsewhere. In a well designed system the resistance through valves and fittings will be of minor significance to the overall head loss, many designers choose to ignore the head loss for valves and fittings at least in the initial stages of a design.
- Much research has been carried out over many years and various formulae to calculate head loss have been developed based on experimental data.

Weisbach first proposed the equation we now know as the Darcy-Weisbach formula orDarcy-Weisbach equation:

$$hf = \frac{4\,f\,LV^2}{2g}$$

where:

hf = head loss (m),f = friction factor

L = length of pipe work (m),d = inner diameter of pipe work (m)

v = velocity of fluid (m/s),g = acceleration due to gravity (m/s²)

g = acceleration due to gravity (ft/s²)

The friction factor fD is not a constant: it depends on such things as the characteristics of the pipe (diameter D and roughness height ε), the characteristics of the fluid (its kinematic viscosity ν [nu]), and the velocity of the fluid flow ⟨v⟩.

It has been measured to high accuracy within certain flow regimes and may be evaluated by the use of various empirical relations, or it may be read from published charts. These charts are often referred to as Moody diagrams, after L. F. Moody, and hence the factor itself is sometimes erroneously called the Moody friction factor.

It is also sometimes called the Blasius friction factor, after the approximate formula he proposed.

Use of Darcy Weisbach equation:

In Fluid Dynamics the Darcy–Weisbach equation is a phenomenological equation, which relates the head loss or pressure loss due to friction along a given length of pipe to the average velocity of the fluid flow for an incompressible fluid.

Pipe roughness:

Roughness coefficient is based on the material of the pipe. For PVC pipe, the standard C value is 150. New steel pipe uses a C value of 140, but with use and corrosion a lower value is typically used. For HDPE pipe, a range of C values between 150 and 160 is typical.

The roughness coefficient is a number that describes how smooth or rough a pipes surface is. This coefficient is used by engineers (or software) to determine friction losses of fluids moving through the pipe.

Relative roughness of pipe:

Relative roughness is the amount of surface roughness that exists inside the pipe. The relative roughness of a pipe is known as the absolute roughness of a pipe divided by the inside diameter of a pipe.Roughness coefficient. A value used in Manning's formula to determine energy losses of flowing water due to pipe or channel wall roughness. Also see friction loss, Manning's formula, and n Factor. There are different systems for calculating the pipe roughness, but typically, the higher the value, the rougher the pipe, and the more losses due to friction.

Material	Typical Manning roughness coefficient
Concrete	0.012
Gravel bottom with sides	
- concrete	0.020
- mortared stone	0.023
- riprap	0.033
Natural stream channels	
Clean, straight stream	0.030
Clean, winding stream	0.040
Winding with weeds and pools	0.050
With heavy brush and timber	0.100
Flood Plains	
Pasture	0.035
Field crops	0.040
Light brush and weeds	0.050
Dense brush	0.070
Dense trees	0.100

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-24

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:Moody's diagram

Introduction : (Maximum 5 sentences)
In Engineering, the Moody chart or Moody diagram is a graph in non-dimensional form that relates the Darcy-Weisbach friction factor f_D, Reynolds number Re, and surface roughness for fully developed flow in a circular pipe

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

Moody's diagram:

The Moody chart or Moody diagram is a graph in non-dimensional form that relates the Darcy-Weisbach friction factor fD, Reynolds number Re, and surface roughness for fully developed flow in a circular pipe.

Introducing the concept of smooth and rough pipes, as shown in Moody chart, we find:

1) For laminar flow, f = 16 / Re

2) For transitional flow, pipes' flow lies outside this region.

3) For smooth turbulent (a limiting line of turbulent flow), all values of relative roughness (ks/d) tend toward this line as R decreases. Blasius equation: f = 0.079 / Re0.25

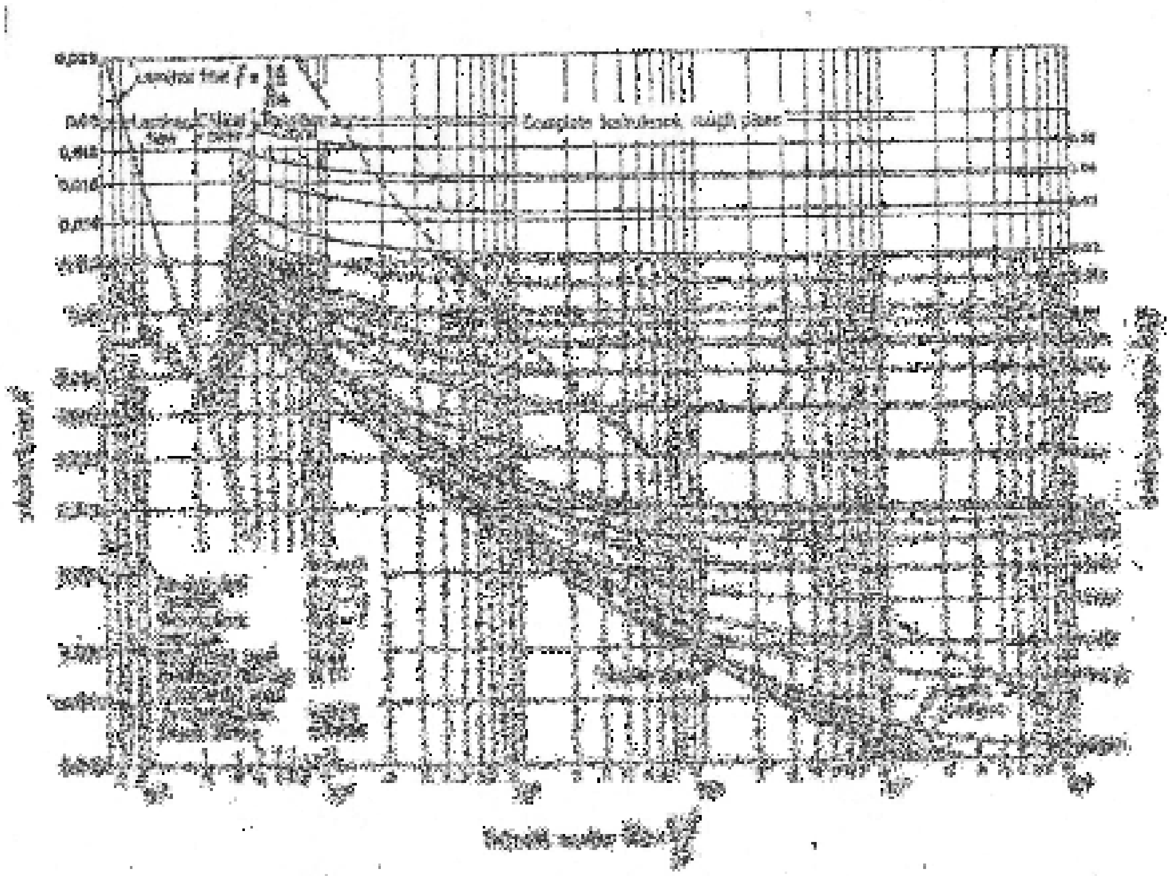
4) For transitional turbulent, it is the region where (f) varies with both (ks/d) & (Re). Most pipes lie in this region.

5) For rough turbulent, (f) is constant for given (ks/d) and is independent of (Re).

Determine the friction coefficient and the head loss due to friction per meter length of the pipe using:

1- Moody chart

2- 2- Smooth pipe formula



Moody chart use:

In case of a laminar fully developed flow through pipes, the friction factor, f is found from the exact solution of the Navier-Stokes equation.In the case of a turbulent flow, friction factor depends on both the Reynolds number and the roughness of pipe surface.

Sir Thomas E. Stanton (1865-1931) first started conducting experiments on a number of pipes of various diameters and materials and with various fluids. Afterwards, a German engineer Nikuradse carried out experiments on flows through pipes in a very wide range of Reynolds number.

A comprehensive documentation of the experimental and theoretical investigations on the laws of friction in pipe flows has been presented in the form of a diagram by L.F. Moody to show the variation of friction factor, f with the pertinent governing parameters, namely, the Reynolds number of flow and the relative roughness of the pipe. This diagram is known as Moody's Chart which is employed till today as the best means for predicting the values of f.

- The friction factor f at a given Reynolds number, in the turbulent region, depends on the relative roughness, defined as the ratio of average roughness to the diameter of the pipe, rather than the absolute roughness.For moderate degree of roughness, a pipe acts as a smooth pipe up to a value of Re where the curve of f vs Re for the pipe coincides with that of a smooth pipe.

This zone is known as the smooth zone of flow .

- The region where f vs Re curves become horizontal showing that f is independent of Re, is known as the rough zone and the intermediate region between the smooth and rough zone is known as the transition zone.
- The position and extent of all these zones depend on the relative roughness of the pipe. In the smooth zone of flow, the laminar sublayer becomes thick, and hence, it covers appreciably the irregular surface protrusions. Therefore all the curves for smooth flow coincide.
- With increasing Reynolds number, the thickness of sublayer decreases and hence the surface bumps protrude through it. The higher is the roughness of the pipe, the lower is the value of Re at which the curve of f vs Re branches off from smooth pipe curve.
- In the rough zone of flow, the flow resistance is mainly due to the form drag of those protrusions. The pressure drop in this region is approximately proportional to the square of the average velocity of flow. Thus f becomes independent of Re in this region.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-25

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:Major and minor losses of flow in pipes

Introduction : (Maximum 5 sentences)
Major Head Loss - head loss or pressure loss - due to friction in pipes and ducts. Minor Head Loss - head loss or pressure loss - due to components as valves, bends, tees and the like in the pipe or duct system.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

Major loss in pipe:

- The friction loss In a uniform, straight sections of pipe, known as "major loss", is caused by the effects of viscosity, the movement of fluid molecules against each other or against the (possibly rough) wall of the pipe.

Minor loss in pipe:

- Losses due to the local disturbances of the flow in the conduits such as changes in cross section, projecting gaskets, elbows, valves and similar items are called minor losses.

Difference between major and minor losses:

- Major losses are head losses due to friction factor and pipe diameter, and can vary depending on the type of pipe used. Minor losses are small Page 2 losses due mainly to bends or valves that disrupt a smooth steady flow

MAJOR LOSSES:

Major losses refer to the losses in pressure head of the flow due to friction effects. Such losses can be evaluated by using the Darcy-Weisbachequation.where f is the Darcy friction factor, L is the length of the pipe segment, v is the flow velocity, D is the diameter of the pipe segment, and g is acceleration due to gravity. it is valid for any fullydeveloped, steady and incompressible flow.

MINOR LOSSES IN PIPES:

The fluid in a typical piping system passes through various fittings, valves, bends, elbows, tees, inlets, exits, enlargements, and contractions in addition to the pipes.

These components interrupt the smooth flow of the fluid and cause additional losses because of the flow separation and mixing they induce. In a typical system with long pipes, these losses are minor compared to the total head loss in the pipes (the major losses) and are called minor losses.

