NEHRU COLLEGE OF ENGINEERING AND RESEARCH CENTRE
(NAAC Accredited)
(Approved by AICTE, Affiliated to APJ Abdul Kalam Technological University, Kerala)

DEPARTMENT OF MECHATRONICS ENGINEERING

COURSE MATERIALS

ME 200 FLUID MECHANICS AND MACHINERY

VISION OF THE INSTITUTION
To mould true citizens who are millennium leaders and catalysts of change through excellence in education.
MISSION OF THE INSTITUTION
NCERC is committed to transform itself into a center of excellence in Learning and Research in Engineering and Frontier Technology and to impart quality education to mould technically competent citizens with moral integrity, social commitment and ethical values. We intend to facilitate our students to assimilate the latest technological know-how and to imbibe discipline, culture and spiritually, and to mould them in to technological giants, dedicated research scientists and intellectual leaders of the country who can spread the beams of light and happiness among the poor and the underprivileged.

ABOUT DEPARTMENT

- Established in: 2013
- Course offered: B.Tech Mechatronics Engineering
- Approved by AICTE New Delhi and Accredited by NAAC
- Affiliated to the University of Dr. A P J Abdul Kalam Technological University.

DEPARTMENT VISION

To develop professionally ethical and socially responsible Mechatronics engineers to serve the humanity through quality professional education.

DEPARTMENT MISSION

1) The department is committed to impart the right blend of knowledge and quality education to create professionally ethical and socially responsible graduates.

2) The department is committed to impart the awareness to meet the current challenges in technology.

3) Establish state-of-the-art laboratories to promote practical knowledge of mechatronics to meet the needs of the society

PROGRAMME EDUCATIONAL OBJECTIVES

I. Graduates shall have the ability to work in multidisciplinary environment with good professional and commitment.

II. Graduates shall have the ability to solve the complex engineering problems by applying electrical, mechanical, electronics and computer knowledge and engage in lifelong learning in their profession.

III. Graduates shall have the ability to lead and contribute in a team with entrepreneur skills, professional, social and ethical responsibilities.

IV. Graduates shall have ability to acquire scientific and engineering fundamentals necessary for higher studies and research.

PROGRAM OUTCOME (PO'S)

Engineering Graduates will be able to:

PO 1. Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

PO 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.

PO 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.

PO 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.

PO 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.

PO 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.

PO 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.

PO 8. Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

PO 9. Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO 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.

PO 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 multidisciplinary environments.

PO 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.

PROGRAM SPECIFIC OUTCOME(PSO'S)

PSO 1: Design and develop Mechatronics systems to solve the complex engineering problem by integrating electronics, mechanical and control systems.

PSO 2: Apply the engineering knowledge to conduct investigations of complex engineering problem related to instrumentation, control, automation, robotics and provide solutions.

COURSE OUTCOME

After the completion of the course the student will be able to

CO 1	**Understand** the fundamental concepts related to mechanics of fluids
CO 2	**Develop** the knowledge on pressure & its measurements
CO 3	**Analyze** about basic fluid equations
CO 4	Acquire **knowledge** on flow measuring instruments
CO 5	**Interpret** principles of fluid machines and devices.
CO 6	**Analyze** existing fluid systems and to apply acquired knowledge on real life problems.

CO VS PO'S AND PSO'S MAPPING

CO	PO1	PO 2	PO3	PO 4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PS0 1	PSO 2
CO 1	3	2	3	2	2	-	-	-	-	-	-	3	1	1
CO 2	3	3	3	2	2	-	-	-	-	-	-	3	1	2
CO 3	3	3	3	3	2	-	-	-	-	-	-	3	1	2
CO 4	3	3	3	3	2	-	-	-	-	-	-	3	2	2
CO 5	3	3	3	3	3	-	-	-	-	-	-	3	2	2
CO 6	3	3	3	3	3	-	-	-	-	-	-	3	2	2

Note: H-Highly correlated=3, M-Medium correlated=2, L-Less correlated=1

SYLLABUS

Course Number	Course Name	L-T-P-Credits	Year of Introduction
ME200	**Fluid mechanics and Machinery**	**3-1-0-4**	**2016**

Prerequisite : Nil

Course Objectives:

- To introduce students, the fundamental concepts related to the mechanics of fluids.
- To understand the basic principles of fluid machines and devices.
- To apply acquired knowledge on real life problems.
- To analyze existing fluid systems and design new fluid systems.

Syllabus

Fundamental Concepts, fluid statics and dynamics, fluid kinematics, boundary layer theory, hydraulic turbines, positive displacement pumps, rotary motion of liquids**,** centrifugal pump, pumping devices.

Expected Outcome

Up on completion of course the students might be in a position to:

i. Analyze flow problems associated with statics, kinematics and dynamics of fluids.
ii. Design and analyze fluid devices such as water turbines and pumps.
iii. Understand and rectify problems faced in practical cases of engineering applications.

Text Book:

1. Modi P. N. and S. M. Seth, *Hydraulics & Fluid Mechanics*, S.B.H Publishers, New Delhi, 2002.
2. Kumar D. S., *Fluid Mechanics and Fluid Power Engineering*, S. K. Kataria & Sons, New Delhi, 1998.

References:

1. J. F. Douglas, "Fluid Mechanics", Pearson education.
2. Cengel Y. A. and J. M. Cimbala, Fluid Mechanics, Tata McGraw Hill, 2013
3. Robert W. Fox and Mc Donald, "Introduction to fluid dynamics", John Wiley and sons
4. K. Subrahmanya, "Theory and applications of fluid mechanics", (TMH)
5. Shames. I. H, "Mechanics of fluids".
6. Jagadish Lal, "Fluid mechanics and Hydraulic machines".
7. R K Bansal, "Hydraulic Machines"

Course Plan

Module	Contents	Hours	Sem. exam marks

I	**Fundamental concepts:** Properties of fluid - density, specific weight, viscosity, surface tension, capillarity, vapour pressure, bulk modulus, compressibility, velocity, rate of shear strain, Newton's law of viscosity, Newtonian and non-Newtonian fluids, real and ideal fluids, incompressible and compressible fluids.	6	15%
II	Fluid statics: Atmospheric pressure, gauge pressure and absolute pressure. Pascal's Law, measurement of pressure - piezo meter, manometers, pressure gauges, energies in flowing fluid, head - pressure, dynamic, static and total head, forces on planar and curved surfaces immersed in fluids, centre of pressure, buoyancy, equilibrium of floating bodies, metacentre and metacentric height.	10	15%
First Internal Exam			
III	**Fluid kinematics and dynamics:** Classification of flow -1D, 2D and 3D flow, steady, unsteady, uniform, non-uniform, rotational, irrotational, laminar and turbulent flow, path line, streak line and stream line. Continuity equation, Euler's equation, Bernoulli's equation. Reynolds experiment, Reynold's number. Hagen- Poiseuille equation, head loss due to friction, friction, Darcy- Weisbach equation, Chezy's formula, compounding pipes, branching of pipes, siphon effect, water hammer transmission of power through pipes (simple problems)	8	15%
IV	**Boundary layer theory:** Basic concepts, laminar andturbulent boundary layer, displacement, momentum, energy thickness, drag and lift, separation of boundary layer. Flow rate measurements- venturi and orifice meters, notches and weirs (description only for notches, weirs and meters), practical applications, velocity measurements- Pitot tube and Pitot –static tube.	10	15%
Second Internal Exam			
V	**Hydraulic turbines :** Impact of jets on vanes - flat, curved, stationary and moving vanes - radial flow over vanes. Impulse and Reaction Turbines – Pelton Wheel constructional features - speed ratio, jet ratio & work done , losses and efficiencies, inward and outward flow reaction turbines- Francis turbine constructional features, work done and efficiencies – axial flow turbine (Kaplan) constructional features, work done and efficiencies, draft tubes, surge tanks, cavitation in turbines.	10	20%

VI	**Positive displacement pumps:** reciprocating pump, indicator diagram, air vessels and their purposes, slip, negative slip and work required and efficiency, effect of acceleration and friction on indicator diagram (no derivations), multi cylinder pumps. **Rotary motion of liquids:** – free, forced and spiral vortex flows, (no derivations), centrifugal pump, working principle, impeller, casings, manometric head, work, efficiency and losses, priming, specific speed, multistage pumps, selection of pumps, pump characteristics.	10	20%
End Semester Exam			

QUESTION BANK

MODULE I				
Q:NO:	**QUESTIONS**	**CO**	**KL**	**PAGE NO:**
1	Define the following properties 1) density 2) weight density 3)specific volume and 4) specific gravity of a fluid	CO1	K2	14
2	Differentiate between 1) liquid and gases 2) real fluids and ideal fluids.	CO1	K2	14
3	What is the difference between dynamic viscosity and kinematic viscosity?	CO1	K2	15
4	Explain the term dynamic viscosity and kinematic viscosity.	CO1	K2	15
5	State the Newtons low of viscosity and gives examples of its application.	CO1	K2	15

6	Define Newtonian and Non-Newtonian fluids.	CO1	K2	16
7	What you understand by the terms isothermal process and adiabatic process?	CO1	K2	16
8	Convert 1kg/s-m dynamic viscosity in poise	CO1	K4	19
9	One litre crude oil weights 9.6 N. Calculate its specific weight, density and specific gravity	CO1	K4	19
10	How does viscosity of a fluid vary with temperature?	CO1	K3	18
11	A plate 0.025 mm distance from a fixed plate, moves at 50 cm/s and require a force of 1.471 N/m^2 to maintain this speed. Determine the fluid viscosity between the plate in the poise.	CO1	K4	20
	MODULE II			
1	Define pressure. Obtain an expression for the pressure intensity at a point in a fluid.	CO2	K2	26
2	State and prove Pascals law.	CO2	K2	30
3	What do you understand by hydrostatic law?	CO2	K2	31
4	Differentiate between absolute and gauge pressure.	CO2	K3	28
5	Differentiate between simple manometer and differential manometer.	CO2	K3	33
6	What you mean by vacuum pressure?	CO2	K2	28
7	What is a manometer? How are they classified?	CO2	K2	33
8	What do you mean by single column manometer	CO2	K2	34
9	What is difference between U- tube differential manometer and in verted U- tube differential manometer?	CO2	K2	35

10	A hydraulic press has a ram of 30 cm diameter and a plunger of 5 cm diameter. Find the weight lifted by the hydraulic press when the force applied to the plunger is 400 N	CO2	K4	35
11	Determine the gauge and absolute pressure at the point which is 2.0 m below the free surface of water. Take atmospheric pressure as 10.1045 N/cm^2	CO2	K4	35
	MODULE III			
1	What are the methods of describing fluid flow?	CO3	K2	44
2	Distinguish between 1) steady flow and unsteady flow 2) uniform and non-uniform flow.	CO3	K3	46
3	Explain the types of fluid flow.	CO3	K2	45
4	Define the equations of continuity. Obtain an expression for continuity equation for a three dimensional flow	CO3	K3	51
5	Define the terms 1) velocity potential function 2) stream function	CO3	K2	47
6	Define the terms 1) vortex flow 2) forced vortex flow 3) free vortex flow	CO3	K2	46
7	The diameter of a pipe at the section 1 and 2 are 15 cm and 20 cm respectively. Find the discharge through the pipe if the velocity of water at section 1 is 4 m/s. determine also velocity at section 2.	CO3	K4	54
8	What is Euler's equation of motion? How will you obtain Bernoulli's equation from it?	CO3	K2	60
9	State Bernoulli's theorem for steady flow of incompressible fluid. Derive an expression for Bernoulli's theorem from first principle.	CO3	K4	61

10	Derive Bernoulli’s equation for the flow of an incompressible frictionless fluid from consideration of momentum.	CO3	K2	61
MODULE IV				
1	What is a venturimeter? Derive an expression for the discharge through a venturimeter.	CO4	K2	82
2	Explain the principle of venturimeter with neat sketch.	CO4	K1	82
3	Discuss the relative merits and demerits of venturimeter with respect to orifice meter.	CO4	K2	83
4	What is a pitot tube? How will you determine the velocity at any point with the help of pitot tube?	CO4	K2	79
5	What is the difference between the pitot tube and pitot static tube?	CO4	K2	79
6	Define the term notch, weir ?	CO4	K2	84
7	How are the notch and weir classified?	CO4	K3	84
8	What are the advantages of triangular notch over rectangular notch?	CO4	K2	84
9	What do you understand by the terms boundary layer and boundary layer theory?	CO4	K2	69
10	Obtain an expression for the boundary shear stress in terms of momentum thickness.	CO4	K2	72
MODULE V				
1	Define the hydraulic turbine.	CO5	K2	88
2	Differentiate between turbines and pumps	CO5	K3	88

3	How will you classify the turbines?	CO5	K3	89
4	Differentiate between impulse and reaction turbines.	CO5	K3	89
5	Differentiate between radial and axial flow turbines.	CO5	K3	97
6	Differentiate between inward and outward flow radial flow turbine.	CO5	K3	93
7	Explain the working of Kaplan turbine with neat fig.	CO5	K2	98
8	Explain the working of Francis turbine with neat fig.	CO5	K2	94
9	What is the basis of selection of a turbine at a particular place?	CO5	K2	89
10	Draw the neat sketches of pelton wheel and Francis turbine.	CO5	K2	91
11	What is a draft tube? what are its functions?	CO5	K2	96
MODULE VI				
1	What is a reciprocating pump? Describe the principle and working of reciprocating pump with neat fig.	CO6	K2	101
2	Differentiate between a single acting and double acting reciprocating pump.	CO6	K3	101
3	Define slip, pressure slip and negative slip of reciprocating pump.	CO6	K2	105
4	How will you classify the reciprocating pump?	CO6	K2	102
5	Define indicator diagram. Draw an indicator diagram.	CO6	K2	106
6	Define a centrifugal pump. Explain the working of a single stage centrifugal pump with neat fig.	CO6	K2	108

7	Differentiate between volute casing and vortex casing.	CO6	K3	109
8	Define the terms suction head and delivery head.	CO6	K2	109
9	Classify efficiencies of a centrifugal pump.	CO6	K3	110
10	What is priming? Why is it necessary?	CO6	K2	109
11	Define cavitation? What are the effects of cavitation?	CO6	K2	112

APPENDIX 1

CONTENT BEYOND THE SYLLABUS

UNIT – I

PROPERTIES OF FLUIDS

Fluid mechanics:

It is the branch of science which deals which behaviour of fluids at rest as well as in motion.

Fluid statics: fluid is a substance which is capable of flowing fluid

Study of fluids at rest.

Fluid kinematics:

Study of fluids in motion where pressure forces are not considered.

Fluid dynamics:

Study of fluids in motion where pressure forces are considered.

Density (or) Mass density:

It is defined as ratio of mass of a fluid to its volume.

Unit kg/m^3.

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

Specific weight (or) weight density:

It is ratio between weight of a fluid to its volume.

$$w = \frac{\text{weight of fluid}}{\text{Volume of fluid}} = \frac{\text{mass of fluid} \times g}{V} = \rho g$$

Specific volume:

$$\text{specific volume} = \frac{\text{volume of fluid}}{\text{mass of fluid}} = V/m$$

$$= \frac{1}{\rho}$$

Specific gravity (or) relative density:

$$S = \frac{\text{Density of fluid}}{\text{Density of water}} = \frac{\rho_{fluid}}{\rho_{water}}$$

Density of water is 1000 kg/m³.

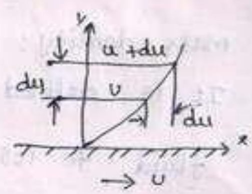
Specific gravity of mercury is 13.6

$S_m = \frac{\rho_m}{1000}$, $\rho_m \rightarrow$ density of mercury

Viscosity:

viscosity is the property of a fluid which offers resistance to movement of one layer of fluid over another adjacent layer of fluid.

Shear stress is directly proportional to viscosity with respect to y.



Unit = NS/m² (dynamic viscosity)

1 pascal = N/m²

= pascal × sec.

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

$$\tau = \mu \cdot \frac{du}{dy}, \quad \frac{\tau}{du/dy} = \mu \qquad \sigma = P/A$$

$$\frac{F/A}{\frac{1}{t} \times \frac{1}{t}} = \frac{N/m^2}{m/s \times \frac{1}{m}} = NS/m^2$$

$1 \text{ Poise} = \frac{1}{10} \, NS/m^2$

Temperature increases → viscosity decrease

Temperature decrease → Viscosity increase

Kinematic viscosity:

$$\text{Kinematic viscosity} = \frac{\text{Dynamic viscosity}}{\text{Density}} = m^2/s$$

$$= \frac{\mu}{\rho}$$

Objectives:

The application of conversation to flow through pipes and hydraulic machines are studied.

To understand importance of dimensional analysis.

To understand importance of various types of flow in pumps & turbine.

FLUID PROPERTIES & FLOW CHARACTERISTIC

Units and dimensions - Properties of fluids mass density, specific weight, specific volume, specific gravity, viscosity, compressibility, vapour pressure, surface tension & capillarity. Flow characteristic. concept of control volume - application of continuity equation, energy equation & momentum equation.

compressibility & bulk modulus:

$$\text{Bulk modulus} = \frac{\text{Increase of Pressure}}{\text{Volumetric strain}}$$

$$\text{compressibility} = 1/k$$

cohesion:

'cohesion' means intermolecular attraction between molecular of same liquid.

Adhesion:

"Adhesion" means attraction between the molecules of a liquid & the molecular liquid. The property enables a liquid to stick to another body.

Problem:

$m = 5\ kg$

$v = 3\ m^3$

$\rho = 1.67\ kg/m^3$

Specific $= 0.6\ m^3/kg$

Surface tension:

It is defined as tensile force acting on surface of a liquid in contact with a gas (or) on the surface between two immisible liquids such as that are contact surface behaves like a membrance under tension.

Surface tension on liquid droplet:

σ = Surface tension of liquid

p = Pressure intensity inside droplets

d = diameter of droplet

Pressure force = Surface tension

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

$d \rightarrow$ diameter decrease pressure intensity increase

Hollow surface:

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

Liquid jet:

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

capillarity:

It is defined as phenomenon of rise (or) fall of a liquid surface in a small tube relative to adjacent general level of when the tube is held vertically. The rise of liquid surface is capillary rise. which fall is defined as capillary depression.

$$S = \frac{\rho_{mercury}}{\rho_{water}}$$

$$13.6 = \frac{\rho_{mer}}{1000}$$

$$\rho_{mercury} = 13600 \ kg/cm^3$$

Expression for capillary rise:

σ = Surface tension of liquid

θ = angle of contact between liquid & glass tube

The weight of liquid of height h in tube,

$$= \frac{\pi d^2}{4} \times \rho \times g \times h \quad \text{--- (1)}$$

Surface Tension force = σ x circumference x angle

$$= \sigma \times \pi d \times \cos\theta \quad \text{--- (2)}$$

eqn (1) = (2)

$$\frac{\pi d^2}{4} \times \rho \times g \times h = \sigma \times \pi d \times \cos\theta$$

$$h = \frac{\sigma \times \pi d \times \cos\theta \times 4}{\pi d^2 \times \rho \times g \times h}$$

$$= \frac{4\sigma \times \pi \times d \cdot \cos\theta}{\pi d^2 \times \rho \times g}$$

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

Expression for capillary fall:

$$h = \frac{4\sigma\cos\theta}{\rho g d} \quad (\theta = 128^\circ \text{ for mercury})$$

The weight of liquid of height tube

$$= \frac{\pi d^2}{4} \times \rho \times g \times h$$

surface force = c x angle of contact

$$= \sigma \times \pi d \times \cos\theta$$

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

Problems :

1) Calculate capillary rise in a glass tube of 2.5 mm dia when immersed vertically in a) water & b) mercury. Take Surface tension $\sigma = 0.0725\ N/m^2$ for water & $\sigma = 0.52\ N/m^2$ for mercury in contact with air. The specific gravity for mercury is 13.6 and angle of contact 130°.

Data :

$d = 2.5\ mm$

$\Rightarrow 2.5 \times 10^{-3}\ mm$

$\sigma = 0.0725\ N/m^2$

$\sigma_{water} = 0.0725\ N/m^2$

$\sigma_{mercury} = 0.52\ N/m^2$

Specific gravity for mercury $= 13.6$

Angle of contact $\theta_m = 130^\circ$

Find :

i) $h_1 = ?$ for water

ii) $h = ?$ for mercury

Soln :

For water,

$$h = \frac{4\sigma \cdot \cos\theta}{\rho g d} = \frac{4 \times 0.0725 \times 1}{1000 \times 9.81 \times 2.5 \times 10^{-3}} = \boxed{0.01182\ m}$$

For mercury,

$$h = \frac{4\sigma \cdot \cos\theta}{\rho g d} = \frac{4 \times 0.52 \times \cos 130}{13.600 \times 13.6 \times 2.5 \times 10^{-3}} = \boxed{-0.0028\ m}$$

vapour pressure & cavitation:

A change from liquid state to gaseous state is known as "Vaporisation". when vaporisation takes place 20°c & Pressure molecular and top of the vessel. This pressure is called "vapour Pressure".

Problem:

1) A flat plate area of $1.5\times10^{6}\ mm^2$ is pulled with a speed of 0.4 m/s relative to another Plate located at a distance of 0.15 mm from it. Find & force and power required to maintain speed If fluid seperating them is having viscosity as 1 poise.

Data:

$A = 1.5\times10^{6}\ mm^2$

$du = 0.4\ m/s$

$dy = 0.15\ mm \Rightarrow 0.15\times10^{-3}\ m$

$u = 1\ \text{Poise} \Rightarrow \frac{1}{10}\ N.s/m^2$

Find:

i) Force

ii) Power

Sol:

$$\tau = \frac{\text{shear force}}{\text{area}}$$

$\tau \times \text{area} = \text{shear force}$

$$\tau = u.du/dy = \frac{1}{10}\times\frac{0.4}{0.15\times10^{-3}}$$

$$\tau = 266.66\ N/m^2$$

Shear force $= \tau \times$ area

$= 266.66 \times 1.25$

Shear force $= 400\,N$

ii) Power.

Power $= F \times u$

$= 400 \times 0.4$

Power $= 160\,W$

8) Calculate the specific weight, density and specific gravity of 1 litre of a liquid which weight 7N.

Data:

Volume $= 1$ lit $\Rightarrow \frac{1}{1000}\,m^3$

Weight $= 7\,N$

1 lit $= \frac{1}{1000}\,m^3$
(or)
1 lit $= 1000\,cm^3$

Find:

i) Specific weight

ii) Density

iii) Specific gravity

Sol:

i)

$$\text{Specific weight} = \frac{\text{weight}}{\text{volume}} = \frac{7}{1/1000} = 7000\ N/m^3$$

ii)

$$\text{Density}\ \rho = \frac{w}{g} = \frac{7000}{9.81} = 713.5\ kg/m^3$$

iii)

$$\text{Specific Gravity} = \frac{\text{Density of liquid}}{\text{Density of water}} = \frac{713.5}{1000} = 0.7135$$

$\because$ Density of water $= 1000\ kg/m^3$

3) Calculate the density, spefic weight & weight of one litre of petrol of specific gravity 0.7

Data :

Volume $= 1\ lit \Rightarrow \frac{1}{1000} = 0.01\ m^3$

Specific gravity $= 0.7$

Find :

i) Density

ii) Specific weight

iii) weight

Sol :

i) Specific Gravity $= \frac{\text{Density of liquid}}{\text{Density of water}}$

Density of liquid $= S_g \times \rho_w$

$= 0.7 \times 1000$

$\rho_{liq} = 700\ kg/m^3$

ii) Specific weight :

$w = \rho \times g$

$= 700 \times 9.81$

$w = 6867\ N/m^3$

iii) weight,

$w = \frac{\text{weight}}{\text{volume}}$

weight $= w \times$ volume

$= 6867 \times 0.01$

weight $= 6.867\ N$

4) Calculate specific weight, density, specific volume & specific gravity.

$$mass = 5\ kg, \quad V = 3\ m^3$$

$$Density = \frac{m}{V} = 5/3 = 1.67\ kg/m^3$$

i) Specific weight $= \rho \times g$

$$= 1.67 \times 9.81$$

$$Specific\ weight = 16.3827\ N/m^3$$

ii) Specific volume $= \frac{V}{m} = 3/5 = 0.6\ m^3/kg$

iii) Specific Gravity,

$$specific\ gravity = \frac{\rho_{liq}}{\rho_{water}}$$

$$= \frac{1.67}{1000}$$

$$Specific\ gravity = 1.67 \times 10^{-3}\ kg/m^3$$

Then,

$$S_g = 0.5, \quad \rho_{liq} = ?$$

$$Specific\ gravity = \frac{\rho_{liq}}{\rho_{water}}$$

$$\rho_{liq} = Specific \times \rho_{water}$$

$$= 0.5 \times 1000$$

$$\rho_{liq} = 500\ kg/m^3$$

MODULE 2

FLUID STATICS

2.0 INTRODUCTION

Fluids are generally found in contact with surfaces. Water in the sea and in reservoirs are in contact with the ground and supporting walls. Atmospheric air is in contact with the ground. Fluids filling vessels are in contact with the walls of the vessels. **Fluids in contact with surfaces exert a force on the surfaces**. The force is mainly due to the specific weight of the fluid in the case of liquids. In the case of gases molecular activity is the main cause of force exerted on the surfaces of the containers. Gas column will also exert a force on the base, but this is usually small in magnitude. **When the whole mass of a fluid held in a container is accelerated or decelerated without relative motion between layers inertia forces also exert a force on the container walls**. This alters the force distribution at stationary or atatic conditions. Surfaces may also be immersed in fluids. A ship floating in sea is an example. In this case the force exerted by the fluid is called buoyant force. This is dealt with in a subsequent chapter. The force exerted by fluids vary with location. The variation of force under static or dynamic condition is discussed in this chapter.

