Mechanics of Fluids (R17A0362)

COURSE FILE

II B. Tech I Semester

(2018-2019)

Prepared By

Ms. Swetha Bala MNVS, Assoc. Prof

Department of Aeronautical Engineering

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

Affiliated to JNTU, Hyderabad, Approved by AICTE - Accredited by NBA & NAAC – 'A' Grade - ISO 9001:2015 Certified

Maisammaguda, Dhulapally (Post Via. Kompally), Secunderabad – 500100, Telangana State, India.

MRCET VISION

- To become a model institution in the fields of Engineering, Technology and Management.
- To have a perfect synchronization of the ideologies of MRCET with challenging demands of International Pioneering Organizations.

MRCET MISSION

To establish a pedestal for the integral innovation, team spirit, originality and competence in the students, expose them to face the global challenges and become pioneers of Indian vision of modern society.

MRCET QUALITY POLICY.

- To pursue continual improvement of teaching learning process of Undergraduate and Post Graduate programs in Engineering & Management vigorously.
- To provide state of art infrastructure and expertise to impart the quality education.

PROGRAM OUTCOMES
(PO's)

Engineering Graduates will be able to:

1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design / development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multi disciplinary environments.
12. **Life- long learning**: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

DEPARTMENT OF AERONAUTICAL ENGINEERING

VISION

Department of Aeronautical Engineering aims to be indispensable source in Aeronautical Engineering which has a zeal to provide the value driven platform for the students to acquire knowledge and empower themselves to shoulder higher responsibility in building a strong nation.

MISSION

The primary mission of the department is to promote engineering education and research. To strive consistently to provide quality education, keeping in pace with time and technology. Department passions to integrate the intellectual, spiritual, ethical and social development of the students for shaping them into dynamic engineers.

QUALITY POLICY STATEMENT

Impart up-to-date knowledge to the students in Aeronautical area to make them quality engineers. Make the students experience the applications on quality equipment and tools. Provide systems, resources and training opportunities to achieve continuous improvement. Maintain global standards in education, training and services.

PROGRAM EDUCATIONAL OBJECTIVES – Aeronautical Engineering

1. **PEO1 (PROFESSIONALISM & CITIZENSHIP):** To create and sustain a community of learning in which students acquire knowledge and learn to apply it professionally with due consideration for ethical, ecological and economic issues.
2. **PEO2 (TECHNICAL ACCOMPLISHMENTS):** To provide knowledge based services to satisfy the needs of society and the industry by providing hands on experience in various technologies in core field.
3. **PEO3 (INVENTION, INNOVATION AND CREATIVITY):** To make the students to design, experiment, analyze, and interpret in the core field with the help of other multi disciplinary concepts wherever applicable.
4. **PEO4 (PROFESSIONAL DEVELOPMENT):** To educate the students to disseminate research findings with good soft skills and become a successful entrepreneur.
5. **PEO5 (HUMAN RESOURCE DEVELOPMENT):** To graduate the students in building national capabilities in technology, education and research

PROGRAM SPECIFIC OUTCOMES – Aeronautical Engineering

1. To mould students to become a professional with all necessary skills, personality and sound knowledge in basic and advance technological areas.
2. To promote understanding of concepts and develop ability in design manufacture and maintenance of aircraft, aerospace vehicles and associated equipment and develop application capability of the concepts sciences to engineering design and processes.
3. Understanding the current scenario in the field of aeronautics and acquire ability to apply knowledge of engineering, science and mathematics to design and conduct experiments in the field of Aeronautical Engineering.
4. To develop leadership skills in our students necessary to shape the social, intellectual, business and technical worlds.

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

II Year B. Tech, ANE-I Sem	L	T/P/D	C
	4	1/-/-	4

R17A0362 - MECHANICS OF FLUIDS

Objectives:

- The student will gain insight into a number of potentially useful phenomena involving movement of fluids.
- He/she will learn to do elementary calculations for engineering application of fluid motion.
- This course also prepares the student for more advanced courses such as Aerodynamics- I & -II.

UNIT I

Fluid Properties: Density, specific weight, specific gravity, surface tension & capillarity, Newton's law of viscosity, incompressible & compressible fluid, numerical problems. Hydrostatic forces on submerged bodies: Pressure at a point, Pascal's law, pressure variation with temperature and height, Center of pressure on vertical, inclined and curved surfaces. Manometers- simple and differential manometers, inverted manometers, micro manometers, Pressure gauges and numerical problems. Buoyancy- Archimedes's Principle, Metacenter, Meta centric height calculations.

UNIT II

Fluid Kinematics: Stream line, path line, streak line, stream surface, stream tube, classification of flows: steady, unsteady, uniform, non uniform, laminar, turbulent flows. One dimensional approximation, examples of real 1-D flows, two dimensional approximation, 2-D flow in wind tunnel, continuity equations for 1-D and 2-D flows both compressible and incompressible, stream function for two dimensional incompressible flows. Vorticity, irrotational flow, Velocity potential function. Introduction to vortex flows.

UNIT III

Fluid Dynamics: Surface & body forces, substantive derivative, local derivative and convective derivative, momentum equation, Euler equation, Bernoulli's equation. Phenomenological basis of Navier-Stokes equation.
Flow measurements: pressure, velocity and mass flow rate, viscosity, Pitot-static tube, venturi meter and orifice meter, viscometers.
Flow of through pipes: major and minor losses.

UNIT IV

Boundary Layer: Introductory concepts of boundary layer, Large Reynolds number flows and Prandtl's boundary layer hypothesis, Qualitative description of Boundary layer thickness and velocity profile on a flat plate and forces due to laminar and turbulent boundary layer. Separation of boundary layer. Methods of preventing separation of boundary layer.

UNIT V

Dimensional and Model Analysis and Forces on submerged bodies:
Statement of Buckingham's π-theorem, Similarity parameters: Dimensionless numbers, Types of similarities, Similarity laws, Model testing and Classification of models. Forces exerted by a flowing fluid on a stationary body, Expressions for drag and lift.

Text Books:
1. Engineering Fluid mechanics – K.L . Kumar, S.Chand & Co.
2. Introduction to Fluid Mechanics and Fluid machines – S.K. Som and G. Biswas
3. Fluid Mechanics and Hydraulic Machines – RK Bansal, Laxmi Publications

Reference Books:
1. Fluid Mechanics – Frank M and White, Mc-Grawhill.
2. Fluid Mechanics- Fox and Mc Donald
3. Fluid Mechanics – E. Rathakrishnan

Outcomes:
- Students can define the governing equations of fluid flow problems.
- It makes the student ready to understand about aerodynamics.
- Students can able to create models for experimental analysis.

Lesson Plan (2018-2019 – I Semester)

UNIT	TOPIC	No. OF CLASSES	Total No. Of Classes
I FLUID PROPERTIES	**Introduction** **Fluid Properties:** Density, specific weight, specific gravity, surface tension & capillarity,	1	16
	Types of fluids and **Newton's law of viscosity** and	1	
	Numerical Problems on Fluid Properties (Viscosity)	2	
	Hydrostatic forces on submerged bodies: Pressure at a point, Pascal's law	1	
	Pressure variation with temperature and height: Center of Pressure and Total Pressure	1	
	Center of pressure on vertical, inclined and curved surfaces.	1	
	Problems on Hydrostatic forces	2	
	Pressure Measuring Devices: Manometers- simple and differential manometers, inverted manometers, micro manometers, Pressure gauges and numerical problems.	2	
	Buoyancy- Archimedes's Principle, Metacenter, Meta centric height calculations.	2	
	Assignment & revision	1	
	Test in Unit 1	1	
2 FLUID KINEMATICS	**Introduction:** Stream line, path line, streak line, stream surface, stream tube	1	12
	Classification of flows: steady, unsteady, uniform, non uniform, laminar, turbulent flows, One dimensional approximation	1	
	Examples of real 1-D flows, two dimensional approximation, 2-D flow in wind tunnel	1	
	Governing Equation: Continuity equation for 1-D and 2-D flows both compressible and incompressible,	1	
	Problems on Continuity equation (1 – D, 2 – D and 3-D)	2	
	Stream Function and Velocity Potential	1	
	Rotational and Irrotational Flows : Angular Velocity components	1	
	Vorticity **Problems on** Stream function, Velocity Potential Function and Vorticity	2	
	Introduction to Vortex Flows	1	
	Assignment & revision	**1**	
	Test in Unit - 2	**1**	

3 FLUID DYNAMICS	**Fluid Dynamics**: Surface & body forces,.	1	14
	Total or substantive derivative: local derivative and convective derivative, Problems on above topic	1	
	Governing equation: Momentum Equation	1	
	Governing equation: Euler equation, Bernoulli's equation. Phenomenological basis of Navier-Stokes equation	1	
	Problems on Bernoulli's Equation	1	
	Flow measurements: Velocity Measurements: Pitot – Static Tube	1	
	Viscosity Measurement: Viscometers	1	
	Mass flow rate: Venturimeter and Orifice Meter	1	
	Problems on Venturi and Orifice meters	2	
	Flow of through pipes: major and minor losses.	2	
	Assignment and revision	1	
	Test in Unit - 3	1	
4 BOUNDARY LAYER	**Boundary Layer**: Introductory concepts of boundary layer, Viscosity and Large Reynolds number flows and Prandtl's boundary layer hypothesis	2	11
	Qualitative description of Boundary layer thickness and expressions for Displacement thickness, Momemtum thickness and Energy thickness	2	
	Velocity profile on a flat plate and forces due to laminar and turbulent boundary layer.	1	
	Problems on Boundary Layer thickness calculation	2	
	Separation of boundary layer. Methods of preventing separation of boundary layer.	1	
	Assignment and revision	1	
	Test in Unit - 3	1	
5 DIMENSIONAL AND MODEL ANALYSIS AND FORCES ON SUBMERGED BODIES	**Dimensional Analysis**: Statement of Buckingham's π-theorem and Significance	1	8
	Similarity parameters: Dimensionless numbers	1	
	Types of similarities: Similarity laws	1	
	Model testing and Classification of models	1	
	Forces exerted by a flowing fluid on a stationary body	1	
	Expressions for drag and lift	1	
	Assignment and revision	1	
	Test in Unit 5	1	
			61

II B.TECH I SEMESTER – AERONAUTICAL ENGINEERING
MECHANICS OF FLUIDS (R17)
MODEL PAPER – I
MAXIMUM MARKS: 70

i. Answer only one question among the two questions in choice.
ii. Each question answer (irrespective of the bits) carries 10M.

1. a. Two plates are placed at a distance of 0.15 mm apart. The lower plate is fixed while the upper plate having the surface area of 1.0 m^2 is pulled at 0.3 m/s. Find the force and the power required to maintain this speed, if the fluid separating them is having viscosity 1.5 poise.
   b. Derive an expression for total pressure and center of pressure for a vertically plane surface immersed in a liquid.
   OR
2. Explain the working of a Bourdon tube pressure gauge using a neat sketch.

3. a. A 40 cm diameter pipe, conveying water, branches into two pipes of diameter 30 cm and 20 cm respectively. If the average velocity in the 40 cm pipe is 3 m/s, find the discharge in this pipe and also determine the velocity in 20 cm pipe. The average velocity in 30 cm pipe is 2 m/s.
   b. The two velocity components are given in the following cases, find the third component such that they satisfy the continuity equation for steady, incompressible flow.
   i. $u = x^3 + y^2 + 2z^2$; $v = -x^2y - yz - xy$
   ii. $u = 2y^2, w = 2xyz$
   OR
4. The velocity vector in a flow field is given as $\boldsymbol{V} = 4x^3\boldsymbol{i} - 10x^2y\boldsymbol{j} + 2t\boldsymbol{k}$. Find the velocity and acceleration of a fluid particle at (2,1,3) at time t = 1.

5. a. A pipe line carrying oil of specific gravity 0.8 changes in diameter from 300 mm at a position A to 500mm at position B which is 5m higher level. If the pressures at A and B are 19.6 N/cm^2 and 14.9 N/cm^2 respectively for a discharge of 150 liters per sec. Find the loss of head and the direction of flow.
   b. What is impulse momentum equation?
   c. What are the uses of dimensional analysis? Explain in brief.
   OR
6. What is the principle of orifice meter? Derive the expression for discharge through an orifice meter.

7. The velocity profile in a laminar boundary layer is given by $\frac{u}{V\infty} = 2\left(\frac{y}{\delta}\right) - 2(\frac{y}{\delta})^3 + \left(\frac{y}{\delta}\right)^4$. Find the expressions for boundary layer thickness, shear intensity and drag force on one side of the plate.
   OR

8. *A man weighing 90kgf descends to the ground from an airplane with the help of a parachute against the resistance of air. The velocity with which the parachute which is hemi – spherical in shape is 20 m/s downwards. Find the diameter of the parachute. Assume C_D = 0.5 and the density of air = 1.25 kg/m^3.*

9. *Derive the relation between shear stress and velocity for the laminar flow between two parallel plates while one plate is stationary and the other is moving with a uniform velocity.*

   OR

10. *Using a neat sketch, show the hydraulic gradient line and energy gradient line for the flow through a pipe. What is the significance and applications of the same?*

II B.TECH I SEMESTER – AERONAUTICAL ENGINEERING
MECHANICS OF FLUIDS (R17)
MODEL PAPER – II
MAXIMUM MARKS: 70

i. Answer only one question among the two questions in choice.
ii. Each question answer (irrespective of the bits) carries 10M.

1. a. Define the term Buoyancy. Explain using neat sketch, the conditions for equilibrium of a submerged body in fluid.
   b. Define Meta center and Meta centric height. Explain the analytical method for determining meta – centric height.
   OR
2. The opening in a dam is 3m wide and 2m high. A vertical sluice gate is used to cover the opening. On the upstream of the gate, the liquid of specific gravity 1.5 lies up to a height of 2m above the top of the gate, whereas on the down stream side, the water is available up to a height of top of the gate. Find the resultant force acting on the gate and the position of center of pressure. Assume that the gate is hinged at the bottom.

3. Derive the 3 – D continuity equation choosing a suitable flow model. Define all the symbols used while deriving it.
   OR
4. a. The 2 – D stream function for a flow is $\psi = 9 + 6x - 4y + 7xy$. Find the velocity potential.
   b. Diffentiate between Eulerian and Lagrangian methods of representing fluid flow.

5. a. A horizontal venturimeter with inlet and throat diameter 20 cm and 10 cm is used to measure the flow of oil of specific gravity 0.8. The discharge of oil through the venturimeter is 60 liters per sec. Find the reading of the oil – mercury differential manometer. Take C_d = 0.98
   b. A pipe of diameter 400mm carries water at a velocity of 25m/s. The pressure at the points A and B are given by 29.43 N/cm^2 and 22.563 N/cm^2 respectively, while the datum head at A and B are 28 m and 30 m respectively. Find the loss of head between A and B.
   OR
6. State Bernoulli's principle. Derive the Bernoulli's equation from Euler's equation of motion.

7. *Derive Von – Karman momentum integral equation.*
   OR
8. Explain boundary layer separation using a neat sketch and state the methods to avoid it.

9. a. *Determine the rate of flow of water through a pipe of diameter 10 cm and length 60 cm when one end of the pipe is connected to a tank and other end of the pipe is open to the atmosphere. The height of the water in the tank from the center of the pipe is 5 cm. The pipe is horizontal and the friction factor is 0.01. Consider minor losses.*

b. Determine the difference in the elevations in the water surfaces in the two tanks which are connected by a horizontal pipe of diameter 300 mm and length 400m. The rate of flow through the pipe is 300 liters per sec. Consider all the losses and take the value of f = 0.008.

OR

10. Derive the relation between shear stress and velocity for the laminar flow between two stationary parallel plates.

II B.TECH I SEMESTER – AERONAUTICAL ENGINEERING
MECHANICS OF FLUIDS (R13)
MODEL PAPER – III
MAXIMUM MARKS: 70

i. Answer only one question among the two questions in choice.
ii. Each question answer (irrespective of the bits) carries 10M.

1. a. Derive an expression for Total Pressure and Center of pressure for an inclined plane surface submerged in a liquid.
   b. An inclined rectangular gate of width 5 m and depth 1.5m is installed to control the discharge of water as shown in fig. **(3.55/127 Bansal)**. The end A is hinged. Determine the force normal to the gate applied at B to open it.
   OR
2. a. Water is flowing through two different pipes to which an inverted differential manometer having an oil of sp. Gr 0.8 is connected. The pressure head in the pipe A is 2m of wate, find the pressure in the pipe B for the manometer readings as shown in fig. **(2.22/54 Bansal)**
   b. A 150mm diameter vertical cylinder rotates concentrically inside another cylinder of diameter 151mm. Both the cylinders are of 250 mm height. The space between the cylinders is filled with a liquid of viscosity 10 poise. Determine the torque required to rotate the inner cylinder at 100 rpm.

3. a. What is a flow net? Describe the uses and limitations of a flow net.
   b. Check whether the flow defined by the stream function ψ = 2xy is irrotational. If so, determine the corresponding velocity potential.
   OR
4. For a steady incompressible flow, check the following values of u and v are possible or not.
   a. $u = 4xy + y^2, v = 6xy + 3x$
   b. $u = 2x^2 + y^2, v = -4xy$

5. a. What is meant by substantial derivative? Derive.
   b. Water is flowing through a pipe 5 cm diameter under a pressure of 29.43 N/cm^2 guage and with a mean velocity of 2m/s. Find the total head or total energy per unit weight of the water at a cross – section which is 5m above the datum line.
   OR
6. a. Differentiate between a model and a prototype.
   b. What are the conditions to be satisfied for both to be in dynamic similarity?
   c. What are the aims and objectives of model studies?

7. a. What is Magnus effect?
   b. Find the diameter of the parachute with which a man of 80 kg descends to the ground form an airplane against the resistance of air with a velocity 25 m/s. Take C_d = 0.5 and density of air = 1.25kg/m^3.

OR

8. Define boundary layer thickness, displacement thickness, momentum thickness and energy thickness. Explain the significance of each.

9. *The rate of flow of water pumped into a pipe ABC which is 200m long is 20 liters per sec. the pipe is laid on an upward slope of 1 in 40. The length of the portion AB is 100m and its diameter is 100mm, while the length of the portion BC is also 100 m but its diameter is 200 mm. The change of diameter at B is sudden. The flow is from A to C. the pressure at A is 19.62N/cm^2 and end C is connected to a tank. Find the pressure at C and draw the hydraulic gradient and total energy line. Take f = 0.008.*

OR

10. *Three pipes of same length L, diameter D and friction factor f are connected in parallel. Determine the diameter of the pipe of length L and friction f which will carry the same discharge for the same head loss. Use Darcy's – Weishbach equation.*

II B.TECH I SEMESTER – AERONAUTICAL ENGINEERING
MECHANICS OF FLUIDS (R17)
MODEL PAPER – IV
MAXIMUM MARKS: 70

i. Answer only one question among the two questions in choice.
ii. Each question answer (irrespective of the bits) carries 10M.

1. a. Explain the working of a U – tube and inverted U – tube manometer.

   b. Calculate the pressure and density of air at a height of 3000m above the sea level give the pressure and temperature at sea level as 10.413 N/cm^2 and 15^{0}C respectively. The temperature lapse rate is given as 0.0065^o K/m. The density of air at sea level is 1.285 kg/m^3.

   OR

2. a. Determine the Bulk modulus of elasticity of a liquid, if the pressure of the liquid is increased from 70 N/cm^2 to 130 N/cm^2. The volume of the liquid is decreased by 0.15 percent.

   b. The velocity distribution for flow over a flat plate is given by $u = \frac{3}{2}y - y^{3/2}$, where u is the point velocity in m/s at a distance *y* meter above the plate. Determine the shear stress at y = 9 cm. Assume dynamic viscosity as 8 poise.

3. Define and derive the expressions for local and convective accelerations.

   OR

4. a. What are the types of displacement that a fluid particle undergoes while in motion of a fluid? Explain using neat sketches.

   b. Water flows through a pipe AB 1.2m diameter at 3 m/s and then passes through a pipe BC 1.5 m diameter. At C, the pipe branches. The branch CD is 0.8 m diameter and carries one – third of the flow in AB. The flow velocity in branch CE is 2.5m/s. Find the volume rate of flow in AB, the velocity in CD and the diameter of CE.

5. 250 liters of water is flowing in a pipe having diameter 300 mm. If the pipe is bent by 135^0, find the magnitude and direction of the resultant force on the bend. The pressure of water flowing is 400 kN/m^2.

   OR

6. a. Explain the working of a capillary tube viscometer.

   b. Explain the working of a concentric – cylinder viscometer.

7. *Using neat sketches, explain the development of lift over a circular cylinder.*

   OR

8. For the velocity profile for laminar boundary layer $\frac{u}{V\infty} = \frac{3}{2}\left(\frac{y}{\delta}\right) - \frac{1}{2}\left(\frac{y}{\delta}\right)^3$. Determine the boundary layer thickness, shear stress, drag force.

9. *a. Three pipes of length 800m, 500m and 400m and of diameters of 500mm, 400mm and 300mm respectively are connected in series. These pipes are to be replaced by a single pipe of length 1700m. Find the diameter of the single pipe.*

   b. What is a siphon? What are its advantages?

   OR

10. *An oil of viscosity 0.1 N/m^2 and relative density 0.9 is flowing through a circular pipe of diameter 50mm and the length of 300m. The rate of flow of fluid through the pipe is 3.5 liters per second. Find the pressure drop in a length of 30 m and also the shear stress at the pipe wall.*

II B.TECH I SEMESTER – AERONAUTICAL ENGINEERING
MECHANICS OF FLUIDS (R13)
MODEL PAPER – V
MAXIMUM MARKS: 70

i. Answer only one question among the two questions in choice.
ii. Each question answer (irrespective of the bits) carries 10M.

1. a. Define vapor pressure and its effects.
   b. State and prove Hydrostatic law.
   c. Explain the variation of viscosity of fluids with temperature.
   OR
2. a. A shaft of diameter 100 mm is rotating inside a journal bearing of diameter 102mm at a pace of 360 rpm. The space between the shaft and the bearing is filled with a lubricating oil of viscosity 5 poise. The length of the bearing is 200mm. Find the power absorbed in the lubricating oil.
   b. A cubical tank has sides of 1.5m. It contains water for the lower 0.6m depth. The upper remaining part is filled with oil of specific gravity 0.9. Calculate for one vertical side of the tank: i. total pressure and (ii) position of center of pressure.

3. A flow field is given by $\boldsymbol{V} = x^2y\boldsymbol{i} + y^2z\boldsymbol{j} - (2xyz + yz^2)\boldsymbol{k}$. Prove that it is a possible case of steady incompressible flow. Calculate the velocity and acceleration at point (2,1,3).
   OR
4. a. Differentiate rotational and irrotational flows. Define vorticity. What are the properties of velocity potential function?
   b. The velocity components in a 2D flow are given by $u = \frac{y^3}{3} + 2x - x^2y$ and $v = xy^2 - 2y - \frac{x^3}{3}$. Show that these components represent a possible case of an irrotational flow.