- Losses caused by fittings, bends, valves, etc
- Minor in comparison to friction losses which are considered major. • Losses are proportional to – velocity of flow, geometry of device.
- The value of K is typically provided for various devices. • Energy lost – units – N.m/N or lb-ft/lb
- K - loss factor - has no units (dimensionless)

. Such losses are generally termed minor losses, with the apparent implication being that the majority of the system loss is associated with the friction in the straight portions of the pipes, the major losses or local losses. In many cases this is true. In other cases the minor losses are greater than the major losses. The minor losses may raised by

- Pipe entrance or exit
- Sudden expansion or contraction
- Bends, elbows, tees, and other fittings
- Valves,open or partially closed
- Gradual expansions or contractions

SuddenEnlargement

Energy lost is because of turbulence. Amount of turbulence depends on the differences in pipe diameters.Gradual Enlargement If the enlargement is gradual (as opposed to our previous case) – the energy losses are less. The loss again depends on the ratio of the pipe diameters and the angle of enlargement.

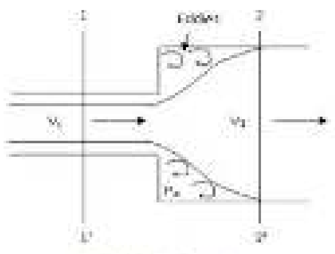


Fig. 1 Sudden Expansion

Note – • If angle increases (in pipe enlargement) – minor losses increase • If angle decreases – minor losses decrease, but you also need a longer pipe to make the transition – that means more FRICTION losses - therefore there is a tradeoff!

Sudden Contraction

Decrease in pipe diameter

Note that the loss is related to the velocity in the second (smaller) pipe! The loss is associated with the contraction of flow and turbulence

The section at which the flow is the narrowest – Vena Contracta • At vena contracta, the velocity is maximum.

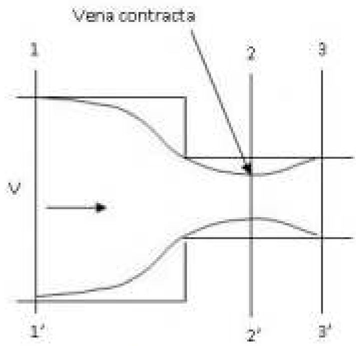


Fig. 2 Sudden Contraction

Gradual Contraction Again a gradual contraction will lower the energy loss (as opposed to sudden contraction). θ is called the cone angle.

Entrance Losses

Fluid moves from zero velocity in tank to v2

Loss of head due to bend in pipe:

This is the energy loss due to bend. When a bend is provided in the pipeline, there is a change in direction of the velocity of flow (figures 3 and 4). Due to this, the flow separates from the walls of the bend and formation of eddies takes place.

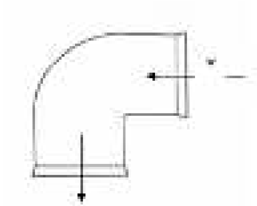


Fig. 3 Sharp Bend (90° elbow)

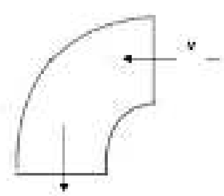


Fig. 4 Smooth Bend

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : III - FLOW THROUGH PIPES

Date of Lecture:

Topic of Lecture:Pipes in series

Introduction : (Maximum 5 sentences)
Discussion on flow through branched pipes (pipes in series or in parallel, or in combination of both), the following principles can be summarized: The friction equation must be satisfied for each pipe. There can be only one value of head at any point.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

Flow in pipe series and parallel.

Flow through pipe in series •When pipes of different diameters are connected end to end to form a pipe line, they are said to be in series. The total loss of energy (or head) will be the sum of the losses in each pipe plus local losses at connections.

Pipe in Series:

Pipes are said to be in series if they are connected end to end (in continuation with each other) so that the fluid flows in a continuous line without any branching. The volume rate of flow through the pipes in series is the same throughout.

Pipes in series are pipes with different diameters and lengths connected together forming a pipe line. Consider pipes in series discharging water from a tank with higher water level to another with lower water level,

Neglecting secondary losses, it is obvious that the total head loss HL between the two tanks is the sum of the friction losses through the pipe line.

Suppose a pipe line consists of a number of pipes of different sizes and lengths. See Fig. 13.37.

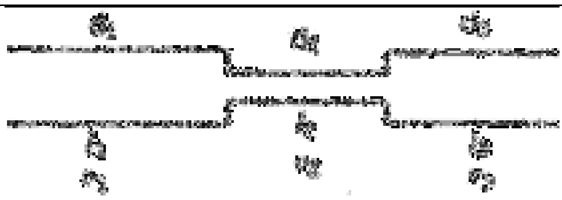


Fig. 13.37, Pipes in series.

Let d1, d2, d3 be the diameters of the component pipes.

Let l1, l2, l3 be the lengths of these component pipes.

Let v1, v2, v3 be the velocities in these pipes.

Friction losses through the pipe line are the sum of friction loss of each pipe.

$$H_L = h_{f1} + h_{f2} + h_{f3} + ...$$

$$H_L = \frac{4f_1 L_1 v_1^2}{2gd_1} + \frac{4f_2 L_2 v_2^2}{2gd_2} + \frac{4f_3 L_3 v_3^2}{2gd_3} + ...$$

OR:

$$H_L = \frac{32 f_1 L_1 Q^2}{\pi^2 g d_1^5} + \frac{32 f_2 L_2 Q^2}{\pi^2 g d_2^5} + \frac{32 f_3 L_3 Q^2}{\pi^2 g d_3^5} + ...$$

Pipes connected in continuation as in this case are said to be connected in series. In this arrangement the rate of discharge Q is the same in all the pipes. Ignoring secondary losses the total loss of head is equal to the sum of the friction losses in the individual pipes.

If h_{f1}, h_{f2}, h_{f3} be the losses of head in the individual pipes the total loss of head h_f is given by,

$$h_f = h_{f1} + h_{f2} + h_{f3} = \frac{4fl_1}{d_1}\cdot\frac{v_1^2}{2g} + \frac{4fl_2}{d_2}\cdot\frac{v_2^2}{2g} + \frac{4fl_3}{d_3}\cdot\frac{v_3^2}{2g}$$

$$= \frac{fl_1 Q^2}{3.0257 d_1^5} + \frac{fl_2 Q^2}{3.0257 d_2^5} + \frac{fl_3 Q^2}{3.0257 d_3^5}$$

If the [illegible] of the pipes are [illegible] the above equation can be so arranged as

$$\frac{l}{d^5} = \frac{l_1}{d_1^5} + \frac{l_2}{d_2^5} + \frac{l_3}{d_3^5} + ...$$

The above relation is called Dupuit's equation.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

L-27

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **III - FLOW THROUGH PIPES**

Date of Lecture:

Topic of Lecture:Pipes in series

Introduction : (Maximum 5 sentences)
Pipes are said to be in parallel when they are so connected that the flow from a pipe branches or divides into two or more separate pipes and then reunite into a single pipe.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

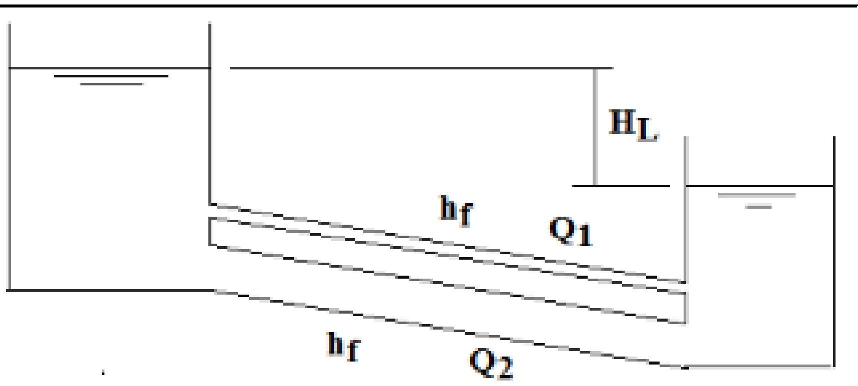
Flow in pipe series and parallel.

Flow through pipe in series •When pipes of different diameters are connected end to end to form a pipe line, they are said to be in series. The total loss of energy (or head) will be the sum of the losses in each pipe plus local losses at connections.

Pipes Connected in Parallel:

- Pipes are said to be in parallel when they are so connected that the flow from a pipe branches or divides into two or more separate pipes and then reunite into a single pipe.
- Pipes in parallel are pipes with different diameters and same lengths, where each pipe is connected separately to increase the discharge.
- Consider pipes in parallel discharging water from a tank with higher water level to another with lower water level, as shown in the figure.
- Neglecting minor losses, it is obvious that the total head loss HL between the two tanks is the same as the friction losses through each pipe.
- The friction losses through all pipes are the same, and all pipes discharge water independently.

  $HL = hf\,1 = hf\,2 = ...$



$$L_1 = L_2 = L$$

$$H_L = \frac{4 f_1 L v_1^2}{2 g d_1} = \frac{4 f_2 L v_2^2}{2 g d_2} = \ldots$$

$$H_L = \frac{32 f_1 L Q_1^{\,2}}{\pi^2 g d_1^{\,5}} = \frac{32 f_2 L Q_2^{\,2}}{\pi^2 g d_2^{\,5}} = \ldots$$

$$Q = Q_1 + Q_2$$

From the above discussion on flow through branched pipes (pipes in series or in parallel, or in combination of both), the following principles can be summarized:

- The friction equation must be satisfied for each pipe.
- There can be only one value of head at any point.
- Algebraic sum of the flow rates at any junction must be zero. i.e., the total mass flow rate towards the junction must be equal to the total mass flow rate away from it.
- Algebraic sum of the products of the flux (Q2) and the flow resistance (the sense being determined by the direction of flow) must be zero in any closed hydraulic circuit.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.465-558
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.368-415

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-28

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **IV-BOUNDARY LAYER** **Date of Lecture:**

Topic of Lecture:– Definition of boundary layer

Introduction : (Maximum 5 sentences)
Boundary layer, in fluid mechanics, thin layer of a flowing gas or liquid in contact with a surface such as that of an airplane wing or of the inside of a pipe.The flow in such boundary layers is generally laminar at the leading or upstream portion and turbulent in the trailing or downstream portion.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

<u>Boundary layer:</u>

Boundary layer, in fluid mechanics, thin layer of a flowing gas or liquid in contact with a surface such as that of an airplane wing or of the inside of a pipe. The flow in such boundary layers is generally laminar at the leading or upstream portion and turbulent in the trailing or downstream portion.

When a fluid rotates and viscous forces are balanced by the Coriolis effect), an Ekman layer forms. In the theory of heat transfer, a thermal boundary layer occurs. A surface can have multiple types of boundary layer simultaneously.