This chapter also deals with pressure exerted by fluids due to the weight and due to the acceleration/deceleration of the whole mass of the fluid without relative motion within the fluid.

Liquids held in containers may or may not fill the container completely. When liquids partially fill a container a free surface will be formed. Gases and vapours always expand and fill the container completely.

2.1 PRESSURE

Pressure is a measure of force distribution over any surface associated with the force. **Pressure is a surface phenomenon and it can be physically visualised or calculated only if the surface over which it acts is specified**. Pressure may be defined as the force acting along the normal direction on unit area of the surface. However a more precise definition of pressure, *P* is as below:

$$P = \lim_{A \to a} (\Delta F/\Delta A) = dF/dA \qquad (2.1.1)$$

F is the resultant force acting normal to the surface area **A**. **'a'** is the limiting area which will give results independent of the area. This explicitly means that **pressure is the ratio of the elemental force to the elemental area normal to it.**

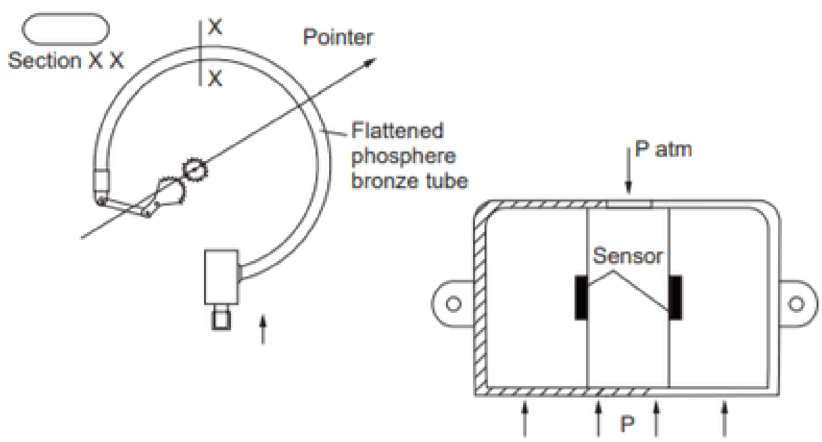
The force dF in the normal direction on the elemental area dA due to the pressure P is

$$dF = P\,dA \qquad (2.1.2)$$

The unit of pressure in the SI system is N/m² also called Pascal (Pa). As the magnitude is small kN/m² (kPa) and MN/m² (Mpa) are more popularly used. **The atmospheric pressure is approximately 10^5 N/m² and is designated as "bar".** This is also a popular unit of pressure. In the metric system the popular unit of pressure is kgf/cm². This is approximately equal to the atmospheric pressure or 1 bar.

2.2 PRESSURE MEASUREMENT

Pressure is generally measured using a sensing element which is exposed on one side to the pressure to be measured and on the other side to the surrounding atmospheric pressure or other reference pressure. The details of some of the pressure measuring instruments are as shown in Fig. 2.2.1.



In the Borden gauge a tube of elliptical section bent into circular shape is exposed on the inside to the pressure to be measured and on the outside to atmospheric pressure. The tube will tend to straighten under pressure. The end of the tube will move due to this action and will actuate through linkages the indicating pointer in proportion to the pressure. Vacuum also can be measured by such a gauge. Under vacuum the tube will tend to bend further inwards and as in the case of pressure, will actuate the pointer to indicate the vacuum pressure. The scale is obtained by calibration with known pressure source.

The pressure measured by the gauge is called gauge pressure. The sum of the gauge pressure and the outside pressure gives the absolute pressure which actually is the pressure measured.

The outside pressure is measured using a mercury barometer (Fortins) or a bellows type meter called Aneroid barometer shown in Fig. 2.2.2. The mercury barometer and bellow type meter have zero as the reference pressure. The other side of the measuring surface in these cases is exposed to vacuum. Hence these meters provide the absolute pressure value.

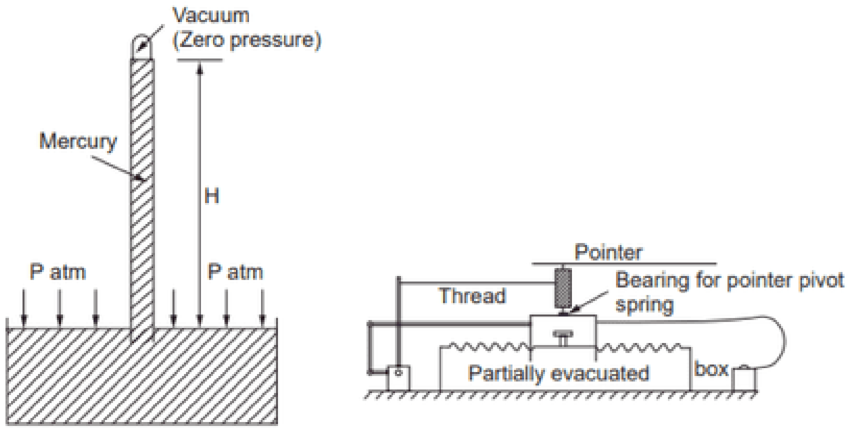


Figure 2.2.2 Barometer

When the pressure measured is above surroundings, then

Absolute pressure = gauge pressure + surrounding pressure

The surrounding pressure is usually the atmospheric pressure.

If the pressure measured is lower than that of surrounding pressure then

Absolute pressure = surrounding pressure – gauge reading

This will be less than the surrounding pressure. This is called Vacuum.

Electrical pressure transducers use the deformation of a flexible diaphragm exposed on one side to the pressure to be measured and to the surrounding pressure or reference pressure on the other side. The deformation provides a signal either as a change in electrical resistance or by a change in the capacitance value. An amplifier is used to amplify the value of the signal.

In this text the mension pressure means absolute pressure. Gauge pressure will be specifically indicated.

Example 2.1. *A gauge indicates 12 kPa as the fluid pressure while, the outside pressure is 150 kPa. Determine the absolute pressure of the fluid. Convert this pressure into kgf/cm²*

Absolute pressure = Gauge pressure + Outside pressure

= 150 + 12 = **162 kPa** or 1.62 bar.

$1.62 \text{ bar} = 1.62 \times 10^5 \text{ N/m}^2$

As $1 \text{ kgf/cm}^2 = 9.81 \text{ N/cm}^2 = 9.81 \times 10^4 \text{ N/m}^2 = 98100 \text{ N/m}^2$

$1.62 \times 10^5 \text{ N/m}^2 = 1.62 \times 10^5/98100 = \mathbf{1.651 \text{ kgf/cm}^2}$

Example 2.2. *A vacuum gauge fixed on a steam condenser indicates 80 kPa vacuum. The barometer indicates 1.013 bar. Determine the absolute pressure inside the condenser. Convert this pressure into head of mercury.*

Barometer reading = 1.013 bar = 101.3 kPa.

Absolute pressure = atmospheric pressure – vacuum gauge reading

Absolute pressure in the condenser = 101.3 – 80 = **21.3 kPa**

101.3 kPa = 760 mm of Hg. (standard atmosphere)

∴ 21.3 kPa = (21.3/101.3) × 760 = **159.8 mm of Hg**

2.3 PASCAL'S LAW

In fluids under static conditions pressure is found to be independent of the orientation of the area. This concept is explained by **Pascal's law** which **states that the pressure at a point in a fluid at rest is equal in magnitude in all directions**. Tangential stress cannot exist if a fluid is to be at rest. This is possible only if the pressure at a point in a fluid at rest is the same in all directions so that the resultant force at that point will be zero.

The proof for the statement is given below.

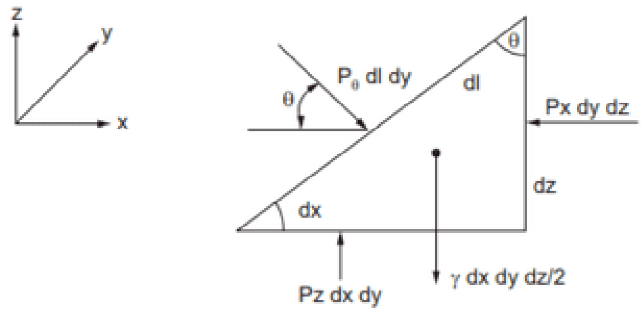


Figure 2.3.1 Pascals law demonstration

Consider a wedge shaped element in a volume of fluid as shown in Fig. 2.3.1. Let the thickness perpendicular to the paper be **dy**. Let the pressure on the surface inclined at an angle θ **to vertical be** P_θ and its length be **dl**. Let the pressure in the x, y and z directions be P_x, P_y, P_z.

First considering the x direction. For the element to be in equilibrium,

$$P_\theta \times dl \times dy \times \cos\theta = P_x \times dy \times dz$$

But, $dl \times \cos\theta = dz$ So, $P_\theta = P_x$

When considering the vertical components, the force due to specific weight should be considered.

$$P_z \times dx \times dy = P_\theta \times dl \times dy \times \sin\theta + 0.5 \times \gamma \times dx \times dy \times dz$$

The second term on RHS of the above equation is negligible, its magnitude is one order less compared to the other terms.

Also, $dl \times \sin\theta = dx$, So, $P_z = P_\theta$

Hence, $\mathbf{P_x = P_z = P_\theta}$

Note that the angle has been chosen arbitrarily and so this relationship should hold for all angles. By using an element in the other direction, it can be shown that

$$\mathbf{P_y = P_\theta \quad \text{and} \quad \text{so } P_x = P_y = P_z}$$

Hence, the pressure at any point in a fluid at rest is the same in all directions. The pressure at a point has only one value regardless of the orientation of the area on which it is measured. This can be extended to conditions where fluid as a whole (like a rotating container) is accelerated like in forced vortex or a tank of water getting accelerated without relative motion between layers of fluid. Surfaces generally experience compressive forces due to the action of fluid pressure.

2.4 PRESSURE VARIATION IN STATIC FLUID (HYDROSTATIC LAW)

It is necessary to determine the pressure at various locations in a stationary fluid to solve engineering problems involving these situations. **Pressure forces are called surface forces. Gravitational force is called body force as it acts on the whole body of the fluid.**

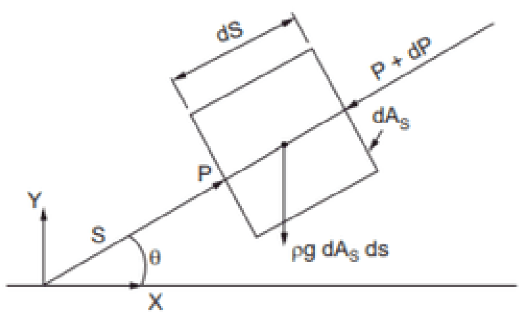


Figure 2.4.1 Free body diagram to obtain hydrostatic law

Consider an element in the shape of a small cylinder of constant area dA_s along the s direction inclined at angle θ to the horizontal, as shown in Fig. 2.4.1. The surface forces are P at section s and $P + dp$ at section $s + ds$. The surface forces on the curved area are balanced. The body force due to gravity acts vertically and its value is $\gamma \times ds \times dA_s$. A force balance in the s direction (for the element to be in equilibrium) gives

$$P \times dA_s - (P + dp) \times dA_s - \gamma \times dA_s \times ds \times \sin\theta = 0$$

Simplifying,

$$dp/ds = -\gamma \times \sin\theta \text{ or, } dp = -\gamma \times ds \times \sin\theta \qquad (2.4.1)$$

This is the fundamental equation in fluid statics. The variation of specific weight γ with location or pressure can also be taken into account, if these relations are specified as (see also section 2.4.2).

$$\gamma = \gamma(P, s)$$

For x axis, $\theta = 0$ and $\sin\theta = 0$.

$$\therefore \quad dP/dx = 0 \quad (2.4.2)$$

In a static fluid with no acceleration, the pressure gradient is zero along any horizontal line i.e., planes normal to the gravity direction.

In y direction, $\theta = 90$ and $\sin\theta = 1$,

$$dP/dy = -\gamma = -\rho g/g_o \quad (2.4.4)$$

Rearranging and integrating between limits y_1 and y

$$\int_{p_1}^{p} dp = -\gamma \int_{y_1}^{y} dy \quad (2.4.5)$$

If γ is constant as in the case of liquids, these being incompressible,

$$P - P_1 = -\gamma \times (y - y_1) = -\rho\, g\, (y - y_1)/g_o \quad (2.4.6)$$

As P_1, y_1 and γ are specified for any given situation, P will be constant if y is constant. This leads to the statement,

The pressure will be the same at the same level in any connected static fluid whose density is constant or a function of pressure only.

A consequence is that the free surface of a liquid will seek a common level in any container, where the free surface is everywhere exposed to the same pressure.

In equation 2.4.6, if $y = y_1$ then $P = P_1$ and $dp = 0$. This result is used very extensively in solving problems on manometers.

2.4.1 Pressure Variation in Fluid with Constant Density

Consider the equation 2.4.6,

$$P - P_1 = -\gamma \times (y - y_1) = \gamma \times (y_1 - y) = \rho\, g\, (y_1 - y)/g_o \quad (2.4.7)$$

As y increases, the pressure decreases and vice versa (**y is generally measured in the upward direction**). In a static fluid, the pressure increases along the depth. If the fluid is incompressible, then **the pressure at any y location is the product of head and specific weight, where head is the y distance** of the point from the reference location.

Example 2.3. *An open cylindrical vertical container is filled with water to a height of 30 cm above the bottom and over that an oil of specific gravity 0.82 for another 40 cm. The oil does not mix with water. If the atmospheric pressure at that location is 1 bar,* ***determine the absolute and gauge pressures*** *at the oil water interface and at the bottom of the cylinder.*

This has to be calculated in two steps, first for oil and then for water.

Density of the oil	$= 1000 \times 0.82 = 820$ kg/m^3
Gauge pressure at interface	$= (\rho \times g \times h)^*_{oil}$
	$= 820 \times 9.81 \times 0.4 =$ **3217.68 N/m²**
Absolute pressure at interface	$= 3217.68 + 1 \times 10^5$ N/m^2
	$= 103217.68$ N/m^2 = **1.0322 bar**
Pressure due to water column	$= \rho \times g \times h = 1000 \times 9.81 \times 0.3 = 2943$ N/m^2

2.5 MANOMETERS

Manometer is a device to measure pressure or mostly difference in pressure using a column of liquid to balance the pressure. It is a basic instrument and is used extensively in flow measurement. It needs no calibration. Very low pressures can be measured using micromanometers. The basic principle of operation of manometers is that at the same level in contigues fluid at rest, the pressure is the same. The pressure due to a constant density liquid (ρ) column if height h is equal to $\rho gh/g_o$. g_o in SI system of units has a numerical value of unity. Hence it is often left out in the equations. For dimensional homogenity g_o should be used. The principle of operation is shown in Fig. 2.5.1 (*a*) and some types of manometers are shown in Fig. 2.5.1 (*b*). In Fig. 2.5.1 (*a*), the pressure inside the conduit is higher than atmospheric pressure. The column of liquid marked *AB* balances the pressure existing inside the conduit. The pressure at point *C* above the atmospheric pressure (acting on the open limb) is given by $h \times (\gamma_1 - \gamma_2)$ where γ_1 and γ_2 are the specific weights of fluids 1 and 2, and h is the height of the column of liquid (*AB*). The pressure at the centre point *D* can be calculated as

$$P_d = P_c - \gamma_2 \times h'$$

Generally the pressure at various points can be calculated using the basic hydrostatic equation $dP/dy = -\gamma$ and continuing the summation from the starting point at which pressure is known, to the end point, where the pressure is to be determined.

Another method of solving is to start from a point of known pressure as datum and adding $\gamma \times \Delta y$ when going downwards and subtracting of $\gamma \times \Delta y$ while going upwards. The pressure at the end point will be the result of this series of operations.

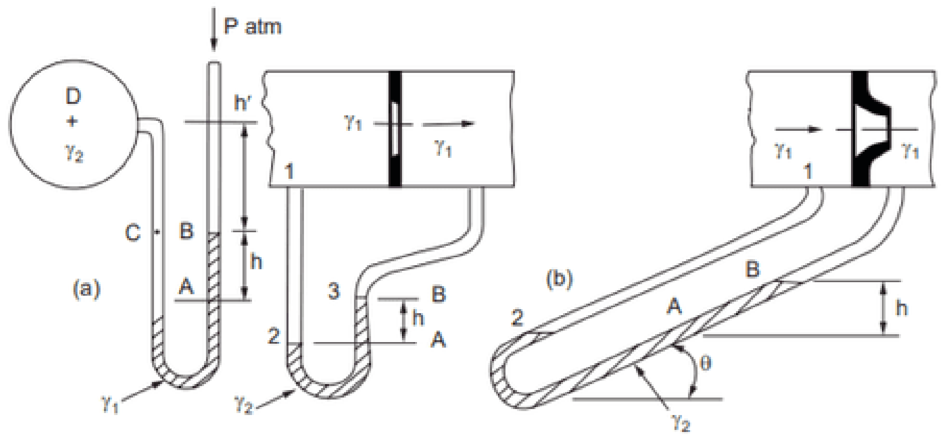


Figure 2.5.1 Types of manometers

$$\Delta P_{1-5} = \gamma_1\,\Delta y_1 + \gamma_2\,\Delta y_2 + \gamma_3\,\Delta y_3 + \gamma_4\,\Delta y_4$$

with proper sign for Δy values.

The advantages of using manometers are (*i*) their simplicity (*ii*) reliability and (*iii*) ease of operation and maintenance and freedom from frequent calibration needed with other types of gauges. As only gravity is involved, horizontal distances need not be considered in the calculation.

The sensitivity of simple manometers can be improved by using inclined tubes (at known angle) where the length of the column will be increased by (1/sin θ) where θ is the angle of inclination with the horizontal (Fig. 2.5.1 (*b*)).

Example 2.6. *A manometer is fitted as shown in Fig. Ex. 2.6. Determine the pressure at point A.*

With respect to datum at B, pressure at left hand side = pressure at right hand side

$P_C = P_B$ Consider the left limb

$P_C = P_a + 0.125 \times 900 \times 9.81 + 0.9 \times 13600 \times 9.81$

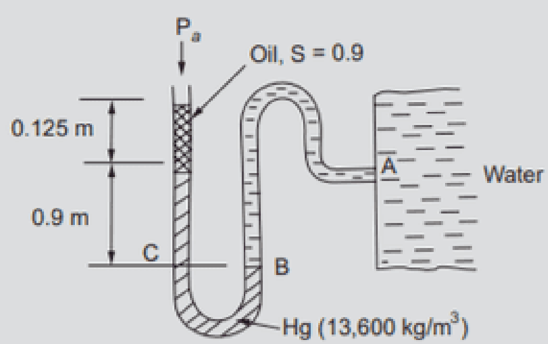
$= P_a + 121178\ \text{N/m}^2$

Consider the right limb $\quad P_A = P_B - 0.9 \times 1000 \times 9.81 = P_a + 121178 - 0.9 \times 1000 \times 9.81$

$= P_a + 112349\ \text{N/m}^2$ Expressed as gauge pressure

$\mathbf{P_A} = 112349\ \text{N/m}^2$

$=$ **112.35 kPa** gauge



Example 2.7. *An inverted U-tube manometer is fitted between two pipes as shown in Fig.Ex.2.7. Determine the pressure at E if P_A = 0.4 bar (gauge)*

$$P_B = P_A - [(0.9 \times 1000) \times 9.81 \times 1.2]$$
$$= 40000 - [(0.9 \times 1000) \times 9.81 \times 1.2] = 29{,}405.2 \text{ N/m}^2$$
$$P_C = P_B - [(0.9 \times 1000) \times 9.81 \times 0.8] = 22342 \text{ N/m}^2$$
$$P_C = P_D = 22342 \text{ N/m}^2$$
$$\mathbf{P_E} = P_D + [1000 \times 9.81 \times 0.8] = 30190 \text{ N/m}^2 = \mathbf{30.19 \text{ kPa (gauge)}}$$

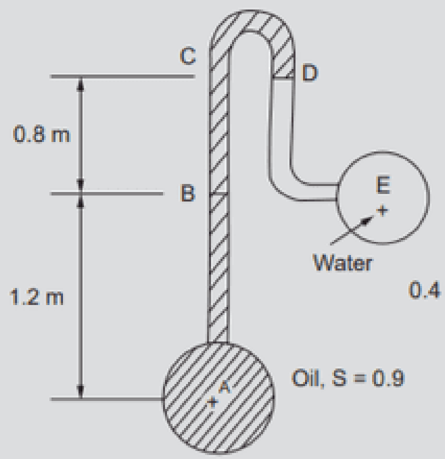


Figure Ex. 2.7

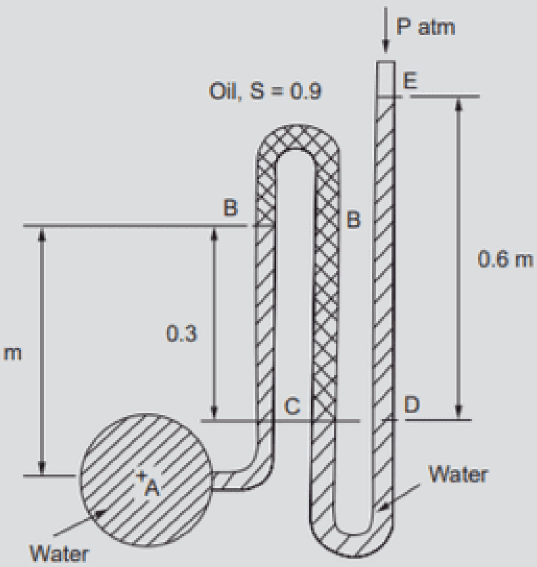


Figure Ex. 2.8

Example 2.8. *A multiple U-tube manometer is fitted to a pipe with centre at A as shown in Fig. Ex.2.8. Determine the pressure at A.*

Pressure at E = atmospheric pressure, P_{atm}

$$P_D = P_{atm} + (1000 \times 9.81 \times 0.6) = P_{atm} + 5886\ P_a \text{ As } P_C = P_D$$
$$P_B = P_C - [0.9 \times 1000 \times 9.81 \times 0.3]$$
$$= P_{atm} + 5886 - 2648.7 = P_{atm} + 3237.3 \text{ Pa}$$
$$\mathbf{P_A} = P_B + [1000 \times 9.81 \times 0.4] = P_{atm} + 7161.3$$
$$= P_{atm} + 7161.3 \text{ N/m}^2 \text{ or } \mathbf{7161.3 \text{ kPa (gauge)}}$$

3.1 CENTROID AND MOMENT OF INERTIA OF AREAS

In the process of obtaining the resultant force and centre of pressure, the determination of first and second moment of areas is found necessary and hence this discussion. The moment of the area with respect to the y axis can be obtained by summing up the moments of elementary areas all over the surface with respect to this axis as shown in Fig. 3.1.1.

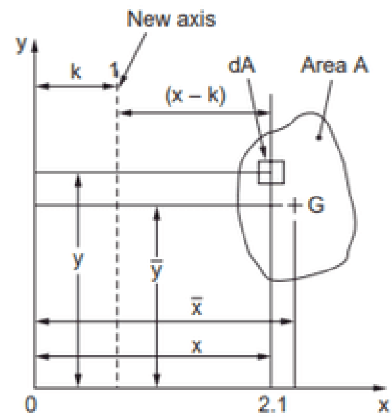


Figure. 3.1.1 First moment and second moment of an area

$$\text{Moment about } y \text{ axis} = \int_A x\, dA \quad (3.1.1)$$

$$\text{Moment about } x \text{ axis} = \int_A y\, dA \quad (3.1.2)$$

The integral has to be taken over the area. If moments are taken with respect to a parallel axis at a distance of k from the y axis equation 3.1.1. can be written as

$$\int_A (x-k)\, dA = \int_A x\, dA - k \int_A dA = \int_A x\, dA - k\, A \quad (3.1.3)$$

As k is a constant, it is possible to choose a value of $\bar{x} = k$, such that the moment about the axis is zero. The moment about the axis through the centre of gravity is always zero.

$$\int_A x\, dA - \bar{x}\, A = 0$$

Such an axis is called **centroidal** y axis. The value of $\bar{x}$ can be determined using

$$\bar{x} = (1/A) \int_A x\, dA \quad (3.1.4)$$

Similarly the centroidal x axis passing at $\bar{y}$ can be located using

$$\bar{y} = (1/A) \int_A y\, dA \quad (3.1.5)$$

The point of intersection of these centroidal axes is known as the **centroid** of the area.

It can be shown that the moment of the area about any line passing through the centroid to be zero.

With reference to the Fig. 3.1.1, the second moment of an area about the y axis. I_y is defined as

$$I_y = \int_A x^2\, dA \quad (3.1.6)$$

Considering an axis parallel to y axis through the centroid and taking the second moment of the area about the axis and calling it as I_G, where $\bar{x}$ is the distance from the axis and the centroid.

$$I_G = \int_A (x - \bar{x})^2 \, dA \tag{3.1.7}$$

$$I_G = \int_A x^2 \, dA - 2\bar{x} \int_A x \, dA + \int_A \bar{x}^2 \, dA$$

By definition $\int_A x^2 \, dA = I_y$, $\int_A x^2 \, dA = \bar{x} \, A$

As $\bar{x}^2$ is constant, $\int_A \bar{x}^2 \, dA = \bar{x}^2 \, A$. Therefore

$$I_G = I_y - 2\bar{x}^2 \, A + \bar{x}^2 \, A = I_y - \bar{x}^2 \, A \tag{3.1.8}$$

or

$$I_y = I_G + \bar{x}^2 A \tag{3.1.9}$$

Similarly

$$I_y = I_G + \bar{y}^2 \, A \tag{3.1.10}$$

The moment of inertia of an area about any axis is equal to the sum of the moment of inertia about a parallel axis through the centroid and the product of the area and the square of the distance between this axis and centroidal axis. These two equations are used in all the subsequent problems.