5. a. Explain the working of a pitot – static tube and mention its purpose.
   b. A pitot – tube is inserted in a pipe of 300 mm diameter. The static pressure in pipe is 100 mm of Hg vacuum. The stagnation pressure at the center of the pipe, recorded by the tube is 0.981 N/cm^2. Calculate the rate of flow of water through the pipe, if the mean velocity of the flow is 0.85 times the central velocity. Take $C_v = 0.98$.
   OR
6. A nozzle of diameter 20 mm is fitted into a pipe of 40 mm. Find the force exerted by the nozzle on the water which is flowing through the pipe at the rate of 1.2 m^3/minute.

7. *Derive the expressions for drag and lift on an arbitrary shaped body place in a uniform field.*
   OR
8. *The velocity distribution in the boundary layer is given by $\frac{u}{V\infty} = \frac{y}{\delta}$. Show that the displacement thickness is δ times the momentum thickness.*

9. *a. Determine the pressure gradient, the shear stress at the surface of the plates and the discharge per meter with for the laminar flow of oil having maximum velocity of 2m/s between tow horizontal parallel fixed plates which are 100 mm apart. Given μ = 2.4525 N-s/m^2.*

   For the problem above, sketch the shear stress and velocity profile.

   OR

10. *a. Define equivalent pipe using necessary illustrations for pipes in series and parallel.*
    b. An old water supply distribution pipe of 250 mm diameter of a city is to be replaced by two parallel pipes of smaller equal diameter having equal lengths and identical friction factor values. Find out the new diameter required.

PREVIOUS QUESTION PAPERS

***** The question paper pattern has changed from R15 to R17 Regulations. Please note that the pattern given in the following papers is R15 pattern***

R15

Code No: R15A0362

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

II B.Tech I Semester Regular/Supplementary Examinations, November 2017

Mechanics of Fluids

(AE)

Roll No										

Time: 3 hours **Max. Marks: 75**

Note: This question paper contains two parts A and B

Part A is compulsory which carriers 25 marks and Answer all questions.

Part B Consists of 5 SECTIONS (One SECTION for each UNIT). Answer FIVE Questions, Choosing ONE Question from each SECTION and each Question carries 10 marks.

PART – A (25 Marks)

1. (a) Define center of pressure. What are the cases for submerged surfaces to determine the total pressure fore and center of pressure? 2 M

(b) Define the terms gauge, vacuum and absolute pressure. A fresh water lake has a maximum depth of 60 m and the mean atmospheric pressure is 91 kPa. Estimate the absolute pressure in kPa at this maximum depth. 3 M

(c) Differentiate between a stream line and a streak line? 2 M

(d) The velocity component in a 2-D flow field for an incompressible fluid are expressed as

$$u = \frac{y^3}{3} + 2x - x^2y \; ; \; v = xy^2 - \frac{x^3}{3} - 2y$$

Obtain an expression for stream function ψ. 3 M

(e) What are the forces acting on fluid in motion? 3 M

(f) Explain the concept of flow through nozzles? 2 M

(g) Define laminar boundary layer and turbulent boundary layer? 2 M

(h) Define momentum thickness and energy thickness with formula? 3 M

(i) Name four points how repeating variables are selected. 2 M

(j) A flat plate of size 2x3 m is submerged in water flowing with velocity of 6 m/s. Find drag and lift if $C_D = 0.04$ and $C_L = 0.2$. 3 M

PART – B (50 Marks)

SECTION – I

2. a) Explain the center of buoyance and metacenter. 4 M

b) Explain briefly i) Newton's law of viscosity

ii) Newtonian and Non Newtonian fluids

iii) Surface tension 6 M

(OR)

3. Explain and differentiate the types of manometers?

SECTION – II

4. a) Prove that velocity potential exists only for irrotational flows of fluids? 5 M
   b) Differentiate between laminar, transient and turbulent flows? 5 M

(OR)

5. a) Derive an expression for 2-D continuity equation for compressible and incompressible flows in Cartesian co-ordinates? 5 M
   b) Write a short note on classification of fluids? 5 M

SECTION – III

6. a) Derive Euler equation and from that Bernoulli's equation? 6 M
   b) Explain the working principle of pitot tube? Derive the equation for measuring the velocity of flow passing through pipe using pitot tube? 4 M

(OR)

7. a) A pipe line carrying oil of specific gravity 0.87 changes in diameter from 200 mm at a position A to 500 mm diameter a position B which is 4 m at high level. If pressure at position A and B are 1.01 bar and 0.6 bar respectively and the discharge is 200 liters/second, determine the loss of head and direction of flow? 6 M
   b) State the reasons for difference in the C_d value in venture meter and orifice meter. 4 M

SECTION – IV

8. a) Explain the boundary layer growth and its characteristics along the thin flat plate with neat diagram? 6 M
   b) For a laminar flow of oil having a dynamic viscosity $\mu = 1.76$ Pa. s in a 0.3 diameter pipe the velocity distribution is parabolic with a maximum point velocity of 3 m/s at a center of pipe. Calculate shear stresses in pipe? 4 M

(OR)

9. Discuss the methods of controlling the separation of boundary layer?

SECTION – V

10. a) How do you ensure that model and prototype are similarly developed. 5 M
    b) Explain the dimensionless numbers. 5 M

(OR)

11. Derive lift and drag forces acting on a stationary body submerged in a moving fluid.

*****

R15

Code No: R15A0362

MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY

(Autonomous Institution – UGC, Govt. of India)

II B. Tech I Semester Supplementary Examinations, May 2018

Mechanics of Fluids

(AE)

Roll No										

Time: 3 hours **Max. Marks: 75**

Note: This question paper contains two parts A and B

Part A is compulsory which carriers 25 marks and Answer all questions.

Part B Consists of 5 SECTIONS (One SECTION for each UNIT). Answer FIVE Questions, Choosing ONE Question from each SECTION and each Question carries 10 marks.

***

PART – A (25 Marks)

12. (a) Differentiate between compressible and incompressible fluid. 2 M

(b) Define ideal fluid and State Newton's law of viscosity . 3 M

(c) Define the rotational and irrotational flow? Write down the condition for irrotational flow? 2 M

(d) Calculate the unknown velocity component so that they satisfy the continuity equitation 3 M

$$u = (2x^2 + 2xy); w = (z^3 - 4xz - 2yz); v = ?$$

(e) Write the assumptions involved in deriving the Bernoulli's equation. 2 M

(f) What are the basic principles used in the analysis of problems of fluid in motion? Write a short note on them. 3 M

(g) Write down the difference between laminar boundary layer and laminar sub-layer? 2 M

(h) Draw the velocity profile in adverse pressure gradient conditions for the flow over a flat plate? Show the conditions involved. 3 M

(i) Define drag and lift. 2 M

(j) What is similitude and write the types of similarities. 3 M

PART – B (50 Marks)

SECTION – I

13. a) Derive an expression for total pressure force and center of pressure for vertical plane surface submerged in liquid. 5 M

b) Determine the total pressure acting on a circular plate of diameter 1.5 m which is placed in water in such a way that center of the plate is 3 m below the free surface of the water. Also, find the position of center of pressure. 5 M

(OR)

14. a) State and prove Pascal's law. 5 M
    b) Differentiate between kinematic and dynamic viscosity. 5 M

SECTION – II

15. a) Prove that stream lines and equi-potential lines are perpendicular to each other. 5 M
    b) Define and distinguish between laminar flow and turbulent flow, uniform flow and non-uniform flow? 5 M

(OR)

16. a) A stream function follow the law $\psi = 4x^2 - 4y$
    Obtain the velocity potential function. 7 M
    b) What do you understand by rotational and vortex flow? 3 M

SECTION – III

17. a) Differentiate between total, local and convective accelerations with one example. 5 M
    b) Explain what do you understand by the terms major and minor energy losses in pipe? 5 M

(OR)

18. a) Explain any two applications of Bernoulli's principle? 4 M
    b) Explain the working of pitot tube. Write the application of pitot tube in aerospace Industry. 6 M

SECTION – IV

19. a) Draw a neat sketch to explain boundary layer growth over a flat plate at zero angle of attack. 5 M
    b) Explain the phenomenon of boundary layer separation? 5 M

(OR)

20. If $\frac{u}{U} = \frac{3}{2}\left(\frac{y}{\delta}\right) - \frac{1}{2}\left(\frac{y}{\delta}\right)^2$, find shear stress, boundary layer thickness and drag coefficient from the fundamental equations of the boundary layer? 10M

SECTION – V

21. a) Explain the forces developed by moving fluid on stationary body. 5 M
    b) A vehicle projected with an area of 6.5 square meter moving at 70 km/hour has a total resistance of 2000 N. of this 25 per cent is due to rolling friction and 5 per cent due to surface friction. The remaining is due to form drag. Calculate the co-efficient of form drag with a density of 1.25 kg/m^3. 5 M

(OR)

22. a) State Buckingham's π-theorem with an example. 5 M
    b) Briefly explain why dimensional analysis is required. 5 M

****

UNIT – I

Key Topics: Fluid – definition, distinction between solid and fluid - Units and dimensions - Properties of fluids -density, specific weight, specific volume, specific gravity, temperature, viscosity, compressibility, vapour pressure, capillarity and surface tension - Fluid statics: concept of fluid statics: Concept of fluid static pressure, absolute and guage pressures, pressure measurement by manometers, and pressure gauges- forces on planes – centre of pressure – bouncy and floatation

1. FLUID MECHANICS

Fluid dynamics is "the branch of applied science that is concerned with the movement of liquids and gases,". Fluid dynamics is one of two branches of fluid mechanics, which is the study of fluids and how forces affect them. (The other branch is fluid statics, which deals with fluids at rest.)

The movement of liquids and gases is generally referred to as "flow," a concept that describes how fluids behave and how they interact with their surrounding environment — for example, water moving through a channel or pipe, or over a surface. Flow can be either steady or unsteady."If all properties of a flow are independent of time, then the flow is steady; otherwise, it is unsteady." *(Reference: "Lectures in Elementary Fluid Dynamics" (University of Kentucky, 2009) J. M. McDonough)*. An example of steady flow would be water flowing through a pipe at a constant rate. On the other hand, a flood or water pouring from an old-fashioned hand pump are examples of unsteady flow.

The gas most commonly encountered in everyday life is air; therefore, scientists have paid much attention to its flow conditions. Wind causes air to move around buildings and other structures, and it can also be made to move by pumps and fans.

One area of particular interest is the movement of objects through the atmosphere. This branch of fluid dynamics is called aerodynamics, which is "the dynamics of bodies moving relative to gases, especially the interaction of moving objects with the atmosphere," according to the American Heritage Dictionary. Problems in this field involve reducing drag on automobile bodies, designing more efficient aircraft and wind turbines, and studying how birds and insects fly.

For further information refer: https://www.livescience.com/47446-fluid-dynamics.html

STATES OF MATTER

There are three states of matter: solids, liquids and gases.

- Both liquids and gases are classified as fluids.

- Fluids do not resist a change in shape. Therefore fluids assume the shape of the container they occupy.

- Liquids may be considered to have a fixed volume and therefore can have a free urface.

Liquids are almost incompressible.

- Conversely, gases are easily compressed and will expand to fill a container occupy.

- We will usually be interested in liquids, either at rest or in motion.

Definition: The strict definition of a fluid is: *A fluid is a substance which conforms continuously under the action of shearing forces.*

If we apply a shear force to a fluid it will deform and take up a state in which no shear force exists. Therefore, we can say: *If a fluid is at rest there can be no shearing forces acting and therefore all forces in the fluid must be perpendicular to the planes in which they act.*

Note here that we specify that the fluid must be at rest. This is because, it is found experimentally that fluids in motion can have slight resistance to shear force. This is the source of **Viscosity.**

Ideal Fluids ad real Fluids:

An ideal fluid is a fluid that has several properties including the fact that it is:

- Incompressible – the density is constant
- Irrotational – the flow is smooth, no turbulence
- Nonviscous –(Inviscid) fluid has no internal friction ($\eta = 0$)

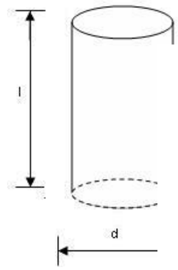
Real fluid: Fluid that have viscosity($\mu > 0$) and their motion known as viscous flow.

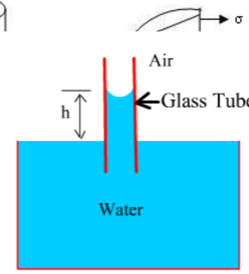
All the fluids in actual practice are real fluids.

2. Fluid properties:

Density: Mass per unit volume is defined as Density (also called as Mass Density). The density of liquid may be considered as constant while that of gases changes with the variation of pressure and temperature.

$$density = \rho = \frac{mass\ of\ fluid}{volume\ of\ fluid} = \frac{m}{V}\ \text{kg/m}^3$$





Specific Weight: Weight per unit volume of fluid is called as Specific Weight (also called as Weight density).

$$\gamma = \frac{weight\ of\ fluid}{volume\ of\ fluid} = \frac{m * g}{V}$$

$$\therefore \gamma = \rho g \quad \text{N/m}^3$$

Where, g is the acceleration due gravity.

Specific Gravity: Specific Gravity is defined as the ratio of the density of a substance to the density of some standard substance at a specified temperature.

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

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

Surface Tension: Surface tension is defined as the force at right angle to any line of unit length in the surface formed by two immiscible liquids. It is denoted by the Greek letter σ (sigma). In MKS units, it is expressed as kgf/m while in SI units as N/m.

Surface tension on liquid droplet:

Figure: surface tension

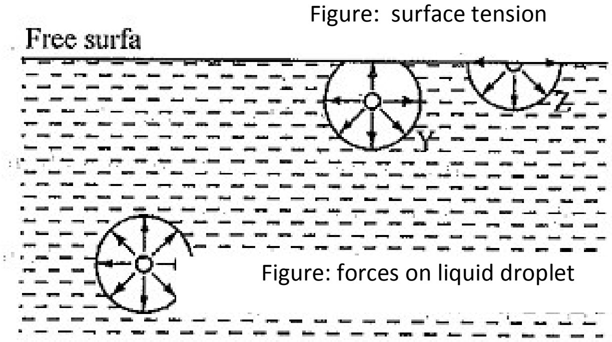


Figure: forces on liquid droplet

Let, σ = Surface tension of the liquid

p = Pressure intensity inside the droplet

d = Dia. of droplet

$$\sigma = \frac{pd}{4}$$

Surface tension on a hollow bubble:

$$\sigma = \frac{pd}{8}$$

Surface tension on a liquid jet:

$$\sigma = \frac{p \times d \times L}{2L}$$

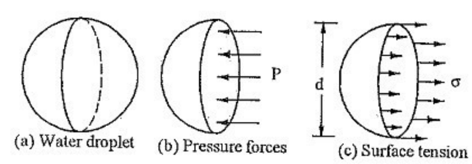


Figure: surface tension on a liquid jet

Capillarity: Capillarity is defined as a phenomenon of rise or fall of the liquid surface relative to the adjacent level of the fluid is called capillarity

Expression for capillary rise or fall:

If θ_c is the angle of contact between liquid and solid, d is the tube diameter, we can determine the capillary rise or depression, h by equating force balance in the z-direction considering surface tension, gravity and pressure. Since the column of fluid is at rest, the sum of all of forces acting on the fluid column is zero.

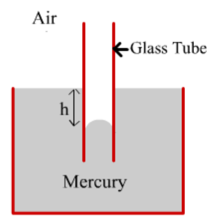
Figure: capillary rise

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

Weight of the liquid column $= \rho g \pi \frac{d^2}{4} h$

Thus, equating the above two equations we get

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

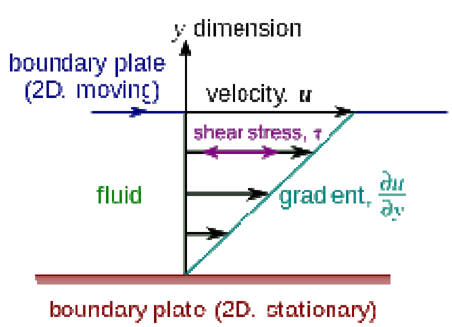


EXERCISE PROBLEMS ON ABOVE PROPERTIES WILL BE DEALT IN THE CLASS

VISCOSITY:

The **viscosity** of a fluid is the measure of its resistance to gradual deformation by shear stress or tensil stress.[1] For liquids, it corresponds to the informal concept of "thickness": for example, honey has a higher viscosity than water.

Viscosity is the property of a fluid which opposes the relative motion between two surfaces of the fluid that are moving at different velocities. In simple terms, viscosity means friction between the molecules of fluid. When the fluid is forced through a tube, the particles which compose the fluid generally move more quickly near the tube's axis and more slowly near its walls; therefore some stress (such as a pressure difference between the two ends of the tube) is needed to overcome the friction between particle layers to keep the fluid moving. For a given velocity pattern, the stress required is proportional to the fluid's viscosity.



The behavior of a fluid in flow is very much related to two intrinsic properties of the fluid: **density** and **viscosity**. For example, a solid body moving through a gas has to overcome a certain resistance which depends on the relative velocity between fluid and solid, the shape of the solid, the density of the gas and its viscosity. The power required to move a fluid through a conduit is a function of the fluid velocity, the diameter of the conduit and the fluid density and viscosity.

Newton's viscosity law:

As in the figure, if the upper plate is kept stationary while the lower plate is set to motion with a velocity u_O, the layer of liquid right next to this plate will also start to move. With time, the motion of the bottom layer of fluid will cause the fluid layers higher up to also move. When steady-state conditions are established, the velocity of the uppermost layer, which is in contact with the stationary plate, will also be zero, while the bottom layer, in contact with the moving plate, will be moving with velocity u_o. If we measure the velocity distribution across the intermediate fluid layers, we find that velocity changes linearly with distance y from the stationary plate:

$$u_x = u_o \frac{y}{Y}$$

Let us assume that we can measure, e.g. by means of a calibrated spring or transducer, the horizontal force ($-F_x$, Fig) which must be applied, in the opposite direction to u_o, to maintain the upper plate at rest; if we divide this force by the surface area of the plate, A, we find that this ratio, called the **shear stress**, is proportional to the velocity of the lower plate, u_o, and inversely proportional to the distance Y between the two plates:

$$\frac{Force}{area} = shear\ stress \propto \frac{u_o}{Y} = \mu \frac{u_o}{Y}$$

the proportionality constant between shear stress and the velocity gradient u_o/Y is called the **viscosity** of the liquid, μ. Since, at steady state conditions, the velocity profile between the two plates is linear, every infinitesimal segment of the line is represented by the same relationship. Therefore,

$$Shear\ stress = \tau_{y,x} = -\mu \frac{du_x}{dy}$$

Newton's viscosity law's states that, the shear stress between adjacent fluid layers is proportional to the velocity gradients between the two layers. The ratio of shear stress to shear rate is a constant, for a given temperature and pressure, and is defined as the**viscosity** or coefficient of **viscosity**.

**** Deatiled explanation of Newton's law and problems on same will be given in the class**

Types of Fluids:

1. Newtonian Fluids: The fluids which obey Newton's Law are called Newtonian Fluids
2. Non - Newtonian Fluids: The fluids which do not obey Newton's Law are called Non - Newtonian Fluids. These are inturn classified as:
   a. Thixotropic fluids
   b. Bingham Plastic
   c. Ideal – Non Newtonian Fluid

Continuum and Free Molecular Flow

- The concept of continuum is a kind of idealization of the continuous description of matter where the properties of the matter are considered as continuous functions of space variables. Although any matter is composed of several molecules, the concept of continuum assumes a continuous distribution of mass within the matter or system with no empty space, instead of the actual conglomeration of separate molecules.
- Describing a fluid flow quantitatively makes it necessary to assume that flow variables (pressure, velocity etc.) and fluid properties vary continuously from one point to another. Mathematical description of flow on this basis has proved to be reliable and treatment of fluid medium as a continuum has firmly become established. For example density at a point is normally define

$$\boldsymbol{\rho = \lim_{\Delta V \to 0} \left[\frac{m}{\Delta V}\right]}$$

Here ΔV is the volume of the fluid element and m is the mass

- If ΔV is very large ρ is affected by the in homogeneities in the fluid medium. Considering another extreme if ΔV is very small, random movement of atoms (or molecules) would change their number at different times. In the continuum approximation point density is defined at the smallest magnitude of ΔV, before statistical fluctuations become significant. This is called continuum limit and is denoted by ΔV_c.

$$\boldsymbol{\rho = \lim_{\Delta V \to \Delta V_C} \left[\frac{m}{\Delta V}\right]}$$

- One of the factors considered important in determining the validity of continuum model is molecular density. It is the distance between the molecules which is characterised by **mean free path (λ)**.
- It is calculated by finding statistical average distance the molecules travel between two successive collisions. *If the mean free path is very small as compared with some characteristic*

length in the flow domain (i.e., the molecular density is very high) then the gas can be treated as a ***continuous medium****. If the mean free path is large in comparison to some characteristic length, the gas cannot be considered continuous and it should be analysed by the* ***molecular theory****.*

A dimensionless parameter known as **Knudsen number**,

$$K_n = \frac{\lambda}{L}$$

where λ is the mean free path and L is the characteristic length. It describes the degree of departure from continuum. *Usually when $K_n > 0.01$, the concept of continuum does not hold good.*

Beyond this critical range of Knudsen number, the flows are known as

- Slip flow or continuum flow **($0.01 < K_n < 0.1$),**
- Transition flow **($0.1 < K_n < 10$)** and
- free-molecule flow **($K_n > 10$)**.

However, for the flow regimes considered in this course, K_n is always less than 0.01 and it is usual to say that the fluid is a continuum. Other factor which checks the validity of continuum is the elapsed time between collisions. The time should be small enough so that the random statistical description of molecular activity holds good.

In continuum approach, fluid properties such as density, viscosity, thermal conductivity, temperature, etc. can be expressed as continuous functions of space and time*.*

For the calculations in Classical Aerodynamics, we always assume the flow to be continuous.

PRESSURE

Pressure (symbol: p or P) is the force applied perpendicular to the surface of an object per unit area over which that force is distributed. Gauge pressure (also spelled *gage* pressure)[a] is the pressure relative to the ambient pressure.

Various units are used to express pressure. Some of these derive from a unit of force divided by a unit of area; the SI unit of pressure, the pascal (Pa), for example, is one newton per square metre; similarly, the pound-force per square inch (psi) is the traditional unit of pressure in the imperial and US customary systems. Pressure may also be expressed in terms of standard atmospheric pressure; the atmosphere (atm) is equal to this pressure, and the torr is defined as $^1/_{760}$ of this.