Boundary layer is formed:

The primary reason for the formation of the boundary layer is Coanda effect. It defines the tendency of a fluid jet to stay attached to a convex surface. This creates a thin layer of fluid near the surface in which the velocity changes from zero at the surface to the free stream value away from the surface.

Causes of Boundary Layer:

When a fluid stream encounters a solid surface that is at rest, the fluid velocity assumes a value of zero at that surface. The velocity then varies from zero at the surface to some larger value sufficiently far from the surface. The development of a boundary layer is caused by the no-slip condition.

Importance of boundary layer:

The boundary layer is a thin zone of calm air that surrounds each leaf. The thickness of the boundary layer influences how quickly gasses and energy are exchanged between the leaf and the surrounding air. A thick boundary layer can reduce the transfer of heat, CO2 and water vapor from the leaf to the environment.

Types of boundary layer:

Boundary layer visualization, showing transition from laminar to turbulent condition.Laminar boundary layers can be loosely classified according to their structure and the circumstances under which they are created. The thin shear layer which develops on an oscillating body is an example of a Stokes boundary layer, while the Blasius boundary layer refers to the well-known similarity solution near an attached flat plate held in an oncoming unidirectional flow and Falkner–Skan boundary layer, a generalization of Blasius profile.

The viscous nature of airflow reduces the local velocities on a surface and is responsible for skin friction. The layer of air over the wing's surface that is slowed down or stopped by viscosity, is the boundary layer. There are two different types of boundary layer flow: laminar and turbulent.

Laminar boundary layer flow

The laminar boundary is a very smooth flow, while the turbulent boundary layer contains swirls or "eddies." The laminar flow creates less skin friction drag than the turbulent flow, but is less stable. Boundary layer flow over a wing surface begins as a smooth laminar flow. As the flow continues back from the leading edge, the laminar boundary layer increases in thickness.

Turbulent boundary layer flow

At some distance back from the leading edge, the smooth laminar flow breaks down and transitions to a turbulent flow. From a drag standpoint, it is advisable to have the transition from laminar to turbulent flow as far aft on the wing as possible, or have a large amount of the wing surface within the laminar portion of the boundary layer. The low energy laminar flow, however, tends to break down more suddenly than the turbulent layer.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-28

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **IV-BOUNDARY LAYER** **Date of Lecture:**

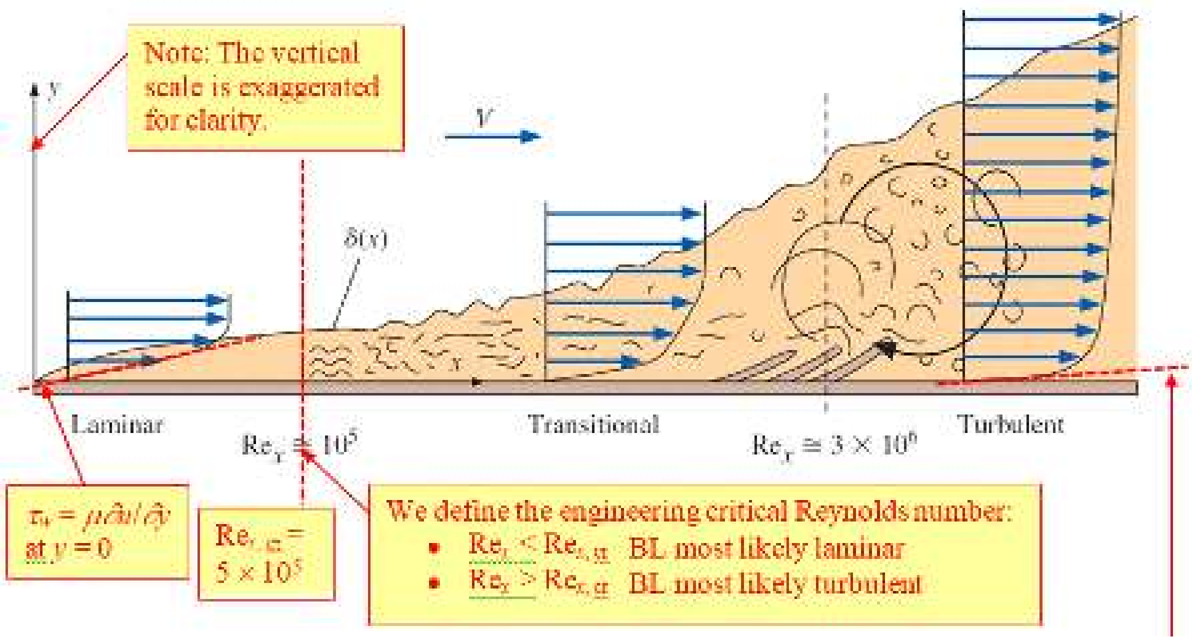
Topic of Lecture:– Boundary layer on a flat plate

Introduction : (Maximum 5 sentences)
Boundary Layer on a Flat Plate. , is unaffected by the edge's presence, and, is, therefore, the same as if the plate were of infinite length. Of course, the flow downstream of the edge is modified as a consequence of the finite length of the plate

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

The Turbulent Flat Plate Boundary Layer



The Turbulent Flat Plate Boundary Layer Velocity Profile:

The time-averaged turbulent flat plate (zero pressure gradient) boundary layer velocity profile is much fuller than the laminar flat plate boundary layer profile, and therefore has a larger slope $\partial u/\partial y$ at the wall, leading to greater skin friction drag along the wall.

The log law:

The log law:
$$\frac{u}{u_*} = \frac{1}{\kappa}\ln\frac{yu_*}{\nu} + B \qquad (10\text{–}83)$$

where

Friction velocity:
$$u_* = \sqrt{\frac{\tau_w}{\rho}} \qquad (10\text{–}84)$$

Column (b) expressions are generally preferred for engineering analysis.

TABLE 10–4

Summary of expressions for laminar and turbulent boundary layers on a smooth flat plate aligned parallel to a uniform stream*

Property	Laminar	(a) Turbulent[†]	(b) Turbulent[‡]
Boundary layer thickness	$\frac{\delta}{x} = \frac{4.91}{\sqrt{\mathrm{Re}_x}}$	$\frac{\delta}{x} \cong \frac{0.16}{(\mathrm{Re}_x)^{1/7}}$	$\frac{\delta}{x} \cong \frac{0.38}{(\mathrm{Re}_x)^{1/5}}$
Displacement thickness	$\frac{\delta^*}{x} = \frac{1.72}{\sqrt{\mathrm{Re}_x}}$	$\frac{\delta^*}{x} \cong \frac{0.020}{(\mathrm{Re}_x)^{1/7}}$	$\frac{\delta^*}{x} \cong \frac{0.048}{(\mathrm{Re}_x)^{1/5}}$
Momentum thickness	$\frac{\theta}{x} = \frac{0.664}{\sqrt{\mathrm{Re}_x}}$	$\frac{\theta}{x} \cong \frac{0.016}{(\mathrm{Re}_x)^{1/7}}$	$\frac{\theta}{x} \cong \frac{0.037}{(\mathrm{Re}_x)^{1/5}}$
Local skin friction coefficient	$C_{f,x} = \frac{0.664}{\sqrt{\mathrm{Re}_x}}$	$C_{f,x} \cong \frac{0.027}{(\mathrm{Re}_x)^{1/7}}$	$C_{f,x} \cong \frac{0.059}{(\mathrm{Re}_x)^{1/5}}$

Note that $C_{f,x}$ is the *local* skin friction coefficient, applied at only *one* value of x.

Laminar:
$$C_f = \frac{1.33}{\mathrm{Re}_L^{1/2}} \qquad \mathrm{Re}_L \lesssim 5\times 10^5 \qquad (11\text{–}19)$$

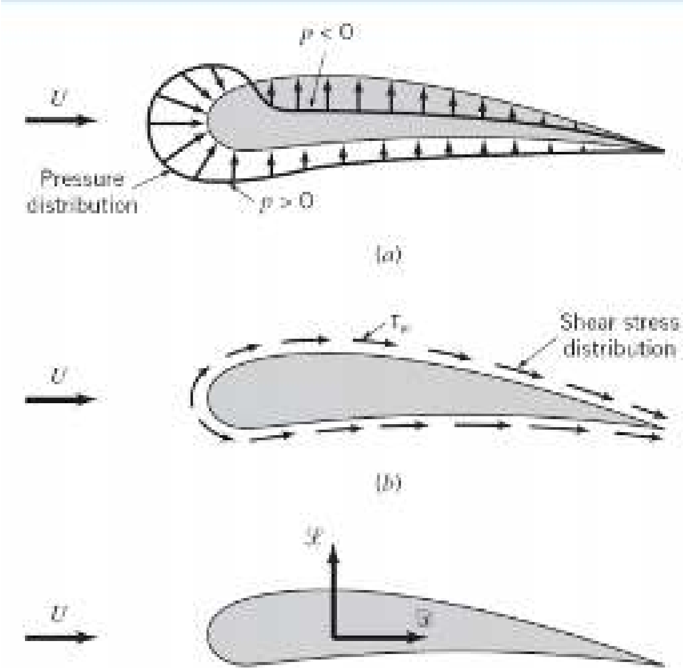
Turbulent:
$$C_f = \frac{0.074}{\mathrm{Re}_L^{1/5}} \qquad 5\times 10^5 \lesssim \mathrm{Re}_L \lesssim 10^7 \qquad (11\text{–}20)$$

Reynolds Number:

The skin-friction coefficients can be calculated using Reynolds number withthese equations a laminar or turbulent boundary layer, respectively.

Lift and Drag Concepts:

When any body moves through a fluid, an interaction between the body and the fluid occurs z This can be described in terms of the stresses-wall shear stresses due to viscous effect and normal stresses due to the pressure P.



