Rajsons Publications Pvt. Ltd.

HYDRAULICS AND FLUID MECHANICS INCLUDING HYDRAULICS MACHINES

DR. P.N. MODI
DR. S.M. SETH

STANDARD BOOK HOUSE (SINCE 1960)

HYDRAULICS AND FLUID MECHANICS

Including

HYDRAULIC MACHINES

(In SI Units)

P. ARUN KUMAR
CIVIL A2
160111732090

By

Dr. P.N. Modi

B.E., M.E., Ph.D
Former Professor of Civil Engineering,
M.R. Engineering College, (Now M.N.I.T), Jaipur
Formerly Principal, Kautilya Institute of Technology and Engineering, Jaipur

and

Dr. S.M. Seth

B.E., M.E., M.I.E., Ph.D (Manchester)
Former Director, National Institute of Hydrology, Roorkee
Presently Principal, Kautilya Institute of Technology and Engineering, Jaipur

STANDARD BOOK HOUSE

Unit of: **RAJSONS PUBLICATIONS PVT. LTD.**

1705-A, Nai Sarak, PB. No. 1074, Delhi-110006 Ph.: +91-(011)-23265506
Show Room: 4262/3, First Lane, G-Floor, Gali Punjabian, Ansari Road, Darya Ganj,
New Delhi-110002 Ph.: +91-(011)43551085 Tele Fax : +91-(011)43551185,
Fax: +91-(011)-23250212

E-mail: sbh10@hotmail.com www.standardbookhouse.com

DOWNLOADED FROM www.CivilEnggForAll.com

Contents

Properties of Fluids

Chapter 1

1.1 INTRODUCTION

A matter exists in either the solid state or the fluid state. The fluid state is further divided into the liquid and the gaseous states. In fact the same matter may exist in any one of the three states viz., solid, liquid and gaseous. For example water, which ordinarily occurs in a liquid state, may also occur under natural conditions in a solid state as ice and in a gaseous state as vapour. The solids, liquids and gases exhibit different characteristics on account of their different molecular structure. All substances consist of vast numbers of molecules separated by empty space. The molecules are continuously moving within the substance and they have an attraction for each other, but when the distance between them becomes very small (of the order of the diameter of the molecule) there is a force of repulsion between the molecules which pushes them apart. In solids the molecules are very closely spaced, but in liquids the spacing between the molecules is relatively large and in gases the space between the molecules is still larger. As such in a given volume a solid contains a large number of molecules, a liquid contains relatively less number of molecules and a gas contains much less number of molecules. It thus follows that in solids the force of attraction between the molecules is large on account of which there is very little movement of molecules within the solid mass and hence solids possess compact and rigid form. In liquids the force of attraction between the molecules is relatively less due to which the molecules can move freely within the liquid mass, but the force of attraction between the molecules is sufficient to keep the liquid together in a definite volume. In gases the force of attraction between the molecules is much less due to which the molecules of gases have greater freedom of movement so that the gases fill completely the container in which they are placed. It may, however, be stated that inspite of the larger mobility and spacing of the molecules of fluids, for mechanical analysis a fluid is considered to be *continuum* i.e., a continuous distribution of matter with no voids or empty spaces. This assumption is justifiable because ordinarily the fluids involved in most of the engineering problems have large number of molecules and the distances between them are small.

Another difference that exists between the solids and the fluids is in their relative abilities to resist the external forces. A solid can resist tensile, compressive and shear forces upto a certain limit. A fluid has no tensile strength or very little of it, and it can resist the compressive forces only when it is kept in a container. When subjected to a shearing force, a fluid deforms continuously as long as this force is applied. The inability of the fluids to resist shearing stress gives them their characteristic property to change shape or to flow. This, however, does not mean that the fluids do not offer any resistance to shearing forces. In fact as the fluids flow there exists shearing (or tangential) stresses between the adjacent fluid layers which result in opposing the movement of one layer over the other. The amount of shear stress in a fluid depends on the magnitude of the rate of deformation of the fluid element. However, if a fluid is at rest no shear force can exist in it.

The two classes of fluids viz., gases and liquids also exhibit quite different characteristics. Gases can be compressed much readily under the action of external pressure and when the external pressure is removed the gases tend to expand indefinitely. On the other hand under ordinary conditions liquids are quite difficult to compress and therefore they may for most purposes be regarded as incompressible. Moreover, even if the pressure acting on a liquid mass is removed, still the cohesion between particles holds them together so that the liquid does not expand indefinitely and it may have a free surface, that is a surface from which all pressure except atmospheric pressure is removed.

1.2 DEFINITION OF A FLUID

In view of the above discussion a *fluid* may be defined as a substance which is capable of flowing. It has no definite shape of its own, but conforms to the shape of the containing vessel. Further even a small amount of shear force exerted on a fluid will cause it to undergo a deformation which continues as long as the force continues to be applied.

A *liquid* is a fluid, which possesses a definite volume, which varies only slightly with temperature and pressure. Since under ordinary conditions liquids are difficult to compress, they may be for all practical purposes regarded as incompressible. It forms a free surface or an interface separating it from the atmosphere or any other gas present.

A *gas* is a fluid, which is compressible and possesses no definite volume but it always expands until its volume is equal to that of the container. Even a slight change in the temperature of a gas has a significant effect on its volume and pressure. However, if the conditions are such that a gas undergoes a negligible change in its volume, it may be regarded as incompressible. But if the change in volume is not negligible the compressibility of the gas will have to be taken into account in the analysis.

A *vapour* is a gas whose temperature and pressure are such that it is very near the liquid state. Thus steam may be considered as a vapour because its state is normally not far from that of water.

The fluids are also classified as *ideal fluids* and *practical* or *real fluids*.

Ideal fluids are those fluids which have no viscosity and surface tension and they are incompressible. As such for ideal fluids no resistance is encountered as the fluid moves. However, in nature the ideal fluids do not exist and therefore, these are only imaginary fluids. The existence of these imaginary fluids was conceived by the mathematicians in order to simplify the mathematical analysis of the fluids in motion. The fluids which have low viscosity such as air, water etc., may however be treated as ideal fluids without much error.

Practical or real fluids are those fluids which are actually available in nature. These fluids possess the properties such as viscosity, surface tension and compressibility and therefore a certain amount of resistance is always offered by these fluids when they are set in motion.

1.3 DEVELOPMENT OF FLUID MECHANICS

Fluid mechanics is that branch of science which deals with the behaviour of the fluids at rest as well as in motion. In general the scope of fluid mechanics is very wide which includes the study of all liquids and gases. But usually it is confined to the study of liquids and those gases for which the effects due to compressibility may be neglected. The gases with appreciable compressibility effects are governed by the laws of Thermodynamics which are however dealt with under the subject of Gas dynamics.

The problems, man encountered in the fields of water supply, irrigation, navigation and water power, resulted in the development of the fluid mechanics. However, with the exception of Archimedes (250 B.C.) Principle which is considered to be as true today as some 2250 years ago, little of the scant knowledge of the ancients appears in modern fluid mechanics. After the fall of Roman Empire (476 A.D.) there is no record of progress made in fluid mechanics until the time of Leonardo da Vinci (1500 A.D.), who designed the first chambered canal lock. However, upto da Vinci's time, concepts of fluid motion must be considered to be more art than science.

Some two hundred years ago mankind's centuries of experience with the flow of water began to crystallize in scientific form. Two distinct Schools of thought gradually evolved in the treatment of fluid mechanics. One, commonly known as Classical Hydrodynamics, deals with theoretical aspects of the fluid flow, which assumes that shearing stresses are non-existent in the fluids, that is, ideal fluid concept. The other known as Hydraulics, deals with the practical aspects of fluid flow which has been developed from experimental findings and is, therefore, more of empirical nature. Notable contributions to theoretical hydrodynamics have been made by Euler, D'Alembert, Navier, Lagrange, Stokes, Kirchoff, Rayleigh, Rankine, Kelvin, Lamb and many others. Many investigators have contributed to the development of experimental hydraulics, notable amongst them being Chezy, Venturi, Bazin, Hagen, Poiseuille, Darcy, Weisbach, Kutter, Manning, Francis and several others.

Although the empirical formulae developed in hydraulics have found useful application in several problems, it is not possible to extend them to the flow of fluids other than water and in the advanced field of aerodynamics. As such there was a definite need for a new approach to the problems of fluid flow—an approach which relied on classical hydrodynamics for its analytical development and at the same time on experimental means for checking the validity of the theoretical analysis. The modern Fluid Mechanics provides this new approach, taking a balanced view of both the theorists and the experimentalists. The generally recognized founder of the modern fluid mechanics is the German Professor, Ludwig Prandtl. His most notable contribution being the boundary layer theory which has had a tremendous influence upon the understanding of the problems involving fluid motion. Other notable contributors to the modern fluid mechanics are Blasius, Bakhmeteff, Nikuradse, Von-Karman, Reynolds, Rouse and many others.

In this book the fundamental principles of fluid mechanics applicable to the problems involving the motion of a particular class of fluids called Newtonian fluids (such as water, air, kerosene, glycerine etc.) have been discussed along with the relevant portions of the experimental hydraulics.

1.4 UNITS OF MEASUREMENT

Units may be defined as those standards in terms of which the various physical quantities like length, mass, time, force, area, volume, velocity, acceleration etc., are measured. The system of units used in mechanics are based upon Newton's second law of motion, which states that force equals mass times acceleration or $F = m \times a$, where F is the force, m is the mass and a is the acceleration. There are in general four systems of units, two in metric (C.G.S. or M.K.S.) system and two in the English (F.P.S.) system. Of the two, one is known as the absolute or physicist's system and the other as the gravitational or engineer's system. The difference between the absolute and gravitational systems is that in the former the standard is the unit of mass. The unit of force is then derived by Newton's law. In the gravitational system the standard is the unit of force and the unit of mass is derived by Newton's law. Table 1.1 lists the various units of measurement for some of the basic or fundamental quantities.

TABLE 1.1

Quantity	Metric Units		English Units	
	Gravitational	Absolute	Gravitational	Absolute
Length	Metre (m)	Metre (m)	Foot (ft)	Foot (ft)
Time	Second (sec)	Second (sec)	Second (sec)	Second (sec)
Mass	Metric slug (msl)	Gram [gm (mass)]	Slug (sl)	Pound [lb (mass)]
Force	kilogram [kg(f)]	Dyne	Pound [lb (f)]	Poundal (pdl)
Temperature	°C	°C	°F	°F

The units of measurement for the various other quantities may be readily obtained with the help of the Table 1.1. Further Table 1.2 below illustrates all the four systems of units in which the units are defined so that one unit of force equals one unit of mass times one unit of acceleration.

TABLE 1.2

Systems of Units	Relationships
Metric Absolute	1 dyne = 1 gram × $\frac{1 \text{ cm}}{\text{sec}^2}$
Metric Gravitational	1 kilogram (f) = 1 metric slug × $\frac{1 \text{ m}}{\text{sec}^2}$
English Absolute	1 poundal = 1 pound × $\frac{1 \text{ ft}}{\text{sec}^2}$
English Gravitational	1 pound (f) = 1 slug × $\frac{1 \text{ ft}}{\text{sec}^2}$

Further the following relationships may be utilized to affect the conversion from one system to another.

1 gram-wt. = 981 dynes : 1 metric slug = 9810 gm (mass)

1 lb-wt.= 32.2 poundals : 1 slug = 32.2 lb (mass)

The use of the different systems of units by the scientists and the engineers and also by the different countries of the world often leads to a lot of confusion. Therefore, it was decided at the Eleventh General Conference of Weights and Measures held in Paris in 1960 to adopt a unified, systematically constituted, coherent system of units for international use. This system of units is called the *International System of Units* and is designated by the abbreviation *SI Units*. More and more countries of the world are now adopting this system of units. There are six base units in SI system of units which are given in Table 1.3.

TABLE 1.3

Quantity	Unit	Symbol
Length	metre	m
Mass	kilogram	kg
Time	second	s
Electric Current	ampere	A
Thermodynamic temperature	kelvin	K
Luminous intensity	candela	cd

In addition to the above noted base units there are two supplementary units which are given in Table 1.4.

TABLE 1.4

Quantity	Unit	Symbol
Plane angle	radian	rad
Solid angle	steradian	sr

The unit of a derived quantity is obtained by taking the physical law connecting it with the basic (or primary or fundamental) quantities and then introducing the corresponding units for the basic quantities. Thus in SI units the unit of force is newton (N) which according to Newton's second law of motion is expressed as 1 N = 1 kg × 1 m/s², i.e., a force of 1 N is required to accelerate a mass of 1 kg by 1 m/s². The units for some of the derived quantities have been assigned special names and symbols. Some of the important derived units with special names, commonly used in Fluid mechanics, in SI and metric gravitational systems of units are given in Table 1.5.

TABLE 1.5

Derived Quantity	System of Units			
	SI		Metric Gravitational	
(1)	Unit (2)	Symbol (3)	Unit (4)	Symbol (5)
Area	square metre	m^2	square metre	m^2
Volume	cubic metre	m^3	cubic metre	m^3
Velocity	metre per second	m/s	metre per second	m/sec
Angular velocity	radian per second	rad/s	radian per second	rad/sec
Acceleration	metre per second square	m/s^2	metre per second square	m/sec^2
Angular acceleration	radian per second square	rad/s^2	radian per second square	rad/sec^2
Frequency	hertz	Hz		1/sec
Discharge	cubic metre per second	m^3/s	cubic metre per second	m^3/sec
Mass density (Specific mass)	kilogram per cubic metre	kg/m^3	metric slug per cubic metre	msl/m^3
Force	newton	N (= $kg.m/s^2$)	Kilogram (f)	kg(f)-
Pressure, Stress, Elastic Modulus	newton per square metre (= pascal)	N/m^2 (= Pa)	kilogram (f) per square metre	$kg(f)/m^2$
Weight density (Specific Weight)	newton per cubic metre	N/m^3	kilogram (f) per cubic metre	$kg(f)/m^3$
Dynamic viscosity	newton second per square metre (= pascal second)	$N.s/m^2$ (= pa. s)	kilogram (f)-second per square metre	kg(f)-sec/m^2
Kinematic viscosity	square metre per second	m^2/s	square metre per second	m^2/sec
Work, Energy, Torque	joule	J (= N.m)	kilogram(f)-metre	kg(f)-m
Quantity of heat	joule	J	kilo-calorie	kcal
Power	watt	W (= J/s)	kilogram(f) - metre per second	kg(f)-m/sec
Surface tension	newton per metre	N/m	kilogram (f) per metre	kg(f)/m

Contd.

Table 1.5 Contd.

(1)	(2)	(3)	(4)	(5)
Momentum	kilogram metre per second	kg. m/s	metric slug - metre per second	msl-m/sec
Moment of momentum	kilogram square metre per second	kg.m/s	metric slug-square metre per second	msl-m^2/sec
Entropy	(i) joule per kilogram kelvin	J/(kg. K)	(i) kilo-calorie per metric slug deg. Centigrade abs.	k cal/msl °C (abs)
	or (ii) joule per kelvin	J/K	or (ii) kilo-calorie per deg. Centigrade abs.	k cal/°C (abs)
Specific heat	joule per kilogram kelvin	J/(kg. K)	kilo-calorie per metric slug deg. Centigrade abs.	k cal/msl °C (abs)
Gas constant	joule per kilogram kelvin	J/(kg.K)	kilogram (f) metre per metric slug deg. Centigrade abs.	kg (f)-m/msl °C (abs)
Thermal conductivity	watt per metre kelvin	W/m.K	kilocalorie per second metre deg. Centigrade abs.	kcal/sec-m °C (abs)

Certain units though outside the International System have been retained for general use in this system also. These units are given in Table 1.6.

TABLE 1.6

Quantity	*Unit*	*Symbol*	*Value in SI Units*
Area of land	area	a	100 m^2
– do–	hectare	ha	10 000 m^2
Time	minute	min	60 s
–do–	hour	h	60 min = 3600 s
–do–	day	d	24 h = 86 400 s
Mass	tonne	t	1000 kg
Volume	litre		110^{-3} m^3 = 1 dm^3
Dynamic viscosity	poise	P	10^{-1} N.s/m^2
–do–	centipoise	cP	10^{-3} N.s/m^2
Kinematic viscosity	stoke	S	10^{-4} m^2/s
Pressure of fluid	bar	bar	100 kN/m^2 = 10^5 Pa

In using SI units certain rules and conventions are to be followed which are as noted below:

(i) Names of units, even when they are named after persons, are not written with first letter capital when written in the spelled form e.g., newton, joule, watt etc.

(ii) The symbols for the units which are named after persons are written with capital first letter of the name e.g., N for newton, J for joule, W for watt, etc.

(iii) The symbols for all other units are written in lower case (small letters), e.g., m for metre, s for second, kg for kilogram, etc.

(iv) A dot is inserted in the space between the symbols for the compound (or combined) units e.g., N.m (for newton metre), kW-h (for kilowatt hour) etc.

(v) Numbers with more than three digits should be written in groups of three with narrow space in between consecutive groups. However, a sequence of four digits is usually not broken e.g.,

50000	should be	written as	50 000
72345.685	should be	written as	72 345.685
0.13579	should be	written as	0.135 79
9810	may be	written as	9 810 or 9810

(vi) The decimal multiples and sub-multiples of the units are formed by using the prefixes. The various prefixes and the corresponding symbols are given in Table 1.7.

TABLE 1.7

Unit multiplier	*Prefix*	*Symbol*	*Unit multiplier*	*Prefix*	*Symbol*
10^{18}	exa	E	10^{-1}	deci	d
10^{15}	peta	P	10^{-2}	centi	c
10^{12}	tera	T	10^{-3}	milli	m
10^{9}	giga	G	10^{-6}	micro	m
10^{6}	mega	M	10^{-9}	nano	n
10^{3}	kilo	k	10^{-12}	pico	p
10^{2}	hecto	h	10^{-15}	femto	f
10	deca	da	10^{-18}	atto	a

The prefixes hecto (h), deca (da), deci (d) and centi (c) are not commonly used and these should be used only when special necessity arises. Further as far as possible prefixes in steps of $10^{\pm 3}$ should only be used to form multiples and sub-multiples of the units. It may also be noted that compound prefixes (or combination of prefixes) are not to be used. For example the correct form for 10^{-9} is the prefix nano (n) and not the combination of prefixes such as milli micro (mμ) or any other combination which is incorrect and should not be used.

In this book both metric gravitational system of units as well as SI units have been used. Some important conversion factors in various systems of units are given in Appendix V.

1.5 MASS DENSITY, SPECIFIC WEIGHT, SPECIFIC VOLUME

Mass density (or specific mass) of a fluid is the mass which it possesses per unit volume. It is denoted by a symbol ρ (Greek 'rho'). In SI units mass density is expressed in kilogram per cubic metre i.e., kg/m^3. In the metric gravitational system of units mass density is expressed in metric slug per cubic metre i.e., msl/m^3 and in the metric absolute system of units it is expressed in gm (mass) per cubic centimetre i.e., gm/cm^3 or gm/cc. The corresponding units in the English gravitational and absolute systems of units are slugs per cubic foot i.e., slugs/ft^3 and pound (mass) per cubic foot i.e., lb (m)/ft^3 respectively. The mass density of water at °C in different systems of units is 1000 kg/m^3, or 102 msl/m^3, or 1 gm/cc, or 1.94 slugs/ft^3, or 62.4 lb(m)/ft^3.

Since a molecule of a substance has a certain mass regardless of its state (solid, liquid or gas), it then follows that the mass density is proportional to the number of molecules in a unit volume of the fluid. As the molecular activity and spacing increase with temperature, fewer molecules exist in a given volume of fluid as temperature rises. Therefore, the mass density of a fluid decreases with increasing temperature. Further by application of pressure a large number of molecules can be forced into a given volume, it is to be expected that the mass density of a fluid will increase with increasing pressure.

Specific weight (also called weight density) of a fluid is the weight it possesses per unit volume. It is denoted by a symbol w or γ (Greek 'gama'). As it represents the force exerted by gravity on a unit volume of fluid, it has units of force per unit volume. In SI units specific weight is expressed in newton per cubic metre i.e., N/m^3. In the metric gravitational system of units specific weight is expressed in kilogram (f) per cubic metre i.e., kg(f)/m^3 and in the metric absolute system of units it is expressed in dynes per cubic centimetre i.e., dynes/cm^3 or dynes/cc. The correponding units in the English gravitational and absolute systems of units are pound (f) per cubic foot i.e., lb (f)/ft^3 and poundal per cubic foot, i.e., pdl/ft^3 respectively. The specific weight of water at 4°C in different units is 9810 N/m^3 (or 9.81 kN/m^3), or 1000 kg(f)/m^3, or 981 dynes/cm^3, or 62.4 lb(f)/ft^3, or (62.4 × 32.2) pdl/ft^3.

The mass density ρ and specific weight w are related as indicated below

$$w = \rho g; \rho = \frac{w}{g} \quad ...(1.1)$$

where g is acceleration due to gravity.

The specific weight depends on the gravitational acceleration and the mass density. Since the gravitational acceleration varies from place to place, the specific weight will also vary. Further as stated earlier the mass density changes with temperature and pressure, hence the specific weight will also depend upon temperature and pressure.

Specific volume of a fluid is the volume of the fluid per unit weight. Thus it is the reciprocal of specific weight. It is generally denoted by v. In SI units specific volume is expressed in cubic metre per newton i.e., m^3/N. In the metric gravitational system of units specific volume is expressed in cubic metre per kilogram (f) i.e., m^3/kg(f) and in the metric absolute system of units it is expressed in cubic centimetre per dyne i.e., cm^3/dyne or cc/dyne. The corresponding units in the English gravitational and absolute systems of units are cubic foot per pound (f) i.e., ft^3/lb(f) and cubic foot per poundal i.e., ft^3/ pdl respectively.

For the problems involving the gas flow *specific volume* is generally defined as the volume of the fluid per unit mass, in which case it is reciprocal of mass density. In SI units the specific volume is then expressed in cubic metre per kilogram i.e., m^3/kg. In the metric gravitational system of units it is expressed in cubic metre per metric slug i.e., m^3/msl and in the metric absolute system of units it is expressed in cubic centimetre per gram(mass) i.e., cm^3/gm(m) or cc/gm(m). The corresponding units in the English gravitational and absolute units are cubic foot per slug i.e., ft^3/slug and cubic foot per pound (mass) i.e., ft^3/lb(m) respectively.

For liquids the mass density, the specific weight and specific volume vary only slightly with the variation of temperature and pressure. It is due to the molecular structure of the liquids in which the molecules are arranged very compactly (in contrast to that of a gas). The presence of dissolved air, salts in solution and suspended matter will slightly increase the values of the mass density and the specific weight of the liquids.

For gases the values of the above properties vary greatly with variation of either temperature, or pressure, or both. It is due to the molecular structure of the gas in which the molecular spacing (i.e., volume) changes considerably on account of pressure and temperature variations.

1.6 SPECIFIC GRAVITY

Specific gravity (sp. gr.) is the ratio of specific weight (or mass density) of a fluid to the specific weight (or mass density) of a standard fluid. For liquids, the standard fluid chosen for comparison is pure water at 4°C (39.2°F). For gases, the standard fluid chosen is either hydrogen or air at some specified temperature and pressure. As specific weight and mass density of a fluid vary with temperature, temperatures must be quoted when specific gravity is used in precise calculations of specific weight or mass density. Being a ratio of two quantities with same units, specific gravity is a pure number independent of the system of units used. The specific gravity of water at the standard temperature (i.e., 4°C), is therefore, equal to 1.0. The specific gravity of mercury varies from 13.5 to 13.6. Knowing the specific gravity of any liquid, its specific weight may be readily calculated by the following relation,

$$w = \text{Sp. gr. of liquid} \times \text{Specific weight of water}$$
$$= (\text{Sp. gr. of liquid}) \times 9\,810 \text{ N/m}^3.$$

1.7 EQUATION OF STATE: THE PERFECT GAS

The density ρ of a particular gas is related to its absolute pressure p and absolute temperature T by the *equation of state*, which for a perfect gas takes the form

$$p = \rho RT; \text{ or } pV = mRT \quad ...(1.2)$$

in which R is a constant called the *gas constant*, the value of which is constant for the gas concerned, and V is the volume occupied by the mass m of the gas. The absolute pressure is the pressure measured above absolute zero (or complete vacuum) and is given by

$$p_{abs} = p_{gage} + p_{atm} \quad \text{(see also Sec. 2.5)}$$

The absolute temperature is expressed in 'kelvin' i.e., K, when the temperature is measured in °C and it is given by

$$T°_{(abs)} = T\,K = 273.15 + t°C$$

No actual gas is perfect. However, most gases (if at temperatures and pressures well away both from the liquid phase and from dissociation) obey this relation closely and hence their pressure, density and (absolute) temperature may, to a good approximation, be related by Eq.1.2. Similarly air at normal temperature and pressure behaves closely in accordance with the equation of state.

It may be noted that the gas constant R is defined by Eq. 1.2 as $p/\rho T$ and, therefore, its dimensional expression is $(FL/M\theta)$. Thus in SI units the gas constant R is expressed in newton-metre per kilogram per kelvin i.e., (N.m/kg. K). Further, since 1 joule = 1 newton × 1 metre, the unit for R also becomes joule per kilogram per kelvin i.e., (J/kg. K). Again, since 1 N = 1 kg × 1 m/s^2, the unit for R becomes (m^2/s^2 K).

In metric gravitational and absolute systems of units, the gas constant R is expressed in kilogram (f)-metre per metric slug per degree C absolute i.e., [kg(f)-m/msl deg. C abs.], and dyne-centimetre per gram (m) per degree C absolute i.e., [dyne-cm/gm(m) deg. C abs.] respectively.

For air the value of R is 287 N-m/kg K, or 287 J/kg K, or 287 m^2/s^2 K.

In metric gravitational system of units the value of R for air is 287 kg(f)-m/msl deg. C abs. Further, since 1 msl = 9.81 kg (m), the value of R for air becomes (287/9.81) or 29.27 kg(f)-m/kg(m) deg. C abs.

Since specific volume may be defined as reciprocal of mass density, the equation of state may also be expressed in terms of specific vloume of the gas as

$$pv = RT \quad ...(1.2\text{ a})$$

in which v is specific volume.

The equation of state may also be expressed as

$$p = wRT \quad ...(1.2\text{ b})$$

in which w is the specific weight of the gas. The unit for the gas constant R then becomes (m/K) or (m/deg. C abs). It may, however, be shown that for air the value of R is 29.27 m/K.

For a given temperature and pressure, Eq. 1.2 indicates that ρR = constant. By Avogadro's hypothesis, all pure gases at the same temperature and pressure have the same number of molecules per unit volume. The density is proportional to the mass of an individual molecule and so the product of R and the 'molecular weight' M is constant for all perfect gases. This product MR is known as the *universal gas constant*. For real gases it is not strictly constant but for monatomic and diatomic gases its variation is slight. If M is the ratio of the mass of the molecule to the mass of a hydrogen atom, MR = 8310 J/kg K.

1.8 VISCOSITY

Viscosity is that property of a fluid by virtue of which it offers resistance to the movement of one layer of fluid over an adjacent layer. It is primarily due to cohesion and molecular momentum exchange between fluid layers, and as flow occurs, these effects appear as shearing stresses between the moving layers of fluid.

Consider two plates (Fig. 1.1) sufficiently large (so that edge conditions may be neglected) placed a small distance Y apart, the space between them being filled with fluid. The lower plate is assumed to be at rest, while the upper one is moved parallel to it with a velocity V by the application of a force F, corresponding to area A, of the moving plate in contact with the fluid. Particles of the fluid in contact with each plate will adhere to it and if the distance Y and velocity V are not too great, the velocity v at a distance y from the lower plate will vary uniformly from zero at the lower plate which is at rest, to V at the upper moving plate. Experiments show that for a large variety of fluids

$$F \sim \frac{AV}{Y}$$

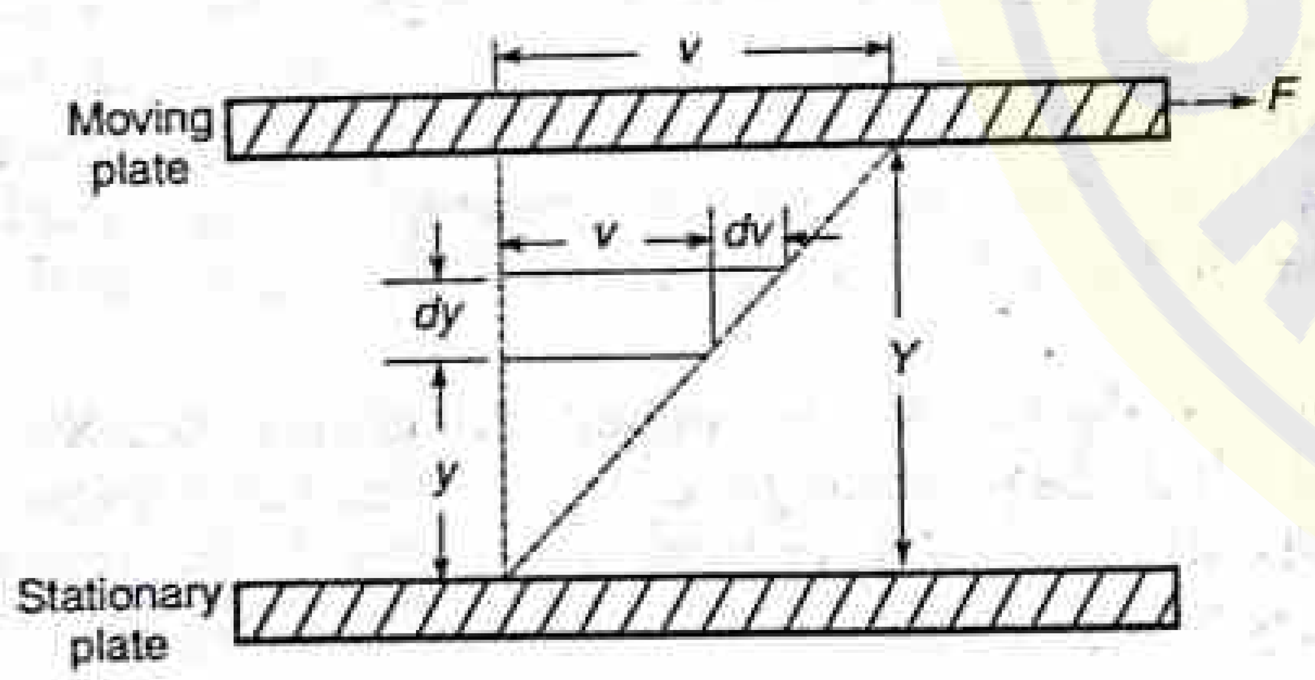


Figure 1.1 *Fluid motion between two parallel plates*

It may be seen from similar triangles in Fig. 1.1 that the ratio V/Y can be replaced by the velocity gradient (dv/dy), which is the rate of angular deformation of the fluid. If a constant of proportionality μ (Greek 'mu') be introduced, the shear strees τ (Greek 'tau') equal to (F/A) between any two thin sheets of fluid may be expressed as

$$\tau = \frac{F}{A} = \mu\frac{V}{Y} = \mu\frac{dv}{dy} \qquad ...(1.3)$$

Equation 1.3 is called Newton's equation of viscosity, and in the transposed form it serves to define the proportionality constant

$$\mu = \frac{\tau}{dv/dy} \qquad ...(1.3\text{ a})$$

which is called the *coefficient of viscosity*, or the *dynamic viscosity* (since it involves force), or simply *viscosity* of the fluid. Thus the dynamic viscosity μ, may be defined as the shear stress required to produce unit rate of angular deformation.

In SI units m is expressed in N.s/m^2, or kg/m.s.

In the metric gravitational system of units, m is expressed in kg(f)-sec/m2 or msl/m-sec. In the metric absolute system of units m is expressed in dyne-sec/m2 or gm(mass)/cm-sec which is also called 'poise' after Poiseuille. The 'centipoise' is one hundredth of a poise.

In the English gravitational system of units m is expressed in lb(f)-sec/ft^2 or slug/ft-sec and in the English absolute system of units it is expressed in pdl-sec/ft^2 or lb(m)/ft-sec.

The numerical conversion from one system to another is as follows:

1 N.s/m^2 = 0.102 kg(f)-sec/m^2 = 10 poise

$$1\frac{\text{lb(f)sec}}{\text{ft}^2} = 479\ \frac{\text{dyne-sec}}{\text{cm}^2} = 479 \text{ poise} = 4.88\ \frac{\text{kg(f)-sec}}{\text{m}^2}$$

In many problems involving viscosity there frequently appears a term dynamic viscosity m divided by mass density ρ. The ratio of the dynamic viscosity μ and the mass density ρ is known as *Kinematic viscosity* and is denoted by the symbol n (Greek 'nu') so that

$$\nu = \frac{\mu}{\rho} \qquad ...(1.4)$$

On analysing the dimensions of the kinematic viscosity it will be observed that it involves only the magnitudes of length and time. The name kinematic viscosity has been given to the ratio (μ/ρ) because kinematics is defined as the study of motion without regard to the cause of the motion and hence it is concerned with length and time only.

In SI units v is expressed in m^2/s. In the metric system of units v is expressed in cm^2/sec or m^2/sec. The unit cm^2/sec is termed as 'stoke' after G.G. Stokes and its one-hundredth part is called 'centistoke'. In the English system of units it is expressed in ft^2/sec.

The numerical conversion from one system to another is as follows:

$$1\ \frac{\text{m}^2}{\text{s}} = 10^4\ \frac{\text{cm}^2}{\text{sec}} = 10^4 \text{ stokes} = 10.764\ \frac{\text{ft}^2}{\text{sec}}$$

The dynamic viscosity μ of either a liquid or a gas is practically independent of the pressure for the range that is ordinarily encountered in practice. However, it varies widely with temperature. For gases, viscosity increases with increase in temperature while for liquids it decreases with increase in temperature. This is so because of their fundamentally different intermolecular characteristics. In liquids the viscosity is governed by the cohesive forces between the molecules of the liquid, whereas in gases the molecular activity plays a dominant role. The kinematic viscosity of liquids and of gases at a given pressure, is essentially a function of temperature.

Common fluids such as air, water, glycerine, kerosene etc., follow Eq. 1.3. There are certain fluids which, however, do not follow Newton's law of viscosity. Accordingly fluids may be classified as Newtonian fluids and non-Newtonian fluids. In a Newtonian fluid there is a linear relation between the magnitude of shear stress and the resulting rate of deformation i.e., the constant of proportionality μ in Eq. 1.3 does not change with rate of deformation. In a non-Newtonian fluid there is a non-linear relation between the magnitude of the applied shear stress and the rate of angular deformation. In the case of a plastic substance which is a non-Newtonian fluid an initial yield stress is to be exceeded to cause a continuous deformation. An ideal plastic has a definite yield stress and a constant linear relation between shear stress and the rate of angular deformation. A thixotropic substance, which is a non-Newtonian fluid, has a non-linear relationship between the shear stress and the rate of angular deformation, beyond an initial yield stress. The printer's ink is an example of a thixotropic liquid.

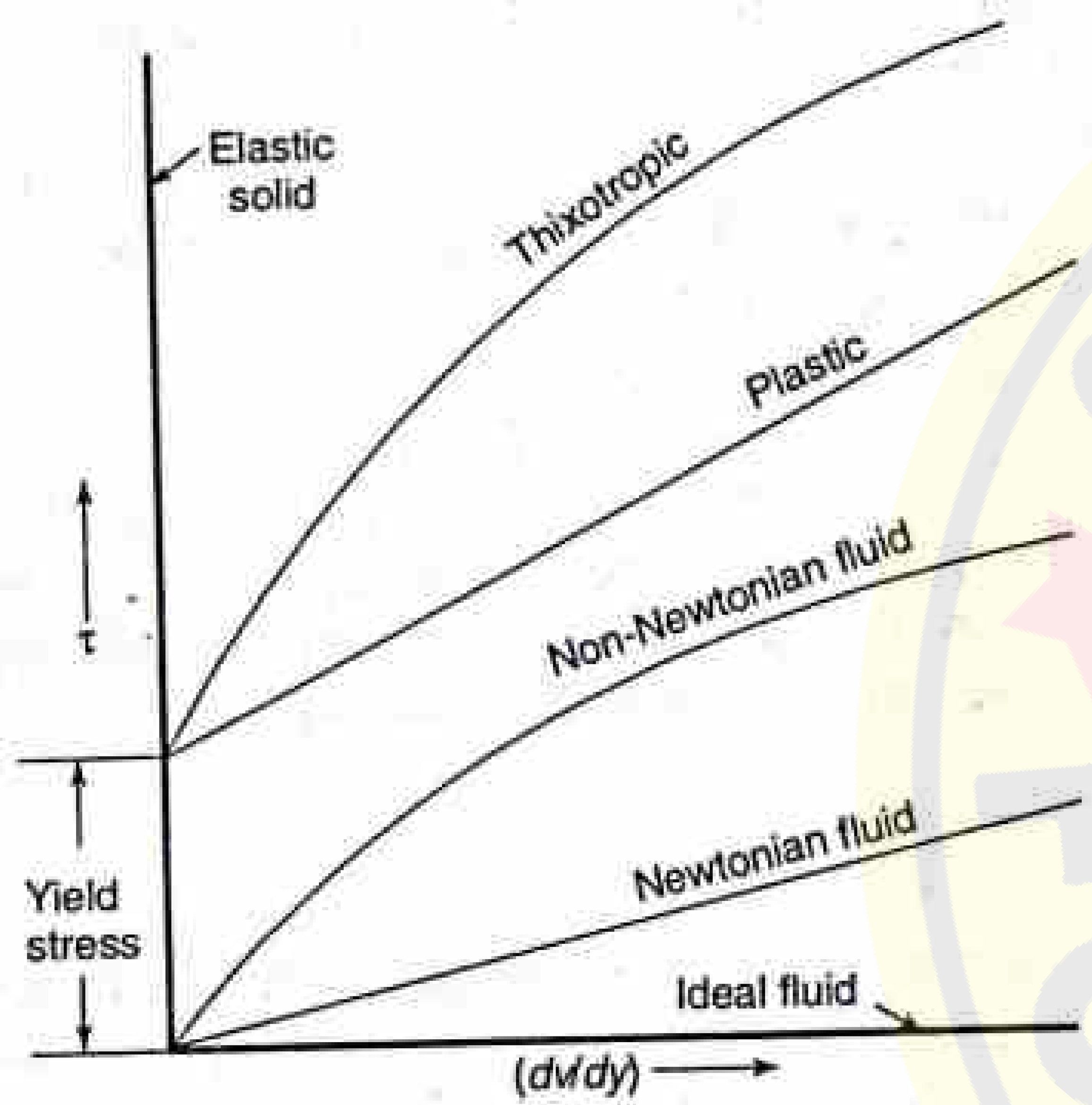


Figure 1.2 *Variation of shear stress with velocity gradient*

An ideal fluid is defined as that having zero viscosity or in other words shear stress is always zero regardless of the motion of the fluid. Thus an ideal fluid is represented by the horizontal axis ($\tau = 0$) in Fig. 1.2 which gives a diagrammatic representation of the Newtonian, non-Newtonian, plastic, thixotropic and ideal fluids. A true elastic solid may be represented by the vertical axis of the diagram.

The fluids with which engineers most often have to deal are Newtonian, that is, their viscosity is not dependent on the rate of angular deformation, and the term 'fluid-mechanics' generally refers only to Newtonian fluids. The study of non-Newtonian fluids is however termed as 'rheology'.

1.9 VAPOUR PRESSURE

All liquids possess a tendency to evaporate or vaporize i.e., to change from the liquid to the gaseous state. Such vaporization occurs because of continuous escaping of the molecules through the free liquid surface. When the liquid is confined in a closed vessel, the ejected vapour molecules get accumulated in the space between the free liquid surface and the top of the vessel. This accumulated vapour of the liquid exerts a partial pressure on the liquid surface which is known as *vapour pressure*

of the liquid. As molecular activity increases with temperature, vapour pressure of the liquid also increases with temperature. If the external absolute pressure imposed on the liquid is reduced by some means to such an extent that it becomes equal to or less than the vapour pressure of the liquid, the boiling of the liquid starts, whatever be the temperature. Thus a liquid may boil even at ordinary temperature if the pressure above the liquid surface is reduced so as to be equal to or less than the vapour pressure of the liquid at that temperature.

If in any flow system the pressure at any point in the liquid approaches the vapour pressure, vaporization of liquid starts, resulting in the pockets of dissolved gases and vapours. The bubbles of vapour thus formed are carried by the flowing liquid into a region of high pressure where they collapse, giving rise to high impact pressure. The pressure developed by the collapsing bubbles is so high that the material from the adjoining boundaries gets eroded and cavities are formed on them. This phenomenon is known as cavitation.

Mercury has a very low vapour pressure and hence it is an excellent fluid to be used in a barometer. On the contrary various volatile liquids like benzene etc., have high vapour pressure.

1.10 COMPRESSIBILITY AND ELASTICITY

All fluids may be compressed by the application of external force, and when the external force is removed the compressed volumes of fluids expand to their original volumes. Thus fluids also possess elastic characteristics like elastic solids. Compressibility of a fluid is quantitatively expressed as inverse of the bulk modulus of elasticity K of the fluid, which is defined as:

$$K = \frac{\text{Stress}}{\text{Strain}} = -\frac{dp}{\left(\frac{dV}{V}\right)} = \frac{\text{Change in pressure}}{\left(\frac{\text{Change in volume}}{\text{Original volume}}\right)} \qquad ...(1.5)$$

Thus bulk modulus of elasticity K is a measure of the incremental change in pressure dp which takes place when a volume V of the fluid is changed by an incremental amount dV. Since a rise in pressure always causes a decrease in volume, dV is always negative, and the minus sign is included in the equation to give a positive value of K.

For example, consider a cylinder containing a fluid of volume V, which is being compressed by a piston. Now if the piston is moved so that the volume V decreases by a small amount dV, then the pressure will increase by amount dp, the magnitude of which depends upon the bulk modulus of elasticity of the fluid, as expressed in Eq. 1.5.

In SI units the bulk modulus of elasticity is expressed in N/m^2. In the metric gravitational system of units it is expressed in either kg(f)/cm^2 or kg(f)/m^2. In the English system of units it is expressed either in lb(f)/in^2 or lb(f)/ft^2. The bulk modulus of elasticity for water and air at normal temperature and pressure is approximately 2.06×10^9 N/m^2 [or 2.1×10^8 kg (f)/m^2] and 1.03×10^5 N/m^2 [or 1.05×10^4 kg (f)/m^2] respectively. Thus air is about 20,000 times more compressible than water. The bulk (volume) modulus of elasticity of mild steel is about 2.06×10^{11} N/m^2 [or 2.1×10^{10} kg(f)/m^2] which shows that water is about 100 times more compressible than steel.

However, the bulk modulus of elasticity of a fluid is not constant, but it increases with increase in pressure. This is so because when a fluid mass is compressed, its molecules become close together and its resistance to further compression increases i.e., K increases. Thus for example, the bulk modulus of water roughly doubles as the pressure is raised from 1 atmosphere to 3500 atmospheres.

The temperature of the fluid also affects the bulk modulus of elasticity of the fluid. In the case of liquids there is a decrease of K with increase of temperature. However, for gases since pressure and temperature are inter-related and as temperature increases, pressure also increases, an increase in temperature results in an increase in the value of K.

For liquids since the bulk modulus of elasticity is very high, the change of density with increase of pressure is very small even with the largest pressure change encountered. Accordingly in the case of liquids the effects of compressibility can be neglected in most of the problems involving the flow of liquids. However, in some special problems such as rapid closure of valve or water hammer, where the changes of pressure are either very large or very sudden, it is necessary to consider the effect of compressibility of liquids.

On the other hand gases are easily compressible and with the change in pressure the mass density of gases changes considerably and hence the effects of compressibility cannot ordinarily be neglected in the problems involving the flow of gases. However, in a few cases where there is not much change in pressure and so gases undergo only very small changes of density, the effects of compressibility may be disregarded e.g., the flow of air in a ventilating system is a case where air may be treated as incompressible.

1.11 SURFACE TENSION AND CAPILLARITY

Due to molecular attraction, liquids possess certain properties such as cohesion and adhesion. *Cohesion* means inter-molecular attraction between molecules of the same liquid. That means it is a tendency of the liquid to remain as one assemblage of particles. *Adhesion* means attraction between the molecules of a liquid and the molecules of a solid boundary surface in contact with the liquid. The property of cohesion enables a liquid to resist tensile stress, while adhesion enables it to stick to another body. *Surface tension* is due to cohesion between liquid particles at the surface, whereas *capillarity* is due to both cohesion and adhesion.

(a) Surface Tension. A liquid molecule on the interior of the liquid body has other molecules on all sides of it, so that the forces of attraction are in equilibrium and the molecule is equally attracted on all the sides, as a molecule at point A shown in Fig. 1.3. On the other hand a liquid molecule at the surface of the liquid, (i.e., at the interface between a liquid and a gas) as at point B, does not have any

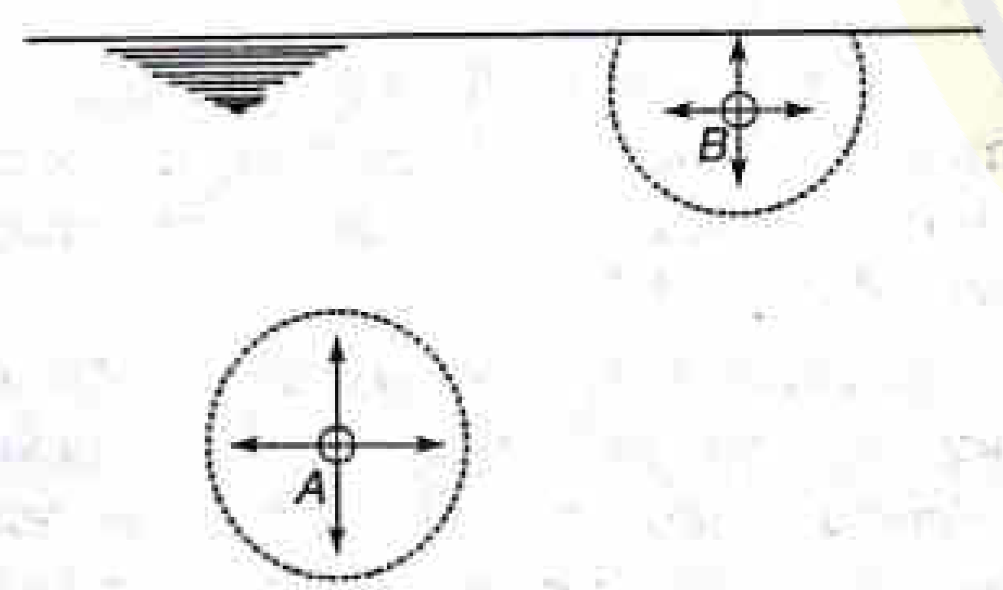


Figure 1.3 *Inter-molecular forces near a liquid surface*

liquid molecule above it, and consequently there is a net downward force on the molecule due to the attraction of the molecules below it. This force on the molecules at the liquid surface, is normal to the liquid surface. Apparently owing to the attraction of liquid molecules below the surface, a film or a special layer seems to form on the liquid at the surface, which is in tension and small loads can be supported over it. For example, a small needle placed gently upon the water surface will not sink but will be supported by the tension at the water surface.

The property of the liquid surface film to exert a tension is called the surface tension. It is denoted by σ (Greek 'sigma') and it is the force required to maintain unit length of the film in equilibrium. In SI units surface tension is expressed in N/m. In the metric gravitational system of units it is expressed in kg(f)/cm or kg(f)/m. In the English gravitational system of units it is expressed in lb(f)/in. or lb(f)/ft.

As surface tension is directly dependent upon inter-molecular cohesive forces, its magnitude for all liquids decreases as the temperature rises. It is also dependent on the fluid in contact with the liquid surface; thus surface tensions are usually quoted in contact with air. The surface tension of water in contact with air varies from 0.0736 N/m [or 0.0075 kg (f)/m] at 19°C to 0.0589 N/m [or 0.006 kg (f)/m] at 100°C. More organic liquids have values of surface tension between 0.0206 N/m [or 0.0021 kg (f)/m] and 0.0304 N/m [or 0.0031 kg (f)/m] and mercury has a value of about 0.4944 N/m [or 0.0504 kg(f)/m], at normal temperature and the liquid in each case being in contact with air.