Manometric units such as the centimetre of water, millimetre of mercury, and inch of mercury are used to express pressures in terms of the height of column of a particular fluid in a manometer.

Pascal's law

Pascal's law (also **Pascal's principle** or the **principle of transmission of fluid-pressure**) is a principle in fluid mechanics that states that a pressure change occurring anywhere in a confined incompressible fluid is transmitted throughout the fluid such that the same change occurs everywhere. The law was established by French mathematician Blaise Pascal in 1647–48.

Pascal's principle is defined as

A change in pressure at any point in an enclosed fluid at rest is transmitted undiminished to all points in the fluid.

This principle is stated mathematically as:

$$\Delta P = \rho g\ (\Delta h)$$

ΔP is the hydrostatic pressure (given in pascals in the SI system), or the difference in pressure at two points within a fluid column, due to the weight of the fluid;

ρ is the fluid density (in kilograms per cubic meter in the SI system);

g is acceleration due to gravity (normally using the sea level acceleration due to Earth's gravity, in SI in metres per second squared);

Δh is the height of fluid above the point of measurement, or the difference in elevation between the two points within the fluid column (in metres in SI).

The intuitive explanation of this formula is that the change in pressure between 2 elevations is due to the weight of the fluid between the elevations. A more correct interpretation, though, is that the pressure change is caused by the change of potential energy per unit volume of the liquid due to the existence of the gravitational field. Note that the variation with height does not depend on any additional pressures. Therefore, Pascal's law can be interpreted as saying that *any* *change in pressure applied* at any given point of the fluid is transmitted *undiminished throughout* the fluid.

****Proof in the class notes**

Refer: https://user.engineering.uiowa.edu/~fluids/Posting/Lecture_Notes/Chapter2.pdf

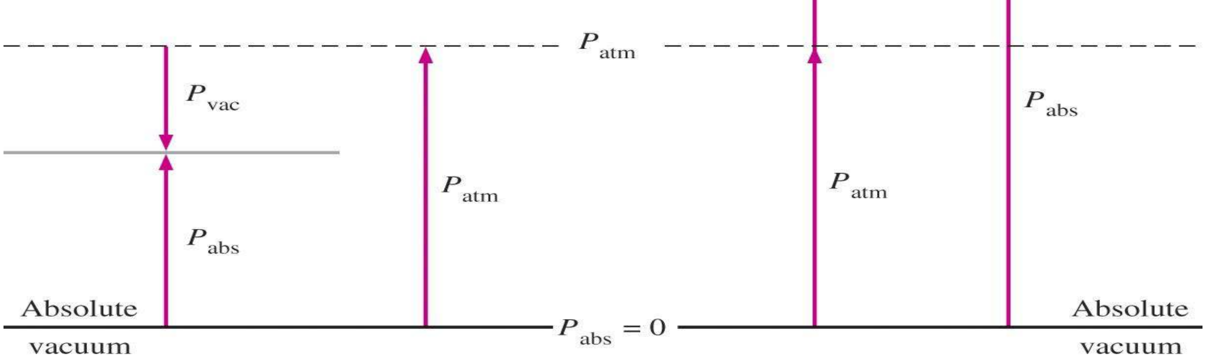
Absolute pressure: The actual pressure at a given position. It is measured relative to absolute vacuum (i.e., absolute zero pressure).

Guage pressure: The difference between the absolute pressure and the local atmospheric pressure. Most pressure-measuring devices are calibrated to read zero in the atmosphere, and so they indicate gage pressure.

Vacuum pressures: Pressures below atmospheric pressure.

$$P_{gage} = P_{abs} - P_{atm}$$

$$P_{vac} = P_{atm} - P_{abs}$$



PRESSURE MEASURING DEVICES – MANOMETERS AND BAROMETERS

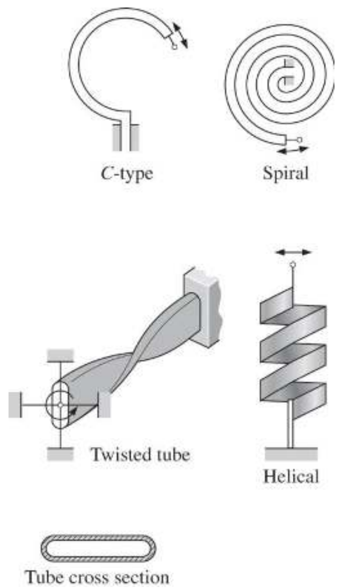
Bourdon tube: Consists of a hollow metal tube bent like a hook whose end is closed and connected to a dial indicator needle.

Pressure transducers: Use various techniques to convert the pressure effect to an electrical effect such as a change in voltage, resistance,or capacitance.

Pressure transducers are smaller and faster, and they can be more sensitive, reliable, and precise than their mechanical counterparts.

Strain-gage pressure transducers: Work by having a diaphragm deflect between two chambers open to the pressure inputs.

Piezoelectric transducers: Also called solid- state pressure transducers, work on the principle that an electric potential is generated in a crystalline substance when it is subjected to mechanical pressure.

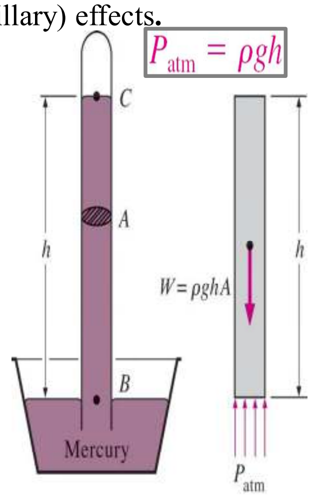


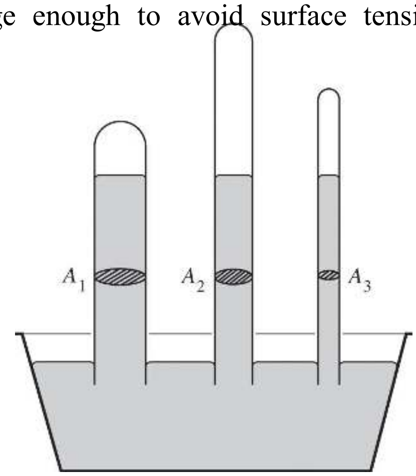
BAROMETER

Atmospheric pressure is measured by a device called a **barometer**; thus, the atmospheric pressure is often referred to as the ***barometric pressure***.

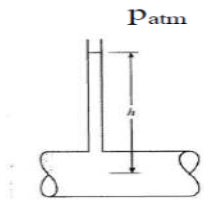
- A frequently used pressure unit is the *standard atmosphere*, which is defined as the pressure produced by a column of mercury 760 mm in height at 0°C ($_{Hg}$ = 13,595 kg/m^3) under standard gravitational acceleration (g = 9.807 m/s^2).

The length or the cross-sectional area of the tube has no effect on the height of the fluid column of a barometer provided that the tube diameter is large enough to avoid surface tension (capillary) effects.





2. Piezometer

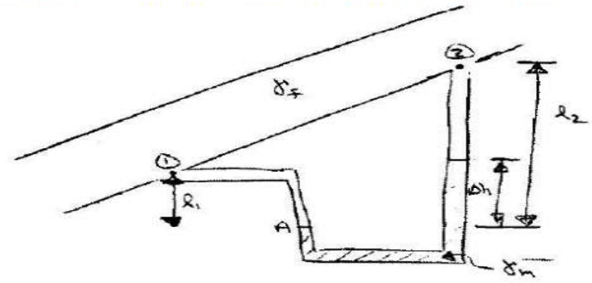


$p_{atm} + \gamma h = p_{pipe} = p$ absolute

$p = \gamma h$ gage

Simple but impractical for large p and vacuum pressures (i.e., $p_{abs} < p_{atm}$). Also for small p and small d, due to large surface tension effects, which could be corrected using $\Delta h = 4\sigma/\gamma d$, but accuracy may be problem if $p/\gamma \approx \Delta h$.

A differential manometer determines the differe[n]
pressures at two points ①and ② when the actua[l]
at any point in the system cannot be determined.



$$p_1+\gamma_f\ell_1-\gamma_m\Delta h-\gamma_f(\ell_2-\Delta h)=p_2$$
$$p_1-p_2=\gamma_f(\ell_2-\ell_1)+(\gamma_m-\gamma_f)\Delta h$$

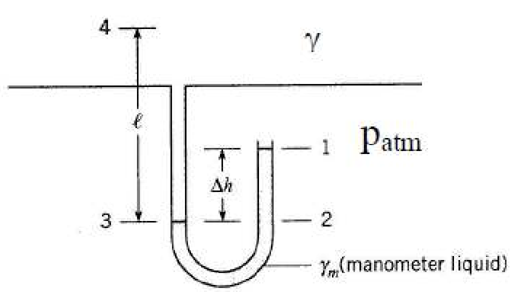
$$\underbrace{\left(\frac{p_1}{\gamma_f}+\ell_1\right)-\left(\frac{p_2}{\gamma_f}+\ell_2\right)}_{\text{difference in piezometric head}}=\left(\frac{\gamma_m}{\gamma_f}-1\right)\Delta h$$

difference in piezometric head

★ if fluid is a gas $\gamma_f << \gamma_m$: $p_1 - p_2 = \gamma_m \Delta h$

★ if fluid is liquid & pipe horizontal $\ell_1 = \ell_2$:
$p_1 - p_2 = (\gamma_m - \gamma_f)\,\Delta h$

3. U-tube or differential manometer



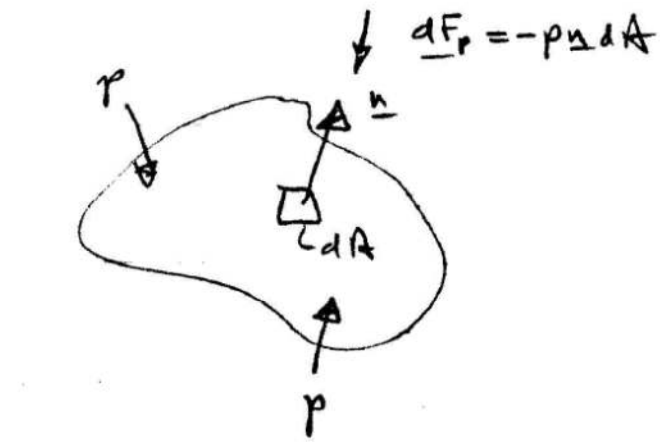
$$p_1 + \gamma_m\Delta h - \gamma l = p_4 \qquad p_1 = p_{atm}$$
$$p_4 = \gamma_m\Delta h - \gamma l \qquad \text{gage}$$
$$= \gamma_w[S_m\Delta h - S\ \ l]$$

for gases $S << S_m$ and can be neglected, i.e., can neglect Δp in gas compared to Δp in liquid in determining $p_4 = p_{pipe}$.

HYDROSTATIC PRESSRE AND CENTER OF PRESSURE

For a static fluid, the shear stress is zero and the only stress is the normal stress, i.e., pressure p. Recall that p is a scalar, which when in contact with a solid surface exerts a normal force towards the surface.

$$\underline{F}_p = -\int_A p\underline{n}dA$$



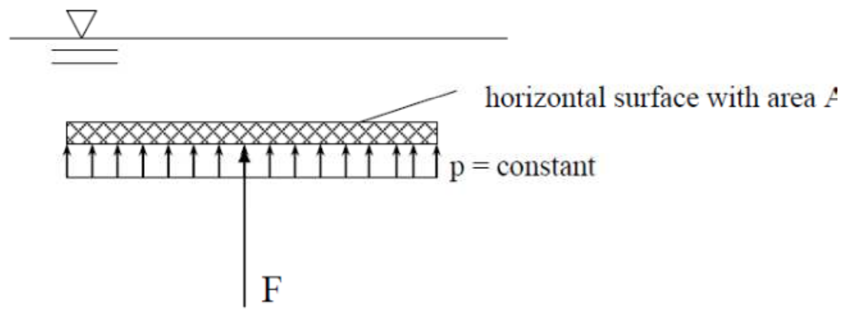
For a plane surface $\underline{n}$ = constant such that we can separately consider the magnitude and line of action of $\underline{F}_p$.

$$\left|\underline{F}_p\right| = F = \int_A p dA$$

Line of action is towards and normal to A through the center of pressure (x_{cp}, y_{cp}).

Unless otherwise stated, throughout the chapter assume p_{atm} acts at liquid surface. Also, we will use gage pressure so that p = 0 at the liquid surface.

Horizontal Surfaces



$$F = \int p dA = pA$$

Line of action is through centroid of A,
i.e., $(x_{cp}, y_{cp}) = (\bar{x}, \bar{y})$

Inclined Surfaces

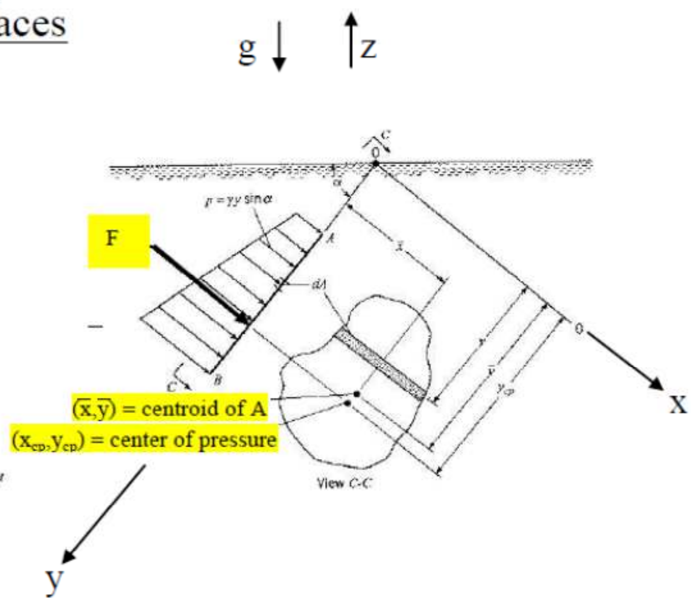


FIGURE 3.10
Distribution of hydrostatic pressure on a plane surface.

$$\frac{dp}{dz} = -\gamma$$

$$\Delta p = -\gamma \Delta z$$

$p - p_0 = -\gamma(z - z_0)$ where $p_0 = 0$ & $z_0 = 0$

$p = -\gamma z$ and $y \cdot \sin\alpha = -z$

$p = \gamma y \cdot \sin\alpha$

$$dF = pdA = \underbrace{\gamma y \sin\alpha}_{p}\, dA$$

γ and sin α are constants

$$F = \int_A pdA = \gamma \sin\alpha \underbrace{\int_A ydA}_{\bar{y}A}$$

$$\bar{y} = \frac{1}{A}\int ydA$$

1st moment of area

$$F = \underbrace{\gamma \sin\alpha\, \bar{y}}\, A$$

$\bar{p}$ = pressure at centroid of A

$$F = \bar{p}A$$

Magnitude of resultant hydrostatic force on plane surface is product of pressure at centroid of area and area of surface.

Center of Pressure

Center of pressure is in general below centroid since pressure increases with depth. Center of pressure is determined by equating the moments of the resultant and distributed forces about any arbitrary axis.

Determine y_{cp} by taking moments about horizontal axis 0-0

$$
\begin{aligned}
y_{cp}F &= \int_A y\,dF \\
&\quad \int_A y\,p\,dA \\
&\quad \int_A y(\gamma y \sin\alpha)\,dA \\
&= \gamma \sin\alpha \underbrace{\int_A y^2 dA}
\end{aligned}
$$

$I_o = 2^{nd}$ moment of area about 0-0
$= $ moment of inertia

transfer equation: $I_o = \bar{y}^2 A + \bar{I}$

$\bar{I}$ = moment of inertia with respect to horizontal centroidal axis

$$y_{cp} F = \gamma \sin\alpha (\bar{y}^2 A + \bar{I})$$

$$y_{cp} (\bar{p}A) = \gamma \sin\alpha (\bar{y}^2 A + \bar{I})$$

$$y_{cp} \gamma \sin\alpha\, \bar{y}A = \gamma \sin\alpha (\bar{y}^2 A + \bar{I})$$

$$y_{cp} \bar{y}A = \bar{y}^2 A + \bar{I}$$

$$y_{cp} = \bar{y} + \frac{\bar{I}}{\bar{y}A}$$

y_{cp} is below centroid by $\bar{I}/\bar{y}A$

$y_{cp} \rightarrow \bar{y}$ for large $\bar{y}$

For $p_o \neq 0$, y must be measured from an equivalent free surface located p_o/γ above $\bar{y}$.

Determine x_{cp} by taking moment about y axis

$$x_{cp}F = \int_A x\,dF = \int_A xp\,dA$$

$$x_{cp}(\gamma \bar{y} \sin \alpha A) = \int_A x(\gamma y \sin \alpha)dA$$

$$x_{cp}\bar{y}A = \underbrace{\int_A xy\,dA}$$

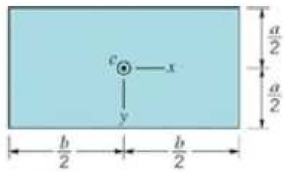
$$I_{xy} = \text{product of inertia}$$

$$= \bar{I}_{xy} + \bar{x}\bar{y}A \quad \text{transfer equation}$$

$$x_{cp}\bar{y}A = \bar{I}_{xy} + \bar{x}\bar{y}A$$

$$x_{cp} = \frac{\bar{I}_{xy}}{\bar{y}A} + \bar{x}$$

For plane surfaces with symmetry about an axis normal to 0-0, $\bar{I}_{xy} = 0$ and $x_{cp} = \bar{x}$.



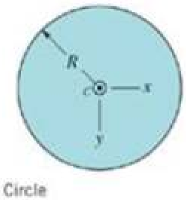
$A = ba$

$I_{xc} = \frac{1}{12} ba^3$

$I_{yc} = \frac{1}{12} ab^3$

$I_{xyc} = 0$

(a) Rectangle

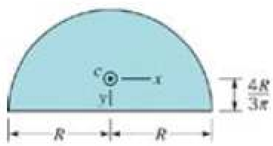


$A = \pi R^2$

$I_{xc} = I_{yc} = \frac{\pi R^4}{4}$

$I_{xyc} = 0$

(b) Circle



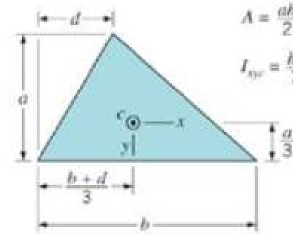
$A = \frac{\pi R^2}{2}$

$I_{xc} = 0.1098R^4$

$I_{yc} = 0.3927R^4$

$I_{xyc} = 0$

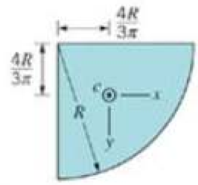
(c) Semicircle



$A = \frac{ab}{2}$ $\quad I_{xc} = \frac{ba^3}{36}$

$I_{xyc} = \frac{ba^2}{72}(b - 2d)$

(d) Triangle

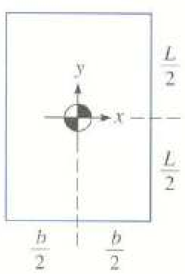


$A = \frac{\pi R^2}{4}$

$I_{xc} = I_{yc} = 0.05488R^4$

$I_{xyc} = -0.01647R^4$

(e) Quarter circle

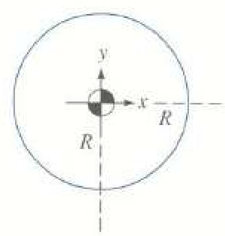


$A = bL$

$I_{xx} = \frac{bL^3}{12}$

$I_{xy} = 0$

(a)

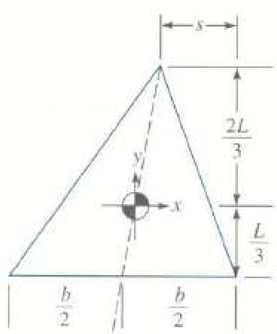


$A = \pi R^2$

$I_{xx} = \frac{\pi R^4}{4}$

$I_{xy} = 0$

(b)

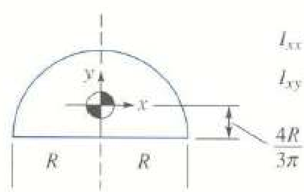


$A = \frac{bL}{2}$

$I_{xx} = \frac{bL^3}{36}$

$I_{xy} = \frac{b(b - 2s)L^2}{72}$

(c)



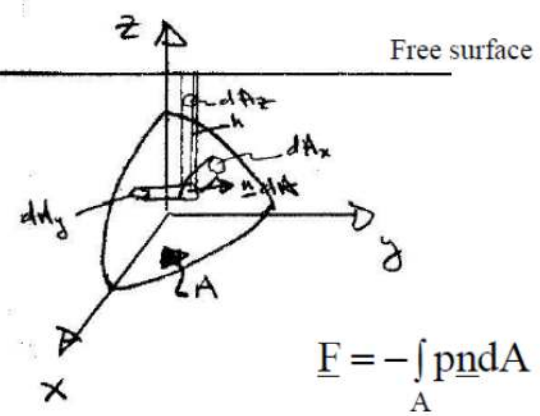
$A = \frac{\pi R^2}{2}$

$I_{xx} = 0.10976R^4$

$I_{xy} = 0$

(d)

Hydrostatic Forces on Curved Surfaces



$$\underline{F} = -\int_A p\underline{n}dA$$

$p = \gamma h$

h = distance below free surface

Horizontal Components (x and y components)

$$F_x = \underline{F} \cdot \hat{i} = -\int_A p\underline{n} \cdot \hat{i}dA = -\int_{A_x} pdA_x$$

dA_x = projection of $\underline{n}dA$ onto vertical plane to x-direction

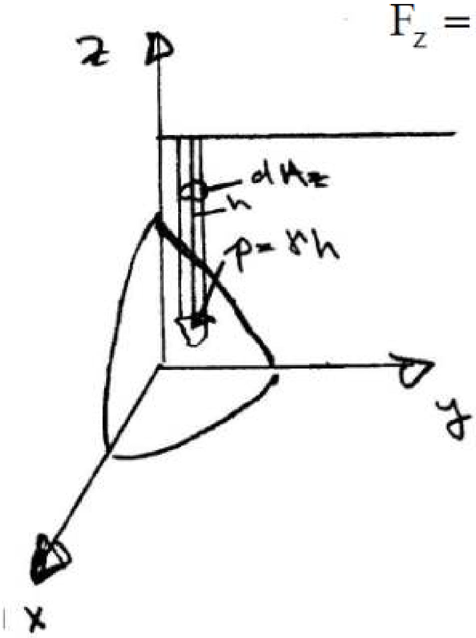
$$F_y = \underline{F} \cdot \hat{j} = -\int_{A_y} pdA_y$$

$dA_y = \underline{n} \cdot \hat{j}dA$ = projection $\underline{n}dA$ onto vertical plane to y-direction

Therefore, the horizontal components can be determined by some methods developed for submerged plane surfaces.