The second moment is used in the determination of the centre of pressure for plane areas immersed in fluids.

The product of inertia is defined as

$$I_{xy} = \int_A xy \, dA = I_{Gxy} + \bar{x} \, \bar{y} \, A \tag{3.1.11}$$

It can be shown that whenever any one of the axes is an axis of symmetry for the area, $I_{xy} = 0$.

4.1 BUOYANCY FORCE

Consider the immersed or floating body shown in Fig. 4.1.1. The total force on the body can be calculated by considering the body to consist of a large number of cylindrical or prismatic elements and calculating the sum of forces on the top and bottom area of each element.

(*i*) **Immersed body.** Consider a prismatic element :

Let the sectional area be dA, Force on the top $dF_1 = dA \, \gamma \, h_1$ and

Force on the base $dF_2 = dA \, \gamma \, h_2$ (cancelling P_{atm}, common for both terms)

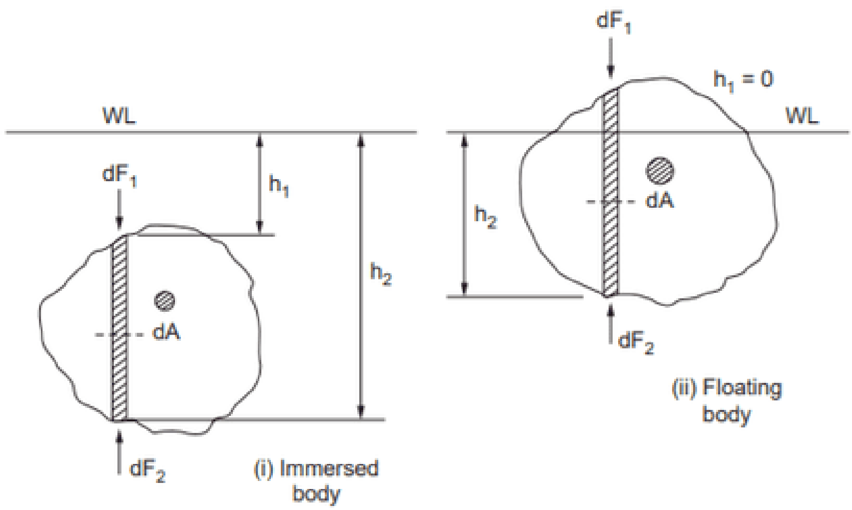


Figure 4.1.1 Proof for Archimedes principle

Net force on the element $(dF_2 - dF_1) = \gamma\, dA\, (h_2 - h_1) = \gamma\, dV$.

where dV is the volume of the element. This force acts upwards. as $h_2 > h_1$

Summing up over the volume, $F = \gamma V$ (or) the weight of the volume of liquid displaced.

(*ii*) **Floating body.** Considering an element of volume dV, Force on the top of the element $dF_1 = dA.\, P_a$ and Force on the base of the element $dF_2 = dA\,(\gamma\, h_2 + P_a)$

$$dF_2 - dF_1 = \gamma\, dA\, h_2 = \gamma\, dV$$

where dV is the volume of the fluid element displaced. Summing up over the area,

$F = \gamma V$, the weight of volume displaced.

It is seen that the equation holds good in both cases – immersed or floating.

Example. 4.1 *A cylinder of diameter 0.3 m and height 0.6 m stays afloat vertically in water at a depth of 1 m from the free surface to the top surface of the cylinder.*

Determine the buoyant force on the cylinder. *Check the value from basics*

Buoyant force = Weight of water displaced

$= (\pi \times 0.3^2/4)\ 0.6 \times 1000 \times 9.81 = 416.06$ N

This acts upward at the centre of gravity G

Check: Bottom is at 1.6 m depth. Top is at 0.6 m depth

Buoyant force = Force on the bottom face – Force on top face

$= (\pi \times 0.3^2/4)\ (1000 \times 9.81 \times 1.6 - 1000 \times 9.81 \times 1.0)$

= 416.16 N

Example. 4.2 ***Determine the maximum weight that may be supported*** *by a hot air balloon of 10 m diameter at a location where the air temperature is 20° C while the hot air temperature is 80° C. The pressure at the location is 0.8 bar. R = 287 J/kg. K.*

The forces that act on the balloon are its weight downward and the buoyant force upwards. The buoyant force equals the weight of the cold surrounding air displaced. The difference between these two gives the maximum weight that may be carried by the balloon. The volume of cold air displaced equals the volume of the balloon. The pressure is assumed to be the same both inside and outside of the balloon.

Volume of balloon = $(4 \times \pi \times 5^3/3) = 523.6\ m^3$

Mass of hot air = $(PV/RT) = 0.8 \times 10^5 \times 523.6/287 \times (273 + 80)$
= 413.46 kg.

Weight of hot air = m.g = 413.46 × 9.81 = 4056 N

Weight of cold air = $[0.8 \times 10^5 \times 523.6 \times 9.81] / [287 \times (273 + 20)]$
= 4886.6 N

Weight that can be carried by the balloon = 4886.6 – 4056
= **830.6 N** (*i.e.*, about 84.67 kg mass under earths gravity)

This should include the weight of the balloon material and fittings.

4.2 STABILITY OF SUBMERGED AND FLOATING BODIES

There are three possible situations for a body when immersed in a fluid.

(*i*) If the **weight of the body is greater** than the weight of the liquid of equal volume then the **body will sink into** the liquid (To keep it floating additional upward force is required).

(*ii*) If the **weight of the body equals** the weight of equal volume of liquid, then the body will submerge and may **stay at any location** below the surface.

(*iii*) If the **weight of the body is less** than the weight of equal volume of liquid, then the body will be partly submerged and **will float in** the liquid.

Comparison of densities cannot be used directly to determine whether the body will float or sink unless the body is solid over the full volume like a lump of iron. However the apparent density calculated by the ratio of weight to total volume can be used to check whether a body will float or sink. If apparent density is higher than that of the liquid, the body will sink. If these are equal, the body will stay afloat at any location. If it is less, the body will float with part above the surface.

A submarine or ship though made of denser material floats because, the weight/volume of the ship will be less than the density of water. In the case of submarine its weight should equal the weight of water displaced for it to lay submerged.

Stability of a body: A ship or a boat should not overturn due to small disturbances but should be stable and return, to its original position. Equilibrium of a body exists when there is no resultant force or moment on the body. A body can stay in three states of equilibrium.

(*i*) **Stable equilibrium: Small disturbances will create a correcting couple** and the body will go back to its original position prior to the disturbance.

(*ii*) **Neutral equilibrium Small disturbances do not create any additional force** and so the body remains in the disturbed position. No further change in position occurs in this case.

(iii) **Unstable equilibrium:** A small **disturbance creates a couple which acts to increase the disturbance** and the body may tilt over completely.

Under equilibrium conditions, two forces of equal magnitude acting along the same line of action, but in the opposite directions exist on a floating/submerged body. These are the gravitational force on the body (weight) acting downward along the centroid of the body and buoyant force acting upward along the centroid of the displaced liquid. Whether floating or submerged, under equilibrium conditions these two forces are equal and opposite and act along the same line.

When the position of the body is disturbed or rocked by external forces (like wind on a ship), the position of the centre of gravity of the body **(with respect to the body)** remains at the same position. But the shape of the displaced volume of liquid changes and so its centre of gravity shifts to a new location. Now these two forces constitute a couple which may correct the original tilt or add to the original tilt. If the couple opposes the movement, then the body will regain or go back to the original position. If the couple acts to increase the tilt then the body becomes unstable. These conditions are illustrated in Fig 4.2.

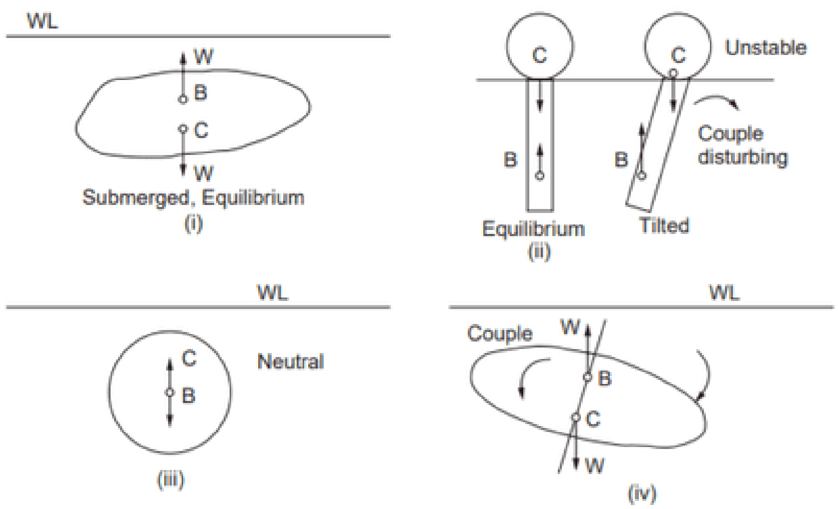


Figure 4.2.1 Stability of floating and submerged bodies

Figure 4.2.1 *(i)* and 4.2.1 *(ii)* shows bodies under equilibrium condition. Point C is the centre of gravity. Point B is the centre of buoyancy. It can be seen that the gravity and buoyant forces are equal and act along the same line but in the opposite directions.

Figure 4.2 *(iii)* shows the body under neutral equilibrium. The centre of gravity and the centre of buoyancy conicide.

Figures 4.2.1 (*iv*) and 4.2.1 (*v*) shows the objects in Figures 4.2.1 (*i*) and 4.2.1 (*ii*) in a slightly disturbed condition. Under such a condition a couple is found to form by the two forces, because the point of application of these forces are moved to new positions. In the case of Figure 4.2.1 (*iv*) the couple formed is opposed to the direction of disturbance and tends to return the body to the original position. This body is in a state of stable equilibrium. The couple is called righting couple. In the case of Figure 4.2.1 (*ii*) the couple formed is in the same direction as the disturbance and hence tends to increase the disturbance. This body is in unstable equilibrium. In the case of figure 4.2.1 (*iii*) no couple is formed due to disturbance as both forces act at the same point. Hence the body will remain in the disturbed position.

In the case of top heavy body (Figure 4.2 (*ii*)) the couple created by a small disturbance tends to further increase the tilt and so the body is unstable.

It is essential that the stability of ships and boats are well established. The equations and calculations are more involved for the actual shapes. Equations will be derived for simple shapes and for small disturbances. (**Note:** For practical cases, the calculations will be elaborate and cannot be attempted at this level.)

4.3 CONDITIONS FOR THE STABILITY OF FLOATING BODIES

(*i*) **When the centre of buoyancy is above the centre of gravity of the floating body, the body is always stable** under all conditions of disturbance. A righting couple is always created to bring the body back to the stable condition.

(*ii*) **When the centre of buoyancy coincides with the centre of gravity,** the two forces act at the same point. A disturbance does not create any couple and so **the body just remains in the disturbed position.** There is no tendency to tilt further or to correct the tilt.

(*iii*) **When the centre of buoyancy is below the centre of gravity** as in the case of ships, **additional analysis is required to establish stable conditions** of floating.

This involves the **concept of metacentre and metacentric height.** When the body is disturbed the centre of gravity still remains on the centroidal line of the body. The shape of the displaced volume changes and the centre of buoyancy moves from its previous position.

The location *M* at which the line of action of buoyant force meets the centroidal axis of the body, when disturbed, is defined as **metacentre.** The distance of this point from the centroid of the body is called metacentric height. This is illustrated in Figure 4.3.1.

If the metacentre is above the centroid of the body, the floating body will be stable. If it is at the centroid, the floating body will be in neutral equilibrium. If it is below the centroid, the floating body will be unstable.

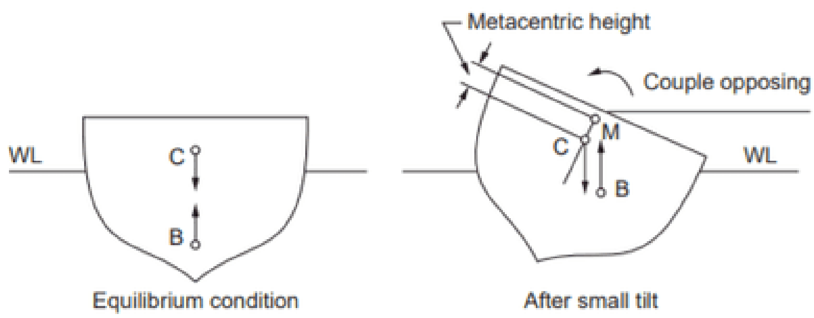


Figure 4.3.1 Metacentric height, stable condition

When a small disturbance occurs, say clockwise, then the centre of gravity moves to the right of the original centre line. The shape of the liquid displaced also changes and the centre of buoyancy also generally moves to the right. If the distance moved by the centre of buoyancy is larger than the distance moved by the centre of gravity, the resulting couple will act anticlockwise, correcting the disturbance. If the distance moved by the centre of gravity is larger, the couple will be clockwise and it will tend to increase the disturbance or tilting.

The distance between the metacentre and the centre of gravity is known is **metacentric height**. The magnitude of the righting couple is directly proportional to the metacentric height. Larger the metacentric height, better will be the stability. Referring to Fig 4.4.1, the centre of gravity *G* is above the centre of buoyancy *B*. After a small clockwise tilt, the centre of buoyancy has moved to *B′*. The line of action of this force is upward and it meets the body centre line at the metacentre *M* which is above *G*. In this case metacentric height is positive and the body is stable. It may also be noted that the couple is anticlockwise. If *M* falls below *G*, then the couple will be clockwise and the body will be unstable.

4.4 METACENTRIC HEIGHT

A floating object is shown in Figure 4.4.1 in section and plan view (part). In the tilted position, the submerged section is *FGHE*. Originally the submerged portion is *AFGHD*. Uniform section is assumed at the water line, as the angle of tilt is small. The original centre of buoyancy *B* was along the centre line. The new location *B′* can be determined by a moment balance. Let it move through a distance *R*. Let the weight of the wedge portion be *P*.

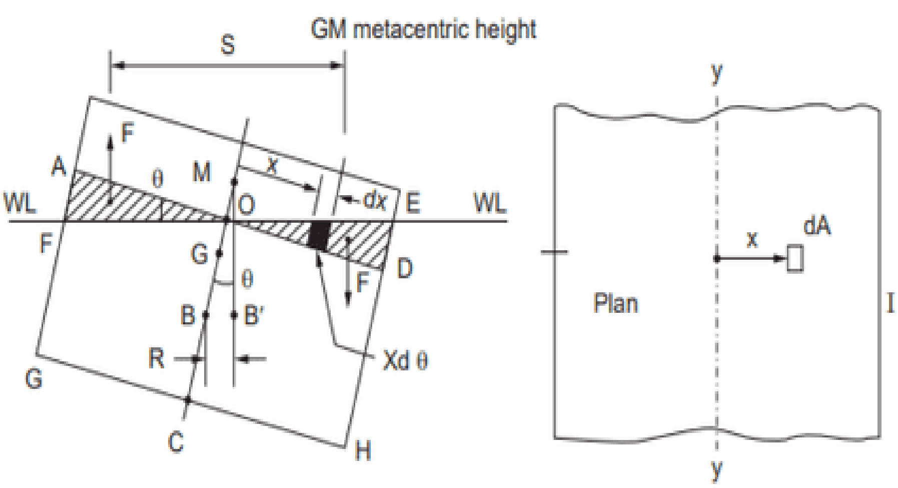


Figure 4.4.1 Metacentric height – derivation

The force system consists of the original buoyant force acting at B and the forces due to the wedges and the resultant is at B' due to the new location of the buoyant force.

Taking moments about B, $P \times S = W \times R$

The moment $P \times S$ can be determined by taking moments of elements displaced about O, the intersection of water surface and centre line.

Consider a small element at x with area dA

The height of the element = $\theta \times x$ (as θ is small, expressed in radians)

The mass of the element $\gamma \times \theta\, dA$ (γ – specific weight). The moment distance is x.

$$\therefore \quad P \times S = \gamma\,\theta \int_A x^2\, dA = \gamma\, I\, \theta \qquad (4.4.1)$$

where $I = \int_A x^2\, dA$, moment of inertia about the axis $y - y$

$$W \times R = \gamma\,\theta\, I, \text{ but } W = V\gamma \text{ where } V \text{ is the displaced volume}$$

$$\therefore \quad \gamma\,\theta\, I = V\gamma R \qquad (4.4.2)$$

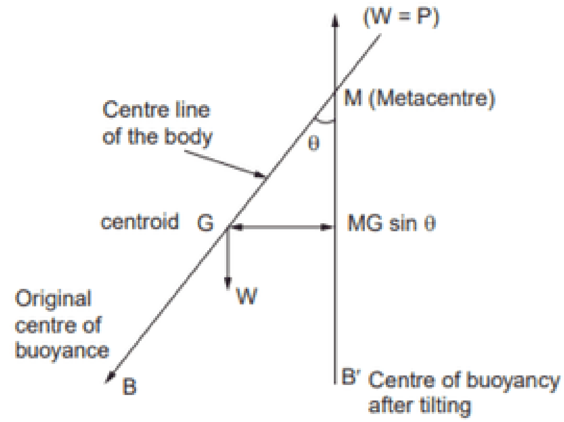
From the triangle MBB′, $R = MB \sin\theta$ or $R = MB\,\theta$

$$\therefore \quad MB = R/\theta = I/V \qquad (4.4.3)$$

Both I and V are known. As $V = W/\gamma$, the metacentric height is given by,

$$MG = MB \pm GB \qquad (4.4.4)$$

GB is originally specified. So the metacentric height can be determined. If G is above B –ve sign is used. If G is below B +ve sign is to be used.



B = W acts vertically along $B'M$. in the upward direction W acts vertically downwards at G. The distance between the couple formed is $MG \sin\theta$. Hence the righting couple

$$= \gamma V\, \overline{MG}\ \theta = W\, \overline{MG}\ \sin\theta.$$

Example. 4.3 *A ship's plan view is in the form of an ellipse with a major axis of 36 m and minor axis of 12 m. The mass of the ship is 1000 tons. The centre of buoyancy is 1.8 m below the water level and the centre of gravity is 0.3 m below the water level.* ***Determine the metacentric height for rolling*** *(y – y axis) and* ***pitching*** *(x – x* axis).

$$MG = (I/V) \pm GB,\ GB = 1.8 - 0.3 = 1.5 \text{ m}$$

For rolling:

$$\mathbf{I} = x\,(bh^3/64) = \boldsymbol{\pi \times 36 \times 12^3/64} = 3053.63 \text{ m}^4,\ V = W/\gamma = m/\rho$$

Considering sea water of density 1030 kg/m^3 and V as the liquid volume displaced.

$$V = 1000{,}000/1030 = 970.87 \text{ m}^3,\ MB = I/V = 3053.63/970.87$$

$$MG = MB - GB = (3053.62/970.87) - 1.5 = 3.15 - 1.5 = \mathbf{1.65\ m}$$

(–ve) sign as B is below G). This is positive and so the ship is stable about rolling by small angles.

For **pitching**

$$\mathbf{MG} = (\pi \times 12 \times 36^3/64 \times 970.87) - 1.5 = \mathbf{28.3\ m}$$

Highly stable in this direction. This situation is for small angles and uniform section at the water line.

MODULE 3

FLUID KINEMATICS AND DYNAMICS

5.0 INTRODUCTION

In the previous three chapters the pressure distribution in static fluids and its effect on surfaces exposed to the fluid was discussed. In this chapter the flow of ideal fluids will be discussed. The main attempt in this chapter is to visualise flow fields. A flow field is a region in which the flow is defined at all points at any instant of time. The means to that is to define the velocities at all the points at different times. It should be noted that the velocity at a point is the velocity of the fluid particle that occupies that point. In order to obtain a complete picture of the flow the fluid motion should be described mathematically. Just like the topography of a region is visualised using the contour map, the flow can be visualised using the velocity at all points at a given time or the velocity of a given particle at different times. It is then possible to also define the potential causing the flow.

Application of a shear force on an element or particle of a fluid will cause continuous deformation of the element. Such continuing deformation will lead to the displacement of the fluid element from its location and this results in fluid flow. The fluid element acted on by the force may move along a steady regular path or randomly changing path depending on the factors controlling the flow. The velocity may also remain constant with time or may vary randomly. In some cases the velocity may vary randomly with time but the variation will be about a mean value . It may also vary completely randomly as in the atmosphere. The study of the velocity of various particles in the flow and the instantaneous flow pattern of the flow field is called flow kinematics or hydrodynamics. Such a study is generally limited to ideal fluids, fluids which are incompressible and inviscid. In real fluid shows, beyond a certain distance from the surfaces, the flow behaves very much like ideal fluid. Hence these studies are applicable in real fluid flow also with some limitations.

5.1 LAGRANGIAN AND EULARIAN METHODS OF STUDY OF FLUID FLOW

In the Lagrangian method a single particle is followed over the flow field, the co-ordinate system following the particle. The flow description is particle based and not space based. A moving coordinate system has to be used. This is equivalent to the observer moving with the particle to study the flow of the particle. This method is more involved mathematically and is used mainly in special cases.

In the Eularian method, the description of flow is on fixed coordinate system based and the description of the velocity etc. are with reference to location and time *i.e.*, $V = V\ (x, y, z, t)$ and not with reference to a particular particle. Such an analysis provides a picture of various parameters at all locations in the flow field at different instants of time. This method provides an easier visualisation of the flow field and is popularly used in fluid flow studies. However the final description of a given flow will be the same by both the methods.

5.2 BASIC SCIENTIFIC LAWS USED IN THE ANALYSIS OF FLUID FLOW

(i) Law of conservation of mass: This law when applied to a control volume states that the net mass flow through the volume will equal the mass stored or removed from the volume. Under conditions of steady flow this will mean that the mass leaving the control volume should be equal to the mass entering the volume. The determination of flow velocity for a specified mass flow rate and flow area is based on the continuity equation derived on the basis of this law.

(ii) Newton's laws of motion: These are basic to any force analysis under various conditions of flow. The resultant force is calculated using the condition that it equals the rate of change of momentum. The reaction on surfaces are calculated on the basis of these laws. Momentum equation for flow is derived based on these laws.

(iii) Law of conservation of energy: Considering a control volume the law can be stated as "the energy flow into the volume will equal the energy flow out of the volume under steady conditions". This also leads to the situation that the total energy of a fluid element in a steady flow field is conserved. This is the basis for the derivation of Euler and Bernoulli equations for fluid flow.

(iv) Thermodynamic laws: are applied in the study of flow of compressible fluids.

5.3 FLOW OF IDEAL / INVISCID AND REAL FLUIDS

Ideal fluid is nonviscous and incompressible. Shear force between the boundary surface and fluid or between the fluid layers is absent and only pressure forces and body forces are controlling.

Real fluids have viscosity and surface shear forces are involved during flow. However the flow after a short distance from the surface is not affected by the viscous effects and approximates to ideal fluid flow. The results of ideal fluid flow analysis are found applicable in the study of flow of real fluids when viscosity values are small.

5.5 COMPRESSIBLE AND INCOMPRESSIBLE FLOW

If the density of the flowing fluid is the same all over the flow field at all times, then such flow is called incompressible flow. Flow of liquids can be considered as incompressible even if the density varies a little due to temperature difference between locations. Low velocity flow of gases with small changes in pressure and temperature can also be considered as incompressible flow. Flow through fans and blowers is considered incompressible as long as the density variation is below 5%. If the density varies with location, the flow is called compressible flow. In this chapter the study is mainly on incompressible flow.

5.4 STEADY AND UNSTEADY FLOW

In order to study the flow pattern it is necessary to classify the various types of flow. The classification will depend upon the constancy or variability of the velocity with time. In the next three sections, these are described. In steady flow the property values at a location in the flow are constant and the values do not vary with time. The velocity or pressure at a point remains constant with time. These can be expressed as $V = V(x, y, z)$, $P = P(x, y, z)$ etc. In steady flow a picture of the flow field recorded at different times will be identical. In the case of unsteady flow, the properties vary with time or $V = V(x, y, z, t)$, $P = P(x, y, z, t)$ where t is time.

In unsteady flow the appearance of the flow field will vary with time and will be constantly changing. In turbulent flow the velocity at any point fluctuates around a mean value, but the mean value at a point over a period of time is constant. For practical purposes turbulent flow is considered as steady flow as long as the mean value of properties do not vary with time.

5.6 LAMINAR AND TURBULENT FLOW

If the flow is smooth and if the layers in the flow do not mix macroscopically then the flow is called laminar flow. For example a dye injected at a point in laminar flow will travel along a continuous smooth line without generally mixing with the main body of the fluid. Momentum, heat and mass transfer between layers will be at molecular level of pure diffusion. In laminar flow layers will glide over each other without mixing.

In turbulent flow fluid layers mix macroscopically and the velocity/temperature/mass concentration at any point is found to vary with reference to a mean value over a time period. For example $u = \overline{u} + u'$ where u is the velocity at an instant at a location and $\overline{u}$ is the average velocity over a period of time at that location and u' is the fluctuating component. This causes

higher rate of momentum/heat/mass transfer. A dye injected into such a flow will not flow along a smooth line but will mix with the main stream within a short distance.

The difference between the flows can be distinguished by observing the smoke coming out of an incense stick. The smoke in still air will be found to rise along a vertical line without mixing. This is the laminar region. At a distance which will depend on flow conditions the smoke will be found to mix with the air as the flow becomes turbulent. Laminar flow will prevail when viscous forces are larger than inertia forces. Turbulence will begin where inertia forces begin to increase and become higher than viscous forces.