The effect of surface tension is illustrated in the case of a droplet as well as a liquid jet. When a droplet is separated initially from the surface of the main body of liquid, then due to surface tension there is a net inward force exerted over the entire surface of the droplet which causes the surface of the droplet to contract from all the sides and results in increasing the internal pressure within the droplet. The contraction of the droplet continues till the inward force due to surface tension is in balance with the internal pressure and the droplet forms into sphere which is the shape for minimum surface area. The internal pressure within a jet of liquid is also increased due to surface tension. The internal pressure intensity within a droplet and a jet of liquid in excess of the outside pressure intensity may be determined by the expressions derived below.

(i) *Pressure intensity inside a droplet.* Consider a spherical droplet of radius r having internal pressure intensity p in excess of the outside pressure intensity. If the droplet is cut into two halves, then the forces acting on one half will be those due to pressure intensity σ on the projected area (πr^2) and the tensile force due to surface tension σ acting around the circumference $(2\pi r)$. These two forces will be equal and opposite for equilibrium and hence we have

$$p(\pi r^2) = \sigma(2\pi r)$$

or

$$p = \frac{2\sigma}{r} \quad \ldots(1.6)$$

Equation 1.6 indicates that the internal pressure intensity increases with the decrease in the size of droplet.

(ii) *Pressure intensity inside a soap bubble.* A spherical soap bubble has two surfaces in contact with air, one inside and the other outside, each one of which contributes the same amount of tensile force due to surface tension. As such on a hemispherical section of a soap bubble of radius r the tensile force due to surface tension is equal to $2\sigma\,(2\pi r)$. However, the pressure force acting on the hemispherical section of the soap bubble is same as in the case of a droplet and it is equal to $p\,(\pi r^2)$. Thus equating these two forces for equilibrium, we have

$$p\,(\pi r^2) = 2\sigma\,(2\pi r)$$

or

$$p = \frac{4\sigma}{r} \quad \ldots(1.6\ a)$$

(iii) *Pressure intensity inside a liquid jet.* Consider a jet of liquid of radius r, length l and having internal pressure intensity p in excess of the outside pressure intensity. If the jet is cut into two halves, then the forces acting on one half will be those due to pressure intensity p on the projected area ($2rl$) and the tensile force due to surface tension σ acting along the two sides ($2l$). These two forces will be equal and opposite for equilibrium and hence we have

$$p(2rl) = \sigma(2l)$$

or

$$p = \frac{\sigma}{r} \quad ...(1.6\text{ b})$$

(b) Capillarity. If molecules of certain liquid possess, relatively, greater affinity for solid molecules, or in other words the liquid has greater adhesion than cohesion, then it will wet a solid surface with which it is in contact and will tend to rise at the point of contact, with the result that the liquid surface is concave upward and the angle of contact θ is less than 90° as shown in Fig. 1.4. For example, if a glass tube of small diameter is partially immersed in water, the water will wet the surface of the tube and it will rise in the tube to some height, above the normal water surface, with the angle of contact θ, being zero. The wetting of solid boundary by liquid results in creating decrease of pressure within the liquid, and hence the rise in the liquid surface takes place, so that the pressure within the column at the elevation of the surrounding liquid surface is the same as the pressure at this elevation outside the column.

On the other hand, if for any liquid there is less attraction for solid molecule or in other words the cohesion predominates, then the liquid will not wet the solid surface and the liquid surface will be depressed at the point of contact, with the result that the liquid surface is concave downward and the angle of contact θ is greater than 90° as shown in Fig. 1.4. For instance if the same glass tube is

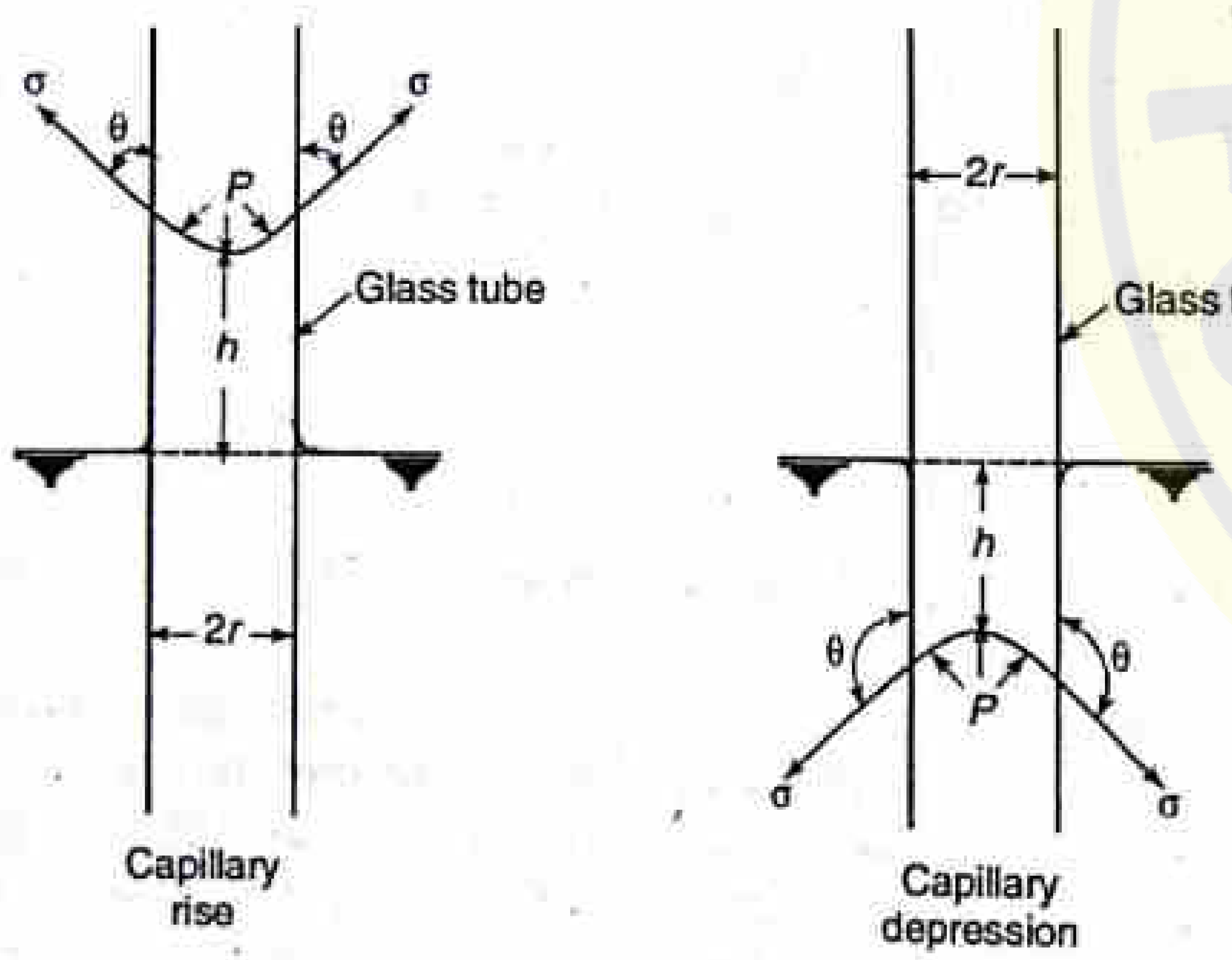


Figure 1.4 *Capillarity in circular glass tubes*

now inserted in mercury, since mercury does not wet the solid boundary in contact with it, the level of mercury inside the tube will be lower than the adjacent mercury level, with the angle of contact θ equal to about 130°. The tendency of the liquids which do not adhere to the solid surface, results in creating an increase of pressure across the liquid surface, (as in the case of a drop of liquid). It is

because of the increased internal pressure, the elevation of the meniscus (curved liquid surface) in the tube is lowered to the level where the pressure is the same as that in the surrounding liquid.

Such a phenomenon of rise or fall of liquid surface relative to the adjacent general level of liquid is known as *capillarity*. Accordingly the rise of liquid surface is designated as capillary rise and the lowering of liquid surface as capillary depression, and it is expressed in terms of m or mm of liquid in SI units, in terms of cm or mm of liquid in the metric system of units and in terms of inch or ft of liquid in the English system of units.

The capillary rise (or depression) can be determined by considering the conditions of equilibrium in a circular tube of small diameter inserted in a liquid. It is supposed that the level of liquid has risen (or fallen) by h above (or below) the general liquid surface when a tube of radius r is inserted in the liquid, see Fig. 1.4. For the equilibrium of vertical forces acting on the mass of liquid lying above (or below) the general liquid level, the weight of liquid column h (or the total internal pressure in the case of capillary depression) must be balanced by the force, at surface of the liquid, due to surface tension σ. Thus equating these two forces we have

$$sw\pi r^2 h = 2\pi r\sigma \cos\theta$$

where w is the specific weight of water, s is specific gravity of liquid, and θ is the contact angle between the liquid and the tube. The expression for h the capillary rise (or depression) then becomes

$$h = \frac{2\sigma\cos\theta}{swr} \quad ...(1.7)$$

As stated earlier, the contact angle θ for water and glass is equal to zero. Thus the value of $\cos\theta$ is equal to unity and hence h is given by the expression

$$h = \frac{2\sigma}{wr} \quad ...(1.8)$$

Equation 1.7 for capillary rise (or depression) indicates that the smaller the radius r the greater is the capillary rise (or depression).

The above obtained expression for the capillary rise (or depression) is based on the assumption that the meniscus or the curved liquid surface is a section of a shpere. This is, however, true only in case of the tubes of small diameters (r < 2.5 mm) and as the size of the tube becomes larger, the meniscus becomes less spherical and also gravitational forces become more appreciable. Hence such simplified solution for computing the capillary rise (or depression) is possible only for the tubes of small diameters. However, with increasing diameter of tube, the capillary rise (or depression) becomes much less. It has been observed that for tubes of diameters 6 mm or more the capillary rise (or depression) is negligible. Hence in order to avoid a correction for the effects of capillarity in manometers, used for measuring pressures, a tube of diameter 6 mm or more should be used.

Another assumption made in deriving Eq. 1.7 is that the liquids and tube surfaces are extremely clean. In practice, however, such cleanliness is virtually never encountered and h will be found to be considerably smaller than that given by Eq. 1.7. In respect of this, Eq. 1.7 will provide a conservative estimate of capillary rise (or depression).

If a tube of radius r is inserted in mercury (sp. gr. s_1) above which a liquid of sp. gr. s_2 lies then by considering the conditions of equilibrium it can be shown that the capillary depression h is given by

$$h = \frac{2\sigma\cos\theta}{rw(s_1 - s_2)} \quad ...(1.9)$$

TABLE 1.8 *Properties of Water at Standard Atmospheric Pressure*

Temperature T		Mass Density ρ		Specific Weight w		Dynamic Viscosity μ			Kinematic Viscosity υ		Bulk Modulus of Elasticity K		Surface Tension (in contact with air) σ		Vapour Pressure Pv	
°C	K	kg/m³	m.slug/m³ or msl/m³	N/m³	kg(f)/m³	N.s/m² or Pa.s	gm(mass)/cm-sec poise	kg(f)-sec/m²	m²/s	cm²/s or stokes	N/m² or Pa	kg(f)/cm²	N/m	kg(f)/m	N/m² or Pa	kg(f)/cm²
						Values below to be multiplied by 10^{-3}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{-4}	Values below to be multiplied by 10^{-6}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{9}	Values below to be multiplied by 10^{4}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{-3}	Values below to be multiplied by 10^{3}	Values below to be multiplied by 10^{-2}
0	273.15	999.9	101.96	9 806	999.9	1.792	1.792	1.827	1.792	1.792	2.04	2.08	7.62	7.77	0.6	0.612
5	278.15	1000.0	101.97	9 807	1000.0	1.519	1.519	1.549	1.519	1.519	2.06	2.10	7.54	7.69	0.9	0.918
10	283.15	999.7	101.94	9 804	999.7	1.308	1.308	1.334	1.308	1.308	2.11	2.15	7.48	7.63	1.2	1.224
15	288.15	999.1	101.88	9 798	999.1	1.140	1.140	1.162	1.141	1.141	2.14	2.18	7.41	7.56	1.7	1.734
20	293.15	998.2	101.79	9 789	998.2	1.005	1.005	1.025	1.007	1.007	2.20	2.24	7.36	7.51	2.5	2.549
25	298.15	997.1	101.68	9 778	997.1	0.894	0.894	0.912	0.897	0.897	2.22	2.26	7.26	7.40	3.2	3.263
30	303.15	995.7	101.53	9 764	995.7	0.801	0.801	0.817	0.804	0.804	2.23	2.27	7.18	7.32	4.3	4.385
35	308.15	994.1	101.37	9 749	994.1	0.723	0.723	0.737	0.727	0.727	2.24	2.28	7.10	7.24	5.7	5.812
40	313.15	992.2	101.18	9 730	992.2	0.656	0.656	0.661	0.661	0.661	2.27	2.31	7.01	7.15	7.5	7.648
45	318.15	990.2	100.97	9 711	990.2	0.599	0.599	0.611	0.605	0.605	2.29	2.34	6.92	7.06	9.6	9.790
50	323.15	988.1	100.76	9 690	988.1	0.549	0.549	0.560	0.556	0.556	2.30	2.35	6.82	6.95	12.4	12.645
55	328.15	985.7	100.51	9 666	985.7	0.506	0.506	0.516	0.513	0.513	2.31	2.36	6.74	6.87	15.8	16.112
60	333.15	983.2	100.26	9 642	983.2	0.469	0.469	0.478	0.477	0.477	2.28	2.32	6.68	6.81	19.9	20.292
65	338.15	980.6	99.99	9 616	980.6	0.436	0.436	0.446	0.444	0.444	2.26	2.30	6.58	6.71	25.1	25.595
70	343.15	977.8	99.71	9 589	977.8	0.406	0.406	0.414	0.415	0.415	2.25	2.29	6.50	6.63	31.4	32.019
75	348.15	974.9	99.41	9 561	974.9	0.380	0.380	0.387	0.390	0.390	2.23	2.27	6.40	6.53	38.8	39.565
80	353.15	971.8	99.10	9 530	971.8	0.357	0.357	0.364	0.367	0.367	2.21	2.25	6.30	6.42	47.7	48.640
85	358.15	968.6	98.77	9 499	968.6	0.336	0.336	0.343	0.347	0.347	2.17	2.21	6.20	6.32	58.1	59.246
90	363.15	965.3	98.43	9 466	965.3	0.317	0.317	0.323	0.328	0.328	2.16	2.20	6.12	6.24	70.4	71.788
95	368.15	961.9	98.09	9 433	961.9	0.299	0.299	0.305	0.311	0.311	2.11	2.15	6.02	6.14	84.5	86.166
100	373.15	958.4	97.73	9 399	958.3	0.284	0.284	0.290	0.296	0.296	2.07	2.11	5.94	6.06	101.3	103.297

Standard Atmospheric Pressure = 101.325 kN/m²; 1 kg(f) = 9.806 65 N; 1 msl = 9.806 65 kg.

TABLE 1.9 *Properties of Some Common Liquids at 20°C and Atmospheric Pressure*

Liquid	Mass Density ρ		Specific Weight w		Dynamic Viscosity μ			Kinematic Viscosity υ		Bulk Modulus of Elasticity K		Surface Tension (in contact with air) σ		Vapour Pressure P_v	
	kg/m³	m.slug/m³ or msl/m³	N/m³	kg(f)/m³	N.s/m² or Pa.s	gm(mass)/cm-sec or poise	kg(f)-s/m²	m²/s	cm²/s or stokes	N/m² or Pa	kg(f)/cm²	N/m	kg(f)/m	N/m² or Pa	kg(f)/cm²
										Values below to be multiplied by 10^{9}	Values below to be multiplied by 10^{4}	Values below to be multiplied by 10^{-2}	Values below to be multiplied by 10^{-3}		
Glycerine	1260	128.48	12356	1260	14.95×10^{-1}	14.95	1.52×10^{-1}	11.87×10^{-4}	11.87	4.35	4.44	6.30	6.42	1.37×10^{-2}	1.40×10^{-7}
Corbon tetra-chloride	1594	162.54	15632	1594	0.97×10^{-3}	0.97×10^{-2}	0.99×10^{-4}	6.04×10^{-7}	6.04×10^{-3}	1.10	1.12	2.67	2.72	1.31×10^{4}	1.34×10^{-1}
Kerosene	800	81.58	7845	800	1.92×10^{-3}	1.92×10^{-2}	1.96×10^{-4}	2.40×10^{-6}	2.40×10^{-2}	1.62	1.65	2.60	2.65	3.30×10^{3}	3.37×10^{-2}
Benzene	879	89.63	8620	879	6.53×10^{-4}	6.53×10^{-3}	6.66×10^{-5}	7.43×10^{-7}	7.43×10^{-3}	1.03	1.05	2.89	2.95	1.00×10^{4}	1.02×10^{-1}
Castor Oil	960	97.89	9414	960	9.80×10^{-1}	9.80	9.99×10^{-2}	10.00×10^{-4}	10.00	1.44	1.47	3.92	4.00	—	—
Ethyl Alcohol	789	80.46	7737	789	1.20×10^{-3}	1.20×10^{-2}	1.22×10^{-4}	1.52×10^{-6}	1.52×10^{-2}	1.21	1.23	2.23	2.27	5.90×10^{3}	6.02×10^{-2}
Mercury	13550	1381.72	132880	13550	1.60×10^{-3}	1.60×10^{-2}	1.63×10^{-4}	1.18×10^{-7}	1.18×10^{-3}	26.20	26.72	51.00	52.00	1.60×10^{-1}	1.63×10^{-6}

Standard atmospheric pressure = 101.325 kN/m²; 1 kg(f) = 9.806 65 N; 1 msl = 9.806 65 kg.

TABLE 1.10 *Properties of Air at Standard Atmospheric Pressure*

Temperature T °C	Temperature T K	Mass Density ρ kg/m³	Mass Density ρ m.slug/m³ or msl/m³	Specific Weight w N/m³	Specific Weight w kg(f)/m³	Dynamic Viscosity μ N.s/m² or pa.s	Dynamic Viscosity μ gm (mass)/cm-sec or poise	Dynamic Viscosity μ kg(f)-sec/m²	Kinematic Viscosity ν m²/s	Kinematic Viscosity ν cm²/s or stokes
-50	223.15	1.582	0.1613	15.514	1.582	1.46×10^{-5}	1.46×10^{-4}	1.489×10^{-6}	0.921×10^{-5}	0.921×10^{-1}
-40	233.15	1.514	0.1544	14.847	1.514	1.51×10^{-5}	1.51×10^{-4}	1.540×10^{-6}	0.998×10^{-5}	0.998×10^{-1}
-30	243.15	1.452	0.1481	14.239	1.452	1.56×10^{-5}	1.56×10^{-4}	1.591×10^{-6}	1.08×10^{-5}	1.08×10^{-1}
-20	253.15	1.394	0.1421	13.670	1.394	1.61×10^{-5}	1.61×10^{-4}	1.642×10^{-6}	1.16×10^{-5}	1.16×10^{-1}
-10	263.15	1.342	0.1368	13.161	1.342	1.67×10^{-5}	1.67×10^{-4}	1.703×10^{-6}	1.24×10^{-5}	1.24×10^{-1}
0	273.15	1.292	0.1317	12.670	1.292	1.72×10^{-5}	1.72×10^{-4}	1.754×10^{-6}	1.33×10^{-5}	1.33×10^{-1}
10	283.15	1.247	0.1272	12.229	1.247	1.76×10^{-5}	1.76×10^{-4}	1.795×10^{-6}	1.42×10^{-5}	1.42×10^{-1}
20	293.15	1.204	0.1228	11.807	1.204	1.81×10^{-5}	1.81×10^{-4}	1.846×10^{-6}	1.51×10^{-5}	1.51×10^{-1}
30	303.15	1.164	0.1187	11.415	1.164	1.86×10^{-5}	1.86×10^{-4}	1.897×10^{-6}	1.60×10^{-5}	1.60×10^{-1}
40	313.15	1.127	0.1149	11.052	1.127	1.91×10^{-5}	1.91×10^{-4}	1.948×10^{-6}	1.69×10^{-5}	1.69×10^{-1}
50	323.15	1.092	0.1114	10.709	1.092	1.95×10^{-5}	1.95×10^{-4}	1.988×10^{-6}	1.79×10^{-5}	1.79×10^{-1}
60	333.15	1.060	0.1081	10.395	1.060	2.00×10^{-5}	2.00×10^{-4}	2.039×10^{-6}	1.89×10^{-5}	1.89×10^{-1}
70	343.15	1.030	0.1050	10.101	1.030	2.05×10^{-5}	2.05×10^{-4}	2.090×10^{-6}	1.99×10^{-5}	1.99×10^{-1}
80	353.15	1.000	0.1020	9.807	1.000	2.09×10^{-5}	2.09×10^{-4}	2.131×10^{-6}	2.09×10^{-5}	2.09×10^{-1}
90	363.15	0.937	0.0992	9.542	0.973	2.13×10^{-5}	2.13×10^{-4}	2.172×10^{-6}	2.19×10^{-5}	2.19×10^{-1}
100	373.15	0.946	0.0965	9.277	0.946	2.17×10^{-5}	2.17×10^{-4}	2.213×10^{-6}	2.30×10^{-5}	2.30×10^{-1}
150	423.15	0.834	0.0850	8.179	0.834	2.38×10^{-5}	2.38×10^{-4}	2.427×10^{-6}	2.85×10^{-5}	2.85×10^{-1}
200	473.15	0.746	0.0761	7.316	0.746	2.57×10^{-5}	2.57×10^{-4}	2.621×10^{-6}	3.45×10^{-5}	3.45×10^{-1}
250	523.15	0.675	0.0688	6.619	0.675	2.75×10^{-5}	2.75×10^{-4}	2.804×10^{-6}	4.08×10^{-5}	4.08×10^{-1}
300	573.15	0.616	0.0628	6.041	0.616	2.93×10^{-5}	2.93×10^{-4}	2.988×10^{-6}	4.75×10^{-5}	4.75×10^{-1}

Standard atmospheric pressure = 101.325 kN/m²; 1 kg(f) = 9.806 65 N; 1 msl = 9.806 65 kg

in which σ is the surface tension of mercury in contact with the liquid and rest of the notation are same as defined earlier.

Further if two vertical parallel plates t distance apart and each of width l are held partially immersed in a liquid of surface tension σ and sp. gr. s, then the capillary rise (or depression) h may be determined by equating the weight of the liquid column h (or the total internal pressure in the case of capillary depression) ($swhlt$) to the force due to surface tension ($2\sigma l\cos\theta$). Thus we have

$$swhlt = 2\sigma l\cos\theta$$

or

$$h = \frac{2\sigma\cos\theta}{swt} \qquad ...(1.10)$$

In Tables 1.8 and 1.9 properties of water and air respectively at different temperatures are listed. Table 1.10 gives the properties of some of the common liquids such as glycerine, kerosene, alcohol, mercury etc., at 20°C.

ILLUSTRATIVE EXAMPLES

Example 1.1. *If 5 m³ of a certain oil weighs 4000 kg(f). Calculate the specific weight, mass density and specific gravity of this oil.*

Solution

$$\text{Specific weight of oil} = \frac{\text{Weight}}{\text{Volume}} = \frac{4000\ \text{kg(f)}}{5\ \text{m}^3} = 800\ \text{kg(f)/m}^3$$

$$\text{Mass density of oil} = \frac{\text{Specific weight of oil}}{\text{Acceleration due to gravity}} = \frac{800\ \text{kg(f)/m}^3}{9.81\ \text{m/sec}^2} = 81.55\ \text{msl/m}^3$$

$$\text{Specific gravity of oil} = \frac{\text{Specific weight of oil}}{\text{Specific weight of water}} = \frac{800\ \text{kg(f)/m}^3}{1000\ \text{kg(f)/m}^3} = 0.8$$

Example 1.2. *If 5 m³ of a certain oil weighs 40 kN, calculate the specific weight, mass density and specific gravity of this oil.*

Solution

$$\text{Specific weight of oil} = \frac{\text{Weight}}{\text{Volume}} = \frac{40\times1000\ \text{N}}{5\ \text{m}^3} = 8000\ \text{N/m}^3$$

$$\text{Mass density of oil} = \frac{\text{Specific weight of oil}}{\text{Acceleration due to gravity}}$$

$$= \frac{8000\ \text{N/m}^3}{9.81\ \text{m/s}^2} = 815.49\ \text{kg/m}^3$$

$$\text{Specific gravity of oil} = \frac{\text{Specific weight of oil}}{\text{Specific weight of water}}$$

$$= \frac{8000\ \text{N/m}^3}{9810\ \text{N/m}^3} = 0.815$$

Example 1.3. *Carbon-tetra chloride has a mass density of 1594 kg/m³. Calculate its mass density, specific weight and specific volume in the metric, and the English gravitational systems of units. Also calculate its specific gravity.*

Solution

Mass density of carbon-tetra chloride $= 1594\ \text{kg/m}^3$

Since $1\ \text{kg} = \frac{1}{9.81}\ \text{msl}$

∴ Mass density of carbon tetra chloride in the metric gravitational system of units

$$= \frac{1594}{9.81} = 162.49\ \text{msl/m}^3$$

Acceleration due to gravity $= 9.81\ \text{m/sec}^2$

∴ Specific weight of carbon-tetra chloride in the metric gravitational system of units $= 162.49 \times 9.81 = 1594\ \text{kg(f)/m}^3$.

Specific volume of carbon-tetra chloride in the metric gravitational system of units

$$= \frac{1}{\text{Specific weight}}$$

$$= \frac{1}{1594} = 6.274 \times 10^{-4}\ \text{m}^3/\text{kg(f)}$$

Since $1\ \text{kg(f)} = 2.205\ \text{lb(f)}$

and $1\ \text{m} = 3.281\ \text{ft}$

∴ Specific weight of carbon-tetra chloride in the English gravitational system of units

$$= \frac{1594 \times 2.205}{(3.281)^3} = 99.51\ \text{lb(f)/ft}^3$$

Acceleration due to gravity $= 32.2\ \text{ft/sec}^2$

∴ Mass density of carbon-tetra chloride in the English gravitational system of units

$$= \frac{99.51}{32.2} = 3.09\ \text{slugs/ft}^3$$

Specific volume of carbon-tetra chloride in the English gravitational system of units

$$= \frac{1}{\text{Specific weight}} = \frac{1}{99.51} = 1.005 \times 10^{-2}\ \text{ft}^3/\text{lb(f)}$$

$$\text{Specific gravity} = \frac{\text{Mass density of carbon tetra chloride}}{\text{Mass density of water}}$$

Mass density of carbon tetra chloride in SI units $= 1594\ \text{kg/m}^3$

Mass density of water in SI units $= 1000\ \text{kg/m}^3$

$$\therefore \text{Specific gravity} = \frac{1594\ \text{kg/m}^3}{1000\ \text{kg/m}^3} = 1.594$$

Example 1.4. *A plate 0.0254 mm distant from a fixed plate, moves at 61 cm/sec and requires a force of 0.2 kg(f)/m² to maintain this speed. Determine the dynamic viscosity of the fluid between the plates.*

Solution

From Eq. 1.3, shear stress

$$\tau = \frac{F}{A} = \mu\frac{dv}{dy} = \mu\frac{V}{Y}$$

$$\tau = \frac{F}{A} = 0.2\ \text{kg(f)/m}^2$$

$$V = 61\ \text{cm/sec} = 0.61\ \text{m/sec}$$

and $Y = 0.0254\ \text{mm} = 2.54 \times 10^{-5}\ \text{m}$

By substituting in the above equation, we get

$$0.2 = \mu \times \frac{0.61}{2.54 \times 10^5}$$

$$\therefore \mu = \frac{0.2 \times 2.54 \times 10^5}{0.61}\ \text{kg(f)-sec/m}^2$$

$$= 8.328 \times 10^{-6}\ \text{kg(f)-sec/m}^2$$

$$= 8.328 \times 10^{-10}\ \text{kg(f)-sec/cm}^2$$

Example 1.5. *At a certain point in castor oil the shear stress is 0.216 N/m² and the velocity gradient 0.216s⁻¹. If the mass density of castor oil is 959.42 kg/m³, find kinematic viscosity.*

Solution

From Eq. 1.3 shear stress

$$\tau = \mu\left(\frac{dv}{dy}\right)$$

$$\tau = 0.216\ \text{N/m}^2;\ \left(\frac{dv}{dy}\right) = 0.216\ \text{s}^{-1}$$

By substitution, we get

$$0.216 = \mu\,(0.216)$$

$\therefore$ $$\mu = 1\ \text{N.s/m}^2$$

$\therefore$ Kinematic viscosity

$$\upsilon = \frac{\mu}{\rho} = \frac{1}{959.42} = 1.042 \times 10^{-3}\ \text{m}^2/\text{s}$$

Example 1.6. *If a certain liquid has viscosity 4.9×10^{-4} kg(f)-sec/m² and kinematic viscosity 3.49×10^{-2} stokes, what is its specific gravity ?*

Solution

Kinematic viscosity $\upsilon = \dfrac{\mu}{\rho}$

or Mass density $\rho = \dfrac{\mu}{\upsilon}$

$$\mu = 4.9 \times 10^{-4}\ \text{kg(f)-sec/m}^2$$

$$\upsilon = 3.49 \times 10^{-2}\ \text{stokes}$$

$$= 3.49 \times 10^{-6}\ \text{m}^2/\text{sec}$$

$\therefore$ $$\rho = \frac{4.9\times10^{4}}{3.49\times10^{6}} = 140.4\ \text{msl/m}^3$$

$\therefore$ Sp. gr. of the liquid $= \dfrac{\text{Mass density of liquid}}{\text{Mass density of water}}$

$$= \frac{140.4}{102} = 1.38$$

Example 1.7. *The kinematic viscosity and specific gravity of a certain liquid are 5.58 stokes (5.58×10^{-4} m²/s) and 2.00 respectively. Calculate the viscosity of this liquid in both metric gravitational and SI units.*

Solution

(a) *Metric gravitational units*

Sp. gr. of the liquid = 2.00

Mass density of water = 102 msl/m³

$\therefore$ Mass density of the liquid = (2 × 102) = 204 msl/m³

Kinematic viscosity of the liquid = 5.58 stokes

$= 5.58 \times 10^{-4}$ m²/sec

$\therefore$ Viscosity of the liquid

$$\mu = \upsilon \times \rho$$

$$= (5.58 \times 10^{-4} \times 204)\ \text{kg(f)-sec/m}^2$$

$$= 0.114\ \text{kg(f)-sec/m}^2$$

(b) *SI units*

Specific gravity of the liquid = 2.00

Mass density of water = 1000 kg/m³

$\therefore$ Mass density of the liquid = (2 × 1000) = 2000 kg/m³

Kinematic viscosity of the liquid = 5.58×10^{-4} m²/s

$\therefore$ Viscosity of the liquid

$$\mu = \upsilon \times \rho$$

$$= (5.58 \times 10^{-4} \times 2000)\ \text{N-s/m}^2$$

$$= 1.116\ \text{N.s/m}^2$$

Example 1.8. *A rectangular plate of size 25 cm by 50 cm and weighing 25 kg(f) slides down a 30° inclined surface at a uniform velocity of 2 m/sec. If the uniform 2 mm gap between the plate and the inclined surface is filled with oil determine the viscosity of the oil.*

Solution

When the plate is moving with a uniform velocity of 2 m/sec, the viscous resistance to the motion is equal to the component of the weight of the plate along the sloping surface. Component of the weight of the plate along the slope = 25 sin 30° = 12.5 kg(f)

Viscous resistance $= (\tau \times A)$

$$= \mu \frac{dv}{dy} \times A = \mu \frac{V}{y} \times A$$

V = 2 m/sec ; $y = 2 \times 10^{-3}$ m; and $A = (0.25 \times 0.5)$ m²

By substituting these values, we get

$$\text{Viscous resistance} = \mu \times \frac{2}{2\times10^{3}}\,(0.25 \times 0.5) = 125\ \mu\text{kg(f)}$$

Equating the two, we get $125\,\mu = 12.5$

$\therefore$ $$\mu = 0.1\ \text{kg(f)-sec/m}^2$$

Example 1.9. *A cubical block of 20 cm edge and weight 20 kg(f) is allowed to slide down a plane inclined at 20° to the horizontal on which there is thin film of oil of viscosity 0.22×10^{-3} kg(f)-s/m². What terminal velocity will be attained by the block if the fill thickness is estimated to be 0.025 mm?*

Solution

The force causing the downward motion of the block is

$$F = W \sin 20° = (20 \times 0.3420) = 6.84\ \text{kg(f)}$$

which will be equal and opposite to shear resistance.

$\therefore$ $$\tau = \frac{F}{A} = \frac{6.84}{(0.20\times0.20)} = 171\ \text{kg(f)/m}^2$$

Further from Eq. 1.3 we have $\tau = \mu\dfrac{dv}{dy} = \mu\dfrac{V}{y}$

$$\mu = 0.22 \times 10^{-3}\ \text{kg(f)-s/m}^2 ;\ y = 0.025\ \text{mm} = 0.025 \times 10^{-3}\ \text{m}$$

Thus by substitution we get $171 = \dfrac{0.22\times10^{-3}\,V}{0.025\times10^{-3}}$

$\therefore$ $$V = 19.43\ \text{m/sec.}$$

Example 1.10. *A cylinder of 0.30 m diameter rotates concentrically inside a fixed cylinder of 0.31 m diameter. Both the cylinders are 0.3 m long. Determine the viscosity of the liquid which fills the space between the cylinders if a torque of 0.98 N.m is required to maintain an angular velocity of 2π rad/s (or 60 r.p.m., since angular velocity $\omega = \frac{2\pi N}{60}$ where N is speed of rotation in r.p.m.).*

Solution

Tangential velocity of the inner cylinder

$$V = r\omega = 0.15 \times 2\pi = 0.942 \text{ m/s}$$

For the small space between the cylinders the velocity profile may be assumed to be linear, then

$$\frac{dv}{dy} = \frac{V}{y} = \frac{0.942}{(0.155-0.15)} = 188.4 \text{ s}^{-1}$$

The torque applied to maintain the constant angular velocity is equal to the torque resisted due to shear stress.

Torque resisted $= \tau \times (2\pi \times 0.15 \times 0.30) \times 0.15$

Thus $0.98 = \tau \times (2\pi \times 0.15 \times 0.30) \times 0.15$

$\therefore \quad \tau = 23.11 \text{ N/m}^2$

From Eq. 1.3 $\quad \tau = \mu \frac{dv}{dy}$

$$\therefore \quad \mu = \frac{\tau}{(dv/dy)} = \frac{23.11}{188.4} = 0.123 \text{ N.s/m}^2$$

Example 1.11. *Through a very narrow gap of height h, a thin plate of large extent is pulled at a velocity V. On one side of the plate is oil of viscosity μ_1 and on the other side oil of viscosity μ_2. Calculate the position of the plate so that (i) the shear force on the two sides of the plate is equal ; (ii) the pull required to drag the plate is minimum.*

Solution

Let y be the distance of the thin plate from one of the surfaces as shown in Fig. Ex. 11.

(i) Force per unit area on the upper surface of the plate

$$= \mu_1 \left(\frac{dv}{dy}\right) = \mu_1 \frac{V}{(h-y)}$$

Force per unit area on the bottom surface of the plate

$$= \mu_2 \left(\frac{dv}{dy}\right) = \mu_2 \frac{V}{y}$$

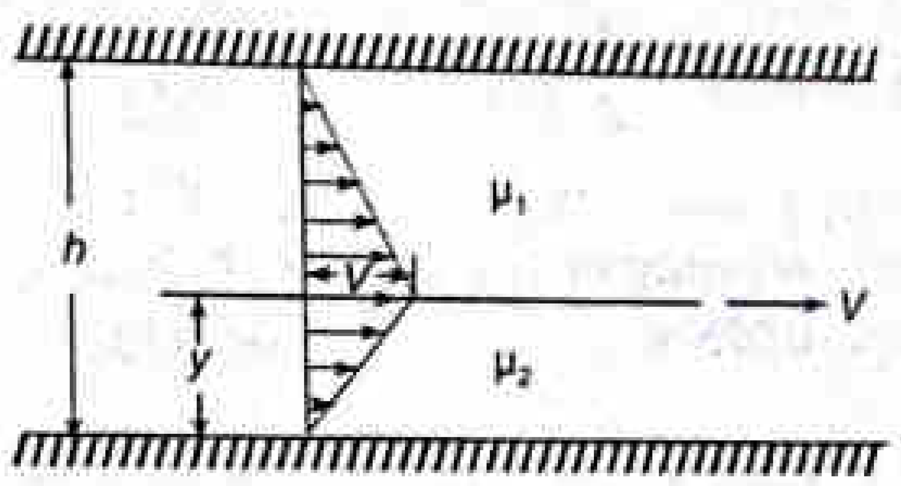


Figure Ex.1.11

Equating the two, we get

$$\mu_1 \frac{V}{h-y} = \mu_2 \frac{V}{y}$$

$$\therefore \quad y = \frac{\mu_2 h}{(\mu_1+\mu_2)}$$

(ii) Let F be the pull per unit area required to drag the plate, then

$$F = \mu_1 \left(\frac{V}{h-y}\right) + \mu_2 \left(\frac{V}{y}\right)$$

F = Sum of the shear forces per unit area on both the surfaces of the plate

For the force F to be minimum

$$\frac{dF}{dy} = 0$$

or $$\frac{dF}{dy} = \frac{\mu_1 V}{(h-y)^2} - \frac{\mu_2 V}{y^2} = 0$$

or $$y = \frac{h}{1+\sqrt{(\mu_1/\mu_2)}}$$

Example 1.12. *If the equation of a velocity profile over a plate is $v = 2y^{2/3}$; in which v is the velocity in m/s at a distance of y metres above the plate, determine the shear stress at y = 0 and y = 0.075 m (or 7.5 cm). Given μ = 0.835 N.s/m² (or 8.35 poise).*

Solution

The velocity profile over the plate is

$$v = 2y^{2/3}$$

$$\therefore \quad \frac{dv}{dy} = 2 \times \frac{2}{3} \times y^{-2/3}\left(\frac{4}{3}\right) y^{-1/3}$$

Shear stress $\quad \tau = \mu\left(\frac{dv}{dy}\right)$

(a) SI units $\tau = 0.835 \times \frac{4}{3}\, y^{-1/3}$

At $y = 0$

$\tau = \infty$ (infinite)

At $y = 0.075$ m

$$\tau = 0.835 \times \frac{4}{3} \times \frac{1}{(0.075)^{1/3}} = 2.64 \text{ N/m}^2$$

(b) Metric gravitational units

$$\mu = 8.35 \text{ poise}$$

$$= \frac{8.35 \times 0.102}{10} \text{ kg(f)-s/m}^2$$

$$= 8.517 \times 10^{-2} \text{ kg(f)-s/m}^2$$

$$\tau = 8.517 \times 10^{-2} \times \frac{4}{3}\, y^{-1/3}$$

At $y = 0$

$\tau = \infty$ (infinite)

At $y = 7.5$ cm $= 0.075$ m

$$\tau = 8.517 \times 10^{-2} \times \frac{4}{3} \times \frac{1}{(0.075)^{1/3}}$$

$$= 0.269 \text{ kg(f)/m}^2$$

Example 1.13. *If the pressure of a liquid is increased from 75 kg(f)/cm² to 140 kg(f)/cm², the volume of the liquid decreases by 0.147 per cent. Determine the bulk modulus of elasticity of the liquid.*

Solution

From Eq. 1.5, bulk modulus of elasticity

$$K = -\frac{dp}{(dV/V)}$$

$$dp = (140 - 75) = 65 \text{ kg(f)/cm}^2$$

and
$$\frac{dV}{V} = -\frac{0.147}{100} = -0.00147$$

$$\therefore \quad K = \frac{65}{0.00147} = 4.42 \times 10^4 \text{ kg(f)/cm}^2$$

Example 1.14. *A liquid compressed in a cylinder has a volume of 0.0113 m³ at 6.87 × 106 N/m² (6.87 MN/m²) pressure and a volume of 0.0112 m³ at 13.73 × 10⁶ N/m² (13.73 MN/m²) pressure. What is its bulk modulus of elasticity?*

Solution

From Eq. 1.5, bulk modulus of elasticity

$$K = -\frac{dp}{(dV/V)}$$

$$dp = (13.73 \times 10^6 - 6.87 \times 10^6) = 6.86 \times 10^6 \text{ N/m}^2$$

$$dV = (0.0112 - 0.0113) = -0.0001/\text{m}^3$$

and
$$V = 0.0113 \text{ m}^3$$

$$\therefore \quad K = \frac{6.86 \times 10^6 \times 0.0113}{0.0001}$$

$$= 7.75 \times 108 \text{ N/m}^2 \text{ (0.775 GN/m}^2\text{)}$$

Example 1.15. *At a depth of 2 kilometres in the ocean the pressure is 840 kg(f)/cm². Assume the specific weight at surface as 1025 kg(f)/m³ and that the average bulk modulus of elasticity is 24 × 10³ kg(f)/cm² for that pressure range. (a) What will be the change in specific volume between that at the surface and at that depth? (b) What will be the specific volume at that depth? (c) What will be the specific weight at that depth?*

Solution

Bulk modulus of elasticity

$$K = -\frac{dp}{(dV/V)}$$

$$K = 24 \times 10^3 \text{ kg(f)/cm}^2$$

and
$$dp = 840 \text{ kg/cm}^2$$

$$\therefore \quad \frac{(dV)}{V} = -\frac{840}{24 \times 10^3} = -0.035$$

The negative sign corresponds to a decrease in the volume with increase in pressure.

The specific volume of the water at the surface of the ocean

$$= \frac{1}{1025} \text{ m}^3\text{/kg(f)}$$

∴ The change in specific volume between that at the surface and at that depth is

$$dV = \frac{0.035}{1025} = 3.41 \times 10^{-5} \text{ m}^3\text{/kg(f)}$$

The specific volume at that depth will be thus equal to

$$V_1 = \left(\frac{1}{1025} - \frac{0.035}{1025}\right) = 9.41 \times 10^{-4} \text{ m}^3\text{/kg(f)}.$$

The specific weight of water at that depth is

$$\frac{1}{V_1} = \frac{1}{9.41 \times 10^4} = 1063 \text{ kg(f)/m}^3.$$

Example 1.16. *What should be the diameter of a droplet of water, if the pressure inside is to be 0.0018 kg(f)/cm² greater than the outside? Given the value of surface tension of water in contact with air at 20°C as 0.0075 kg(f)/m.*

Solution

The internal pressure intensity p in excess of the outside pressure is given by Eq. 1.6 as

$$p = \frac{2\sigma}{r}$$

or
$$r = \frac{2\sigma}{p}$$

or
$$2r = \frac{4\sigma}{p}$$

$$\sigma = 0.0075 \text{ kg (f)/m}$$

and
$$p = 0.0018 \text{ kg(f)/cm}^2$$

By substitution, we get

$$2r = d = 4 \times \frac{0.0075}{100} \times \frac{1}{0.0018} \text{ cm}$$

$$= \frac{4 \times 0.0075 \times 10}{100 \times 0.0018} \text{ mm}$$

$$= 1.67 \text{ mm.}$$

Example 1.17. *What is the pressure within a droplet of water 0.05 mm in diameter at 20°C, if the pressure outside the droplet is standard atmospheric pressure of 1.03 kg(f)/cm² ? Given σ = 0.0075 kg(f)/m for water at 20°C.*

Solution

From Eq. 1.6 the internal pressure intensity p in excess of the outside pressure is given as

$$p = \frac{2\sigma}{r}$$

$$\sigma = 0.0075 \text{ kg(f)/m}$$

$$= \frac{0.0075}{100} \text{ kg(f)/cm}$$

$$r = \frac{0.05}{2} = 0.025 \text{ mm} = \frac{0.025}{10} \text{ cm}$$

By substitution, we get $p = 2 \times \frac{0.0075}{100} \times \frac{10}{0.025} = 0.06 \text{ kg(f)/cm}^2$

The pressure intensity outside the droplet of water = 1.03 kg(f)/cm²

$\therefore$ The pressure intensity within the droplet of water = (1.03 + 0.06) = 1.09 kg(f)/cm².

Example 1.18. *Calculate the capillary rise in a glass tube of 2 mm diameter when immersed in (a) water, (b) mercury. Both the liquids being at 20°C and the values of the surface tensions for water and mercury at 20°C in contact with air are respectively 0.0075 kg(f)/m and 0.052 kg(f)/m.*

Solution

From Eq. 1.7 the capillary rise (or depression) is given as $h = \frac{2\sigma\cos\theta}{swr}$

(a) For water

$$\theta = 0, \cos\theta = 1$$
$$\sigma = 0.0075 \text{ kg(f)/m}$$
$$= \frac{0.0075}{100} \text{ kg(f)/cm}$$
$$sw = 1000 \text{ kg(f)/m}^3$$
$$= 0.001 \text{ kg(f)/cm}^3$$
$$r = 1 \text{ mm} = 0.1 \text{ cm}$$

By substitution, we get $h = 2 \times \frac{0.0075}{100} \times \frac{1}{0.001 \times 0.1} = 1.5$ cm.

(b) For mercury

$$\theta = 130° \times \cos\theta = -0.6428$$
$$\sigma = 0.052 \text{ kg(f)/m}$$
$$= \frac{0.052}{100} \text{ kg(f)/cm}$$
$$sw = (13.6 \times 1000) \text{ kg(f)/m}^3$$
$$= (13.6 \times 0.001) \text{ kg(f)/cm}^3$$
$$r = 1 \text{ mm} = 0.1 \text{ cm}$$

By substitution, we get $h = 2 \times \frac{0.052}{100} \times \frac{(-0.6428)}{13.6 \times 0.001 \times 0.1} = -0.492$ cm.

The negative sign in the case of mercury indicates that there is capillary depression.

Note. Often the value of contact angle for mercury is taken as 180°; in which case $\cos\theta = -1$ and the capillary depression becomes

$$h = -\frac{2 \times 0.052 \times 1}{100 \times 13.6 \times 0.001 \times 0.1} = -0.765 \text{ cm.}$$

Example 1.19. *Determine the minimum size of glass tubing that can be used to measure water level, if the capillary rise in the tube is not to exceed 0.25 cm. Take surface tension of water in contact with air as 0.0075 kg(f)/m.*

Solution

From Eq. 1.8 capillary rise $h = \frac{2\sigma}{swr}$ $\sigma = 0.0075 \text{ kg(f)/m} = \frac{0.0075}{100} \text{ kg(f)/cm}$

$$sw = 1000 \text{ kg(f)/m}^3 = 0.001 \text{ kg(f)/cm}^3$$
$$h = 0.25 \text{ cm}$$

By substitution, we get $0.25 = \frac{2 \times 0.0075}{0.001 \times 100 \times r}$

$\therefore$ $r = 0.6$ cm

Thus minimum diameter of the tube is 1.2 cm.

Example 1.20 *In measuring the unit surface energy of a mineral oil (sp. gr. 0.85) by the bubble method, air is forced to form a bubble at the lower end of a tube of internal diameter 1.5 mm immersed at a depth of 1.25 cm in the oil. Calculate the unit surface energy if the maximum bubble pressure is 15 kg(f)/m².*

Solution

The effective pressure attributable to surface tension is

$$p = \left[15 - \frac{(0.85\times1000\times1.25)}{100}\right] = 4.375 \text{ kg(f)/m}^2$$

From Eq. 1.6 $\qquad p = \dfrac{2\sigma}{r}$

The radius of the bubble is taken equal to that of the tube, thus by substitution, we get

$$4.375 = \frac{2\times\sigma}{0.75\times10^{-3}}$$

$$\therefore \qquad \sigma = 0.001\,64 \text{ kg(f)/m}.$$

Example 1.21. *Calculate the capillary effect in mm in a glass tube 3 mm in diameter when immersed in (a) water (b) mercury. Both the liquids are at 20°C and the values of the surface tensions for water and mercury at 20°C in contact with air are respectively 0.0736 N/m and 0.51 N/m. Contact angle for water = 0° and for mercury = 130°.*

Solution

From Eq. 1.7 the capillary rise (or depression) is given as $h = \dfrac{2\sigma\cos\theta}{swr}$

(a) For water $\theta = 0, \cos\theta = 1$

$$\sigma = 0.0736 \text{ N/m}$$

$$sw = 9810 \text{ N/m}^3$$

$$r = \frac{3}{2} = 1.5 \text{ mm} = 1.5\times10^{-3} \text{ m}$$

By substitution, we get $h = \dfrac{2\times0.0736\times1}{9810\times1.5\times10^{-3}}$

$$= 1.00\times10^{-2} \text{ m}$$

$$= 10 \text{ mm}$$

(b) For mercury $\theta = 130°$, $\cos\theta = -0.6428$

$$\sigma = 0.51 \text{ N/m}$$

$$sw = (13.6\times9810) \text{ N/m}^3$$

$$r = \frac{3}{2} = 1.5 \text{ mm} = 1.5\times10^{-3} \text{ m}$$

By substitution, we get $h = \dfrac{2\times0.51\times(-0.6428)}{13.6\times9810\times1.5\times10^{-3}}$

$$= -3.276\times10^{-3} \text{ m}$$

$$= -3.276 \text{ mm}$$

The negative (–) sign in the case of mercury indicates that there is capillary depression.