Drag, D: the resultant force in the direction of the upstream velocity z

Lift, L: the resultant force normal to the upstream velocity

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

L-30

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : IV-BOUNDARY LAYER **Date of Lecture:**

Topic of Lecture:– - Thickness and classification- Displacement thickness

Introduction : (Maximum 5 sentences)

It is defined as the distance from the solid boundary measured in y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by δ

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Thickness and classification:

It is defined as the distance from the solid boundary measured in y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by δ

Various types of boundary layer thickness:

a. Displacement thickness(δ*),

b. Momentum thickness(θ),

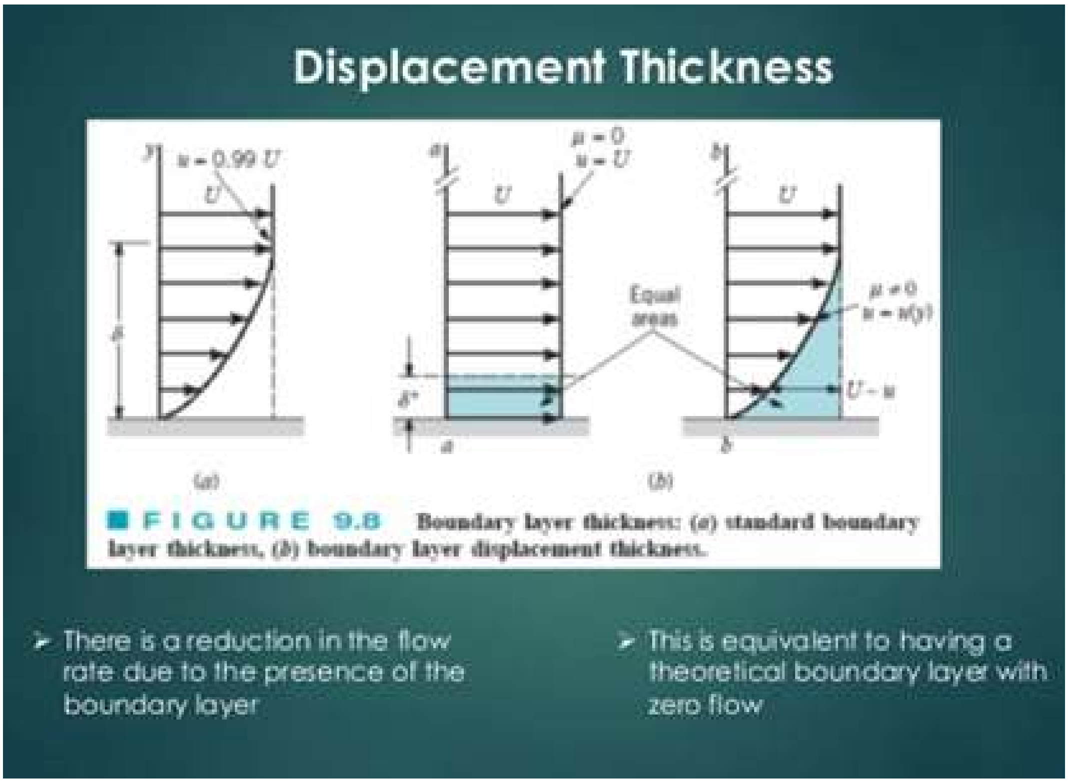
c. Energy thickness(δ**)

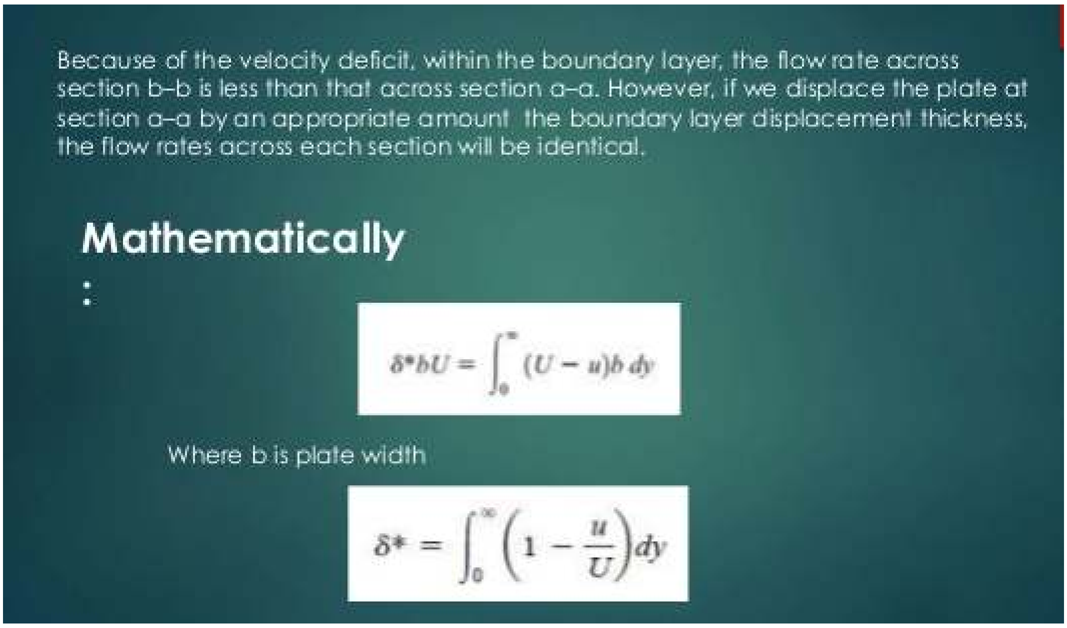
Displacement thickness:

The displacement thickness (δ) is defined as the distance by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation.

It is defined as the distance measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation. It is denoted by δ^*. it is also defined as :The distance perpendicular to the boundary by which the free stream is displaced due to the formation of boundary layer.

$$\delta^* = \int [\, 1 - (u/U) \,]\, dy$$





Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. 1Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

L-31

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : IV-BOUNDARY LAYER **Date of Lecture:**

Topic of Lecture:– - Thickness and classification- Momentum thickness

Introduction : (Maximum 5 sentences)

It is defined as the distance from the solid boundary measured in y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by δ

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

<u>Thickness and classification:</u>

It is defined as the distance from the solid boundary measured in y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by δ

Various types of boundary layer thickness:

a. Displacement thickness(δ*),

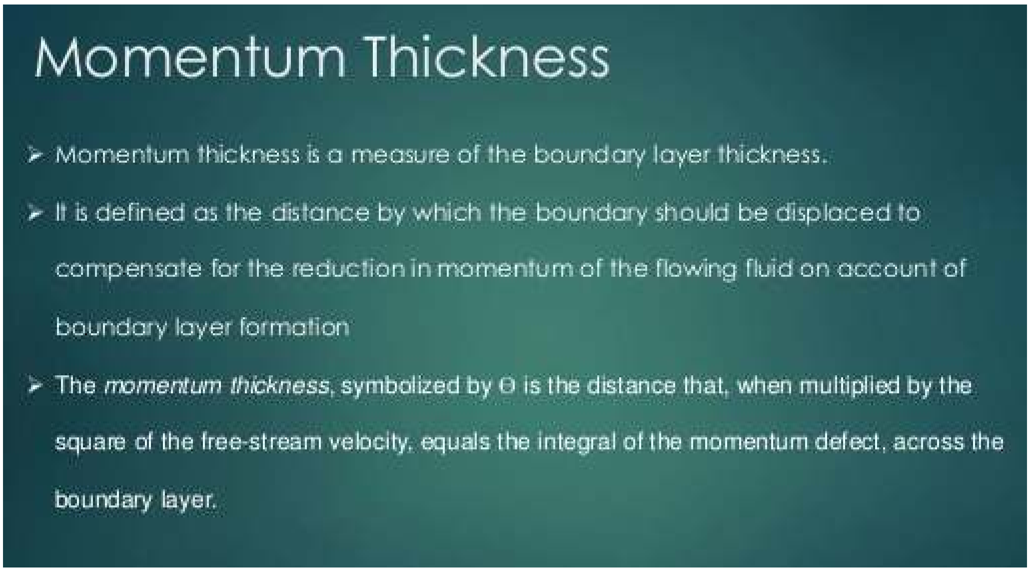
b. Momentum thickness(θ),

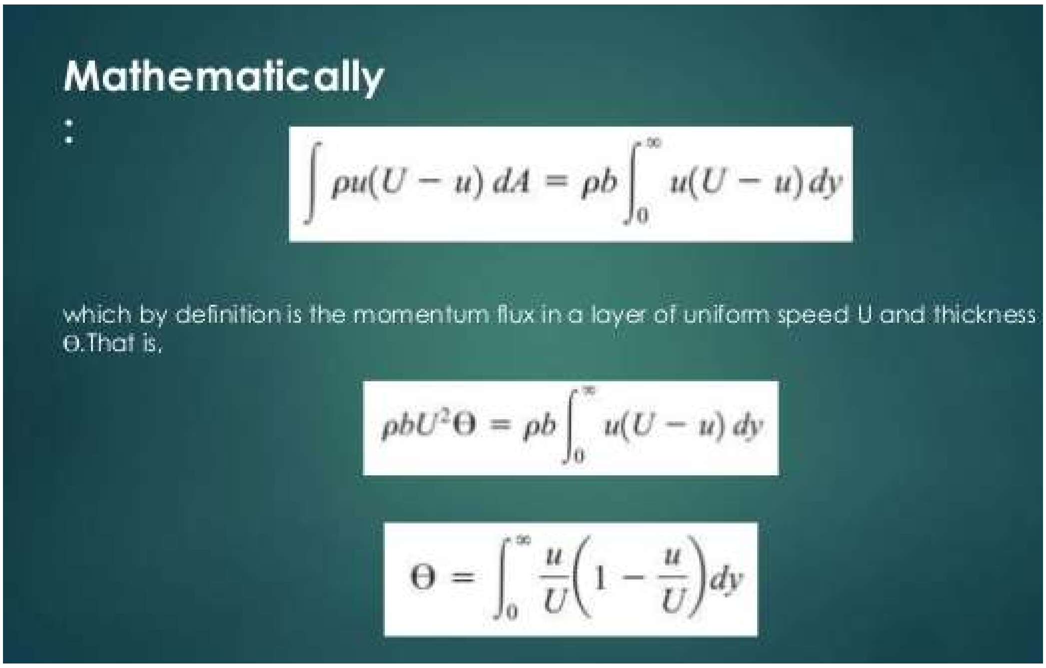
c. Energy thickness(δ**)

Momentum thickness:

The momentum thickness (θ) is defined as the distance by which the boundary should be displaced to compensate for the reduction in momentum of the flowing fluid on account of boundary layer formation. It is defined as the distance measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in flow rate on account of boundary layer formation. It is denoted by δ^*. It is also defined as the distance perpendicular to the boundary by which the free stream is displaced due to the formation of boundary layer.

$$\theta = \int [(u/U) - (u/U)^2] \, dy$$





Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

L-32

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **IV-BOUNDARY LAYER** **Date of Lecture:**

Topic of Lecture:– - Thickness and classification- Energy thickness(δ)**

Introduction : (Maximum 5 sentences)

It is defined as the distance from the solid boundary measured in y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by δ

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Thickness and classification:

It is defined as the distance from the solid boundary measured in y-direction to the point, where the velocity of fluid is approximately equal to 0.99 times the free stream velocity (U) of the fluid. It is denoted by δ

Various types of boundary layer thickness:

a. Displacement thickness(δ*),

b. Momentum thickness(θ),

c. Energy thickness(δ**)

Energy thickness:

It is defined as the distance measured perpendicular to the boundary of the solid body by which the boundary should be displaced to compensate for the reduction in kinetic energy of the flowing fluid on account of boundary layer formation. It is denoted by δ^{**}.

$$\delta^{**} = \int_0^{\delta} \frac{u}{U}\left[1 - \frac{u^2}{U^2}\right] dy$$

The energy thickness (δ**) is defined as the distance by which the boundary should be displaced to compensate for the reduction in kinetic energy of the flowing fluid on account of boundary layer formation.