The horizontal component of force acting on a curved surface is equal to the force acting on a vertical projection of that surface including both magnitude and line of action.

<u>Vertical Components</u>



$$F_z = \underline{F} \cdot \hat{k} = -\int_A p\underline{n} \cdot \hat{k} dA$$

$$= -\int_{A_z} p dA_z \qquad p = \gamma h$$

h=distance below free surface

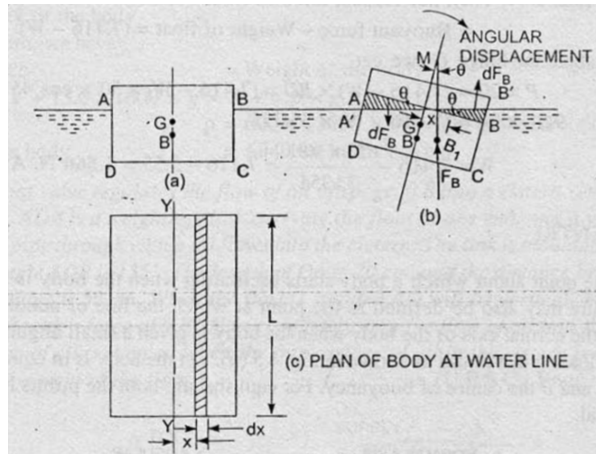
$$= \gamma \int_{A_z} h dA_z = \gamma V$$

= weight of fluid above surface A

The vertical component of force acting on a curved surface is equal to the net weight of the column of fluid above the curved surface with line of action through the centroid of that fluid volume.

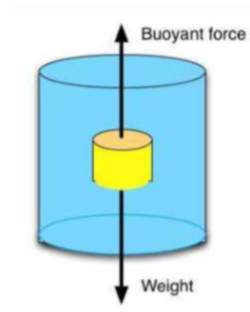
Buoyancy or Buoyant Force

The Archimedes principle states that the buoyant force on a partially immersed body or a submerged body is equal to the weight of liquid displaced by the body and acts vertically upward through the centroid of the displaced volume.



Meta-centre: the point about which a body starts oscillating when the body is tilted by a small angle is called as the centre of buoyancy.

Meta-centric Height: the distance between the meta-centre of a floating body and centre of gravity of the body is called meta-centric height.



For equilibrium, $F_B = w$

Due to symmetry of the situation, displaced volume remains unaltered and hence the buoyancy force

Thus, $F_B^{'} = F_B$

For couple calculation, F_B can be equivalently taken as a sum of ΔF_B (upward) due to added volume of fluid OBB' and ΔF_B (downward) due to decreased volume OAA'

Let ΔC be the couple due to these forces. Taking an element of area dA on the surface OBB' at a distance x from the center line (as shown in fig). Corresponding volume element is $x\Delta\theta dA$ and the buoyant force is $2\rho g x\Delta\theta dA$. This produces the couple $2\rho g x\Delta\theta dA$ (due to symmetrically located element on OA')

Therefore, integrating

$$\Delta C = 2\rho g\Delta\theta$$

$$\int_{area\mathrm{OB'}} x^2 dA = \rho g\Delta\theta$$

$$\int_{free\ surface} x^2 dA = \rho g\Delta\theta\, I_{yy}$$

Now we have $F_B^{'} r = wr = \Delta c$ where r is moment arm of $F_B^{'}$ about B.

Therefore, $wx = \rho g\Delta\theta I_{yy} \Rightarrow x = \frac{\rho g\Delta\theta I_{yy}}{w}$

Now, from figure $MB = \frac{x}{sin\Delta\theta} = \frac{\rho g \Delta\theta I_{yy}}{w sin\Delta\theta} = \frac{\rho g I_{yy}}{w}$ since $sin\Delta\theta \rightarrow \Delta\theta$

Thus $MG + BG = \frac{\rho g I_{yy}}{w}$

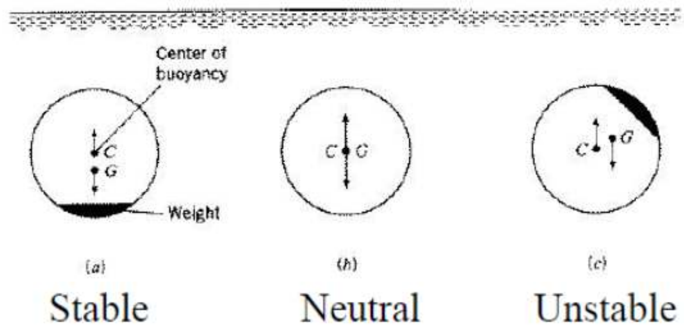
Meta-centric height is given by $\therefore\ MG = \frac{\rho g I_{yy}}{w} - BG$

Stability of Immersed and Floating Bodies

Here we'll consider transverse stability. In actual applications both transverse and longitudinal stability are important.

Immersed Bodies

FIGURE 3.15
Conditions of stability for immersed bodies. (a) Stable. (b) Neutral. (c) Unstable.



Static equilibrium requires: $\sum F_v = 0$ and $\sum M = 0$

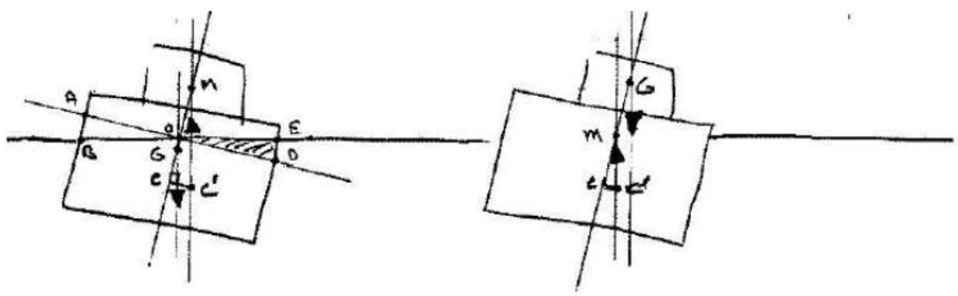
$\sum M = 0$ requires that the centers of gravity and buoyancy coincide, i.e., C = G and body is neutrally stable

If C is above G, then the body is stable (righting moment when heeled)

If G is above C, then the body is unstable (heeling moment when heeled)

Floating Bodies

For a floating body the situation is slightly more complicated since the center of buoyancy will generally shift when the body is rotated depending upon the shape of the body and the position in which it is floating.



Positive $\overline{GM}$ Negative $\overline{GM}$

The center of buoyancy (centroid of the displaced volume) shifts laterally to the right for the case shown because part of the original buoyant volume AOB is transferred to a new buoyant volume EOD.

The point of intersection of the lines of action of the buoyant force before and after heel is called the metacenter M and the distance GM is called the metacentric height. If GM is positive, that is, if M is above G, then the ship is stable; however, if GM is negative, the ship is unstable.

UNIT 2

FLUID KINEMATICS

Fluid Kinematics - Flow visualization - lines of flow - types of flow - velocity field and acceleration - continuity equation (one and three dimensional differential forms)- Equation of streamline – stream function – velocity potential function – circulation – flow net

Rotaional and Irrotational flows, vorticity

Introduction:

Stream line is the imaginary line whose tangent at any point gives the velocity of the particle moving along the line. Mathematically written as

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

Where $\vec{V}$ is the velocity vector,

$d\vec{S}$ is the infinitesimally small line segment of streamline.

A **stream surface** is defined as a continuous surface that is everywhere tangent to a specified vector.

Stream tube is a bundle of streamlines where there is no velocity of particles across the tube.

Path line is the trajectory of a particle in the fluid flow. For a steady uniform flow, path line coincides with the stream line. Path lines can intersect each other while the stream lines cannot.

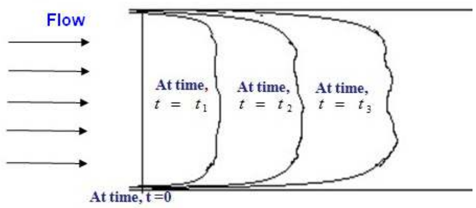
Streak line is the line joining the particles that have passed through a point in the flow field at any instant of time. The equation for a streak line is

$$\vec{S} = f[F(\overrightarrow{S_1}, t). t]$$

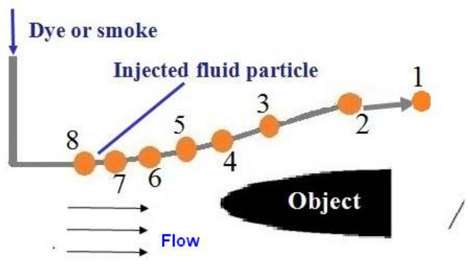
Where $\vec{S}$ is the streak line,

$\overrightarrow{S_1}$ is the point through which the points are passed,

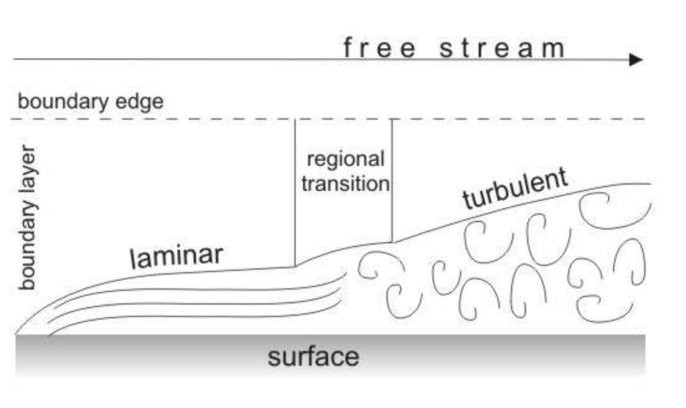
t is the instant of time.



(a) Timelines



(c) Streakline



classification of flows:

steady and unsteady flow

In a steady flow, the properties of fluid do not change with time at any point. Whereas in an unsteady flow, fluid properties change with time. Here, streamlines change with respect to the time in unsteady flow. In **Eulerian approach**, a steady flow can be mathematically written as

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

and

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

Uniform and non-uniform flow

The flow is defined as uniform flow when in the flow field the velocity and other hydrodynamic parameters do not change from point to point at any instant of time. When the velocity and other hydrodynamic parameters changes from one point to another the flow is defined as non-uniform flow. A uniform flow can be mathematically represented as

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

laminar and turbulent

The flow of fluid in the form of smooth layers is laminar and the flow with highly disordered fluid particles is turbulent. This type of flow mainly depends on the Reynolds number R_e.

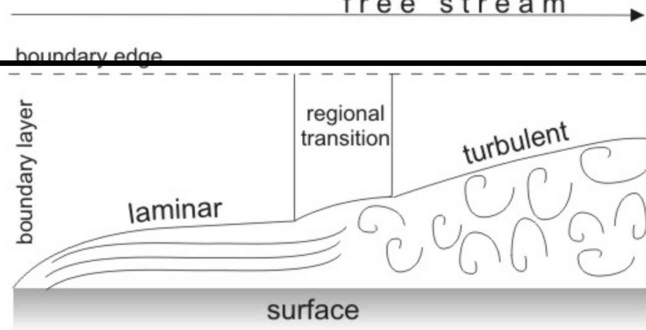


Figure: laminar and turbulent flows

laminar - when R_e < 2300

transient - when 2300 < R_e < 4000

turbulent - when R_e > 4000

Transient flow is the region where the flow changes from laminar to turbulent.

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.

(f) Graphical data analysis techniques: Profile plots, vector plots and contour plots are few important techniques in which fluid flow properties can be analyzed. The profile plot (Fig. 3.1.2-a) indicates the variation of any scalar property (such as pressure, temperature and density) along some desired direction in a flow field. Using this plot, it is possible to examine the relative behavior of all variables in a multivariate data set.

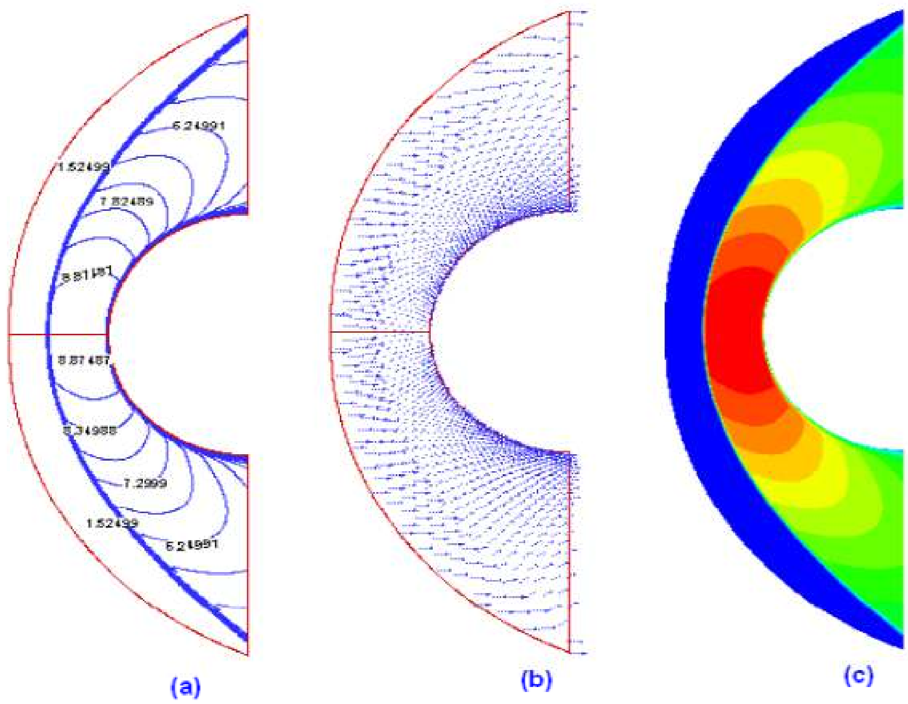


Fig. 3.1.2: Graphical representation of data analysis technique: (a) profile plot, (b) vector plot, (c) contour plot.

A vector plot (Fig. 3.1.2-b) is an array of arrows indicating the magnitude and direction of a vector property at an instant of time. Although, streamlines indicate the direction of instantaneous velocity field, but does not directly indicate the magnitude of velocity. A useful flow pattern for both experimental and computational fluid flow is the 'vector plot' that indicates the magnitude and direction of instantaneous vector property.

A contour plot (Fig. 3.1.2-c) is a two-dimensional plot of a three-dimensional surface showing lines where the surface intersects planes of constant elevation. Thus, they are curves with constant values of scalar property (or magnitude of vector property) at an instant of time. They can be filled in with either colors or sheds of gray representing the magnitude of the property.

3.1 Continuity equation

Continuity equation represents the law of conservation of mass.

In general for unsteady flow the continuity equation is

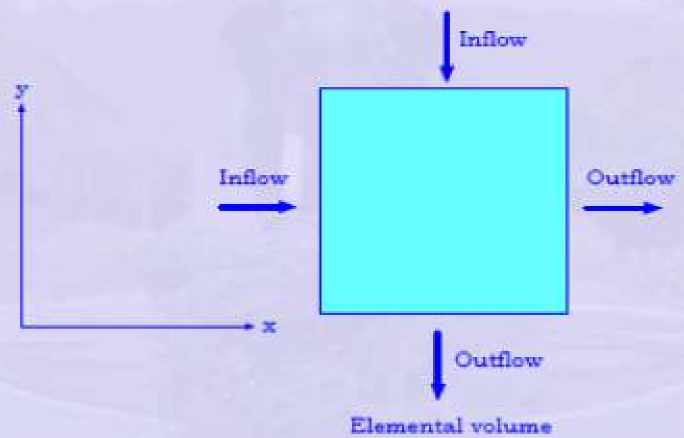
(Mass flow rate into the system) - (Mass flow rate out of the system) = Rate of change of storage.

For steady state condition

(Mass flow rate into the system) - (Mass flow rate out of the system) = 0.

Example: Inflow: The flow that is coming into a system or an elemental volume such as rainfall in y direction, flow entering into the river or a channel.

Outflow: The flow escaping from the system such as evaporation, seepage, water released from a system.



Generally, the mass balance is written in all the three directions namely x, y and z.

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

in which

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

ρ is the mass density of the fluid. If the mass density is cons tan t the above equation can be rewritten as

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

If *v=0, w=0 i.e.,* for one dim ensional flow it reduces to

$$\frac{\partial \rho u}{\partial x}=0$$

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

$$\frac{\partial \rho u}{\partial x} * \text{elemental area} = \text{constant}$$

Integrating one gets

UA = constant

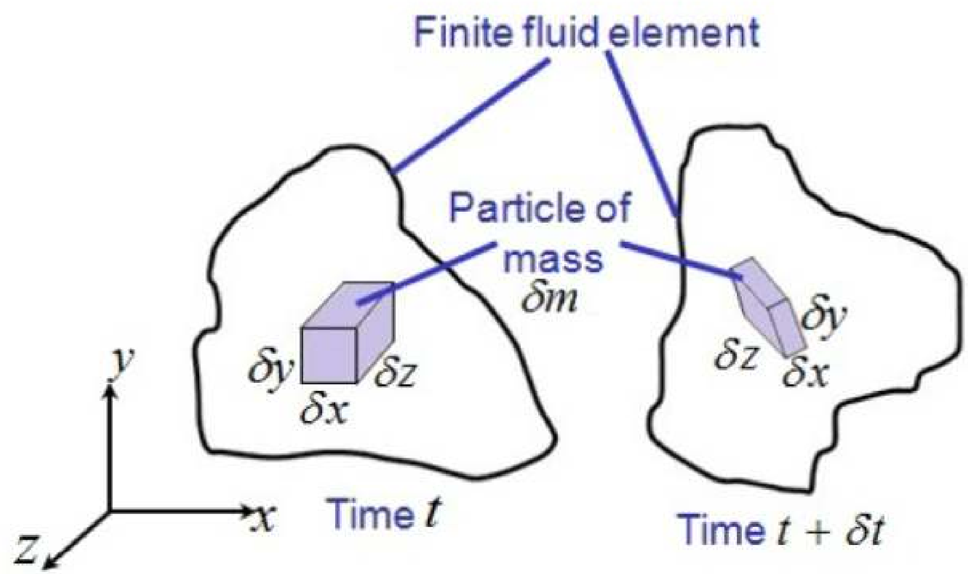
$\therefore$ Volume rate could be expressed as m^3/s. This is generally known as flow rate or discharge (Q) and expressed as cubic meter / second and is abbreviated as cumec (m^3/s).

$$Q = \text{Area} * \text{Velocity} = A\overline{V}$$

****Numerical Problems on Continuity Equation to be solved in the class**

Kinematic Description of Fluid Flow

In order to illustrate this concept, we consider a typical fluid element of certain volume at any arbitrary time as shown in Fig. 3.2.1. After certain time interval, it has moved and changed its shape as well as orientation drastically. However, when we limit our attention to an infinitesimal particle of volume $d\forall (= \delta x.\delta y.\delta z)$ at time t and $(t+\delta t)$ within the fluid element, it may be observed that the change of its shape is limited to only stretching/shrinking and rotation with its sides remaining straight even though there is a drastic change in the finite fluid element. Thus, the particle motion in a fluid flow can be decomposed into four fundamental components i.e. *translation, rotation, linear strain* and *shear strain* as shown in Fig. 3.2.2. When the fluid particle moves in space from one point to another, it is referred as *translation. Rotation* of the fluid particle can occur in any of the orthogonal axis. In the case of *linear strain*, the particle's side can stretch or shrink. When the angle between the sides of the particle changes, it is called as *shear strain*.



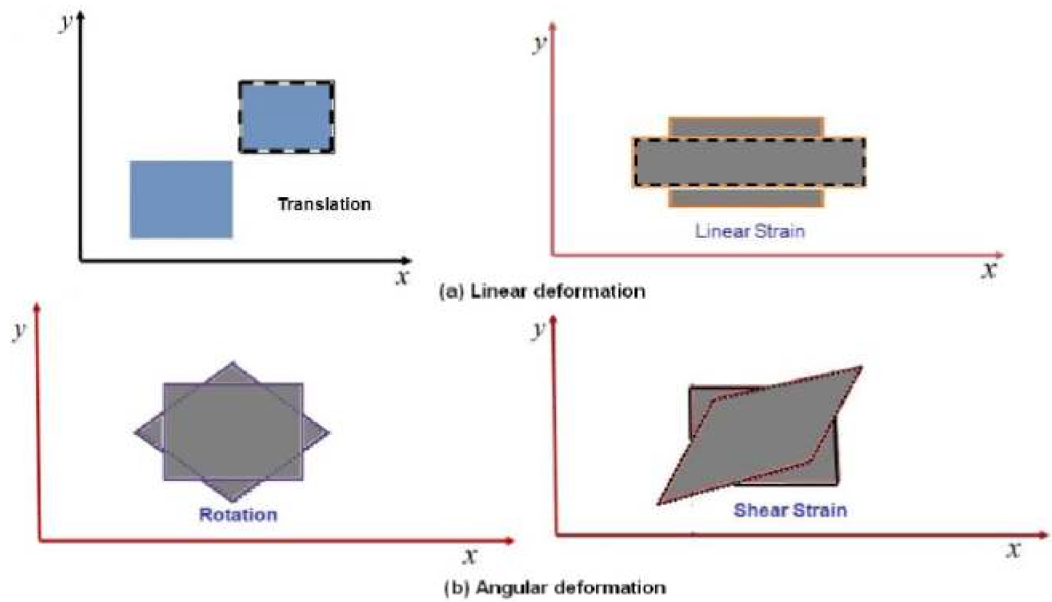


Fig. 3.2.2: Basic deformations of fluid mass: (a) Linear deformation; (b) Angular deformation.