5.7 CONCEPTS OF UNIFORM FLOW, REVERSIBLE FLOW AND THREE DIMENSIONAL FLOW

If the velocity value at all points in a flow field is the same, then the flow is defined as uniform flow. The velocity in the flow is independent of location. Certain flows may be approximated as uniform flow for the purpose of analysis, though ideally the flow may not be uniform.

If there are no pressure or head losses in the fluid due to frictional forces to be overcome by loss of kinetic energy (being converted to heat), the flow becomes reversible. The fluid can be restored to its original condition without additional work input. For a flow to be reversible, no surface or fluid friction should exist. The flow in a venturi (at low velocities) can be considered as reversible and the pressures upstream and downstream of the venturi will be the same in such a case. The flow becomes irreversible if there are pressure or head losses.

If the components of the velocity in a flow field exist only in one direction it is called one dimensional flow and $V = V(x)$. Denoting the velocity components in x, y and z directions as u, v and w, in one dimensional flow two of the components of velocity will be zero. In two dimensional flow one of the components will be zero or $V = V(x, y)$. In three dimensional flow all the three components will exist and $V = V(x, y, z)$. This describes the general steady flow situation. Depending on the relative values of u, v and w approximations can be made in the analysis. In unsteady flow $V = V(x, y, z, t)$.

5.8 VELOCITY AND ACCELERATION COMPONENTS

The components of velocity can be designated as

$$u = \frac{dx}{dt},\ v = \frac{dy}{dt} \quad \text{and} \quad w = \frac{dz}{dt}$$

where t is the time and dx, dy, dz are the displacements in the directions x, y, z.

In general as $u = u(x, y, z, t)$, $v = v(x, y, z, t)$ and $w = w(x, y, z, t)$

Defining acceleration components as

$$a_x = \frac{du}{dt},\ a_y = \frac{dv}{dt} \text{ and } a_z = \frac{dw}{dt}, \quad \text{as } u = u\,(x, y, z, t)$$

$$a_x = \frac{\partial u}{\partial x}\frac{\partial x}{\partial t} + \frac{\partial u}{\partial y}\frac{\partial y}{\partial t} + \frac{\partial u}{\partial z}\frac{\partial z}{\partial t} + \frac{\partial u}{\partial t}$$

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

Similarly,
$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t}$$

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

The first three terms in each case is known as convective acceleration terms, because these represent the convective act of moving from one position to another. The last term is known as local accleration term, because the flow at a point is changing with time. Under steady flow conditions, only the convective acceleration terms will exist.

5.12 STREAM LINES, STREAM TUBE, PATH LINES, STREAK LINES AND TIME LINES

The analytical description of flow velocity is geometrically depicted through the concept of stream lines. The velocity vector is a function of both position and time. If at a fixed instant of time a curve is drawn so that it is tangent everywhere to the velocity vectors at these locations

then the curve is called a stream line. Thus stream line shows the mean direction of a number of particles in the flow at the same instant of time. **Stream lines are a series of curves drawn tangent to the mean velocity vectors of a number of particles in the flow. Since stream lines are tangent to the velocity vector at every point in the flow field, there can be no flow across a stream line.**

A bundle of neighbouring stream lines may be imagined to form a passage through which the fluid flows. Such a passage is called a stream tube. Since the stream tube is bounded on all sides by stream lines, there can be no flow across the surface. Flow can be only through the ends. A stream tube is shown diagrammatically in Figure 5.12.1.

Under steady flow condition, the flow through a stream tube will be constant along the length.

Path line is the trace of the path of a single particle over a period of time. Path line shows the direction of the velocity of a particle at successive instants of time. In steady flow path lines and stream lines will be identical.

Streak lines provide an instantaneous picture of the particles, which have passed through a given point like the injection point of a dye in a flow. In steady flow these lines will also coincide with stream lines.

Path lines and streak lines are shown in Figure 5.12.1.

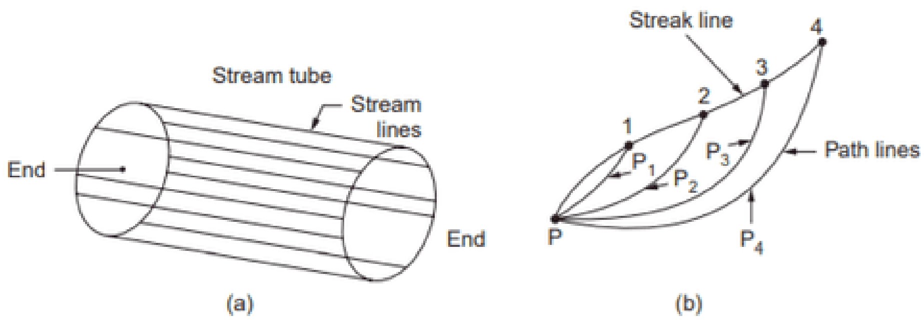


Figure 5.12.1 Stream tube, Path lines and Streak lines

Particles P_1, P_2, P_3, P_4, starting from point P at successive times pass along path lines shown. At the instant of time considered the positions of the particles are at 1, 2, 3 and 4. A line joining these points is the streak line.

If a number of adjacent fluid particles in a flow field are marked at a given instant, they form a line at that instant. This line is called time line. Subsequent observations of the line may provide information about the flow field. For example the deformation of a fluid under shear force can be studied using time lines.

5.13 CONCEPT OF STREAM LINE

In a flow field if a continuous line can be drawn such that the tangent at every point on the line gives the direction of the velocity of flow at that point, such a line is defined as a stream line. In steady flow any particle entering the flow on the line will travel only along this line. This leads to visualisation of a stream line in laminar flow as the path of a dye injected into the flow.

There can be no flow across the stream line, as the velocity perpendicular to the stream line is zero at all points. The flow along the stream line can be considered as one dimensional flow, though the stream line may be curved as there is no component of velocity in the other directions. Stream lines define the flow paths of streams in the flow. The flow entering between two stream lines will always flow between the lines. The lines serve as boundaries for the stream.

5.14 CONCEPT OF STREAM FUNCTION

Refer to Fig. 5.14.1 showing the flow field, co-ordinate system and two stream lines.

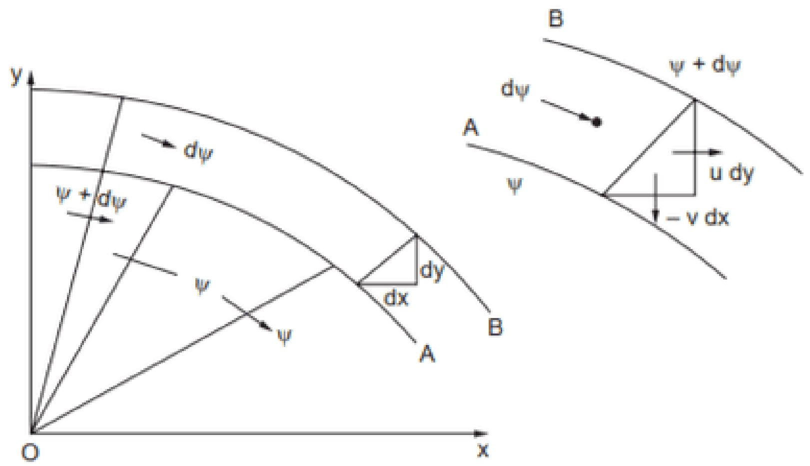


Figure. 5.14.1 Stream function—Definition

Stream function is a mathematical expression that describes a flow field. The definition is based on the continuity principle. It provides a means of plotting and interpreting flow fields. Considering the stream line A in figure, the flow rate across any line joining 0 and any point on A should be the same as no flow can cross the stream line A. Let the slow rate be denoted as ψ. Then ψ is a constant of the streamline A. If ψ can be described by an equation in x and y then stream line A can be plotted on the flow field. Consider another stream line B close to A. Let the flow between stream lines A and B be $d\psi$. The flow across any line between A and B will be dψ. Now taking components in the x and y directions,

$$d\psi = u\, dy - v\, dx \tag{5.14.1}$$

If the stream function ψ can be expressed as $\psi = \psi(x, y)$ (as it has a value at every point) then

$$d\psi = \frac{\partial \psi}{\partial x} dx + \frac{\partial \psi}{\partial y} dy \tag{5.14.2}$$

and comparing the above two equations, it is seen that

$$u = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = -\frac{\partial \psi}{\partial x} \tag{5.14.3}$$

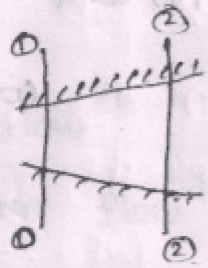
Continuity equation:

The equation based on the principle of conservation of mass is called "continuity equation".

$$Q_1 = \rho_1 A_1 V_1$$

$$Q_2 = \rho_2 A_2 V_2$$

$$Q_1 = Q_2$$



The fluid of water $\rho_2 = \rho_1$

$$A_1 V_1 = A_2 V_2$$

where,

ρ_1 = Density at section 1-1

A_1 = Area of pipe at section 1-1

V_1 = Average velocity at cross section 1-1

Q = discharge

Problems :

1) The diameter of a pipe at the secton 1-2 are 10cm & 15cm respectivly. Find the discharge the pipe if velocity of water flowing through a pipe at section 1 as 5 m/s. Determine also the velocity at section 2.

Data :

$D_1 = 10$ cm $\Rightarrow$ 1 cm = 100 m

$\Rightarrow 0.1$ m

$D_2 = 15$ cm

$\Rightarrow 0.15$ m

$v_1 = 5$ m/s

Find :

i) $v_2 = ?$

ii) $Q_1 = ?$, $Q_2 = ?$

Solution :

$$A_1 = \pi/4 (d_1^2) = \pi/4 (0.1)^2 = 7.83 \times 10^{-3} \; m^2$$

$$A_2 = \pi/4 (D_2)^2 = \pi/4 (0.15)^2 = 0.01767 \; m^2$$

$$A_1 V_1 = A_2 V_2$$

$$V_2 = \frac{A_1 V_1}{A_2} = \frac{7.83 \times 10^{-3} \times 5}{0.01767} = 2\ m/s$$

$$Q_1 = A_1 V_1$$

$$= 7.83 \times 10^{-3} \times 5$$

$$Q_1 = 0.03915\ m^3/s$$

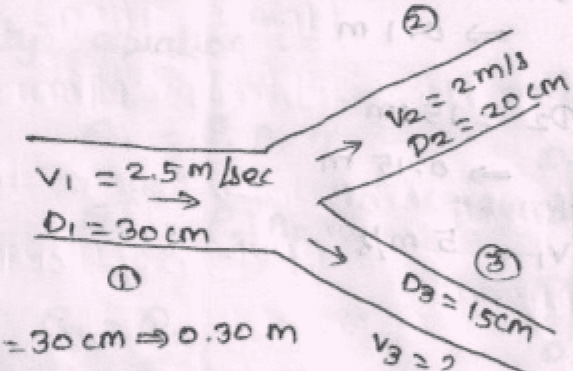
$$Q_2 = A_2 V_2$$

$$= 0.01767 \times 2$$

$$Q_2 = 0.03534\ m^3/s$$

8) A 30 cm diameter pipe, conveying water, branches into two pipes of diameters 20 cm and 15 cm respectively. If the average velocity in the 30 cm diameter pipe is 2.5 m/s. Find the discharge in this pipe. Also determine the velocity in 15 cm pipe if the average velocity in 20 cm diameter pipe is 2 m/s.

Data:



$D_1 = 30$ cm $\Rightarrow 0.30$ m

$D_2 = 20$ cm $\Rightarrow 0.20$ m

$D_3 = 15$ cm $\Rightarrow 0.15$ m

Find:

$V_3 = ?$

Solution :

$$A_1 = \pi/4 (d_1)^2$$

$$= \pi/4 (0.30)^2$$

$$A_1 = 0.07068 \ m^2$$

$$A_2 = \pi/4 (d_2)^2$$

$$= \pi/4 (0.20)^2$$

$$A_2 = 0.0314 \ m^2$$

$$A_3 = \pi/4 (d_3^2)$$

$$= \pi/4 (0.15)^2$$

$$A_3 = 0.01767 \ m^2$$

Discharge,

$$Q_1 = Q_2 + Q_3$$

$$Q_1 = A_1 \times V_1$$

$$= 0.07068 \times 2.5$$

$$\boxed{Q_1 = 0.1767 \ m^3/s}$$

$$Q_2 = A_2 \times V_2$$

$$= 0.0314 \times 2$$

$$\boxed{Q_2 = 0.0628 \ m^3/s}$$

$$\therefore Q_1 = Q_2 + Q_3$$

$$Q_3 = Q_1 - Q_2$$

$$= 0.1767 - 0.0628$$

$$\boxed{Q_3 = 0.1139 \ m^3/s}$$

$$Q_3 = A_3 \times V_3$$

$$V_3 = \frac{Q_3}{A_3}$$

$$= \frac{0.1139}{0.01767}$$

$$\boxed{V_3 = 6.44 \text{ m/s}}$$

6.1 FORMS OF ENERGY ENCOUNTERED IN FLUID FLOW

Energy associated with a fluid element may exist in several forms. These are listed here and the method of calculation of their numerical values is also indicated.

6.1.2 Potential Energy

This energy is due to the position of the element in the gravitational field. While a zero value for *KE* is possible, the value of potential energy is relative to a chosen datum. The value of potential energy is given by

$$PE = mZ\, g/g_o \text{ Nm} \qquad (6.1.3)$$

Where m is the mass of the element in kg, Z is the distance from the datum along the gravitational direction, in m. The unit will be (kg m m/s^2) × (Ns^2/kgm) *i.e.,* Nm. The specific potential energy (per kg) is obtained by dividing equation 6.1.3 by the mass of the element.

$$PE = Z\, g/g_0 \text{ Nm/kg} \qquad (6.1.3.\ b)$$

This gives the physical quantity of energy associated with 1 kg due to the position of the fluid element in the gravitational field above the datum. As in the case of the kinetic energy, the value of *PE* also is expressed as head of fluid, *Z*.

$$PE = Z\,(g/g_o)\,(g_o/g) = Z\ m. \qquad (6.1.4)$$

6.1.1 Kinetic Energy

This is the energy due to the motion of the element as a whole. If the velocity is V, then the kinetic energy for m kg is given by

$$KE = \frac{mV^2}{2g_o} \text{ Nm} \tag{6.1.1}$$

The unit in the SI system will be Nm also called Joule (J)

{(kg m^2/s^2)/(kg m/N s^2)}

The same referred to one kg (specific kinetic energy) can be obtained by dividing 6.1.1 by the mass m and then the unit will be Nm/kg.

$$KE = \frac{V^2}{2g_o}, \text{ Nm/kg} \tag{6.1.1b}$$

In fluid flow studies, it is found desirable to express the energy as the head of fluid in m. This unit can be obtained by multiplying equation (6.1.1) by g_o/g.

$$\text{Kinetic head} = \frac{V^2}{2g_o}\frac{g_o}{g} = \frac{V^2}{2g} \tag{6.1.2}$$

The unit for this expression will be $\frac{m^2 s^2}{s^2 m}$ = m

Apparently the unit appears as metre, but in reality it is Nm/N, where the denominator is weight of the fluid in N.

The equation in this form is used at several places particularly in flow of liquids. But the energy associated physically is given directly only be equation 6.1.1.

The learner should be familiar with both forms of the equation and should be able to choose and use the proper equation as the situation demands. **When different forms of the energy of a fluid element is summed up to obtain the total energy, all forms should be in the same unit.**

This form will be used in equations, but as in the case of KE, one should be familiar with both the forms and choose the suitable form as the situation demands.

6.1.3 Pressure Energy (Also Equals Flow Energy)

The element when entering the control volume has to flow against the pressure at that location. The work done can be calculated referring Fig. 6.1.1.

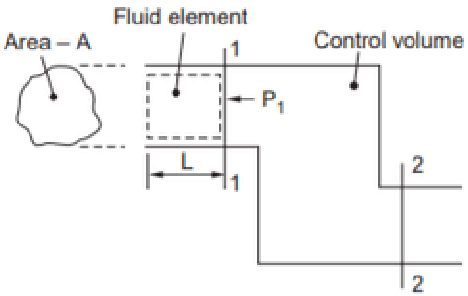


Figure 6.1.1 Flow work calculation

The boundary of the element of fluid considered is shown by the dotted line, Force = $P_1 A$, distance to be moved = L, work done = $P_1 AL = P_1\ mv$ as AL = volume = mass × specific volume, v. ∴ flow work = $P\ mv$.

The pressure energy per kg can be calculated using $m = 1$. The flow energy is given by

$$FE = P.v = P/\rho,\ \text{Nm/kg} \qquad (6.1.5)$$

Note: $$\frac{N}{m^2}\frac{m^3}{kg} \rightarrow \frac{Nm}{kg}$$

As in the other cases, the flow energy can also expressed as head of fluid.

$$FE = \frac{P}{\rho}\frac{g_o}{g},\ \text{m} \qquad (6.1.5a)$$

As specific weight $\gamma = \rho\ g/g_o$, the equation is written as,

$$FE = P/\gamma,\ \text{m} \qquad (6.1.5b)$$

It is important that in any equation, when energy quantities are summed up consistent forms of these set of equations should be used, that is, all the terms should be expressed either as head of fluid or as energy (J) per kg. These are the three forms of energy encountered more often in flow of incompressible fluids.

6.1.4 Internal Energy

This is due to the thermal condition of the fluid. This form is encountered in compressible fluid flow. For gases (above a datum temperature) $IE = c_v\ T$ where T is the temperature above the datum temperature and c_v is the specific heat of the gas at constant volume. The unit for internal energy is J/kg (Nm/kg). When friction is significant other forms of energy is converted to internal energy both in the case of compressible and incompressible flow.

6.3 EULER'S EQUATION OF MOTION FOR FLOW ALONG A STREAM LINE

Consider a small element along the stream line, the direction being designated as *s*.

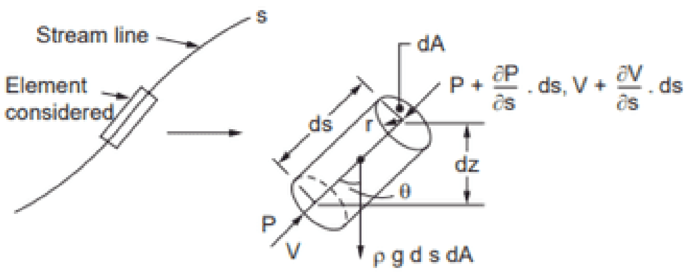


Figure 6.3.1 Euler's equation of Motion – Derivation

The net force on the element are the body forces and surface forces (pressure). These are indicated in the figure. Summing this up, and equating to the change in momentum.

$$PdA - \{P + (\partial P/\partial s\} \, dA - \rho g \, dA \, ds \cos\theta = \rho \, dA \, ds \, a_s \qquad (6.3.1)$$

where a_s is the acceleration along the *s* direction. This reduces to,

$$\frac{1}{\rho}\frac{\partial P}{\partial s} + g\cos\theta + a_s = 0 \qquad (6.3.2)$$

(**Note**: It will be desirable to add g_o to the first term for dimensional homogeneity. As it is, the first term will have a unit of N/kg while the other two terms will have a unit of m/s². Multiplying by g_o, it will also have a unit of m/s²).

$$a_s = dV/dt, \text{ as velocity, } V = f(s, t), (t = \text{time}).$$

$$dV = \frac{\partial V}{\partial s} ds + \frac{\partial V}{\partial t} dt \quad \text{dividing by } dt,$$

$$\frac{dV}{dt} = \frac{\partial V}{\partial s}\frac{ds}{dt} + \frac{\partial V}{\partial t} \qquad \text{As } \frac{ds}{dt} = V,$$

and as cos θ = dz/ds, equation 6.3.2 reduces to,

$$\frac{1}{\rho}\frac{\partial P}{\partial s} + g\frac{\partial z}{\partial s} + V\frac{\partial V}{\partial s} + \frac{\partial V}{\partial t} = 0 \qquad (6.3.2.\ a)$$

For steady flow $\partial V/\partial t = 0$. Cancelling ∂s and using total derivatives in place of partials as these are independent quantities.

$$\frac{dp}{\rho} + gdz + VdV = 0 \qquad (6.3.3)$$

(**Note**: in equation 6.3.3 also it is better to write the first term as $g_o.dp/\rho$ for dimensional homogeneity).

This equation after dividing by g, is also written as,

$$\frac{dp}{\gamma} + d\left(\frac{V^2}{2g}\right) + dz = 0 \quad \text{or} \quad d\left[\frac{P}{\gamma} + \frac{V^2}{2g} + z\right] = 0 \qquad (6.3.4)$$

which means that the quantity within the bracket remains constant along the flow.

This equation is known as Euler's equation of motion. The assumptions involved are:

1. Steady flow
2. Motion along a stream line and
3. Ideal fluid (frictionless)

In the case on incompressible flow, this equation can be integrated to obtain Bernoulli equation.

6.4 BERNOULLI EQUATION FOR FLUID FLOW

Euler's equation as given in 6.3.3 can be integrated directly if the flow is assumed to be incompressible.

$$\frac{dP}{\rho} + gdz + VdV = 0, \quad \text{as } \rho = \text{constant}$$

$$\frac{P}{\rho} + gz + \frac{V^2}{2} = \text{const. or } \frac{P}{\rho} + z\left(\frac{g}{g_0}\right) + \frac{V^2}{2g_0} = \text{Constant} \qquad (6.4.1)$$

The constant is to be evaluated by using specified boundary conditions. The unit of the terms will be energy unit (Nm/kg).

In SI units the numerical value of g_o = 1, kg m/N s^2. Equation 6.4.1 can also be written as to express energy as head of fluid column.

$$\frac{P}{\gamma} + z + \frac{V^2}{2g} = \text{constant} \qquad (6.4.2)$$

(γ is the specific weight N/m^3). In this equation all the terms are in the unit of head of the fluid.

The constant has the same value along a stream line or a stream tube. The first term represents (flow work) pressure energy, the second term the potential energy and the third term the kinetic energy.

This equation is extensively used in practical design to estimate pressure/velocity in flow through ducts, venturimeter, nozzle meter, orifice meter etc. In case energy is added or taken out at any point in the flow, or loss of head due to friction occurs, the equations will read as,

$$\frac{P_1}{\rho} + \frac{V_1^2}{2g_o} + \frac{z_1 g}{g_o} + W - \frac{h_f g}{g_o} = \frac{P_2}{\rho} + \frac{V_2^2}{2g_o} + \frac{z_2 g}{g_o} \qquad (6.4.3)$$

where W is the energy added and h_f is the loss of head due to friction.

In calculations using SI system of units g_o may be omitting as its value is unity.

Example 6.1 *A liquid of specific gravity 1.3 flows in a pipe at a rate of 800 l/s, from point 1 to point 2 which is 1 m above point 1. The diameters at section 1 and 2 are 0.6 m and 0.3 m respectively. If the pressure at section 1 is 10 bar,* ***determine the pressure at section 2.***

Using Bernoulli equation in the following form (6.4.2)

$$\frac{P}{\gamma} + z + \frac{V^2}{2g} = \text{constant},$$

Taking the datum as section 1, the pressure P_2 can be calculated.

$V_1 = 0.8 \times 4/\pi \times 0.6^2 = 2.83$ *m/s*, $V_2 = 0.8 \times 4/\pi \times 0.3^3 = 11.32$ *m/s*

$P_1 = 10 \times 10^5$ N/m^2, γ = sp. gravity × 9810. Substituting.

$$\frac{10 \times 10^5}{9810 \times 1.3} + 0 + \frac{2.83^2}{2 \times 9.81} = \frac{P_2}{9810 \times 1.3} + 1 + \frac{11.32^2}{2 \times 9.81}$$

Solving, $\mathbf{P_2 = 9.092}$ **bar** (9.092×10^5 N/m^2).

As P/γ is involved directly on both sides, gauge pressure or absolute pressure can be used without error. However, it is desirable to use absolute pressure to avoid negative pressure values (or use of the term vacuum pressure).

Example 6.2 *Water flows through a horizontal venturimeter with diameters of 0.6 m and 0.2 m. The guage pressure at the entry is 1 bar.* ***Determine the flow rate*** *when the throat pressure is 0.5 bar (vacuum). Barometric pressure is 1 bar.*

Using Bernoulli's equation in the form,

$$\frac{P_1}{\gamma} + Z_1 + \frac{V_1^2}{2g} = \frac{P_2}{\gamma} + Z_2 + \frac{V_2^2}{2g}$$

and noting $Z_1 = Z_2$, $P_1 = 2 \times 10^5$ N/m^2 (absolute)

$P_2 = 0.5 \times 10^5$ N/m^2 (absolute), $\gamma = 9810$ N/m^3

$V_1 = Q \times 4/(\pi \times 0.60^2) = 3.54\ Q$, $V_2 = Q \times 4/(\pi \times 0.20^2) = 31.83Q$

$$\frac{2 \times 10^5}{9810} + 0 + \frac{3.54^2}{2 \times 9.81} Q^2 = \frac{0.5 \times 10^5}{9810} + 0 + \frac{31.83^2\, Q^2}{2 \times 9.81}$$

Solving, $\mathbf{Q = 0.548\ m^3/s}$, $V_1 = 1.94$ m/s, $V_2 = 17.43$ m/s.