SUMMARY OF MAIN POINTS

1. Mass density or specific mass of a fluid is the mass per unit volume of the fluid. It is denoted by ρ.
2. Specific weight or weight density of a fluid is the weight per unit volume of the fluid. It is denoted by w or γ. Thus
$$w \text{ (or } \gamma) = \rho g$$
3. Specific volume of a fluid is the volume per unit weight (or volume per unit mass) of the fluid. It is denoted by v and it is reciprocal of specific weight (or specific mass).
Thus
$$v = \frac{1}{w}; \quad \text{or } v = \frac{1}{\rho}$$
4. Shear stress τ is proportional to the velocity gradient $\dfrac{dv}{dy}$. Thus
$$\tau = \mu\frac{dv}{dy}$$
where μ is the constant of proportionality which is called coefficient of viscosity, or dynamic viscosity.
The SI units of μ are N.s/m², kg/m.s, or poise (= g/cm.s) or Pa.s,
(where Pa = pascal = N/m²).
5. Kinematic viscosity v is given by $v = \dfrac{\mu}{\rho}$.
The SI units of v are m²/s, or stoke (= cm²/s).
6. The mass density ρ of a perfect gas is related to its absolute pressure p and absolute temperature T by equation of state as
$$p = \rho RT$$
where R = gas constant. For air in SI units
$$R = 287\frac{\text{N.m}}{\text{kg°K}} = 287\frac{\text{J}}{\text{kg°K}} = 287\frac{\text{m}^2}{\text{S}^2\text{°K}}$$
and in metric units
$$R = (287/9.81) \text{ or } 29.27 \frac{\text{kg(f)°m}}{\text{kg(m)°K}}$$
7. Compressibility of a fluid is reciprocal of the bulk modulus of elasticity K of the fluid which is defined as $K = \dfrac{dp}{\left(\dfrac{dV}{V}\right)}$.
8. Surface tension is the force required to maintain unit length of a film at the liquid surface in equlibrium. It is denoted by σ.
The relation between surface tension σ and difference of pressure p between inside and outside of :
(i) a liquid drop of radius r is given as
$$p = \frac{2\sigma}{\gamma};$$
(ii) a soap bubble of radius r is given as
$$p = \frac{2\sigma}{r}; \text{ and}$$
(iii) a liquid jet of radius r is given as
$$p = \frac{\sigma}{r}$$
9. Capillary rise (or depression) h of a liquid of specific weight w in a tube of radious r is given by
$$h = \frac{2\sigma\cos\theta}{wr}.$$
The value of contact angle θ for water and glass is equal to zero and for mercury and glass is 130°.

PROBLESMS

1.1 If a certain liquid has a mass density of 129 msl/m^3, what are the values of its specific weight, specific gravity and specific volume in metric gravitational and metric absolute systems of units.

[*Ans.* 1266 kg(f)/m^3; 1.266; 7.9 × 10^{-4} m^3/kg(f); 1242 dynes/cm^3; 1266; 8.05 × 10^{-4} cm^3/dyne]

1.2 If 5.27 m^3 of a certain oil weighs 44 kN, calculate the specific weight ,mass density and specific gravity of the oil.

[*Ans.* 8349 N/m^3; 851.09 kg/m^3; 0.851]

1.3 The specific gravity of a liquid is 3.0, what are its specific weight, specific mass and specific volume.

[*Ans.* 3000 kg(f)/m^3; 305.8 msl/m^3; 0.33 × 10^{-3} m^3/kg(f); 29.43 kN/m^3; 3000 kg/m^3; 3.398 × 10^{-5} m^3/N]

1.4 A certain liquid has a dynamic viscosity of 0.073 poise and specific gravity of 0.87. Compute the kinematic viscosity of the liquid in stokes and also in m^2/s.

[*Ans.* 0.0839 stokes; 0.0839 × 10^{-4} m^2/s]

1.5 If a certain liquid has a viscosity of 0.048 poise and kinematic viscosity 3.50 × 10^{-2} stokes, what is its specific gravity? [*Ans.* 1.371]

1.6 In a stream of glycerine in motion at a certain point the velocity gradient is 0.25 s^{-1}. If for fluid ρ = 129.3 msl/m^3 (1268.4 kg/m^3) and v = 6.30 × 10^{-4} m$^{2/s}$, calculate the shear stress at the point.

[*Ans.* 0.02036 kg (f)/m^2 ; 0.19977 N/m^2]

1.7 If the equation of a velocity distribution over a plate is given by $v = 2y - y^2$, in which v is the velocity in m/s at a distance y, measured in metres above the plate, what is the velocity gradient at the boundary and at 7.5 cm and 15 cm from it? Also determine the shear stress at these points if absolute viscosity μ = 8.60 poise.

[*Ans.* 2 sec^{-1}; 1.85 sec^{-1}; 1.70 sec^{-1}; 0.175 kg(f)/m^2; 0.162 kg(f)/m^2; 0.149 kg(f)m^2]

1.8 A body weighing 441.45 N with a flat surface area of 0.093 m^2 slides down lubricated inclined plane making a 30° angle with the horizontal. For viscosity of 0.1 N.s/m^2 and body speed of 3m/s, determine the lubricant film thickness.

[*Ans.* 0.126 mm]

1.9 A hydraulic lift consists of a 25 cm diameter ram which slides in a 25.015 cm diameter cylinder, the annular space being filled with oil having a kinematic viscosity of 0.025 cm^2/sec and specific gravity of 0.85. If the rate of travel of the ram is 9.15 m/min, find the frictional resistance when 3.05 m of the ram is engaged in the cylinder. [*Ans.* 1.055 kg(f)]

1.10 A cylinder 0.1 m diameter rotates in an annular sleeve 0.102 m internal diameter at 100 r.p.m. The cylinder is 0.2 m long. If the dynamic viscosity of the lubricant between the two cylinders is 1.0 poise, find the torque needed to drive the cylinder against viscous resistance. Assume that Newton's Law of viscosity is applicable and the velocity profile is linear.

[*Ans.* 0.1645 N.m]

1.11 A fluid compressed in a cylinder has a volume of 0.011 32 m^3 at a pressure of 70.30 kg(f)/cm^2. What should be the new pressure in order to make its volume 0.011 21 m^3 ? Assume bulk modulus of elasticity K of the liquid as 703 0 kg(f)/cm^2. [*Ans.* 138.61 kg(f)/cm^2]

1.12 If the volume of a liquid decreases by 0.2 per cent for an increase of pressure from 6.867 MN/m^2 to 15.696 MN/m^2, what is the value of the bulk modulus of the liquid ?

[*Ans.* 44.145 × 108 N/m^2]

1.13 Obtain an expression for the bulk modulus of elasticity K of a fluid in terms of the pressure P, and the mass density ρ. [*Ans.* $K = \rho(dp/d\rho)$]

1.14 A soap bubble 51 mm in diameter has an internal pressure in excess of outside pressure of 0.000 21 kg(f)/cm^2. Calculate the tension in the soap film. [**Hint.** $\rho = (4\sigma/r)$]

[*Ans.* 0.0134 kg(f)/m]

1.15. If the pressure inside a droplet of water is 196.2 N/m^2 in excess of the external pressure, what is the diameter of the droplet? Given the value of surface tension of water in contact with air at 20°C as 0.073 58 N/m. [*Ans.* 1.5 mm]

1.16 A glass tube 0.25 mm in diameter contains mercury column with air above the mercury at 20°C. The surface tension of mercury in contact with air is 0.051 kg(f)/m. What will be the capillary depression of the mercury if angle of contact θ = 130° and sp. gr. of mercury = 13.6.

[*Ans.* 3.86 cm]

1.17 A glass tube 0.25 mm in diameter contains a mercury column with water above the mercury. The temperature is 20°C at which the surface tension of mercury in contact with water is 0.037 kg(f)/m. What will be the capillary depression of the mercury? Take angle of contact θ = 130°.

[**Hint.** Use Eq. 1.9] [*Ans.* 3.02 cm]

1.18 Calculate the capillary rise h in a glass tube of 3 mm diameter when immersed in water at 20°C. Take s for water at 20°C as 0.007 5 kg(f)/m. What will be the percentage increase in the value of h if the diameter of the glass tube is 2 mm? [*Ans.* 10 mm; 50 per cent]

1.19 Show that for two vertical parallel plates t distance apart, held partially immersed in a liquid of surface tension σ and specific weight w, the capillary rise h is given by the expression

$$h = \frac{2\sigma\cos\theta}{wt}$$

in which θ is the angle of contact.

Hence calculate the maximum capillary rise of water at 20°C to be expected between two vertical clean glass plates spaced 1 mm apart. Take σ = 0.073 58 N/m. [*Ans.* 15 mm]

1.20 A capillary tube having inside diameter 5 mm is dipped in water at 20°C. Determine the height of water which will rise in the tube. Take σ = 0.075 gm(f)/cm and θ = 60°. Specific weight of water at 20°C = 998 kg(f)/m^3. [*Ans.* 3 mm]

1.21 By how much does the pressure in a cylindrical jet of water 4 mm in diameter exceed the pressure of the surrounding atmosphere if σ = 0.0075 kg(f)/m.

[**Hint.** Use Eq. 1.6 b]

[*Ans.* 3.75 kg(f)m^2]

1.22 Calculate the capillary effect in mm in a glass tube of 4 mm diameter when immersed in (i) water, and (ii) mercury, both at 20°C. The values of s of water and mercury at 20°C in contact with air are respectively 0.007 5 kg(f)/m and 0.052 kg(f)/m. The contact angle for water θ = 0° and for mercury θ = 130°.

[*Ans.* (i) 7.5 mm; (ii) – 2.46 mm]

1.23 Name the characteristic fluid properties to which the following phenomena are attributable: (i) rise of sap in a tree; (ii) spherical shape of a drop of a liquid; (iii) cavitation; (iv) flow of a jet of oil in an unbroken stream; (v) water hammer.

Fluid Pressure and its Measurement

Chapter 2

2.1 FLUID PRESSURE AT A POINT

'Pressure' or 'intensity of pressure' may be defined as the force exerted on a unit area. If F represents the total force uniformly distributed over an area A, the pressure at any point is $p = (F/A)$. However, if the force is not uniformly distributed, the expression will give the average value only. When the pressure varies from point to point on an area, the magnitude of pressure at any point can be obtained by the following expression

$$p = \frac{dF}{dA}$$

where dF represents the force acting on an infinitesimal area dA. In SI units pressure is expressed in N/m^2 (or pascal), and in metric gravitational units it is expressed in kg(f)/cm² or kg(f)/m².

A fluid is a substance which is capable of flowing. As such when a certain mass of fluid is held in static equilibrium by confining within solid boundaries, it exerts forces against boundary surfaces. The forces so exerted always act in the direction normal to the surface in contact. This is so because a fluid at rest cannot sustain shear stress and hence the forces cannot have tangential components. The normal force exerted by a fluid per unit area of the surface is called the fluid pressure. It may, however, be noted that even if an imaginary surface is assumed within a fluid body, the fluid pressure and pressure force on the imaginary surface are exactly the same as those acting on any real surface. This is in accordance with Newton's third law of motion, viz., action and reaction exist in pairs.

2.2 VARIATION OF PRESSURE IN A FLUID

Consider a small fluid element of size $\delta x \times \delta y \times \delta z$ at any point in a static mass of fluid as shown in Fig. 2.1. Since the fluid is at rest, the element is in equilibrium under the various forces acting on it. The forces acting on the element are the pressure forces on its faces and the self-weight of the element. Let p be the pressure intensity at the mid point O of the element. Then the pressure intensity on the left hand face of the element is $\left[p-\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]$ and the pressure intensity on the right hand face of the element is $\left[p+\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]$. The corresponding pressure forces on the left hand and the right hand

faces of the element are $\left[p-\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]\cdot \delta y \delta z$ and $\left[p+\left(\frac{\partial p}{\partial x}\right)\frac{\delta x}{2}\right]\delta y \delta z$ respectively. Likewise the pressure intensities and the corresponding pressure forces on the other faces of the element may be obtained as shown in Fig. 2.1. Further if w is the specific weight of the fluid then the weight of the element acting vertically downwards is $(w\delta x\, \delta y\, \delta z)$. Since the element is in equilibrium under these forces, the algebraic sum of the forces acting on it in any direction must be zero. Thus considering the forces acting on the element along x, y and z axes the following Equations are obtained

$$\Sigma F_x = 0$$

or

$$\left(p-\frac{\partial p}{\partial x}\frac{\delta x}{2}\right)\cdot \delta y\, \delta z-\left(p+\frac{\partial p}{\partial x}\frac{\delta x}{2}\right)\cdot \delta y \delta z = 0$$

or

$$\frac{\partial p}{\partial x} = 0 \qquad ...(2.1)$$

$$\Sigma F_y = 0$$

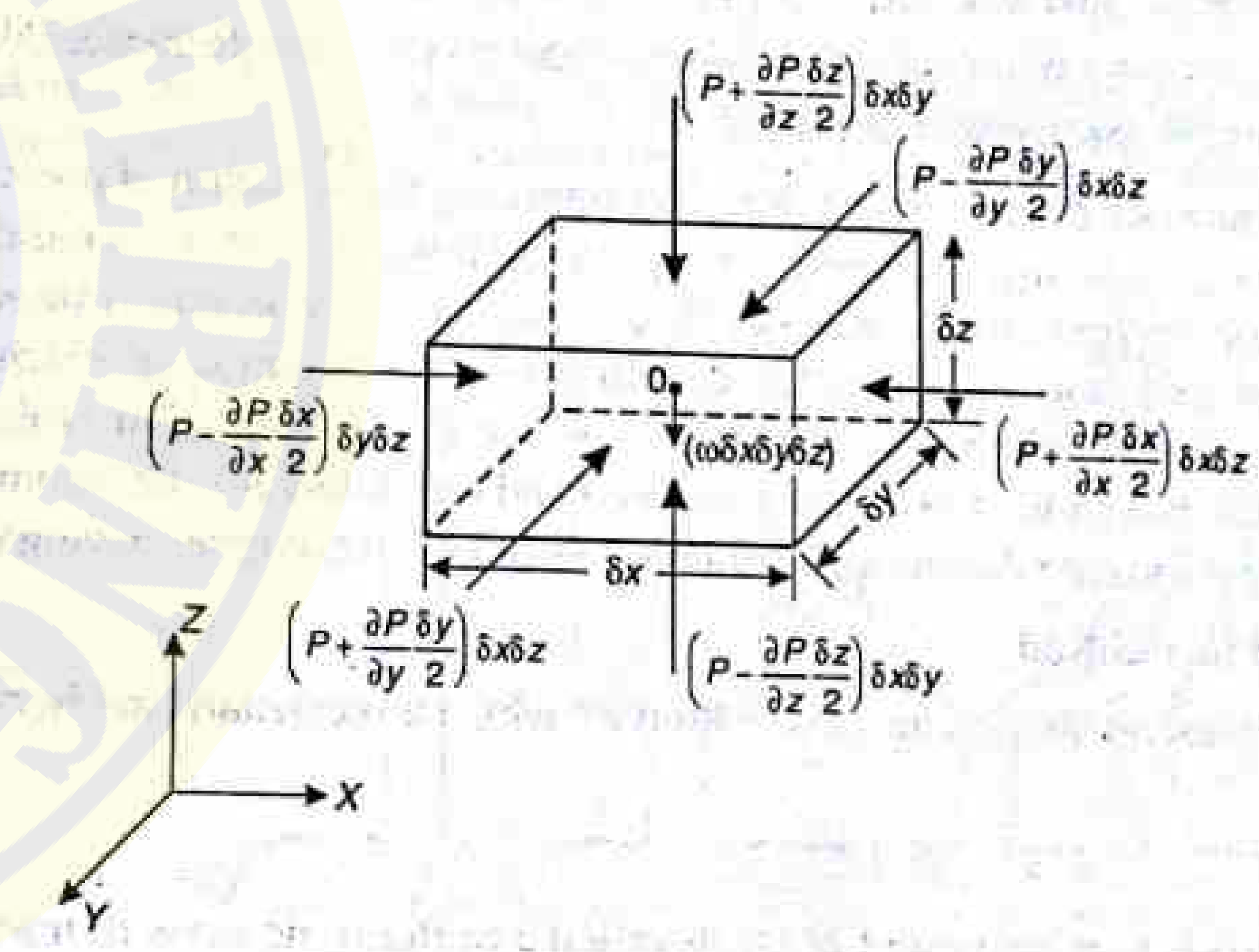


Figure 2.1 *Fluid element with forces acting on it in a static mass of fluid*

or

$$\left(p-\frac{\partial p}{\partial y}\frac{\delta y}{2}\right)\cdot \delta x\, \delta z-\left(p+\frac{\partial p}{\partial y}\frac{\delta y}{2}\right)\cdot \delta x\, \delta z = 0$$

or

$$\frac{\partial p}{\partial y} = 0 \qquad ...(2.2)$$

$$\Sigma F_z = 0$$

$$\left(p - \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x\,\delta y - \left(p + \frac{\partial p}{\partial z}\frac{\delta z}{2}\right)\delta x\,\delta y - (w\,\delta x\,\delta y\,\delta z) = 0$$

or

$$\frac{\partial p}{\partial z} = -w \quad ...(2.3)$$

Equations 2.1, 2.2 and 2.3 indicate that the pressure intensity p at any point in a static mass of fluid does not vary in x and y directions and it varies only in z direction. Hence the partial derivative in eq. 2.3 may be reduced to total (or exact) derivative as follows.

$$\frac{dp}{dz} = -w = -\rho g \quad ...(2.4)$$

In vector notation Eq. 2.4 may be expressed as

$$-\text{grad}\,p = wk = \rho g k$$

where k is unit vector parallel to z axis.

The minus sign (–) in the above equation signifies that the pressure decreases in the direction in which z increases i.e., in the upward direction.

Equation 2.4 is the basic differential equation representing the variation of pressure in a fluid at rest, which holds for both compressible and incompressible fluids. Equation 2.4 indicates that within a body of fluid at rest the pressure increases in the downward direction at the rate equivalent to the specific weight w of the liquid. Further if $dz = 0$, then dp is also equal to zero; which means that the pressure remains constant over any horizontal plane in a fluid. Integration of Eq. 2.4 yields the pressure at any point in a fluid at rest, which is discussed separately for the incompressible fluids such as liquids having constant density and compressible fluids such as gases having variable density.

Pressure at a Point in a Liquid

A liquid may be considered as incompressible fluid for which w is constant and hence integration of Eq. 2.4 gives

$$p = -wz + C \quad ...(2.5)$$

in which p is the pressure at any point at an elevation z in the static mass of liquid and C is the constant of integration. Liquids have a free surface at which the pressure of atmosphere acts. Thus as shown in Fig. 2.2 for a point lying in the free surface of the liquid $z = (H + z_0)$ and if p_a is the atmospheric pressure at the liquid surface then from Eq. 2.5 the constant of integration $C = [p_a + w(H + z_0)]$. Substituting this value of C in Eq. 2.5, it becomes

$$P = -wz + [p_a + w(H + z_0)] \quad ...(2.6)$$

Now if a point is lying in the liquid mass at a vertical depth h below the free surface of the liquid then as shown in Fig. 2.2 for this point $z = (H + z_0 - h)$ and from Eq. 2.6

$$p = p_a + wh \quad ...(2.7)$$

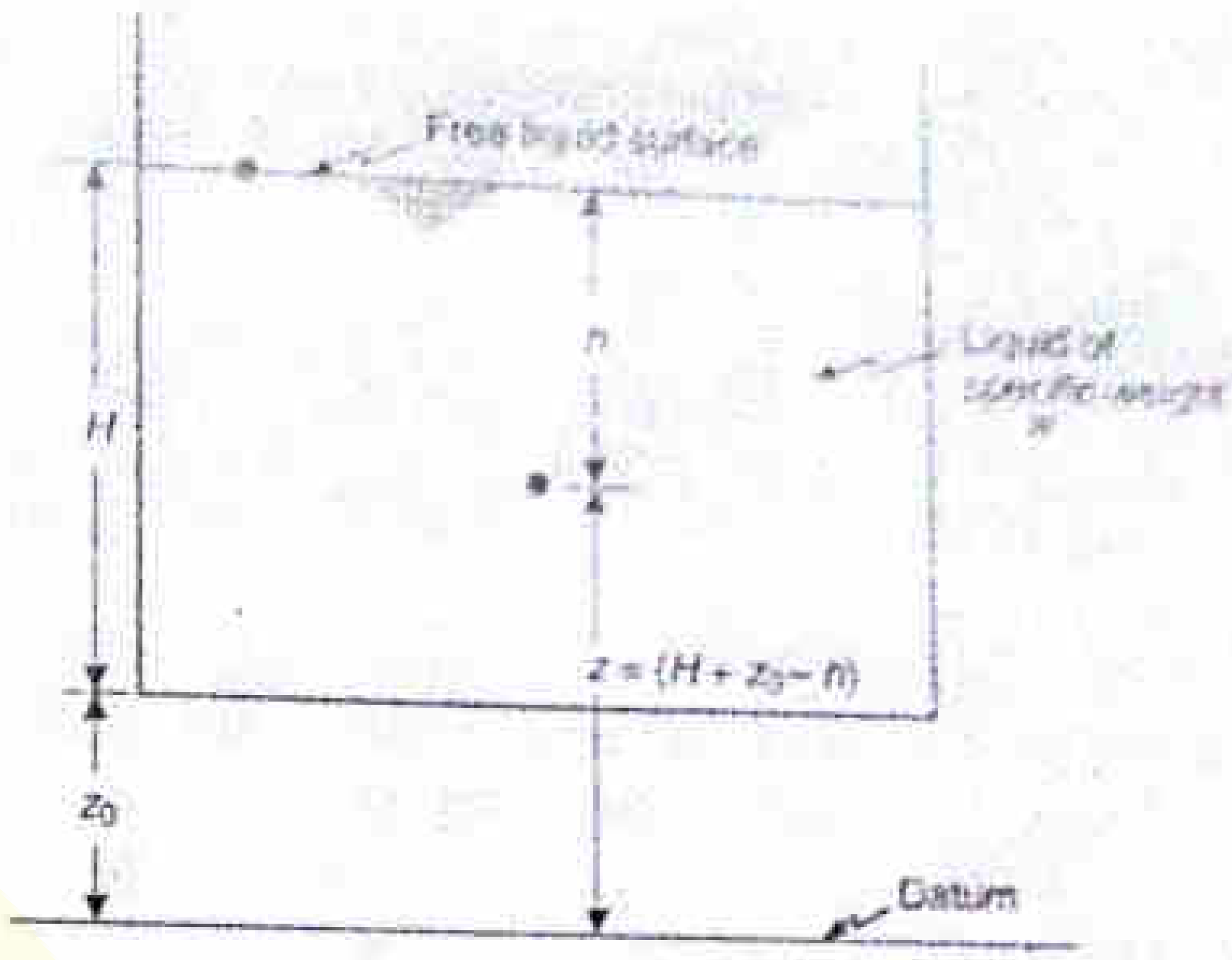


Figure 2.2 *Pressure at a point in a liquid*

It is evident from Eq. 2.7 that the pressure at any point in a static mass of liquid depends only upon the vertical depth of the point below the free surface and the specific weight of the liquid, and it does not depend upon the shape and size of the bounding container. This fact is illustrated in Fig. 2.3 in which although the containers of different shapes are interconnected, so that the total weight

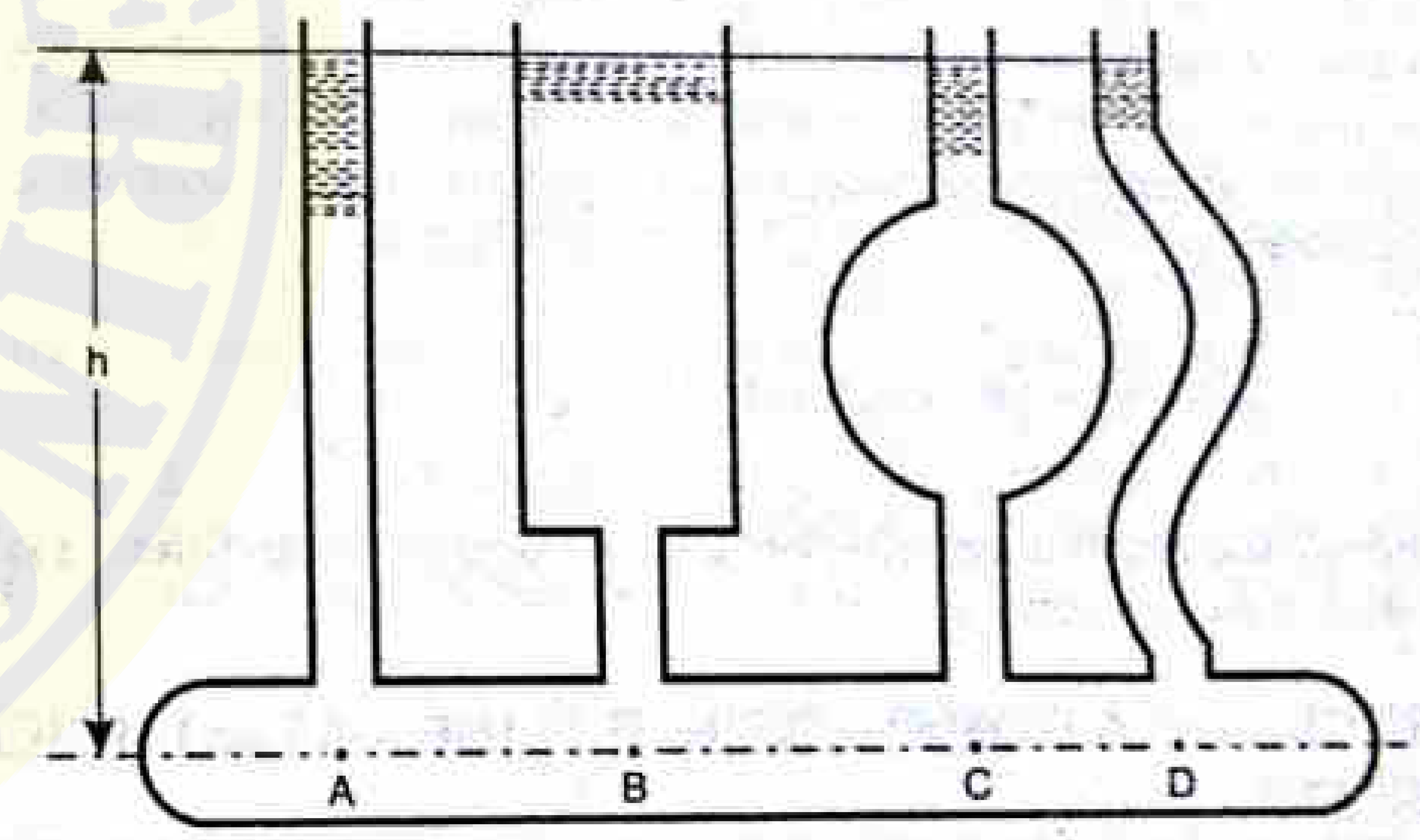


Figure 2.3 *Interconnected containers of different shapes*

of the liquid in each part differs, yet the pressures at points A, B, C and D lying on the same horizontal level and at the same vertical depth h below the free surface of the liquid will be same.

Since the atmospheric pressure at a place is constant, at any point in a static mass of liquid, often only the pressure in excess of the atmospheric pressure is considered, in which case Eq. 2.7 becomes

$$p = wh \quad ...(2.8)$$

Pressure Head. The vertical height of the free surface above any point in a liquid at rest is known as *pressure head*. From Eq. 2.8 the pressure head may be expressed as

$$h = \frac{p}{w} \quad ...(2.9)$$

Since the pressure at any point in a liquid depends on the height of the free surface above the point, it is convenient to express a fluid pressure in terms of pressure head. The pressure is then expressed in terms of metres (or centimetres) of a liquid column.

Equation 2.8 can be used to obtain a relationship between the heights of columns of different liquids which would develop the same pressure at any point. Thus, if h_1 and h_2 are the heights of the columns of liquids of specific weights w_1 and w_2 required to develop the same pressure p, at any point; then from Eq. 2.8

$$p = w_1h_1 = w_2h_2 \quad ...(2.10)$$

If S_1 and S_2 are the specific gravities of the two liquids and w is the specific weight of water then since $w_1 = S_1 w$ and $w_2 = S_2 w$, Eq. 2.10 may also be written as

$$S_1h_1 = S_2h_2 \quad ...(2.10\ a)$$

Further if h_1 and h_2 are the depths of two points below the free surface in a static mass of liquid of specific weight w and p_1 and p_2 are the respective pressure intensities at these points, then from Eq. 2.8 the pressure difference between these points is obtained as

$$(p_1 - p_2) = w(h_1 - h_1)$$

Thus it may be stated that the difference in pressure at any two points in a static mass of liquid varies directly as the difference in depth (or elevation) of the two points.

Pressure at any Point in a Compressible Fluid. For a compressible fluid since the density varies with the pressure, Eq. 2.4 can be integrated only if the relation between w and p is known. Moreover, in a compressible fluid there being no free surface, the integration of Eq. 2.4 gives the variation of pressure between any two points lying in a static mass of fluid. Thus if p_1 and p_2 are the pressure intensities at two points which are at elevations z_1 and z_2 above an arbitrarily assumed datum, then integration of Eq. 2.4 yields

$$\int_{p_1}^{p_2} \frac{dp}{w} = -\int_{z_1}^{z_2} dz = (z_1 - z_2)$$

The left hand side of the above expression has been evaluated using different relations between w and p as indicated in the Section 2.3.

2.3 EQUILIBRIUM OF A COMPRESSIBLE FLUID—ATMOSPHERIC EQUILIBRIUM

Equation 2.4 expresses the conditions for equilibrium of any fluid, which may be written as

$$dp = -wdz = -\rho g dz \quad ...(2.4)$$

This is a general relationship which can be applied to both incompressible as well as compressible fluids. However, for a compressible fluid the above equation can be integrated, to obtain the value of pressure at any point in the fluid, provided it can be assumed that the mass density ρ is either constant or it is a function of pressure (or elevation) only. The mass density ρ may be assumed to be constant if the variation in elevation is not great. But if the variation in elevation is great, ρ can no longer be considered constant, in which case the manner in which ρ varies with pressure p (or elevation) must be known. The variation of pressure and mass density of the air in the atmosphere with elevation is one such problem which is encountered in aeronautics and meteorology. Similarly in oceanography too the problem of the variation of pressure and mass density with depth is of concern, since at great depth there is a small increase in the density of the fluid.

In order to obtain the variation of pressure with elevation in the atmosphere, some of the relations between p and ρ which may be used in Eq. 2.4 are as described below.

(a) Isothermal State for Atmosphere or Isothermal Atmosphere. If it is assumed that the variation of p and ρ is according to isothermal condition, i.e., the temperature of atmosphere is assumed to have a constant value (which may however be true only over a relatively small vertical distance), then according to Boyle's law

$$\frac{p}{\rho} = \frac{p_0}{\rho_0} \quad ...(2.11)$$

where p_0 and ρ_0 are the values of the pressure and density of the gas at initial condition at some reference level, for example at the earth's surface i.e., at $z = 0$.

From Eq. 2.4

$$\frac{dp}{\rho} = -gdz$$

By substituting for ρ from Eq. 2.11

$$\frac{dp}{p}\frac{p_0}{\rho_0} = -gdz$$

Since the value of g decreases by only about 0.1% for about 300 m increase in altitude it may be assumed constant. Thus integration of above equation gives

$$-gz = \frac{p_0}{\rho_0}\log_e p + C$$

If p_1 is the pressure at height z_1, then

$$-gz_1 = \frac{p_0}{\rho_0}\log_e p_1 + C$$

Thus eliminating C from these equations

$$(z - z_1) = \frac{p_0}{\rho_0 g}\log_e\left(\frac{p_1}{p}\right) \quad ...(2.12)$$

Equation 2.12 expresses the relation between altitude and pressure when the air is isothermal. The ratio $(p_0/\rho_0 g)$ represents the height of a fluid column of constant specific weight $(\rho_0 g)$. It is called the *equivalent height of a uniform atmosphere.*

Further at $z = 0$ since $p = p_0$ the integration constant

$$C = -\frac{p_0}{\rho_0}\log_e p_0$$

Hence

$$z = -\frac{p_0}{\rho_0 g}\log_e\left(\frac{p}{p_0}\right) \quad ...(2.13)$$

For a perfect gas from equation of state

$$p_0 = \rho_0 RT_0 \ ; \ \text{or} \ \frac{p_0}{\rho_0} = RT_0$$

Thus substituting the values of (p_0/ρ_0) in Eq. (2.13)

$$z = -\frac{RT_0}{g}\log_e\left(\frac{p}{p_0}\right)$$

or

$$\frac{p}{p_0} = \exp\left(-\frac{gz}{RT_0}\right) \qquad ...(2.14)$$

From Eq. 2.14 it may be seen that when (gz/RT_0) is small this expression approximates to that for a fluid of constant density.

(b) Adiabatic State for Atmosphere or Adiabatic (or Isentropic) Atmosphere. It is well-known fact that the temperature in the atmosphere decreases rapidly as one goes to higher altitude. Thus if it is assumed that no heat is added or taken away from a certain column of air, then this column is said to be under adiabatic conditions.

For an adiabatic process

$$\frac{p}{\rho^k} = \frac{p_0}{\rho_0^k} ; \ \text{or} \ \rho = \rho_0\left(\frac{p}{p_0}\right)^{1/k} \qquad ...(2.15)$$

in which k is the adiabatic exponent (or adiabatic index) defined as the ratio of the specific heat at constant pressure C_p to the specific heat at constant volume C_v

Substituting this value in Eq. 2.4 one obtains

$$-gdz = \frac{dp}{\rho_0}\left(\frac{p_0}{p}\right)^{1/k}$$

or

$$-dz = \frac{1}{\rho_0 g}\left(\frac{p_0}{p}\right)^{1/k} dp$$

or

$$-dz = \frac{p_0}{\rho_0 g}\left(\frac{p}{p_0}\right)^{-\frac{1}{k}} d\left(\frac{p}{p_0}\right)$$

As stated earlier g may be assumed to be constant and hence by integrating the above expression one obtains

$$-z = \frac{p_0}{\rho_0 g}\left(\frac{p}{p_0}\right)^{1-\frac{1}{k}}\left(\frac{1}{1-\frac{1}{k}}\right) + C$$

if $p = p_0$ when $z = z_0$ then integration constant

$$C = -z_0 - \left(\frac{k}{k-1}\right)\frac{p_0}{\rho_0 g}$$

Hence

$$(z - z_0) = \left(\frac{k}{k-1}\right)\frac{p_0}{\rho_0 g}\left[1-\left(\frac{p}{p_0}\right)^{\frac{k-1}{k}}\right] \qquad ...(2.16)$$

from which

$$\frac{p}{p_0} = \left[1-\Delta z\frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right]^{\frac{k}{k-1}} \qquad ...(2.17)$$

where $(z - z_0) = \Delta z$

Substituting the value of (p/p_0) in Eq. (2.15)

$$\frac{\rho}{\rho_0} = \left[1-\Delta z\frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right]^{\frac{1}{k-1}} \qquad ...(2.18)$$

Again for a perfect gas from equation of state

$$p_0 = \rho_0 RT_0 ; \ \text{or} \ \frac{p_0}{\rho_0} = RT_0$$

Substituting the value of (p_0/ρ_0) in Eq. (2.17)

$$\frac{p}{p_0} = \left[1-\Delta z\frac{g}{RT_0}\left(\frac{k-1}{k}\right)\right]^{\frac{k}{k-1}} \qquad ...(2.19)$$

If z is small, all but the first two terms of the binomial expression of the right hand side of Eq. 2.17 may be neglected and then

$$\frac{p}{p_0} = 1 - \frac{\rho_0 g \Delta z}{p_0}$$

or

$$p = p_0 - \rho_0 g\,\Delta z$$

This corresponds to the relationship $p + wz$ = constant (eq. 2.5) for fluid of constant density. Thus for small differences of height say less than about 300 m in the atmosphere, the fluid may be considered to be of constant density without an appreciable error being introduced.

Further dividing Eq. 2.17 by Eq. 2.18 one obtains

$$\frac{p}{\rho} = \frac{p_0}{\rho_0}\left[1-\Delta z\frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right]$$

or
$$RT = \frac{p_0}{\rho_0}\left[1-\Delta z\frac{\rho_0 g}{p_0}\left(\frac{k-1}{k}\right)\right] \quad ...(2.20)$$

since $\frac{p}{\rho} = RT$ from the eq. of state.

Equation 2.20 gives the relation between the absolute temperature T and altitude z. Now if $T = T_0$ at $z = z_0$ (or $\Delta z = 0$) then from Eq. 2.20

$$RT_0 = \frac{p_0}{\rho_0}$$

Further if $(T_0 - T) = \Delta T$ for a difference in elevation Δz then

$$R(\Delta T) = g(\Delta z)\left(\frac{k-1}{k}\right) \quad ...(2.21)$$

For air, R = 29.27 m-kg(f)/kg(m)°C absolute and k = 1.405, then for Δz = 100 m, the variation in temperature ΔT may be obtained from Eq. 2.21 as

$$R(\Delta T) = \frac{9.81\times100\times0.405}{1.405}$$

$$R = 29.27 \text{ m-kg (f)/kg (m)°C absolute}$$
$$= 29.27 \times 9.81 \text{ m}^2/\text{sec}^2 \text{ °C absolute}$$

$$\therefore \quad \Delta T = \frac{9.81\times100\times0.405}{29.27\times9.81\times1.405} = 0.985 \text{ °C}.$$

This shows that under adiabatic condition, the absolute temperature decreases by about 0.985°C for each 100 m increase in elevation. However, if the atmosphere is stable, the temperature drop with altitude is generally somewhat less than that computed above.

(c) Polytropic State for Atmosphere or Polytropic Atmosphere. In a more generalized way the variation of atmospheric pressure p with ρ may be considered according to a polytropic process, in which case, we have

$$\frac{p}{\rho^n} = \frac{p_0}{\rho_0^n}; \text{ or } \rho = \rho_0\left(\frac{p}{p_0}\right)^{1/n}$$

where n is a positive constant.

Again substituting this value in Eq. 2.4 and integrating as indicated earlier the following expression may be obtained

$$\frac{p}{p_0} = \left[1-\frac{g(z-z_0)}{RT_0}\frac{n-1}{n}\right]^{\frac{n}{n-1}} \quad ...(2.22)$$

Evidently Eq. 2.22 is more general being applicable for any value of n, except for the particular case of $n = 1.0$; and from this, eq. 2.19 can be obtained by considering $n = k$. Equation 2.22 represents the variation of pressure with elevation, which has been plotted in Fig. 2.4 for different values of n. The actual variation of pressure with elevation in the atmosphere is also plotted in Fig. 2.4, from which it may be seen that in the atmosphere the actual value of n usually varies between n = 1.2 (wet adiabatic process) to n = 1.4 (dry adiabatic process). As shown in Fig. 2.4 the value of n depends on the *temperature lapse rate* $(\partial T/\partial z)$. Combining Eq. 2.22 with the equation of state $p = \rho RT$ and the equation (p/ρ^n) = constant, we have

$$\frac{T}{T_0} = \left[1-\frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right] \quad ...(2.23)$$

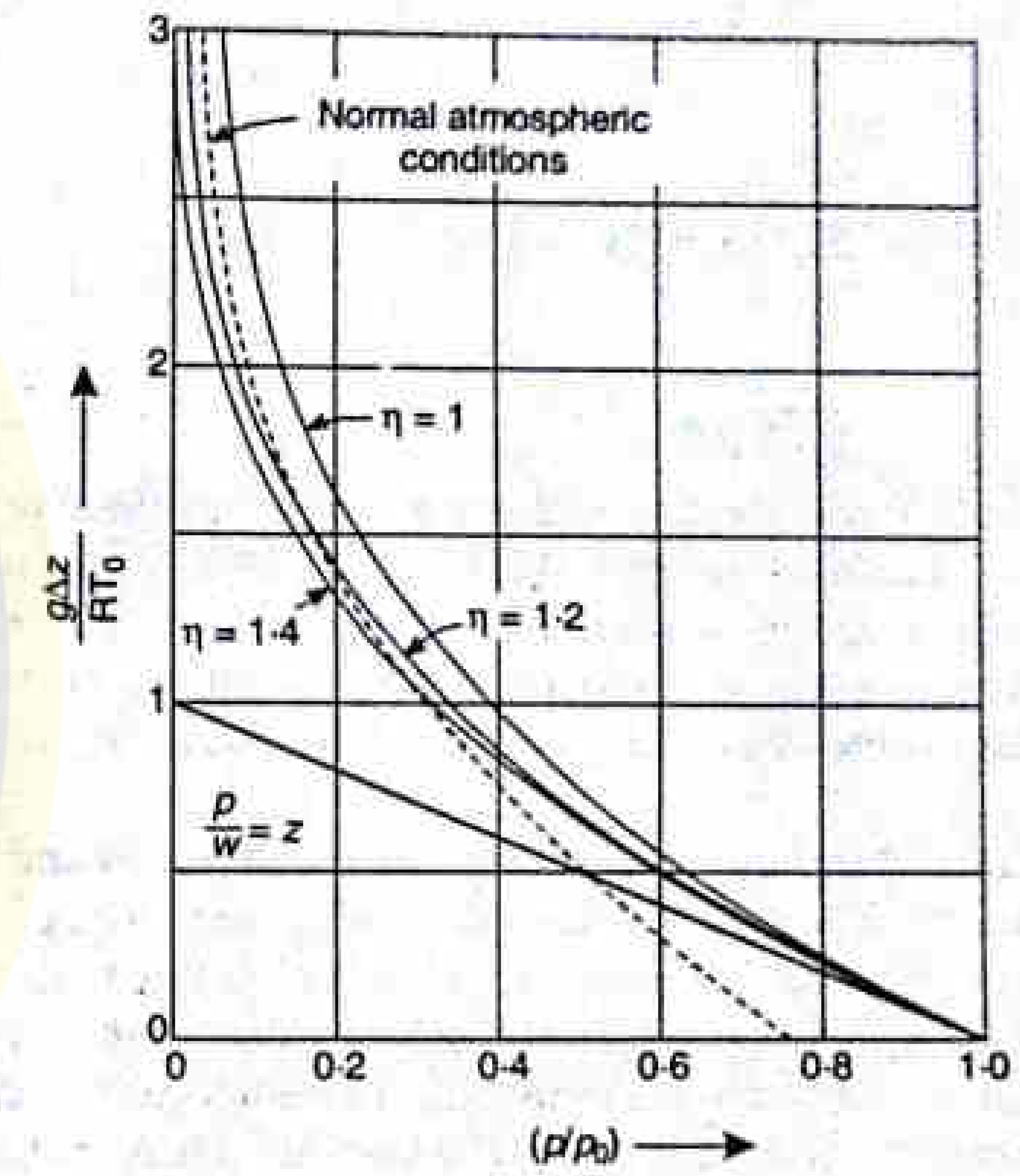


Figure 2.4 *Variation of pressure with elevation in the atmosphere*

Differentiating with respect to z, we obtain

$$\frac{1}{T_0}\frac{\partial T}{\partial z} = -\frac{g}{RT_0}\left(\frac{n-1}{n}\right)$$

or
$$\frac{g}{R}\left(\frac{n-1}{n}\right) = -\frac{\partial T}{\partial z} = \lambda$$

where $\lambda = -\left(\frac{\partial T}{\partial z}\right)$

$$\therefore \quad n = \frac{1}{1-(R\lambda/g)} \qquad ...(2.24)$$

For the first 11 000 m above the ground the temperature in the atmosphere decreases uniformly, that is $-(\partial T/\partial z) = \lambda$ = constant. The observed value of λ in this region of atmosphere is about 6.56°C/ 1000 m

$$\therefore \quad \frac{R\lambda}{g} = \frac{29.27 \text{ m kg(f)}}{\text{kg(m)°C}} \times \frac{6.56°\text{C}}{1000 \text{ m}} \times \frac{1}{9.81} \frac{\text{sec}^2}{\text{m}}$$

$$= \frac{29.27 \times 0.00656}{9.81} \frac{\text{kg(f)-sec}^2}{\text{kg(m) m}}$$

Since 1 kg(f) = 9.81 kg (m) $\frac{\text{m}}{\text{sec}^2}$

$$\therefore \quad \frac{R\lambda}{g} = 29.27 \times 0.006\,56 = 0.192$$

and $$n = \frac{1}{1-0.192} = 1.238.$$

From about 11 000 m to 32 000 m the temperature in the atmosphere remains constant at about −56.5°C in which case $n = 1$, that is, isothermal condition may be assumed and then Eq. 2.14 applies. Beyond 32 000 m the temperature rises again. Since n depends on the temperature lapse rate λ, which varies with the altitude, it is obvious that in the atmosphere n varies with altitude. As such the assumption of a constant value of n for all the altitudes in the atmosphere may lead to erroneous results.

(d) Standard Atmosphere. It is very well known that the densities and the temperatures in the atmosphere vary continuously. They change from day to day and from place to place on the earth. In addition, moisture too plays a part in this variation, though it has a relatively small influence on density ρ and hence it is generally neglected in practical calculations. For aeronautical purpose, particularly for comparison of aircraft performance at different locations and on different days the difficulties that may arise from variations in ρ must be avoided. Therefore, an international *standard atmosphere* has been chosen. The standard atmosphere is defined by certain values of n, p_0 and T_0 which provides a set of data reasonably representative of the actual atmosphere. The standard atmosphere as approved by ICAO (International Civil Aviation Organisation) is based on the following values at sea level : p_o = 10 332 kg(f)/m^2 ; t_0 = 15°C ; w_0 = 1.226 kg(f)/m^3 ; ρ_0 = 0.125 msl/m^3 ; R = 29.27 m.kg(f)/kg(m) °C. In the lower atmosphere or *troposphere* the temperature of the air decreases linearly with altitude at an average rate of λ = 6.56° C/1000 m, which continues upto an altitude of about 11 000 m. For a considerable distance above this elevation the temperature of air remains constant at about −56.5°C. This upper region where constant temperature prevails is usually referred to as the *stratosphere*, which extends from about 11000 m to 32000 m. However, there is no sharp demarcation between these two strata, but the transition from one to the other is rather gradual. This zone of transition is called the *tropopause*. As discussed earlier, in the troposhpere the functional relation between p and ρ may be assumed to be polytropic with n = 1.238 while in the stratosphere on account of the temperature being constant the functional relation between p and ρ is isothermal.

2.4 PRESSURE, SAME IN ALL DIRECTIONS — PASCAL'S LAW

The pressure at any point in a fluid at rest has the same magnitude in all directions. In other words, when a certain pressure is applied at any point in a fluid at rest, the pressure is equally transmitted in all the directions and to every other point in the fluid. This fact was established by B. Pascal, a French Mathematician in 1653, and accordingly it is known as Pascal's Law.