Linear Motion and Deformation

Translation is the simplest type of fluid motion in which all the points in the fluid element have same velocity. As shown in Fig. 3.2.3-a, the particle located as point O will move to O' during a small time interval δt. When there is a presence of velocity gradient, the element will tend to deform as it moves. Now, consider the effect of single velocity gradient $(\partial u/\partial x)$ on a small cube having sides $\delta x, \delta y$ and δz and volume $\delta \mathcal{V} = \delta x\, \delta y\, \delta z$. As shown in Fig. 3.2.3-b, the x-component of velocity of O and B is u. Then, x-component of velocity of points A and C would be, $u + (\partial u/\partial x)\delta x$, which causes stretching of AA' by an amount $\left[(\partial u/\partial x)\delta x\, \delta t\right]$ as shown in Fig.3.2.3-c. So, there is a change in the volume element $\delta \mathcal{V} = \left(\dfrac{\partial u}{\partial x}\delta x\right)(\delta y\, \delta z)\delta t$

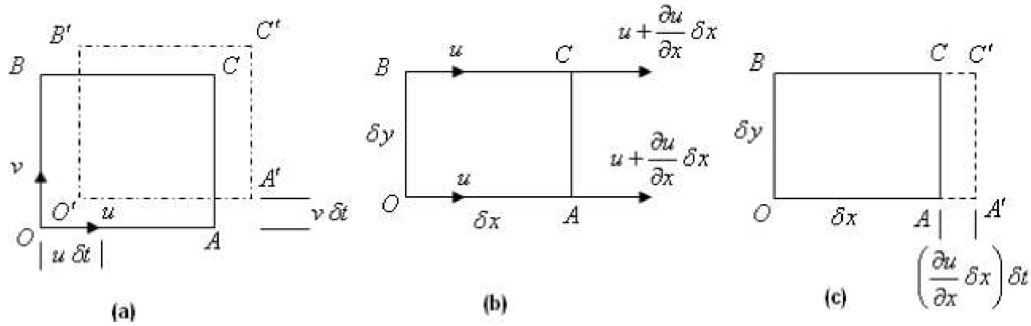


Fig. 3.2.3: Linear deformation of a fluid element.

Rate at which the volume $\delta\forall$ changes per unit volume due to the velocity gradient $(\partial u/\partial x)$ is

$$\frac{1}{\delta\forall}\frac{d(\delta\forall)}{dt}=\lim_{\delta t\to 0}\left[\frac{(\partial u/\partial x)\delta t}{\delta t}\right]=\frac{\partial u}{\partial x} \tag{3.2.1}$$

In the presence of other velocity gradients $(\partial v/\partial y)$ and $(\partial w/\partial z)$, Eq. (3.2.1) becomes,

$$\frac{1}{\delta\forall}\frac{d(\delta\forall)}{dt}=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=\nabla.\vec{V} \tag{3.2.2}$$

If we look closely to the unit of velocity gradients $(\partial u/\partial x)$, $(\partial v/\partial y)$ and $(\partial w/\partial z)$, then they resemble to unit of *strain rate* and the deformation is associated in the respective directions of orthogonal coordinates in which the components of the velocity lie. Thus, the *linear strain* (Fig. 3.2.2-a) is defined as the rate of increase in length to original length and the *linear strain rates* are expressed as,

$$\varepsilon_{xx}=\frac{\partial u}{\partial x},\ \varepsilon_{yy}=\frac{\partial v}{\partial y},\ \varepsilon_{zz}=\frac{\partial w}{\partial z} \tag{3.2.3}$$

The *volumetric strain rate/volumetric dilatation rate* is defined as the rate of increase of volume of a fluid element per unit volume.

$$\frac{1}{\forall}\frac{d\forall}{dt}=\varepsilon_{xx}+\varepsilon_{yy}+\varepsilon_{zz}=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z} \tag{3.2.4}$$

In an incompressible fluid, the *volumetric dilatation rate* is zero because the fluid element volume cannot change without change in fluid density.

Angular Motion and Deformation

The variations of velocity in the direction of velocity is represented by the partial derivatives $(\partial u/\partial x)$, $(\partial v/\partial y)$ and $(\partial w/\partial z)$, which causes linear deformation in the sense that shape of the fluid element does not change. However, cross variations of derivatives such as $(\partial u/\partial y)$, $(\partial v/\partial z)$, $(\partial w/\partial x)$ will cause the fluid element to rotate. These motions lead to angular *deformation* which generally changes the shape of the element.

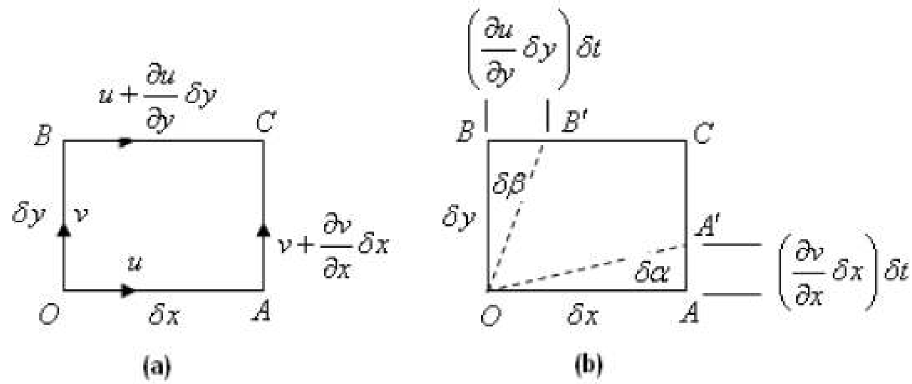


Fig. 3.2.4: Angular deformation of a fluid element.

Let us consider the angular motion in *x-y* plane in which the initial shape is given by *OACB*, as shown in Fig. 3.2.4-a. The velocity variations cause the rotation and angular deformation so that the new positions become *OA'* and *OB'* after a time interval δt. Then the angles AOA' and BOB' are given by $\delta\alpha$ and $\delta\beta$, respectively as shown in Fig. 3.2.4-b. Thus, the angular velocities of line *OA* and *OB* are,

$$\omega_{OA} = \lim_{\delta t\to 0}\left(\frac{\delta\alpha}{\delta t}\right) = \frac{\partial v}{\partial x}; \quad \omega_{OB} = \lim_{\delta t\to 0}\left(\frac{\delta\beta}{\delta t}\right) = \frac{\partial u}{\partial y}$$

$$\text{For small angles, } \tan\delta\alpha \approx \delta\alpha = \frac{(\partial v/\partial x)\,\delta x\,\delta t}{\delta x} = \frac{\partial v}{\partial x}\delta t \qquad (3.2.5)$$

$$\text{and} \quad \tan\delta\beta \approx \delta\beta = \frac{(\partial u/\partial y)\,\delta y\,\delta t}{\delta y} = \frac{\partial u}{\partial y}\delta t$$

When, both $(\partial v/\partial x)$ and $(\partial u/\partial y)$ are positive, then both ω_{OA} and ω_{OB} will be in counterclockwise direction. Now, the *rotation* of the fluid element about z-direction (i.e *x-y* plane) ω_z can be defined as the average of ω_{OA} and ω_{OB}. If counterclockwise rotation is considered as positive, then

$$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \qquad (3.2.6)$$

In a similar manner, the rotation of the fluid element about *x and y* axes are denoted as ω_x and ω_y, respectively.

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right); \ \ \omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) \qquad (3.2.7)$$

These three components can be combined to define the rotation vector $(\vec{\omega})$ in the form as,

$$\vec{\omega} = \omega_x \hat{i} + \omega_y \hat{j} + \omega_z \hat{k}$$

$$\text{or, } \vec{\omega} = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\hat{i} + \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\hat{j} + \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\hat{k} \qquad (3.2.8)$$

$$\text{or, } \vec{\omega} = \frac{1}{2}\begin{vmatrix} \hat{i} & \hat{j} & \hat{k} \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ u & v & w \end{vmatrix} = \frac{1}{2}\nabla \times \vec{V}$$

It is observed from Eq.(3.2.6) that the fluid element will rotate about *z*-axis, as an *undeformed* block, only when, $\omega_{OA} = -\omega_{OB}$ i.e. $(\partial v/\partial x) = -(\partial u/\partial y)$. Otherwise it will be associated with angular deformation which is characterized by *shear strain rate*. When the fluid element undergoes shear deformation (Fig. 3.2.2-b), the *average shear strain rates* expressed in different cartesian planes as,

$$\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right); \ \varepsilon_{yz} = \frac{1}{2}\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right); \ \varepsilon_{zx} = \frac{1}{2}\left(\frac{\partial w}{\partial z} + \frac{\partial u}{\partial x}\right) \qquad (3.2.9)$$

Strain rate as a whole constitute a symmetric second order tensor i.e.

$$\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{yx} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{zx} & \varepsilon_{zy} & \varepsilon_{zz} \end{pmatrix} \qquad (3.2.10)$$

Vorticity

In a flow field, v*orticity* is related to fluid particle velocity which is defined as *twice* of rotation vector i.e.

$$\zeta = 2\vec{\omega} = \nabla \times \vec{V} = \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\hat{i} + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\hat{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\hat{k} \qquad (3.2.11)$$

Thus, the curl of the velocity vector is equal to the vorticity. It leads to two important definitions:

- If $\nabla \times \vec{V} \neq 0$ at every point in the flow, the flow is called as rotational. It implies that the fluid elements have a finite angular velocity.
- If $\nabla \times \vec{V} = 0$ at every point in the flow, the flow is called as irrotational. It implies that the fluid elements have no angular velocity rather the motion is purely translational.

Irrotational Flow

In Eq.(3.2.11) , if $\nabla \times \vec{V} = 0$ is zero, then the rotation and vorticity are zero. The flow fields for which the above condition is applicable is known as irrotational flow. The condition of *irrotationality* imposes specific relationship among the velocity gradients which is applicable for *inviscid* flow. If the rotations about the respective orthogonal axes are to be zero, then, one can write Eq. (3.2.11) as,

$$\omega_z = 0 \Rightarrow \frac{\partial v}{\partial x} = \frac{\partial u}{\partial y};\ \omega_y = 0 \Rightarrow \frac{\partial u}{\partial z} = \frac{\partial w}{\partial x};\ \omega_x = 0 \Rightarrow \frac{\partial w}{\partial y} = \frac{\partial v}{\partial z} \qquad (3.2.12)$$

A general flow field would never satisfy all the above conditions. However, a uniform flow field defined in a fashion, for which $u = U$ (a constant); $v = 0$; $w = 0$, is certainly an example of an irrotational flow because there are no velocity gradients. A fluid flow which is initially irrotational may become rotational if viscous effects caused by solid boundaries, entropy gradients and density gradients become significant.

Circulation

It is defined as the line integral of the tangential velocity component about any closed curve fixed in the flow i.e.

$$\Gamma = \oint \vec{V}.d\vec{s} \tag{3.2.13}$$

where, $d\vec{s}$ is an elemental vector tangent to the curve and with length ds with counterclockwise path of integration considered as positive. For the closed curve path *OACB* as shown in Fig. 3.2.4-a, we can develop the relationship between circulation and vorticity as follows;

$$\delta\Gamma = u\,\delta x + \left(v + \frac{\partial v}{\partial x}\delta x\right)\delta y - \left(u + \frac{\partial u}{\partial y}\delta y\right)\delta x - v\,\delta y$$

$$\text{or, } \delta\Gamma = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\delta x\,\delta y = 2\omega_z \delta x\,\delta y \tag{3.2.14}$$

$$\text{Then, } \Gamma = \oint \vec{V}.d\vec{s} = \int_A 2\omega_z\,dA = \int_A \left(\nabla \times \vec{V}\right)_z dA$$

Hence, *circulation* around a closed contour is equal to *total vorticity* enclosed within it. It is known as *Stokes theorem* in two dimensions.

Stream Function

The idea of introducing stream function works only if the continuity equation is reduced to two terms. There are 4-terms in the continuity equation that one can get by expanding the vector equation (3.3.1) i.e.

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{3.3.2}$$

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

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{3.3.3}$$

Here, the striking idea of stream function works that will eliminate two velocity components u and v into a single variable (Fig. 3.3.1-a). So, the *stream function* $\{\psi(x,y)\}$ relates to the velocity components in such a way that continuity equation (3.3.3) is satisfied.

$$\begin{aligned} u=\frac{\partial \psi}{\partial y};\ v=-\frac{\partial \psi}{\partial x} \\ \text{or}, \vec{V}=\frac{\partial \psi}{\partial y}\hat{i}-\frac{\partial \psi}{\partial x}\hat{j} \end{aligned} \tag{3.3.4}$$

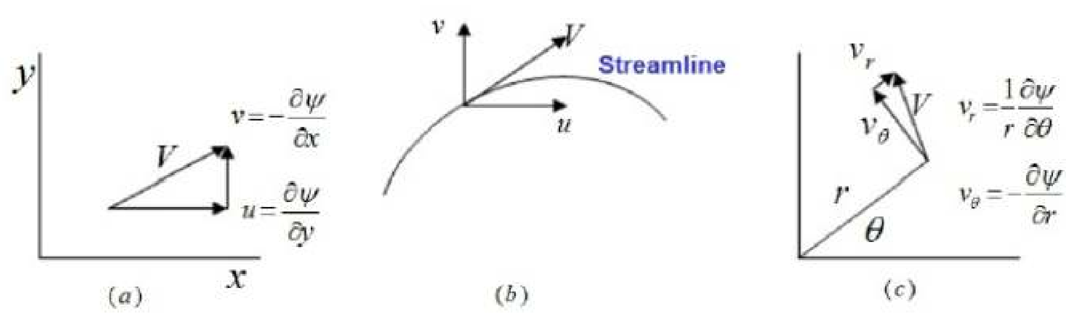


Fig. 3.3.1: Velocity components along a streamline.

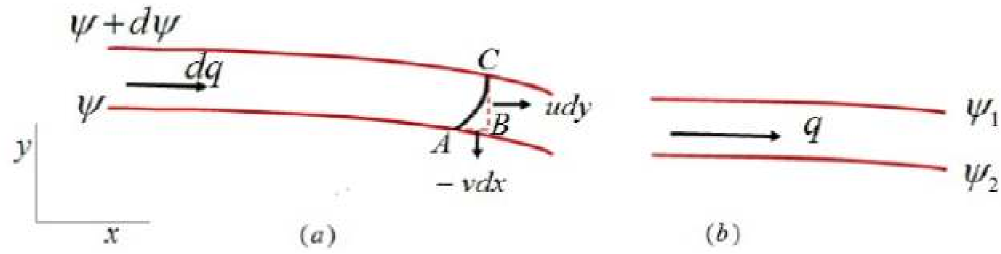


Fig. 3.3.2: Flow between two streamlines.

The following important points can be noted for stream functions;

1. The lines along which ψ is constant are called as *streamlines*. In a flow field, the tangent drawn at every point along a streamline shows the direction of velocity (Fig. 3.3.1-b). So, the slope at any point along a streamline is given by,

$$\frac{dy}{dx} = \frac{v}{u} \qquad (3.3.5)$$

Referring to the Fig. 3.3.2-a, if we move from one point (x, y) to a nearby point $(x+dx, y+dy)$, then the corresponding change in the value of stream function is $d\psi$ which is given by,

$$d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy = -v\,dx + u\,dy \qquad (3.3.6)$$

Along a line of constant ψ,

$$\begin{aligned} d\psi &= -v\,dx + u\,dy = 0 \\ \text{or, } \frac{dy}{dx} &= \frac{v}{u} \end{aligned} \qquad (3.3.7)$$

The Eq. (3.3.5) is same as that of Eq. (3.3.7). Hence, it is the defining equation for the streamline. Thus, infinite number streamlines can be drawn with constant ψ. This family of streamlines will be useful in visualizing the flow patterns. It may also be noted that *streamlines are always parallel to each other*.

2. The numerical constant associated to ψ, represents the volume rate of flow. Consider two closely spaced streamlines ψ and $(\psi + d\psi)$ as shown in Fig. 3.3.2-a. Let $d\dot{q}$ represents the volume rate of flow per unit width perpendicular to *x-y* plane, passing between the streamlines. At any arbitrary surface *AC*, this volume flow must be equal to net outflow through surfaces *AB* and *BC*. Thus,

$$d\dot{q} = -v\,dx + u\,dy = \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy = d\psi \qquad (3.3.8)$$

$$\text{or, } d\dot{q} = d\psi$$

Hence, the volume flow rate $(\dot{q})$ can be determined by integrating Eq. (3.3.8) between streamlines ψ_1 and ψ_2 as follows;

$$\dot{q} = \int_{\psi_1}^{\psi_2} d\psi = \psi_2 - \psi_1 \qquad (3.3.9)$$

So, the change in the value of stream function is equal to volume rate of flow. If the upper streamline ψ_2 has a value greater than the lower one ψ_1, then the volume flow rate is positive i.e. flow takes place from left to right (Fig. 3.3.2-b).

3. In cylindrical coordinates, the continuity equation for a steady, incompressible, plane, two-dimensional flow, reduces to

$$\frac{1}{r}\frac{\partial (r v_r)}{\partial r} + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} = 0 \qquad (3.3.10)$$

The respective velocity components v_r and v_θ are shown in Fig. 3.3.1-c. The stream function $\{\psi(r,\theta)\}$ that satisfies Eq. (3.3.10), can then be defined as,

$$v_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta};\ v_\theta = -\frac{\partial \psi}{\partial r} \qquad (3.3.11)$$

4. In a steady, plane compressible flow, the stream function can be defined by including the density of the fluid. But, the change in the stream function is equal to mass flow rate $(\dot{m})$.

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

$$d\dot{m} = -\rho v\, dx + \rho u\, dy = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy = d\psi \qquad (3.3.12)$$

$$\text{or, } d\dot{m} = d\psi$$

5. One important application in a two-dimensional plane is the *inviscid and irroational* flow where, there is no velocity gradient and $\omega_z = 0$. Then, the vorticity vector becomes,

$$\zeta = 2\omega_z \hat{k} = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right) \hat{k} = 0$$

$$\text{or, } \left[\frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial x} \right) - \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial y} \right) \right] \hat{k} = 0 \qquad (3.3.13)$$

$$\text{or, } \frac{\partial^2 \psi}{\partial x^2} - \frac{\partial^2 \psi}{\partial y^2} = 0$$

$$\text{or, } \nabla^2 \psi = 0$$

This is a second order equation and is quite popular in mathematics and is known as *Laplace equation* in a two-dimensional plane.

Velocity Potential

An irrotational flow is defined as the flow where the vorticity is zero at every point. It gives rise to a scalar function (ϕ) which is similar and complementary to the stream function (ψ). Let us consider the equations of irrortional flow and scalar function (ϕ). In an irrotational flow, there is no vorticity $(\vec{\xi})$

$$\vec{\xi} = \nabla \times \vec{V} = 0 \qquad (3.3.14)$$

Now, take the vector identity of the scalar function (ϕ),

$$\nabla \times (\nabla \phi) = 0 \qquad (3.3.15)$$

i.e. a vector with zero curl must be the gradient of a scalar function or, curl of the gradient of a scalar function is identically zero. Comparing, Eqs. (3.3.14) and (3.3.15), we see that,

$$\vec{V} = \nabla \phi \qquad (3.3.16)$$

Here, ϕ is called as *velocity potential function* and its gradient gives rise to velocity vector. The knowledge ϕ immediately gives the velocity components. In cartesian coordinates, the *velocity potential function* can be defined as, $\phi = \phi(x, y, z)$ so that Eq. (3.3.16) can be written as,

$$u\hat{i} + v\hat{j} + w\hat{k} = \frac{\partial \phi}{\partial x}\hat{i} + \frac{\partial \phi}{\partial y}\hat{j} + \frac{\partial \phi}{\partial z}\hat{k} \qquad (3.3.17)$$

So, the velocity components can be written as,

$$u = \frac{\partial \phi}{\partial x};\ v = \frac{\partial \phi}{\partial y};\ w = \frac{\partial \phi}{\partial z} \qquad (3.3.18)$$

In cylindrical coordinates, if $\phi = \phi(r, \theta, z)$, then

$$V_r = \frac{\partial \phi}{\partial r};\ V_\theta = \frac{\partial \phi}{\partial \theta};\ V_z = \frac{\partial \phi}{\partial z} \qquad (3.3.19)$$

Further, if the flow is incompressible i.e. $\rho = \text{constant}$ and $\left(\partial\rho/\partial t\right) = 0$, then continuity equation can be written as,

$$\frac{\partial\rho}{\partial t} + \nabla.\left(\rho\vec{V}\right) = 0$$
$$\text{or,}\ \rho\left(\nabla.\vec{V}\right) = 0 \qquad (3.3.20)$$
$$\text{or,}\ \nabla.\vec{V} = 0$$

Therefore, for a flow which is incompressible and irrotational, Eqs. (3.3.16) and (3.3.20) can be combined to yield a second order *Laplace equation* in a three-dimensional plane.

$$\nabla.\left(\nabla\phi\right) = 0$$
$$\text{or,}\ \nabla^2\phi = 0 \qquad (3.3.21)$$
$$\text{or,}\ \frac{\partial^2\phi}{\partial x^2} + \frac{\partial^2\phi}{\partial y^2} + \frac{\partial^2\phi}{\partial z^2} = 0$$

Thus, any irrotational, incompressible flow has a velocity potential and stream function (for two-dimensional flow) that both satisfy *Laplace equation*. Conversely, any solution of *Laplace equation* represents both velocity potential and stream function (two-dimensional) for an irrotational, incompressible flow.

An irrotational flow allows a velocity potential to be defined and leads to simplification of fundamental equations. Instead of dealing with the velocity components u, v and w as unknowns, one can deal with only one parameter ϕ, for a given problem. Since, the irrotational flows are best described by velocity potential, such flows are called as *potential flows*. In these flows, the lines with constant ϕ, is known as *equipotential lines*. In addition, a line drawn in space such that $\nabla\phi$ is the tangent at every point is defined as a *gradient line* and thus can be called as *streamline*.

Stream Function vs Velocity Potential

The velocity potential is analogous to stream function in a sense that the derivatives of both ϕ and ψ yield the flow field velocities. However, there are distinct differences between ϕ and ψ :

- The flow field velocities are obtained by differentiating ϕ in the same direction as the velocities, whereas, ψ is differentiated normal to the velocity direction.
- The velocity potential is defined for irrotational flows only. In contrast, stream function can be used in either rotational or irrotational flows.
- The velocity potential applies to three-dimensional flows, whereas the stream function is defined for two dimensional flows only.

It is seen that the stream lines are defined as lines of constant ψ which are same as gradient lines and perpendicular to lines of constant ϕ. So, the equipotential lines and stream lines are mutually perpendicular. In order to illustrate the results more clearly, let us consider a two-dimensional, irrotational, incompressible flow in Cartesian coordinates.

For a streamline, $\psi(x, y) = \text{constant}$, and the differential of ψ is zero.

$$\begin{aligned} d\psi &= \frac{\partial \psi}{\partial x}dx + \frac{\partial \psi}{\partial y}dy = 0 \\ \text{or, } d\psi &= -v\,dx + u\,dy = 0 \\ \text{or, } \left(\frac{dy}{dx}\right)_{\psi=\text{constant}} &= \frac{v}{u} \end{aligned} \tag{3.3.22}$$

Similarly, for an equipotential line, $\phi(x, y) = \text{constant}$, and the differential of ϕ is zero.

$$\begin{aligned} d\phi &= \frac{\partial \phi}{\partial x}dx + \frac{\partial \phi}{\partial y}dy = 0 \\ \text{or, } d\phi &= u\,dx + v\,dy = 0 \\ \text{or, } \left(\frac{dy}{dx}\right)_{\phi=\text{constant}} &= -\frac{u}{v} \end{aligned} \tag{3.3.23}$$

Combining Eqs. (3.3.22) and (3.3.23), we can write,

$$\left(\frac{dy}{dx}\right)_{\psi=\text{constant}} = -\frac{1}{(dy/dx)_{\phi=\text{constant}}} \tag{3.3.24}$$

Hence, the streamlines and equipotential lines are mutually perpendicular.