7.8 DARCY–WEISBACH EQUATION FOR CALCULATING PRESSURE DROP

In the design of piping systems the choice falls between the selection of diameter and the pressure drop. The selection of a larger diameter leads to higher initial cost. But the pressure drop is lower in such a case which leads to lower operating cost. So in the process of design of piping systems it becomes necessary to investigate the pressure drop for various diameters of pipe for a given flow rate. Another factor which affects the pressure drop is the pipe roughness. It is easily seen that the pressure drop will depend directly upon the length and inversely upon the diameter. The velocity will also be a factor and in this case the pressure drop will depend in the square of the velocity (refer Bernoulli equation).

Hence we can say that

$$\Delta p \propto \frac{LV^2}{2D} \tag{7.8.1}$$

The proportionality constant is found to depend on other factors. In the process of such determination Darcy defined or friction factor f as

$$f = 4\,\tau_0/(\rho u_m^{\,2}/2g_0) \qquad (7.8.2)$$

This quantity is dimensionless which may be checked.

Extensive investigations have been made to determine the factors influencing the friction factor.

It is established that in laminar flow f depends only on the Reynolds number and it is given by

$$f = \frac{64}{\text{Re}} \qquad (7.8.3)$$

In the turbulent region the friction factor is found to depend on Reynolds number for smooth pipes and both on Reynolds number and roughness for rough pipes. Some empirical equations are given in section 7.4 and also under discussions on turbulent flow. The value of friction factor with Reynolds number with roughness as parameter is available in Moody diagram, given in the appendix. Using the definition of Darcy friction factor and conditions of equilibrium, expression for pressure drop in pipes is derived in this section. Consider an elemental length L in the pipe. The pressures at sections 1 and 2 are P_1 and P_2.

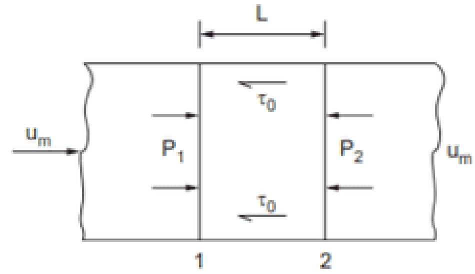


Figure 7.8.1

The other force involved on the element is the wall shear τ_0.

Net pressure force in the element is $(P_1 - P_2)$

Net shear force in the element is $\tau_0\, \pi\, DL$

Force balance for equilibrium yields

$$(P_1 - P_2)\,\frac{\pi D^2}{4} = \tau_0\, \pi\, DL \qquad (7.8.4)$$

From the definition friction factor

$$f = 4\,\tau_0 / (\rho\, u_m^{\,2}/ 2\, g_0)$$

$$\tau_0 = \frac{f\,\rho\, u_m^{\,2}}{8\, g_0}$$

Substituting and letting $(P_1 - P_2)$ to be ΔP.

$$\Delta P\,.\,\frac{\pi D^2}{4} = \frac{f\,\rho\, u_m^{\,2}}{8\, g_0}\,.\,\pi\, DL$$

This reduces to $$\Delta P = \frac{f\, L\, u_m^{\,2}\, \rho}{2\, g_0\, D} \qquad (7.8.5)$$

This equation known as Darcy-Weisbach equation and is generally applicable in most of the pipe flow problems. As mentioned earlier, the value of f is to be obtained either from equations or from Moody diagram. The diameter for circular tubes will be the hydraulic diameter D_h defined earlier in the text.

It is found desirable to express the pressure drop as head of the flowing fluid.

In this case as $$h = \frac{P}{\gamma} = \frac{P\, g_0}{\rho\, g}$$

$$\Delta h = h_f = \frac{f\, L\, u_m^{\,2}}{2\, g\, D} \tag{7.8.6}$$

The velocity term can be replaced in terms of volume flow and the equation obtained is found useful in designs as Q is generally specified in designs.

$$u_m = \frac{4\, Q}{\pi\, D^2},\ u_m^{\,2} = \frac{16\, Q^2}{\pi^2\, D^4}$$

Substituting in (7.8.6), we get

$$h_f = \frac{8\, f\, L\, Q^2}{\pi^2\, g\, D^5} \tag{7.8.7}$$

It is found that $h_f \propto \dfrac{Q^2}{D^5}$

Another coefficient of friction C_f is defined as $C_f = f/4$

In this case $$h_f = \frac{4C_f\, L\, u_m^{\,2}}{2\, g\, D} \tag{7.8.8}$$

Now a days equation 7.8.5 are more popularly used as value of f is easily available.

7.9 HAGEN–POISEUILLE EQUATION FOR FRICTION DROP

In the case of laminar flow in pipes another equation is available for the calculation of pressure drop. The equation is derived in this section.

Refer to section (7.7) equation (7.7.1)

$$u = -\frac{1}{\mu}\frac{dp}{dL}\frac{R^2}{4}\left[1-\left(\frac{r}{R}\right)^2\right]$$

$\frac{dP}{dL}$ can be approximated to $\Delta P/L$ as the pressure drop is uniform along the length L under steady laminar flow

Using eqn (7.7.2), $u_{max} = -\frac{dP}{dL}\frac{1}{\mu}\cdot\frac{R^2}{4} = 2u_m$

$$\therefore \quad -\frac{dP}{dL} = \frac{8\,u_m\,\mu}{R^2}$$

$$\therefore \quad -\frac{dP}{dL} = \frac{8u_m\mu}{R^2} = \frac{32u_m\mu}{D^2}$$, Substituting for $-\frac{dP}{dL}$ as $\frac{\Delta P}{L}$

$$\Delta P = \frac{32\,\mu\,u_m L}{D^2} \tag{7.9.1}$$

This can also be expressed in terms of volume flow rate Q as

$$Q = \frac{\pi D^2}{4}\,.\,u_m$$

$\therefore \quad u_m = 4Q/\pi D^2$, substituting

$$\Delta P = 128\,\mu\,L\,Q/\pi\,D^4 \tag{7.9.2}$$

Converting ΔP as head of fluid

$$h_f = \frac{32\,\upsilon u_m\,Lg_0}{gD^2} \tag{7.9.3}$$

This equation is known as Hagen-Poiseuille equation

g_0 is the force conversion factor having a value of unity in the SI system of unit. Also $(\mu / \rho) = \nu$.

Equations 7.9.1, 7.9.2 and 7.9.3 are applicable for laminar flow only whereas Darcy-Weisbach equation (7.8.6) is applicable for all flows

7.10 SIGNIFICANCE OF REYNOLDS NUMBER IN PIPE FLOW

Reynolds number is the ratio of inertia force to viscous force. The inertia force is proportional to the mass flow and velocity *i.e.*, ($\rho u.u$). The viscous force is proportional to $\mu(du/dy)$ or $\mu\ u/D$, dividing

$$\frac{\text{inertia force}}{\text{viscous force}} = \frac{\rho u u D}{\mu u} = \frac{\rho u D}{\mu} = \frac{uD}{v}$$

Viscous force tends to keep the layers moving smoothly one over the other. Inertia forces tend to move the particles away from the layer. When viscous force are sufficiently high so that any disturbance is smoothed down, laminar flow prevails in pipes. When velocity increases, inertia forces increase and particles are pushed upwards out of the smoother path. As long as Reynolds number is below 2,300, laminar flow prevails in pipes. The friction factor in flow is also found to be a function of Reynolds number (in laminar flow, $f = 64/\text{Re}$).

MODULE 4

BOUNDARY LAYER THEORY

7.0 PARAMETERS INVOLVED IN THE STUDY OF FLOW THROUGH CLOSED CONDUITS

In the previous chapter, the energy level changes along the flow was discussed. The losses due to wall friction in flows was not discussed. In this chapter the determination of drop in pressure in pipe flow systems due to friction is attempted.

Fluids are conveyed (transported) through closed conduits in numerous industrial processes. It is found necessary to design the pipe system to carry a specified quantity of fluid between specified locations with minimum pressure loss. It is also necessary to consider the initial cost of the piping system.

The flow may be laminar with fluid flowing in an orderly way, with layers not mixing macroscopically. The momentum transfer and consequent shear induced is at the molecular level by pure diffusion. Such flow is encountered with very viscous fluids. Blood flow through the arteries and veins is generally laminar. Laminar condition prevails upto a certain velocity in fluids flowing in pipes.

The flow turns turbulent under certain conditions with macroscopic mixing of fluid layers in the flow. At any location the velocity varies about a mean value. Air flow and water flow in pipes are generally turbulent.

The flow is controlled by (*i*) pressure gradient (*ii*) the pipe diameter or hydraulic mean diameter (*iii*) the fluid properties like viscosity and density and (*iv*) the pipe roughness. The velocity distribution in the flow and the state of the flow namely laminar or turbulent also influence the design. Pressure drop for a given flow rate through a duct for a specified fluid is the main quantity to be calculated. The inverse-namely the quantity flow for a specified pressure drop is to be also worked out on occasions.

The basic laws involved in the study of incompressible flow are (*i*) Law of conservation of mass and (*ii*) Newton's laws of motion.

Besides these laws, modified Bernoulli equation is applicable in these flows.

7.1 BOUNDARY LAYER CONCEPT IN THE STUDY OF FLUID FLOW

When fluids flow over surfaces, the molecules near the surface are brought to rest due to the viscosity of the fluid. The adjacent layers also slow down, but to a lower and lower extent. This slowing down is found limited to a thin layer near the surface. The fluid beyond this layer is not affected by the presence of the surface. The fluid layer near the surface in which there is a general slowing down is defined as boundary layer. The velocity of flow in this layer increases from zero at the surface to free stream velocity at the edge of the boundary layer. The development of the boundary layer in flow over a flat plate and the velocity distribution in the layer are shown in Fig. 7.1.1.

Pressure drop in fluid flow is to overcome the viscous shear force which depends on the velocity gradient at the surface. Velocity gradient exists only in the boundary layer. The study thus involves mainly the study of the boundary layer. The boundary conditions are (*i*) at the wall surface, (zero thickness) the velocity is zero. (*ii*) at full thickness the velocity equals the free stream velocity (*iii*) The velocity gradient is zero at the full thickness. Use of the concept is that the main analysis can be limited to this layer.

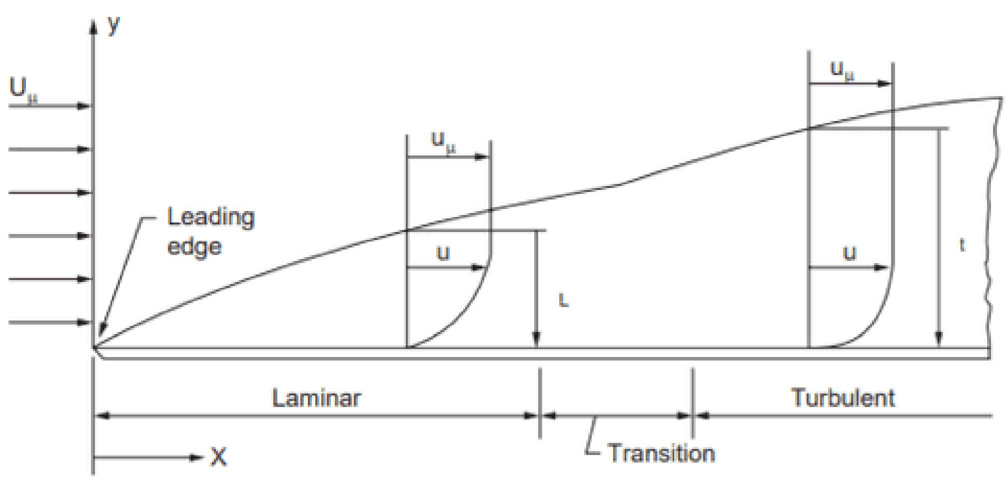


Figure 7.1.1 Boundary Layer Development (flat-plate)

7.2 BOUNDARY LAYER DEVELOPMENT OVER A FLAT PLATE

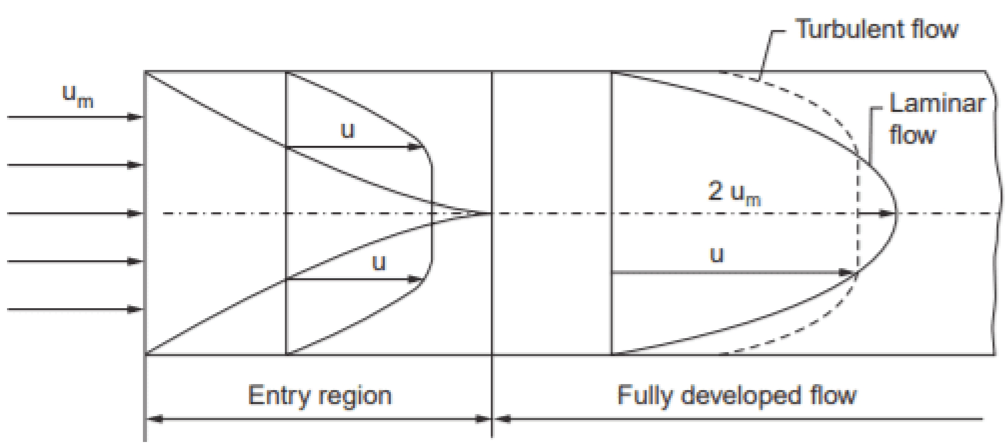
The situation when a uniform flow meets with a plane surface parallel to the flow is shown in Fig. 7.1.1. At the plane of entry (leading edge) the velocity is uniform and equals free stream velocity. Beyond this point, the fluid near the surface comes to rest and adjacent layers are retarded to a larger and larger depth as the flow proceeds.

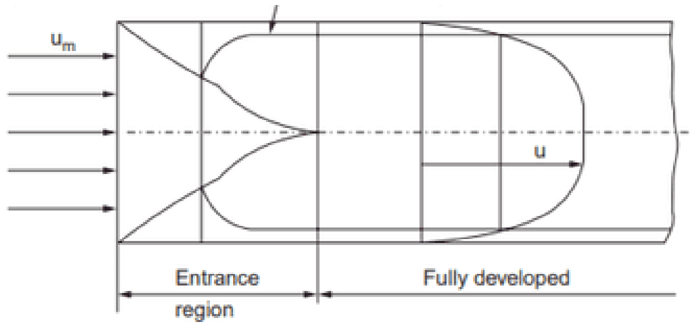
The thickness of the boundary layer increases due to the continuous retardation of flow. The flow initially is laminar. There is no intermingling of layers. Momentum transfer is at the molecular level, mainly by diffusion. The viscous forces predominate over inertia forces. Small disturbances are damped out. Beyond a certain distance, the flow in the boundary layer becomes

turbulent with macroscopic mixing of layers. Inertia forces become predominant. This change occurs at a value of Reynolds number (given $Re = ux/v$, where v is the kinematic viscosity) of about 5×10^5 in the case of flow over flat plates. Reynolds number is the ratio of inertia and viscous forces. In the turbulent region momentum transfer and consequently the shear forces increase at a more rapid rate.

7.3 DEVELOPMENT OF BOUNDARY LAYER IN CLOSED CONDUITS (PIPES)

In this case the boundary layer develops all over the circumference. The initial development of the boundary layer is similar to that over the flat plate. At some distance from the entrance, the boundary layers merge and further changes in velocity distribution becomes impossible. The velocity profile beyond this point remains unchanged. The distance upto this point is known as entry length. It is about $0.04\ Re \times D$. The flow beyond is said to be fully developed. The velocity profiles in the entry region and fully developed region are shown in Fig. 7.3.1*a*. The laminar or turbulent nature of the flow was first investigated by Osborn Reynolds in honour of





(b) Turbulent flow

Figure 7.3.1 Boundary layer development (pipe flow)

whom the dimensionless ratio of inertia to viscous forces is named. The flow was observed to be laminar till a Reynolds number value of about 2300. The Reynolds number is calculated on the basis of diameter (ud/v). In pipe flow it is not a function of length. As long as the diameter

is constant, the Reynolds number depends on the velocity for a given flow. Hence the value of velocity determines the nature of flow in pipes for a given fluid. The value of the flow Reynolds number is decided by the diameter and the velocity and hence it is decided at the entry itself. The development of boundary layer in the turbulent range is shown in Fig. 7.3.1*b*. In this case, there is a very short length in which the flow is laminar. This length, x, can be calculated using the relation $ux/v = 2000$. After this length the flow in the boundary layer turns turbulent. A very thin laminar sublayer near the wall in which the velocity gradient is linear is present all through. After some length the boundary layers merge and the flow becomes fully developed. The entry length in turbulent flow is about 10 to 60 times the diameter.

The velocity profile in the fully developed flow remains constant and is generally more flat compared to laminar flow in which it is parabolic.

7.4 FEATURES OF LAMINAR AND TURBULENT FLOWS

In laminar region the flow is smooth and regular. The fluid layers do not mix macroscopically (more than a molecule at a time). If a dye is injected into the flow, the dye will travel along a straight line. Laminar flow will be maintained till the value of Reynolds number is less than of the critical value (2300 in conduits and 5×10^5 in flow over plates). In this region the viscous forces are able to damp out any disturbance.

The friction factor, f for pipe flow defined as $4\tau_s/(\rho u^2/2g_o)$ is obtainable as $f = 64/\text{Re}$ where τ_s is the wall shear stress, u is the average velocity and Re is the Reynolds number. In the case of flow through pipes, the average velocity is used to calculate Reynolds number. The dye path is shown in Fig. 7.4.1.

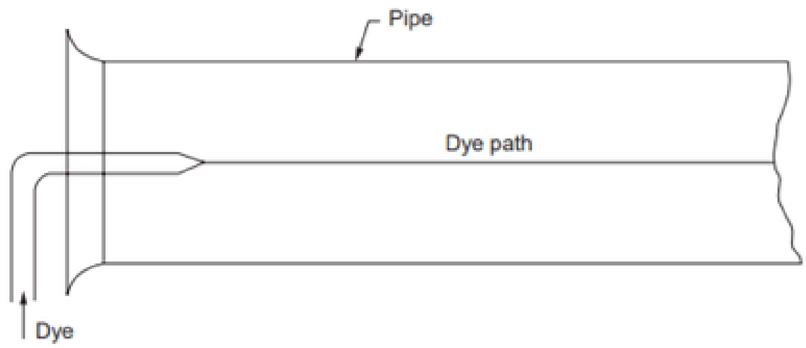


Figure 7.4.1 Reynolds Experiment

In turbulent flow there is considerable mixing between layers. A dye injected into the flow will quickly mix with the fluid. Most of the air and water flow in conduits will be turbulent. Turbulence leads to higher frictional losses leading to higher pressure drop. The friction factor is given by the following empirical relations.

$$f = 0.316/\text{Re}^{0.25} \quad \text{for Re} < 2 \times 10^4 \qquad (7.4.1)$$

$$f = 0.186/\text{Re}^{0.2} \quad \text{for Re} > 2 \times 10^4 \qquad (7.4.2)$$

These expressions apply for smooth pipes. In rough pipes, the flow may turn turbulent below the critical Reynolds number itself. The friction factor in rough pipe of diameter D, with a roughness height of ε, is given by

$$f = 1.325/[\ln \{(\varepsilon/3.7D) + 5.74/\text{Re}^{0.9}\}]^2 \qquad (7.4.3)$$

10.1 BOUNDARY LAYER THICKNESS

In the solution of the basic equations describing the flow namely continuity and momentum equations of the boundary layer, one boundary is provided by the solid surface. The need for the other boundary is met by edge of the boundary layer determined by the thickness. The determination of the velocity variation along the layer enables the determination

of velocity gradient. This is made possible by these two boundary conditions. Once the velocity gradient at the surface is determined, the shear stress can be determined using the equation

$$\tau = \mu \frac{du}{dy} \tag{10.1.1}$$

This leads to the determination of resistance due to the flow.

10.1.1 Flow Over Flat Plate

The simplest situation that can be analyzed is the flow over a flat plate placed parallel to uniform flow velocity in a large flow field. The layer near the surface is retarded to rest or zero. velocity. The next layer is retarded to a lower extent. This proceeds farther till the velocity equals the free stream velocity. As the distance for this condition is difficult to determine, **the boundary layer thickness is arbitrarily defined as the distance from the surface where the velocity is 0.99 times the free stream velocity.**

There are two approaches for the analysis of the problem.

1. Exact method : Solution of the differential equations describing the flow using the boundary conditions. It is found that this method can be easily applied only to simple geometries.

2. Approximate method : Formulation of integral equations describing the flow and solving them using an assumed velocity variation satisfying the boundary conditions. This method is more versatile and results in easier solution of problems. The difference between the results obtained by the exact method and by the integral method is found to be within acceptable limits.

At present several computer softwares are available to solve almost any type of boundary, and the learner should become familiar with such softwares if he is to be current.

10.1.2 Continuity Equation

The flow of fluid over a flat plate in a large flow field is shown in Fig. 10.1.1. The flow over the top surface alone is shown in the figure.

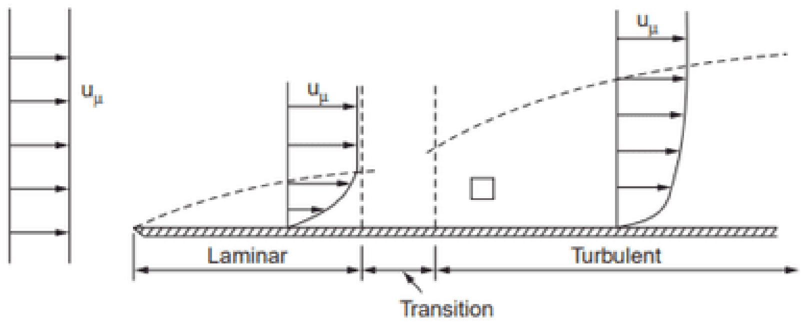


Figure 10.1.1 Formation of boundary layer over flat plate

The velocity is uniform in the flow field having a value of u_∞. Boundary layer begins to form from the leading edge and increases in thickness as the flow proceeds. This is because the viscosity effect is felt at layers more and more removed from the surface. At the earlier stages the flow is regular and layers keep their position and there is no macroscopic mixing between layers. **Momentum transfer resulting in the retarding force is by molecular diffusion between layers. This type of flow is called laminar flow** and analysis of such flow is somewhat simpler. **Viscous effects prevail over inertial effects in such a layer**. Viscous forces maintain orderly flow. **As flow proceeds farther, inertial effects begin to prevail over viscous forces resulting in macroscopic mixing between layers. This type of flow is called turbulent flow**. Higher rates of momentum transfer takes place in such a flow. For the formulation of the differential equations an element of size $dx \times dy \times 1$ is considered.

An enlarged sectional view of the element is shown in Fig. 10.1.2.

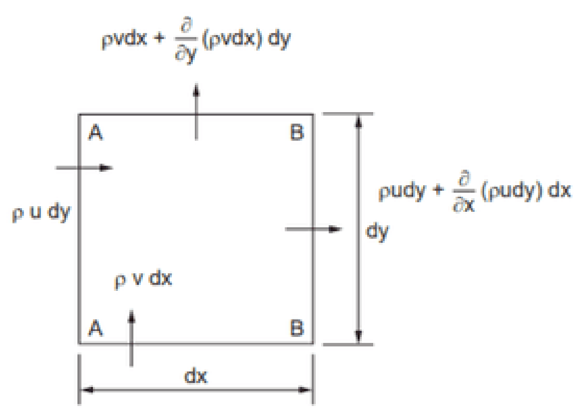


Figure 10.1.2 Enlarged view of element in the boundary layer

The assumptions are (*i*) flow is incompressible or density remains constant, (*ii*) flow is steady, (*iii*) there is no pressure gradient in the boundary layer.

Continuity equation is obtained using the principle of conservation of mass. Under steady flow conditions the net mass flow across the element should be zero. Under unsteady conditions, the net mass flow should equal the change of mass in the elemental volume considered. The values of velocities are indicated in the figure. The density of the fluid is ρ. Unit time and unit *Z* distance are assumed. **Time is not indicated in the equations.**

Flow in across face *AA*, $\rho u dy \times 1 = \rho u dy$

Flow out across face *BB*, $\rho u dy + \frac{\partial}{\partial x}(\rho u dy)\, dx$

Net flow in the *x* direction = $\frac{\partial (\rho u)}{\partial x}\, dxdy$

Similarly the net flow in the *y* direction is given by $\frac{\partial (\rho v)}{\partial x}\, dxdy$

Under steady conditions the sum is zero. Also for incompressible flow density is constant. Hence

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (10.1.2)$$

This is known as continuity equation for steady incompressible flow. If *u* decreases, $\frac{\partial u}{\partial x}$ is – ve and so $\frac{\partial v}{\partial y}$ should be positive. The algebraic sum of *x* and *y* directional flows is zero.