The **Energy thickness**, is the distance by which a surface would have to be moved parallel to its normal vector towards the reference plane in an in viscosity fluid stream of velocity to give the same total kinetic energy as exists between the surface and the reference plane in a real fluid.The definition of the energy thickness for compressible flow is based on mass flow rate:The definition for incompressible flow can be based on volumetric flow rate, as the density is constant Where and are the density and velocity in the 'free stream' outside the boundary layer, and is the coordinate normal to the wall.

Shape factor:

A shape factor is used in boundary layer flow to determine the nature of the flow.where H is the shape factor, is the displacement thickness and θ is the momentum thickness. The higher the value of H, the stronger the adverse pressure gradient.

A high adverse pressure gradient can greatly reduce the Reynolds number at which transition into turbulence may occur.

Conventionally, $H = 2.59$ (Blasius boundary layer) is typical of laminar flows, while $H = 1.3$ - 1.4 is typical of turbulent flows.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-33

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : IV-BOUNDARY LAYER **Date of Lecture:**

Topic of Lecture:– Boundary layer separation and control

Introduction : (Maximum 5 sentences)

Whenever there is relative movement between a fluid and a solid surface, whether externally round a body, or internally in an enclosed passage, a boundary layer exists with viscous forces present in the layer of fluid close to the surface. Boundary layers can be either laminar or turbulent.It causes buffeting of aircraft structures and control surfaces.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

Separation boundary layer:

The loss of kinetic energy is recovered from the intermediate fluid layer in contact with the layer adjacent to solid surface through momentum exchange process. Thus the velocity of the layer goes on decreasing.

Along the length of the solid body, at a certain point a stage may come when the boundary layer may not be able to keep sticking to the solid body if it can't provide kinetic energy to overcome the resistance offer by the solid body, the boundary layer will be separated from the surface. This phenomenon is called boundary layer separation. The point on the body at which the boundary layer is on the verge of separation from the surface is called as the point of separation. **Effect of pressure gradient on the boundary layer separation:**

Effect of pressure gradient ($dpdx$) on the boundary layer separation can be explained by considering the flow over a curved surface. The area of flow decreases and hence velocity increases. This means that flow gets accelerated in this region.

Due to increase in the velocity, the pressure decreases in the direction of the flow and hence pressure gradient ($dpdx$) is negative.

Location of separation point:

The separation point is determined from the condition, $(\partial u/\partial y)_{y=0}=0$

For a given velocity profile, it can be determine whether the boundary layer has separated or verge of separation or will not separate from the following condition.

1.If $(\partial u/\partial y)_{y=0}$ is negative…the flow has separated.

2.If $(\partial u/\partial y)_{y=0} = 0$ the flow is on the verge of separation.

3.If $(\partial u/\partial y)_{y=0}$ is positive ….the flow will not separate or flow will remain attached with the surface.

Methods of preventing the separation of boundary layer:

When the boundary layer separates from the surface, a certain portion adjacent to the surface has a back flow and eddies are continuously formed in this region and hence continuous loss of energy takes place. Thus separation of boundary layer is undesirable and attempts should be made to avoid separation by various methods.

The following are the methods for preventing the separation of boundary layer:

1. Suction of the slow moving fluid by a suction slot.
2. Supplying additional energy from a blower.
3. Providing a bypass in the slotted wing.
4. Rotating boundary in the direction of flow.
5. Providing small divergence in a diffuser.
6. Providing guide-blades in a bend.
7. Providing a trip-wire ring in the laminar region for the flow over a sphere.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-34

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **IV-BOUNDARY LAYER** **Date of Lecture:**

Topic of Lecture:– Drag in flat plate

Introduction : (Maximum 5 sentences)

The friction coefficient represents the resistance to fluid flow over a flat plate. It is proportional to the drag force acting on the plate. The drag coefficient for a flat surface is equivalent to the mean friction coefficient.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Drag force of water:

A drag force is the resistance force caused by the motion of a body through a fluid, such as water or air. A drag force acts opposite to the direction of the oncoming flow velocity. This is the relative velocity between the body and the fluid.

Turbulent boundary layer on a flat plate:

The thickness of the boundary layer, drag force on one side of the plate and co-efficient of drag due to turbulent boundary layer on a smooth plate at zero pressure gradient are determined as incase of laminar boundary layer provider the velocity profile is known. Blasius on the basis of the experiment given the following velocity profile for a turbulent boundary layer.

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^n$$

Where n=1/7 for $R_e < 10^7$ but more than 5×10^5

$$\frac{u}{U} = \left(\frac{y}{\delta}\right)^{\frac{1}{7}}$$

The above equation is not applicable very near the boundary, where the thin laminar sub-layer of thickness δ` exist. Here velocity distribution is influenced by viscous effects.

The value of $\tau 0$ for flat plate is taken as

$$\tau_0 = 0.0225\ U^2 \left(\frac{\mu}{\rho \delta U}\right)^{\frac{1}{4}}.$$

Analysis of turbulent boundary layer:

(a).If reynold number is more than 5×105 and less than 107 the thickness of boundary layer and drag co-efficient are given as:

$$\delta = \frac{0.37x}{(R_{e_x})^{1/5}} \text{ and } C_D = \frac{0.072}{(R_{e_L})^{1/5}}$$

Where x=distance from the leading edge

Rex = reynold number for length x

Rel = reynold number at the end of the plate

(b).If reynold number is more than 107 but less than 109 , schlichting gave the empirical equation as

$$C_D = \frac{0.455}{\left(\log_{10} R_{e_L}\right)^{2.58}}$$

The friction coefficient represents the resistance to fluid flow over a flat plate. It is proportional to the drag force acting on the plate. The drag coefficient for a flat surface is equivalent to the mean friction coefficient.

The average modern automobile achieves a drag coefficient of between 0.25 and 0.3. SUVs, with their typically boxy shapes, typically achieve a Cd=0.35–0.45. The drag coefficient of a vehicle is affected by the shape of body of the vehicle.

The drag coefficient Cd is equal to the drag D divided by the quantity: density r times half the velocity V squared times the reference area A. The drag coefficient then expresses the ratio of the drag force to the force produced by the dynamic pressure times the area.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-35

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : IV-BOUNDARY LAYER **Date of Lecture:**

Topic of Lecture:– Drag and lift coefficients

Introduction : (Maximum 5 sentences)

The drag coefficient is always associated with a particular surface area. The drag coefficient of any object comprises the effects of the two basic contributors to fluid dynamic drag: skin friction and form drag. The drag coefficient of a lifting airfoil or hydrofoil also includes the effects of lift-induced drag.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Drag and lift coefficients:

- The average modern automobile achieves a drag coefficient of between 0.25 and 0.3. SUVs, with their typically boxy shapes, typically achieve a Cd=0.35–0.45. The drag coefficient of a vehicle is affected by the shape of body of the vehicle.
- Drag depends directly on the mass of the flow going past the aircraft. The drag also depends in a complex way on two other properties of the air: its viscosity and its compressibility. These factors affect the wave drag and skin frictions which are described above.
- The lift coefficient (CL) is a dimensionless coefficient that relates the lift generated by a lifting body to the fluid density around the body, the fluid velocity and an associated reference area.
- A lifting body is a foil or a complete foil-bearing body such as a fixed-wing aircraft.
- CL is a function of the angle of the body to the flow, its Reynolds number and its Mach number. The section lift coefficient cl refers to the dynamic lift characteristics of a two-dimensional foil section, with the reference area replaced by the foil chord.
- Lift coefficient may also be used as a characteristic of a particular shape (or cross-section)

of an airfoil. In this application it is called the section lift

Drag force on a flat plate due to boundary layer:

$$F_D = \int \Delta F_D = \int_0^L \tau_0 \times b \times dx.$$

Local co-efficient of drag:

$$C_D{}^* = \frac{\tau_0}{\frac{1}{2}\rho U^2}$$

Average co-efficient of drag:

$$C_D = \frac{F_D}{\frac{1}{2}\rho A U^2}$$

Minimum drag coefficient:

When the Reynolds number equal to 0.73×10^6, the experimental drag coefficient reduced to a minimum value, which is hereafter called the minimum drag coefficient (MDC).

Von karman momentum Integral equation:

Applying the basic integral conservation principles of mass and momentum to a length of boundary layer, ds, yields the Karman momentum integral equation that will prove very useful in quantifying the evolution of a steady, planar boundary layer, whether laminar or turbulent.

Develop approximations to the exact solution by eliminating negligible contributions to the solution using scale analysis. Identify and formulate the physical interpretation of the mathematical terms in solutions to fluid dynamics problems

It is known that von Karman's momentum equation yields better results for accelerated flow than for decelerated flow, and the position of the separation point determined by the von Karman momentum equation is usually located downstream of the separation point computed by an exact solution.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

L-36

LECTURE HANDOUTS

CIVIL | **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : IV-BOUNDARY LAYER **Date of Lecture:**

Topic of Lecture:– Drag and lift coefficients

Introduction : (Maximum 5 sentences)

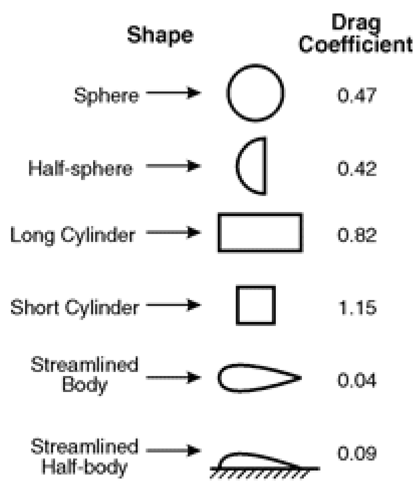
The drag coefficient is always associated with a particular surface area. The drag coefficient of any object comprises the effects of the two basic contributors to fluid dynamic drag: skin friction and form drag. The drag coefficient of a lifting airfoil or hydrofoil also includes the effects of lift-induced drag.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Drag and lift coefficients:

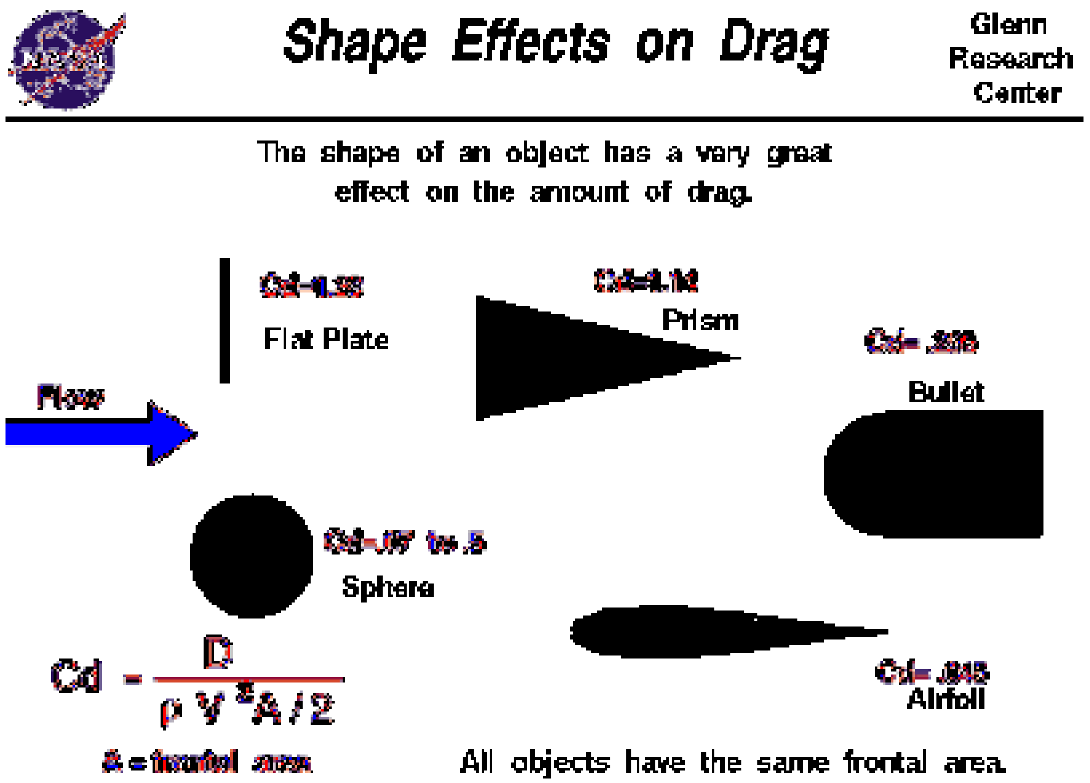
- A lifting body is a foil or a complete foil-bearing body such as a fixed-wing aircraft.
- CL is a function of the angle of the body to the flow, its Reynolds number and its Mach number. The section lift coefficient cl refers to the dynamic lift characteristics of a two-dimensional foil section, with the reference area replaced by the foil chord.



Measured Drag Coefficients

- Lift coefficient may also be used as a characteristic of a particular shape (or cross-section) of an airfoil. In this application it is called the section lift

<u>Drag Coefficient For Different Shapes:</u>



Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.611-656
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.425-513

Course Teacher

Verified by HOD

L-37

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty : Mr.M.Gopinath

Unit : V-SIMILITUDE AND MODEL STUDY

Date of Lecture:

Topic of Lecture: Dimensional analysis

Introduction : (Maximum 5 sentences)
Dimensional analysis is a mathematical technique used to predict physical parameters that influence the flow in fluid mechanics, heat transfer in thermodynamics, and so forth. The analysis involves the fundamental units of dimensions MLT: mass, length, and time.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)
- ✓ No Prerequisite knowledge required

Dimensional Analysis:

The systematic procedure of identifying the variables in aphysical phenomena and correlating them to form a set of dimensionless group isknown as dimensional analysis.

Dimensional analysis is a means of simplifying a physical problem by appealing todimensional homogeneity to reduce the number of relevant variables.

It is particularly useful for:

- ❖ Presenting and interpreting experimental data;
- ❖ Attacking problems not amenable to a direct theoretical solution;
- ❖ Checking equations;
- ❖ Establishing the relative importance of particular physical phenomena;
- ❖ Physical modelling.

Dimensional Homogeneity:

If an equation truly expresses a proper relationship amongvariables in a physical process, then it will be dimensionally homogeneous. Theequations are correct for any system of units and consequently each group of terms inthe equation must have the same dimensional representation. This is also known asthe law of dimensional homogeneity.

Dimensional variables: These are the quantities, which actually vary during a givencase and can be plotted against each other.

Dimensional constants: These are normally held constant during a given run. But,they may vary from case to case.

Pure constants: They have no dimensions, but, while performing the mathematical manipulation, they can arise.

DIMENSIONS:

Dimensions and Units:

A dimension is the type of physical quantity.A unit is a means of assigning a numerical value to that quantity.SI units are preferred in scientific work.

Primary Dimensions

In fluid mechanics the primary or fundamental dimensions, together with their SI units are:

- ❖ Mass M (kilogram, kg)
- ❖ Length L (metre, m)
- ❖ Time T (second, s)
- ❖ Temperature Θ (kelvin, K)

In other areas of physics additional dimensions may be necessary. The complete set specifiedby the SI system consists of the above plus

- ❖ Electric current I (ampere, A)
- ❖ Luminous intensity C (candela, cd)
- ❖ Amount of substance n (mole, mol)

Dimensions of Derived Quantities Dimensions of common derived mechanical quantities are given in the following table.

- ❖ A typical fluid mechanics problem typical fluid mechanics problem in which experimentation is required consider the experimentation is required consider the steady flow of an steady flow of anincompressible Newtonian fluid through a long, smooth incompressible Newtonian fluid through a long, smoothwalled, horizontal, circular pipe.
- ❖ An important characteristic of this system, which would be interest to an engineer designing a pipeline, is thepressure drop per unit length that develops along the pipeas a result of friction

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

L-38

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **V-SIMILITUDE AND MODEL STUDY**

Date of Lecture:

Topic of Lecture:Dimensional analysis

Introduction : (Maximum 5 sentences)
Dimensional analysis is a mathematical technique used to predict physical parameters that influence the flow in fluid mechanics, heat transfer in thermodynamics, and so forth. The analysis involves the fundamental units of dimensions MLT: mass, length, and time.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

Dimensional Analysis:

The systematic procedure of identifying the variables in physical phenomena and correlating them to form a set of dimensionless group isknown as dimensional analysis.

- ❖ The first step in the planning of an experiment to study this problem would be to decide the factors, or variables, that will have an effect on the pressure drop. ϖPressure drop Pressure dro p unit length $\Delta p = f (D , \rho , \mu , V) \lambda \rceil$ Pressure drop per unit length depends on FOUR variables: sphere size (D); speed (V); fluid density (sphere size (D); speed (V); fluid density density (ρ); fluid viscosity); fluid viscosity viscosity (m)
- ❖ To perform the experiments in a meaningful and systematic manner, it would be necessary to change the variable, such as the velocity, which holding all other constant, and measure the corresponding pressure drop. ϖ Difficulty to Difficulty to determine the functional relationship between determines the functional relationship between the pressure drop and the various facts that influence it.
- ❖ Fortunately, there is a much simpler approach to the problem that will eliminate the difficulties described above.

DIMENSIONS AND DERIVED QUANTITIES

Dimensions of common derived mechanical quantities are given in the following table.

	Quantity	Common Symbol(s)	Dimensions
Geometry	Area	A	L^2
	Volume	V	L^3
	Second moment of area	I	L^4
Kinematics	Velocity	U	LT^{-1}
	Acceleration	a	LT^{-2}
	Angle	θ	1 (i.e. dimensionless)
	Angular velocity	ω	T^{-1}
	Quantity of flow	Q	L^3T^{-1}
	Mass flow rate	$\dot{m}$	MT^{-1}
Dynamics	Force	F	MLT^{-2}
	Moment, torque	T	ML^2T^{-2}
	Energy, work, heat	E, W	ML^2T^{-2}
	Power	P	ML^2T^{-3}
	Pressure, stress	p, τ	$ML^{-1}T^{-2}$
Fluid properties	Density	ρ	ML^{-3}
	Viscosity	μ	$ML^{-1}T^{-1}$
	Kinematic viscosity	ν	L^2T^{-1}
	Surface tension	σ	MT^{-2}
	Thermal conductivity	k	$MLT^{-3}\Theta^{-1}$
	Specific heat	c_p, c_v	$L^2T^{-2}\Theta^{-1}$
	Bulk modulus	K	$ML^{-1}T^{-2}$

Alternative Choices For Primary Dimensions

The choice of primary dimensions is not unique. It is not uncommon – and it may sometimes be more convenient – to choose force F as a primary dimension rather than mass, and have a {FLT} system rather than {MLT}.

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-39

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath =

Unit : V-SIMILITUDE AND MODEL STUDY

Date of Lecture:

Topic of Lecture:Rayleigh's method

Introduction : (Maximum 5 sentences)
Rayleigh's method of dimensional analysis is a conceptual tool used in physics, chemistry, and engineering. This form of dimensional analysis expresses a functional relationship of some variables in the form of an exponential equation.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

METHODS OF DIMENSIONAL ANALYSIS

There are two methods of dimensional analysis used.

(i) Rayleigh's method

(ii) Buckingham π Theorem

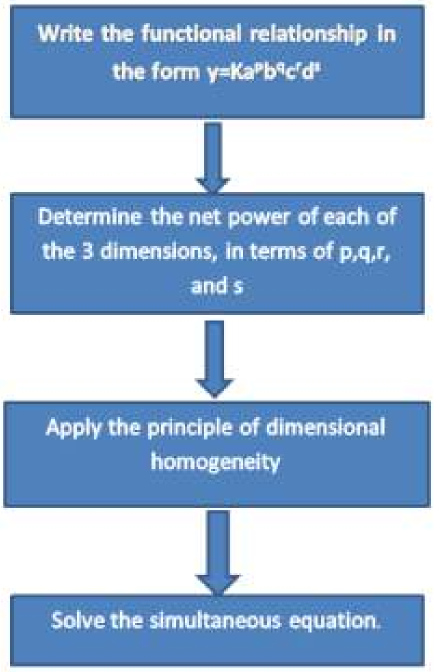
RAYLEIGH'S METHOD

In this method, the expression is determined for a variable depending upon maximum three or four variablesonly. If the number of independent variables becomes more than four, it is very difficult to find theexpression for the dependent variable. So, a functional relationship between variables is expressed inexponential form of equations.

The method involves the following steps:

- Gather all the independent variables that are likely to influence the dependent variable.
- If R is a variable that depends upon independent variables R1, R2, R3, ..., Rn, then the functional equation can be written as R = F(R1, R2, R3, ..., Rn).
- Write the above equation in the form R = C R1a R2b R3c ... Rnm, where C is a dimensionless constant and a, b, c, ..., m are arbitrary exponents.
- Express each of the quantities in the equation in some base units in which the solution is required.

- ➢ By using dimensional homogeneity, obtain a set of simultaneous equations involving the exponents a, b, c, ..., m.
- ➢ Solve these equations to obtain the value of exponents a, b, c, ..., m.
- ➢ Substitute the values of exponents in the main equation, and form the non-dimensional parameters by grouping the variables with like exponents.
- ➢ A basic method to dimensional analysis method and can be simplified to yield dimensionless groups controlling the phenomenon. Flow chart below shows the procedures.