To prove this statement, consider an infinitesimal wedge shaped element of fluid at rest as a free body. The element is arbitrarily chosen and has the dimensions as shown in Fig. 2.5. Since in a fluid at rest there can be no shear forces, the only forces acting on the free-body are the normal pressure

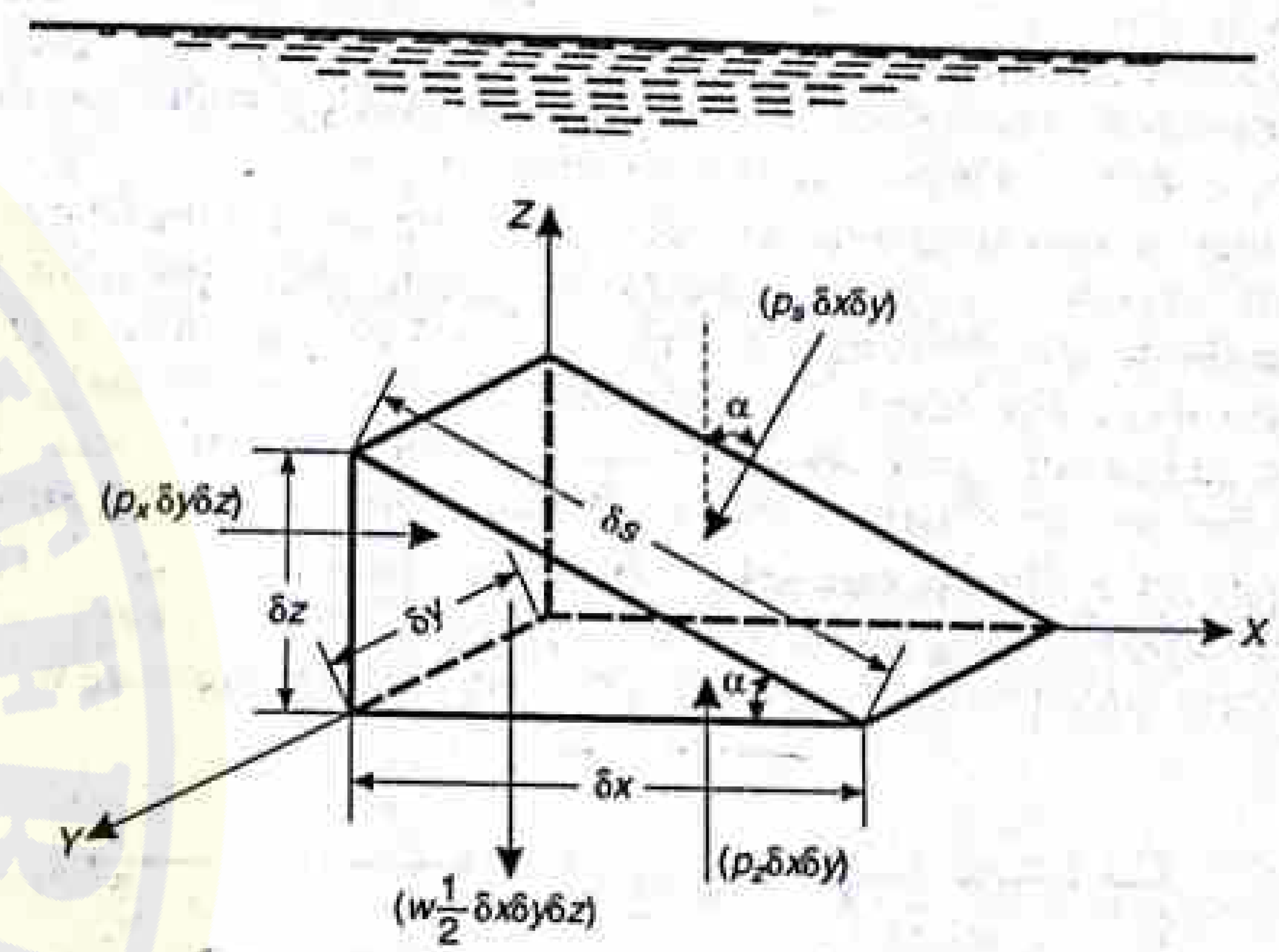


Figure 2.5 *Free-body diagram of a wedge-shaped element of fluid*

forces exerted by the surrounding fluid on the plane surfaces, and the weight of the element. As the element is in equilibrium, the sum of the force components on the element in any direction must be equal to zero. So, the equations of equilibrium in the x and z directions are respectively,

$$p_x\ \delta y\ \delta z - p_s\ \delta_s \delta y\ \sin\alpha = 0$$

$$p_z\ \delta x\ \delta y - p_s\ \delta s\ \delta y\ \cos\alpha - w\frac{1}{2}\ \delta x\ \delta y\ \delta z = 0$$

in which p_x, p_z, p_s are the average pressures on the three faces and w is the specific weight of the fluid. Since $\delta z = \delta s \sin\alpha$ and $\delta x = \delta s \cos\alpha$ the above equations simplify to

$$p_x\ \delta y\ \delta z - p_s\ \delta y\ \delta z = 0$$

$$p_z\ \delta x\ \delta y - p_s\ \delta x\ \delta y - \frac{1}{2} w\ \delta x\ \delta y\ \delta z = 0$$

The third term of the second equation is much smaller than the other two terms (since it involves a product of three infinitesimal quantities), and hence it may be neglected. Now by dividing the equations by $\delta y \delta z$ and $\delta x \delta y$ respectively and taking the limit, so that the element is reduced to a point, it follows from the equations that $p_s = p_x = p_z$.

Since angle α is chosen arbitrarily, this equation proves that the pressure is the same in all directions at a point in a static fluid.

The Pascal's law is made use of in the construction of machines such as hydraulic press, hydraulic jack, hydraulic lift, hydraulic crane, hydraulic rivetter etc., in which by the application of relatively small forces considerably larger forces are developed.

2.5 ATMOSPHERIC, ABSOLUTE, GAGE AND VACUUM PRESSURES

The atmospheric air exerts a normal pressure upon all surfaces with which it is in contact, and it is known as *atmospheric pressure*. The atmospheric pressure varies with the altitude and it can be measured by means of a barometer. As such it is also called the barometric pressure. At sea level under normal conditions the equivalent values of the atmospheric pressure are 10.1043×10^4 N/m^2 ; or 1.03 kg(f)/cm^2 ; or 10.3 m of water ; or 76 cm of mercury.

Fluid pressure may be measured with respect to any arbitrary datum. The two most common datums used are (i) absolute zero pressure and (ii) local atmospheric pressure. When pressure is measured above absolute zero (or complete vacuum), it is called an *absolute pressure*. When it is measured either above or below atmospheric pressure as a datum, it is called *gage pressure*. This is because practically all pressure gages read zero when open to the atmosphere and read only the difference between the pressure of the fluid to which they are connected and the atmospheric pressure.

If the pressure of a fluid is below atmospheric pressure it is designated as vacuum pressure (or suction pressure on negative gage pressure) ; and its gage value is the amount by which it is below that of the atmospheric pressure. A gage which measures vacuum pressure is known as vacuum gage.

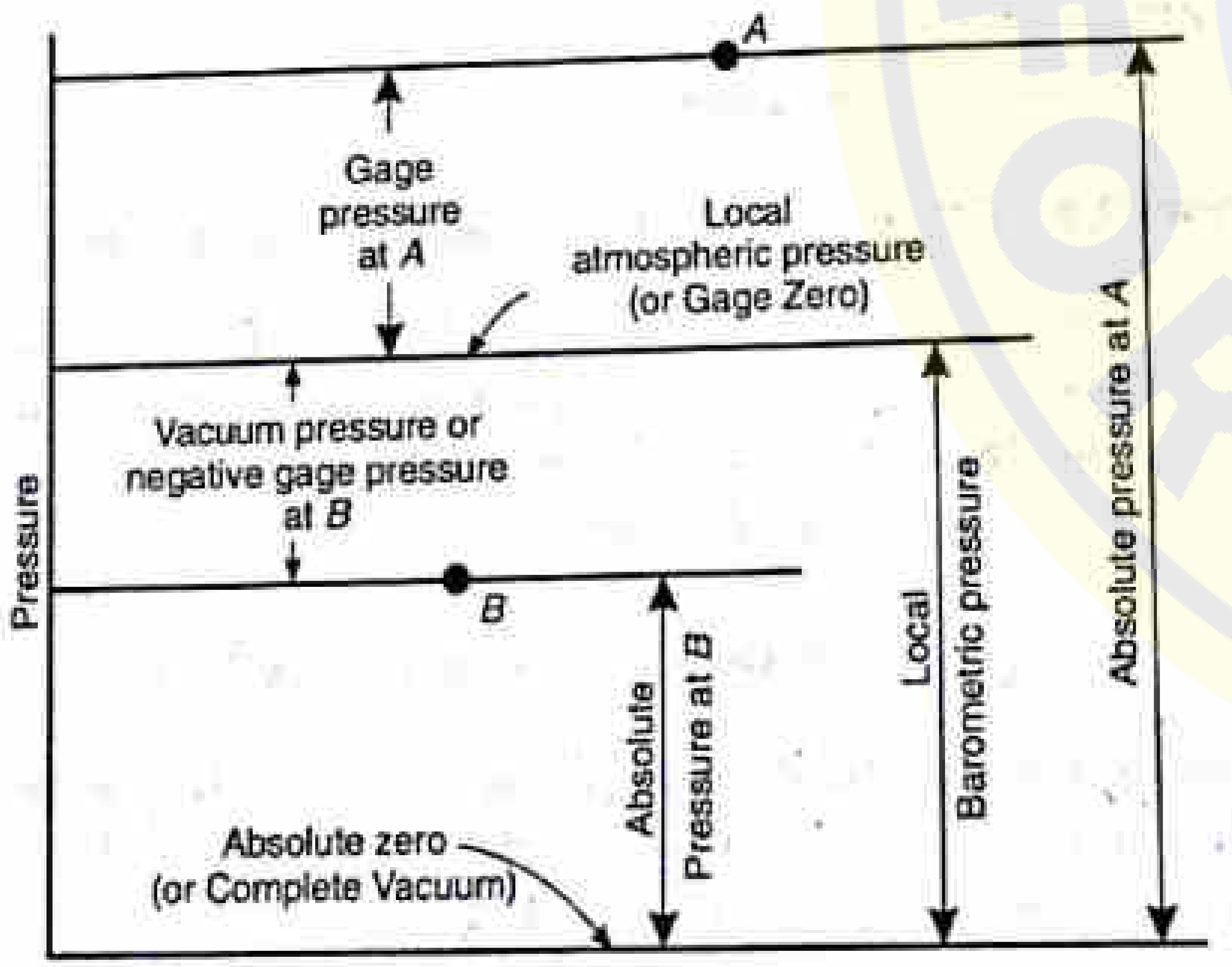


Figure 2.6 *Relationship between absolute, gage and vacuum pressures*

All values of absolute pressure are positive, since in the case of fluids the lowest absolute pressure which can possibly exist corresponds to absolute zero or complete vacuum. However, gage pressures are positive if they are above that of the atmosphere and negative if they are vacuum pressures. Figure 2.6 illustrates the relation between absolute gage and vacuum pressures.

From the foregoing discussion it can be seen that the following relations hold :

Absolute Pressure = Atmospheric Pressure + Gage Pressure ...(2.25)

Absolute Pressure = Atmospheric Pressure – Vacuum Pressure ...(2.26)

2.6 MESUREMENT OF PRESSURE

The various devices adopted for measuring fluid pressure may be broadly classified under the following two heads:

(1) Manometers

(2) Mechanical Gages.

Manometers. Manometers are those pressure measuring devices which are based on the principle of balancing the column of liquid (whose pressure is to be found) by the same or another column of liquid. The manometers may be classified as

(a) Simple Manometers.

(b) Differential Manometers.

Simple Manometers are those which measure pressure at a point in a fluid contained in a pipe or a vessel. On the other hand *Differential Manometers* measure the difference of pressure between any two points in a fluid contained in a pipe or a vessel.

Simple Manometers. In general a simple manometer consists of a glass tube having one of its ends connected to the gage point where the pressure is to be measured and the other remains open to atmosphere. Some of the common types of simple manometers are as noted below:

(i) Piezometer.

(ii) U-tube Manometer.

(iii) Single Column Manometer.

(i) *Piezometer.* A piezometer is the simplest form of manometer which can be used for measuring moderate pressures of liquids. It consists of a glass tube (Fig. 2.7) inserted in the wall of a pipe or a vessel, containing a liquid whose pressure is to be measured. The tube extends vertically upward to such a height that liquid can freely rise in it without overflowing. The pressure at any point in the liquid is indicated by the height of the liquid in the tube above that point, which can be read on the scale attached to it. Thus, if w is the specific weight of the liquid, then the pressure at point m in Fig. 2.7 (a) is $p_m = wh_m$. In other words, h_m is the pressure head at m. Piezometers measure gage pressure only, since the surface of the liquid in the tube is subjected to atmospheric pressure.

From the foregoing principles of pressure in homogeneous liquid at rest, it is obvious that the location of the point of insertion of a piezometer makes no difference. Hence as shown in Fig. 2.7 (a) piezometers may be inserted either in the top, or the side, or the bottom of the container, but the liquid will rise to the same level in the three tubes.

Negative gage pressures (or pressures less than atmospheric) can be measured by means of the piezometer shown in Fig. 2.7 (b). It is evident that if the pressure in the container is less than the atmospheric no column of liquid will rise in the ordinary piezometer. But if the top of the tube is bent downward and its lower end dipped into a vessel containing water (or some other suitable liquid) [Fig. 2.7 (b)], the atmospheric pressure will cause a column of the liquid to rise to a height h in the tube, from which the magnitude of the pressure of the liquid in the container can be obtained.

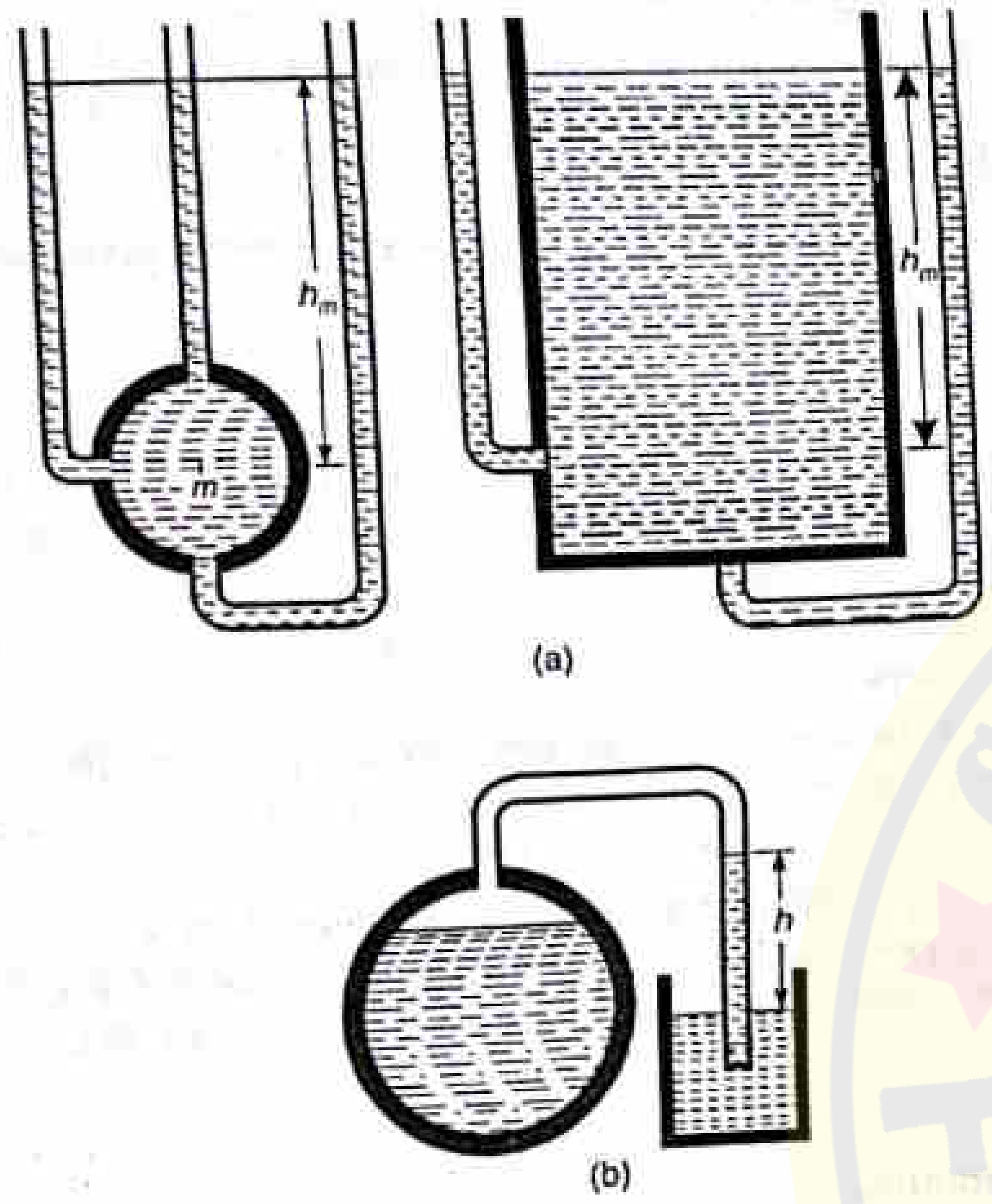


Figure 2.7 *Piezometers*

Neglecting the weight of the air caught in the portion of the tube, the pressure on the free surface in the container is the same as that at free surface in the tube which from Eq. 2.8 may be expressed as $p = -wh$, where w is the specific weight of the liquid used in the vessel. Conversely $-h$ is the pressure head at the free surface in the container.

Piezometers are also used to measure pressure heads in pipes where the liquid is in motion. Such tubes should enter the pipe in a direction at right angles to the direction of flow and the connecting end should be flush with the inner surface of the pipe. All burrs and surface roughness near the hole must be removed, and it is better to round the edge of the hole slightly. Also, the hole should be small, preferably not larger than 3 mm.

In order to prevent the capillary action from affecting the height of the column of liquid in a piezometer, the glass tube having an internal diameter less than 12 mm should not be used. Moreover for precise work at low heads the tubes having an internal diameter of 25 mm may be used.

(ii) *U-tube Manometer*. Piezometers cannot be used when large pressures in the lighter liquids are to be measured, since this would require very long tubes, which cannot be handled conveniently. Furthermore gas pressures cannot be measured by means of piezometers because a gas forms no free atmospheric surface. These limitations imposed on the use of piezometers may be overcome by the use of U-tube manometers. A U-tube manometer consists of a glass tube bent in U-shape, one

end of which is connected to the gage point and the other end remains open to the atmosphere (Fig. 2.8). The tube contains a liquid of specific gravity greater than that of the fluid of which the pressure is to be measured.

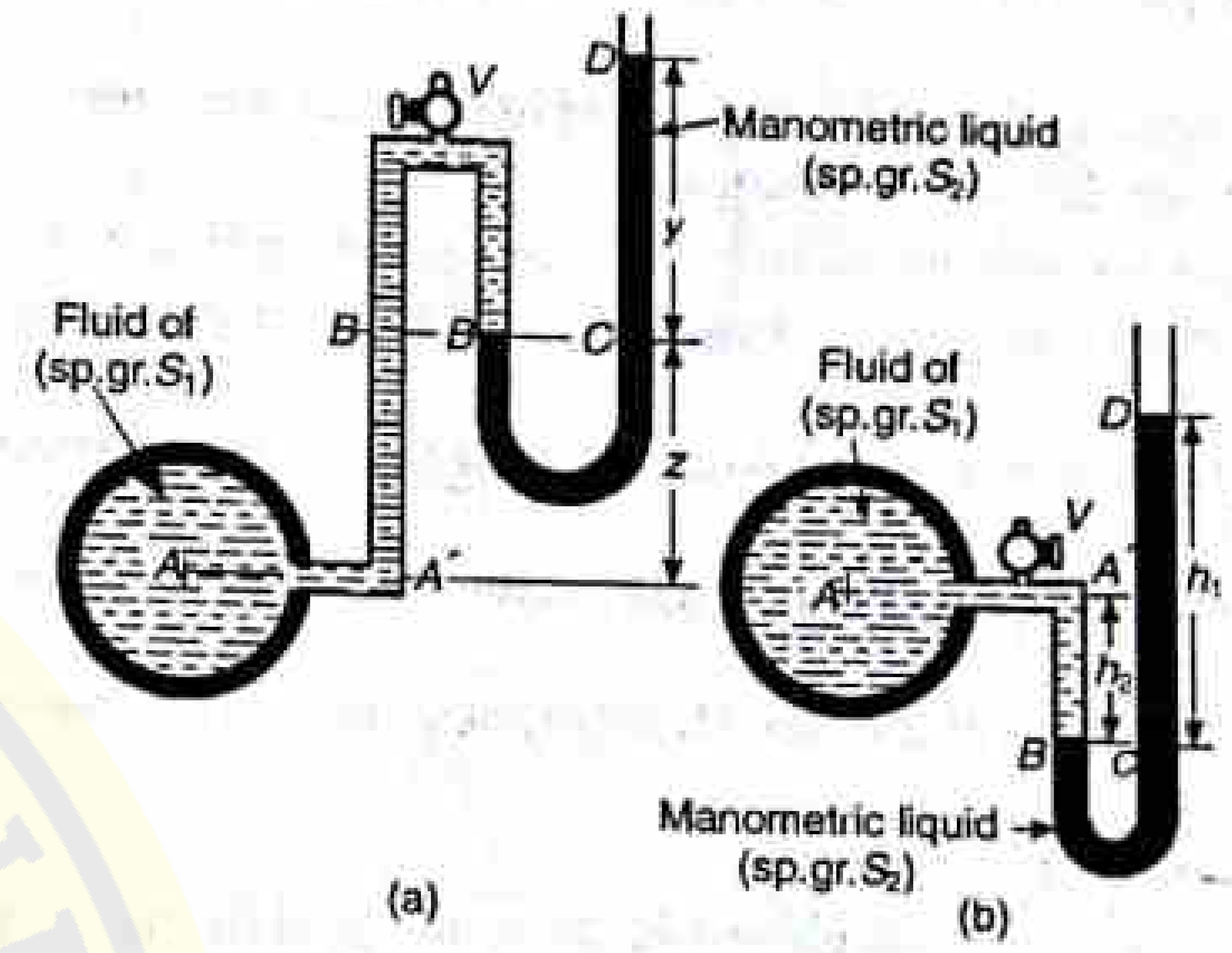


Figure 2.8 *U-tube simple manometer*

Sometimes more than one liquid may also be used in the manometer. The liquids used in the manometers should be such that they do not get mixed with the fluids of which the pressures are to be measured. Some of the liquids that are frequently used in the manometers are mercury, oil, salt solution, carbon disulphide, carbon tetrachloride, bromoform and alcohol. Water may also be used as a manometric liquid when the pressures of gases or certain coloured liquids (which are immiscible with water) are to be measured. The choice of the manometric liquid, however, depends on the range of pressure to be measured. For low pressure range, liquids of lower specific gravities are used and for high pressure range, generally mercury is employed.

When one of the limbs of the U-tube manometer is connected to the gage point, the fluid from the container or pipe A will enter the connected limb of the manometer thereby causing the manometric liquid to rise in the open limb as shown in Fig. 2.8. An air relief valve V is usually provided at the top of the connecting tube which permits the expulsion of all air from the portion $A'B$ and its place taken by the fluid in A. This is essential because the presence of even a small air bubble in the portion $A'B$ would result in an inaccurate pressure measurement.

In order to determine the pressure at A, a gage equation may be written as described below. Although any units of pressure or head may be used in the gage equation, it is generally convenient to express all the terms in metres of the fluid whose pressure is to be measured. The following general procedure may be adopted to obtain the gage equation :

(1) Start from either A or from the free surface in the open end of the manometer and write the pressure there in an appropriate unit (say metres of water or other fluid or N/m^2 or kg(f)/cm^2 or kg(f)/m^2). If the pressure is unknown (as at A) it may be expressed in terms of an appropriate symbol. On the other hand the pressure at the free surface in the open end (which is equal to atmospheric pressure) may be taken as zero. So that equation formed in each case will represent the gage pressure.

(2) To the pressure found above, add the change in pressure (in the same units) which will be caused while proceeding from one level to another adjacent level of contact of liquids of different specific gravities. Use positive sign if the next level of contact is lower than the first and negative if it is higher. The pressure heads in terms of the heights of columns of same liquid may be obtained by using Eq. 2.10 (a).

(3) Continue the process as in (2) until the other end of the gage is reached and equate the expression to the pressure at that point, known or unknown.

The expression will contain only one unknown viz., the pressure at A, which may thus be evaluated.

Thus for the manometer arrangement shown in Fig. 2.8 (a) the gage equation may be written as mentioned below.

Starting from A, if p_A is the unknown pressure intensity at A, w is the specific weight of water and S_1 is the specific gravity of the liquid in the container, then the pressure head at $A = \frac{p_A}{wS_1}$ (in terms of liquid at A). Since all points lying at the same horizontal level in the same continuous static mass of liquid have same pressure.

Pressure at A = Pressure at A'.

From A' to B' there being increase in elevation, pressure head decreases, so that pressure head at $B' = \left(\frac{p_A}{wS_1} - z\right)$. Again from the above enunciation,

Pressure at B' = Pressure at B = Pressure at C.

From C to D there being increase in elevation, pressure head decreases. If S_2 represents the specific gravity of the manometric liquid then from Eq. 2.10(a) the pressure head, in terms of liquid at A, equivalent to the column CD (= y) of the manometric fluid $= y\left(\frac{S_2}{S_1}\right)$. Thus pressure head at $D = \left(\frac{p_A}{wS_1} - z - y\frac{S_2}{S_1}\right)$. But at D there being atmospheric pressure, the pressure head = 0, in terms of the gage pressure. As such equating the pressure heads at D, the gage equation becomes

$$\frac{p_A}{wS_1} - z - \frac{yS_2}{S} = 0$$

or

$$\frac{p_A}{wS_1} = z + \frac{yS_2}{S_1} \quad ...(2.27)$$

Equation 2.27 represents the pressure heads in terms of the liquid at A. However, if the pressure heads are expressed in terms of water the following equation is obtained

$$\frac{p_A}{w} = zS_1 + yS_2 \quad ...(2.28)$$

Evidently Eq. 2.28 may be obtained directly from Eq. 2.27 by multiplying its both the sides by S_1, the specific gravity of liquid at A.

If A contains a gas, its specific weight being quite small, the specific gravity S_1 of the gas (with respect to water) will be so small that zS_1 may be neglected. In which case the pressure head of the gas at A in terms of water is given by

$$\frac{p_A}{w} = yS_2 \quad ...(2.29)$$

Figure 2.8 (b) shows another arrangement for measuring pressure at A by means of a U-tube manometer. By following the same procedure as indicated above the gage equation for this arrangement can also be written, which will be in terms of liquid at A as,

$$\frac{p_A}{wS_1} = h_1\frac{S_2}{S_1} - h_2 \quad ...(2.30)$$

and in terms of water as

$$\frac{p_A}{w} = h_1S_2 - h_2S_1 \quad ...(2.31)$$

Again if A contains gas, the specific gravity S_1 of the gas (with respect to water) being so small that h_2S_1 may be neglected. In which case Eq. 2.31 becomes

$$\frac{p_A}{w} = h_1S_2 \quad ...(2.32)$$

A U-tube manometer can also be used to measure negative or vacuum pressure. For measurement of small negative pressure, a U-tube manometer without any manometric liquid may be used, which is as shown in Fig. 2.9 (a). It is evident that for the pressure at A being negative (i.e., less than atmospheric pressure) the liquid surface in the open limb of the manometer will be below A. The pressure at A may be determined from the gage equation as obtained below.

At C the pressure head = 0, in terms of gage pressure. Further, pressure at C = pressure at B. From B to A', there being increase in elevation, the pressure head decreases, so that pressure head at $A' = 0 - h$ (in terms of liquid at A). Again, Pressure at A' = Pressure at A.

Now if p_A is the pressure intensity at A, w is the specific weight of water, and S_1 is the specific gravity of the liquid at A, then the pressure head at $A = \frac{p_A}{wS_1}$ (in terms of liquid at A). Thus in terms of liquid at A,

$$\frac{p_A}{wS_1} = -h \quad ...(2.33)$$

and in terms of water

$$\frac{p_A}{w} = -S_1h \quad ...(2.34)$$

For measuring negative pressures of larger magnitude a manometric liquid having higher specific gravity is employed, for which the arrangement as shown in Fig. 2.9 (b) or Fig. 2.9 (c) may be employed.

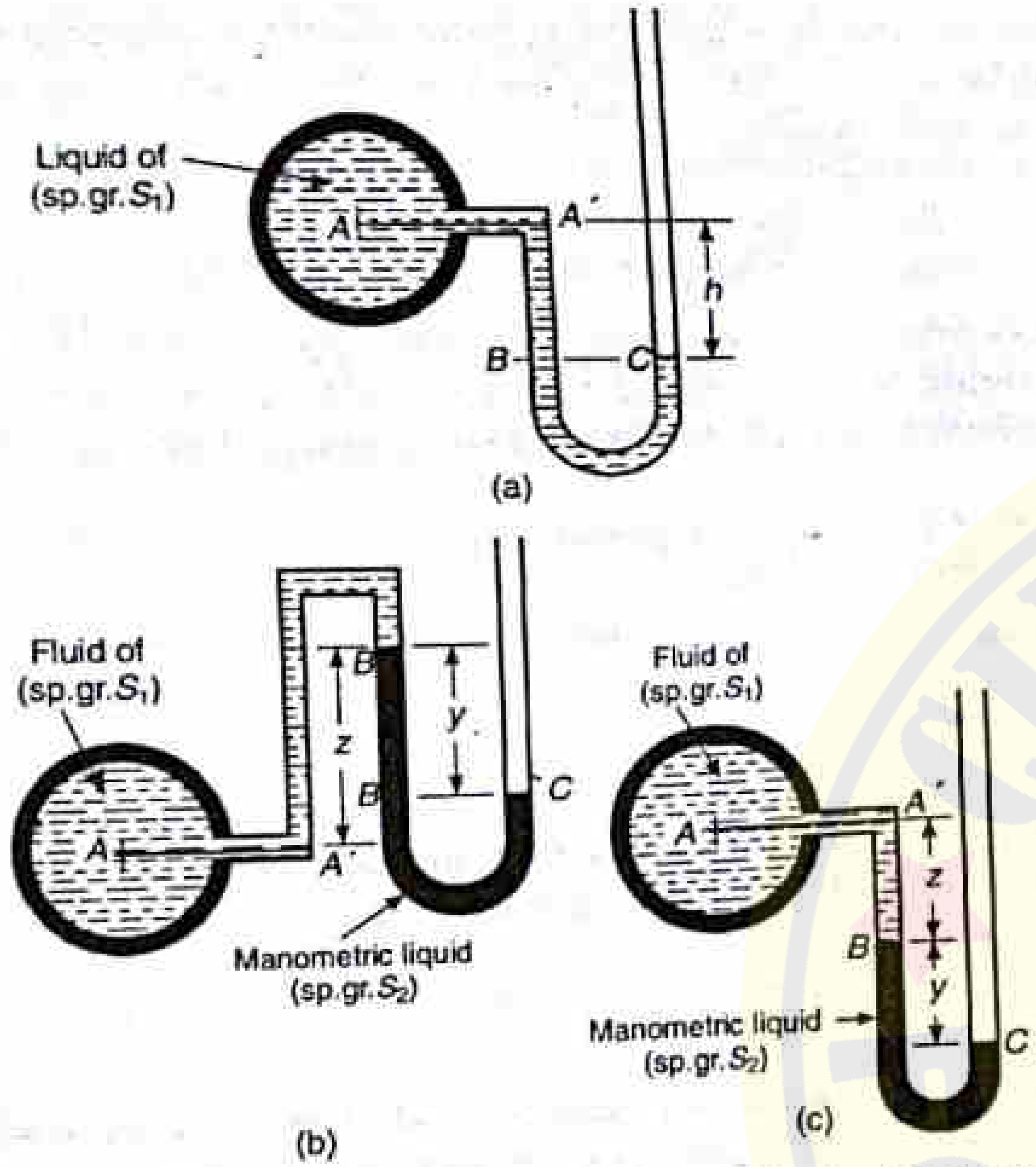


Figure 2.9 *Measurement of negative pressure by U-tube simple manometer*

If the specific gravity of the fluid at A is S_1 (with respect to water) and the specific gravity of the manometric liquid is S_2, the gage equation for the arrangement of Fig. 2.9 (b) may be written in terms of liquid at A as,

$$\frac{p_A}{wS_1} = z - y\frac{S_2}{S_1} \qquad ...(2.35)$$

and in terms of water as

$$\frac{p_A}{w} = zS_1 - yS_2 \qquad ...(2.36)$$

Similarly, for the arrangement shown in Fig. 2.9 (c) the gage equation may be written in terms of liquid at A as,

$$\frac{p_A}{wS_1} = -z - y\frac{S_2}{S_1} \qquad ...(2.37)$$

and in terms of water as

$$\frac{p_A}{w} = -zS_1 - yS_2 \qquad ...(2.38)$$

The arrangement of Fig. 2.9 (c) is generally preferred to that of Fig. 2.9 (b).

(iii) *Single Column Manometer*. The U-tube manometers described above usually require readings of fluid levels at two or more points, since a change in pressure causes a rise of liquid in one limb of the manometer and a drop in the other. This difficulty may however be overcome by using single column manometers. A single column manometer is a modified form of a U-tube manometer in which a shallow reservoir having a large cross-sectional area (about 100 times) as compared to the area of the tube is introduced into one limb of the manometer, as shown in Fig. 2.10. For any variation in pressure, the change in the liquid level in the reservoir will be so small that it may be neglected, and the pressure is indicated approximately by the height of the liquid in the other limb. As such only one reading in the narrow limb of the manometer need be taken for all pressure measurements. The narrow limb of the manometer may be vertical as in Fig. 2.10 (a) or it may be inclined as in Fig. 2.10 (b). The inclined type is useful for the measurement of small pressures. As explained later since no reading is required to be taken for the level of liquid in the reservoir, it need not be made of transparent material.

When the manometer is not connected to the container, the surface of the manometric liquid in the reservoir will stand at level 0 – 0, and since it is subjected to a pressure due to a column of fluid of height y and specific gravity S_1, the surface of the manometric liquid in the tube will stand at B, at a height h_1 above 0 – 0, such that from Eq. 2.10 (a)

$$yS_1 = h_1S_2 \qquad ...(2.39)$$

where S_2 is the specific gravity of the manometric liquid. This is known as the normal position of the manometric liquid. On being connected to the container at the gage point, the high pressure fluid will enter the reservoir, due to which there will be a drop in the manometric liquid surface in the reservoir by a distance Δy and a consequent rise in the tube by a distance h_2, above B. If A and a are the cross-sectional areas of the reservoir and the tube respectively, then,

$$A(\Delta y) = ah_2 \qquad ...(2.40)$$

If p_A is the pressure intensity at A and w is the specific weight of water, then starting from D the following gage equation in terms of water is obtained

$$0 + (h_2 + h_1 + \Delta y)S_2 - (\Delta y)S_1 - yS_1 = \frac{p_A}{w} \qquad ...(2.41)$$

Introducing Eqs (2.39) and (2.40) in Equation (2.41) it becomes

$$\frac{p_A}{w} = h_2\left[S_2 + (S_2 - S_1)\frac{a}{A}\right] \qquad ...(2.42)$$

By making the reservoir sufficiently large, the ratio $\left(\frac{a}{A}\right)$ can be made so small that Δy is negligible and height $(h_1 + h_2)$ is a measure of the pressure head at the contact level $C - C$ in the reservoir. In which case Eq. 2.42 reduces to

$$\frac{p_A}{w} = h_2S_2 \qquad ...(2.43)$$

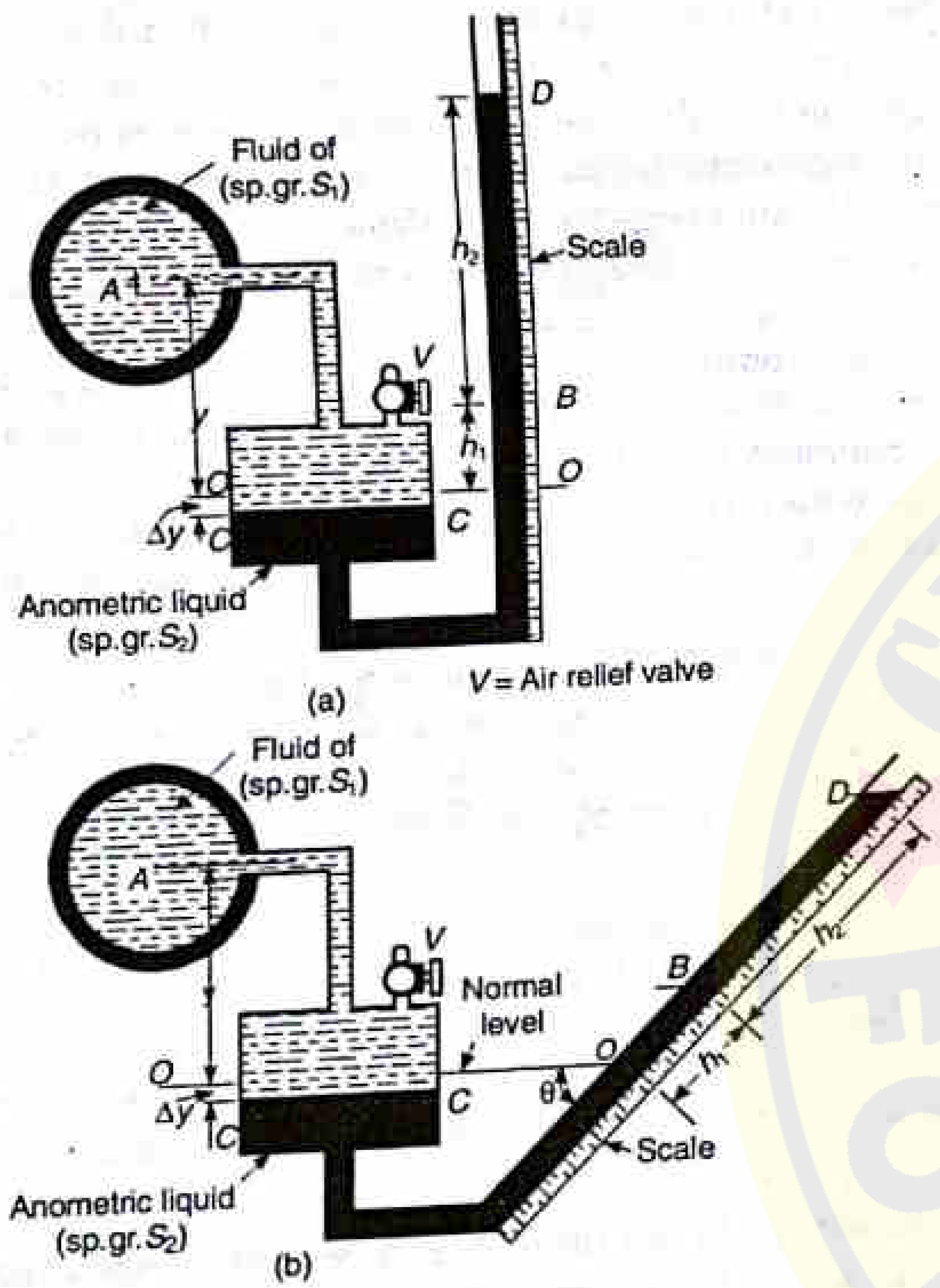


Figure 2.10 *Single column manometer*

so that only one reading of the height of level of liquid in the narrow tube is required to be taken to obtain the pressure head at A. However, if Δy is appreciable, then since the terms within brackets on the right side of Eq. 2.42 are constant, the scale on which h_2 is read can be so graduated as to correct for Δy so that again only one reading of the height of liquid level in the narrow tube is required to be taken, which will directly give the pressure head at A.

A single tube manometer can be made more sensitive by making its narrow tube inclined as shown in Fig. 2.10 (b). With this modification the distance moved by the liquid in the narrow tube shall be comparatively more, even for small pressure intensity at A. As before when the manometer is not connected to the container, the manometric liquid surface in the reservoir will stand at level 0 – 0 and that in the tube will stand at B, such that

$$yS_1 = (h_1 \sin \theta)\, S_2 \qquad \text{...(2.44)}$$

Due to high pressure fluid entering the reservoir, the manometric liquid surface will drop to level $C-C$ by a distance Δy, and it will travel a distance BD equal to h_2 in the narrow tube. Thus

$$A\,(\Delta y) = a h_2 \qquad \text{...(2.45)}$$

Again starting from D the gage equation in this case becomes

$$0 + (h_1 + h_2)(\sin \theta)\, S_2 + \Delta y S_2 - \Delta y S_1 - y S_1 = \frac{p_A}{w} \qquad \text{...(2.46)}$$

Introducing Eqs 2.44 and 2.45 in Eq. 2.46 it becomes

$$\frac{p_A}{w} = h_2 \left[S_2 \sin\theta + (S_2 - S_1)\frac{a}{A} \right] \qquad \text{...(2.47)}$$

Again if the ratio is negligible then Eq. 2.47 reduces to

$$\frac{p_A}{w} = (h_2 \sin \theta)\, S_2 \qquad \text{...(2.48)}$$

Single column manometers can also be employed to measure the negative gage pressures. If the pressure at A in the container is negative, the manometric liquid surface in the reservoir will be raised by a certain distance and consequently there will be drop in the liquid surface in the tube. Again by adopting the same procedure the gage equations for the negative pressure measurement can also be obtained.

Differential Manometers. For measuring the difference of pressure between any two points in a pipeline or in two pipes or containers, a differential manometer is employed. In general a differential manometer consists of a bent glass tube, the two ends of which are connected to each of the two gage points between which the pressure difference is required to be measured. Some of the common types of differential manometers are as noted below:

(i) Two–Piezometer Manometer.
(ii) Inverted U-Tube Manometer.
(iii) U- Tube Differential Manometer.
(iv) Micromanometer.

(i) *Two-Piezometer Manometer.* As the name suggests this manometer consists of two separate piezometers which are inserted at the two gage points between which the difference of pressure is required to be measured. The difference in the levels of the liquid raised in the two tubes will denote the pressure difference between the two points. Evidently this method is useful only if the pressure at each of the two points is small. Moreover it cannot be used to measure the pressure difference in gases, for which the other types of differential manometers described below may be employed.

(ii) *Inverted U-tube Manometer.* It consists of a glass tube bent in U-shape and held inverted as shown in Fig. 2.11. Thus it is as if two piezometers described above are connected with each other at top. When the two ends of the manometer are connected to the points between which the pressure difference is required to be measured, the liquid under pressure will enter the two limbs of the manometer, thereby causing the air within the manometer to get compressed. The presence of the compressed air results in restricting the heights of the columns of liquids raised in the two limbs of the manometer. An air cock as shown in Fig. 2.11, is usually provided at the top of the inverted U-tube which facilitates the raising of the liquid columns to suitable level in both the limbs by driving out a portion of the compressed air. It also permits the expulsion of air bubbles which might have been entrapped somewhere in the pipeline.

If p_A and p_B are the pressure intensities at points A and B between which the inverted U-tube manometer is connected, then corresponding to these pressure intensities the liquid will rise above points A and B upto C and D in the two limbs of the manometer as shown in Fig. 2.11. Now if w represents the specific weight of water and S_1 represents the specific gravity of the liquid at A or B,

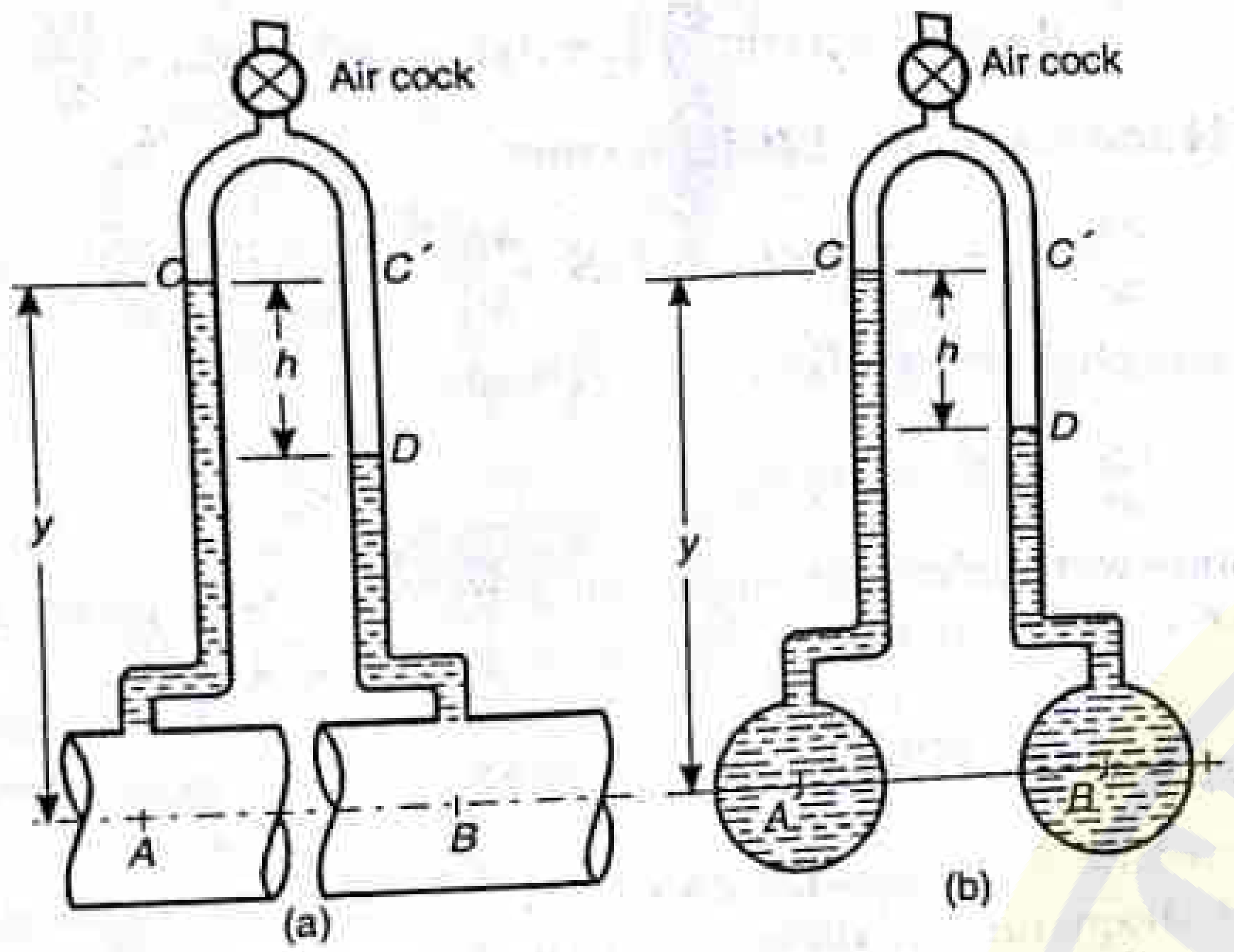


Figure 2.11 *Inverted U-Tube differential manometer*

then commencing from A, the pressure head at C in terms of water is equal to $\left(\frac{p_A}{w} - yS_1\right)$. Since points C and C' are at the same horizontal level and in the same continuous static mass of fluid

Pressure at C = Pressure at C'

Between points C' and D there is a column of compressed air. Since the specific weight of air is negligible as compared with that of liquid, the weight of air column between C' and D may be neglected. Hence,

Pressure at C' = Pressure at D

From D to B there being decrease in elevation, the pressure head increases so that the pressure head at B is equal to pressure head at D plus $(y - h)\, S_1$. Thus the gage equation may be expressed as

$$\frac{p_A}{w} - yS_1 + (y-h)\,S_1 = \frac{p_B}{w}$$

or

$$\frac{p_A}{w} - \frac{p_B}{w} = hS_1 \qquad \text{...(2.49)}$$

Inverted U-tube manometers are suitable for the measurement of small pressure difference in liquids. Sometimes instead of air, the upper part of this manometer is filled with a manometric liquid which is lighter than the liquid for which the pressure difference is to be measured and is immiscible with it. Such an arrangement is shown in Fig. 2.12. As explained later the use of manometric liquid in this manometer results in increasing the sensitivity of the manometer.