This is called Flow NET

*****Problems on Veolocity potential, Stream Function, Rotational and irrotational flows will be given in the class***

UNIT – 3

FLUID DYNAMICS

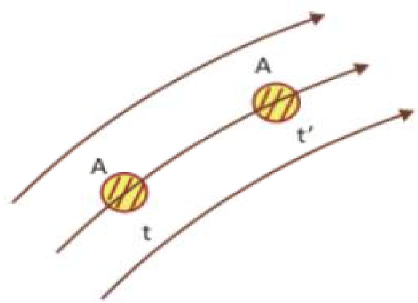
Part 1:

Fluid Dynamics: Surface & body forces, substantive derivative, local derivative and convective derivative, momentum equation, Euler equation, Bernoulli's equation. Phenomenological basis of Navier-Stokes equation.

Eulerian and Lagrangian Approach

Consider 2D flow of a fluid.

$\overline{V} = iV_x + jV_y$



(Fig. 9a)

There are two approaches to describe the motion of a fluid and its associated properties.

1. Lagrangian approach
2. Eulerian approach

Lagrangian approach:

Identify (or label) a material of the fluid; track (or follow) it as it moves, and monitor change in its properties. The properties may be velocity, temperature, density, mass, or concentration, etc in the flow field.

Refer the above-figure. The 'material' or 'particle' of the fluid 'A' at time t has moved to some other location at time t'. Its property, say temperature, is recorded, as the material moves in the flow-field:

$$\left.\begin{matrix} t_1 \rightarrow T_1 \\ t_2 \rightarrow T_2 \\ t_n \rightarrow T_n \end{matrix}\right]$$

Note that the recorded temperatures are associated with the same fluid particle, but at different locations and at different times.

Think of a temperature sensor attached to a balloon, both having negligible mass and floating in the atmosphere and recording the atmosphere-temperature or the temperature of the flow-field. In such case, the following temperature-data are recorded by the sensor**:**

Location	time	temperature
$(x_1 y_1 z_1)$	t_1	T_1
(x_2, y_2, z_2)	t_2	T_2
⋮	⋮	⋮
(x_n, y_n, z_n)	t_n	T_n

The time change of the temperature in such a measurement is denoted as $\frac{DT}{Dt}$, which is called material derivative or substantial derivative. It reflects time change in the temperature (or any other properties) of the labeled /marked/tagged fluid particles as observed by an observer moving with the fluid. Lagrangian approach is also called "particle based approach"**.**

Eulerian approach (a field approach)

Identify (or label) a certain fixed location in the flow field and follow change in its property, as different materials pass through that location. In such case, the following property, say temperature is recorded by the sensor **:**

$$\left.\begin{matrix} t_1 \rightarrow T_1 \\ t_2 \rightarrow T_2 \\ t_n \rightarrow T_n \end{matrix}\right]$$

Note that the recorded temperatures are associated with the fixed location in the flow-fluid, having different fluid elements at different times.

The time- change of the temperature in such a measurement is denoted as $\left.\frac{\partial T}{\partial t}\right]_{(x,y,z)}$ which is called the partial derivativeof the temperature with respect to time. Note that the suffix (x,y,z) implies that the observer records the change in the property at the fixed location (x,y,z).

$\left(\frac{\partial T}{\partial t}\right)$ is also called the local rate of change of that property (temperature in this case).

Substantial/Total derivative:

A Substantial derivative is the time derivative – rate of change – of a property following a fluid particle 'p'. It is also called as Material derivative and is a Lagrangian concept. Mathematically represented as

$$\frac{D}{Dt} \equiv \frac{\partial}{\partial t} + \vec{v} \cdot \nabla$$

Velocity and Acceleration Field

Since the 'continuum' assumption holds well for fluids, the description of any fluid property (such as density, pressure, velocity, acceleration etc.) can be expressed as a function of location. These representation as a function of spatial coordinates is called as "field representation" of the flow. One of the most important fluid variables is the velocity field. It is a vector function of position and time with components u, v and w as scalar variables i.e.

$$\vec{V} = u(x, y, z, t)\hat{i} + v(x, y, z, t)\hat{j} + w(x, y, z, t)\hat{k} \qquad (3.1.1)$$

The magnitude of the velocity vector i.e. $|\vec{V}| = \sqrt{(u^2 + v^2 + w^2)}$, is the speed of fluid. The total time derivative of the velocity vector is the acceleration vector field $(\vec{a})$ of the flow which is given as,

$$\vec{a} = \frac{d\vec{V}}{dt} = \frac{d\{u(x, y, z, t)\}}{dt}\hat{i} + \frac{d\{v(x, y, z, t)\}}{dt}\hat{j} + \frac{d\{w(x, y, z, t)\}}{dt}\hat{k} \qquad (3.1.2)$$

For instance, the scalar time derivative of u is expressed as,

$$\begin{aligned}\frac{d\{u(x,y,z,t)\}}{dt} &= \frac{\partial u}{\partial t}+\frac{\partial u}{\partial t}\frac{dx}{dt}+\frac{\partial u}{\partial t}\frac{dy}{dt}+\frac{\partial u}{\partial t}\frac{dz}{dt} \\ &= \frac{\partial u}{\partial t}+u\frac{\partial u}{\partial t}+v\frac{\partial u}{\partial t}+w\frac{\partial u}{\partial t} \\ &= \frac{\partial u}{\partial t}+(\vec{V}.\nabla)u\end{aligned} \qquad (3.1.3)$$

When u is replaced with v and w in the above equation, then the corresponding expressions would be,

$$\begin{aligned}\frac{d\{v(x,y,z,t)\}}{dt} &= \frac{\partial v}{\partial t}+(\vec{V}.\nabla)v \\ \frac{d\{w(x,y,z,t)\}}{dt} &= \frac{\partial w}{\partial t}+(\vec{V}.\nabla)w\end{aligned} \qquad (3.1.4)$$

Now, summing them into a vector quantity, one may write Eq. (3.1.2) in compact form as,

$$\begin{aligned}&\vec{a}=\frac{d\vec{V}}{dt}=\frac{\partial \vec{V}}{\partial t}+\left(u\frac{\partial \vec{V}}{\partial x}+v\frac{\partial \vec{V}}{\partial y}+w\frac{\partial \vec{V}}{\partial z}\right)=\frac{\partial \vec{V}}{\partial t}+(\vec{V}.\nabla)\vec{V} \\ &\text{where, } \vec{V}=u\frac{\partial}{\partial x}+v\frac{\partial}{\partial y}+w\frac{\partial}{\partial z} \text{ and } \nabla=\frac{\partial}{\partial x}\hat{i}+\frac{\partial}{\partial y}\hat{j}+\frac{\partial}{\partial z}\hat{k}\end{aligned} \qquad (3.1.5)$$

In the above equation, $\frac{\partial \vec{V}}{\partial t}$ is called as "local acceleration" and the second part i.e. $\left(u\frac{\partial \vec{V}}{\partial x}+v\frac{\partial \vec{V}}{\partial y}+w\frac{\partial \vec{V}}{\partial z}\right)$ is called a "convective acceleration". The total time derivative is called as "substantial/material" derivative. This field concept can be applied to any variable (vector or scalar). For example, one may write the total derivative for pressure and temperature field as,

$$\begin{aligned}\frac{dp}{dt} &= \frac{\partial p}{\partial t}+\left(u\frac{\partial p}{\partial x}+v\frac{\partial p}{\partial y}+w\frac{\partial p}{\partial z}\right)=\frac{\partial p}{\partial t}+(\vec{V}.\nabla)p \\ \frac{dT}{dt} &= \frac{\partial T}{\partial t}+\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\frac{\partial T}{\partial t}+(\vec{V}.\nabla)T\end{aligned} \qquad (3.1.6)$$

MOMENTUM EQUATION

Newton's second law of motion states that the rate of change of linear momentum for a material region (system) is equal to the sum of external forces acting on the system. For a particle of mass dm, this law can written as

$$\mathrm{d}\mathbf{F} = \frac{\mathrm{d}}{\mathrm{d}t}\left(\mathrm{v}\,\mathrm{d}m\right), \qquad \mathrm{d}m = \rho\,\mathrm{d}\Omega \tag{6.1}$$

Hence, for a finite material region, this law takes the form

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega} \mathrm{v}\rho\,\mathrm{d}\Omega = \mathbf{F} \qquad \text{or} \quad \frac{\mathrm{d}\mathbf{P}}{\mathrm{d}t} = \mathbf{F} \tag{6.2}$$

where **P** is the linear momentum of the system. The intensive property corresponding to **P** is $\phi \equiv \mathbf{v}$. Hence, from Reynolds transport theorem for a fixed control volume

$$\frac{\mathrm{d}\mathbf{P}}{\mathrm{d}t} = \underbrace{\frac{\partial}{\partial t}\int_{\Omega} \rho \mathrm{v}\,\mathrm{d}\Omega}_{\substack{\text{Rate of change of}\\ \text{momentum in the CV}}} + \underbrace{\int_{S} \rho \mathrm{v}\mathrm{v}\cdot \mathrm{d}\mathbf{A}}_{\substack{\text{Rate of efflux of momentum}\\ \text{across the control surface}}} \tag{6.3}$$

Net force **F** on the control volume can be expressed as sum of the surface force, $\mathbf{F}_S$ (pressure, viscous stress), and body force, $\mathbf{F}_B$ (gravity, electromagnetic, centrifugal, Coriolis etc.), i.e.

$$\mathbf{F} = \mathbf{F}_S + \mathbf{F}_B \tag{6.4}$$

The surface force $\mathbf{F}_S$ essentially represents microscopic momentum flux across a surface and can be expressed as

$$\mathbf{F}_S = \int_S \boldsymbol{\tau}\cdot \mathrm{d}\mathbf{A} \tag{6.5}$$

where $\boldsymbol{\tau}$ is the stress tensor. Body force $\mathbf{F}_B$ can be expressed as

$$\mathbf{F}_B = \int_{\Omega} \rho\mathbf{b}\ \mathrm{d}\Omega \tag{6.6}$$

where **b** is body force per unit mass. Combining (6.2)-(6.6), the integral form of momentum equation can be written as

$$\frac{\partial}{\partial t}\int_{\Omega} \rho\mathrm{v}\ \mathrm{d}\Omega + \underbrace{\int_S \rho\mathrm{v}\mathrm{v}\cdot \mathrm{d}\mathbf{A}}_{\text{Convective flux}} = \underbrace{\int_S \boldsymbol{\tau}\cdot \mathrm{d}\mathbf{A}}_{\text{Diffusive flux}} + \int_{\Omega} \rho\mathbf{b}\ \mathrm{d}\Omega \tag{6.7}$$

Note that since momentum is a vector quantity, its convective and diffusive fluxes are the scalar products of second order tensors $\rho\mathrm{v}\mathrm{v}$ and $\boldsymbol{\tau}$ with the surface vector d**A**.

MOMENTUM EQUATION IN DIFFRENTIAL FORM

For a fixed control volume, order of temporal differentiation and integration in Eq. (6.7) can be interchanged. Transform the convective and diffusive terms using Gauss divergence theorem, i.e.

$$\frac{\partial}{\partial t}\int_{\Omega} \rho \mathbf{v} d\Omega = \int_{\Omega} \frac{\partial(\rho \mathbf{v})}{\partial t} d\Omega, \quad \int_{S} \rho \mathbf{v}\mathbf{v} \cdot d\mathbf{A} = \int_{\Omega} \nabla \cdot (\rho \mathbf{v}\mathbf{v}) d\Omega \quad \text{and} \quad \int_{S} \boldsymbol{\tau} \cdot d\mathbf{A} = \int_{\Omega} \nabla \cdot \boldsymbol{\tau}\, d\Omega \qquad (6.8)$$

Substitution of Eq. (6.8) into Eq. (6.7) yields

$$\int_{\Omega} \left[\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla.(\rho \mathbf{v}\mathbf{v}) - \nabla.\boldsymbol{\tau} - \rho \mathbf{b} \right] d\Omega = 0 \qquad (6.9)$$

Equation (6.9) holds for any control volume which is possible only if the integrand vanishes everywhere, i.e.

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla.(\rho \mathbf{v}\mathbf{v}) = \nabla.\boldsymbol{\tau} + \rho \mathbf{b} \qquad (6.10)$$

Equation (6.10) is referred as the *conservative* or *strong conservation* form of momentum equation. It is also known as *Cauchy equation of motion*.

The integral form of momentum equation (6.7) or its differential form represented by Eq. (6.10) is applicable to an inertial control volume. Similar forms can be derived for moving control volumes and non-inertial reference frames (Batchelor 1973, Panton 2005, Kundu and Cohen 2008).

Further, using the identity

$$\nabla \cdot (\rho \mathbf{v}\mathbf{v}) = [\nabla \cdot (\rho \mathbf{v})] \mathbf{v} + \rho \mathbf{v} \cdot \nabla \mathbf{v} \qquad (6.11)$$

and chain rule of differentiation, we get

$$\frac{\partial(\rho \mathbf{v})}{\partial t} + \nabla.(\rho \mathbf{v}\mathbf{v}) = \left[\frac{\partial \rho}{\partial t} + \nabla.(\rho \mathbf{v}) \right] \mathbf{v} + \rho \left[\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right]$$

Using continuity equation (5.4), the first term on the RHS of the preceding equation vanishes. Thus, Eq. (6.10) takes the following form:

$$\rho \left[\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right] = \nabla.\boldsymbol{\tau} + \rho \mathbf{b} \qquad \text{or} \qquad \rho \frac{D\mathbf{v}}{Dt} = \nabla.\boldsymbol{\tau} + \rho \mathbf{b} \qquad (6.12)$$

where the operator (D/D*t*) denotes the material or particle derivative. Equation (6.12) is referred to as the *non-conservative form* of the momentum equation.

BERNOULLI'S EQUATION:

Prinicple of Conservation of Energy: Energy remains constant in a system

Bernoulli's equation is the alternate form of energy equation

The Bernoulli's equation is one of the most useful equations that is applied in a wide variety of fluid flow related problems. This equation can be derived in different ways, e.g. by integrating Euler's equation along a streamline, by applying first and second laws of thermodynamics to steady, irrotational, inviscid and incompressible flows etc. In simple form the Bernoulli's equation relates the pressure, velocity and elevation between any two points in the flow field. It is a scalar equation and is given by:

$$\underbrace{\frac{p}{\rho g}}_{\text{pressure head}} + \underbrace{\frac{V^2}{2g}}_{\text{velocity head}} + \underbrace{z}_{\text{static head}} = \underbrace{H}_{\text{total head}} = \text{constant} \qquad (6.7)$$

Each term in the above equation has dimensions of length (i.e., meters in SI units) hence these terms are called as pressure head, velocity head, static head and total heads respectively. Bernoulli's equation can also be written in terms of pressures (i.e., Pascals in SI units) as:

$$\underbrace{p}_{\text{static pressure}} + \underbrace{\rho\frac{V^2}{2}}_{\text{velocity pressure}} + \underbrace{\rho g z}_{\text{pressure due to datum}} = \underbrace{p_T}_{\text{total pressure}} \qquad (6.8)$$

Bernoulli's equation is valid between any two points in the flow field when the flow is *steady, irrotational, inviscid and incompressible*. The equation is valid along a streamline *for rotational, steady and incompressible flows*. Between any two points 1 and 2 in the flow field for irrotational flows, the Bernoulli's equation is written as:

$$\frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \qquad (6.9)$$

Bernoulli's equation can also be considered to be an alternate statement of conservation of energy (1st law of thermodynamics). The equation also implies the possibility of conversion of one form of pressure into other. For example, neglecting the pressure changes due to datum, it can be concluded from Bernoulli's equation that the static pressure rises in the direction of flow in a diffuser while it drops in the direction of flow in case of nozzle due to conversion of velocity pressure into static pressure and vice versa. Figure 6.2 shows the variation of total, static and velocity pressure for steady, incompressible and inviscid, fluid flow through a pipe of uniform cross-section.

Since all real fluids have finite viscosity, i.e. in all actual fluid flows, some energy will be lost in overcoming friction. This is referred to as head loss, i.e. if the fluid were to rise in a vertical pipe it will rise to a lower height than predicted by Bernoulli's equation. The head loss will cause the pressure to decrease in the flow direction. If the head loss is denoted by H_l, then Bernoulli's equation can be modified to:

$$\frac{p_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{V_2^2}{2g}+z_2+H_1 \qquad (6.10)$$

Figure 6.2 shows the variation of total, static and velocity pressure for steady, incompressible fluid flow through a pipe of uniform cross-section without viscous effects (solid line) and with viscous effects (dashed lines).

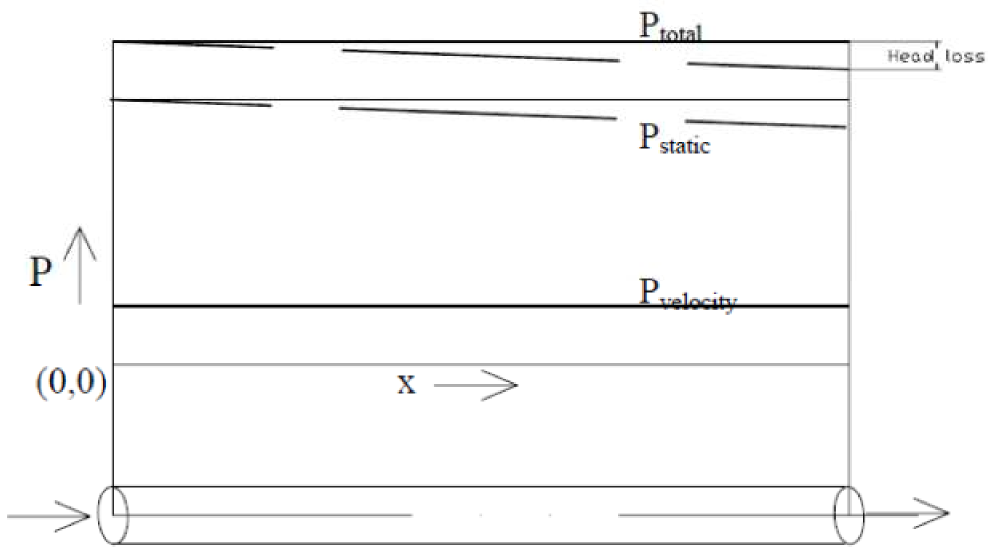


Fig. 6.2. *Application of Bernoulli equation to pipe flow*

Since the total pressure reduces in the direction of flow, sometimes it becomes necessary to use a pump or a fan to maintain the fluid flow as shown in Fig. 6.3.

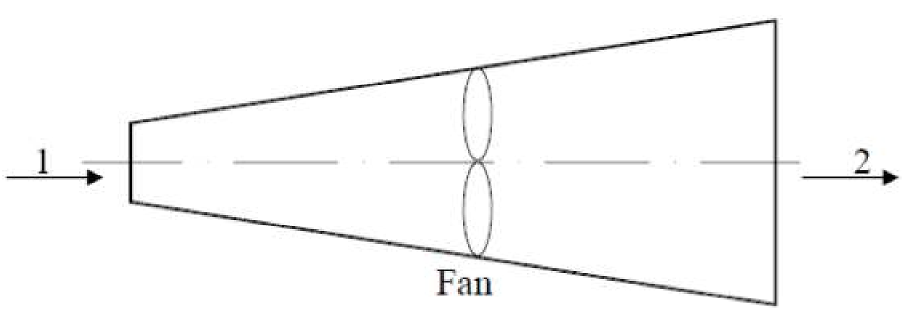


Fig. 6.3. *Air flow through a duct with a fan*

Energy is added to the fluid when fan or pump is used in the fluid flow conduit (Fig. 6.3), then the modified Bernoulli equation is written as:

$$\frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 + H_p = \frac{p_2}{\rho g} + \frac{V_2^2}{2g} + z_2 + H_1 \qquad (6.11)$$

where H_p is the gain in head due to fan or pump and H_l is the loss in head due to friction. When fan or pump is used, the power required (W) to drive the fan/pump is given by:

$$W = \left(\frac{\dot{m}}{\eta_{fan}}\right)\left(\frac{(p_2 - p_1)}{\rho} + \frac{(V_2{}^2 - V_1{}^2)}{2} + g(z_2 - z_1) + \frac{gH_1}{\rho}\right) \qquad (6.12)$$

where $\dot{m}$ is the mass flow rate of the fluid and η_{fan} is the energy efficiency of the fan/pump. Some of the terms in the above equation can be negligibly small, for example, for air flow the potential energy term $g(z_1-z_2)$ is quite small compared to the other terms. For liquids, the kinetic energy term $(v_2{}^2-v_1{}^2)/2$ is relatively small. If there is no fan or pump then W is zero.

6.1.4. Pressure loss during fluid flow:

The loss in pressure during fluid flow is due to:

a) Fluid friction and turbulence
b) Change in fluid flow cross sectional area, and
c) Abrupt change in the fluid flow direction

Normally pressure drop due to fluid friction is called as major loss or frictional pressure drop Δp_f and pressure drop due to change in flow area and direction is called as minor loss Δp_m. The total pressure drop is the summation of frictional pressure drop and minor loss. In most of the situations, the temperature of the fluid does not change appreciably along the flow direction due to pressure drop. This is due to the fact that the temperature tends to rise due to energy dissipation by fluid friction and turbulence, at the same time temperature tends to drop due to pressure drop. These two opposing effects more or less cancel each other and hence the temperature remains almost constant (assuming no heat transfer to or from the surroundings).

Evaluation of frictional pressure drop:

When a fluid flows through a pipe or a duct, the relative velocity of the fluid at the wall of the pipe/duct will be zero, and this condition is known as a *no-slip condition*. The no-slip condition is met in most of the common fluid flow problems (however, there are special circumstances under which the no-slip condition is not satisfied). As a result of this a velocity gradient develops inside the pipe/duct beginning with zero at the wall to a maximum, normally at the axis of the conduit. The velocity profile at any cross section depends on several factors such as the type of fluid flow (i.e. laminar or turbulent), condition of the walls (e.g. adiabatic or non-adiabatic) etc. This velocity gradient gives rise to shear stresses ultimately resulting in frictional pressure drop.