Rayleigh Method has the following limitations:

1. The premise that an exponential relationship exists between the variables.
2. Rayleigh method Becomes laborious if variables are more than fundamental dimensions (MLT)
3. Rayleigh method doesn't provide any information regarding number of dimensionless groups to be obtained as a result of dimension analysis.
4. Rayleigh method is not always so straightforward. Consider the situation of flow over a U-notched weir.
5. When a large number of variables are involved, Rayleigh's method becomes lengthy

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-40

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty : **Mr.M.Gopinath**

Unit : **V-SIMILITUDE AND MODEL STUDY**

Date of Lecture:

Topic of Lecture:Buckingham's Pi-theorem method

Introduction : (Maximum 5 sentences)

Rayleigh's method of dimensional analysis is a conceptual tool used in physics, chemistry, and engineering. This form of dimensional analysis expresses a functional relationship of some variables in the form of an exponential equation.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

METHODS OF DIMENSIONAL ANALYSIS

There are two methods of dimensional analysis used.

(i) Rayleigh's method

(ii) Buckingham π Theorem

BUCKINGHAM'S Pi THEORY

Rayleigh method is not helpful when the number of independent variables is more than three or four.

This difficultyis eliminated in Buckingham π Theorem

It states that if there are 'n' variables in a dimensionally homogeneous equation and ifthese variables contain 'm' fundamental dimensions (M, L, T), then they are grouped into (n - m),dimensionless independent π-terms.

Let X1, X2, X3, ...,Xn are the variables involved in a physical phenomenon. Let X1 be the variables involved in a physical phenomenon. Let X1 be the dependent variables and X2, X3, ... Xn are the independent variables on which X1 depends.

Then X1 is a function of X2, X3, ... Xn andmathematically, it is expressed as

$$X1 = f(X2, X3, \ldots Xn) \text{ …….. (1)}$$

Equation (1) can also be written as

$$F_1 (X_1 , X_2 , X_3 , . . . X_n) = 0 \ldots\ldots\ldots (2)$$

This equation is a dimensionally homogeneous equation. It contains n variables. If thereare 'm' fundamental dimensions then according to Buckingham-π-theorem, equation (2) can bewritten interms in which number of π -terms is equal to (n - m). Hence, equation (2) becomes

$$F (\pi_1, \pi_2, ... \pi_{n-m}) = 0 \ldots\ldots\ldots (3)$$

Each of π terms is dimensionless and independent of the system. Division or multiplicationby a constant does not change the character of the π term. Each of π term contains (m +1)variables, where m is the number of Fundamental dimensions and is also called repeating variables.Let 'm' in the above case X2, X3 and X4 are repeating variables, if the fundamental dimensions(M, L, T) = 3 then each π term is written as

$$\pi_1 = X_2^{a1}, X_3^{b1}, X_4^{c1} . X^1$$

$$\pi_1 = X_2^{a2}, X_3^{b2}, X_4^{c2} . X^5$$

$$\pi_{n-m} = X_2^{an-m}, X_3^{bn-m}, X_4^{cn-m} . X^n \ldots\ldots\ldots (4)$$

Each equation is solved by the principle of dimensional homogeneity and values of a1, b1, c1etc. are obtained. These values are substituted in equation (4) and values of π1, π2, π3...... πn-m areobtained. These values of π's are substituted in equation (3). The final equation for thephenomenon is obtained by expressing any one of the π-terms as a function of others as

$$\Pi_1 = \varphi [\pi_2, \pi_3, \ldots\ldots \pi_{n-m}]$$

$$\Pi_2 = \varphi [\pi_1, \pi_3, \ldots\ldots \pi_{n-m}]$$

SELECTION OF REPEATING VARIABLES

There is no separate rule for selecting repeating variables. Hut the number of repeatingvariables is equal to the fundamental dimensions of the problem. Generally, ρ, υ, l or ρ, υ, D) arechoosen as repeating variables.

It means, one refers to fluid property (ρ), one refers to flow property (υ) and the other onerefers to geometric property (l or D). In addition to this, the following points should be kept inmind while selecting the repeating variables:

1. No variables should be dimensionless.
2. The selected two repeating variables should not have the same dimensions.
3. The selected repeating variables should be independent as far as possible.

STEPS TO BE FOLLOWED IN BUCKINGHAM Π METHOD

1. First, the variables involved in a given analysis are listed to study about given phenomenon thoroughly.
2. Then, these variables are expressed in terms of primary dimensions.
3. Next, the repeating variables are chosen according to the hint given in selection of repeating

variables. Once, the repeating variables should be checked either those are independent or dependent variables because all should be independent variables.

4. Then the dimensionless parameters are obtained by adding one at a time repeating variables.

5. The number of π-terms involved in dimensional analysis is calculated using, n - m = Number of π terms.

Where, n = Total number of variables involved in given analysis.

m = Number of fundamental variables.

6. Finally, each equation in exponential form is solved which means the coefficients of exponents are found by comparing both sides exponents. Then these dimensionless parameters are recombined and arranged suitably.

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

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-41

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **V-SIMILITUDE AND MODEL STUDY**

Date of Lecture:

Topic of Lecture:Similitude and models

Introduction : (Maximum 5 sentences)
The model is the small scale replica of the actual structure or machine. The actual structure or machine is called Prototype. It is not necessary that the models should be smaller than the prototypes (though in most of cases it is), they may be larger than the prototype.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)

- ✓ No Prerequisite knowledge required

MODEL ANALYSIS

- ❖ For predicting the performance of the hydraulic structures (such as dams, spillways etc.) or hydraulicmachines (such as turbines, pumps etc.), before actually constructing or manufacturing.
- ❖ Models of the structures or machines are made and tests are performed on them to obtain the desiredformation.
- ❖ The model is the small scale replica of the actual structure or machine. The actual structure or machineis called Prototype. It is not necessary that the models should be smaller than the prototypes (thoughin most of cases it is), they may be larger than the prototype.
- ❖ The study of models of actual machines is called Model analysis. Model analysis is actually an experimental method of finding solutions of complex flow problems. Exact analytical solutions arepossible only for a limited number of flow problems.

The followings are the advantages of the dimensional and model analysis:

- ❖ The performance of the hydraulic structure or hydraulic machine can be easily predicted, in advance,from its model.
- ❖ With the help of dimensional analysis, a relationship between the variables influencing a flow problemin terms of dimensionless parameters is obtained. This relationship helps in conducting

tests on themodel.

- ❖ The merits of alternative designs can be predicted with the help of model testing. The most economicaland safe design may be, finally, adopted.
- ❖ The tests performed on the models can be utilized for obtaining, in advance, useful information aboutthe performance of the prototypes only if a complete similarity exists between the model and theprototype.

SIMILITUDE-TYPES OF SIMILARITIES

Similitude is defined as the similarity between the model and its prototype in every respect, which meansthat the model and prototype have similar properties or model and prototype are completely similar.

Three types of similarities must exist between the model and prototype. They are

1 Geometric Similarity,

2. Kinematic Similarity, and

3. Dynamic Similarity.

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

Lm = Length of model,

bm = Breadth of model,

Dm - Diameter of model,

Am - Area of model,

Vm = Volume of model,

LP, bp, Dp, Ap, Vp = Corresponding values of the prototype,

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

$Lp/Lm = bp/bm = Dp/Dm =$ Lr

Lr is called the scale ratio

Kinematic Similarity:

- ➢ Kinematic similarity means the similarity of motion between model and prototype.
- ➢ Thus kinematic similarityis said to exist between the model and the prototype if the ratios velocity and acceleration at thecorresponding points in the model and at the corresponding points in the prototype are the same.
- ➢ Since velocity and acceleration are vector quantities, hence not only the ratio of magnitude of velocity andacceleration at the corresponding points in model and prototype should be same, but the directions ofvelocity and accelerations at the corresponding points in the model and prototype also should be parallel.
- ➢ All the direction of the velocities in the model and prototype should be same.

Dynamic Similarity:

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

TYPES OF FORCES ACTING IN MOVING FLUID

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

- Inertia force, Fi
- Viscous force, Fv
- Gravity force, Fg
- Pressure force, Fp
- Surface tension force, Fy
- Elastic force, Fe

Inertia force (fv):

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

Viscous Force (Fv):

It is equal to the product of shear stress (x) due to viscosity and surface area of the flow. It is present in fluidflow problems where viscosity is having an important role to play.

Gravity Force (Fg):

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

Pressure Force (Fp):

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

Surface Tension Force (Fs):

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

Elastic Force (Fe):

It is equal to the product of elastic stress and area of the flowing fluid.For a flowing fluid, the above-mentioned forces may not always be present. And also the forces, which arein a fluid flow problem, are not of equal magnitude. There are always one or two forces whichdominate the other forces. These dominating forces govern the flow of fluid.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

L-42

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **V-SIMILITUDE AND MODEL STUDY**

Date of Lecture:

Topic of Lecture:Similitude and models

Introduction : (Maximum 5 sentences)
The model is the small scale replica of the actual structure or machine. The actual structure or machine is called Prototype. It is not necessary that the models should be smaller than the prototypes (though in most of cases it is), they may be larger than the prototype.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

MODEL LAWS OR SIMILARITY LAWS

- For the dynamic similarity between the model and the prototype, the ratio of the corresponding forcesacting at the corresponding points in the model and prototype should be equal. The ratio of the forcesare dimensionless numbers.
- It means for dynamic similarity between the model and prototype, the dimensionless numbers shouldbe same for model and the prototype.
- But it is quite difficult to satisfy the condition that all the dimensionless numbers (i.e., Re, Fe, We, Euand M) are the same for the model and prototype. Hence models are designed on the basis of ratio ofthe force, which is dominating in the phenomenon.

The laws on which the models are designed for dynamic similarity are called model laws or laws of similarity.

The Followings are the model laws:

- Reynolds's model law
- Froude model law
- Euler model law
- Weber model law
- Mach model law

Reynolds's Model Law:

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

(i) Pipe flow

(ii) Resistance experienced by sub-marines, airplanes, fully immersed bodies etc.As defined earlier that Reynolds number is the ratio of inertia force and viscous force, and hence fluid flowproblems where viscous forces alone are predominant, the models are designed for dynamic similarity onReynolds law, which states that the Reynolds number for the model must be equal to the Reynolds numberfor the prototype.

Let

Vm = Velocity of fluid in model

ρm= Density of fluid in model

Lm = Length or linear dimension of the model

µm = Viscosity or fluid in model

And (Vm,ρm, Lm and µm) are the corresponding values of velocity, density, linear dimension andviscosity of fluid in prototype.

Then according to Reynolds's model law,

$$\mathbf{Re(p) = Re(m)}$$

Froude Model Law:

Froude model law is the law in which the models are based on Froude number which means for dynamicsimilarity between the model and prototype, the Froude number for both of them should be equal.Froude model law is applicable when the gravity force is only predominant force which controls the flowin addition to the force of inertia.

Froude model law is applied in the following fluid flow problems:

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

$$\mathbf{(Fe)\ model = (Fe)\ prototype}$$

Euler's Model Law:

- Euler's model law is the law in which the models are designed on Euler's number which means for dynamicsimilarity between the model and prototype, the Euler number for model arid prototype should be equal.
- Euler's model law is applicable when the pressure forces are alone predominant in addition to the inertiaforce. According to this law:

$$\mathbf{(Eu)model = (Eu)prototype}$$

- Euler's model law is applied for fluid flow problems where flow is taking place in a closed pipe in whichcase turbulence is fully developed so that viscous forces are negligible and gravity force and surfacetension force is absent. This law is also used where the phenomenon of cavitation takes place.