Again if p_A and p_B are the pressure intensities at points A and B between which the inverted U-tube manometer is connected, then corresponding to these pressure intensities the liquid will rise above points A and B upto C and D in the two limbs of the manometer as shown in Fig. 2.12. Now

if w represents the specific weight of water and S_1 and S_2 are the specific gravities of the liquid at A or B and the manometric liquid (in the upper part of the manometer) respectively, then commencing

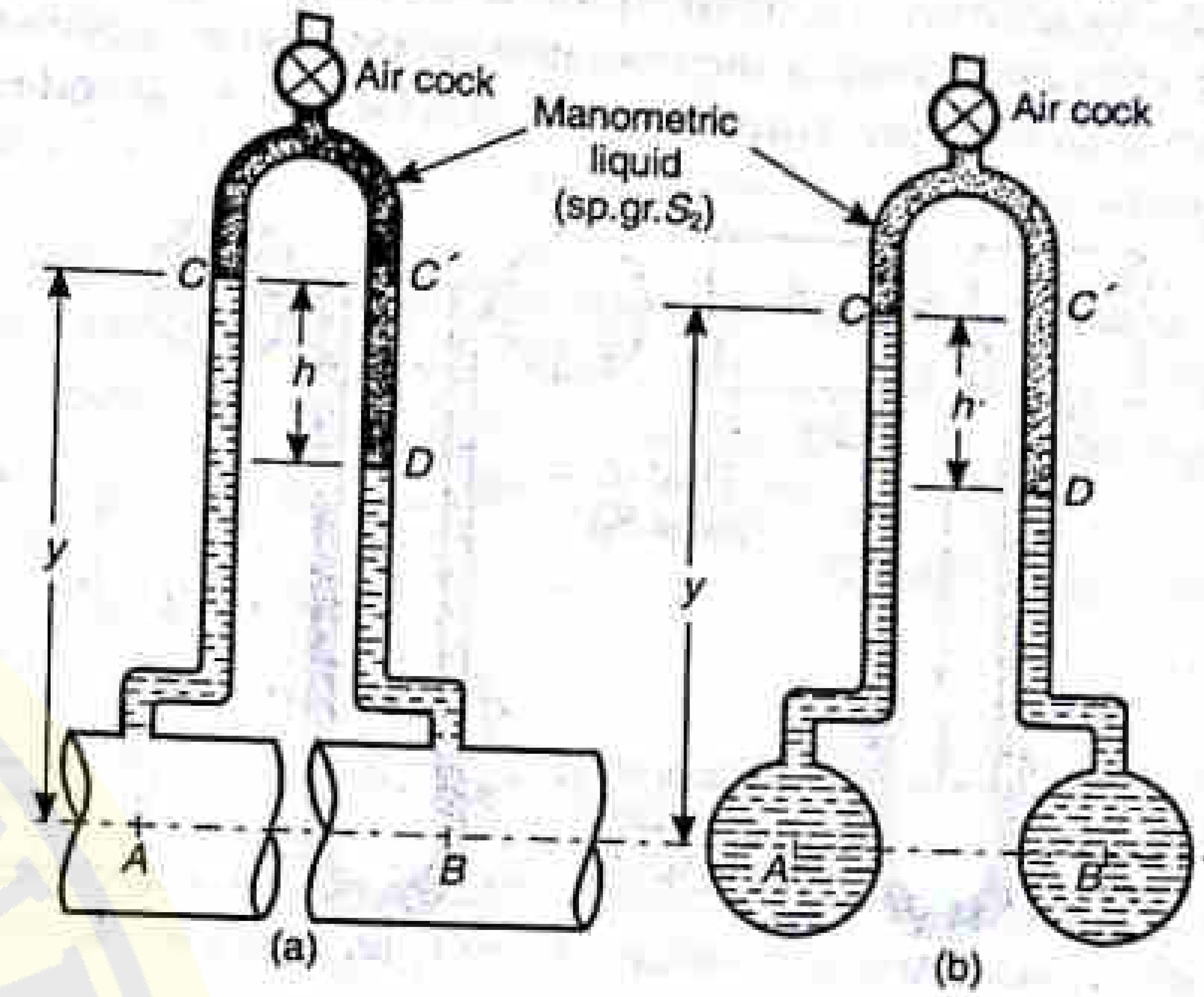


Figure. 2.12 *Inverted U-tube differential manometer with light manometric liquid*

from A, the pressure head at C in terms of water is equal to $\left(\frac{p_A}{w} - yS_1\right)$. Since points C and C' are at the same horizontal level and in the same continuous static mass of liquid,

∴ Pressure at C = Pressure at C'

From C' to D there being decrease in elevation, the pressure head increases, so that the pressure head at D in terms of water is equal to

$$\left[\left(\frac{p_A}{w} - yS_1\right) + hS_2\right]$$

Further from D to B there is decrease in elevation and hence the gage equation becomes

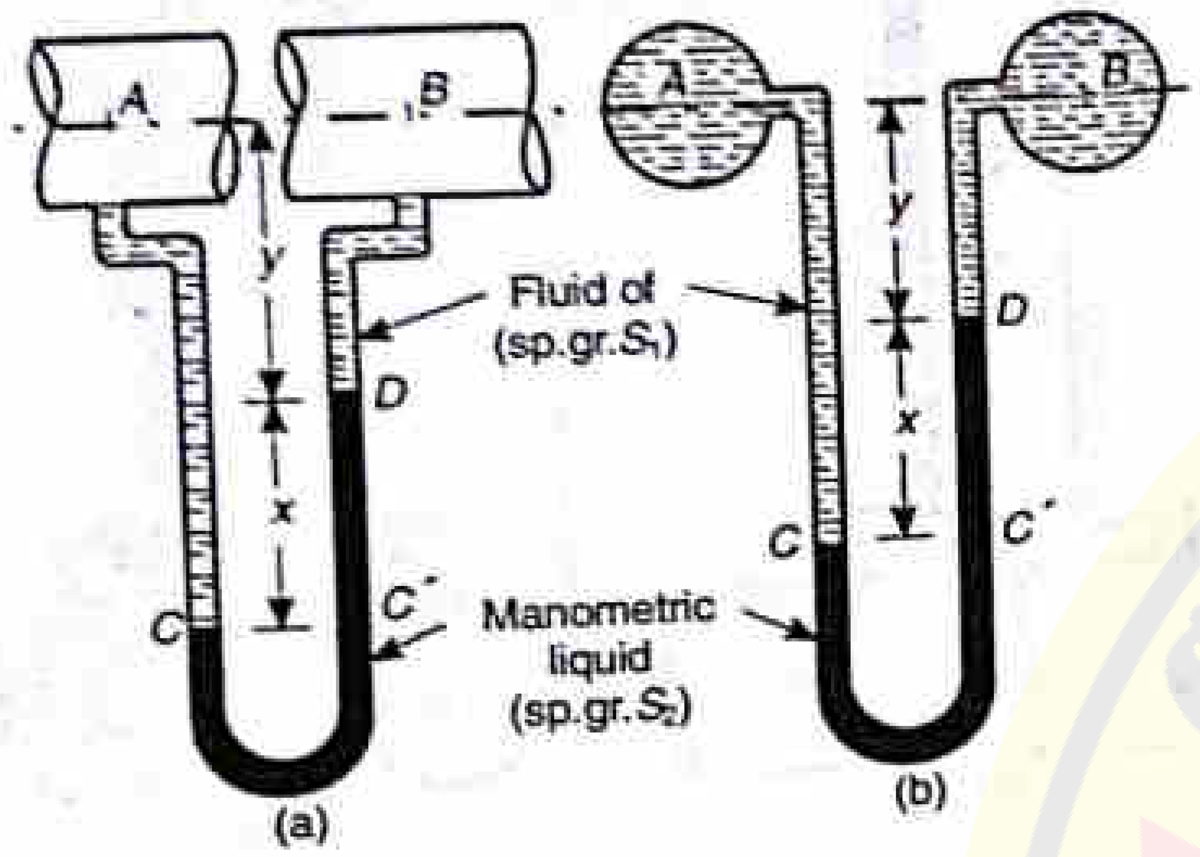
$$\frac{p_A}{w} - yS_1 + hS_2 + (y-h)\,S_1 = \frac{p_B}{w}$$

or

$$\frac{p_A}{w} - \frac{p_B}{w} = h\,(S_1 - S_2) \qquad \text{...(2.50)}$$

It is evident from Eq. 2.50 that as the specific gravity of the manometric liquid approaches that of the liquid at A or B $(S_1 - S_2)$ approaches zero and large values of h will be obtained even for small pressure differences, thus increasing the sensitivity of the manometer. Another arrangement for increasing the sensitivity of these manometers is to incline the gage tubes so that a vertical gage difference h is transposed into a reading which is magnified by $\frac{1}{\sin\theta}$, where θ is angle of inclination with the horizontal.

(iii) *U-Tube Differential Manometer.* It consists of glass tube bent in U-shape, the two ends of which are connected to the two gage points between which the pressure difference is required to be measured. Figure 2.13 shows such an arrangement for measuring the pressure difference between any two points A and B. The lower part of the manometer contains a manometric liquid which is heavier than the liquid for which the pressure difference is to be measured and is immiscible with it.



Fogure 2.13 *U-tube Differential Manometer*

When the two limbs of the manometer are connected to the gage points A and B, then corresponding to the difference in the pressure intensities p_A and p_B the levels of manometric liquid in the two limbs of the manometer will be displaced through a distance x as shown in Fig. 2.13. By measuring this difference in the levels of the manometric liquid, the pressure difference $(p_A - p_B)$ may be computed as indicated below.

If S_1 and S_2 are the specific gravities of the liquid at A or B and the manometric liquid respectively, then by commencing at A where the pressure is p_A, the pressure head at C in terms of water is equal to $\left[\frac{p_A}{w}+(y+x)S_1\right]$, in which w is the specific weight of water. Further, since points C and C' are at the same level and are lying in the same continuous static mass of liquid,

Pressure at C = Pressure at C'

Further from C' to D there being an increase in the elevation, the pressure head decreases, so that the pressure head at D in terms of water is equal to $\left[\frac{p_A}{w}+(y+x)S_1-xS_2\right]$. Similarly from D to B there is an increase in elevation and hence the gage equation becomes

$$\left[\frac{p_A}{w}+(y+x)S_1-xS_2-yS_1\right] = \frac{p_B}{w}$$

or

$$\frac{p_A}{w}-\frac{p_B}{w} = x(S_2-S_1) \qquad ...(2.51)$$

If, for example, the manometric liquid is mercury ($S_2 = 13.6$) and the liquid at A or B is water ($S_1 = 1$) then the difference in pressure heads at the points A and B is 12.6 times the deflection x of the manometric liquid in the two limbs of the manometer. As such the use of mercury as manometric liquid in U-tube manometer is suitable for measuring large pressure differences. However, for small pressure differences, mercury makes precise measurement difficult, and hence for such cases it is common to use a liquid which is only slightly heavier than the liquid for which the pressure difference is to be measured.

Often the points A and B between which the pressure difference is to be measured are not at the same level, as shown in Fig. 2.14. For such cases also, by adopting the same procedure, the following gage equation may be obtained in order to compute the pressure difference between the points A and B.

$$\left[\frac{p_A}{w}+(z+y+x)S_1-xS_2-yS_3\right] = \frac{p_B}{w}$$

or

$$\frac{p_A}{w}-\frac{p_B}{w} = [x(S_2-S_1)+y(S_3-S_1)-zS_1] \qquad ...(2.52)$$

Equation 2.52 is, however, a general equation, which may be modified to derive the equations for different conditions. Thus, for example, if there is same liquid at A and B, then since $S_1 = S_3$, Eq. 2.52 becomes

$$\frac{p_A}{w}-\frac{p_B}{w} = [x(S_2-S_1)-zS_1] \qquad ...(2.53)$$

Further if A and B are at the same level, then since $z = 0$, Eq. 2.53 becomes same as Eq. 2.51 which is quite obvious.

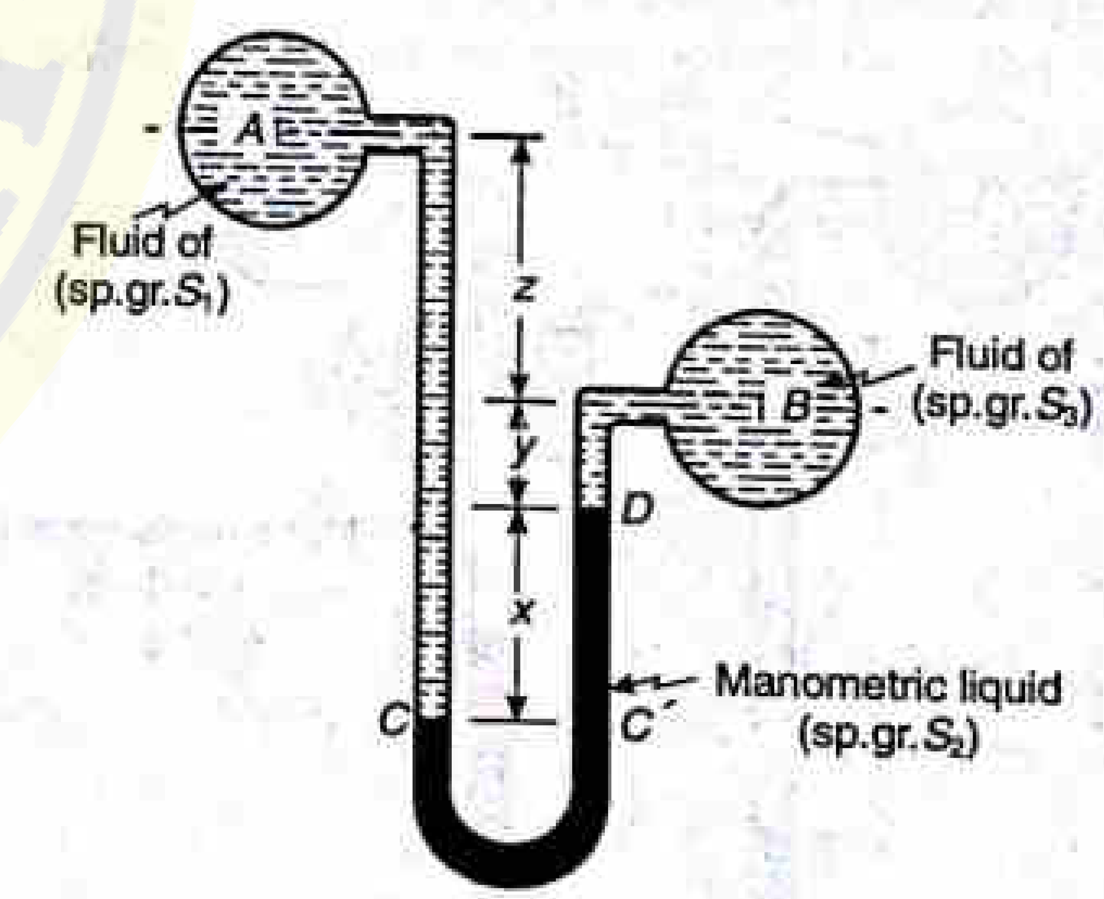


Figure 2.14 *U-tube differential manometer between two points at different levels*

(iv) *Micromanometers.* For the measurement of very small pressure differences, or for the measurement of pressure differences with very high precision, special forms of manometers called micromanometers are used. A wide variety of micromanometers have been developed, which either magnify the readings or permit the readings to be observed with greater accuracy. One simple type of micromanometer consists of a glass U-tube, provided with two transparent basins of wider sections at the top of the two limbs, as shown in Fig. 2.15. The manometer contains two manometric liquids of different specific gravities and immiscible with each other and with the fluid for which the pressure difference is to be measured. Before the manometer is connected to the pressure points A and B, both the limbs are subjected to the same pressure. As such the heavier manometric liquid of sp. gr. S_1 will occupy the level DD' and the lighter manometric liquid of sp. gr. S_2 will occupy the level CC'. When the manometer is connected to the pressure points A and B where the pressure intensities are p_A and p_B respectively, such that $p_A > p_B$ then the level of the lighter manometric liquid will fall in the left basin and rise in the right basin by the same amount Δy. Similarly the level of the heavier manometric liquid will fall in the left limb to point E and rise in the right limb to point F. If A and a are the cross-sectional areas of the basin and the tube respectively, then since the volume of the liquid displaced in each basin is equal to the volume of the liquid displaced in each limb of the tube the following expression may be readily obtained

$$A\,(\Delta y) = a\left(\frac{x}{2}\right) \qquad \text{...(2.54)}$$

Further if w is specific weight of water, then, starting from point A the following gage equation in terms of water column may be obtained,

$$\frac{p_A}{w} + (y_1 + \Delta y)\,S_3 + \left(y_2 - \Delta y + \frac{x}{2}\right)S_2 - xS_1 - \left(y_2 - \frac{x}{2} + \Delta y\right)S_2 - (y_1 - \Delta y)\,S_3 = \frac{p_B}{w}$$

Substituting the value of Δy from Eq. 2.54 and simplifying the above equation it becomes

$$\frac{p_A}{w} - \frac{p_B}{w} = \left[S_1 - S_2\left(1 - \frac{a}{A}\right) - S_3\,\frac{a}{A}\right] \qquad \text{...(2.55)}$$

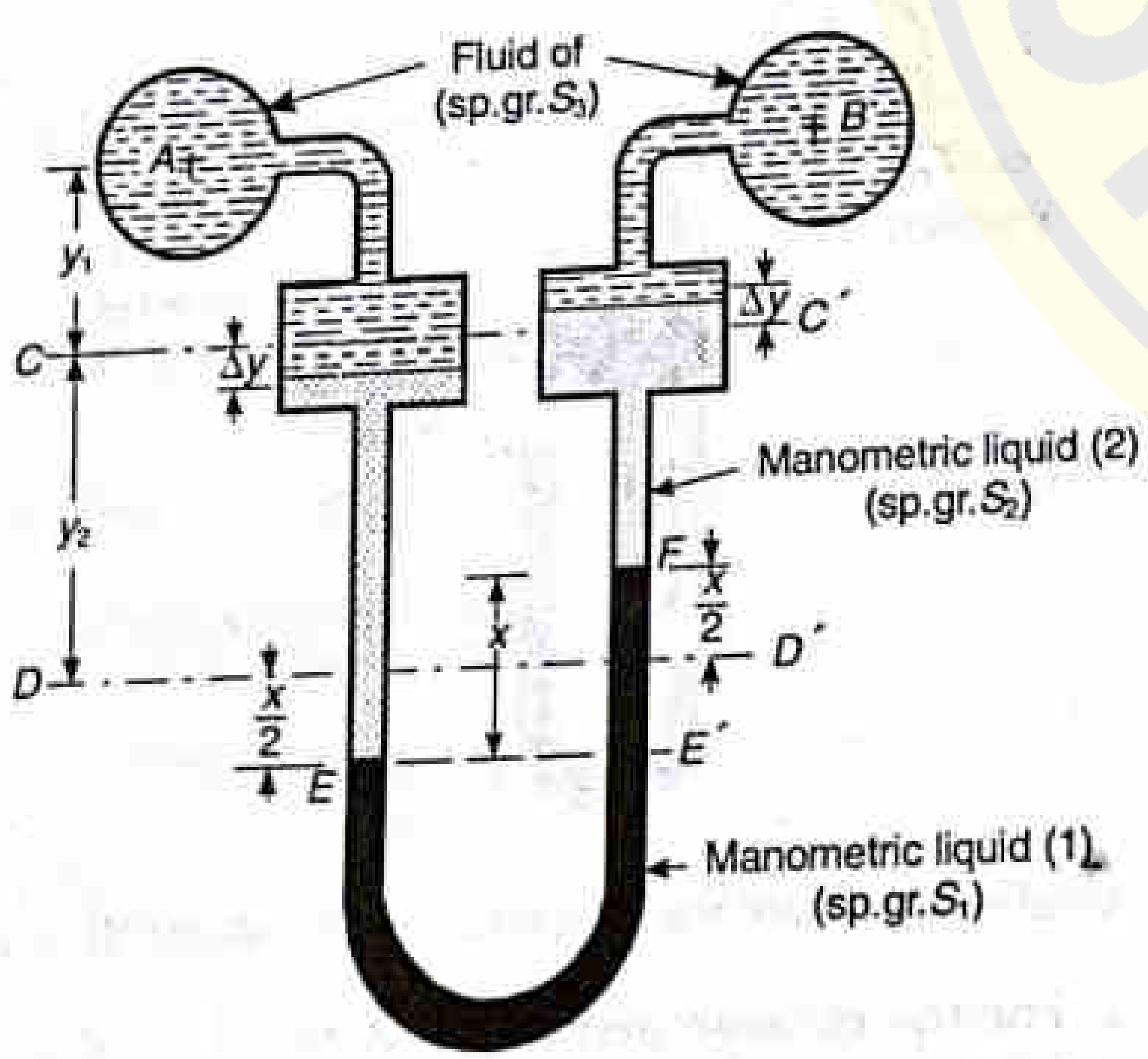


Figure 2.15 *Micromanometer*

The quantities within brackets on right side of Eq. 2.55 are constant for a particular manometer. Thus by measuring x and substituting in Eq. 2.55 the pressure difference between any two points can be known.

If the cross-sectional area of the basin is large as compared with the cross-sectional area of the tube, then the ratio a/A is very small and the Eq. 2.55 reduces to

$$\frac{p_A}{w} - \frac{p_B}{w} = x\,(S_1 - S_2) \qquad \text{...(2.56)}$$

By selecting the two manometric liquids such that their specific gravities are very nearly equal then a measurable value of x may be achieved even for a very small pressure difference between the two points.

In a number of other types of micromanometers the pressure difference to be measured is balanced by the slight raising or lowering (on a micrometer screw) of one arm of the manometer whereby a meniscus is brought back to its original position. The micromanometers of this type are those invented by Chattock, Small and Krell, which are sensitive to pressure differences down to less than 0.0025 mm of water. However the disadvantage with such manometers is that an appreciable time is required to take a reading and they are therefore suitable only for completely steady pressures.

Mechanical Gages. Mechanical gages are those pressure measuring devices, which embody an elastic element, which deflects under the action of the applied pressure, and this movement mechanically magnified, operates a pointer moving against a graduated circumferential scale. Generally these gages are used for measuring high pressures and where high precision is not required. Some of the mechanical pressure gages which are commonly used are as noted below:

(i) Bourdon Tube Pressure Gage
(ii) Diaphragm Pressure Gage
(iii) Bellows Pressure Gage
(iv) Dead-weight Pressure Gage

(i) *Bourdon Tube Pressure Gage.* It is the most common type of pressure gage which was invented by E. Bourdon (1808–84). The pressure responsive element in this gage is a tube of steel or bronze which is of elliptic cross-section and is curved into a circular arc. The tube is closed at its outer end,

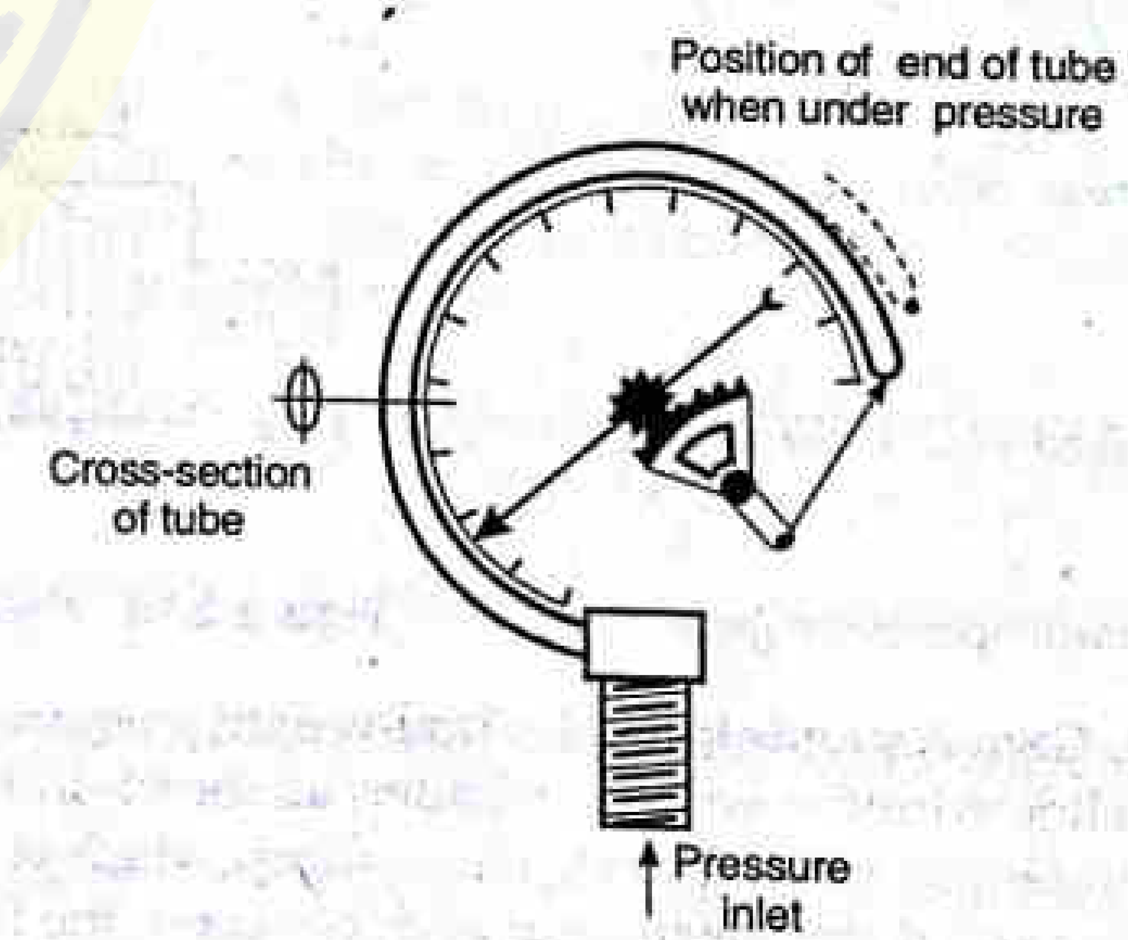


Figure 2.16 *Bourdon tube pressure gage*

and this end of the tube is free to move. The other end of the tube, through which the fluid enters, is rigidly fixed to the frame as shown in Fig. 2.16. When the gage is connected to the gage point, fluid under pressure enters the tube. Due to increase in internal pressure, the elliptical cross-section of the tube tends to become circular, thus causing the tube to straighten out slightly. The small outward movement of the free end of the tube is transmitted, through a link, quadrant and pinion, to a pointer which by moving clockwise on the graduated circular dial indicates the pressure intensity of the fluid. The dial of the gage is so calibrated that it reads zero when the pressure inside the tube equals the local atmospheric pressure, and the elastic deformation of the tube causes the pointer to be displaced on the dial in proportion to the pressure intensity of the fluid. By using tubes of appropriate stiffness, gages for wide range of pressures may be made. Further by suitably modifying the graduations of the dial and adjusting the pointer Bourdon tube vacuum gages can also be made. When a vacuum gage is connected to a partial vacuum, the tube tends to close, thereby moving the pointer in anti-clockwise direction, indicating the negative or vacuum pressure. The gage dials are usually calibrated to read newton per square metre (N/m^2), or pascal (Pa), or kilogram (f) per square centimetre [$kg(f)/cm^2$]. However other units of pressure, such as metres of water or centimetres of mercury, are also frequently used.

(ii) *Diaphragm Pressure Gage.* The pressure responsive element in this gage is an elastic steel corrugated diaphragm. The elastic deformation of the diaphragm under pressure is transmitted to a pointer by a similar arrangement as in the case of Bourdon tube pressure gage (see Fig. 2.17). However, this gage is used to measure relatively low pressure intensities. The Aneroid barometer operates on a similar principle.

(iii) *Bellows Pressure Gage.* In this gage the pressure responsive element is made up of a thin metallic tube having deep circumferential corrugations. In response to the pressure changes this elastic element expands or contracts, thereby moving the pointer on a graduated circular dial as shown in Fig. 2.18.

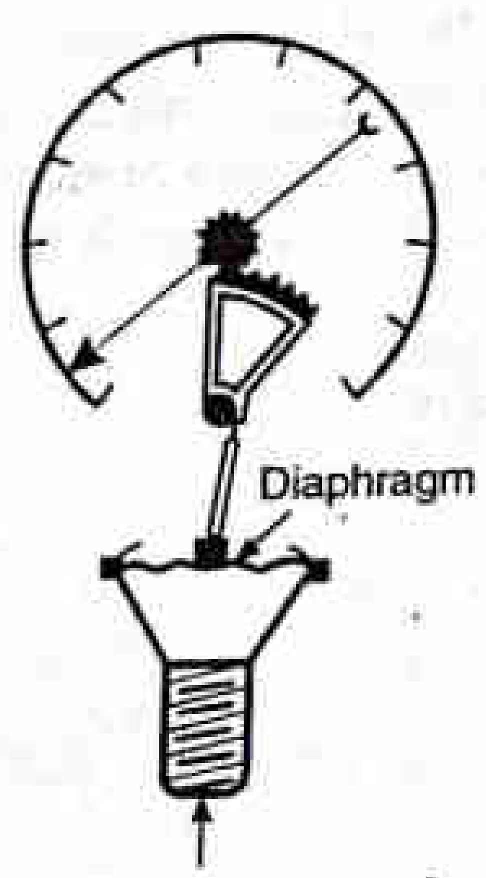


Figure 2.17 *Diaphragm pressure gage*

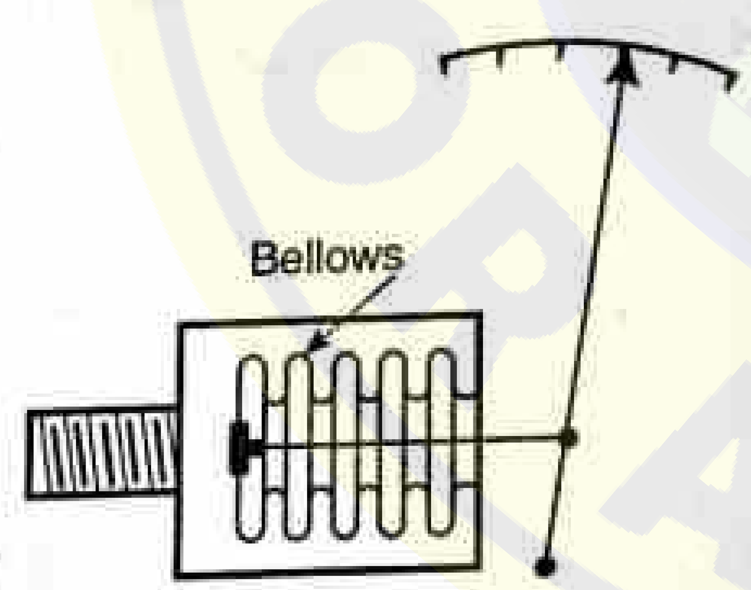


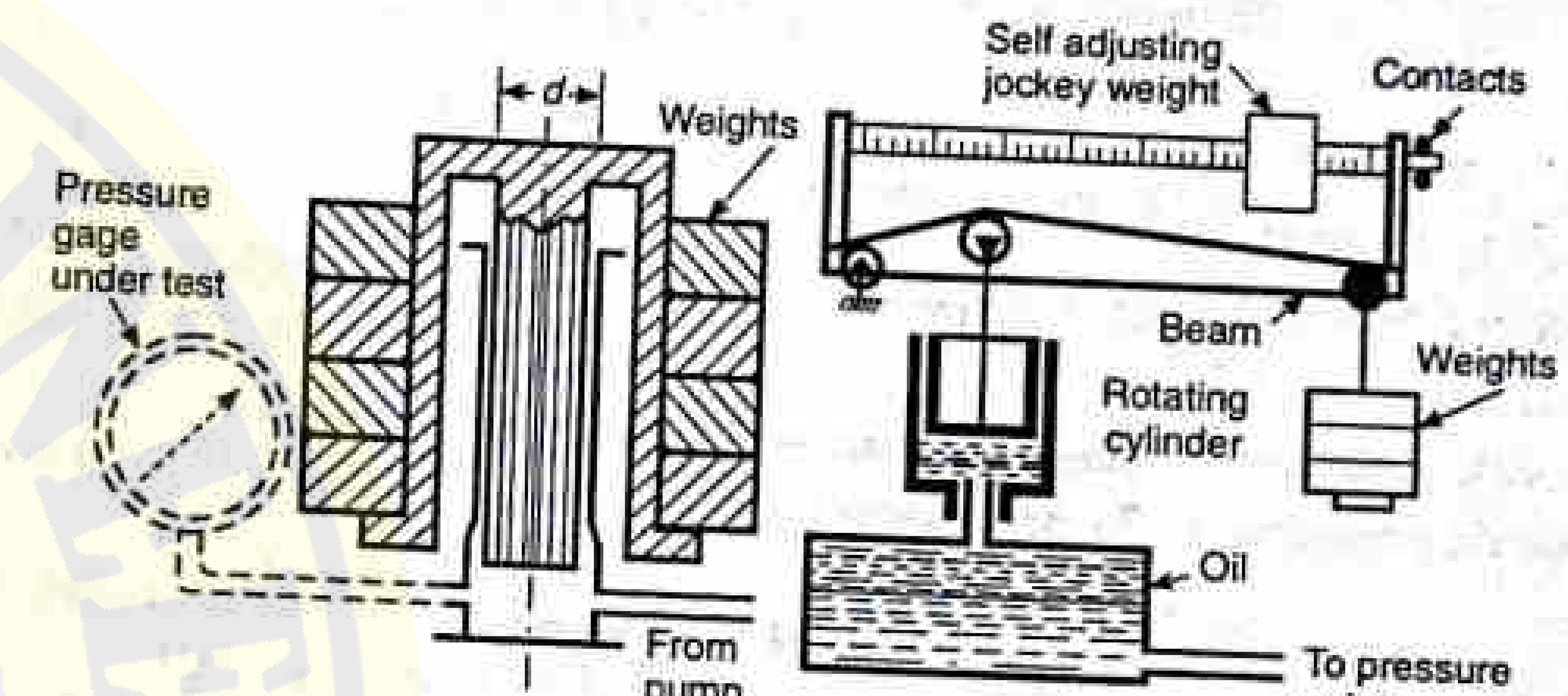
Figure 2.18 *Bellows pressure gage*

(iv) *Dead-Weight Pressure Gage.* A simple form of a dead-weight pressure gage consists of a plunger of diameter d, which can slide within a vertical cylinder, as shown in Fig. 2.19. The fluid under pressure, entering the cylinder, exerts a force on the plunger, which is balanced by the weights loaded on the top of the plunger. If the weight required to balance the fluid under pressure is W, then the pressure intensity p of the fluid may be determined as,

$$p = W\Big/\left(\frac{\pi}{4}d^2\right)$$

The only error that may be involved is due to frictional resistance offered to motion of the plunger in the cylinder. But this error can be avoided if the plunger is carefully ground, so as to fit with the least permissible clearance in the cylinder. Moreover, the whole mass can be rotated by hand before final readings are taken.

Dead-weight gages are generally not used so much to measure the pressure intensity at a particular point as to serve as standards of comparison. Hence as shown in Fig. 2.19, a pressure gage which is to be checked or calibrated is set in parallel with the dead-weight gage. Oil under pressure is pumped into the gages, thereby lifting the plunger and balancing it against the oil pressure by loading it with known weights. The pressure intensity of the oil being thus known, the attached pressure gage can either be tested for its accuracy or it can be calibrated.



(a) Dead weight pressure gage used for calibration.

(b) Dead weight pressure gage used for measuring pressure at a point

Figure 2.19 *Dead-weight pressure gage*

A dead-weight gage which can be used for measuring pressure at a point with more convenience is also shown in Fig. 2.19. In this gage a lever, same as in some of the weighing machines, is provided to magnify the pull of the weights. The load required to balance the force due to fluid pressure is first roughly adjusted by hanging weights from the end of the main beam. Then a smaller jockey weight is slided along to give precise balance. In more precise type of gage the sliding motion may be contrived automatically by an electric motor.

2.7 GENERAL COMMENTS ON CONNECTIONS FOR MANOMETERS AND GAGES

The following points should be kept in view while making connections for the various pressure measuring devices:

(i) At the gage point the hole should be drilled normal to the surface and it should flush with the inner surface.

(ii) The diameters of the holes at the gage points should be about 3 mm to 6 mm.

(iii) The holes should not disturb the internal surface and no burrs or irregularities must be left.

(iv) There should be no air pockets left over in the connecting tubes, which should be completely filled with the liquid. The presence of air bubbles can easily be detected if the connecting tubes are made of polythene or similar transparent material.

ILLUSTRATIVE EXAMPLES

Example 2.1. *Express pressure intensity of 7.5 kg (f)/cm² in all pressure units. Take the barometer reading as 76 cm of mercury.*

Solution

(A) *Gage Units*

(a) $p = 7.5$ kg(f)/cm²

(b) $p = 7.5 \times 10^4$ kg(f)/m²

(c) $h = \frac{p}{w} = \frac{7.5 \times 10^4}{1000} = 75$ m of water

(d) $h = \frac{p}{w} = \frac{7.5 \times 10^4}{13.6 \times 1000} = 5.51$ m of mercury

(e) $p = 9\,810 \times 75 = 73.575 \times 10^4$ N/m²

(B) *Absolute Units*

$$\text{Absolute pressure} = \text{Gage pressure} + \text{Atmospheric pressure}$$

$$\text{Atmospheric pressure} = 76 \text{ cm of mercury}$$

$$= \frac{76 \times 13.6}{100} = 10.34 \text{ m of water}$$

$$= \frac{76 \times 13.6 \times 1000}{100} = 1.034 \times 10^4 \text{ kg(f)/m}^2$$

$$= \frac{76 \times 13.6 \times 1000}{100 \times 10^4} = 1.034 \text{ kg(f)/cm}^2$$

$$= \frac{76 \times 13.6 \times 9810}{100} = 10.14 \times 10^4 \text{ N/m}^2$$

(a) Absolute pressure $= (7.5 + 1.034)$
$= 8.534$ kg(f)/cm²

(b) Absolute pressure $= (7.5 \times 10^4 + 1.034 \times 10^4)$
$= 8.534 \times 10^4$ kg(f)/m²

(c) Absolute pressure head $= (75 + 10.34)$
$= 85.34$ m of water

(d) Absolute pressure head $= (5.51 + 0.76)$
$= 6.27$ m of mercury

(e) Absolute pressure $= \frac{6.27}{0.76}$
$= 8.25$ atmospheres

(f) Absolute pressure $= (73.58 \times 10^4 + 10.14 \times 10^4)$
$= 83.72 \times 10^4$ N/m²

Example 2.2. *Find the depth of a point below water surface in sea where pressure intensity is 1.006 MN/m². Specific gravity of sea water = 1.025*

Solution

Depth of sea water above the point is

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

$$p = 1.006 \text{ MN/m}^2$$
$$= 1.006 \times 10^6 \text{ N/m}^2$$
$$w = (1.025 \times 9\,810) \text{ N/m}^2$$
$$= 1.006 \times 10^4 \text{ N/m}^3$$

$$h = \frac{1.006 \times 10^6}{1.006 \times 10^4} = 100 \text{ m}$$

Example 2.3. *Convert a pressure head of 100 m of water to (a) kerosene of specific gravity 0.81, (b) carbon tetrachloride of specific gravity 1.6.*

Solution

From Eq. 2.10 (a)

$$h_1 S_1 = h_2 S_2$$

Thus by substitution

(a) $100 \times 1 = h_2 \times 0.81$

$$h_2 = \frac{100}{0.81}$$
$$= 123.46 \text{ m of kerosene}$$

(b) $100 \times 1 = h_2 \times 1.6$

$$h_2 = \frac{100}{1.6} = 62.5 \text{ m of carbon tetrachloride}$$

Example 2.4. *The left leg of a U-tube mercury manometer is connected to a pipe-line conveying water, the level of mercury in the leg being 0.6 m below the center of pipe-line, and the right leg is open to atmosphere. The level of mercury in the right leg is 0.45 m above that in the left leg and the space above mercury in the right leg contains Benzene (specific gravity 0.88) to a height of 0.3 m. Find the pressure in the pipe.*

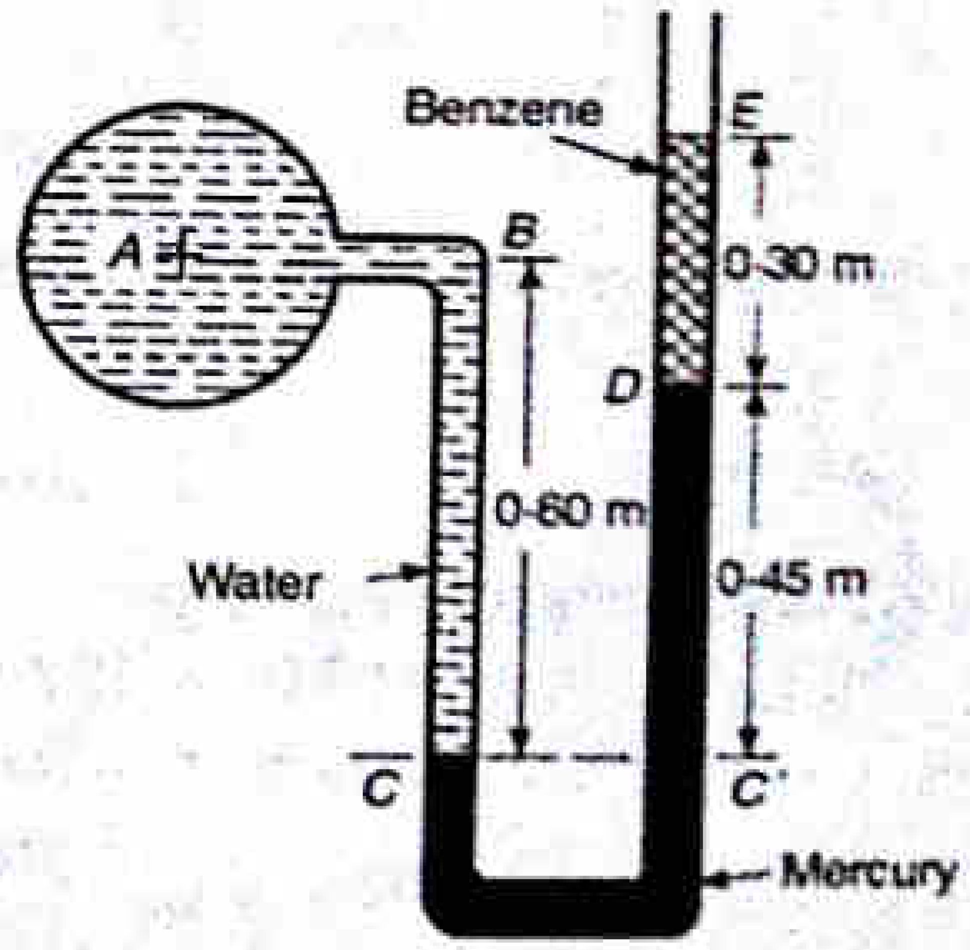


Figure Ex. 2.4

Solution

In the accompanying figure the *pressures at C and C′* are equal. Thus computing the pressure heads at C and C′ from either side and equating the same, we get

$$\frac{p_A}{w} + 0.6 = 0.45 \times 13.6 + 0.3 \times 0.88$$

or

$$\frac{p_A}{w} = 5.784 \text{ m of water}$$

$$\therefore \quad p_A = (5.784 \times 9\,810)$$
$$= 5.674 \times 10^4 \text{ N/m}^2$$

or
$$p_A = (5.784 \times 1000)$$
$$= 5.784 \times 10^3 \text{ kg (f)/m}^2$$
$$= 0.578\,4 \text{ kg (f)/cm}^2$$

Example 2.5. *As shown in the accompanying figure, pipe M contains carbon tetrachloride of specific gravity 1.594 under a pressure of 1.05 kg (f)/cm² and pipe N contains oil of specific gravity 0.8. If the pressure in the pipe N is 1.75 kg (f)/cm² and the manometric fluid is mercury, find the difference x between the levels of mercury.*

Solution

Equate the pressure heads at Z and Z′ as shown in the fig. Ex. 2.5

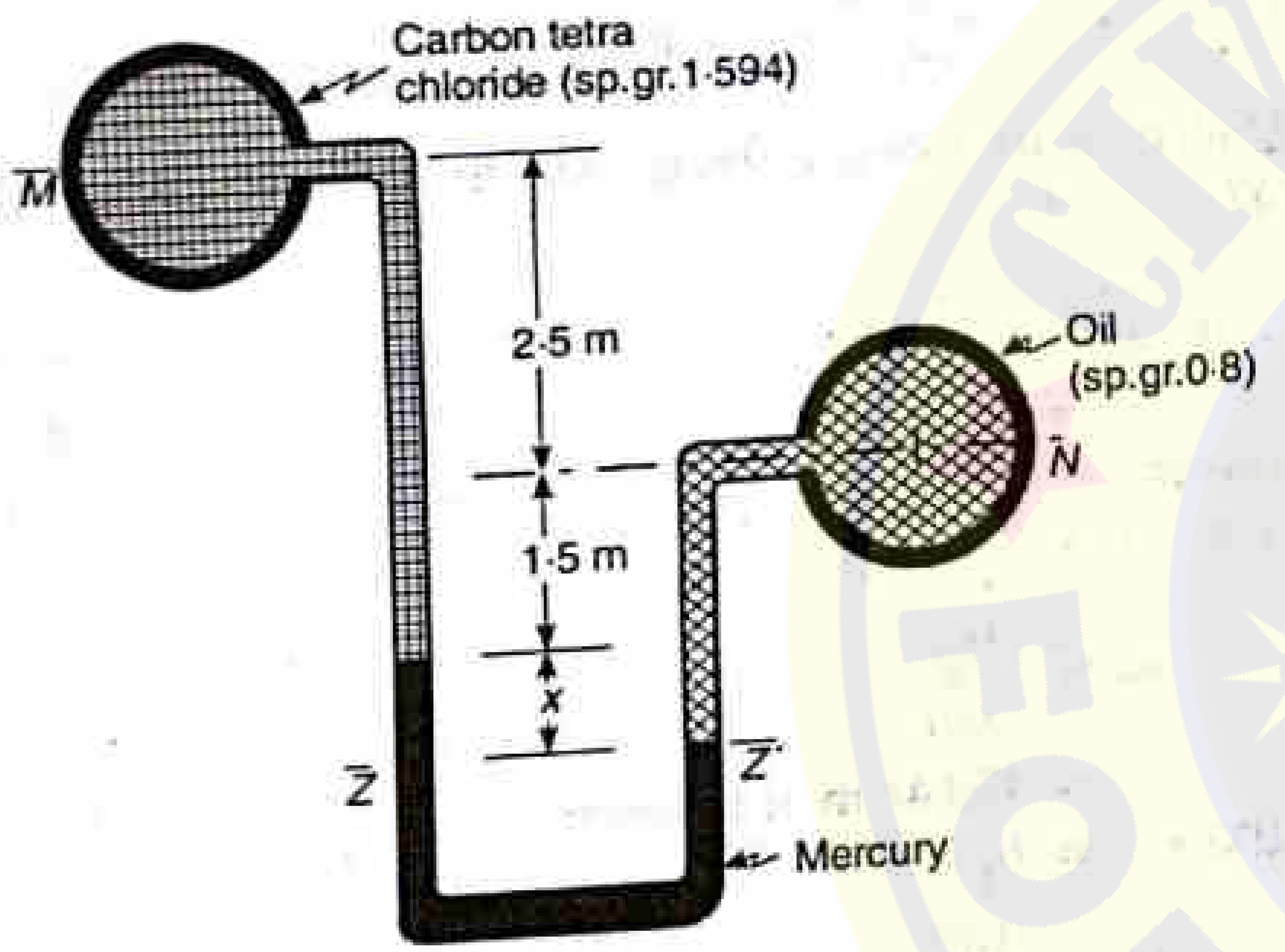


Figure Ex. 2.5

Pressure head at Z in terms of water column

$$= \left[\frac{1.05 \times 10^4}{1000} + (2.5 + 1.5) \times 1.594 + x(13.6)\right] = 16.876 + 13.6x$$

Similarly pressure head at Z′ in terms of water column

$$= \left[\frac{1.75 \times 10^4}{1000} + (1.5 \times 0.8) + x(0.8)\right] = 18.7 + 0.8x$$

Thus equating the two, we get

$$16.876 + 13.6x = 18.7 + 0.8x$$

or
$$12.8x = 1.824$$

$$\therefore \quad x = \frac{1.824}{12.8} = 0.1425 \text{ m} = 14.25 \text{ cm}$$

Example 2.6. *The pressure between two points A and B in a pipe conveying oil of specific gravity measured by an inverted U-tube. The column connected to point B stands 1.6 m higher than that at po A commercial pressure gage attached directly to the pipe at A reads 1.125 kg (f)/cm²; determine its re when attached directly to the pipe at B.*

Solution

The difference of pressure head between the two points A and B is equal to 1.6 m of oil.

$$\therefore \quad \frac{p_B}{w} - \frac{p_A}{w} = (1.6 \times 0.8) = 1.28 \text{ m of water}$$

But
$$p_A = 1.125 \text{ kg (f)/cm}^2$$

$$\therefore \quad \frac{p_A}{w} = \frac{1.125 \times 10^4}{1000} = 11.25 \text{ m of water}$$

Thus by substitution, we get

$$\frac{p_B}{w} - 11.25 = 1.28$$

or
$$\frac{p_B}{w} = 12.53 \text{ m of water}$$

$$\therefore \quad p_B = (12.53 \times 1000) \text{ kg (f)/m}^2$$
$$= \frac{12.53 \times 1000}{10^4} = 1.253 \text{ kg (f)/cm}^2$$

Example 2.7. *Two pipes as shown in Fig. Ex. 2.7 convey toluene of specific gravity 0.875 and water respectively. Both the liquids in the pipes are under pressure. The pipes are connected to a U-tube manometer and the hoses connecting the pipes to the tubes are filled with the corresponding liquids. Find the difference of pressure in two pipes if the level of manometric liquid having a specific gravity 1.25 is 2.25 m higher in the right limb than the lower level of toluene in the left limb of the manometer.*

Solution

Let p_A and p_B be the pressure intensities at the centre of the pipes A and B respectively.