The Darcy-Weisbach equation is one of the most commonly used equations for estimating frictional pressure drops in internal flows. This equation is given by:

$$\Delta p_f = f\frac{L}{D}\left(\frac{\rho V^2}{2}\right) \quad (6.13)$$

where f is the dimensionless friction factor, L is the length of the pipe/duct and D is the diameter in case of a circular duct and hydraulic diameter in case of a noncircular duct. The friction factor is a function of Reynolds number, $Re_D = \left(\frac{\rho VD}{\mu}\right)$ and the relative surface of the pipe or duct surface in contact with the fluid.

For steady, fully developed, laminar, incompressible flows, the Darcy friction factor f (which is independent of surface roughness) is given by:

$$f = \frac{64}{Re_D} \quad (6.14)$$

For turbulent flow, the friction factor can be evaluated using the empirical correlation suggested by Colebrook and White is used, the correlation is given by:

$$\frac{1}{\sqrt{f}} = -2\log_{10}\left[\frac{k_s}{3.7D} + \frac{2.51}{(Re_D)\sqrt{f}}\right] \quad (6.15)$$

Where k_s is the average roughness of inner pipe wall expressed in same units as the diameter D. Evaluation of f from the above equation requires iteration since f occurs on both the sides of it.

ASHRAE suggests the following form for determination of friction factor,

$$f_1 = 0.11\left(\frac{k_s}{D} + \frac{0.68}{Re_D}\right)^{0.25} \quad (6.16)$$

If f_1 determined from above equation equals or exceeds 0.018 then f is taken to be same as f_1. If it is less than 0.018 then f is given by:

$$f = 0.85f_1 + 0.0028 \quad (6.17)$$

Another straightforward equation suggested by Haaland (1983) is as follows:

$$\frac{1}{f^{1/2}} \approx -1.8\log_{10}\left[\frac{6.9}{Re_D} + \left(\frac{k_s/D}{3.7}\right)^{1.11}\right] \quad (6.18)$$

Evaluation of minor loss, Δp_m:

The process of converting static pressure into kinetic energy is quite efficient. However, the process of converting kinetic energy into pressure head involves losses. These losses, which occur in ducts because of bends, elbows, joints, valves etc. are called minor losses. This term could be a misnomer, since in many cases these are more significant than the losses due to friction. For almost all the cases, the minor losses are determined from experimental data. In turbulent flows, the loss is proportional to square of velocity. Hence these are expressed as:

$$\Delta p_m = K\frac{\rho V^2}{2} \tag{6.19}$$

Experimental values for the constant K are available for various valves, elbows, diffusers and nozzles and other fittings. These aspects will be discussed in a later chapter on distribution of air.

Questions:

1. Is the flow incompressible if the velocity field is given by $\vec{V} = 2x^3\vec{i} - 6x^2y\vec{j} + t\vec{k}$? (Answer)

2. Derive the expression of fully developed laminar flow velocity profile through a circular pipe using control volume approach. (Answer)

3. A Static-pitot (Fig. Q3) is used to measure the flow of an inviscid fluid having a density of 1000 kg/m^3 in a 100 mm diameter pipe. What is the flow rate through the duct assuming the flow to be steady and incompressible and mercury as the manometer fluid? (Solution)

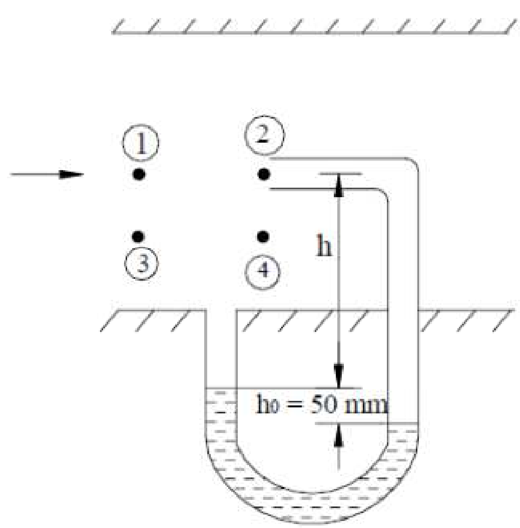


***Fig. Q3**. Figure of problem 3*

4. Calculate the pressure drop in 30 m of a rectangular duct of cross section 12.5 mm X 25 mm when saturated water at 60^{0}C flows at 5 cm/s? (Solution) Hint: Lundgrem

determined that for rectangular ducts with ratio of sides 0.5 the product of f.Re=62.19.

5. A fluid is flowing though a pipeline having a diameter of 150 mm at 1 m/s. The pipe is 50 m long. Calculate the head loss due to friction? (Solution) (Density and viscosity of fluid are 850 kg/m^3 and 0.08 kg/m.s respectively)

6. A fluid flows from point 1 to 2 of a horizontal pipe having a diameter of 150 mm. The distance between the points is 100 m. The pressure at point 1 is 1 MPa and at point 2 is 0.9 MPa. What is the flow rate? (Solution) (Density and kinematic viscosity of fluid are 900 kg/m^3 and 400 X 10^{-6} m^2/s respectively)

7. Three pipes of 0.5 m, 0.3 m and 0.4 m diameters and having lengths of 100 m, 60 m and 80 m respectively are connected in series between two tanks whose difference in water levels is 10 m as shown in Fig. Q7. If the friction factor for all the pipes is equal to 0.05, calculate the flow rate through the pipes. (Solution)

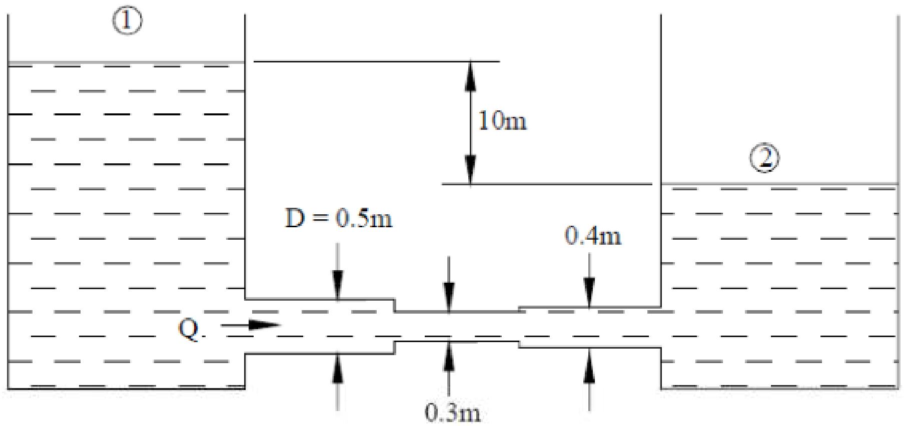


Fig. Q7. *Figure of problem 7*

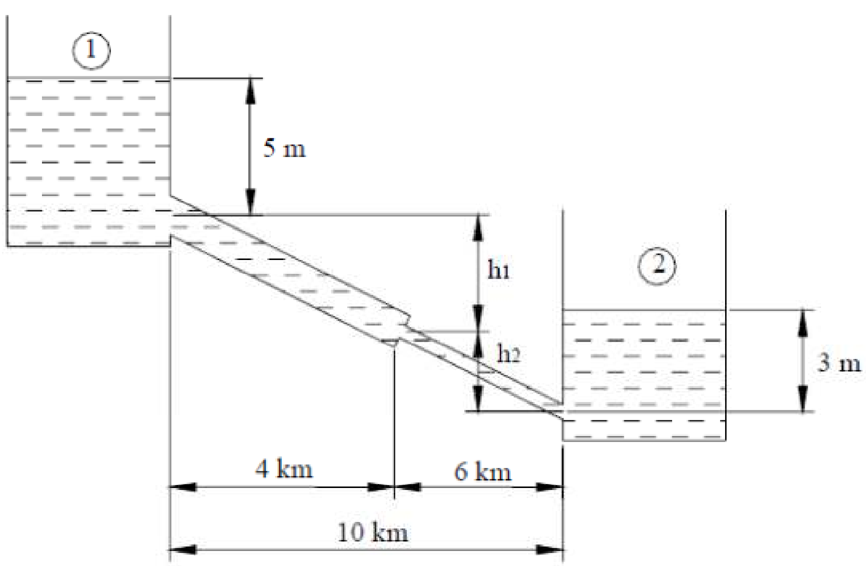


Fig. Q8. *Figure of problem 8*

8. Two reservoirs 10 kms apart is connected by a pipeline which is 0.25 m in diameter in the first 4 kms, sloping at 5 m per km, and the remaining by a 0.15 m diameter sloping at 2 m per km as is shown in Fig. Q8. The levels of water above the pipe openings are 5 m and 3 m in the upper and lower reservoirs respectively. Taking f = 0.03 for both pipes and neglecting contraction and exit losses at openings calculate the rate of discharge through the pipelines. (Solution)

9. A 10 cm hose with 5 cm discharges water at 3 m^3/min to the atmosphere as is shown in Fig. Q9. Assuming frictionless flow, calculate the force exerted on the flange bolts. (Solution)

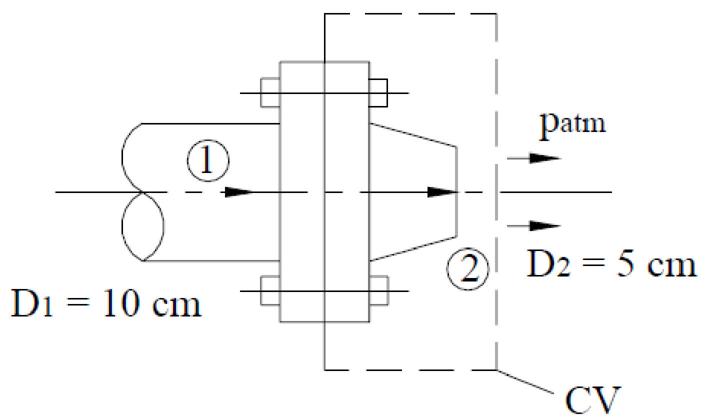


Fig. Q9. *Figure of problem 9*

Fluid flow Measurement

In the physical world, mechanical engineers are frequently required to monitor or control the flow of various fluids through pipes, ducts and assorted vessels. This fluid can range from thick oils to light gasses. While some techniques work better with some groups of fluids, and less well with others, some are not at all suitable for some applications. In this primer on fluid flow instrumentation we will look at a wide variety of flow transducers and their application in the physical world.

1.0 Fluid flow measurement

Fluid flow measurement can encompass a wide variety of fluids and applications. To meet this wide variety of applications the instrumentation industry has, over many years, developed a wide variety of instruments. The earliest known uses for flow come as early as the first recorded history. The ancient Sumerian cities of UR and Kish, near the Tigris and Euphrates rivers (around 5000 B.C.) used water flow measurement to manage the flow of water through the aqueducts feeding their cities. In this age the a simple obstruction was placed in the water flow, and by measuring the height of the water flowing over the top of the obstruction, these early engineers could determine how much water was flowing. In 1450 the Italian art architect Battista Alberti invented the first mechanical anemometer. It consisted of a disk placed perpendicular to the wind, and the force of the wind caused it to rotate. The angle of inclination of the disk would then indicate the wind velocity. This was the first recorded instrument to measure wind speed. An English inventor, Robert Hooke reinvented this device in 1709, along with the Mayan Indians around that same period of time. Today we would look down our noses at these crude methods of flow measurement, but as you will see, these crude methods are still in use today.

1.1 Types of flow measurement devices

Fluid flow devices fall into a number of device categories as well as fluid classes. In general we can split the fluids into two classes; gasses and liquids. Within these two broad classes are a number of special classes that one should be careful of. Flammable liquids and gasses require special handling, as do those that are at temperature extremes (cold or hot). When selecting a transducer you should be cautious that the device you are selecting is compatible with the fluid and conditions you hare working with. A few examples would be acids, food grade liquids, and DI water. Surprisingly de-ionized water is an extremely harsh liquid that can cause serious headaches.

The physical measurement devices come in a number of classifications. While the following classifications do not match any industry standards, they serve to break the transducers down into some reasonably functional groups. These are:

Obstruction flow meters
Velocity flow meters – Including Moving Member meters
Positive Displacement meters
Variable area meters
Electronic meters

We will spend some time at each category, looking at the particular devices that fall into that category. Some of these devices will work with a wide array of fluids, while others have significant limitations. This tutorial should help you understand what these restrictions are and when to use or not use a particular meter.

2.0 Obstruction flow meters

Obstruction flow meters are the simples and oldest of the measurement classes. One of the first obstruction flow meters was used by the ancient Samarians. In order to measure the amount of water flowing through an aquaduct, they would place a board across the flow, and measure how high the water was when it flowed over the top of the board. In this way they could easily calculate how much water was flowing in the duct. This was modified in later times to a device called a "notch" weir.

2.1 Notch weir

Notch weirs are classified by the shape of their notch; rectangular weirs; triangular, or **V-notch**, weirs; trapezoidal weirs; and parabolic weirs.

http://waterknowledge.colostate.edu/v_notch.htm

The picture above shows a V-notch weir. The edge the water cascades over is called the crest and the overflowing water sheet is called the nappe. Today weirs are still used to determine flows from open water sources such as streams. A typical 90° V-notch will be beveled at 45° so the edge is less than 0.08" thick, and the angle of the notch will be

precisely 90°. Water flow over the weir is calculated by the equation : $Q=2.49h_1^{2.48}$, where h_1 = head on the weir in ft and Q = discharge over weir in ft^3/s.
It is easy to see that this is a simple measurement technique can be used on nearly any open flowing body of water. Its simply a matter of building a large enough weir plate. It is just as obvious that this technique wont work in an enclosed pipe, and it certainly wont work for gasses. The measurement of head is the height of the water above the lowest portion of the weir, and should be made at least four times that height, back from the weir.

2.2 Orifice plate

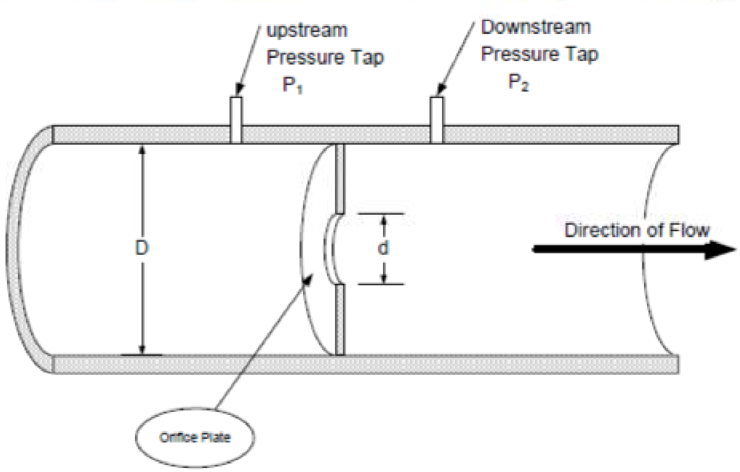
The equivalent of the notch weir in a tube would be an orifice plate. This flow device is created by inserting an obstructing plate, usually with a round hole in the middle, into the pipe and measuring the pressure on each side of the orifice. This is again a very simple device that has been in use for measuring both gas flow and liquid flow for decades.

These plates are generally installed by trapping it between two pipe flanges. Pressure taps on each flange allow you to easily measure the pressure differential across the plate. This pressure differential, along with the dimensions of the plate, are combined with certain fluid properties to determine the flow through the pipe.

The calculation for incompressible (liquid) flow is described by the incompressible Bernoulli equation, as long as the flow is sub-sonic (< mach 0.3).

$$\Delta P = \frac{1}{2}\rho V_2^2 - \rho V_1^2$$

Given the following physical layout, you can modify this formula to take into account the dimensional information rather than the velocity. Also the equation above assumes perfectly laminar flow, which is generally not the case in the real world. Flows in pipe will have a certainly amount of turbulence, which acts to convert kinetic flow energy into heat. This effect is taken into account by adding a new term



to the equation called a *discharge coefficient* (C_d). The resulting equation shows how the area and this new coefficient are applied to get a flow rate (Q).

$$Q = C_d \sqrt{\frac{2(P_1 - P_2)}{\rho}} \times \frac{A_2}{\sqrt{1 - \left(\frac{A_2}{A_1}\right)^2}}$$

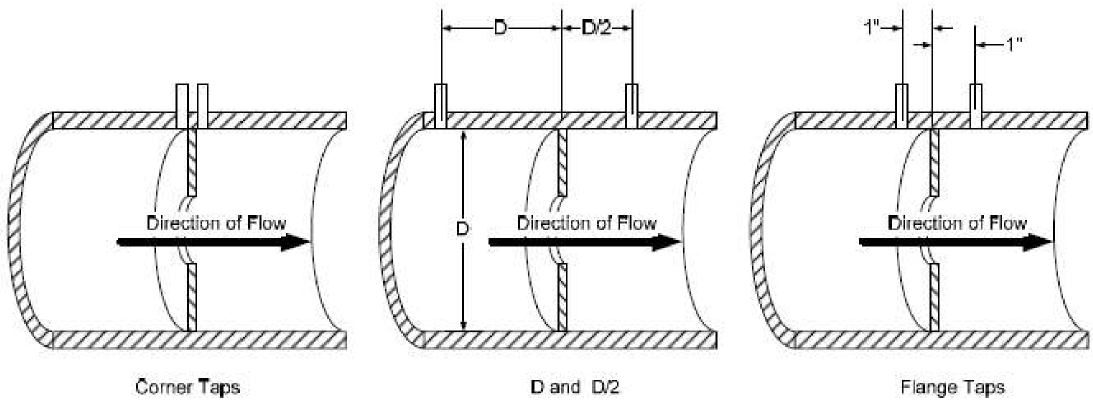
Since the actual flow profile at location 2 (downstream) is quite complex, making the effective value of A_2 uncertain, a substitution is made and a new coefficient C_f is put in place of the area and C_d. The new equation looks like:

$$Q = C_f A_o \sqrt{\frac{2\Delta P}{\rho}}$$

As you can see the formula has been simplified significantly, now only requiring the value of C_f, the area of the orifice (A_o), the density of the fluid and the differential pressure to obtain the volumetric flow rate. The only problem now is the value of C_f. The flow coefficient is found experimentally and is tabulated in numerous reference books. This value ranges from 0.6 to 0.9 for most orifices, and the value depends on the orifice and pipe diameters as well as the Reynolds Number.

Discharge Coefficient - c_d				
Diameter Ratio	Reynolds Number - *Re*			
d / D	104	105	106	107
0.2	0.6	0.595	0.594	0.594
0.4	0.61	0.603	0.598	0.598
0.5	0.62	0.608	0.603	0.603
0.6	0.63	0.61	0.608	0.608
0.7	0.64	0.614	0.609	0.609

This takes care of the incompressible flow, but what about compressible flow, such as air. In this case the orifice flowmeter becomes much more difficult to calculate. The placement of the pressure taps even effect the calculations.



Without going into detail here are the equations for calculating the flow for a gas in pipes larger than 5CM in diameter. These are based on ISO 1991 and 1998.

Corner Pressure Taps: $L_1 = L'_2 = 0$
D and D/2 Pressure Taps: $L_1 = 1$ and $L'_2 = 0.47$
Flange Pressure Taps: $L_1 = L'_2 = 0.0254/D$ where D is in meters

$$Q_m = \frac{eC\,A_{throat}\sqrt{2\rho\Delta p}}{\sqrt{1-\beta^4}} \qquad Q_a = \frac{Q_m}{\rho} \qquad Q_s = Q_a\frac{P_1 T_{std}}{P_{std} T}$$

$$\rho = \frac{P_1}{RT} \qquad e = 1-(0.41+0.35\beta^4)\frac{\Delta P}{K P_1} \qquad \beta = \frac{d}{D}$$

$$\mathrm{Re}_D = \frac{V_{pipe} D}{\nu} \qquad \mathrm{Re}_d = \frac{V_{throat}\, d}{\nu} \qquad \nu = \frac{\mu}{\rho}$$

$$w = \frac{\sqrt{1-\beta^4} - C\beta^2}{\sqrt{1-\beta^4} + C\beta^2}\Delta P \qquad K_m = \frac{2w}{\rho V_{pipe}^2} \qquad V_{pipe} = \frac{Q_a}{A_{pipe}}$$

$$V_{throat} = \frac{Q_a}{A_{throat}} \qquad A_{pipe} = \frac{\pi}{4}D^2 \qquad A_{throat} = \frac{\pi}{4}d^2$$

Discharge Coefficient:

$$C = 0.5961 + 0.0261\beta^2 - 0.216\beta^8 + 0.000521\left(\frac{10^6\beta}{Re_D}\right)^{0.7}$$

$$+\left(0.0188 + 0.0063\left(\frac{19000\beta}{Re_D}\right)^{0.8}\right)\left(\frac{10^6}{Re_D}\right)^{0.3}\beta^{3.5}$$

$$+\left(0.043 + 0.08e^{-10L_1} - 0.123e^{-7L_1}\right)\left(1 - 0.11\left(\frac{19000\beta}{Re_D}\right)^{0.8}\right)\frac{\beta^4}{1-\beta^4}$$

$$-0.031\left(\frac{2L'_2}{1-\beta} - 0.8\left(\frac{2L'_2}{1-\beta}\right)^{1.1}\right)\beta^{1.3}$$

If $D < 0.07112$ m (2.8 inch), then add the following (where D is in meters):

$$+0.011(0.75-\beta)\left(2.8 - \frac{D}{0.0254}\right)$$

Variables for the above equations:

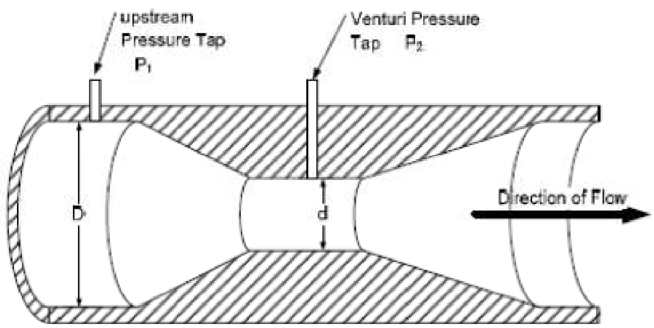
Dimensions: F=Force, L=Length, M=Mass, T=Time, t=temperature

A_{pipe} =Pipe Area [L^2], A_{throat} =Throat Area [L^2], C=Discharge Coefficient
d=Throat Diameter [L], D=Pipe Diameter [L], e=Gas Expansibility
k=Equivalent Roughness of Pipe Material [L]
K=Gas Isentropic Exponent, K_m=Minor Loss Coefficient
M=Mass Flowrate [M/T]
P_1 =Upstream Absolute Pressure [F/L^2], P_2 =Downstream Absolute Pressure [F/L^2]
ΔP =Differential Pressure [F/L^2] = P_1-P_2
P_{std} =Standard Absolute Pressure =14.73 psia = 1.016×10^5 N/m^2
Q_a =Actual Volumetric Flowrate [L^3/T]
Q_s =Volumetric Flowrate at Standard Pressure and Temperature [L^3/T]
R=Gas Constant (used to compute gas density) = 8312/W N-m/kg-K
Re_d =Reynolds Number based on d, Re_D =Reynolds Number based on D
T=Gas Temperature [t] (converted automatically to absolute)
T_{std} =Standard Absolute Temperature=520 ^{0}R=288.9K
V_{pipe}=Gas Velocity in Pipe [L/T], V_{throat} =Gas Velocity in Throat [L/T]
w=Static Pressure Loss [F/L^2], W=Molecular Weight of Gas [gram/mole]
β =Ratio d/D, ρ=Gas Density [M/L^3], μ =Gas Dynamic Viscosity [F-T/L^2]
ν =Gas Kinematic Viscosity [L^2/T]

The primary disadvantage of the orifice type flow meter is that there is a significant pressure drop across the plate, which is not recoverable. For this reason selection of this meter must only be used where you can afford the pressure drop without affecting the rest of the system operations. This is also the reason that the next flow meter type was developed.