Weber Model Law:

- Weber model law is the law in which models are based on Weber's number, which is the ratio of thesquare root of inertia force to surface tension force.
- Hence where surface tension effects redominate in addition to inertia force, the dynamic similaritybetween these model and prototype is obtained by equating the Weber number of the model and itsprototype.Hence according to this law:

$$(We)_{model} = (We)_{prototype}$$

Weber model law is applied in following cases:

- Capillary rise in narrow passages
- Capillary movement of water in soil
- Capillary waves in channels
- Flow over weirs for small head

Mach Model Law:

- Mach model law is the law in which models are designed on Mach number, which is the ratio of the squareroot of inertia force to elastic force of a fluid.
- Hence where the forces due to elastic compression predominate in addition to inertia force, the dynamicsimilarity between the model and its prototype is obtained by equating the Mach number of the model andits prototype.Hence according to this law:

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

Mach model law is applied in the following cases:

- Flow of aeroplane and projectile through air at supersonic speed, ie., at a velocity more than the velocity of sound.
- Aerodynamic testing
- Under water testing of torpedoes
- Water hammer problems

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE

(An Autonomous Institution)

(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-43

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : **19CED07 / MECHANICS OF FLUIDS**

Course Faculty :**Mr.M.Gopinath**

Unit : **V-SIMILITUDE AND MODEL STUDY**

Date of Lecture:

Topic of Lecture:Scale effect

Introduction : (Maximum 5 sentences)
Scale effect occurs when a prototype hydraulic process is simulated at a laboratory scale due to dissatisfaction of similarity laws. It might lead to considerable deviation when the model scour depth is extrapolated to prototype value.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)
- ✓ No Prerequisite knowledge required

SCALE EFFECT

- ❖ Scale effect occurs when a prototype hydraulic process is simulated at a laboratory scale due to dissatisfaction of similarity laws. It might lead to considerable deviation when the model scour depth is extrapolated to prototype value.
- ❖ Three popular experimental approaches including predictionequation-targeted flume test, series-model test and similitude-model test are reviewed with emphasis upon their merits and limitations in reducing or alleviating scale effects.
- ❖ Scale laws guiding scour physical-model design are discussed for performing cost-effective model tests. An empirical equation is further derived from data of clear-water pier scour experiments to examine the test results in up-scale extrapolating.
- ❖ It is suggested that scale effect in a scour physical model test could be efficiently reduced if both the mobility similarity of bed particles and Froude number similarity are satisfied simultaneously.
- ❖ Scale effects arise due to force ratios which are not identical between a model and its real-world prototype and result in deviations between the up-scaled model and prototype observations.
- ❖ This review article considers mechanical, Froude and Reynolds model–prototype similarities,

describes scale effects for typical hydraulic flow phenomena and discusses how scale effects are avoided, compensated or corrected. Four approaches are addressed to obtain model–prototype similarity, to quantify scale effects and to define limiting criteria under which they can be neglected.

- ❖ The principal contribution of this study is a delineation of length-scale effects on the capacity of small hydraulic models, and micro-models, to simulate features of the flow field around a single dike in a flat-bed channel.
- ❖ Of particular interest in this regard are length-scale effects on flow-thalweg alignment around a dike and the extent of the flow-separation region downstream of a dike.
- ❖ Those flow features are important in channel-control activities, such as used to facilitate navigation in alluvial channels. The study's findings, though, are of direct relevance to the use of hydraulic models generally.
- ❖ These are inspectional analysis, dimensional analysis, calibration and scale series, which are applied to landslide generated impulse waves.
- ❖ Tables include both limiting criteria to avoid significant scale effects and typical scales of physical hydraulic engineering models for a wide variety of hydraulic flow phenomena.
- ❖ The article further shows why it is challenging to model sediment transport and distensible structures in a physical hydraulic model without significant scale effects. Possible future research directions are finally suggested.

SCALE EFFECTS:

Similitude limits admit scale effects whose consequences increase as model scales (prototype/model ratio) increase. For hydraulic modeling, the usual increases and distortions are as follow:

1. Large length scales, with increasing significance of viscous and surface-tension effects.
2. Distortion of vertical-length scale relative to horizontal-length scale.
3. Amplification of bed-sediment size, or bed roughness.
4. Exaggeration of channel slope.

Video Content / Details of website for further learning (if any):

https://www.youtube.com/watch?v=H8M8-SJAKTo

https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

L-44

LECTURE HANDOUTS

CIVIL **II/III**

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : V-SIMILITUDE AND MODEL STUDY

Date of Lecture:

Topic of Lecture:Distorted models

Introduction : (Maximum 5 sentences)
Distorted model is the model in which the scale ratio in the Cross section of the structure will not be the same i.e, (Hm/Hp) may not be equal to (Bm/Bp) [the word distortion means change in dimensions.

Prerequisite knowledge for Complete understanding and learning of Topic:
(Max. Four important topics)
- ✓ No Prerequisite knowledge required

CLASSIFICATION OF MODELS

Generally, hydraulic models are classified into two types.

- Undistorted models.
- Distorted models

1. UNDISTORTED MODELS

The model which is geometrically similar to its prototype is known as undistorted models. In such models,the conditions of similitude are fully satisfied. So, the results obtained from the model are used to predictthe performance of the prototype easily. Based on this, design,construction and interpretation of (he modelare simpler.

2. DISTORTED MODELS

A model which is not geometrically similar to its prototype but it may be similar in appearance with itsprototype. So, different scale ratios are used for linear dimensions such as length, breadth and height. Usually, the following distortions may occur in distorted models:

- Geometrical distortion.
- Material distortion.
- Distortion of hydraulic quantities

Geometrical distortion:

The distortion occurs either in dimensions or in configuration. It can be corrected by using different scalevalues for vertical and horizontal dimensions.

Material distortion:

It arises due to the use of different materials for the model and prototype. To avoid this, the same materialshave to be used as much as possible.

Distortion of hydraulic quantities:

Due to uncontrollable hydraulic quantities, the distortion may occur. Example: Velocity, discharge etc.

Reasons of adopting distorted models

- ❖ To maintain accuracy.
- ❖ To maintain turbulent flow.
- ❖ To accommodate available facilities
- ❖ To obtain suitable bed materials.
- ❖ To obtain required roughness condition.

Advantages of distorted models

- ❖ Accurate measurements can be possible.
- ❖ Surface tension can be minimized as much as possible.
- ❖ The operation is simplified due to small model size.
- ❖ Reynolds number of flow is increased sufficiently.

Disadvantages of distorted models

- ❖ Exit pressure and velocity distributions are not true.
- ❖ A model wave may differ from that of prototype.
- ❖ Both extrapolation and interpolation of results are difficult

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD

MUTHAYAMMAL ENGINEERING COLLEGE
(An Autonomous Institution)
(Approved by AICTE, New Delhi, Accredited by NAAC & Affiliated to Anna University)
Rasipuram - 637 408, Namakkal Dist., Tamil Nadu

L-45

LECTURE HANDOUTS

CIVIL

II/III

Course Name with Code : 19CED07 / MECHANICS OF FLUIDS

Course Faculty :Mr.M.Gopinath

Unit : V-SIMILITUDE AND MODEL STUDY

Date of Lecture:

Topic of Lecture:Distorted models

Introduction : (Maximum 5 sentences)
Distorted model is the model in which the scale ratio in the Cross section of the structure will not be the same i.e, (Hm/Hp) may not be equal to (Bm/Bp) [the word distortion means change in dimensions.

Prerequisite knowledge for Complete understanding and learning of Topic: (Max. Four important topics)

- ✓ No Prerequisite knowledge required

CLASSIFICATION OF MODELS

Generally, hydraulic models are classified into two types.

- Undistorted models.
- Distorted models
- Dimensional analysis is a method of dimensions. It is a mathematical technique used in research work for design and for conducting model tests. It deals with the dimensions of the physical quantities involved in the phenomenon.
- All physical quantities are measured by comparison, which is made with respect to an arbitrarily fixed value. Length L, mass M and time T are three fixed dimensions which are of importance in Fluid Mechanics.
- If in any problem of fluid mechanics, heat is involved then temperature is also taken as fixed dimension. These fixed dimensions are called fundamental dimensions or fundamental quantity.

DIMENSIONLESS NUMBERS:

Dimensionless numbers are those numbers .which are obtained by dividing the inertia force by viscous forceor gravity force or pressure force or surface tension force or elastic force. As this is a ratio of one force tothe other force, it will be a dimensionless number.

These dimensionless numbers also called non-dimensional parameters. The followings are the importantdimensionless numbers:

- Reynold's number
- Froude's number
- Euler's number
- Weber's number
- Mach's number

Reynold's Number (Re):

It is defined as the ratio of inertia force of a flowing fluid and r viscous force of the fluid. The expression for Reynold's number is obtained as

$$R_e = \sqrt{\frac{inertia\ force}{viscous\ force}}$$

$$Re = \frac{V*L}{v} \qquad [v = \frac{\mu}{\rho}]$$

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

$$R_e = \frac{V*d}{v} \quad (\text{ or }) \quad = \frac{\rho V d}{\mu}$$

Froude's Number (Fe):

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

$$Fe = \sqrt{\frac{inertia\ force}{gravity\ force}} = \sqrt{\frac{V}{Lg}}$$

Euler's Number (Eu):

It is defined as the square root of the ratio of the inertia force of a flowing fluid to the pressure force. Mathematically, it is expressed as,

$$E_u = \sqrt{\frac{inertia\ force}{pressure\ force}} = \frac{V}{\sqrt{\frac{p}{\rho}}}$$

Weber's Number (We):

It is defined as the square root of the ratio of the inertia force of a flowing fluid to the surface tension force. Mathematically, it is expressed as,

$$W_e = \sqrt{\frac{inertia\ force}{surface\ tension\ force}}$$

$$= \frac{V}{\sqrt{\frac{\sigma}{\rho L}}}$$

Mach's Number (M):

Mach's number is defined as the square root of the ratio of the inertia force of aflowing fluid to the elastic force. Mathematically, it is defined as,

$$M = \sqrt{\frac{inertia\ force}{elastic\ force}}$$

$$= \frac{M}{C}$$

Video Content / Details of website for further learning (if any):
https://www.youtube.com/watch?v=H8M8-SJAKTo
https://www.youtube.com/watch?v=2YHvzX6BagE

Important Books/Journals for further learning including the page nos.:

1. Fluid mechanics & Hydraulic machines By Bansal R.K in the page No.559-610
2. Hydraulics and Fluid Mechanics,By Modi P.N. and Seth S.M in the page No.520-586

Course Teacher

Verified by HOD