Since points D and D′ lie at the same horizontal plane and in the same continuous static mass of liquid, the pressure heads at these two points will be equal.

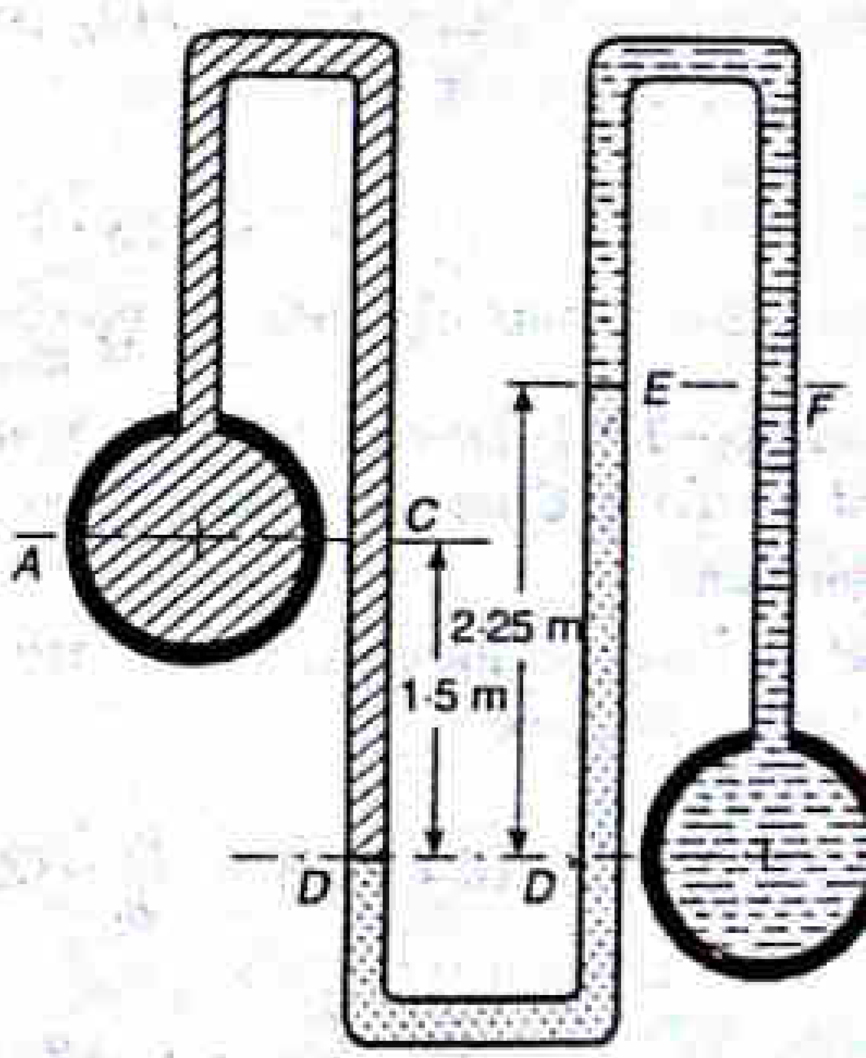


Figure Ex. 2.7

$$\text{Pressure head at } D = \frac{p_A}{w} + (1.5 \times 0.875)$$

$$\text{Pressure head at } D' = \frac{p_B}{w} - 2.25 + (2.25 \times 1.25)$$

The pressure heads have been calculated in terms of water column. Equating the pressure heads, we get

$$\frac{p_A}{w} + (1.5 \times 0.875) = \frac{p_B}{w} - 2.25 + (2.25 \times 1.25)$$

or
$$\frac{p_A}{w} + 1.3125 = \frac{p_B}{w} + 0.5625$$

or
$$\frac{p_A}{w} - \frac{p_B}{w} = 0.75$$

or
$$(p_B - p_A) = (0.75 \times 1000) = 750 \text{ kg(f)/m}^2$$

or
$$(p_B - p_A) = 0.75 \times 9\,810 = 7\,357.5 \text{ N/m}^2 = 7.357\,5 \text{ kN/m}^2$$

Example 2.8. *Two tanks are filled with water of specific weight 9.81 kN/m³. The bottoms of the tanks are connected to an inverted U-tube containing oil weighing 7.85 kN/m³. Find the difference in pressure between the two tanks when the manometer gives a reading of 0.8 m.*

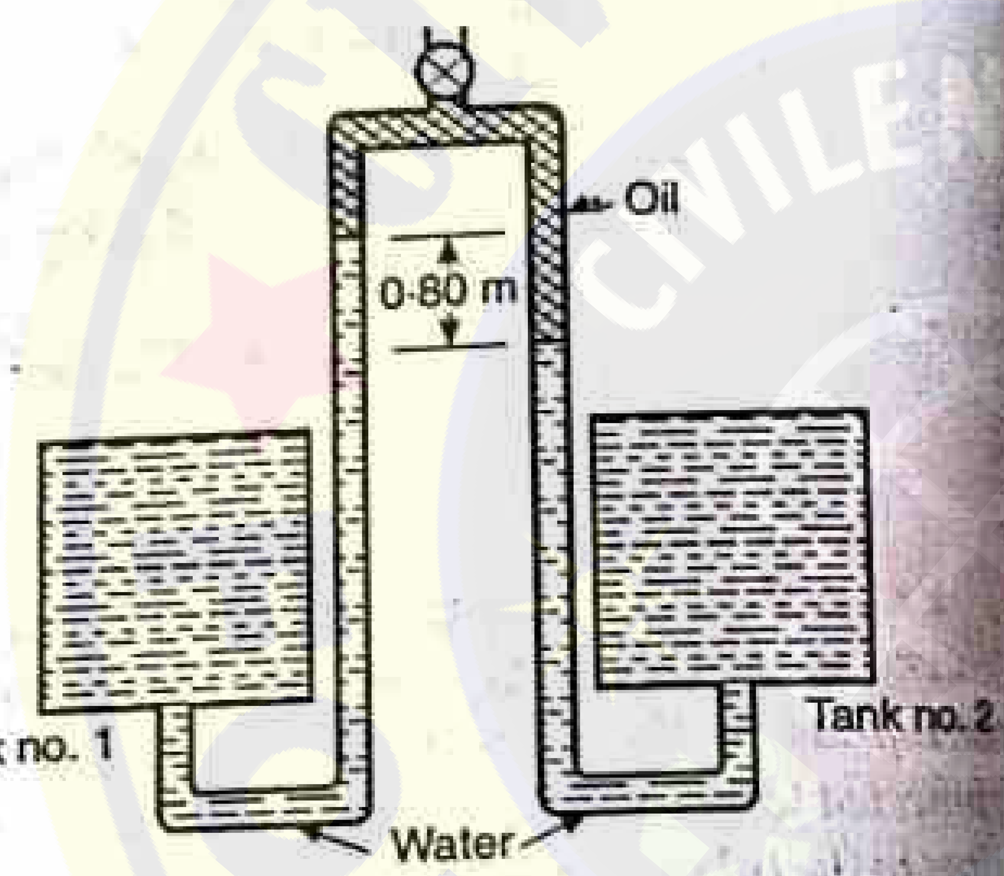


Figure Ex. 2.8

Solution

Using Eq. 2.50, the difference of pressure head in the two tanks is given by

$$\frac{p_1}{w} - \frac{p_2}{w} = x(S_1 - S_2)$$

$$= 0.8\left(1 - \frac{7.85}{9.81}\right)$$

$$= 0.16 \text{ m of water} = 16 \text{ cm of water}$$

Example 2.9. *In the accompanying figure, fluid A is water, fluid B is oil of specific gravity 0.85, Z = 0.7 m and y = 1.5 m. Compute pressure difference between m and n.*

Solution

Let the height of the common surface above the point m be x. Since pressure head at T = pressure head at T'; we have

$$\frac{p_m}{w} - x - (Z \times 0.85) = \frac{p_n}{w} - (Z + x - y)$$

or
$$\frac{p_m}{w} - \frac{p_n}{w} = y - Z(1 - 0.85)$$

$$= 1.5 - 0.7(0.15)$$

$$= (1.5 - 0.105)$$

$$= 1.395 \text{ m of water}$$

or
$$(p_m - p_n) = \frac{1.395 \times 1000}{10^4} = 0.1395 \text{ kg(f)/cm}^2$$

or
$$(p_m - p_n) = 1.395 \times 9.810 = 13.685 \text{ kN/m}^2$$

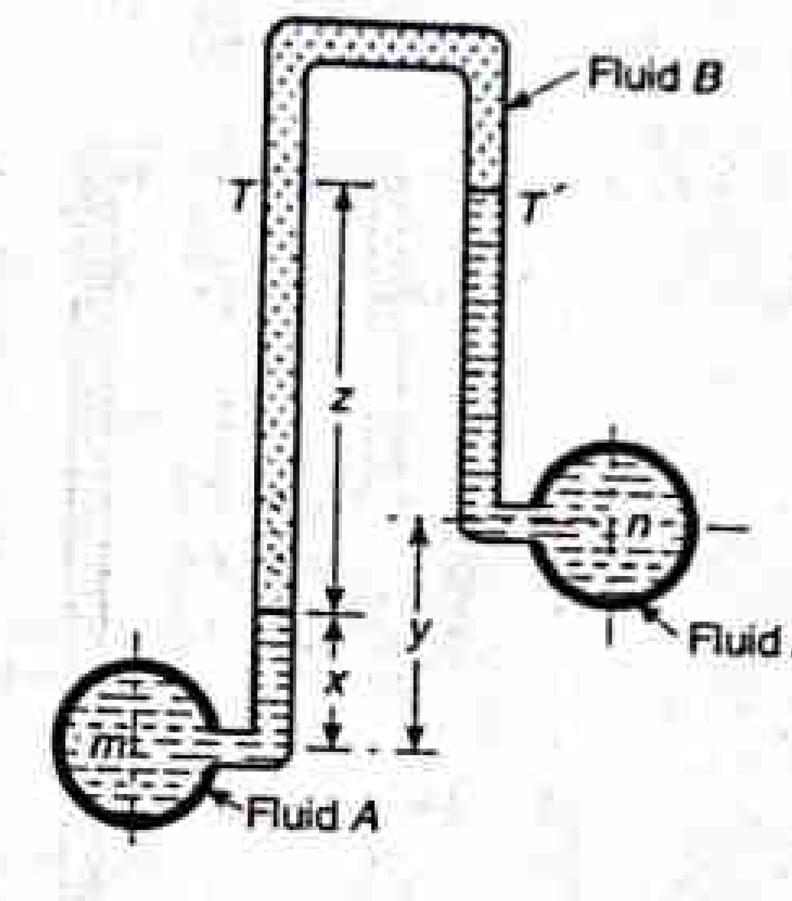


Figure Ex. 2.9

Example 2.10 (*a*) *Water fills the vessels shown in the Fig. Ex. 2.10 and a portion of the connecting tube. If the manometric liquid is oil of specific gravity 0.9, find the diference in pressure intensity at m and n when h = 1.25 m and Z = 0.3 m.*

(*b*) *If in the same figure instead of water there is mercury and the manometric liquid used has a specific gravity of 1.6, find the difference in pressure intensity at m and n when h = 0.6 m and Z = 1.0 m.*

Solution

(a) Pressure head at X = Pressure head at X'

Thus if the height of the common surface between oil and water in the right limb above point m is y, then

$$\frac{p_m}{w} - Z - y = \frac{p_n}{w} - (h + y) - Z(0.9) \quad \text{...(i)}$$

or
$$\frac{p_n}{w} - \frac{p_m}{w} = h - Z(1 - 0.9)$$

$$= 1.25 - 0.3(1 - 0.9)$$

$$= 1.22 \text{ m of water}$$

or
$$p_n - p_m = \frac{1.22 \times 1000}{10^4} = 0.122 \text{ kg(f)/cm}^2$$

or
$$p_n - p_m = 1.22 \times 9\,810 = 11\,968.2 \text{ N/m}^2 = 11.968\,2 \text{ kN/m}^2$$

(b) Substituting the corresponding values in Eq. (i) noted above, we get

$$\frac{p_m}{w} - (Z + y) \times 13.6 = \frac{p_n}{w} - (h + y) \times 13.6 - Z\,(1.6)$$

or
$$\frac{p_m}{w} - \frac{p_n}{w} = Z\,(13.6 - 1.6) - h\,(13.6)$$

$$= 1.0\,(13.6 - 1.6) - 0.6\,(13.6)$$

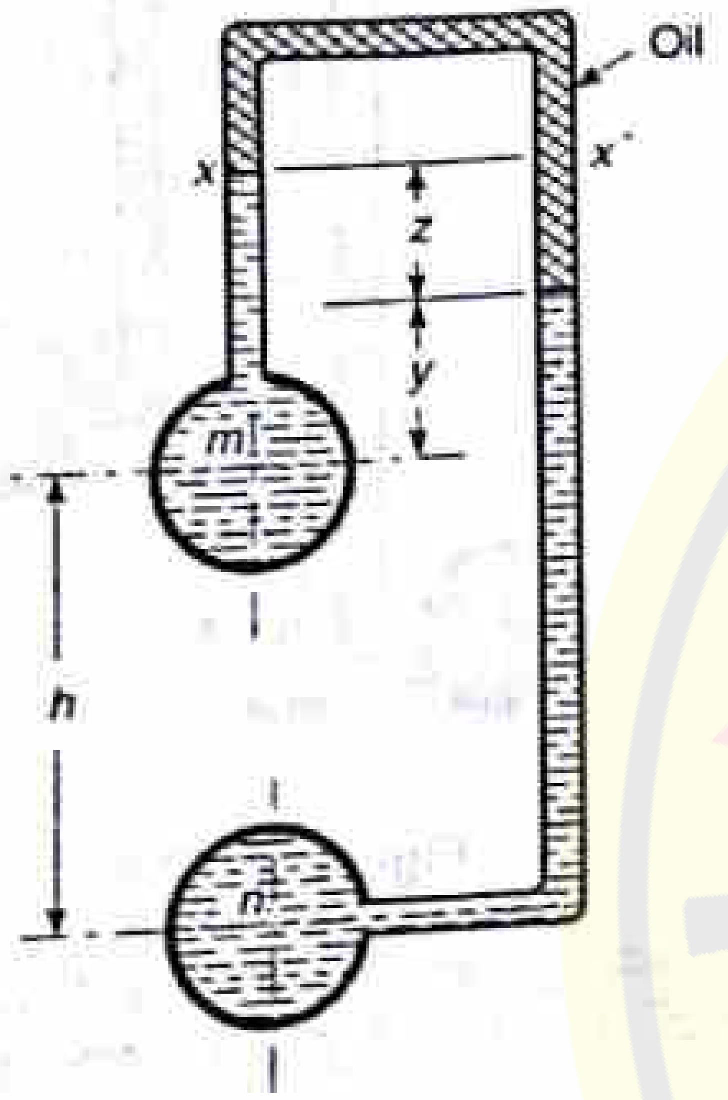


Figure Ex.2.10

$$= (12.0 - 8.16) = 3.84 \text{ m of water}$$

or
$$(p_m - p_n) = \frac{3.84 \times 1000}{10^4} = 0.384 \text{ kg(f)/cm}^2$$

or
$$(p_m - p_n) = 3.84 \times 9\,810 = 37\,670.4 \text{ N/m}^2 = 37.670\,4 \text{ kN/m}^2.$$

Example 2.11. *Figure Ex. 2.11 shows a differential gage. X and Y are connected to two different sources of pressure. With the equal pressure at X and Y, tops of kerosene columns stand at the common level J-J and water at 0-0. Points X and Y are at the same level. Find the difference of pressure head between X and Y in mm of water if h is 0.3 m. Take the reservoir cross-section 100 times that of glass tube.*

Solution

Let A and a be the cross-sectional areas of the reservoir and the glass tube respectively.

$\therefore$
$$(A/a) = 100$$

Let l be the vertical height of kerosene column above 0–0 in the right limb. Also let Z and Z' be the two points in the left and the right limbs respectively at the same level and at height $(h/2)$ above J-J as shown in Fig. Ex. 2.11.

$\therefore$ Pressure head at Z = Pressure head at Z'.

The pressure head at Z will be equal to $\frac{p_x}{w} - t - y - \left(l - \frac{y}{2}\right) 0.82 - h\,(0.82)$

where t is the vertical depth of the points X and Y below the common surface between kerosene and water in the right limb.

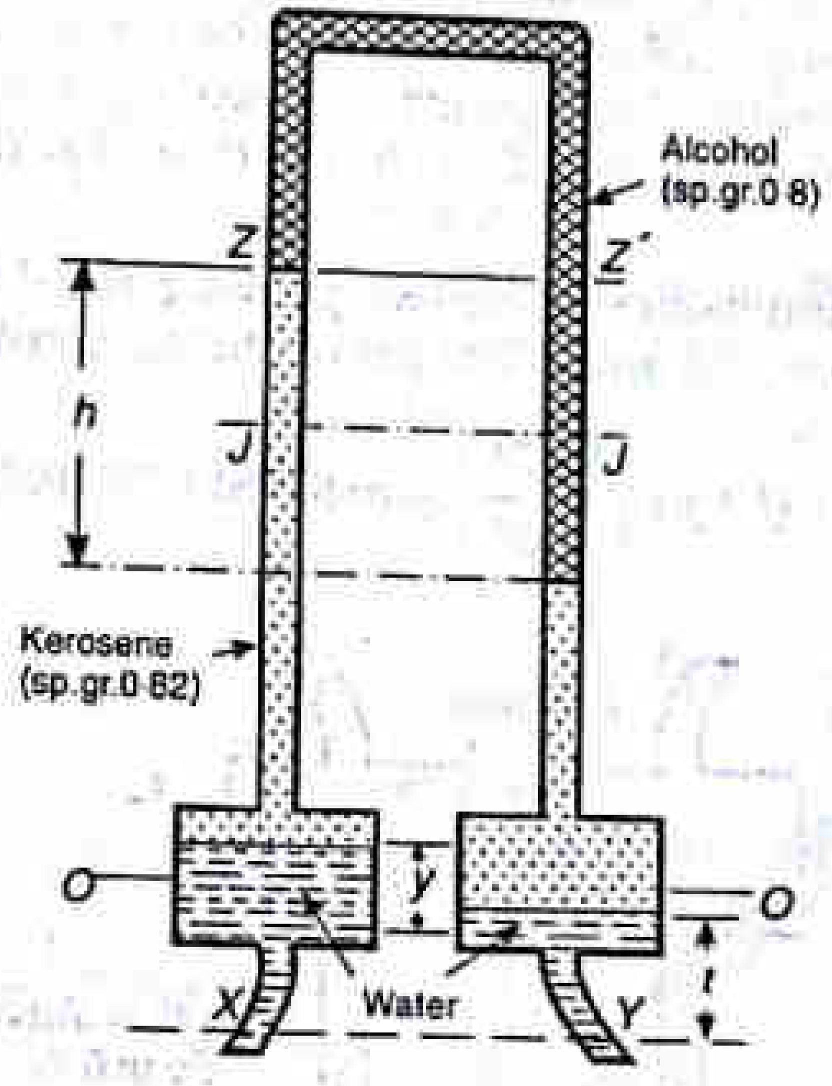


Figure Ex.2.11

Similarly the pressure head at point Z' will be equal to $\frac{p_y}{w} - t - \left(l + \frac{y}{2}\right) 0.82 - h\,(0.80)$

Thus, we have

$$\frac{p_x}{w} - t - y - \left(l - \frac{y}{2}\right) 0.82 - h\,(0.82)$$

$$\frac{p_y}{w} - t - \left(l + \frac{y}{2}\right) 0.82 - h\,(0.8)$$

or
$$\frac{p_x}{w} - \frac{p_y}{w} = h\,(0.82 - 0.80) + y(1 - 0.82)$$

Further
$$A \times y = a \times h$$

and
$$h = 0.3 \text{ m}$$

$\therefore$
$$y = \frac{a}{A} \times h = \frac{0.3}{100} = 0.003 \text{ m}$$

By substituting these values, we get

$$\frac{p_x}{w} - \frac{p_y}{w} = 0.3(0.82 - 0.80) + 0.003(1 - 0.82)$$

$$= 6.54 \times 10^{-3} \text{ m} = 6.54 \text{ mm of water.}$$

Example 2.12. *A two-liquid double column enlarged -ends manometer is used to measure a high precision ressure difference between two points of a pipeline containing gas under pressure. The basins are partly filled ith methyl alcohol of specific gravity 0.78 and the lower portion of the U–tube is filled with mercury of pecific gravity 13.6. The specific weight of the gas which is methane is 0.476 kg(f)/m³. Find the pressure ifference if the U-tube reading is 30 mm and the diameter of the basin is 15 times that of the U-tube.*

Solution

Let p_1 and p_2 be the pressure intensities at the two points 1 and 2 in the pipeline. Thus equating he pressure heads at the two points Z and Z′, as shown in the accompanying figure, we get

$$\frac{p_1}{w} + (l + y)\left(\frac{0.476}{1000}\right) + m\,(0.78) + 30\,(0.78)$$

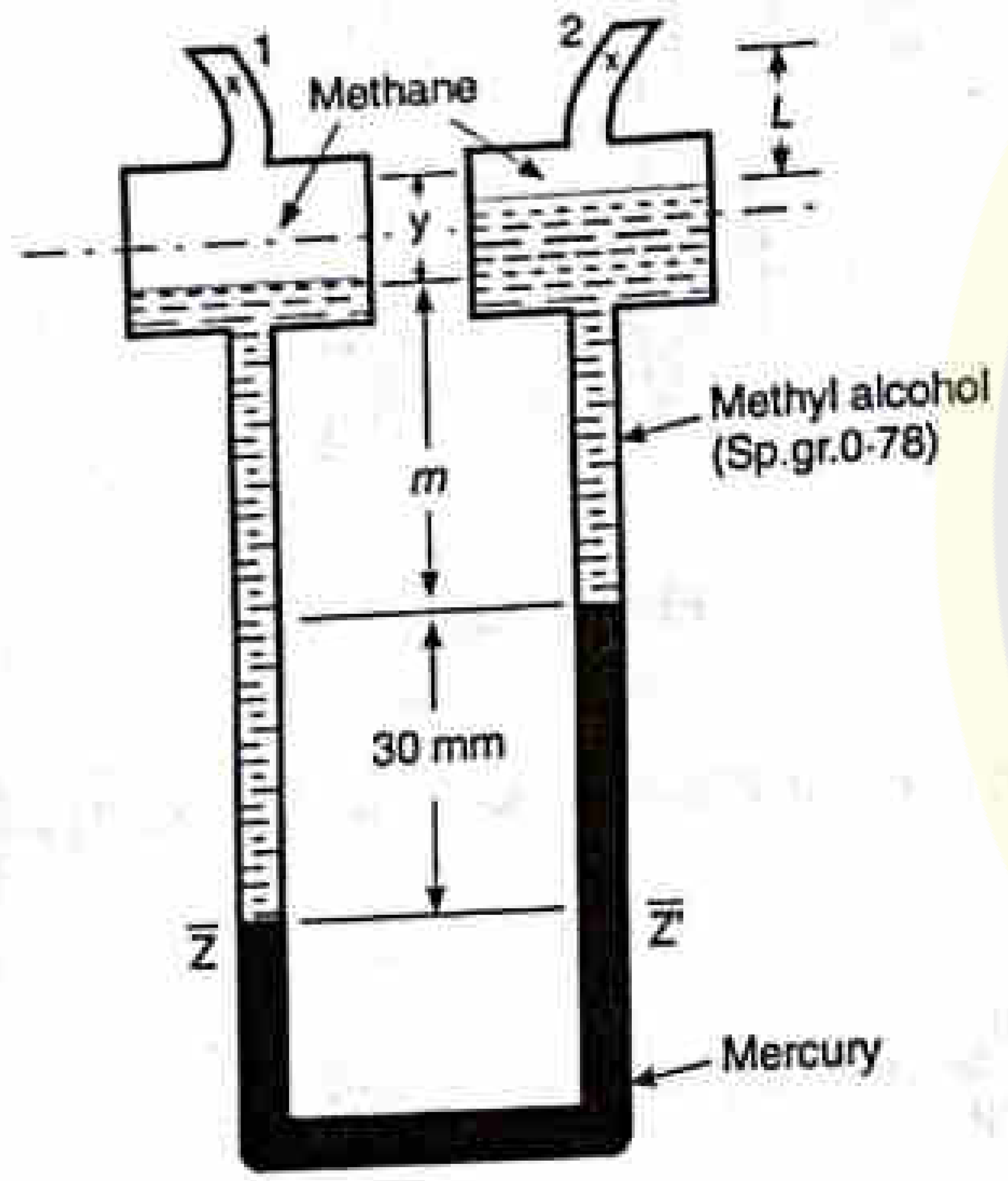


Figure Ex. 2.12

$$= \frac{p_2}{w} + l\left(\frac{0.476}{1000}\right) + (m + y)\,0.78 + 30\,(13.6)$$

or
$$\frac{p_1}{w} - \frac{p_2}{w} = 30(13.6 - 0.78) + y\,(0.78 - 0.476 \times 10^{-3})$$

$$= 384.6 + y\,(0.779\,5)$$

Further
$$A \times y = 30 \times a$$

or
$$y = 30\,\frac{a}{A} \text{ mm}$$

But
$$A = (15)^2\,a = 225\,a$$

$$\therefore \quad y = \frac{30}{225}$$

By substituting these values, we get

$$\frac{p_1}{w} - \frac{p_2}{w} = 384.6 + \frac{30}{225}(0.7795)$$

$$= 384.704 \text{ mm of water}$$

$$= 38.470\,4 \text{ cm of water}$$

Example 2.13. *A manometer consists of an inclined glass tube which is connected to a metal cylinder standing upright, and manometric liquid fills the apparatus to a fixed zero mark on the tube when both cylinder and the tube are open to the atmosphere. The upper end of the cylinder is then connected to a gas supply at a pressure p and the liquid rises in the tube.*

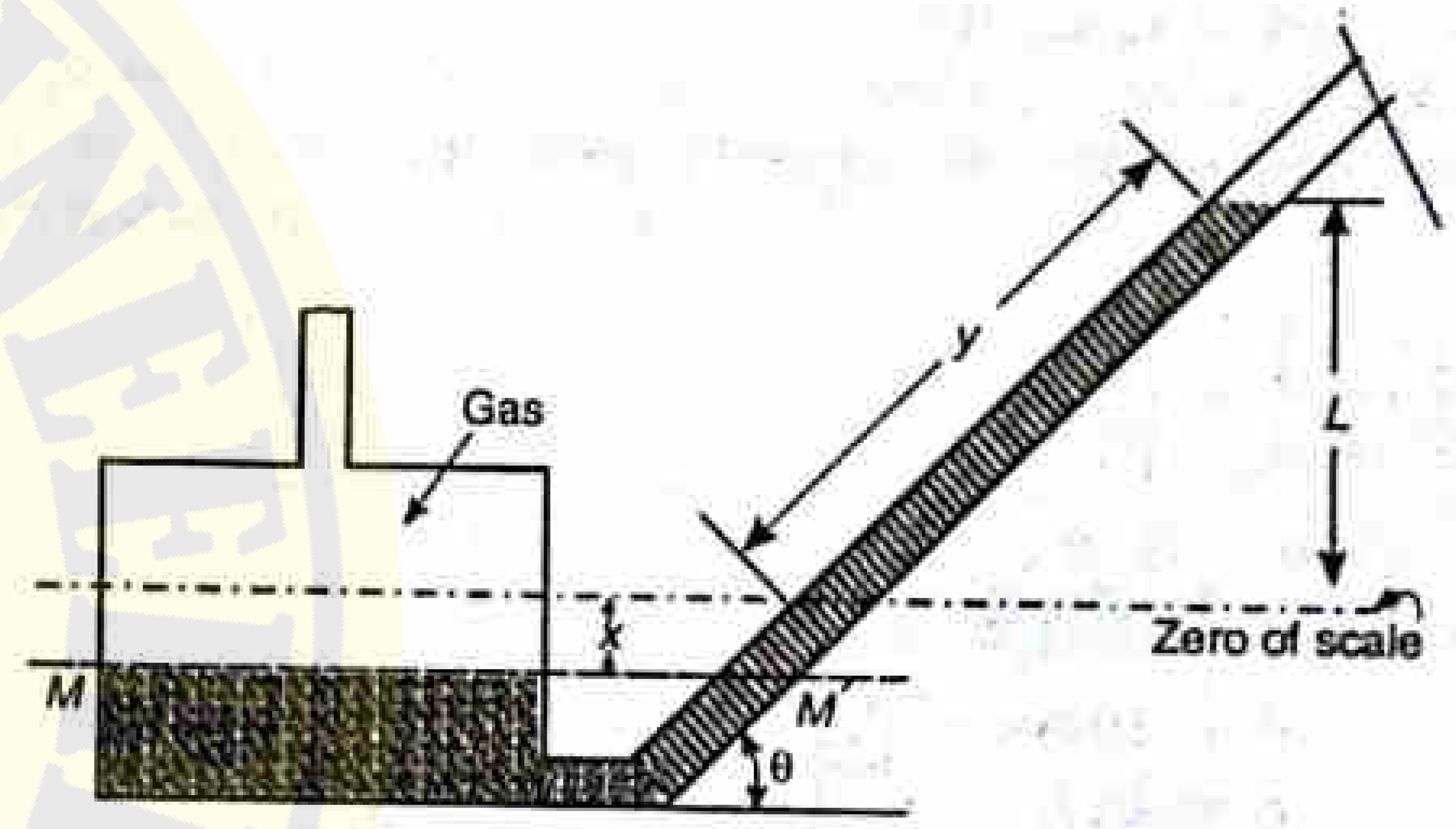


Figure Ex. 2.13

Find an expression for the pressure p in cm of water when the liquid reads y cm in the tube, in terms of the inclination θ of the tube, the specific gravity of the liquid S, and the ratio ρ of the diameter of the cylinder to the diameter of the tube. Hence determine the value of ρ so that the error due to disregarding the change in level in the cylinder will not exceed 0.1% when θ = 30°.

Solution

When the level of liquid rises in the tube by y cm, the level of the liquid will fall in the reservoir by certain amount; say x, which will be given by

$$x\left(\frac{\pi D^2}{4}\right) = y\left(\frac{\pi d^2}{4}\right)$$

or
$$x = y\left(\frac{d}{D}\right)^2 = \frac{y}{\rho^2}$$

Further the vertical rise corresponding to y cm of rise $= l = (y \sin\theta)$.

Thus equating the pressure heads at the two points M and M', and neglecting the height of the gas column, we get

$$\frac{p}{w} = xS + (y\sin\theta)\,S$$

$$= \left(\frac{y}{\rho^2} + y\sin\theta\right) S$$

$$= y\,(\rho^{-2} + \sin\theta)S$$

where p and w are the pressure intensity of the gas and the specific weight of water respectively.

When the change in the level of the liquid in the cylinder is disregarded, the pressure head of the gas will become

$$\frac{p}{w} = (y\sin\theta)S$$

For the error due to disregarding the change in level of liquid in the cylinder not to exceed 0.1%,

$$\frac{99.9}{100}\left(\sin\theta + \frac{1}{\rho^2}\right) yS = (y\sin\theta)\,S$$

But $\theta = 30°$

Thus

$$\frac{99.9}{100}\left(\frac{1}{2} + \frac{1}{\rho^2}\right) = \frac{1}{2}$$

or $\dfrac{0.999}{\rho^2} = \dfrac{0.1}{2\times100}$

or $\rho^2 = 1998$

$\therefore$ $\rho = 44.7.$

Example 2.14. *For a gage pressure at A of –0.15 kg(f)/cm², determine the specific gravity of the gage liquid B in the Fig. Ex. 2.14.*

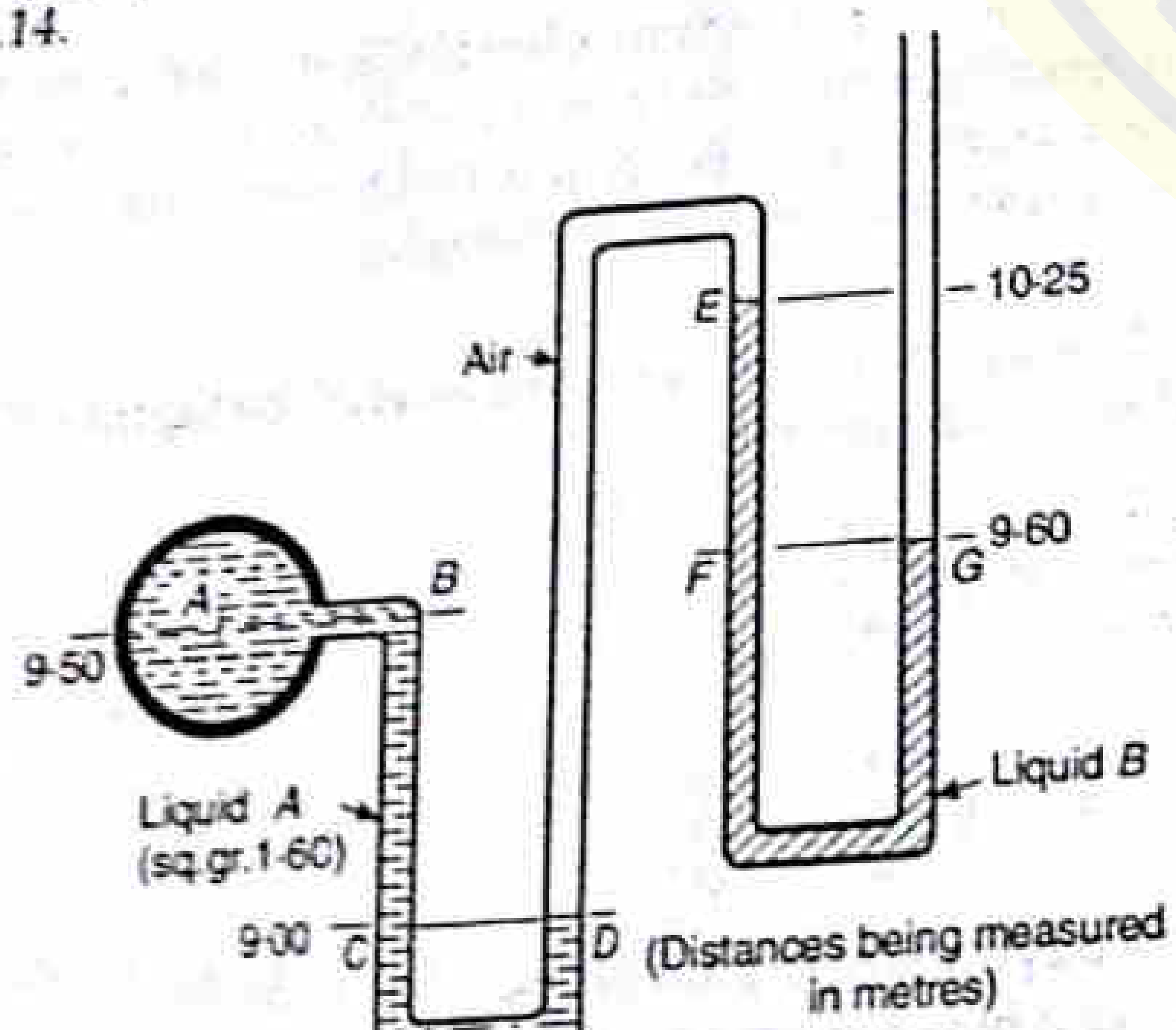


Figure Ex. 2.14

Solution

Pressure at C = Pressure at D

Thus

$-(0.15)\times10^4 + (1000\times1.6\times0.5) = p_D$

or $p_D = -0.7\times103\ \text{kg(f)/m}^2$

Between points D and E, since there is an air column which can be neglected. Thus

$$p_D = p_E$$

Also, pressure at F = pressure at G

But point G being at atmospheric pressure,

$$p_G = 0 = p_F$$

Thus $p_F = p_E + S\times1000(10.25 - 9.60) = 0$

or $-0.7\times103 + S\times1000\,(0.65) = 0$

$\therefore$ $S = \dfrac{0.70}{0.65} = 1.077.$

Example 2.15. *The pressure head at level A – A is 0.1 m of water and the unit weight of gas and air are 0.5607 kg(f) m³ and 1.2608 kg(f)/m³ respectively. Determine the reading of the water in the U-tube gage which measures the gas pressure at level B in the Fig. 2.15.*

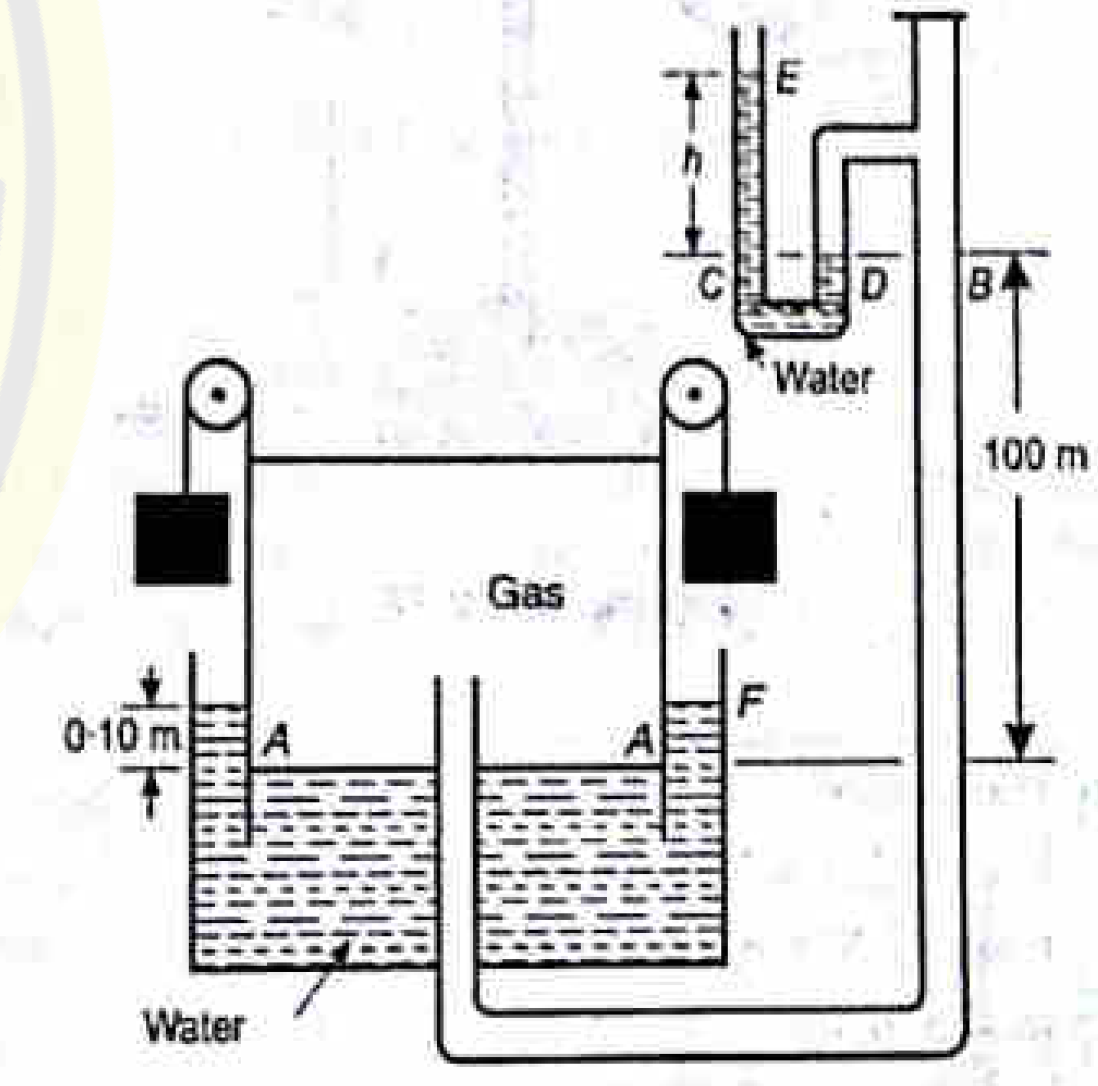


Figure Ex. 2.15

Solution

Assume that the values of specific weight for air and gas remain constant for the 100 m difference of elevation. Because the specific weights of gas and air are of the same order of magnitude, the change in atmospheric pressure with altitude must be taken into account.

Thus Absolute pressure at C = Absolute pressure at D or Atmospheric pressure $p_E + 1000\times h$ = Absolute pressure at $A - (0.5607\times100)$

The absolute pressure at A will now be evaluated in terms of the atmospheric pressure at E, obtaining first the atmospheric pressure at F and then p_A.

Absolute pr. p_A = [atmospheric pr. P_E +1.2608 (h +100 – 0.10)] + (0.10 × 1000)

By substituting the value of absolute pr. p_A in the above equation and cancelling atmospheric pr. p_E, we get

$$1000 \times h = 1.2608\,(h + 99.9) + (1000 \times 0.10) - (0.5607 \times 100)$$

or
$$(1000 - 1.2608)h = (1.2608 \times 99.9) + 100 - 56.07 = 169.884$$

or
$$h = \frac{169.884}{998.7392}\text{ m of water} = \frac{169.884 \times 100}{998.7392}\text{ cm of water}$$
$$= 17.0098 \text{ cm of water.}$$

Example 2.16. *Point A is 0.25 m below the surface of the liquid of specific gravity 1.25, in the vessel as shown in Fig. Ex. 2.16. What is the pressure at A if a liquid of specific gravity 1.36 rises 2.1 m in the tube?*

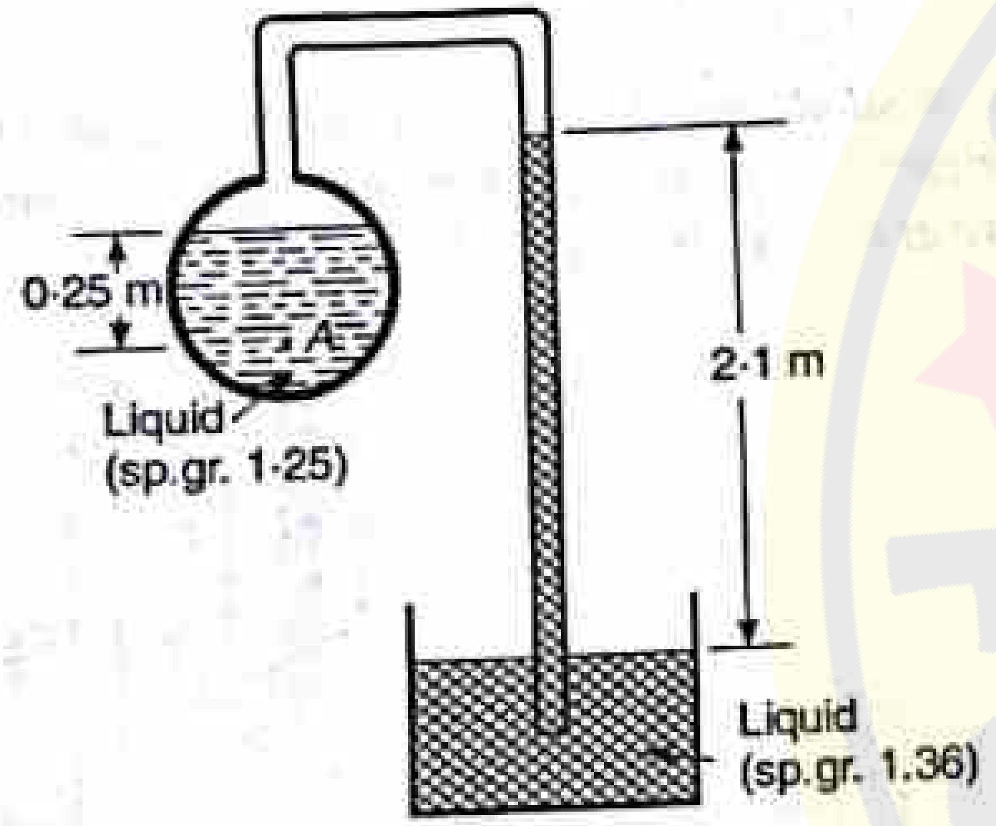


Figure Ex. 2.16

Solution

The pressure of the air in the vessel

$$= -(2.1 \times 1.36) \text{ m of water}$$
$$= -2.856 \text{ m of water.}$$

Thus pressure at point A in the vessel

$$= [-2.856 + 0.25 \times 1.25] \text{ m of water} = [-2.856 + 0.312\,5] \text{ m of water}$$
$$= -2.543\,5 \text{ m of water}$$
$$= -(2.543\,5 \times 9\,810)\text{N/m}^2$$
$$= -24.952 \text{ kN/m}^2$$
$$= -254\,3.5 \text{ kg(f)/ m}^2$$
$$= -0.254\,35 \text{ kg(f)/cm}^2$$

Example 2.17. *The cylinder and tubing shown in Fig. Ex. 2.17 contain oil of specific gravity 0.902. For a gage reading of 2 kg (f)/cm², what is the total weight of the piston and the slab placed on it ?*

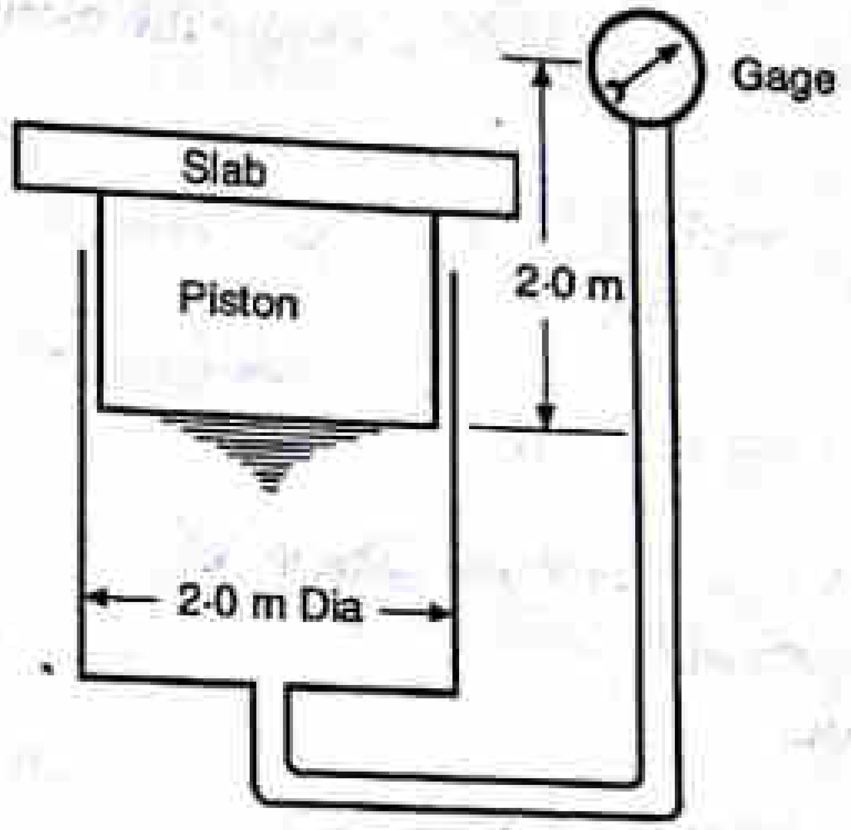


Figure Ex. 2.17

Solution

Let W be the total weight of the piston and the slab.