10.1.3 Momentum Equation

The equation is based on Newton's second law of motion. The net force on the surface of the element should equal the rate of change of momentum of the fluid flowing through the element. Here *x* directional forces are considered with reference to the element shown in Fig. 10.1.3. The flows are indicated on the figure unit time and unit *Z* distance are assumed. The density of the fluid is ρ

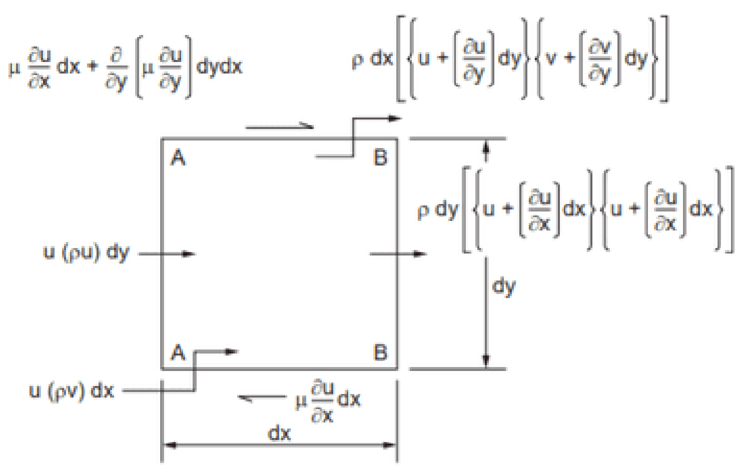


Figure 10.1.3 Momentum analysis

Consider the momentum flow in the x direction :

Across AA momentum flow = $u\ (\rho u)\ dy$

Across BB momentum flow = $u\ (\rho u)\ dy + \frac{\partial}{\partial x}\{u(\rho u)\ dy\}\ dx$

Taking the difference, the net flow is (as ρ is constant) (u^2 is written as $u \times u$)

$$\frac{\partial}{\partial x}[u(\rho u)dy]dx = \rho\ dxdy\left[u\frac{\partial u}{\partial x} + u\frac{\partial u}{\partial x}\right]$$

Considering the flow in the y direction, the net x directional momentum flow is

$$\frac{\partial}{\partial y}[u(\rho v)dy]dx = \rho\ dxdy\left[u\frac{\partial u}{\partial y} + v\frac{\partial u}{\partial y}\right]$$

Summing up, the net momentum flow is

$$\rho\ dxdy\left[u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \left\{u\left[\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right]\right\}\right]$$

From continuity equation, the second set in the above equation is zero. Hence net x directional momentum flow is

$$\left[u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right]\rho\ dxdy$$

It was assumed that no body forces or pressure forces are present. Only surface forces due to viscosity is considered.

At the bottom surface shear $= dx\ \mu\frac{\partial u}{\partial y}$

At the top surface shear $= dx\ \mu\frac{\partial u}{\partial y} + \frac{\partial}{\partial y}\left[\mu\frac{\partial u}{\partial y}dx\right]dy$

The net shear on the element is $\mu\frac{\partial^2 u}{\partial y^2}dxdy$, noting $v = \mu/\rho$

Equating, and simplifying, $u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = v\frac{\partial^2 u}{\partial y^2}$ (10.1.3)

This is known as momentum equation for the boundary layer. v is also called as momentum diffusivity. In case of pressure gradient along the flow $-\frac{1}{\rho}\frac{\partial P}{\partial x}$ has to added on the RHS.

10.2 TURBULENT FLOW

As flow preceeds farther along the flat plate, inertia forces begin to prevail and viscous forces are unable to keep the flow in an orderly way. **Reynolds number is the ratio of inertia force to viscous force. As inertia force increases Reynolds number increases and the flow becomes turbulent**. Generally the limiting Reynolds number for laminar flow over flat plate is taken as 5×10^5 (for internal flow the critical Reynolds number is 2000).

Turbulent flow is characterized by the variation of velocity with time at any location. The velocity at any location at any time, can be represented by

$$u = \bar{u} + u'$$

where u is the instantaneous velocity, $\bar{u}$ is the average over time and u' is the fluctuating component. The flow is steady as u' is constant at any location. An accurate velocity profile known as universal velocity profile, having different distributions at different heights is available. However it is too cmplex for use with integral method at our level of discussion.

One seventh power law has been adopted as a suitable velocity distribution for turbulent flow.

$$\frac{u}{u_\infty} = \left(\frac{y}{\delta}\right)^{1/7} \quad (10.2.1)$$

Substituting in the integral momentum equation 10.1.2, boundary layer thickness is obtained as

$$\delta = 0.382\, x/\text{Re}_x^{\,0.2} \quad (10.2.2)$$

For combined laminar and turbulent flow,

$$\delta_L = (0.381x/\text{Re}_L^{\,0.2}) - (10256/\text{Re}_L) \quad (10.2.2\ a)$$

The friction coefficient is obtianed as

$$C_{fx} = 0.0594/\text{Re}_x^{\,0.2} \quad (10.2.3)$$

for combined laminar turbulent flow

$$C_{fL} = 0.074\text{Re}^{-0.2} - 1742\text{Re}_L^{\,-1} \quad (10.2.4)$$

Displacement thickness is obtained as $\delta_d = \delta/8$

10.3 FLOW SEPARATION IN BOUNDARY LAYERS

Boundary layer is formed in the case of flow of real fluids. Viscous forces exist in such flows. The shear stress at the wall is given by

$$\tau_w = \mu \left.\frac{du}{dy}\right|_{y=0}$$

, The wall shear cannot be zero. Hence at $y = 0$, $\frac{du}{dy}$ cannot be zero. This means that the velocity gradient at the wall cannot be zero.

Separation of flow is said to occur when the direction of the flow velocity near the surface is opposed to the direction of the free stream velocity, which means $(du/dy) \le 0$. Such a situation does not arise when there is no pressure gradient opposed to the flow direction, *ie.*, the pressure downstream of flow is higher compared to the pressure upstream. An example is subsonic diffuser. In the direction of flow the pressure increases. The increase in area along the flow causes a pressure rise.

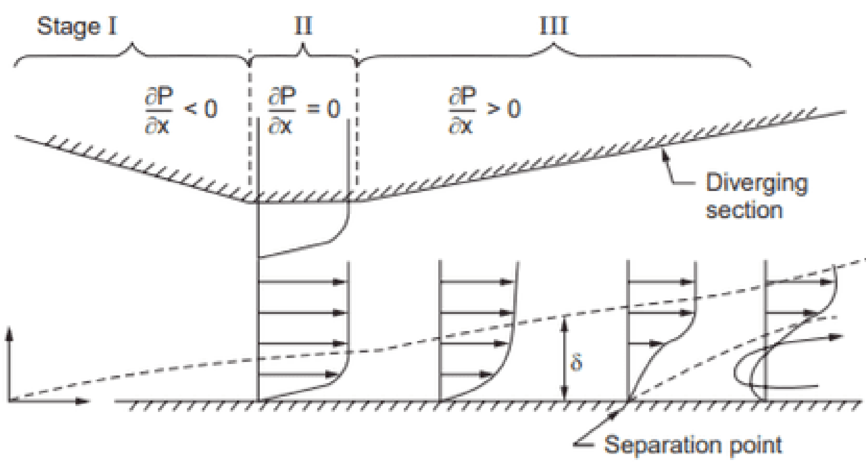


Figure 10.3.1 Flow separation

If (dp/dx) increases to the extent that it can overcome the shear near the surface, then separation will occur. Such a pressure gradient is called adverse pressure gradient. In the case of incompressible flow in a nozzle a favorable pressure gradient exists. Separation will not occur in such flows. In the case of diverging section of a diffuser, separation can occur if the rate of area increase is large. This is shown in Fig. 10.3.1. In turbulent flow, the momentum near the surface is high compared to laminar flow. Hence turbulent layer is able to resist separation better than laminar layer.

In the case of flow over spheres, cylinders, blunt bodies, airfoils etc., there is a change in flow area due to the obstruction and hence an adverse pressure gradient may be produced. Simple analytical solutions are not available to determine exactly at what conditions separation will occur. Experimental results are used to predict such conditions.

Flow Measurements

11.2 VELOCITY MEASUREMENTS

The measurement of velocity at a point or a number of points throughout a section in a flow stream is often needed to establish the velocity profile. Measurement of velocity at a point is almost impossible, since any sensing device has a finite dimension. However, if the area of flow occupied by the sensing device is relatively small compared to the total area of flow stream, then it may be considered that the velocity measured is the velocity at a point. It is essential that the presence of the sensing device in the flow stream does not afffect the flow being measured. **Velocity is usually measured indirectly by measuring the difference between the stagnation and static pressures** (pitot tube) or by the rotational speed of wheels (vane anemometer) or by the temperature drop on a thin cylindrical wire in cross flow (hot wire anemometer) and also by optical systems. Velocity is also measured directly, in some instances, by determining the distance travelled by a group of fluid particles during a measured time interval.

11.2.1 Pitot Tube

If a small bore hollow tube bent at 90° is placed in a flow stream with its end facing upstream, fluid will rise in the vertical side of the tube as shown in Fig 11.2.1 (*a*). This method is used as pick-up in velocity measurment.

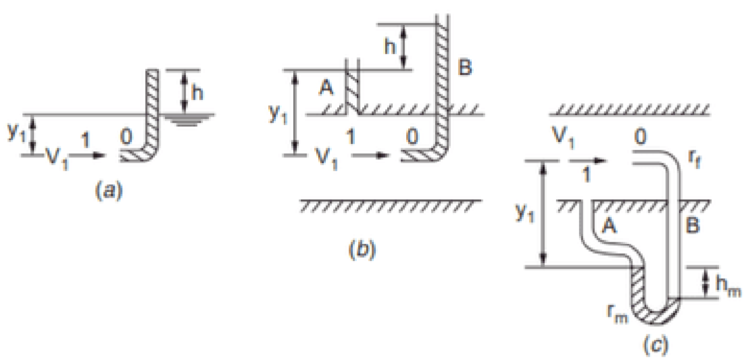


Figure 11.2.1 Pitot tube arrangements

If Bernoulli equation is applied between a point, 1 upstream at the submerged end of the tube and a point, 0 at the other end of the tube, then leaving out P_{atm} on both sides

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} = \frac{P_0}{\rho g} \tag{11.2.1}$$

Since stagnation condition exists within the tube

$P_0 = \rho g(y + h)$, at point 1, the static pressure is $P_1 = \rho g y_1$

Substituting and rearranging equation (11.1)

$$\frac{V_1^2}{2g} = \frac{(P_0 - P_1)}{\rho g} = \frac{\rho g[(y_1 + h) - y_1]}{\rho g} = h\ ,$$

$$\therefore \quad V_1 = \sqrt{2gh} \tag{11.2.2}$$

Note that **h is the head expressed as the column of flowing fluid.**

For velocity measurement in ducts a different arrangement of pick ups is necessary. A typical method is illustrated in Fig. 11.2.1 (*b*). A tapping perpendicular to the flow gives the static pressure. The tube connection at this point is called static tube/probe. The pitot probe held facing upstream measures the total pressure.

The static tube *A* and pitot tube *B* are connected to a *U* tube manometer as shown in Fig. 11.2.1 (*c*) for measurement of velocity in a pipe. Equating the pressure at the left and right side limbs of the manometer,

$$P_1 + \rho g y_1 + \rho_m g\, h_m = P_0 + \rho g y_1$$

$$h_m g(\rho_m - \rho) = P_0 - P_1 \tag{11.2.3}$$

where ρ and ρ_m are the densities of flowing and manometric fluids. Substituting for $(P_0 - P_1)$ from equation 11.2.1,

$$h_m g\,(\rho_m - \rho) = \frac{\rho V_1^2}{2}$$

$\therefore$ The velocity of fluid near the tip of the pitot at section 1 is

$$V_1 = \sqrt{2gh_m(\rho_m - \rho)/\rho} = \sqrt{2gh_m((\rho_m/\rho) - 1)} \tag{11.2.4}$$

ρ_m/ρ is to be replaced by s_m/s in terms of specific gravities.

11.3 VOLUME FLOW RATE MEASUREMENT

Volume flow rate in pipes can be measured either using direct measuring devices such as watermeter or rotameters (float meters) or using a constriction or elbow meters which produce a measurable pressure difference that can be used to determine the flow rate.

Flow meters (watermeter or rotameter) may be calibrated either by the manufacturer or by the user before installation. The same fluid and same range of flows as in the actual installation should be used for the calibration.

In the case of constriction meters Bernoulli equation and continuity equation are applied between the upstream and downstream sections of the constriction to obtain an expression for the flow rate.

11.3.1 Rotameter (Float Meter)

The rotameter is a device whose indication is essentially linear with flow rate. This device is also called as variable area meter or float meter. In this device a flot moves freely inside a tapered tube as shown in Fig. 11.3.1

The flow takes place upward through the tube. The following forces act on the float (*i*) downward gravity force (*ii*) upward buoyant force (*iii*) pressure and (*iv*) viscous drag force.

For a given flow rate, the float assumes a position inside the tube where the forces acting on it are in equilibrium. Through careful design, the effects of changes in viscosity or density may be minimized, leaving only the pressure forces as the main variable. Pressure force depends on flow rate and area available for flow. Hence the position of the float indicates the flow rate.

A major limitation in using rotameters is that these have to be installed in vertical position only. Also it cannot be used with liquids containing large number of solid particles and at high pressure conditions. It is also expensive. The advantage is that its capacity to measure the flow rate can be easily changed by changing the float or the tube.

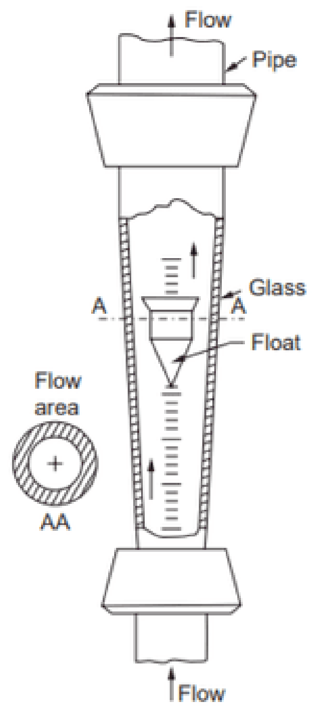


Figure 11.3.1 Rotameter

11.3.3 Venturi, Nozzle and Orifice Meters

Venturi, Nozzle and Orifice meters are the three obstruction type meters commonly used for the measurement of flow through pipes. In each case the meter acts as an obstacle placed in the path of the flowing fluid causing local changes in pressure and velocity as shown in Fig. 11.3.3.

Applying Bernoulli and continuity equations between sections 11.1 and 11.2

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + Z_2 \text{ and } V_1A_1 = V_2A_2$$

Solving these equations,

$$V_2 = \frac{1}{\sqrt{1-(A_2/A_1)^2}} \sqrt{2g[(p_1/\rho g + Z_1) - (P_2/\rho g + Z_2)]}$$

By connecting a manometer to the tappings at sections 11.1 and 11.2 the difference in pressure levels $\left[\left(\frac{P_1}{\rho g} + Z_1\right) - \left(\frac{P_2}{\rho g} + Z_2\right)\right]$ can be measured by the manometer reading, Δh.

$$\therefore \quad V_2 = \frac{1}{\sqrt{1-(A_2/A_1)^2}} \sqrt{2g\Delta h}$$

$$\therefore \quad \text{Flow rate} \quad Q = \frac{A_2}{\sqrt{1-(A_2/A_1)^2}} \sqrt{2g\Delta h}$$

Refer equations 11.2.3 and 11.2.2.

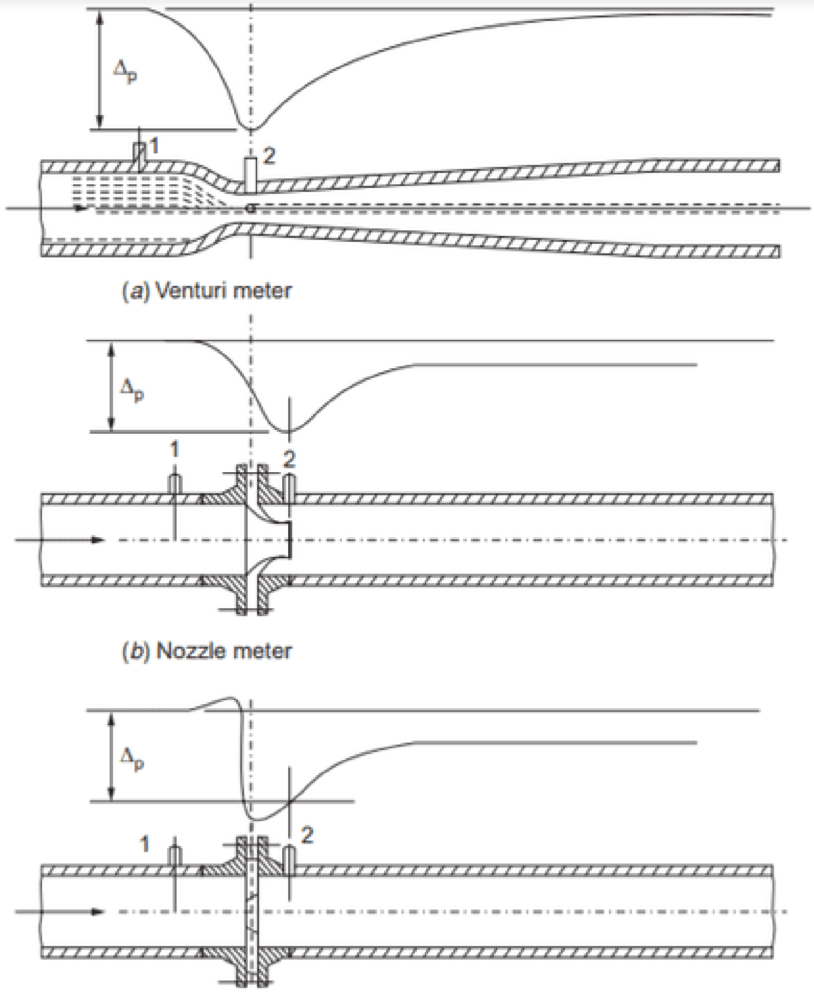


Figure 11.3.3 Pressure variation in obstruction type meters

This equation needs a modifying coefficient as viscous effects and boundary roughness as well as the velocity of approach factor that depend on the diameter ratio have been neglected. The coefficient is defined by,

$$\therefore \qquad Q_{\text{actual}} = Q_{\text{theoretical}} \times C_d$$

where C_d is the coefficient of discharge. Venturimeter is a highly accurate device with discharge coefficient falling within a narrow range depending on the finish of the entrance cone. C_d for venturi meters is in the range 0.95 to 0.98.

The approach curve in the nozzle flow meter must be proportioned to prevent separation between the flow and the well. A parallel section is used to ensure that flow fills the throat. C_d for flow nozzle is in the range 0.7 to 0.9 depending on diameter ratio and Reynolds number to some extent.

Orificemeter is the simple and cheap device compared to the other two. But sudden area of contraction in this device leads to higher pressure loss. The range for coefficient of discharge is 0.6 to 0.65. The value depends on the diameter ratio. Higher the value D_2/D_1 lower the value of the coefficient. In both the above cases for $Re > 10^5$ the effect of Re on C_d is marginal.

11.4 FLOW MEASUREMENT USING ORIFICES, NOTCHES AND WEIRS

Flow out of open tanks are measured using orifices. Flow out of open channels is measured using weirs. Flow from open channels and tanks is due to gravity and the change in velocity produced is due to the change in head.

11.4.1 Discharge Measurement Using Orifices

Fig. 11.4.1 shows an orifice in an open tank through which the flow takes place. Applying Bernoulli equation between points 1 and 2

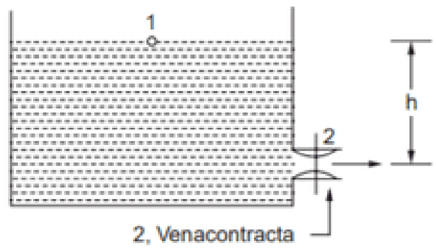


Figure 11.4.1 Orifice meter

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + Z_2$$

The velocity a point 1 is zero and the pressures at 1 and 2 are both atmospheric.

$$\therefore \quad Z_1 - Z_2 = V_2^2/2g$$

$$V_2 = \sqrt{2g(Z_1 - Z_2)} = \sqrt{2gh}$$

The theoretical flow rate is given by $Q_t = A_2 \sqrt{2gh}$

where A_2 is the area of cross-section at section 11.2. The actual flow rate is given by

$$Q_{\text{actual}} = C_d A_0 \sqrt{2gh}$$

where A_0 is the area of orifice and C_d is the coefficient of discharge.

$$C_d = Q_{\text{actual}}/Q_{\text{theoretical}}$$

The values of C_d depends upon the contraction of the jet fromt the orifice to section 2 and on nonideal flow effects such as head losses which depend upon the roughness of the inside surface of the tank near the orifice and the flow rate. Typical value for C_d is 0.62.

Cofficient of velocity (C_v). There is alsways some loss of energy due to viscous effects in real fluid flows. Due to these effects, the actual flow velocity through the orifice will always be less than the theoretical possible velocity. The velocity coefficients C_v is defined as follows.

$$C_v = \frac{\text{Actual velocity of jet at venacontracta}}{\text{Theoritical velocity}} = \frac{V}{\sqrt{2gh}}$$

The value of C_v varies from 0.95 to 0.99 for different orifices depending on their shape and size.

Coefficient of contraction C_c. As water leaves an open tank through an orifice, the stream lines converge and the area just outside the orifice is lower compared to the area of the orifice. This section is called as vena contracta. Area of jet at the vena contracta is less than the area of the orifice itself due to convergence of stream lines. The coefficient of contraction C_c is defined as follows

$$C_c = \frac{\text{Area of the jet at vena contracta}}{\text{Area of orifice}} = \frac{a_c}{a}$$

The value of coefficient of contraction varies from 0.61 to 0.69 depending on the shape and size of the orifice.

Coefficient of discharge (C_d) Coefficient of discharge is defined as

$$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} = \frac{\text{Actual area}}{\text{Theoretical area}} \times \frac{\text{Actual velocity}}{\text{Theoretical velocity}} = C_c \times C_v$$

Average value of C_d for orifices is 0.62.

11.4.2 Flow Measurements in Open Channels

Rectangular and triangular weirs are used to measure the flow in an open channel. A rectangular notch is shown in Fig. 11.4.2. A weir extends to the full width of the channel while a notch occupies a smaller width.

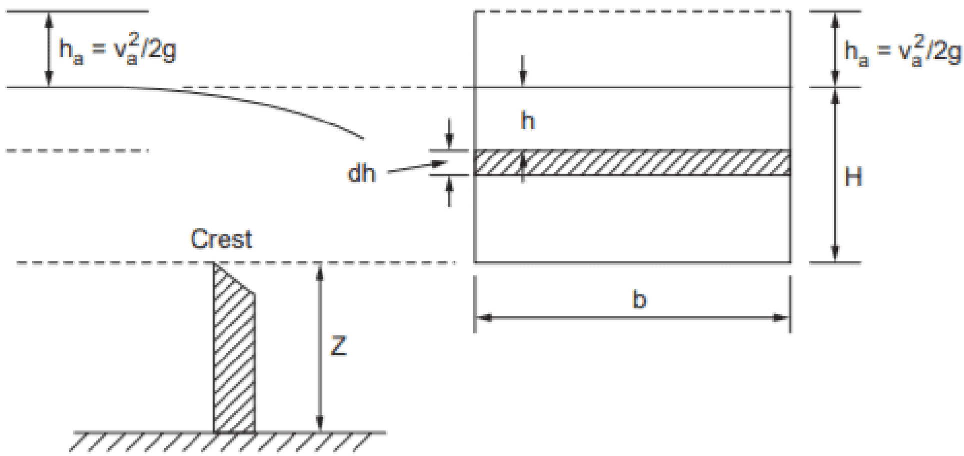


Figure 11.4.2 Rectangular weir

Figure 11.4.2 Rectangular weir

Rectangular weir. Bernoulli equation is applied between upstream and drown stream of the weir. Consider a rectangular strip as shown in figure, with height dh and width B at a height h above the strip.

Flow rate through the elemental strip = $dq = C_d\,(B\,dh)\,\sqrt{2gh}$

Integrating between the weir tip and the water level

Total discharge $\quad Q = \int_0^H C_d\,B dh\,\sqrt{2gh} = C_d\,B\,\sqrt{2g}\,\int_0^H h^{1/2}\,dh$

$$Q = C_d B\sqrt{2g}\left[\frac{h^{3/2}}{3/2}\right]_0^H \qquad Q = \frac{2}{3}C_d B\sqrt{2g}\,H^{3/2} \qquad (11.4.1)$$

The value of C_d depends the approach velocity which in turn depends on the ratio of head H and crest height z. The value of C_d is given by

$$C_d = 0.611 + 0.075\frac{H}{z} \qquad (11.4.2)$$

A trapezoidal weir with side slope of 1 horizontal to 4 vertical is used to compensate for flow reduction due to end contraction at the corners. It is called **Cipolletti weir.** The flow equation is the same with B as bottom width. The value of C_d will however be different.

Discharge over a triangular notch. A triangular notch is called *V* notch as shown in Fig. 11.4.3.

Consider an elemental strip *dh*, the discharge through the elemental strip *dh* is

$$dq = C_d \left(2\,(H-h) \tan\frac{\theta}{2} dh \right) \sqrt{2gh}$$

Total discharge $$Q = \int_0^H C_d \left(2(H-h) \tan\frac{\theta}{2} dh \right) \sqrt{2gh}$$

$$= 2\, C_d \sqrt{2g} \tan\frac{\theta}{2} \int_0^H (H-h)\, h^{1/2} dh$$

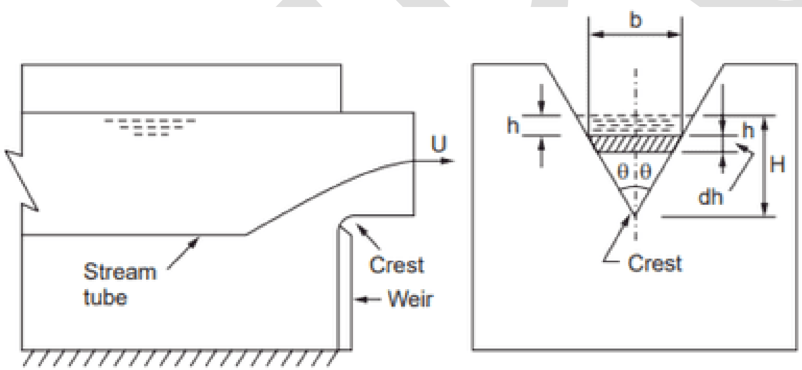


Figure 11.4.3 Triangular notch

$$Q = 2C_d \sqrt{2g} \tan\frac{\theta}{2} \left[\frac{Hh^{3/2}}{3/2} - \frac{h^{5/2}}{5/2} \right] = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2} \qquad (11.4.3)$$

MODULE 6

Hydraulic turbines

14.0 INTRODUCTION

Most of the electrical generators are powered by turbines. Turbines are the primemovers of civilisation. Steam and Gas turbines share in the electrical power generation is about 75%. About 20% of power is generated by hydraulic turbines and hence thier importance. Rest of 5% only is by other means of generation. Hydraulic power depends on renewable source and hence is ever lasting. It is also non polluting in terms of non generation of carbon dioxide.