2.3 Venturi Flow meter

The venturi flow meter, while considered an obstruction flow meter, is less of an obstruction than the orifice type. It still does have a certain amount of pressure drop, but it is significantly less than the orifice type meter.



Once again, as long as the incompressible fluid velocity is well below the supersonic point (< mach .3), the Bernoulli equation can be used.

$$\Delta P = \frac{1}{2}\rho V_2^2 - \rho V_1^2$$

From continuity we can substitute the throat velocity (V_2) out of the above equation, yielding the following:

$$\Delta P = \frac{1}{2}\rho V_1^2\left[\left(\frac{A_1}{A_2}\right)^2 - 1\right]$$

Solving for the upstream velocity and multiplying by the cross sectional area gives the volumetric flow rate Q.

$$Q = \sqrt{\frac{2\Delta P}{\rho}}\frac{A_1}{\sqrt{\left(\frac{A_1}{A_2}\right)^2 - 1}}$$

Ideal fluids would obey this equation, however small amounts of energy are converted into heat within the viscous boundary layers, and tend to lower the actual velocity of real fluids. A discharge coefficient C is typically introduced to account for the viscosity of the fluid.

$$Q = C\sqrt{\frac{2\Delta P}{\rho}}\frac{A_1}{\sqrt{\left(\frac{A_1}{A_2}\right)^2 - 1}}$$

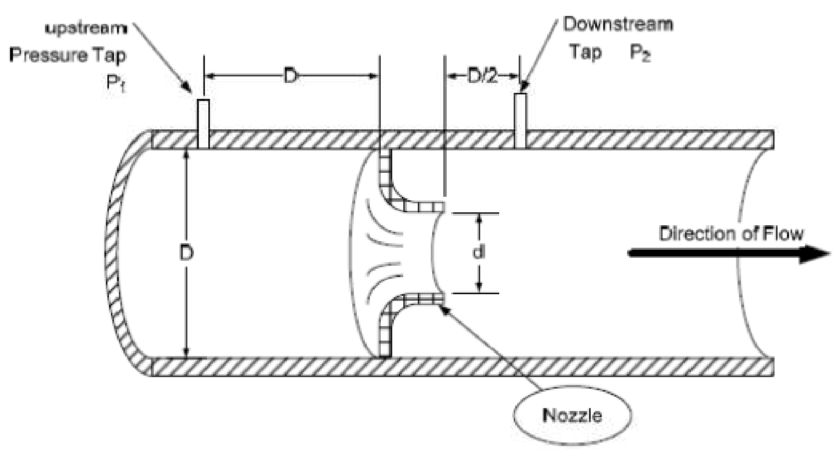
C is found to depend on the Reynolds Number of the flow, and usually lies between .90 and .98 for smoothly tapering venturis.

For air flow you can use the same calculation and assume that the gas is incompressible. The density needs to be adjusted appropriately using the ideal gas formula.

$$\rho = \frac{P}{RT}$$ Where R is the gas constant (287 J/Kg/K for air)

2.4 Nozzle Flow meter

A flow nozzle consists of a restriction with an elliptical contour approach section that terminates in a cylindrical throat section. Pressure drop between the locations one pipe diameter upstream and one-half pipe diameter downstream is measured. Flow nozzles provide an intermediate pressure drop between orifice plates and venturi tubes; also, they are applicable to some slurry systems that would be otherwise difficult to measure.



The flow calculations for the long radius nozzle are similar to that of the orifice plate, with the exception of the values of the discharge coefficient. The following table shows some standard values for this value.

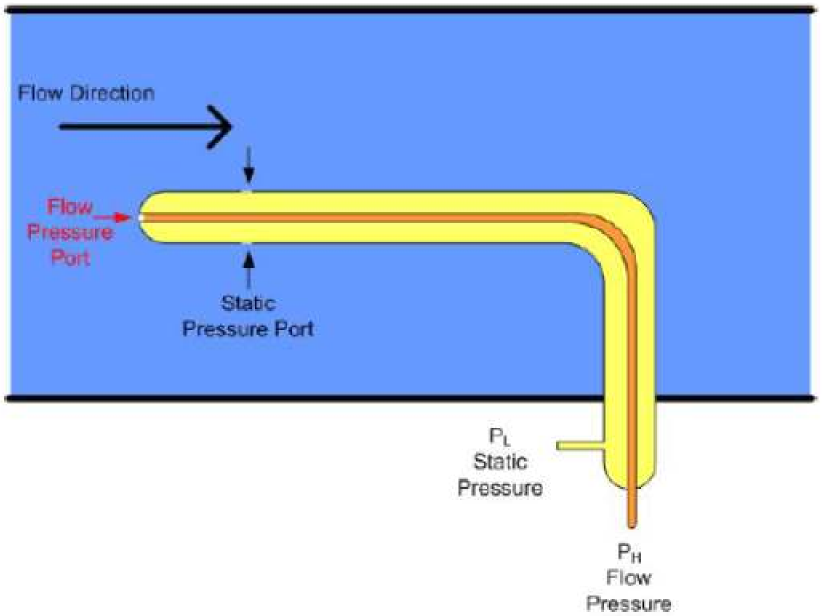
Discharge Coefficient - c_d				
Diameter Ratio	Reynolds Number - *Re*			
d / D	104	105	106	107
0.2	0.968	0.988	0.994	0.995
0.4	0.957	0.984	0.993	0.995
0.6	0.95	0.981	0.992	0.995
0.8	0.94	0.978	0.991	0.995

3.0 velocity flow measurement devices

Velocity flow measurement techniques allow for the measurement of total flow by measuring the velocity of the fluid within a fixed area duct or pipe. The technique uses a measuring probe to determine the velocity of the fluid in the *center portion* of the pipe. It is important to understand that with all fluid flows, there are boundary layer effects at the interface between the walls of the duct or pipe and the fluid flowing through it. For this technique to provide reasonably accurate results, the velocity measurement of the flow must be made well within the duct, to minimize the effects of the boundary layers. For this reason ducts or piles of small diameter typically do not fair well with this technique. The technique also requires that you be in a laminar flow environment. The results in a turbulent flow area suffer in stability and accuracy. It is possible to calculate the location where the flow in a pipe or duct is fully laminar, but for most applications a general rule of thumb is sufficient. That rule is to make the measurement at least 10 pipe diameters upstream and 20 pipe diameters downstream of any junction, elbow or other flow disturbing point in the pipe.

3.1 Pitot Tube

The Pitot tube is a simple device that allows for the measurement of the flow pressure in a moving fluid. This device is a section of tube that measures the pressure at the tip and the pressure at the side of the tube. Reading this differential pressure and applying Bernoulli's equation will allow for the calculation of the fluid velocity.



The above diagram shows how the Pitot tube is constructed of two tubes, one inside the other, to create a static pressure port and a flow pressure port. Applying Bernoulli's equation we get:

$$P_S + r\left(\frac{V^2}{2}\right) = P_F \qquad \text{Where} \quad \begin{aligned} r &= \text{Density} \\ V &= \text{Velocity} \\ P_S &= \text{Static Pressure} \\ P_F &= \text{Flow Pressure} \end{aligned}$$

If we solve for the velocity we get the following equation:

$$V^2 = \frac{2(P_F - P_S)}{r}$$

With the velocity of the fluid now known, you can simply multiply it by the area of the duct to get the total volume flow.

This process is extremely useful in locations where there is a significant volume in a large duct or pipe. The differential pressure between the two ports is typically quite small for air flow, and the use of water monometers is a common method of measuring the pressure differential. Small differential pressure transducers are also quite common when an electronic readout is required or desired. Liquid flows can have significantly larger pressure differentials. As with the obstruction flow meters, the fluid that is within the pipe or duct will be on the pressure taps. If this fluid has any nasty properties, you need to take the appropriate steps to protect personnel and equipment. Not all fluids are compatible with all pressure transducers and care must be taken to ensure that an appropriate material is used for all wetted parts.

3.2 Hot Wire / Film probes

While Pitot tubes work well for high flow rates in gasses, and a variety of flow rates in liquids, the technique fails for low air velocities in gasses. To solve this gap in velocity measurement technology, the hot wire and hot film probes were developed.

http://www.omega.com/pptst/HHF42.html

This technique is fairly straight forward in concept, but much more difficult in operation. The theory is that if you place a resistance wire in the flow of air (or other gas) and heat the wire with a fixed current, the voltage across the wire will indicate the resistance of the wire. If you know the properties of the wire you can deduce what its temperature is. Knowing this information, you can determine how much heat is being carried away by the moving stream of gas flowing across the wire or film. Simple… maybe. The difficulty with this is that the density, temperature and actual makeup of the gas flowing affect the heat absorption as well as the flow. This has been handled in a number of ways, but the most straightforward is to use two wires. One in the flow and one out of the flow, and make your measurement based on the difference of these two values. A second method is to make an assumption that the reading is being made in "standard air" which has a known coefficient of absorption. Using this method the only values that are needed are hot wire value and the temperature of the air prior to the hot wire.

Hot wire probes are extremely fast response devices. With a wire size in the micrometers, the probe can respond to temperature changes at rates faster than 1 millisecond. This makes this type of probe ideal for studies of turbulent flows. Scientific level meters are available from a number of companies that will respond to these high rates of change, but the price is generally in the thousands. Smaller hand held units that respond much slower are available for a few hundred dollars and are a good solution to a low flow application. The accuracy of these devices is typically around 1% or so and are generally designed for use in air, although most can be calibrated for other gasses as well.

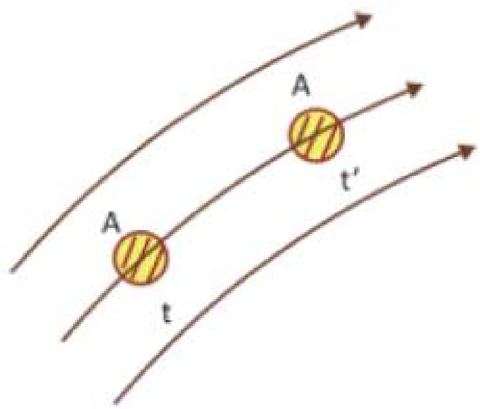
APPENDIX – FOR INFORMATION

FLUID FLOW MODELS – EULERIAN AND LAGRANGIAN APPROACH

There are two approaches to describe the motion of a fluid and its associated properties. 1. Lagrangian approach 2. Eulerian approach

Lagrangian approach:

Identify (or label) a material of the fluid; track (or follow) it as it moves, and monitor change in its properties. The properties may be velocity, temperature, density, mass, or concentration, etc in the flow field.



Refer the above-figure.

The 'material' or 'particle' of the fluid 'A' at time t has moved to some other location at time t'. Its property, say temperature, is recorded, as the material moves in the flow-field: Note that the recorded temperatures are associated with the same fluid particle, but at different locations and at different times.

t_1 -> T_1

t_2 -> T_2

t_3 -> T_3…..

Think of a temperature sensor attached to a balloon, both having negligible mass and floating in the atmosphere and recording the atmosphere-temperature or the temperature of the flow-field.

In such case, the following temperature-data are recorded by the sensor: The time change of the temperature in such a measurement is denoted as which is called material derivative or substantial derivative.

Location	Time	Temperature
X_1 , Y_1, Z_1	t_1	T_1
X_2 , Y_2, Z_2	t_2	T_2
X_3 , Y_3, Z_3	t_3	T_3

It reflects time change in the temperature (or any other properties) of the labeled /marked/tagged fluid particles as observed by an observer moving with the fluid.

Lagrangian approach is also called "particle based approach".

Eulerian approach (a field approach)

Identify (or label) a certain fixed location in the flow field and follow change in its property, as different materials pass through that location. In such case, the following property, say temperature is recorded by the sensor **:**

$t_1 \rightarrow T_1$

$t_2 \rightarrow T_2$

$t_n \rightarrow T_n$…..

Note that the recorded temperatures are associated with the fixed location in the flow-fluid, having different fluid elements at different times.

The time- change of the temperature in such a measurement is denoted as $\left.\frac{\partial T}{\partial t}\right]_{(x,y,z)}$ which is called the partial derivative of the temperature with respect to time. Note that the suffix (x,y,z) implies that the observer records the change in the property at the fixed location (x,y,z). is also called the local rate of change of that property (temperature in this case).

Based on the above, the following 4 approaches can be deduced for Mathematical modeling

1. Control volume fixed in space
2. Control volume moving in space
3. Infinetisimal small Control volume fixed in space
4. Infinetisimal small control volume moving in space

For further more reading refer Fundamentals of Aerodynamics by John.D. Anderson

FUNDAMENTAL PRINCIPLES GOVERNING THE FLUID FLOW

There are three fundamental principles governing the fluid flow. Viz:

1. Principle of conservation of Mass -- Mathematical model for the same is C**ontinuity Equation**
2. Principle of conservation of Energy -- Mathematical model for the same is **Energy Equation**
3. Principle of conservation of Momentum -- Mathematical model for the same is **Momentum Equation**

MATHEMATICAL EQUATIONS DERIVED FROM THE FUNDAMENTAL PRINCIPLES – GOVERNING EQUATIONS

2.1 Continuity Equation (integral and differential forms)

Here, the control volume is fixed in space, with the flow moving through it . the volume V and control surface S are now constant with time, and the mass of fluid contained within the control volume can change as a function of time (due to unsteady fluctuations of the flow field).

Consider a given area A arbitrarily oriented in a flow field as shown in the figure 7.1. In this figure we are looking at an edge view of area A.

Let A be small enough such that the flow velocity V is uniform across A. Consider the fluid elements with velocity V that pass through A. In time *dt* after crossing A, they have moved a distance V*dt* and have swept out the shaded volume shown in figure 7.1 . This volume is equal to the base area A times the height of the cylinder $V_n dt$, where V_n is the component of velocity normal to A i.e.,

$$\text{Volume} = (V_n\, dt)A$$

The mass inside the shaded volume is therefore

$$\text{Mass} = \rho(V_n\, dt)A \qquad \text{.......... (7.1)}$$

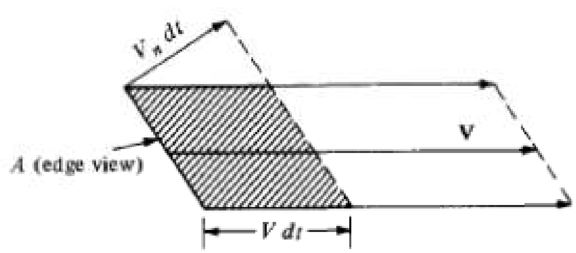
This is the mass that has swept past A in time *dt*. By definition, the mass flow through A is the mass crossing A per second (e.g., kilograms per second, slugs per second).

Let ṁ denote mass flow. From equation (7.1)

$$\dot{m} = \frac{\rho(V_n\, dt)A}{dt}$$

Or

$$\boxed{\dot{m} = \rho V_n A} \qquad \text{.......... (7.2)}$$



sketch for discussion of mass flow through area A in a flow field.

Equation (7.2) Demonstrates that mass flow through A is given by the Product

Area × density × component of flow velocity *normal* to the area

A related concept is that of *mass flux*, defined as the mass flow *per unit area*.

$$\text{Mass flux} = \frac{\dot{m}}{A} = \rho V_n$$ ……………………………………………….. (7.3)

Typical units of mass flux are kg/(s.m^2) and slug/(s.ft^2).

The concepts of mass flow and mass flux are important. Note from equation (7.3) the mass flux across a surface is equal to the product of density times the component of velocity perpendicular to the surface.

In a more general sense, if V is the magnitude of velocity in an arbitrary direction, the product ρV is physically the mass flux (mass flow per unit area) across an area oriented perpendicular to the direction of V.

We are now ready to apply our first physical principle to a finite control volume fixed in space.

Physical Principle: Mass can neither be created nor destroyed.

Consider a flow field wherein all properties vary with spatial location and time e.g. $\rho=\rho(x,y,z,t)$. In this flow field, consider the fixed finite control volume shown in below figure. At a point on the control surface, the flow velocit **V** and the vector elemental surface area is **dS.** Also d$\mathcal{V}$ is an elemental volume inside the control volume. Applied to this control volume, the above physical principle means

$$\begin{matrix}\text{Net mass flow } \textit{out} \text{ of control} \\ \text{volume through surface } S\end{matrix} = \begin{matrix}\text{time rate of decrease of} \\ \text{mass inside control volume } \mathcal{V}\end{matrix}$$

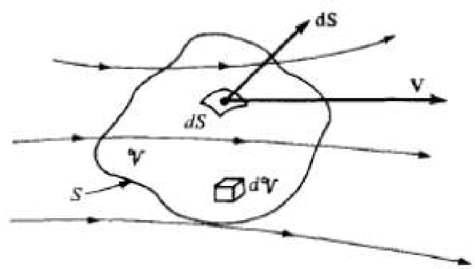
…………………………(7.4a)

Or $B=C$ ………………………………………………………………………………………(7.4b)

Where B and C are just convenient symbols for the left and right sides, respectively, of equation (7.4a). first let us obtain an expression for B in terms of the quantities shown in figure 7.2 below. From equation (7.2), the elemental mass flow across the area dS is

$$\rho V_n \, dS = \rho \mathbf{V} \cdot \mathbf{dS}$$

Examining the figure 7.2, note that by convention, **dS** always point in a direction out of the control Volume. Hence, when V also point out of the control volume(as shown in figure 7.2), the product ρ***V. dS*** is posititve.



2.2 Finite Control volume fixed in space

Moreover, when V points out of the control volume, the mass flow is physically leaving the control volume; i.e., it is an *outflow*. Hence, a positive $\rho\mathbf{V}\cdot\mathbf{dS}$ denotes an outflow. In turn, when V points into the control volume, $\rho\mathbf{V}\cdot\mathbf{dS}$ is *negative* . moreover, when V points inward, the mass flow is physically entering the control volume; i.e., it is an inflow. Hence , a negative ρV. dS denotes an ***inflow.*** The net mass flow out of the entire control surface S is the Summation over S of the elemental mass flow, in the limit, this becomes a surface integral, which is physically the left side of equations(7.4a&b); i.e.,

$$B = \oiint_S \rho \mathbf{V} \cdot \mathbf{dS} \qquad \text{………………………………………………….(7.5)}$$

Now consider the right side of equations (7.4a&b). the mass contained within the elemental volume d*V* is

$$\rho\, d\mathcal{V}$$

Hence, the total mass inside the control volume is

$$\oiiint_{\mathcal{V}} \rho\, d\mathcal{V}$$

The time rate of increase of mass inside *V* is then

$$\frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho\, d\mathcal{V}$$

In turn, the time rate of decrease of mass inside *V* is the negative of the above; i.e.,

$$-\frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho \, d\mathcal{V} = C$$

……………………………………………………………………………..……(7.6)

Thus, substituting equations (7.5) and (7.6) into (7.4 b), we have

$$\oiint_{S} \rho \mathbf{V} \cdot \mathbf{dS} = -\frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho \, d\mathcal{V}$$

Or

$$\boxed{\frac{\partial}{\partial t}\oiiint_{\mathcal{V}} \rho \, d\mathcal{V} + \oiint_{S} \rho \mathbf{V} \cdot \mathbf{dS} = 0}$$

…………………………………………………….. (7.7)

The above equation is called the *Continuity equation*. It is one of the most fundamental equations of *fluid dynamics.* It expresses the continuity equation in integral form.

However, equations (7.7) is fixed in space, the limits of integration are also fixed. Hence the time derivative can be placed inside the volume integral and equation (7.7) can be written as

$$\oiiint_{\mathcal{V}} \frac{\partial \rho}{\partial t} d\mathcal{V} + \oiint_{S} \rho \mathbf{V} \cdot \mathbf{dS} = 0$$

………………………………………… (7.8)

Applying the divergence theorem, we can express the right-hand term of equation (7.8) as

$$\oiint_{S} (\rho \mathbf{V}) \cdot \mathbf{dS} = \oiiint_{\mathcal{V}} \nabla \cdot (\rho \mathbf{V}) \, d\mathcal{V}$$

……………………………………………. (7.9)

Substituting Equation (7.9) into (7.8), we obtain

$$\oiiint_{\mathcal{V}} \frac{\partial \rho}{\partial t} d\mathcal{V} + \oiiint_{\mathcal{V}} \nabla \cdot (\rho \mathbf{V}) \, d\mathcal{V} = 0$$

Or

$$\oiiint_{\mathcal{V}} \left[\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V})\right] d\mathcal{V} = 0$$

…………………………………………………… (7.10)

However, the finite control volume is arbitrarily drawn in space, there is no reason to expect cancellation of one region by the other. Hence, the only way for the integral in equation (7.10) to be zero for an arbitrarily control volume is for the integrand to be zero at all points within the control volume. Thus, from equation (7.10), we have

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0$$

…………………………………………………………..………(7.11)

Equation (7.11) is the continuity equation in the form of a partial differential equation. This equation relates the flow field variables at a point in the flow, as opposed to the equation (7.7), which deal with a finite space.

It is important to emphasize the difference between unsteady and steady flows. In an unsteady flow, the flow-field variables are a function of both spatial location and time, e.g,

$$\rho = \rho(x, y, z, t)$$

This means that if you lock your eyes on one fixed point in space, the density at that point will change with time. Such unsteady fluctuations can be caused by time-varying boundaries. Equations (7.7) and (7.10) hold for such unsteady flows. On the other hand, the vast majority of practical aerodynamic problems involve steady flow. Here, the flow-field variables are a function of spatial location only, e.g.

$$\rho = \rho(x, y, z)$$

The density at that point will be a fixed value, invariant with time. For steady flow $\partial/\partial t = 0$, and hence Equations (7.7) and (7.10) reduce to

$$\oiint_S \rho \mathbf{V} \cdot \mathbf{dS} = 0$$

…………………………………………………………………….
(7.12)

And

$$\nabla \cdot (\rho \mathbf{V}) = 0$$

……………………………………………………………………………….(7.13)