Since pr. below the piston = pr. at the same level in the vertical tube, we have

$$\frac{W}{\frac{\pi}{4}(2)^2} = 2 \times 10^4 + (2 \times 1000 \times 0.902)$$

$$\therefore \quad W = \pi(2 \times 10^4 + 0.180\,4 \times 10^4)$$
$$= 6.85 \times 10^4 \text{ kg(f).}$$

Example 2.18. *The compartments B and C shown in Fig. Ex. 2.18 are closed and filled with air. The barometer reads 99.572 kN/m². When the gages A and D read as indicated, what should be the value of x for gage E, if mercury is the manometric liquid in each U-tube gage ?*

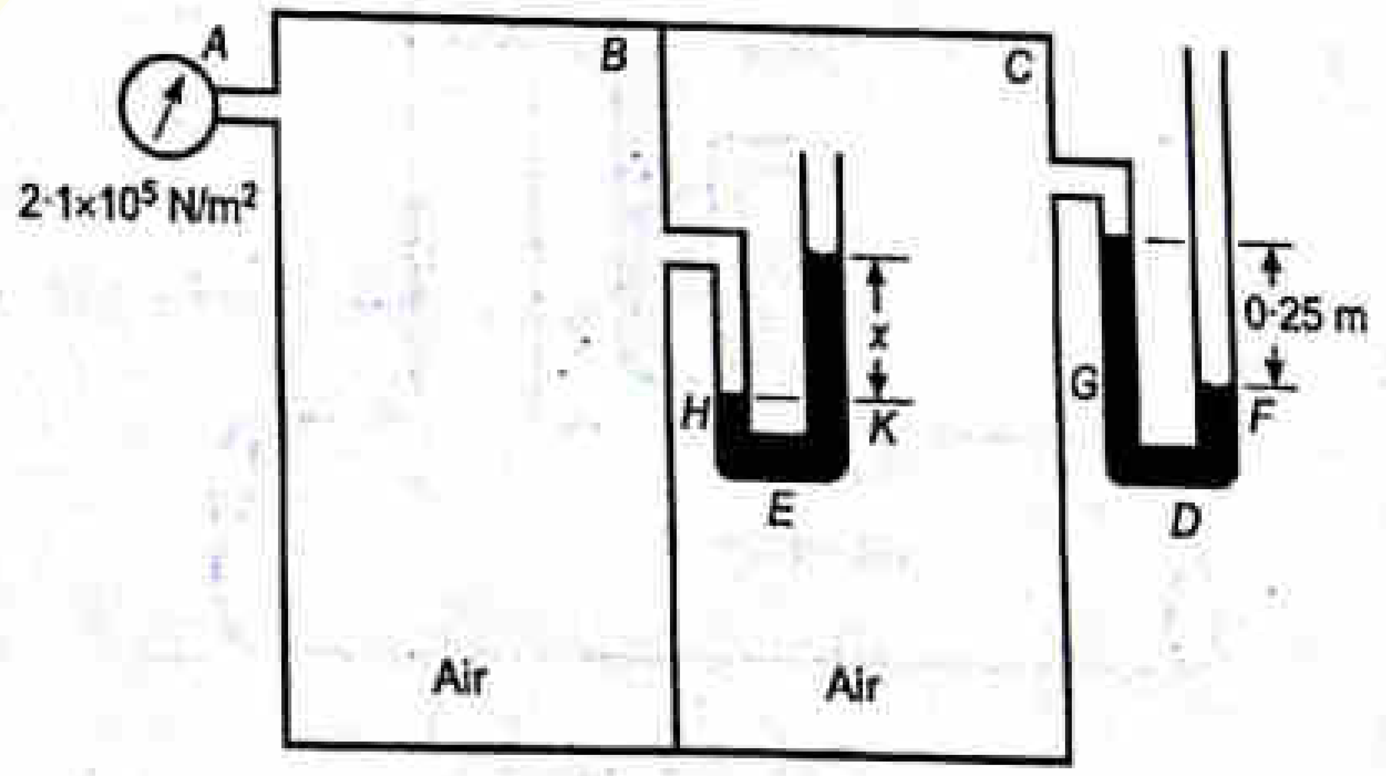


Figure Ex. 2.18

Solution

Pressure at F = Pressure at G. Thus if p is the pressure intensity of air in the compartment C in terms of absolute units, then

$$\frac{p_c}{w} + 0.25 \times 13.6 = \frac{99.572 \times 10^3}{9810}$$

or
$$\frac{p_c}{w} = (10.15 - 3.4)$$
$$= 6.75 \text{ m of water (absolute)}$$

Further pressure at K = Pressure at H

Thus in terms of absolute units

$$6.75 + x\,(13.6) = \left[\frac{2.1 \times 10^5}{9810} + \frac{99.572 \times 10^3}{9810}\right]$$

or
$$x = \frac{1}{13.6}(31.56 - 6.75) = 1.824 \text{ m}$$

Example 2.19. *The pressure of illuminating gas is measured as shown in Fig. 2.19. The U-tube at the gas main shows a pressure of 75 mm of water. What difference should the upper tube show in mm of water? The right hand columns of both gages are open to the atmosphere. The density of both air and gas may be assumed constant throughout the 100 m of elevation. Air weighs 1.33 kg(f)/m³ and the gas weighs 0.56 kg(f)/m³.*

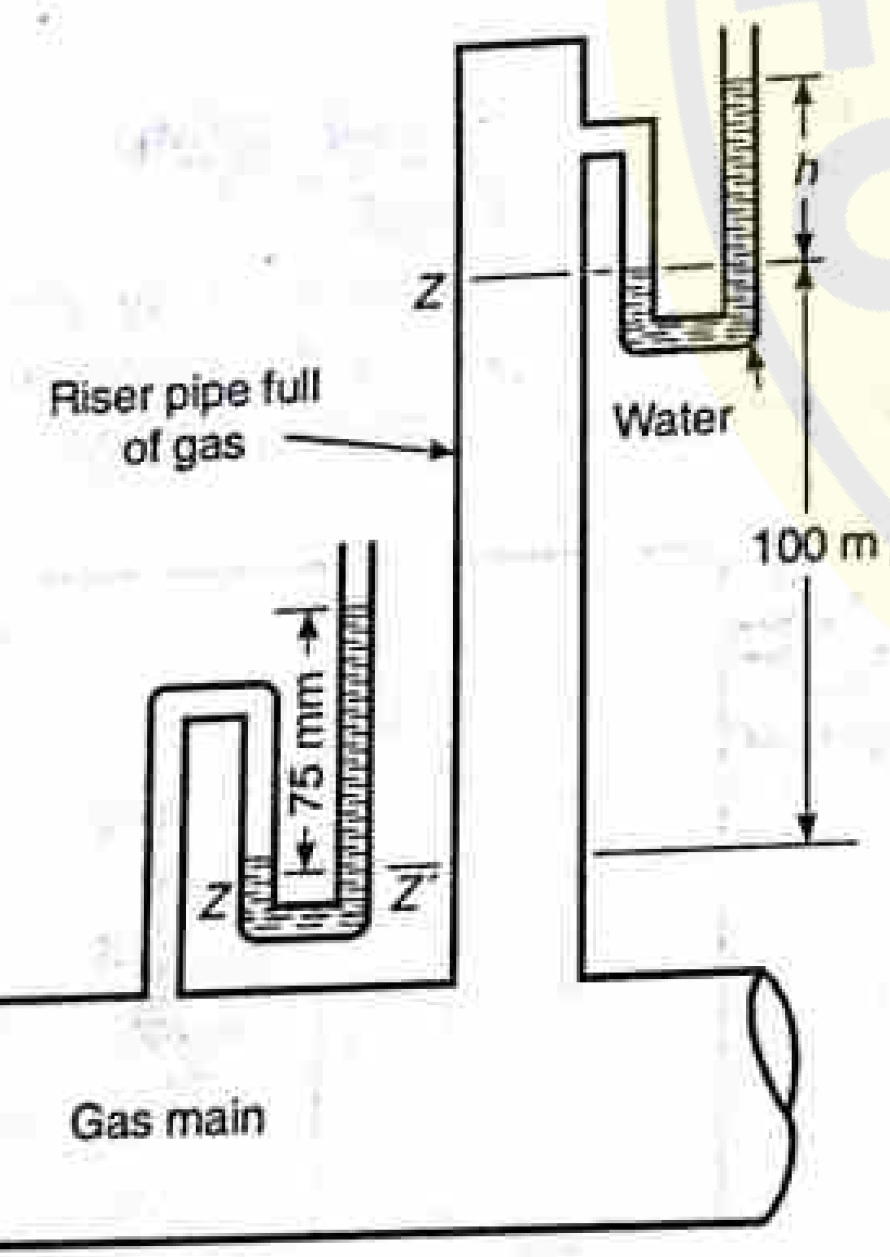


Figure Ex. 2.19

Solution

Let p_a represent the intensity of the atmospheric pressure in kg(f)/m² at an elevation of 100 m.

Thus the absolute pressure of the gas at point Z will be equal to

$$(p_a + h \times 1000) \text{ kg(f)/m}^2$$

The pressure at point Z′ in the riser pipe at 100 m below Z will be given by

$$\text{Pressure at } Z' = \text{Pressure at } Z + (100 \times 0.56) = p_a + h \times 1000 + (100 \times 0.56)$$

Further

$$\text{Pressure at } Z' = \text{Pressure at } Z''$$

Thus

$$(p_a + h \times 1000) + (100 \times 0.56) = [p_a + (1.33 \times 100)] + \frac{75}{1000} \times 1000$$

or
$$1000h = (133 + 75 - 56)$$
$$\therefore \quad h = 0.152 \text{ m}$$

Example 2.20. *For the multiple differential manometer shown in Fig. Ex. 2.20, if points A, B and C are at the same elevation, what is the difference in pressure heads in terms of water column between A and B, between A and C and between B and C?*

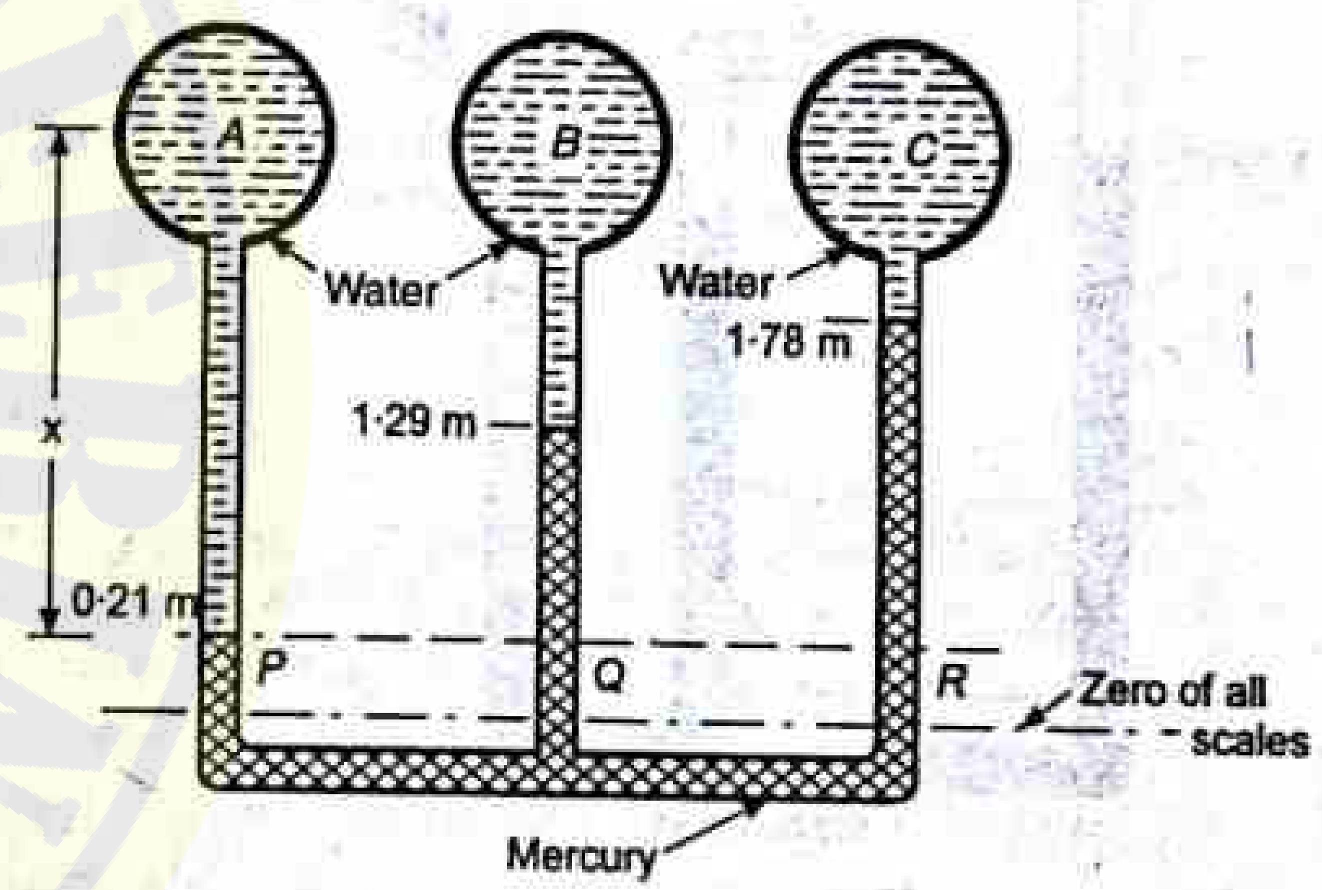


Figure Ex. 2.20

Solution

$$\text{Pr. at } P = \text{Pr. at } Q = \text{Pr. at } R$$

Thus if the depth of the point P below $A = x$, then

$$\frac{p_A}{w} + x = \frac{p_B}{w} + (x - 1.08) + (1.08 \times 13.6)$$

$$\frac{p_C}{w} = + (x - 1.57) + (1.57 \times 13.6)$$

Thus

$$\frac{p_A}{w} - \frac{p_B}{w} = 1.08\,(13.6 - 1)$$
$$= 13.608 \text{ m of water}$$

Similarly

$$\frac{p_A}{w} - \frac{p_C}{w} = 1.57\,(13.6 - 1)$$

$$= 19.782 \text{ m of water}$$

and

$$\frac{p_B}{w} - \frac{p_C}{w} = (19.782 - 13.608)$$

$$= 6.174 \text{ m of water}$$

Example 2.21. *For a compound manometer shown in Fig. Ex. 2.21, what is gage pressure at C if the manometric fluid is mercury and if the fluid in the pipe and in the tubing which connects the two U-tubes is water?*

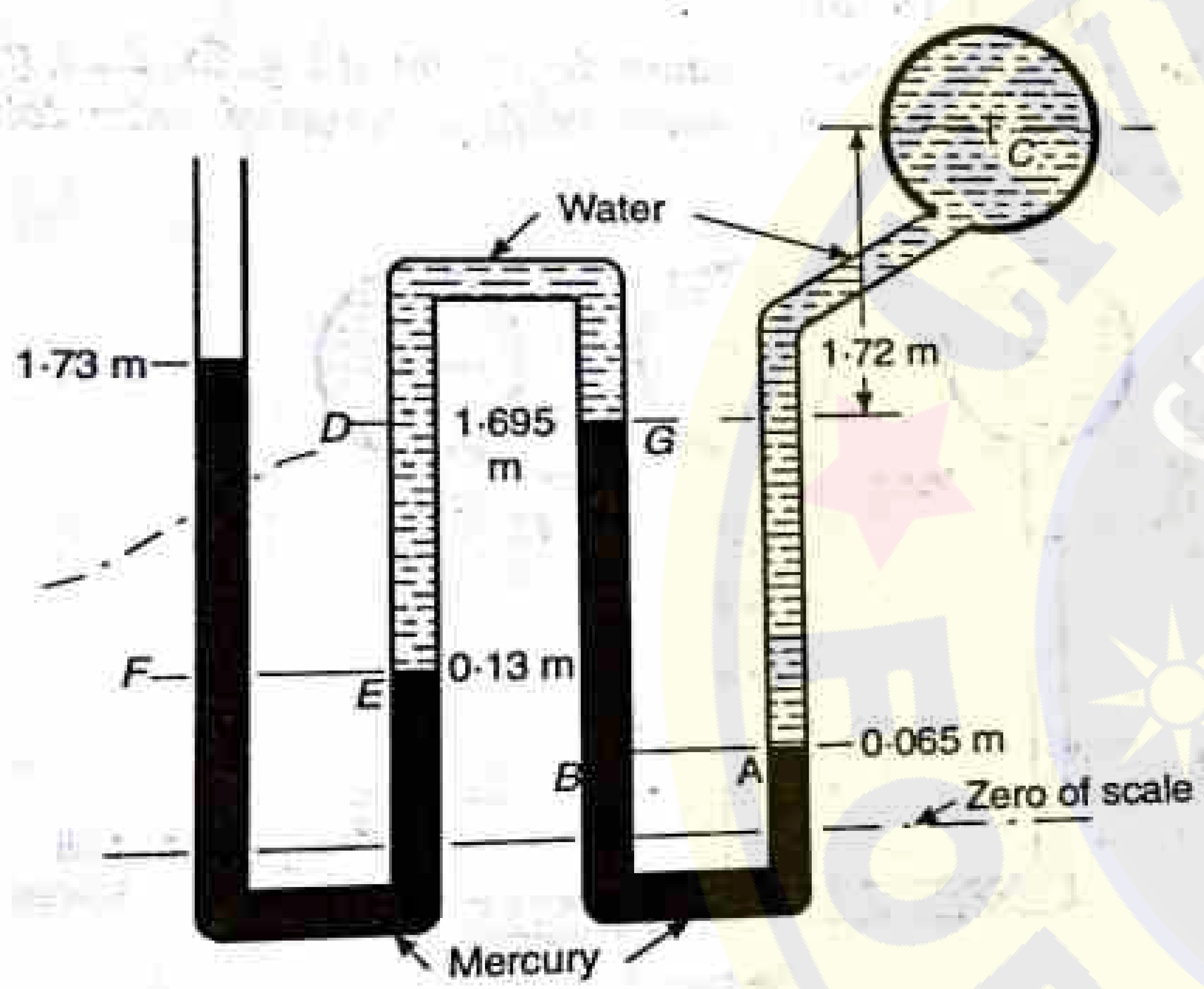


Figure Ex. 2.21

Solution

Pressure at A, p_A = Pressure at B, p_B

Thus we have

$$\frac{p_A}{w} = \frac{p_C}{w} + 1.72 + 1.695 - 0.065 = \frac{p_B}{w}$$

Further

Pressure at G,

$$\frac{p_G}{w} = \frac{p_B}{w} - (1.695 - 0.065)\,13.6$$

$$= \frac{p_C}{w} + (1.72 + 1.695 - 0.065) - (1.695 - 0.065) \times 13.6$$

Also

Pressure at G, p_G = Pressure at D, p_D

and Pressure at E, p_E = Pressure at F, p_F

But

$$\frac{p_E}{w} = \frac{p_D}{w} + (1.695 - 0.13)$$

and

$$\frac{p_F}{w} = (1.73 - 0.13) \times 13.6$$

Thus by substitution, we get

$$= \frac{p_C}{w} + (1.72 + 1.695 - 0.065) - (1.695 - 0.065) \times 13.6$$

$$= (1.73 - 0.13) \times 13.6 - (1.695 - 0.13)$$

or

$$\frac{p_C}{w} + 3.35 - 22.168 = 21.76 - 1.565$$

$$\therefore \quad \frac{p_C}{w} = (43.928 - 4.915)$$

$$= 39.013 \text{ m of water}$$

i.e.,

$$p_C = 39.013 \times 10^3 \text{ kg(f)/m}^2$$

$$= 3.901\,3 \text{ kg(f)/cm}^2$$

Example 2.22. *An empty cylindrical bucket, 0.3 m in diameter and 0.5 m long whose wall thickness and weight can be considered as negligible is forced with its open end first into water until its lower edge is 4 m below the surface. What force will be required to maintain this position, assuming the trapped air to remain at constant temperature during the entire operation? Atmospheric pressure = 1.03 kg(f) / cm².*

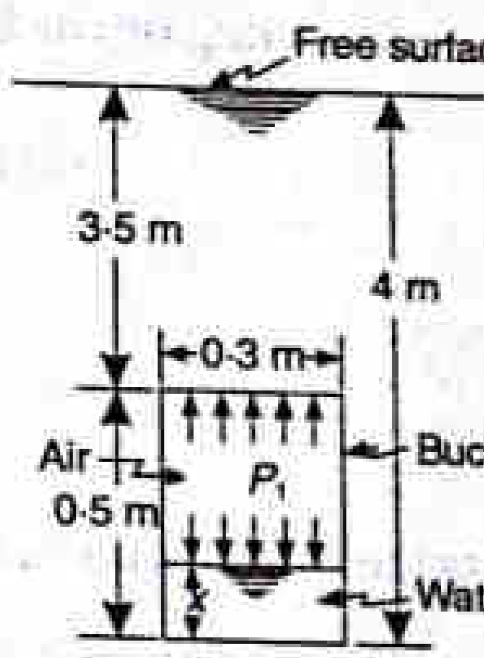


Figure Ex. 2.22

Solution

Let P_a be the atmospheric pressure and P_1 be the absolute pressure of the compressed air trapped in the bucket. If the depth of water raised in the bucket is x, then since the temperature of the air remains constant, according to isothermal condition.

$$p_a \times \frac{\pi}{4}(0.3)^2 \times 0.5 = p_1 \times \frac{\pi}{4} \times (0.3)^2\,(0.5 - x)$$

or

$$p_1 = \left(\frac{0.5}{0.5 - x}\right) p_a \qquad \text{...(i)}$$

Also

$$p_1 = p_a + (4 - x) \times 1000 \qquad \text{...(ii)}$$

From Eqs. (i) and (ii), we get

$$\left(\frac{0.5}{0.5 - x}\right) p_a = p_a + (4 - x) \times 1000$$

Since

$$p_a = 1.03 \times 10^4 \text{ kg(f)/m}^2\text{, we get}$$

$$\left(\frac{0.5}{0.5-x}\right) \times 1.03 \times 10^4 = 1.03 \times 10^4 + (4-x) \times 1000$$

or

$$\left(\frac{0.5}{0.5-x}\right) 1.03 \times 104 = (4-x) \times 1000$$

$$\therefore \quad x = 0.136\,4 \text{ m}$$

Substituting the value of x in (i), we get

$$p_1 = \left(\frac{0.5}{0.5-0.1364}\right) \times 1.03 \times 10^4$$

$$= 1.416 \times 10^4 \text{ kg(f)/m}^2$$

$$= 1.416\,4 \text{ kg(f)/cm}^2$$

The force tending to move the bucket in the upward direction

$$P_1 = p_1 \times \frac{\pi}{4}(0.3)^2$$

$$= 1.416\,4 \times 10^4 \times \frac{\pi}{4} \times (0.3)^2$$

$$= 1\,001.194 \text{ kg(f)}$$

The force acting on the bucket in the downward direction

$$P_2 = [1.03 \times 10^4 + 1000 \times 3.5] \frac{\pi}{4} \times (0.3)^2$$

$$= 13.8 \times 10^3 \times \frac{\pi}{4} \times (0.3)^2$$

$$= 975.464 \text{ kg(f)}$$

Thus the force required to maintain the bucket in this position

$$F = (P_1 - P_2)$$

$$= (1000.194 - 975.464)$$

$$= 24.73 \text{ kg(f)}$$

Example 2.23. *Petrol of specific gravity 0.8 flows upwards though a vertical pipe. A and B are two points in the pipe, B being 0.3 m higher than A. Connections are led from A and B to a U-tube containing mercury. If the difference of pressure between A and B is 0.18 kg(f)/cm², find the reading shown by the differential mercury manometer gage.*

Solution

Let p_A and p_B be the pressure intensities at the points A and B. If the reading of the gage is x, then as shown in Fig. Ex. 2.23.

$$\frac{p_A}{w} + (x+y) \times 0.8 = \frac{p_B}{w} + (0.3+y)\,0.8 + (x \times 13.6)$$

$$\left(\frac{p_A}{w} - \frac{p_B}{w}\right) = x\,(13.6 - 0.8) + (0.3 \times 0.8)$$

But

$$\left(\frac{p_A}{w} - \frac{p_B}{w}\right) = \frac{0.18 \times 10^4}{1000} = 1.8 \text{ m}$$

Hence

$$1.8 = 12.8\,x + 0.24$$

$$\therefore \quad x = \frac{1.56}{12.8} = 0.122 \text{ m or } 12.2 \text{ cm.}$$

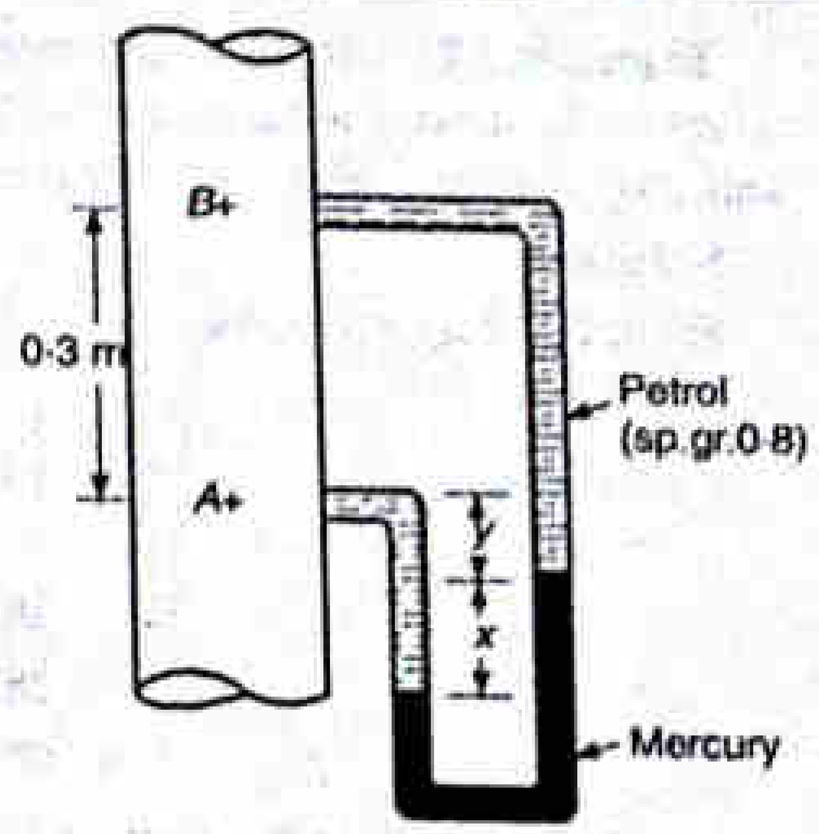


Figure Ex. 2.22

Example 2.24. *Assuming adiabatic conditions, find the temperature drop for an altitude of 2000 metres above the earth's surface. Assume the earth's surface temperature as 15°C, the gas constant R for air 289 m²/sec² per °C abs and n for standard polytropic atmosphere = 1.24. Also find the atmospheric pressure at that altitude, if it is given that it is 1.033 kg(f)/cm² at earth's surface.*

Solution

From Eq. 2.23 viz.,

$$\frac{T}{T_0} = \left[1 - \frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]$$

the temperature drop is given by

$$\frac{T_0 - T}{T_0} = \frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right); \; T_0 = (273+15) = 288° \text{ abs.}$$

By substituting the given values, we get

$$\frac{288-T}{288} = \frac{9.81 \times 2000}{289 \times 288} \times \frac{1.24-1}{1.24}$$

$$\therefore \quad T = 274.86° \text{ abs.}$$

$\therefore$ Temperature drop = (288 – 274.86) = 13.14°C.

Further from Eq. 2.22 the variation of atmospheric pressure with altitude is given as

$$\frac{p}{p_0} = \left[1 - \frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]^{\frac{n}{n-1}}$$

By substituting the given values, we get

$$\frac{p}{1.033} = \left[1 - \frac{9.81 \times 2000}{289 \times 288} \times \left(\frac{1.24-1}{1.24}\right)\right]^{\left(\frac{1.24}{1.24-1}\right)}$$

$$p = 0.812 \text{ kg(f)/cm}^2.$$

Example 2.25. *At the top of a mountain the temperature is –5°C and a mercury barometer reads 56.6 cm, whereas the reading at the foot of the mountain is 74.9 cm. Assuming dry adiabatic conditions with R = 287 joule/[kg (m) deg C abs], calculate the height of the mountain.*

Solution

From Eq. 2.23, we have

$$\frac{T}{T_0} = \left[1-\frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]$$

$$R = 287 \text{ joule }/[\text{kg(m) deg C abs}]$$
$$= 287 \text{ m}^2/(\text{sec}^2 \text{ deg C abs})$$
$$T = (273-5) = 268°\text{C abs.}$$

For dry abiabatic conditions, n =1.4

Thus by substitution, we get

$$\frac{268}{T_0} = \left[1-\frac{9.81(z-z_0)}{287\times T_0}\times\left(\frac{1.4-1}{1.4}\right)\right]$$

$$\therefore \quad T_0 = \left[268+\frac{9.81\times(z-z_0)\times 0.4}{287\times 1.4}\right]$$

Further from Eq. 2.22

$$\frac{p}{p_0} = \left[1-\frac{g(z-z_0)}{RT_0}\left(\frac{n-1}{n}\right)\right]^{\frac{n}{n-1}}$$

By substitution, we get

$$\frac{56.6}{74.9} = \left[1-\frac{9.81\times(z-z_0)\times 0.4\times 287\times 1.4}{(287\times 1.4)[(268\times 287\times 1.4)+9.81\times(z-z_0)\times 0.4]}\right]^{\frac{1.4}{0.4}}$$

or
$$\left(\frac{56.6}{74.9}\right)^{\frac{0.4}{1.4}} = \left[1-\frac{9.81\times(z-z_0)\times 0.4}{(268\times 287\times 1.4)+9.81\times(z-z_0)\times 0.4}\right]$$

or
$$0.923 = \frac{268\times 287\times 1.4}{(268\times 287\times 1.4)+9.81\times(z-z_0)\times 0.4}$$

$$\therefore \quad (z-z_0) = 2289 \text{ m.}$$

Example 2.26. *Figure Ex. 2.26 shows a device used to indicate the level inside a fuel tank. It is in the form of a U-tube with the crossing over. The U-tube is partially filled with a manometric fluid of specific weight w_2 which is heavier than the fuel whose specific weight is w_1. When the U-tube and indicating tube are vertical and the tank is full of fuel the reading on the scale is unity.*

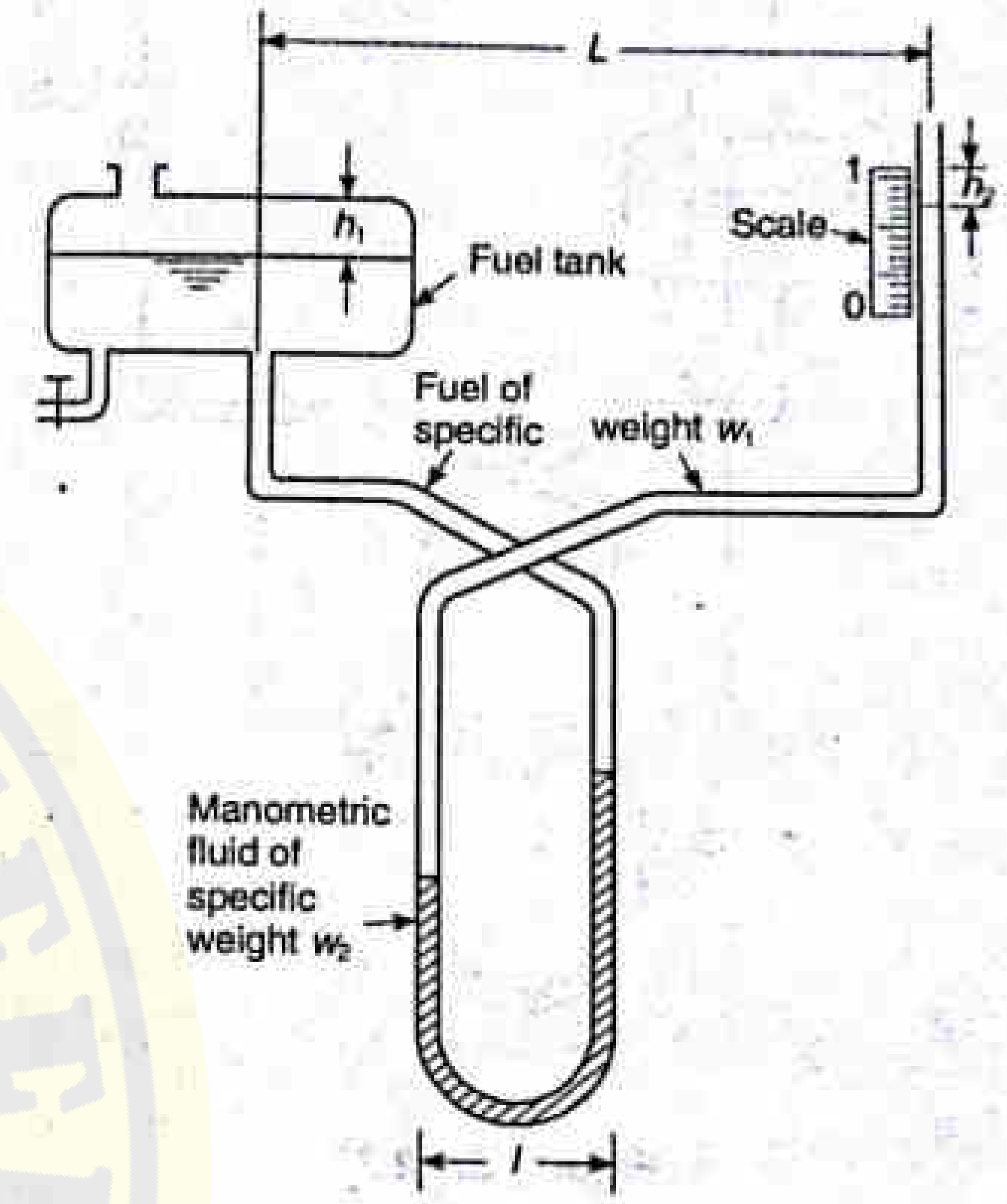


Figure Ex. 2.26

(a) *Determine the relation between h_1, the distance through which the fuel level is lower in the tank and h_2, the distance through which the fuel level falls in the indicating tube.*

(b) *For what value of the ratio $\frac{l}{L}$ will the reading of the device remains unaltered when the system is given small tilt in the vertical plane?*

Solution

(a) When the fuel tank is full, let the manometric liquid in the limb of the U-tube connected to the fuel tank be at B and that in the other limb of the U-tube be at A (see Fig. 2.26).

Equating the pressures at point A and A', we get

$$w_1X = w_1Y + w_2Z \quad \text{...(i)}$$

When the level of fuel in the tank is reduced by h_1 and that in the indicating tube is reduced by h_2, the level of manometric liquid in the limb of the U-tube connected to the indicating tube will be raised from A to C by h_2, and that in the limb of the U-tube connected to the fuel tank will be reduced from B to D by h_2.

Thus equating the pressures at points C and C', we get

$$w_1(X-2h_2) = w_1(Y-h_1+h_2)+w_2(Z-2h_2) \quad \text{...(ii)}$$

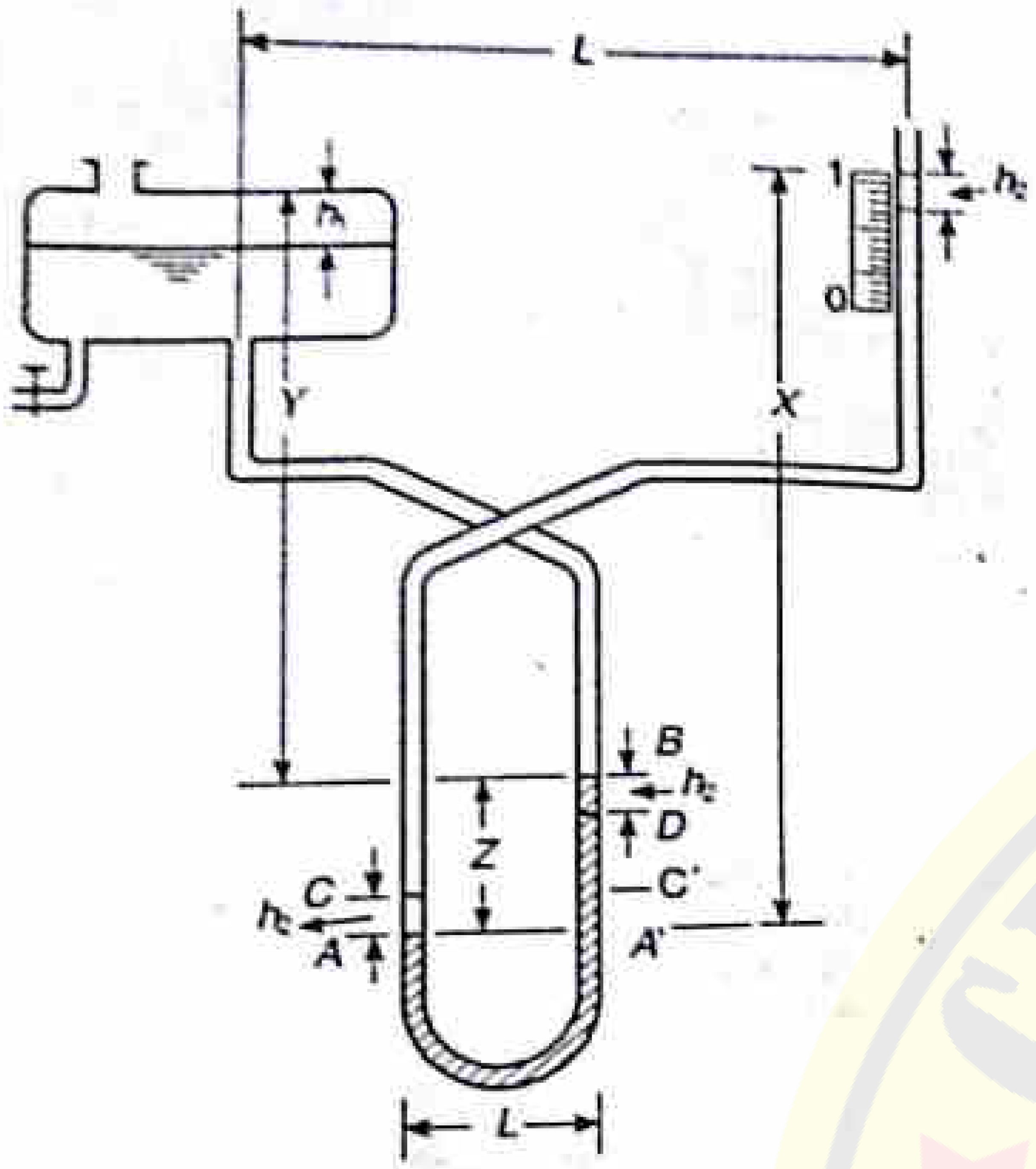


Figure Ex. 2.26

Subtracting Eq. (ii) from (i), we get

$$2w_1h_2 = w_1h_1 - w_1h_2 + 2w_2h_2$$

or

$$\frac{h_1}{h_2} = 1 + \left(\frac{w_2}{w_1} - 1\right)$$

which is the required relation between h_1 and h_2.

(b) For the readings of the device to remain unaltered when the system is given a small tilt θ in the vertical plane, sin θ or tan θ may be expressed as

$$\frac{Z}{l} = \frac{(X-Y)}{L}$$

or

$$\frac{l}{L} = \frac{Z}{X-Y}$$

From Eq. (i), we have

$$\frac{Z}{X-Y} = \frac{w_1}{w_2}$$

∴

$$\frac{l}{L} = \frac{w_1}{w_2}$$

SUMMARY OF MAIN POINTS

1. Pressure or intensity of pressure is defined as the force per unit area.
2. The pressure p at any point in a static mass of fluid (or fluid at rest) varies only in z or vertical direction as indicated by the following equation

$$\frac{dp}{dz} = -w = -\rho g \qquad ...(i)$$

which represents the basic law of hydrostatics.

(i) A liquid may be considered as incompressible fluid for which w or ρ is constant and hence integration of the above equation gives pressure p at any point in a static mass of liquid (or liquid at rest) as

$$p = -wz + C$$

where C is the constant of integration.

(ii) For any point lying in a static mass of liquid (or liquid at rest) at a vertical depth h below the free surface of the liquid, $z = -h$, and at the free surface of the liquid i.e., at $z = 0$, the pressure is equal to the atmospheric pressure p_a, the above equation gives

$$p = wh + p_a$$

and the pressure in excess of the atmospheric pressure is

$$p = wh$$

(iii) The vertical height of the free surface above any point in a liquid at rest is known as pressure head which may be expressed from the above equation as

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

3. For a compressible fluid (gas) since the density ρ or w varies with the pressure p, Eq. (i) can be integrated to determine the pressure p at any ponit only if a relation between p and ρ or w is known. The following two relations between p and ρ or w are considered.

(i) If a static mass of compressible fluid is assumed to undergo isothermal compressible, then $\frac{p}{\rho} = \frac{p_0}{\rho_0}$ = constant.

By introduing this equation in Eq. (i), it is integrated and the resulting equation is combined with the equation of state to obtain the pressure p at a height z in a static mass of compressible fluid undergoing isothermal compression as

$$p = p_0 \exp\left(-\frac{gz}{RT_0}\right)$$

where

p_0 = absolute pressure of the fluid at initial condition at some reference level ;

z = height of the point above the reference level ;

R = gas constant ; and

T_0 = absolute temperature of the fluid at initial condition, which, however, remains constant during the isothermal compression.

(ii) If a static mass of compressible fluid is assumed to undergo adiabatic compression, then $\frac{p}{\rho^k} = \frac{p_0}{\rho_0^k}$ = constant. By introducing this equation in Eq. (i), it is integrated and the resulting equation is combined with the equation of state to obtain the pressure p at a height z in a static mass of compressible fluid undergoing adiabatic compression as

$$p = p_0\left[1 - \frac{gz}{RT_0}\left(\frac{k-1}{k}\right)\right]^{\frac{k}{k-1}}$$

where k is the adiabtic exponent or adiabatic index, and the other notations are same as indicated earlier.

Since in this case temperature is not constant it is given by

$$T = T_0\left[1 - \frac{gz}{RT_0}\left(\frac{k-1}{k}\right)\right]$$

The rate at which the temperature changes with elevation or height z is known as Temperature Lapse rate λ, which is given by

$$\lambda = \frac{\partial T}{\partial z} = -\frac{g}{R}\left(\frac{k-1}{k}\right)$$

(a) For $k = 1$, temperature lapse rate $\lambda = 0$. This condition pertains to isothermal compression during which the temperature remains constant and hence it does not vary with elevation or height z.

(b) For $k > 1$, temperature lapse rate λ is negative, which means that temperature decreases with the increase of height z.

4. The Pascal's law states that the pressure at any point in a fluid at rest has the same magnitude in all directions.

5. Fluid pressure may be measured with respect to (i) absolute zero pressure, and (ii) local atmospheric pressure.

   When pressure is measured above absolute zero or complete vacuum it is called absolute pressure, and when pressure is measured either above or below atmospheric pressure it is called gage pressure. If the pressure of a fluid is below atmospheric pressure it is designated as vacuum pressure or suction pressure or negative gage pressure.

   Thus the relation between absolute pressure, atmospheric pressure and gage pressure may be expressed as

$$p_{abs} = p_{atm} + p_{gage}$$
$$p_{abs} = p_{atm} - p_{gage}$$

6. Manometer is a device used for measuring pressure at a point in a fluid.

   Manometers are classified as (a) Simple manometers, and (b) Differential manometers.

   (a) Simple manometers are used for measuring pressure at a point in a fluid contained in a pipe or vessel. (b) Differential manometers are used for measuring the difference of pressure between any two points in a fluid contained in a pipe or a vessel or in two different pipes or two different vessels.

7. Micromanometers are used for the measurement of very small pressure differences or for the measurment of pressure differences with very high precision.

8. Mechanical gages are those pressure measuring devices which embody an elastic element which deflect under the action of the applied pressure, and this movement mechanically magnified, operates a pointer moving against a graduated circumferential scale. These gages are generally used for measuring high pressures and where high precision is not required. Bourdon Tube Pressure gage is the most commonly used mechanical gage which may be used for measuring both pressures above atmospheric pressure as well as pressures below atmospheric pressure or vacuum pressures.

PROBLEMS

2.1 Explain the terms—intensity of pressure and pressure head.

2.2 Prove that the pressure is the same in all directions at a point in a static fluid.

2.3 State Pascal's Law and give some examples where this principle is applied.

2.4 Define the terms gage pressure, vacuum pressure and absolute pressure. Indicate their relative positions on a chart.

2.5 Briefly explain the principle employed in the manometers used for the measurement of pressure.

2.6 Differentiate between simple and differential type of manometers.

2.7 Describe with the help of neat sketches different types of manometers.

2.8 State the advantages of mechanical pressure gages over the manometers.

2.9 Explain how vacuum pressure can be measured with the help of a U-tube manometer.

2.10 Describe with a neat sketch a micromanometer used for very precise measurement of small pressure difference between two points.

2.11 Express a pressure intensity of 5 kg (f)/cm² in all possible units. Take the barometer reading as 76 cm of mercury.

[*Ans.* Gage units : 5 kg(f)/cm² ; 5 × 10⁴ kg(f)/m²; 50 m of water; 49.05 × 10⁴ N/m²; 3.68 m of mercury;

Absolute units: 6.034 kg(f)/cm²; 6.034 × 10⁴ kg(f)/m²; 59.19 × 10⁴ N/m² ; 60.34 m of water ; 4.44 m of mercury ; 5.842 atmospheres]

2.12 Find the depth of a point below free surface in a tank containing oil where the pressure intensity is 9 kg (f)/cm². Specific gravity of oil is 0.9. [*Ans.* 100 m]

2.13 Convert a pressure head of 15 m of water to (a) metres of oil of specific gravity 0.750 ; (b) metres of mercury of specific gravity 13.6.

[*Ans.* (a) 20 m; (b) 1.103 m]

2.14 A U-tube containing mercury has its right limb open to atmosphere. The left limb is full of water and is connected to a pipe containing water under pressure, the centre of which is in level with the free surface of mercury. Find the pressure of the water in the pipe above atmosphere, if the difference of level of mercury in the limbs is 5.08 cm. [*Ans.* 0.064 kg(f)/cm²]

2.15 The pressure between two points M and N in a pipe conveying oil of specific gravity 0.9 is measured by an inverted U-tube and the column connected to point N stands 1.5 m higher than that at point M. A commercial pressure gage attached directly to the pipe at M reads 2 kg(f)/cm²; determine its reading when attached directly to the pipe at N.

[*Ans.* 2.135 kg(f)/cm²]

2.16 A pipe containing water at 172 kN/m² pressure is connected by a differential gage to another pipe 1.5 m lower than the first pipe and containing water at high pressure. If the difference in heights of the two mercury columns of the gage is equal to 75 mm, what is the pressure in the lower pipe? Specific gravity of mercury is 13.6. [*Ans.* 196 kN/m²]

2.17 Two pressure points in a water pipe are connected to a manometer which has the form of an inverted U-tube. The space above the water in the two limbs of the manometers is filled with toluene of specific gravity 0.875. If the difference of level of water columns in the two limbs is equal to 0.12 m what is the corresponding difference of pressure expressed in (a) kg(f)/cm²; (b) N/m².

[*Ans.* (a) 0.0015 kg(f)/cm²; (b) 147.15 N/m²]

2.18 A U-tube differential gage is attached to two sections A and B in a horizontal pipe in which oil of specific gravity 0.8 is flowing. The deflection of the mercury in the gage is 60 cm, the level nearer to A being the lower one. Calculate the difference of pressure in kg(f)/cm² between the sections A and B.