14.1 HYDRAULIC POWER PLANT

The main components of hydraulic power plant are (*i*) The storage system. (*ii*) Conveying system (*iii*) Hydraulic turbine with control system and (*iv*) Electrical generator

The storage system consists of a reservoir with a dam structure and the water flow control in terms of sluices and gates etc. The reservoir may be at a high level in the case of availability of such a location. In such cases the potential energy in the water will be large but the quantity of water available will be small. The conveying system may consist of tunnels, channels and steel pipes called penstocks. Tunnels and channels are used for surface conveyance. Penstocks are pressure pipes conveying the water from a higher level to a lower level under pressure. The penstock pipes end at the flow control system and are connected to nozzles at the end. The nozzles convert the potential energy to kinetic energy in free water jets. These jets by dynamic action turn the turbine wheels. In some cases the nozzles may be replaced by guide vanes which partially convert potential energy to kinetic energy and then direct the stream to the turbine wheel, where the remaining expansion takes place, causing a reaction on the turbine runner. Dams in river beds provide larger quantities of water but with a lower potential energy.

The reader is referred to books on power plants for details of the components and types of plants and their relative merits. In this chapter we shall concentrate on the details and operation of hydraulic turbines.

14.2 CLASSIFICATION OF TURBINES

The main classification depends upon the type of action of the water on the turbine. These are

(*i*) **Impulse turbine** (*ii*) **Reaction Turbine**. In the case of impulse turbine all the potential energy is converted to kinetic energy in the nozzles. The imulse provided by the jets is used to turn the turbine wheel. The pressure inside the turbine is atmospheric. This type is found suitable when the available potential energy is high and the flow available is comparatively low. Some people call this type as tangential flow units. Later discussion will show under what conditions this type is chosen for operation.

(*ii*) In reaction turbines the available potential energy is progressively converted in the turbines rotors and the reaction of the accelerating water causes the turning of the wheel. These are again divided into radial flow, mixed flow and axial flow machines. Radial flow machines are found suitable for moderate levels of potential energy and medium quantities of flow. The axial machines are suitable for low levels of potential energy and large flow rates. The potential energy available is generally denoted as "head available". With this terminology plants are designated as "high head", "medium head" and "low head" plants.

14.4 TURBINE EFFICIENCIES

The head available for hydroelectric plant depends on the site conditions. Gross head is defined as the difference in level between the reservoir water level (called head race) and the level of water in the stream into which the water is let out (called tail race), both levels to be observed at the same time. During the conveyance of water there are losses involved. The difference between the gross head and head loss is called the net head or effective head. It can be measured

by the difference in pressure between the turbine entry and tailrace level. The following efficiencies are generally used.

1. Hydraulic efficiency : It is defined as the ratio of the power produced by the turbine runner and the power supplied by the water at the turbine inlet.

$$\eta_H = \frac{\text{Power produced by the runner}}{\rho\, Q\, g\, H} \qquad (14.4.1)$$

where Q is the volume flow rate and H is the net or effective head. Power produced by the runner is calculated by the Euler turbine equation $P = Q\rho\,[u_1 V_{u1} - u_2 V_{u2}]$. This reflects the runner design effectiveness.

2. Volumetric efficiency : It is possible some water flows out through the clearance between the runner and casing without passing through the runner.

Volumetric efficiency is defined as the ratio between the volume of water flowing through the runner and the total volume of water supplied to the turbine. Indicating Q as the volume flow and ΔQ as the volume of water passing out without flowing through the runner.

$$\eta_v = \frac{Q - \Delta Q}{Q} \qquad (14.4.2)$$

To some extent this depends on manufacturing tolerances.

3. Mechanical efficiency : The power produced by the runner is always greater than the power available at the turbine shaft. This is due to mechanical losses at the bearings, windage losses and other frictional losses.

$$\eta_m = \frac{\text{Power available at the turbine shaft}}{\text{Power produced by the runner}} \qquad (14.4.3)$$

4. Overall efficiency : This is the ratio of power output at the shaft and power input by the water at the turbine inlet.

$$\eta_0 = \frac{\text{Power available at the turbine shaft}}{\rho\, QgH} \qquad (14.4.4)$$

Also the overall efficiency is the product of the other three efficiencies defind

$$\eta_0 = N_H\ N_m N_v \qquad (14.4.5)$$

14.6 PELTON TURBINE

This is the only type used in high head power plants. This type of turbine was developed and patented by L.A. Pelton in 1889 and all the type of turbines are called by his name to honour him.

A sectional view of a horizontal axis Pelton turbine is shown in figure 14.6.1. The main components are (1) The runner with the (vanes) buckets fixed on the periphery of the same. (2) The nozzle assembly with control spear and deflector (3) Brake nozzle and (4) The casing.

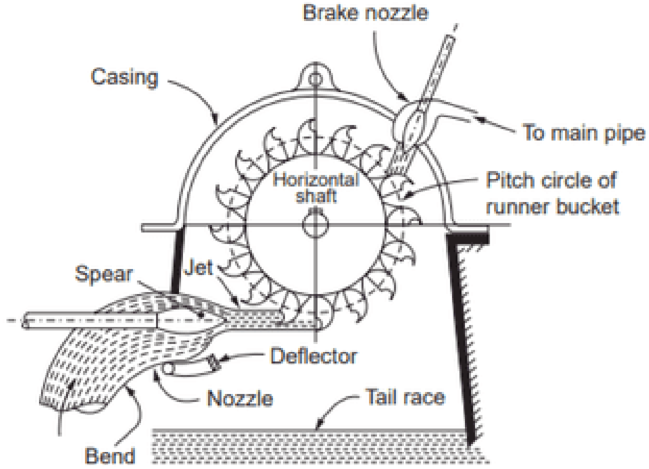


Figure 14.6.1 Pelton turbine

The rotor or runner consists of a circular disc, fixed on suitable shaft, made of cast or forged steel. Buckets are fixed on the periphery of the disc. The spacing of the buckets is decided by the runner diameter and jet diameter and is generally more than 15 in number. These buckets in small sizes may be cast integral with the runner. In larger sizes it is bolted to the runner disc.

The buckets are also made of special materials and the surfaces are well polished. A view of a bucket is shown in figure 14.6.2 with relative dimensions indicated in the figure. Originally spherical buckets were used and pelton modified the buckets to the present shape. It is formed in the shape of two half ellipsoids with a splilter connecting the two. A cut is made in the lip to facilitate all the water in the jet to usefully impinge on the buckets. This avoids interference of the incoming bucket on the jet impinging on the previous bucket. Equations are available to calculate the number of buckets on a wheel. The number of buckets, Z,

$$Z = (D/2d) + 15$$

where D is the runner diameter and d is the jet diameter.

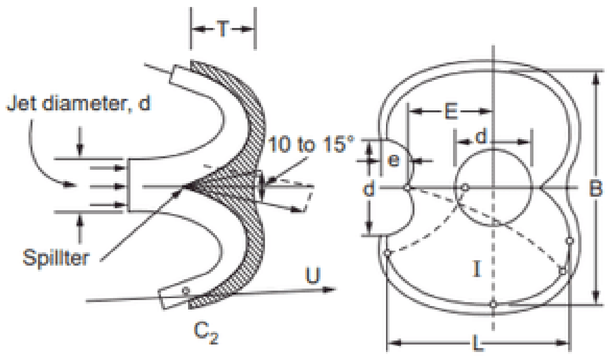


Figure 14.6.2 Pelton turbine bucket

Bucket and wheel dimensions

D/d	*B/d*	*L/d*	*T/d*	Notch width
14 – 16	2.8 – 4	2.5 – 2.8	0.95	1.1 *d* + 5 mm

The nozzle and controlling spear and deflector assembly

The head is generally constant and the jet velocity is thus constant. A fixed ratio between the jet velocity and runner peripheral velocity is to be maintained for best efficiency. The nozzle is designed to satisfy the need. But the load on the turbine will often fluctuate and some times sudden changes in load can take place due to electrical circuit tripping. The velocity of the jet should not be changed to meet the load fluctuation due to frequency requirements. The quantity of water flow only should be changed to meet the load fluctuation. A governor moves to and fro a suitably shaped spear placed inside the nozzle assembly in order to change the flow rate at the same time maintaining a compact circular jet.

When load drops suddenly, the water flow should not be stopped suddenly. Such a sudden action will cause a high pressure wave in the penstock pipes that may cause damage to the system. To avoid this a deflector as shown in figure 14.6.3 is used to suddenly play out and deflect the jet so that the jet bypasses the buckets. Meanwhile the spear will move at the safe rate and close the nozzle and stop the flow. The deflector will than move to the initial position. Even when the flow is cut off, it will take a long time for the runner to come to rest due to the high inertia. To avoid this a braking jet is used which directs a jet in the opposite direction and stops the rotation. The spear assembly with the deflector is shown in figure 14.6.3. Some other methods like auxiliary waste nozzle and tilting nozzle are also used for speed regulation. The first wastes water and the second is mechanically complex. In side **the casing** the pressure is atmospheric and hence no need to design the casing for pressure. It mainly serves the purpose of providing a cover and deflecting the water downwards. The casing is cast in two halves for case of assembly. The casing also supports the bearing and as such should be sturdy enough to take up the load.

When the condition is such that the specific speed indicates more than one jet, a vertical shaft system will be adopted. In this case the shaft is vertical and a horizontal nozzle ring with several nozzle is used. The jets in this case should not interfere with each other.

14.7 REACTION TURBINES

The functioning of reaction turbines differs from impulse turbines in two aspects.

1. In the impulse turbine the potential energy available is completely converted to kinetic energy by the nozzles before the water enters the runner. The pressure in the runner is constant at atmospheric level.

In the case of reaction turbine the potential energy is partly converted to kinetic energy in the stater guide blades. The remaining potential energy is gradually converted to kinetic energy and absorbed by the runner. The pressure inside the runner varies along the flow.

2. In the impulse turbine only a few buckets are engaged by the jet at a time.

In the reaction turbine as it is fully flowing all blades or vanes are engaged by water at all the time. The other differences are that reaction turbines are well suited for low and medium heads (300 m to below) while impulse turbines are well suited for high heads above this value.

Also due to the drop in pressure in the vane passages in the reaction turbine the relative velocity at outlet is higher compared to the value at inlet. In the case of impulse turbine there is no drop in pressure in the bucket passage and the relative velocity either decreases due to surface friction or remains constant. In the case of reaction turbine the flow area between two blades changes gradually to accomodate the change in static pressure. In the case of impulse turbine the speed ratio for best efficiency is fixed as about 0.46. As there is no such limitation, reaction turbines can be run at higher speeds.

14.7.1 Francis Turbines

Francis turbine is a radial inward flow turbine and is the most popularly used one in the medium head range of 60 to 300 m. Francis turbine was first developed as a purly radial flow turbine by James B. Francis, an American engineer in 1849. But the design has gradually changed into a mixed flow turbine of today.

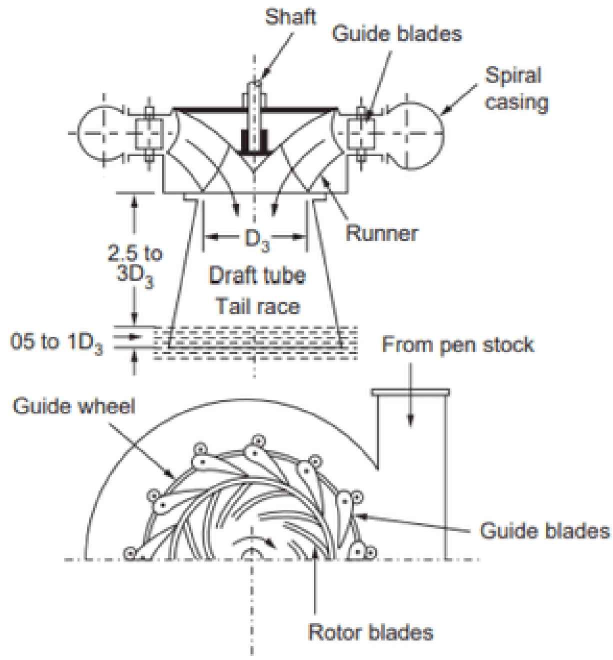


Figure 14.7.1 Typical sectional and front view of a modern Francis turbine.

A sectional view of a typical Francis turbine of today is shown in figure 14.7.1.

The main components are (*i*) The spiral casing (*ii*) Guide vanes (*iii*) Runner (*iv*) Draft tube and (*v*) Governor mechanism. Most of the machines are of vertical shaft arrangement while some smaller units are of horizontal shaft type.

14.7.1.1 Spiral Casing

The spiral casing surrounds the runner completely. Its area of cross section decreases gradually around the circumference. This leads to uniform distribution of water all along the circumference of the runner. Water from the penstock pipes enters the spiral casing and is distributed uniformly to the guide blades placed on the periphery of a circle. The casing should be strong enough to withstand the high pressure.

14.7.1.2 Guide Blades

Water enters the runner through the guide blades along the circumference. The number of guide blades are generally fewer than the number of blades in the runner. These should also be not simple multiples of the runner blades. The guide blades in addition to guiding the water at the proper direction serves two important functions. The water entering the guide blades are imparted a tangential velocity by the drop in pressure in the passage of the water through the blades. **The blade passages act as a nozzle in this aspect**.

The guide blades rest on pivoted on a ring and can be rotated by the rotation of the ring, whose movement is controlled by the governor. In this way the area of blade passage is changed to vary the flow rate of water according to the load so that the speed can be maintained constant. The variation of area between guide blades is illustrated in Figure 14.7.2. The control mechanism will be discussed in a later section.

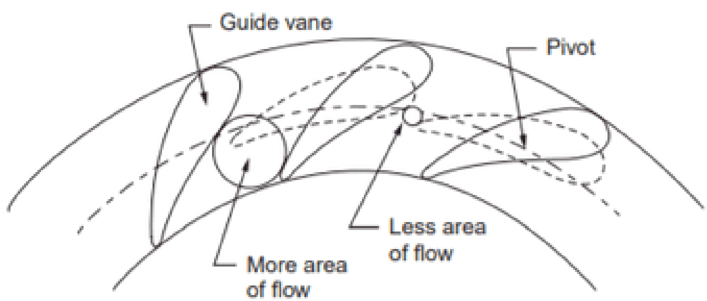


Figure 14.7.2. Guide vanee and giude wheel

14.7.1.3 The Runner

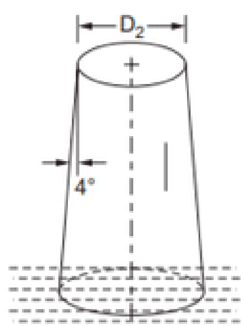
The runner is circular disc and has the blades fixed on one side. In high speed runners in which the blades are longer a circular band may be used around the blades to keep them in position.

The shape of the runner depends on the specific speed of the unit. These are classified as (*a*) slow runner (*b*) medium speed runner (*c*) high speed runner and (*d*) very high speed runner.

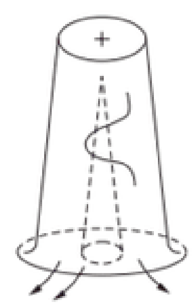
The shape of the runner and the corresponding velocity triangles are shown in figure 14.7.3. The development of mixed flow runners was necessitated by the limited power capacity of the purely radial flow runner. A larger exit flow area is made possible by the change of shape from radial to axial flow shape. This reduces the outlet velocity and thus increases efficiency. As seen in the figure the velocity triangles are of different shape for different runners. It is seen from the velocity triangles that the blade inlet angle β_1 changes from acute to obtuse as the speed increases. The guide vane outlet angle α_1 also increases from about 15° to higher values as speed increases.

14.7.1.4 Draft Tube

The turbines have to be installed a few meters above the flood water level to avoid innundation. In the case of impulse turbines this does not lead to significant loss of head. In the case of reaction turbines, the loss due to the installation at a higher level from the tailrace will be significant. This loss is reduced by connecting a fully flowing diverging tube from the turbine outlet to be immersed in the tailrace at the tube outlet. This reduces the pressure loss as the pressure at the turbine outlet will be below atmospheric due to the arrangement. **The loss in effective head is reduced by this arrangement. Also because of the diverging section of the tube the kinetic energy is converted to pressure energy which adds to the effective head.** The draft tube thus helps (1) to regain the lost static head due to higher level installation of the turbine and (2) helps to recover part of the kinetic energy that otherwise may be lost at the turbine outlet. A draft tube arrangement is shown in Figure 14.7.1 (as also in figure 14.7.5). Different shapes of draft tubes is shown in figure 14.7.6.



Straight divergent tube



Moody's bell mouthed tube

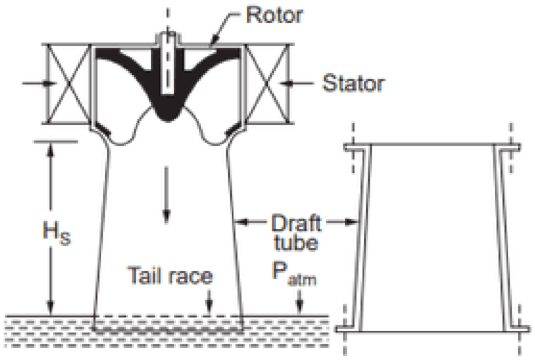


Figure 14.7.5 Draft tube

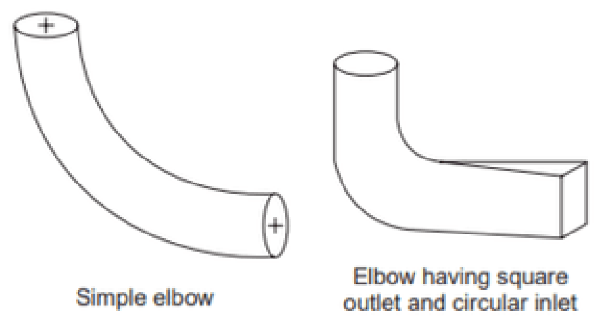


Figure 14.7.6 Various shapes of draft tubes

14.8 AXIAL FLOW TURBINES

The popular axial flow turbines are the Kaplan turbine and propeller turbine. In propeller turbine the blades are fixed. In the Kaplan turbines the blades are mounted in the boss in bearings and the blades are rotated according to the flow conditions by a servomechanism maintaining constant speed. In this way a constant efficiency is achieved in these turbines. The system is costly and where constant load conditions prevail, the simpler propeller turbines are installed.

In the discussions on Francis turbines, it was pointed out that as specific speed increases (more due to increased flow) the shape of the runner changes so that the flow tends towards axial direction. This trend when continued, the runner becomes purely axial flow type.

There are many locations where large flows are available at low head. In such a case the specific speed increases to a higher value. In such situations axial flow turbines are gainfully employed. A sectional view of a kaplan turbines in shown in figure 14.8.1. These turbines are suited for head in the range 5 – 80 m and specific speeds in the range 350 to 900. The water from supply pipes enters the spiral casing as in the case of Francis turbine. Guide blades direct the water into the chamber above the blades at the proper direction. The speed governor in this case acts on the guide blades and rotates them as per load requirements. The flow rate is changed without any change in head. The water directed by the guide blades enters the runner which has much fewer blades (3 to 10) than the Francis turbine. The blades are also rotated by the governor to change the inlet blade angle as per the flow direction from the guide blades, so that entry is without shock. As the head is low, many times the draft tube may have to be elbow type. The important dimensions are the diameter and the boss diameter which will vary with the chosen speed. At lower specific speeds the boss diameter may be higher.

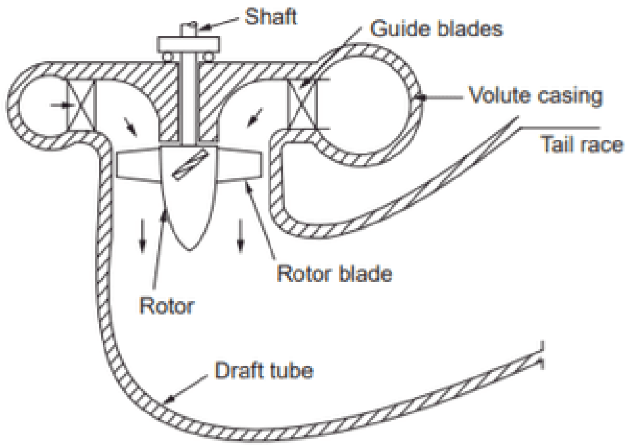


Figure 14.8.1 Sectional view of kaplan furbine

The number of blades depends on the head available and varies from 3 to 10 for heads from 5 to 70 m. As the peripheral speed varies along the radius (proportional to the radius) the blade inlet angle should also vary with the radius. Hence twisted type or Airfoil blade section has to be used. The speed ratio is calculated on the basis of the tip speed as $\phi = u/\sqrt{2gH}$ and varies from 1.5 to 2.4. The flow ratio lies in the range 0.35 to 0.75.

Typical velocity diagrams at the tip and at the hub are shown in Figure 14.8.2. The diagram is in the axial and tangential plane instead of radial and tangential plane as in the other turbines.

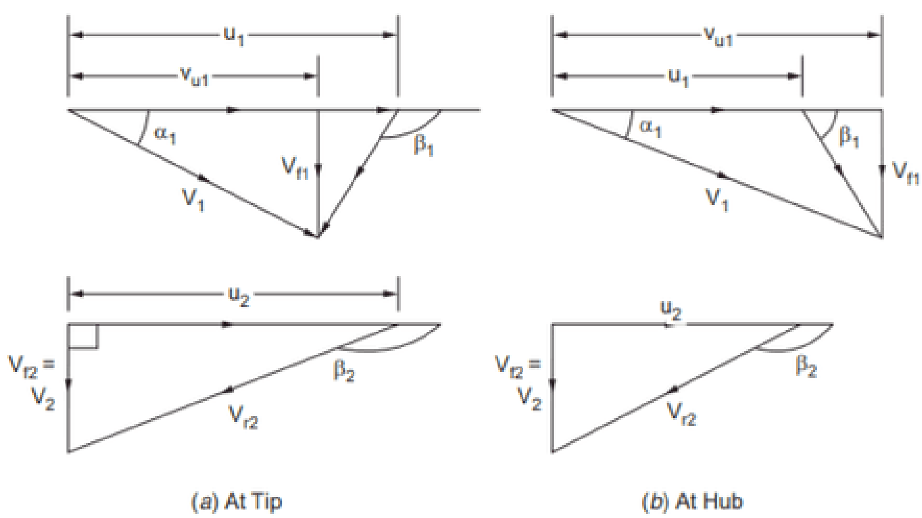


Figure 14.8.2 Typical velocity diagrams for Kaplan turbine

Work done = $u_1 V_{u1}$ (Taken at the mean diameter)

$$\eta_H = \frac{u_1 V_{u1}}{g H}$$

All other relations defined for other turbines hold for this type also. The flow velocity remains constant with radius. As the hydraulic efficiency is constant all along the length of the blades, $u_1 V_{u1}$ = Constant along the length of the blades or V_{u1} decreases with redius.

Kaplan turbine has a flat characteristics for variation of efficiency with load. Thus the part load efficiency is higher in this case. In the case of propeller turbine the part load efficiency suffers as the blade angle at these loads are such that entry is with shock.

14.9 CAVITATION IN HYDRAULIC MACHINES

If at any point in the flow the pressure in the liquid is reduced to its vapour pressure, the liquid will then will boil at that point and bubbles of vapour will form. As the fluid flows into a region of higher pressure the bubbles of vapour will suddenly condense or collapse. This action produces very high dynamic pressure upon the adjacent solid walls and since the action is continuous and has a high frequency the material in that zone will be damaged. Turbine runners and pump impellers are often severely damaged by such action. The process is called cavitation and the damage is called cavitation damage. In order to avoid cavitation, the absolute pressure at all points should be above the vapour pressure.

Cavitation can occur in the case of reaction turbines at the turbine exit or draft tube inlet where the pressure may be below atmospheric level. In the case of pumps such damage may occur at the suction side of the pump, where the absolute pressure is generally below atmospheric level.

In addition to the damage to the runner cavitation results in undesirable vibration noise and loss of efficiency. The flow will be disturbed from the design conditions. In reaction turbines the most likely place for cavitation damage is the back sides of the runner blades near their trailing edge. The critical factor in the installation of reaction turbines is the vertical distance from the runner to the tailrace level. For high specific speed propeller units it may be desirable to place the runner at a level lower than the tailrace level.

To compare cavitation characteristics a cavitation parameter known as Thoma cavitation coefficient, σ, is used. It is defined as

$$\sigma = \frac{h_a - h_r - z}{h} \qquad (14.8.1)$$

where h_a is the atmospheric head h_r is the vapour pressure head, z is the height of the runner outlet above tail race and h is the total operating head. The minimum value of σ at which cavitation occurs is defined as critical cavitation factor σ_e. Knowing σ_c the maximum value of z can be obtained as

$$z = h_a - h_v - \sigma_e h \qquad (14.8.2)$$

σ_c is found to be a function of specific speed. In the range of specific speeds for Francis turbine σ_c varies from 0.1 to 0.64 and in the range of specific speeds for Kaplan turbine σ_c varies from 0.4 to 1.5. The minimum pressure at the turbine outlet, h_0 can be obtained as

$$h_0 = h_a - z - \sigma_c H \qquad (14.8.3)$$

There are a number of correlations available for the value of σ_c in terms of specific speed, obtained from experiments by Moody and Zowski. The constants in the equations depends on the system used to calculate specific speed.

For Francis runners $\sigma_c = 0.006 + 0.55\ (N_s/444.6)^{1.8}$ (14.8.4)

For Kaplan runners $\sigma_c = 0.1 + 0.3\ [N_s/444.6]^{2.5}$ (14.8.5)

Other empirical corrlations are

Francis runner
$$\sigma_c = 0.625 \left[\frac{N_s}{380.78}\right]^2 \qquad (14.8.6)$$

For Kaplan runner
$$\sigma_c = 0.308 + \frac{1}{6.82}\left(\frac{N_s}{380.78}\right)^2 \qquad (14.8.7)$$

MODULE 6

RECIPROCATING PUMPS

16.0 INTRODUCTION

There are two main types of pumps namely the dynamic and positive displacement pumps. Dynamic pumps consist of centrifugal, axial and mixed flow pumps. In these cases pressure is developed by the dynamic action of the impeller on the fluid. Momentum is imparted to the fluid by dynamic action. This type was discussed in the previous chapter. Positive displacement pumps consist of reciprocating and rotary types. These types of pumps are discussed in this chapter. In these types a certain volume of fluid is taken in an enclosed volume and then it is forced out against pressure to the required application.

16.1 COMPARISON

Dynamic pumps	Positive displacement pumps
1. Simple in construction.	More complex, consists of several moving parts.
2. Can operate at high speed and hence compact.	Speed is limited by the higher inertia of the moving parts and the fluid.
3. Suitable for large volumes of discharge at moderate pressures in a single stage.	Suitable for fairly low volumes of flow at high pressures.
4. Lower maintenance requirements.	Higher maintenance cost.
5. Delivery is smooth and continuous.	Fluctuating flow.

16.2 DESCRIPTION AND WORKING

The main components are:

1. Cylinder with suitable valves at inlet and delivery.
2. Plunger or piston with piston rings.
3. Connecting rod and crank mechanism.

4. Suction pipe with one way valve.
5. Delivery pipe.
6. Supporting frame.
7. Air vessels to reduce flow fluctuation and reduction of acceleration head and friction head.

A diagramatic sketch is shown in Fig. 16.2.1.

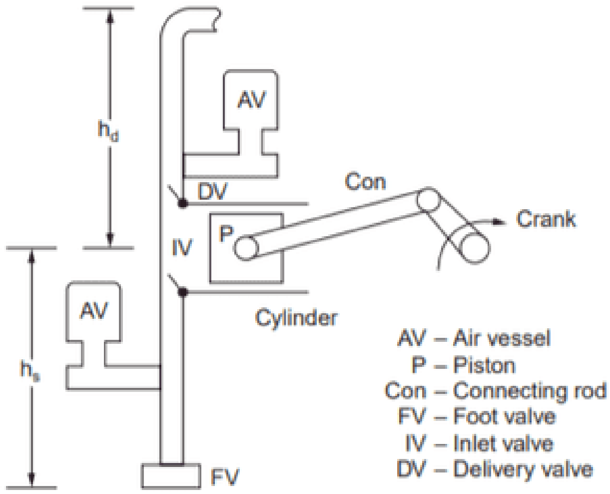


Figure 16.2.1 Diagramatic view of single acting reciprocating pump

Figure 16.2.1 Diagramatic view of single acting reciprocating pump

The action is similar to that of reciprocating engines. As the crank moves outwards, the piston moves out creating suction in the cylinder. Due to the suction water/fluid is drawn into the cylinder through the inlet valve. The delivery valve will be closed during this outward stroke. During the return stroke as the fluid is incompressible pressure will developed immediately which opens the delivery valve and closes the inlet valve. During the return stroke fluid will be pushed out of the cylinder against the delivery side pressure. The functions of the air vessels will be discussed in a later section. The volume delivered per stroke will be the product of the piston area and the stroke length. In a single acting type of pump there will be only one delivery stroke per revolution. Suction takes place during half revolution and delivery takes place during the other half. As the piston speed is not uniform (crank speed is uniform) the discharge will vary with the position of the crank. The discharge variation is shown in figure 16.2.2.

In a single acting pump the flow will be fluctuating because of this operation.

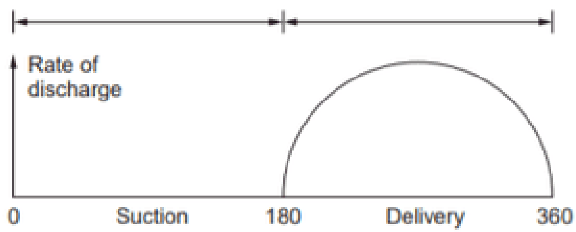


Figure 16.2.2 Flow variation during crank movement of single acting pump

Fluctuation can be reduced to some extent by double acting pump or multicylinder pump.

The diagramatic sketch of a double acting pump is shown in figure 16.2.3.

In this case the piston cannot be connected directly with the connecting rod. A gland and packing and piston rod and cross-head and guide are additional components. There will be nearly double the discharge per revolution as compared to single acting pump. When one side of the piston is under suction the other side will be delivering the fluid under pressure. As can be noted, the construction is more complex.

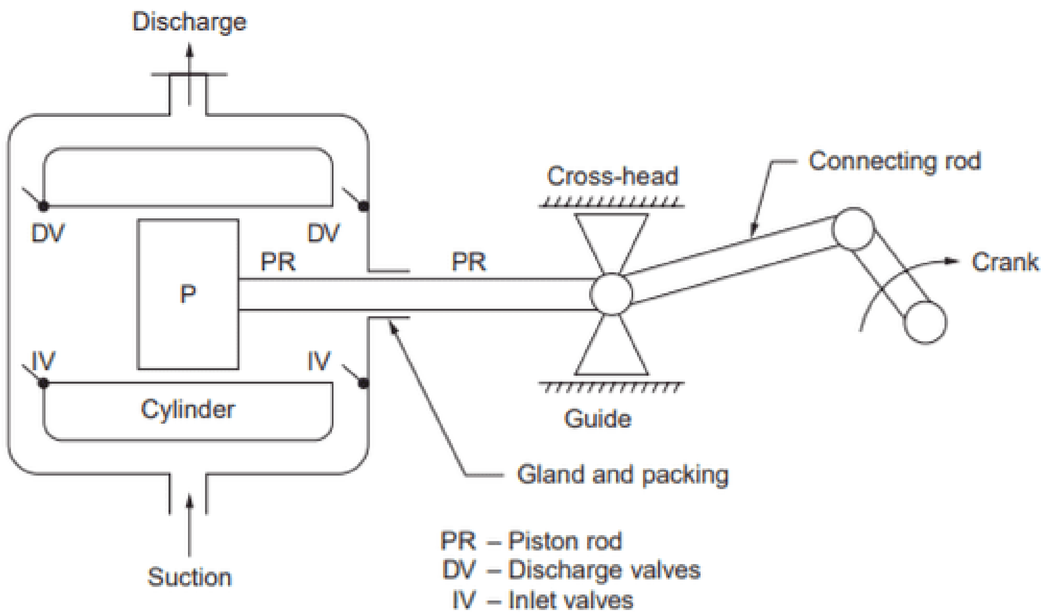


Figure 16.2.3 Diagramatic view of a double action pump

16.3 FLOW RATE AND POWER

Theoretical flow rate per second for single acting pump is given by, $Q_{SA} = \dfrac{L\,A\,N}{60}$ m^3/s

(16.3.1)

Where L is the length of stroke, A is the cylinder or piston area and N is the revolution per minute. It is desirable to express the same in terms of crank radius and the angular velocity as simple harmonic motion is assumed.

$$\omega = \frac{2\pi N}{60},\ N = \frac{60\,\omega}{2\pi},\ r = \frac{L}{2}$$

$$Q_{SA} = \frac{2r\,.\,A \times 60\,\omega}{2\pi \times 60} = \frac{A\omega r}{\pi}\ \text{m}^3/\text{s} \qquad (16.3.1a)$$

In double acting pumps, the flow will be nearly twice this value. If the piston rod area is taken into account, then

$$Q_{DA} = \frac{A\,L\,N}{60} + (A - A_{pr})\,\frac{L\,N}{60}\ \text{m}^3/\text{s} \qquad (16.3.2)$$

Compared to the piston area, the piston rod area is very small and neglecting this will lead to an error less than 1%.

$$\therefore \quad Q_{DA} = \frac{2A\,L\,N}{60} = \frac{2A\,w\,r}{\pi}\ \text{m}^3/\text{s}. \qquad (16.3.2a)$$

For multicylinder pumps, these expressions, (16.3.1), (16.3.1*a*), (16.3.2), and (16.3.2*a*) are to be multiplied by the number of cylinders.

16.3.1 Slip

There can be leakage along the valves, piston rings, gland and packing which will reduce the discharge to some extent. This is accounted for by the term slip.

$$\text{Percentage of slip} = \frac{Q_{th} - Q_{ac}}{Q_{th}} \times 100 \qquad (16.3.3)$$

Where Q_{th} is the theoretical discharge given by equation (16.3.1) and 2 and Q_{ac} is the measured discharge.

$$\text{Coefficient of discharge, } C_d = \frac{Q_{ac}}{Q_{th}} \qquad (16.3.4)$$

It has been found in some cases that $\mathbf{Q_{ac} > Q_{th}}$, due to operating conditions. In this case the slip is called **negative slip**. When the delivery pipe is short or the delivery head is small and the accelerating head in the suction side is high, the delivery valve is found to open before the end of suction stroke and the water passes directly into the delivery pipe. Such a situation leads to negative slip.

Theoretical power $= mg(h_s + h_d)$ W

where m is given by $Q \times \delta$. (16.3.5)

Example 16.1 *A single acting reciprocating pump has a bore of 200 mm and a stroke of 350 mm and runs at 45 rpm. The suction head is 8 m and the delivery head is 20 m. Determine the theoretical discharge of water and power required. If slip is 10%, what is the actual flow rate ?*

Theoretical flow volume $Q = \frac{L\,A\,N}{60} = \frac{0.35 \times \pi \times 0.2^2}{4} \times \frac{45}{60}$

$= 8.247 \times 10^{-3}$ m³/s or 8.247 l/s or 8.247 kg/s

Theoretical power = (mass flow/s) × head in $m \times g$ Nm/s or W

$= 0.9 \times 8.247 \times (20 + 8) \times 9.81$

= 2039 W or 2.039 kW

$$\text{Slip} = \frac{Q_{th} - Q_{ac}}{Q_{th}},\ 0.1 = \frac{8.247 - Q_{ac}}{8.247}$$

$\therefore$ $\mathbf{Q_{actual} = 7.422}$ **l/s**

The actual power will be higher than this value due to both solid and fluid friction.

Example 16.2 *A double acting reciprocating pump has a bore of 150 mm and stroke of 250 mm and runs at 35 rpm. The piston rod diameter is 20 mm. The suction head is 6.5 m and the delivery head is 14.5 m. The discharge of water was 4.7 l/s. Determine the slip and the power required.*

$$\mathbf{Q} = \frac{L A_1 N}{60} + \frac{L A_2 N}{60} = \frac{L N}{60} [A_1 + A_2]$$

$$= \frac{0.25 \times 35}{60} \left[\frac{\pi \times 0.15^2}{4} + \frac{\pi}{4} (0.15^2 - 0.02^2) \right]$$

$$= \frac{0.25 \times 35 \times \pi}{60 \times 4} [2 \times 0.15^2 - 0.02^2]$$

$$= \mathbf{5.108 \times 10^{-3}\ m^3/s\ or\ 5.108\ l/s\ or\ 5.108\ kg/s}$$

It piston rod area is not taken into account

$$Q = 5.154 \text{ l/s.}$$

An error of 0.9% rather negligible.

$$\textbf{Slip} = \frac{5.108 - 4.7}{5.108} \times 100 = \mathbf{7.99\%}$$

Theoretical power $= m\,g\,h = 4.7 \times 9.81 \times (14.5 + 6.5)$ W = **968 W**

The actual power will be higher than this value due to mechanical and fluid friction.

16.4 INDICATOR DIAGRAM

The pressure variation in the cylinder during a cycle consisting of one revolution of the crank. When represented in a diagram is termed as indicator diagram. The same is shown in figure 16.4.1.

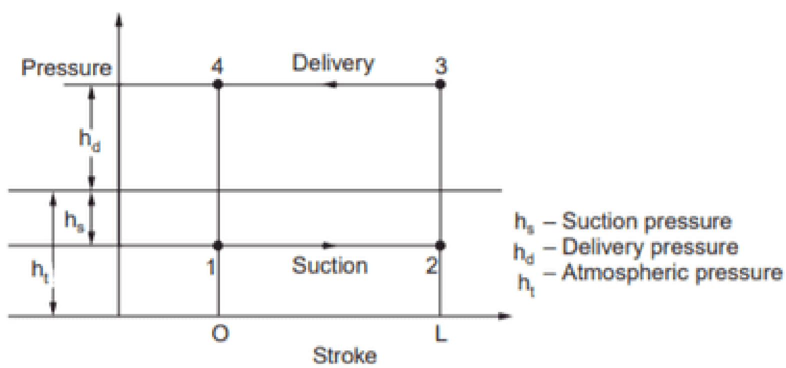


Figure 16.4.1 Indicator diagram for a crank revolution

Figure represents an ideal diagram, assuming no other effects are involved except the suction and delivery pressures. Modifications due to other effects will be discussed later in the section.

Point 1 represents the condition as the piston has just started moving during the suction stroke. 1-2 represents the suction stroke and the pressure in the cylinder is the suction pressure below the atmospheric pressure. The point 3 represents the condition just as the piston has started moving when the pressure increases to the delivery pressure. Along 3-4 representing the delivery stroke the pressure remains constant. The area enclosed represents the work done during a crank revolution to some scale

$$\text{Power} = Q\,\rho\,g(h_s + h_d) = \rho\,g\,L\,A\,N\,(h_s + h_d)/60 \qquad (16.4.1)$$

ROTODYNAMIC PUMPS

15.0 INTRODUCTION

Liquids have to be moved from one location to another and one level to another in domestic, agricultural and industrial spheres. The liquid is more often water in the domestic and agriculture spheres. In industries chemicals, petroleum products and in some cases slurries have to be moved, by pumping. **Three types of pumps** are in use.

(1) **Rotodynamic pumps** which move the fluid by dynamic action of imparting momentum to the fluid using mechanical energy. (2) **Reciprocating pumps** which first trap the liquid in a cylinder by suction and then push the liquid against pressure. (3)**Rotary positive displacement pumps** which also trap the liquid in a volume and push the same out against pressure.

Reciprocating pumps are limited by the **low speed of operation** required and **small volumes it can handle.**

Rotary positive displacement pumps are limited **by lower pressures** of operation and **small volumes** these can handle. Gear, vane and lobe pumps are of these type. **Rotodynamic pumps** *i.e.* centrifugal and axial flow pumps can be operated at **high speeds** often directly coupled to electric motors. These can handle from **small volumes to very large volumes.** These pumps **can handle corrosive and viscous, fluids and even slurries**. The **overall efficiency is high** in the case of these pumps. Hence these are found to be the most popular pumps in use. Rotodynamic pumps can be of **radial flow, mixed flow and axial flow** types according to the flow direction. **Radial flow** or purely centrifugal pumps generally handle **lower volumes at higher pressures. Mixed flow** pumps handle comparatively **larger volumes** at **medium range of pressures. Axial flow pumps** can handle **very large volumes,** but the **pressure** against which these pumps operate is **limited**. The overall efficiency of the three types are nearly the same.

15.1 CENTRIFUGAL PUMPS

These are so called because energy is imparted to the fluid by centrifugal action of moving blades from the inner radius to the outer radius. The **main components** of centrifugal pumps are (1) **the impeller,** (2) **the casing** and (3) **the drive shaft with gland and packing.**

Additionally suction pipe with one way valve (foot valve) and delivery pipe with delivery valve completes the system.

The liquid enters the eye of the impeller axially due to the suction created by the impeller motion. The impeller blades guide the fluid and impart momentum to the fluid, which increases the total head (or pressure) of the fluid, causing the fluid to flow out. The fluid comes out at a high velocity which is not directly usable. The casing can be of simple volute type or a diffuser can be used as desired. The volute is a spiral casing of gradually increasing cross section. A part of the kinetic energy in the fluid is converted to pressure in the casing.

Figure 15.1.1 shows a sectional view of the centrifugal pump.

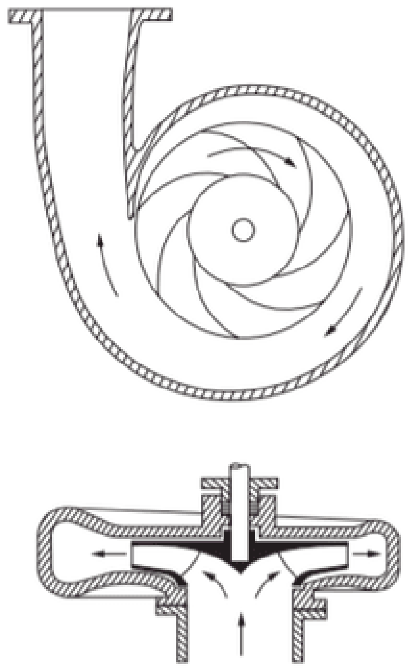


Figure 15.1.1 Volute type centrifugal pump.

Gland and packing or so called stuffing box is used to reduce leakage along the drive shaft. By the use of the volute only a small fraction of the kinetic head can be recovered as useful static head. A diffuser can diffuse the flow more efficiently and recover kinetic head as useful static head. A view of such arrangement is shown in figure 15.1.2. Diffuser pump are also called as turbine pumps as these resembles Francis turbine with flow direction reversed.

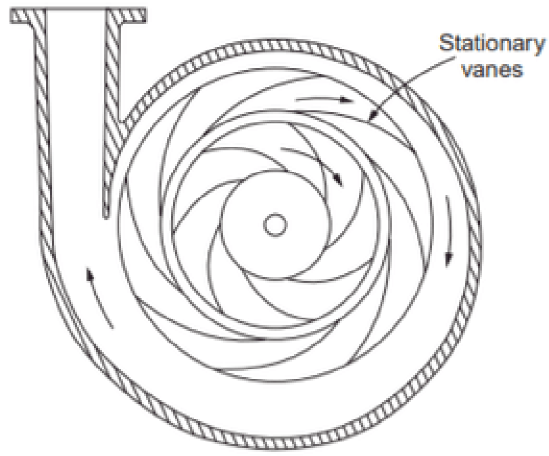


Figure 15.1.2 Diffuser pump.

15.1.1 Impeller

The impeller consists of a disc with blades mounted perpendicularly on its surface. The blades may of three different orientations. These are **(*i*) Radial, (*ii*) Backward curved, and (*iii*) Forward curved.** Backward and forward refers to the direction of motion of the disc periphery. Of these the most popular one is the backward curved type, due to its desirable characteristics, which reference to the static head developed and power variation with flow rate. This will be discussed in detail later in this chapter.

A simple disc with blades mounted perpendicularly on it is called **open impeller.** If another disc is used to cover the blades, this type is called **shrouded impeller.** This is more popular with water pumps. Open impellers are well adopted for use with dirty or water containing solids. The **third type** is just the **blades spreading out from the shaft.** These are used to pump slurries. Impellers may be of cast iron or bronzes or steel or special alloys as required by the application. In order to maintain constant radial velocity, the width of the impeller will be wider at entrance and narrower at the exit. This may be also noted from figure 15.1.1.

The blades are generally cast integral with the disc. Recently even plastic material is used for the impeller. **To start delivery of the fluid the casing and impeller should be filled with the fluid without any air pockets. This is called priming.** If air is present the there will be only compression and no delivery of fluid. In order to release any air entrained an air valve is generally provided **The one way foot value keeps the suction line and the pump casing filled with water.**

15.1.2 Classification

As already mentioned, centrifugal pumps may be classified in several ways. On the **basis of speed** as low speed, medium speed and high speed pumps. On the basis of **direction of flow** of fluid, the classification is **radial flow**, **mixed flow** and **radial flow.** On the basis of **head** pumps may be classified as **low head** (10 m and below), **medium head** (10-50 m) **and high head** pumps. Single entry type and double entry type is another classification. Double entry pumps have blades on both sides of the impeller disc. This leads to reduction in axial thrust and increase in flow for the same speed and diameter. Figure 15.1.3 illustrates the same. When the head required is high and which cannot be developed by a single impeller, **multi staging** is used. In deep well submersible pumps the diameter is limited by the diameter of the bore well casing. In this case multi stage pump becomes a must. In multi stage pumps several impellers are mounted on the same shaft and the outlet flow of one impeller is led to the inlet of the next impeller and so on. The total head developed equals the sum of heads developed by all the stages.

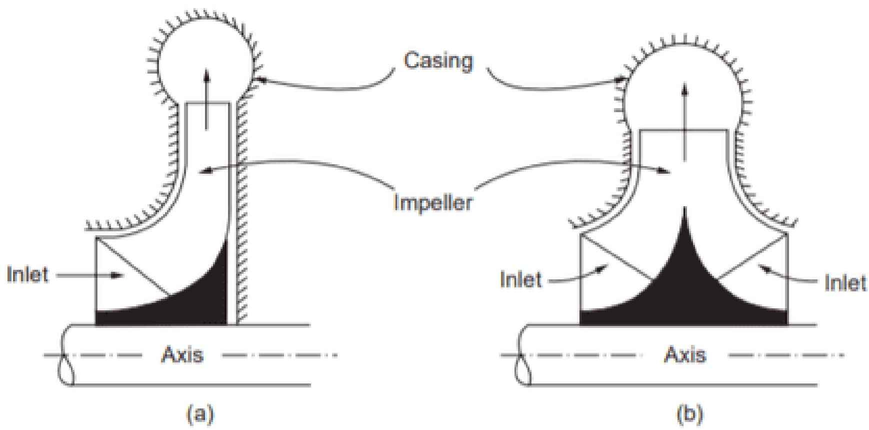


Figure 15.1.3 Single and double entry pumps

Pumps may also be operated in parallel to obtain large volumes of flow. The characteristics under series and parallel operations is discussed later in the chapter. The classification may also be based on the **specific speed** of the pump. In chapter 9 the dimensionless parameters have been derived in the case of hydraulic machines. The same is also repeated in example 15.1. The expression for the dimensionless specific speed is given in equation 15.1.1.

$$N_s = \frac{N\sqrt{Q}}{(gH)^{3/4}} \qquad (15.1.1)$$

More often dimensional specific speed is used in practise. In this case

$$N_s = \frac{N\sqrt{Q}}{H^{3/4}} \qquad (15.1.1a)$$

The units used are : N in rpm, Q in m³/s, and H in meter.

Typical values are given in table 15.1

Table 15.1 Specific speed classification of pumps.

Flow direction	speed	Dimensional specific speed	Non Dimensional specific speed
Radial	Low	10 – 30	1.8 – 5.4
	Medium	30 – 50	5.4 – 9.0
	High	50 – 80	9.0 – 14.0
Mixed flow		80 – 160	14 – 29
Axial flow		100 – 450	18 – 81

The best efficiency is obtained for the various types of pumps in this range of specific speeds indicated.

15.4 PUMP CHARACTERISTICS

We have seen that the theoretical head

$$H_{th} = \frac{u_2 V_{u2}}{g} \quad \text{and} \quad V_{u2} = V_{f2} \cot \beta_2$$

$$V_{f2} = \frac{Q}{A}, \text{ where } A \text{ is the circumferential area.}$$

$$u_2 = \pi DN.$$

Substituting these relations in the general equation. We can write

$$H_{th} = \pi^2 D^2 N^2 - \left(\frac{\pi DN}{A} . \cot \beta_2\right) Q.$$

For a given pump, D, A, β_2 and N are fixed . So at constant speed we can write

$$\mathbf{H_{th} = k_1 - k_2 Q} \qquad (15.4.1)$$

where k_1 and k_2 are constants and

$$k_1 = \pi^2 D^2 N^2 \text{ and } k_2 = \left(\frac{\pi DN}{A} . \cot \beta_2\right)$$

Hence at constant speed this leads to a drooping linear characteristics for backward curved blading. This is shown by curve 1 in Figure 15.4.1.

The slip causes drop in the head, which can be written as $\sigma V_{u2} u_2/g$. As flow increases this loss also increases. Curve 2 shown the head after slip. The flow will enter without shock only at the design flow rate. At other flow rates, the water will enter with shock causing losses. This lose can be expressed as

$$\mathbf{h_{shock} = k_3 (Q_{th} - Q)^2}$$

The reduced head after shock losses is shown in curve 5. The shock losses with flow rate is shown by curve 3. The mechanical losses can be represented by $\mathbf{h_f = k_4 Q^2}$. The variation is shown by curve 4. With variation of speed the head characteristic is shifted near paralley with the curve 5 shown in Figure 15.4.1.

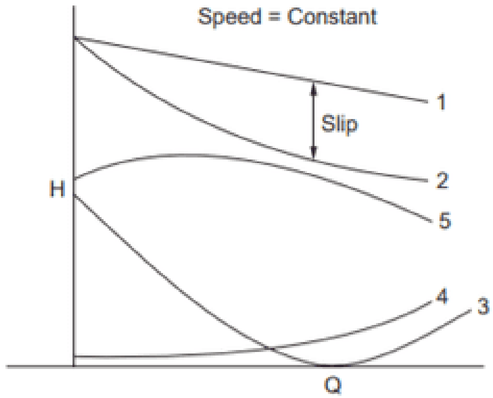


Figure 15.4.1 Characteristics of a centrifugal pump