[*Ans.* 0.768 kg(f)/cm²]

2.19 In Fig. P 2.19 the areas of the plunger A and cylinder B are 40 and 4 000 cm² respectively. The weight of cylinder B is 4 100 kg(f). The vessel and the connecting passages are filled with oil of specific gravity 0.750. What force F is required for equilibrium, if the weight of A is neglected ? [*Ans.* 40.85 kg(f)]

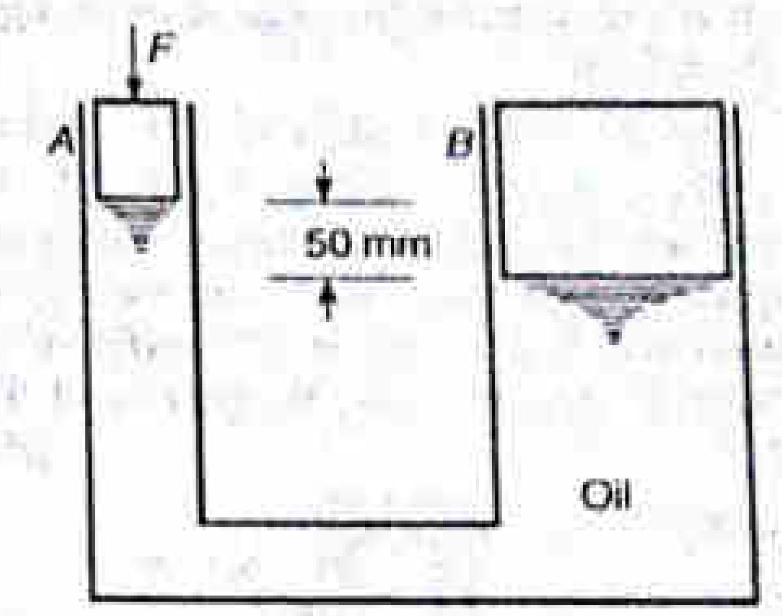


Fig. P. 2.19

2.20 Vessels A and B contain water under pressures of 274.68 kN/m² [2.8 kg(f)/cm²] and 137.34 kN/m² [1.4 kg(f)/cm²] respectively. What is the deflection of the mercury in the diffrential gage shown in Fig. P. 2.20 ? [*Ans.* 1.27 m]

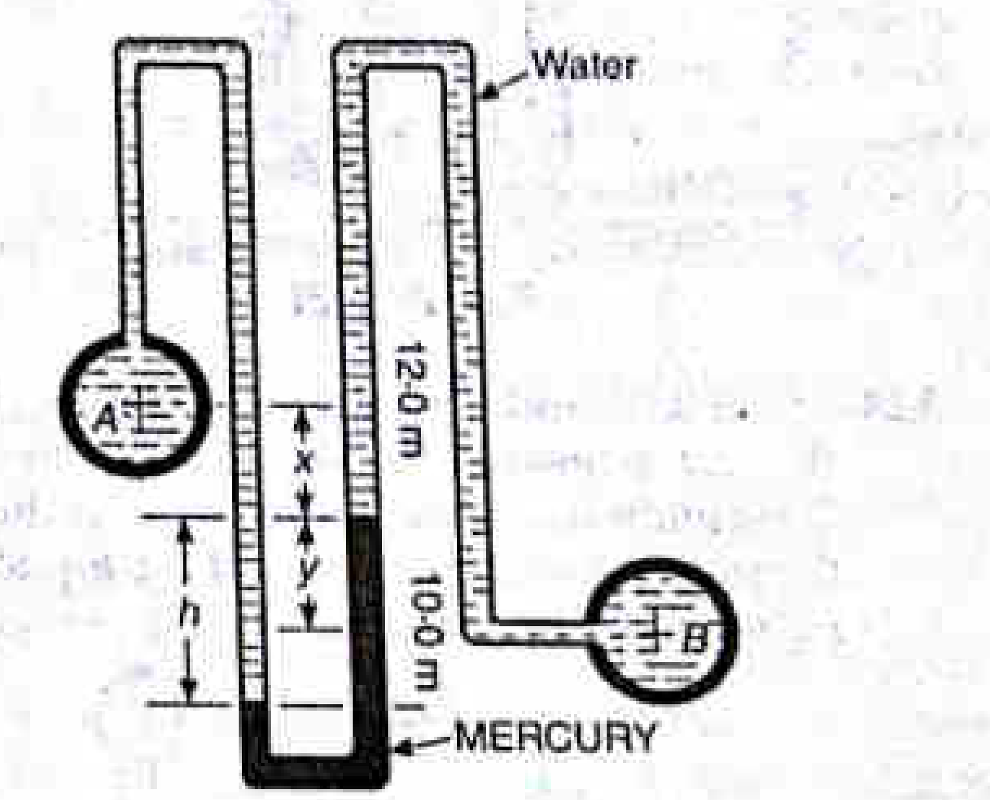


Fig. P. 2.20

2.21 The tank in the accompanying figure contains oil of specific gravity 0.750. Determine the

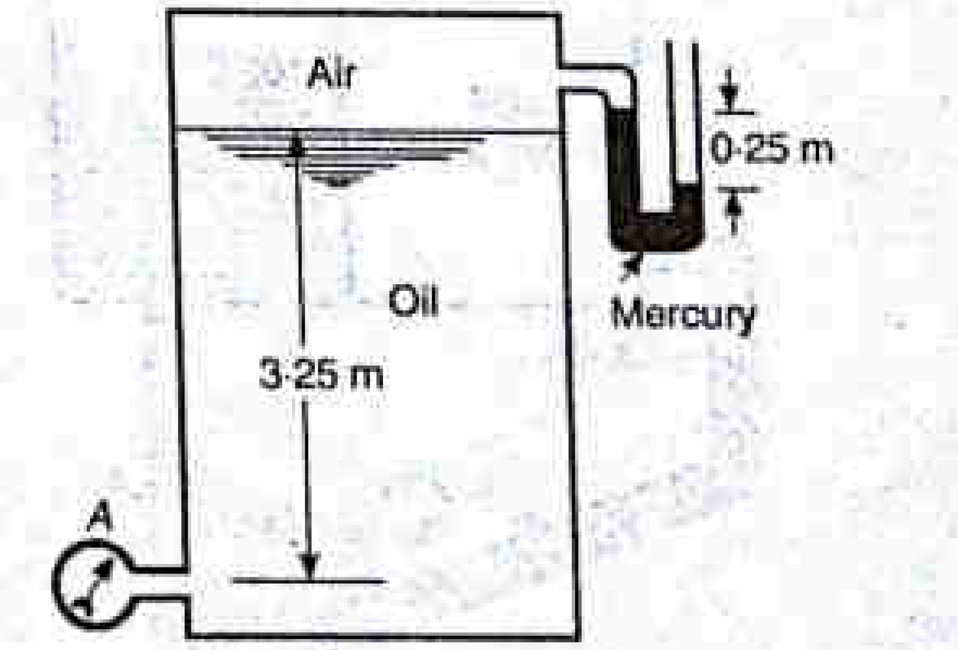


Fig. P. 2.21

reading of pressure gage A in (a) kg(f)/cm^2 ; (b) kN/m^2.

[*Ans.* (a) – 0.096 kg(f)/cm^2; (b) – 9.442 kN/m^2]

2.22 In the manometer shown in Fig. P. 2.22 the liquid on the left side is carbon-tetrachloride of specific gravity 1.60 and liquid on the right side is mercury. If $(p_A - p_B)$ is 525 kg(f)/m^2 (5 150.25 N/m^2), find the specific gravity of the liquid X. [*Ans.* 0.75]

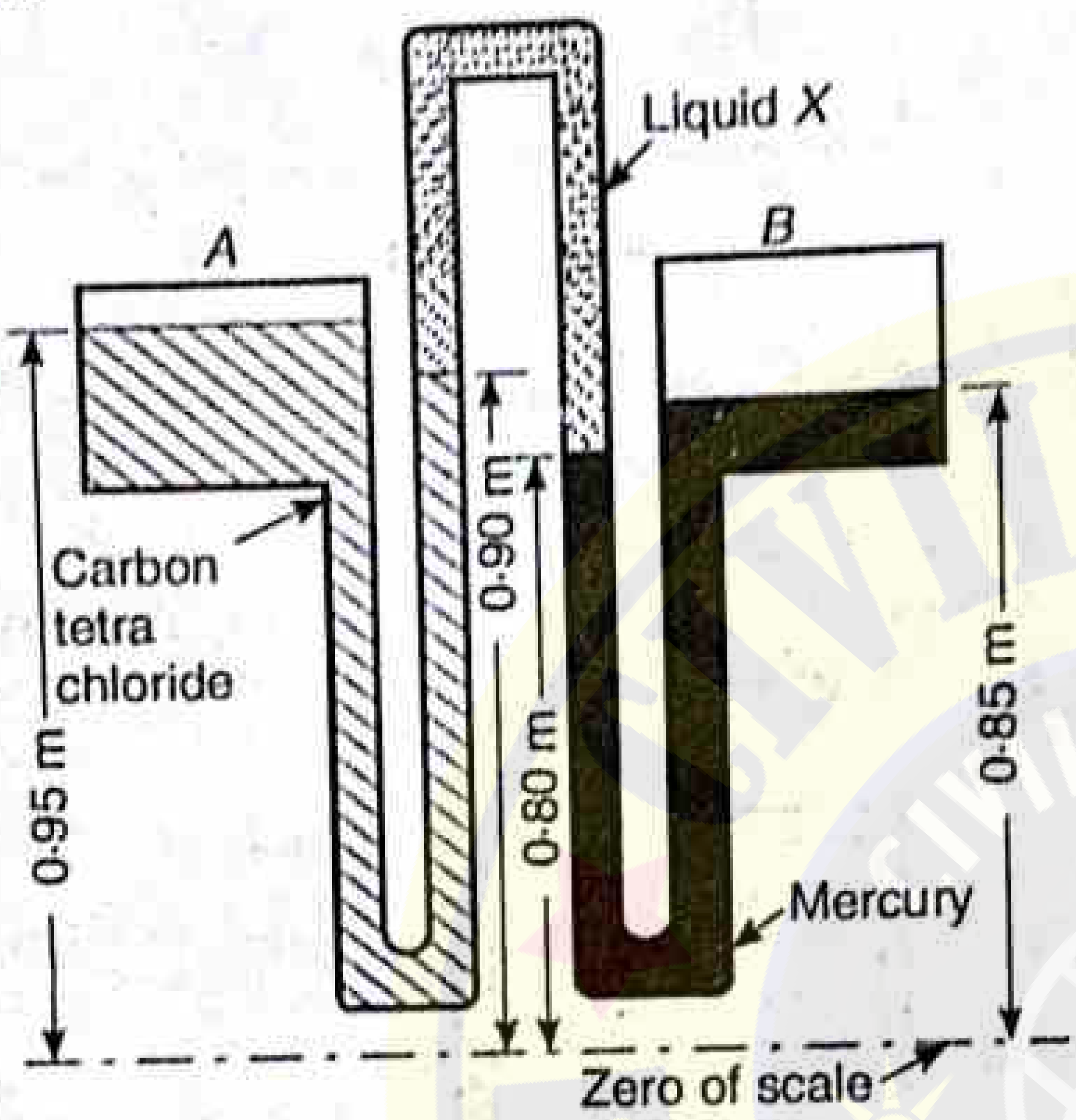


Fig. P. 2.22

2.23 In the left hand tank shown in the Fig. P. 2.23, the air pressure is –0.23 m of mercury. Determine the elevation of the gage liquid in the right hand column at A, if the liquid in the right hand tank is water. [*Ans.* 94.62 m]

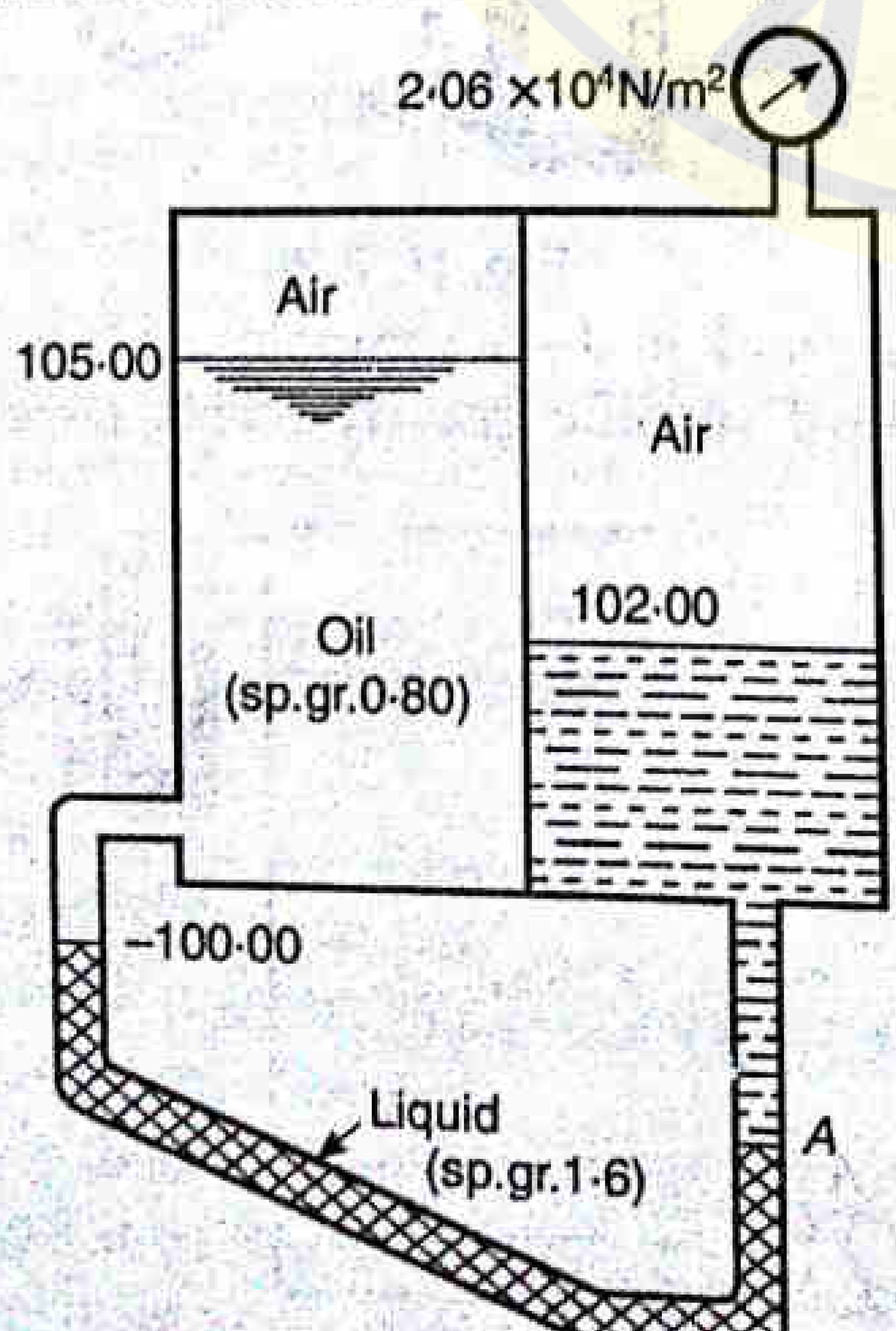


Fig. P. 2.23

2.24 As shown in Fig. P. 2.24, the pipe and connection B are full of oil of specific gravity 0.9 under pressure. If the U-tube contains mercury, find the elevation of point A in metres. [*Ans.* 112.76 m]

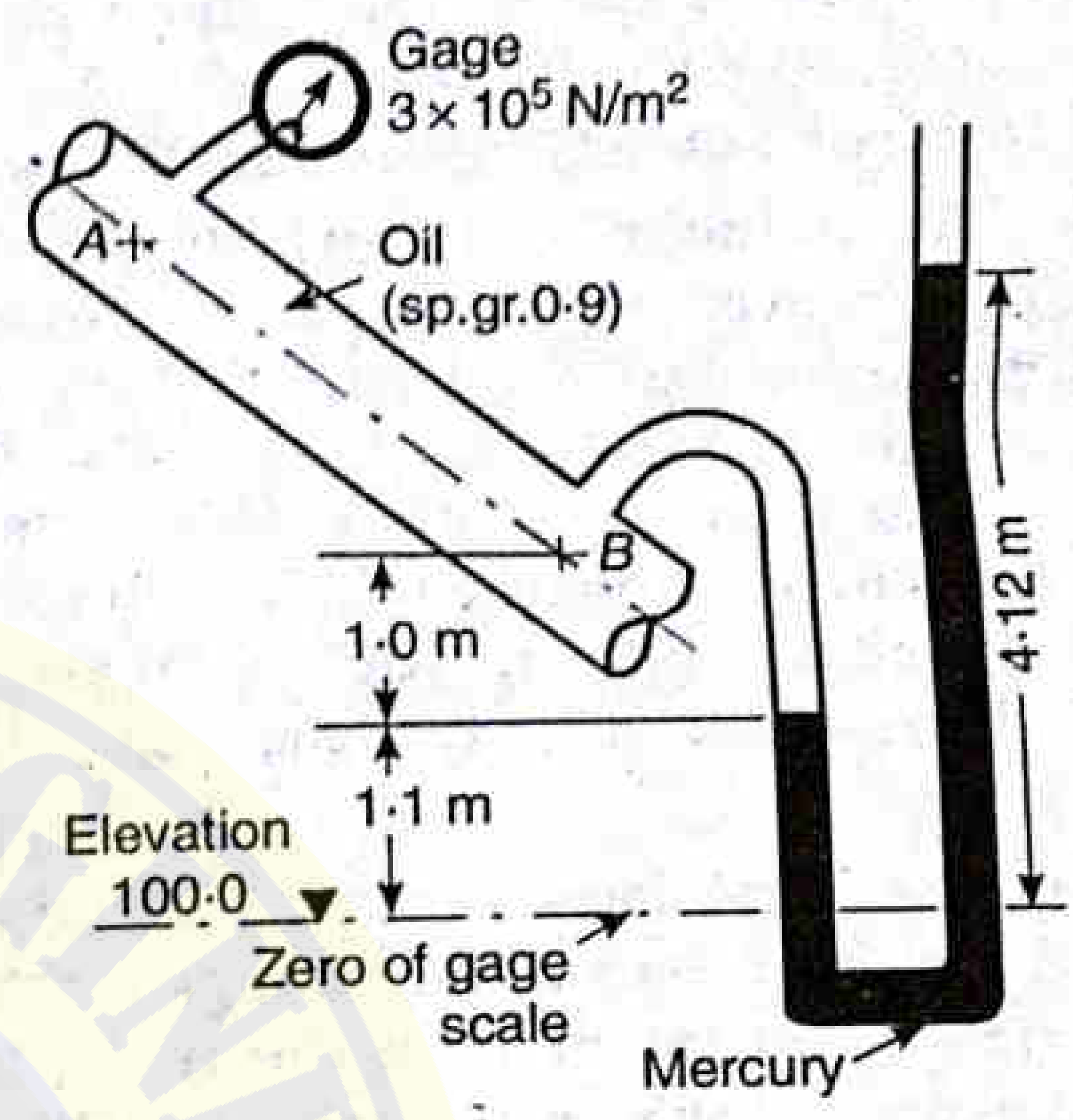


Fig. P. 2.24

2.25 Neglecting the friction between the piston A and the gas tank, find the gage reading at B in metres of water. Assuming gas and air to be of constant specific weight and equal to 5.501 N/m^2 and 12.267 N/m^3 respectively. [*Ans.* 0.101 m]

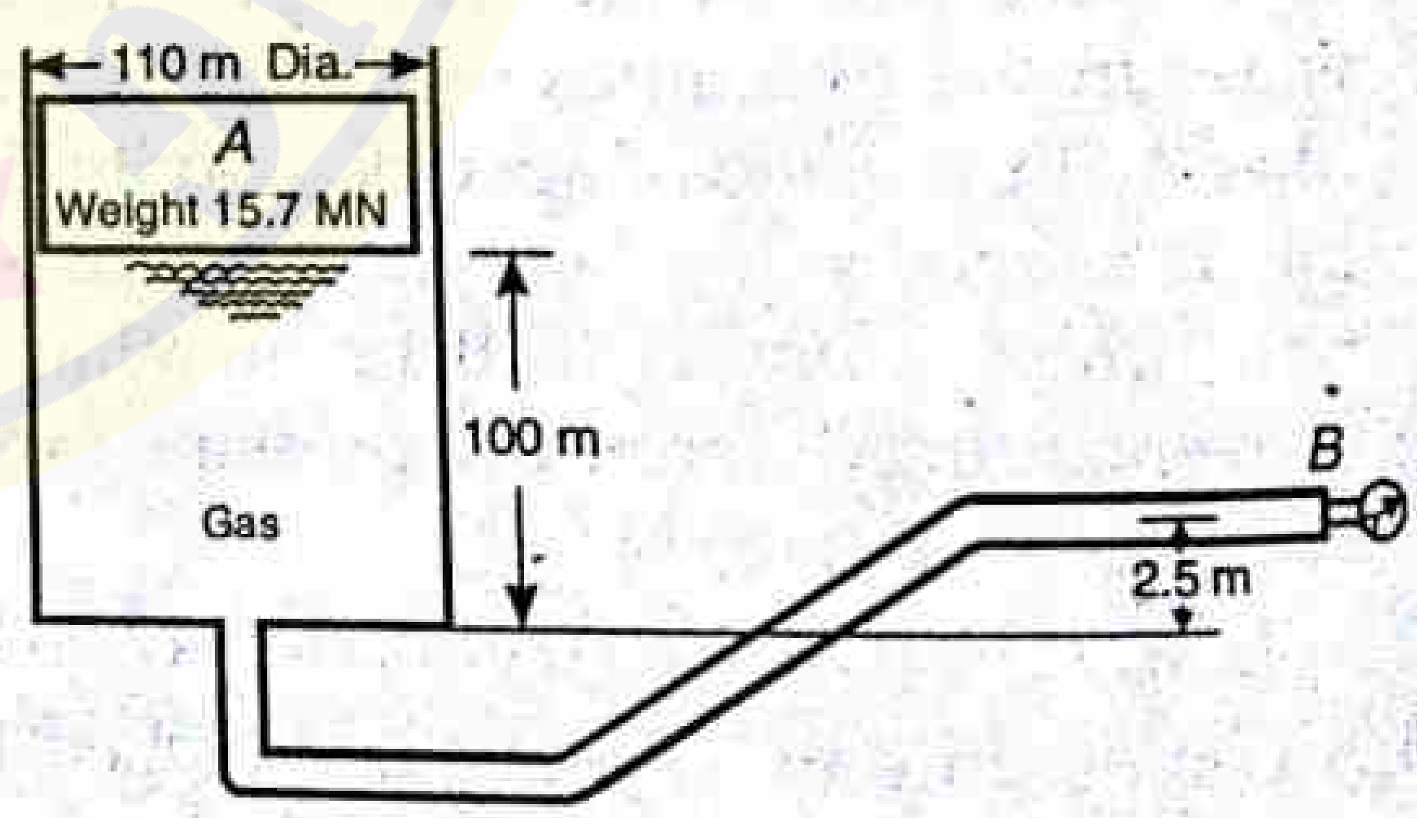


Fig. P. 2.25

2.26 A micro-manometer consists of two cylindrical bulbs A and B each 1000 sq.mm cross-sectional area which are connected by a U-tube with vertical limbs each of 25 sq.mm cross-sectional area. A liquid of specific gravity 1.2 is filled in A and another liquid of specific gravity 0.9 is filled in B, the surface of separation being in the limb attached to B. Find the displacement of the surface of separation when the pressure on the surface in B is greater than that in A by an amount equal to 15 mm head of water. [*Ans.* 42.6 mm]

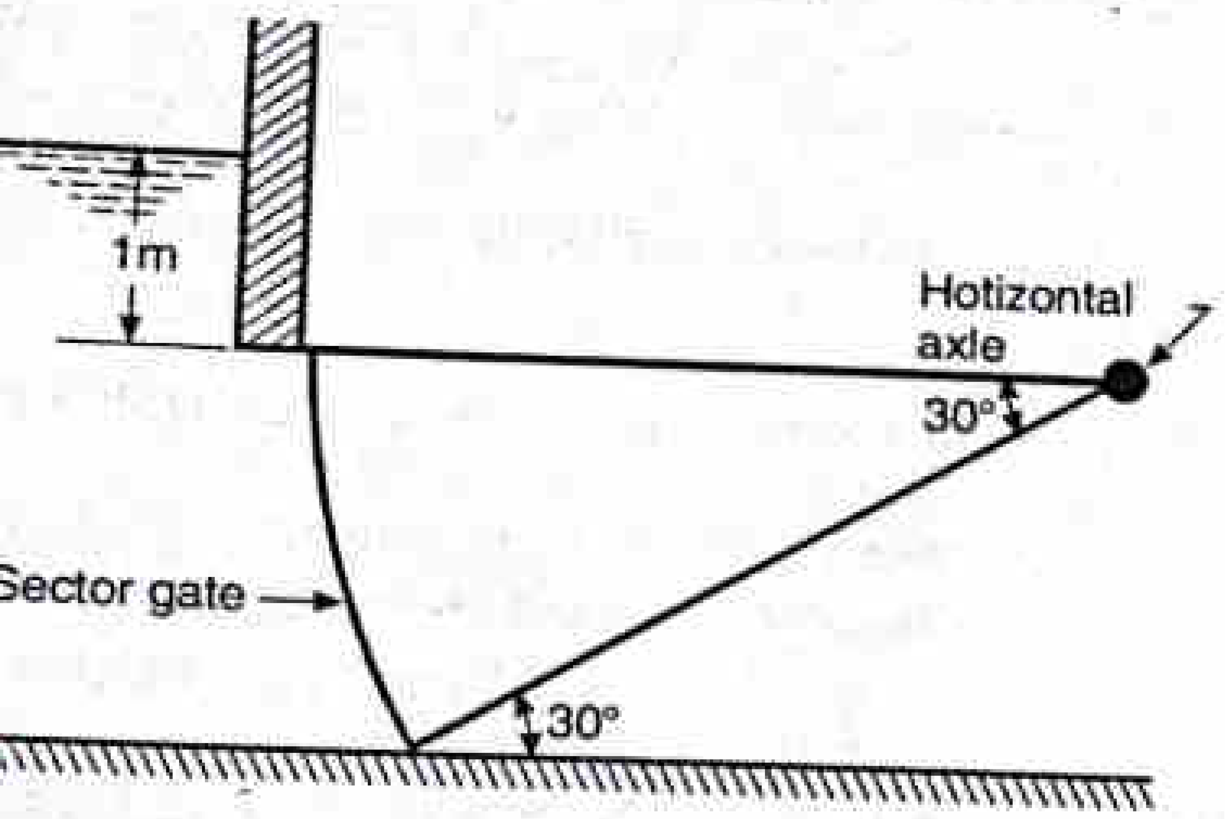


Figure P. 3.18

3.19 For a 2.5 m length of gate AB shown in Fig. P. 3.19 below, find the compression in strut CD due to water pressure if points B, C and D are hinged. [*Ans.* 6.92 × 10^3 kg(f) (67.885 kN)]

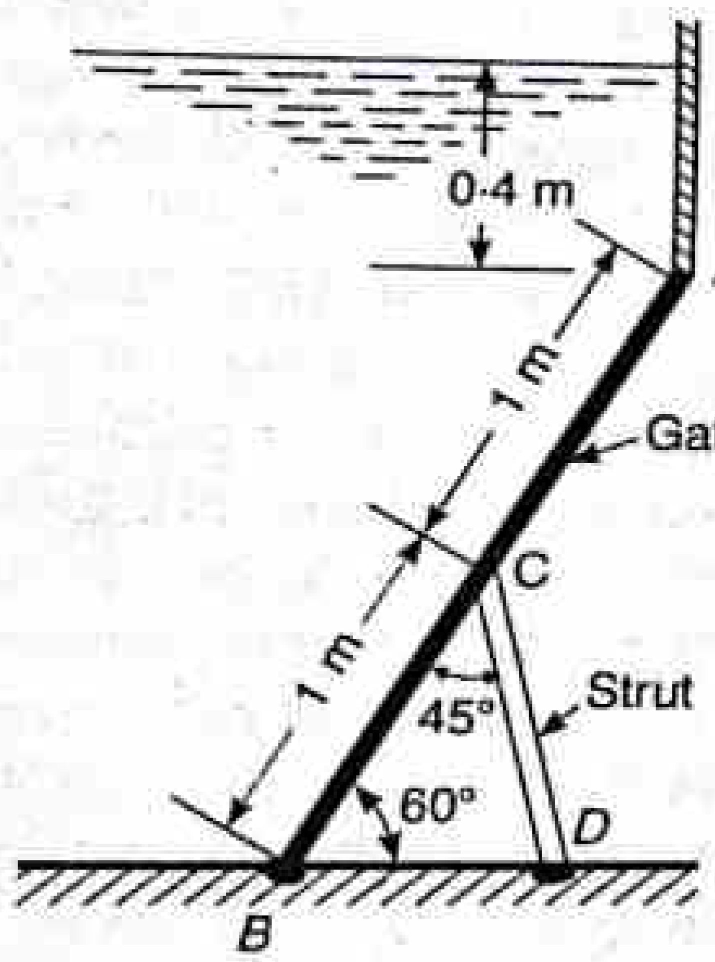


Figure P. 3.19

3.20 Find the dimension Z so that the total compression in rod BD shown in Fig. P. 3.20 will not be more than 88.29 kN [9×10^3 kg(f)]. The length of the board perpendicular to the paper may be taken as 1.25 m. [*Ans.* 1.94 m]

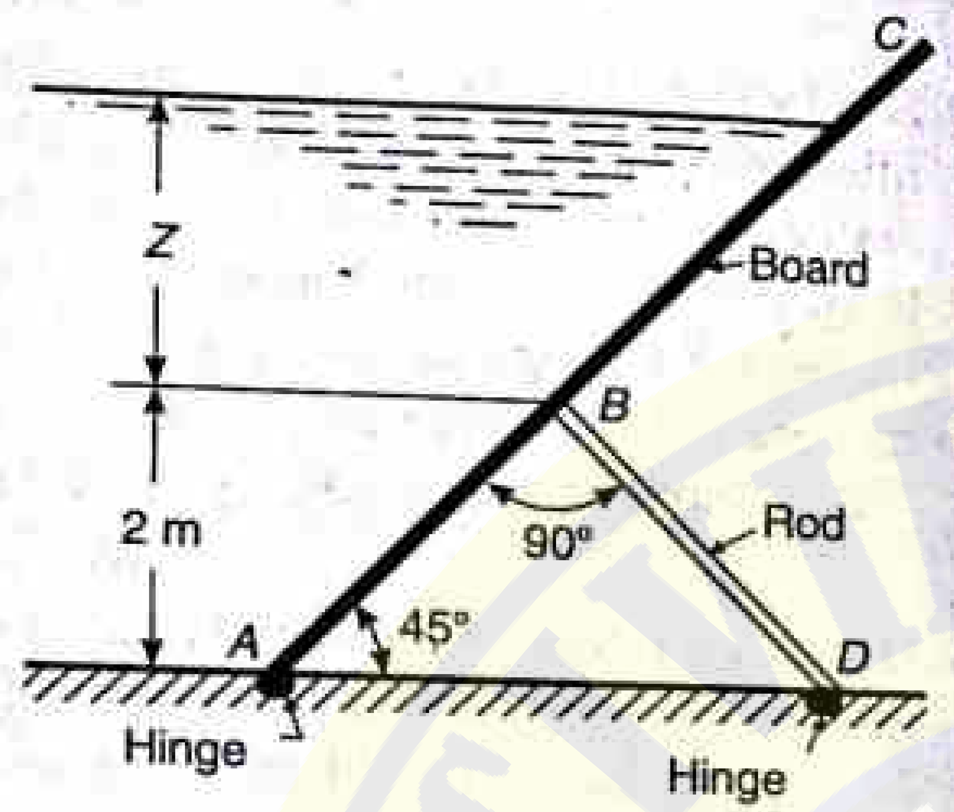


Figure P. 3.20

3.21 Calculate the depth of centre of pressure below the free surface of water of a plane lamina having a shape of a circular ring with outer and inner diameters $2a$ and a respectively. The lamina is immersed vertically with its top edge below the free surface at depth a.

[*Ans.* 2.156 a]

Buoyancy and Floatation

Chapter **4**

4.1 BUOYANCY, BUOYANT FORCE AND CENTRE OF BUOYANCY

When a body is immersed in a fluid either wholly or partially it is subjected to an upward force which tends to lift (or buoy) it up. This tendency for an immersed body to be lifted up in the fluid, due to an upward force opposite to the action of gravity, is known as *buoyancy*. The force tending to lift up the body under such conditions is known as *buoyant force* (*or force of buoyancy or upthrust*). The point of application of the force of buoyancy on the body is known as *centre of buoyancy*.

The magnitude of the buoyant force can be determined by the well-known *Archimedes' principle*, which states that *when a body is immersed in a fluid either wholly or partially, it is buoyed or lifted up by a force which is equal to the weight of the fluid displaced by the body*. It is due to this upward force acting on a body immersed in a fluid, either wholly or partially, that there occurs an apparent loss in the weight

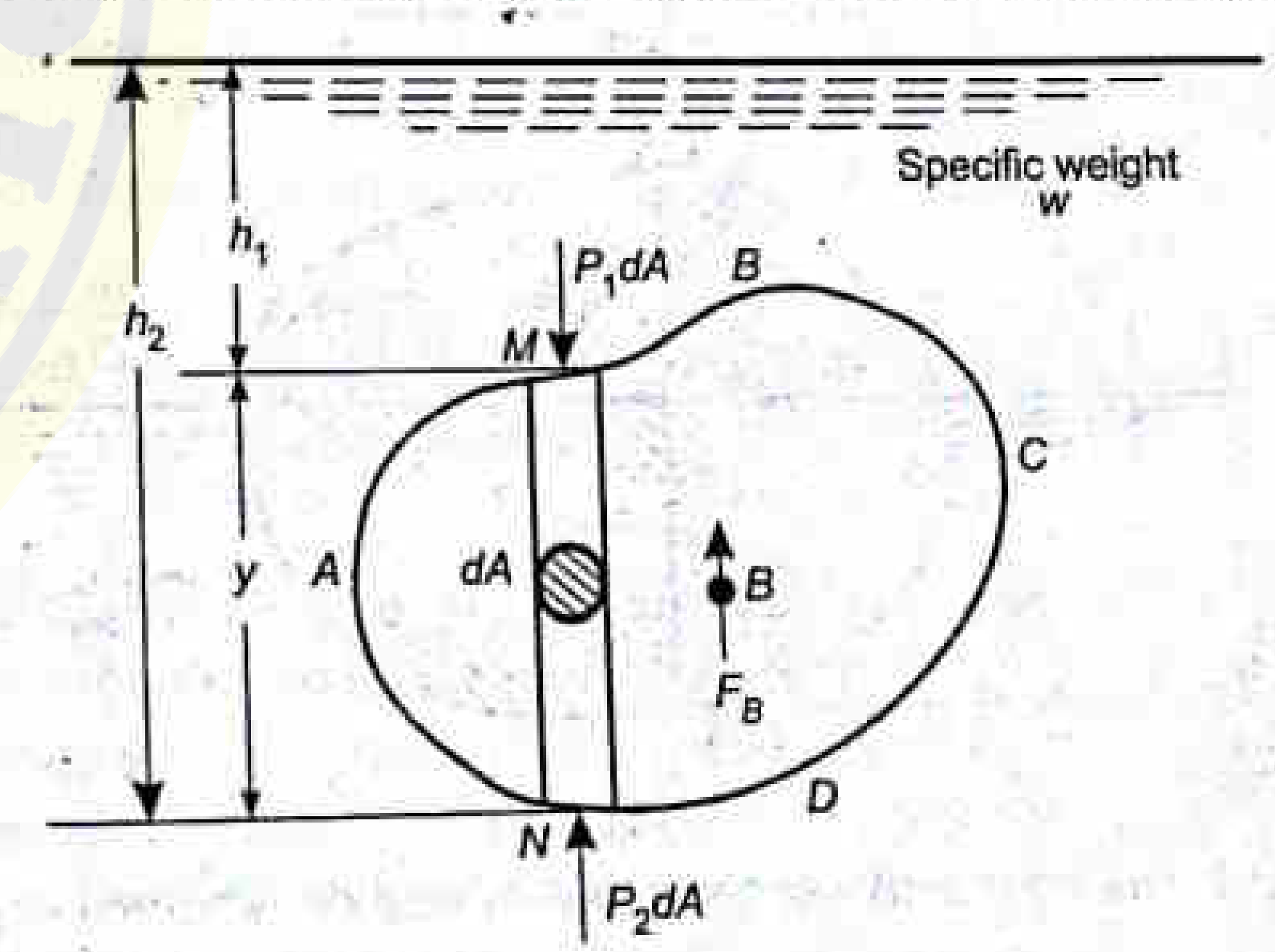


Figure 4.1 *Buoyant force on a wholly submerged body*

of the body. According to Archimedes' principle it is therefore known that the buoyant force is equal to the weight of the fluid displaced by the body. The buoyant force is exerted on a body immersed in a fluid, either wholly or partially, on account of the various portions of the boundary surface of the body being exposed to varying pressure intensities, which may be demonstrated by considering the pressure exerted by the surrounding fluid on the boundary surface of a submerged body as discussed below.

Consider a wholly submerged body $ABCD$ as shown in Fig. 4.1. The resultant horizontal pressure on the body is equal to zero because the horizontal pressure on the vertical projection from both sides are equal in the magnitude and opposite in direction. In order to obtain the resultant force acting on the body in the vertical direction, consider an elementary vertical prism MN of cross-section dA out of the whole body. On the top end of the prism MN a pressure force $(p_1\,dA)$ is exerted which is acting in the vertical downward direction. Similarly on its bottom end a pressure force $(p_2 dA)$ is exerted which is acting in the vertical upward direction. As shown in Fig. 4.1, $p_1 = wh_1$, $p_2 = wh_2$ and $(h_2 - h_1) = y$, where w is the specific weight of the fluid. Further since $p_2 > p_1$, the difference between the upward and downward pressure forces is a net upward force which is equal to the buoyant force dF_B on the vertical prism MN. Thus

$$dF_B = (p_2\,dA - p_1\,dA) = w\,(h_2 - h_1)\,dA = wydA$$

If $d\,d\bar{V}$ represents the volume of the vertical prism MN then

$$d\bar{V} = (ydA)$$

and

$$dFB = wd\bar{V}$$

The buoyant force FB on the entire submerged body $ABCD$ is obtained by integrating dF_B. Therefore

$$F_B = +\int dF_B = +\int wd\bar{V} = (w\bar{V}) \qquad ...(4.1)$$

where $\bar{V}$ is the volume of the submerged body. Equation 4.1 thus indicates that the buoyant force exerted on a submerged body is equal to the weight of the fluid displaced by the submerged body. The buoyant force acts vertically upwards through the centre of buoyancy which evidently coincides with the centroid of the volume of the fluid displaced. Moreover for a wholly submerged body of homogeneous composition the centre of buoyancy will coincide with the centre of gravity of the body.

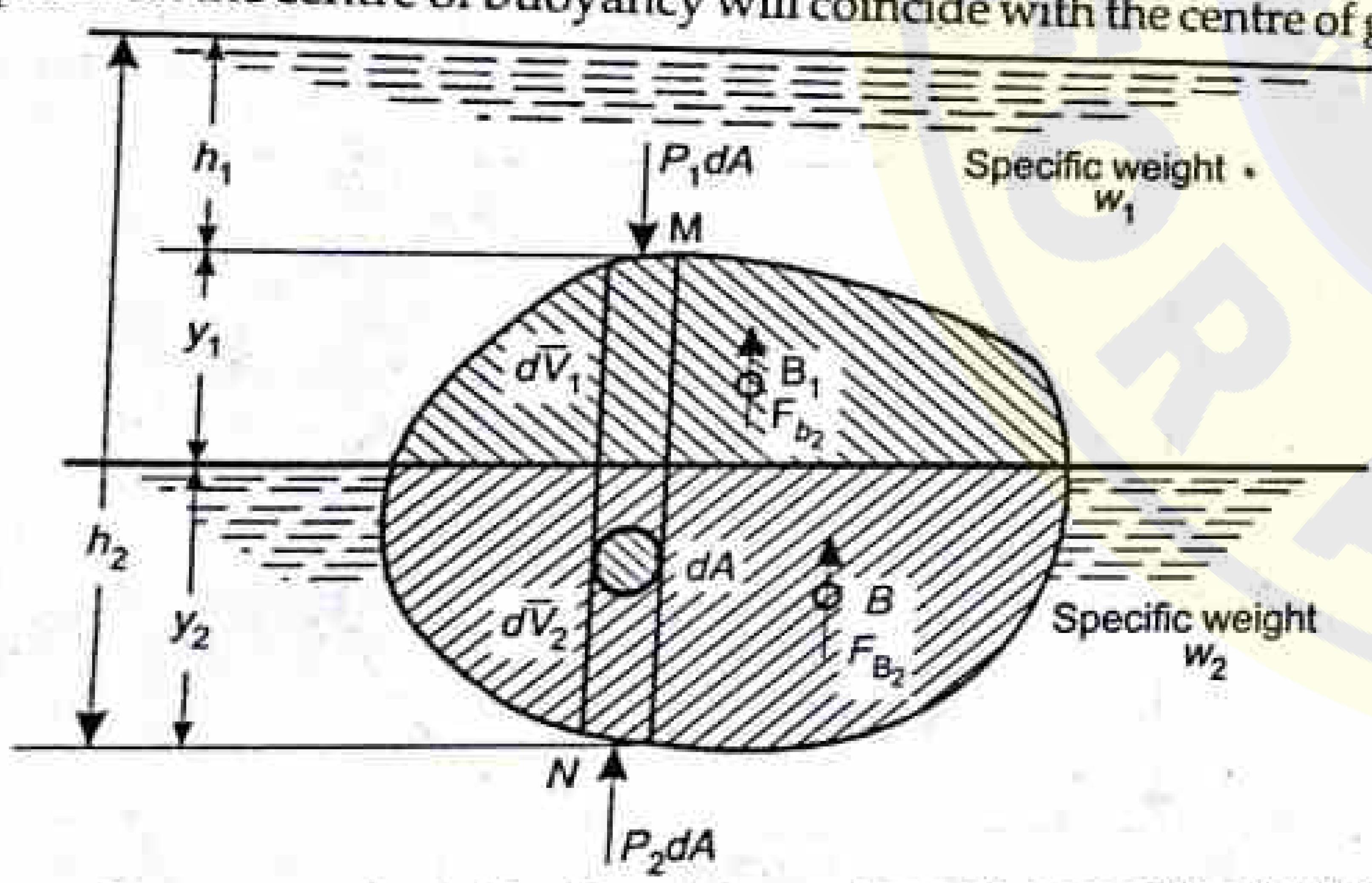


Figure 4.2 *Buoyant forces on a body floating at the surface of separation between two fluids*

When a body floats at a surface of separation (or interface) between two immiscible fluids of specific weights w_1 and w_2 as shown in Fig. 4.2, the buoyant force dF_B on a vertical prism MN of cross-section 4 is

$$dF_B = (p_2dA - p_1dA)$$
$$= [w_1\,(h_1 + y_1) + w_2 y_2 - (w_1\,h)]dA$$

or

$$dF_B = (w_1 y_1 + w_2 y_2)dA$$

or

$$dF_B = (w_1 d\bar{V}_1 + w_2 d\bar{V}_2)$$

in which $d\bar{V}_1$ and $d\bar{V}_2$ represent volumes of the elementary prism submerged in fluids of specific weights w_1 and w_2 respectively. The total buoyant force is

$$F_B = +\int dF_B = +\int (w_1 d\bar{V} + w_2 d\bar{V}_2)$$
$$= (w_1\bar{V}_1 + w_2\bar{V}_2)$$

which is the total weight of the displaced fluids. In this case it is therefore observed that the total buoyant force consists of the sum of the two components which correspond to the weight of the displaced fluids. The centre of buoyancy for each of these components obviously coincides with the centroid of the corresponding volume of fluid displaced, and may be located independently.

When a body is floating at the free surface of a liquid, it remains partially submerged in the liquid (as shown in Fig. 4.3), with the top portion of the body remaining in contact with air and its bottom portion being submerged in the liquid. In this case since the specific weight of air is negligible as compared with the specific weight of liquid, the weight of the air displaced by the top portion of the body may be neglected. Therefore same considerations as above show that the buoyant force exerted on a partially submerged body is, equal to the weight of the liquid displaced by the body, acting at the centre of buoyancy which obviously coincides with the centroid of the volume of the liquid displaced.

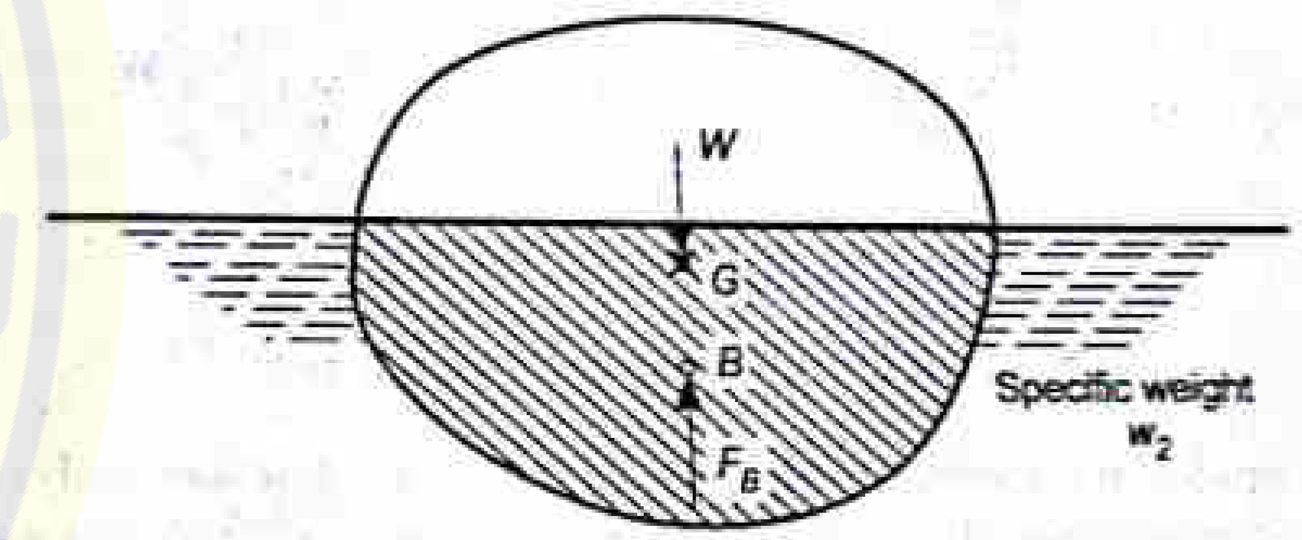


Figure 4.3 *Buoyant force on a body floating at the free surface of the liquid*

For a body immersed (either wholly or partially) in a fluid, the self-weight of the body always acts in the vertical downward direction. As such if a body floating in fluid is to be in equilibrium the buoyant force must be equal to the weight of the body i.e.,

$$F_B = W \qquad ...(4.2)$$

Further the lines of action of both, the buoyant force and the weight of the body must lie along the same vertical line, so that their moment about any axis is zero.

Equation 4.2 represents the *principle of floatation* which states that *the weight of a body floating in a fluid is equal to the buoyant force which in turn is equal to the weight of the fluid displaced by the body*. Thus for a body immersed in a fluid if the buoyant force exceeds the weight of the body, the body will rise until its weight equals the buoyant force. On the other hand if the weight of an immersed body exceeds the buoyant force, the body will tend to move downward and it may finally sink.

4.2 METACENTRE AND METACENTRIC HEIGHT

Consider a body floating in a liquid. If it is statically in equilibrium, it is acted upon by two forces viz., the weight of the body W acting at the centre of gravity G of the body and the buoyant force F acting at the centre of buoyancy B. The forces F_B and W are equal and opposite and as shown in Fig. 4.4 the points G and B lie along the same vertical line which is the vertical axis of the body.

Let this body be tilted slightly or it undergoes a small angular displacement (or angle of heel) θ. It is assumed that the position of the centre of gravity G remains unchanged relative to the body. The centre of buoyancy B, however, does not remain fixed relative to the body. This is so because as the body is tilted the portion of the body immersed on the right hand side increases while that on the left hand side decreases and hence the centre of buoyancy moves to a new position B_1. In the tilted position of the body the buoyant force acts in a vertical upward direction at B_1. Now if a vertical line is drawn through the new centre of buoyancy B_1 (or in other words the line of action of the buoyant force is extended) it intersects the axis of the body BG at point M, which is known as *metacentre*. Thus metacentre

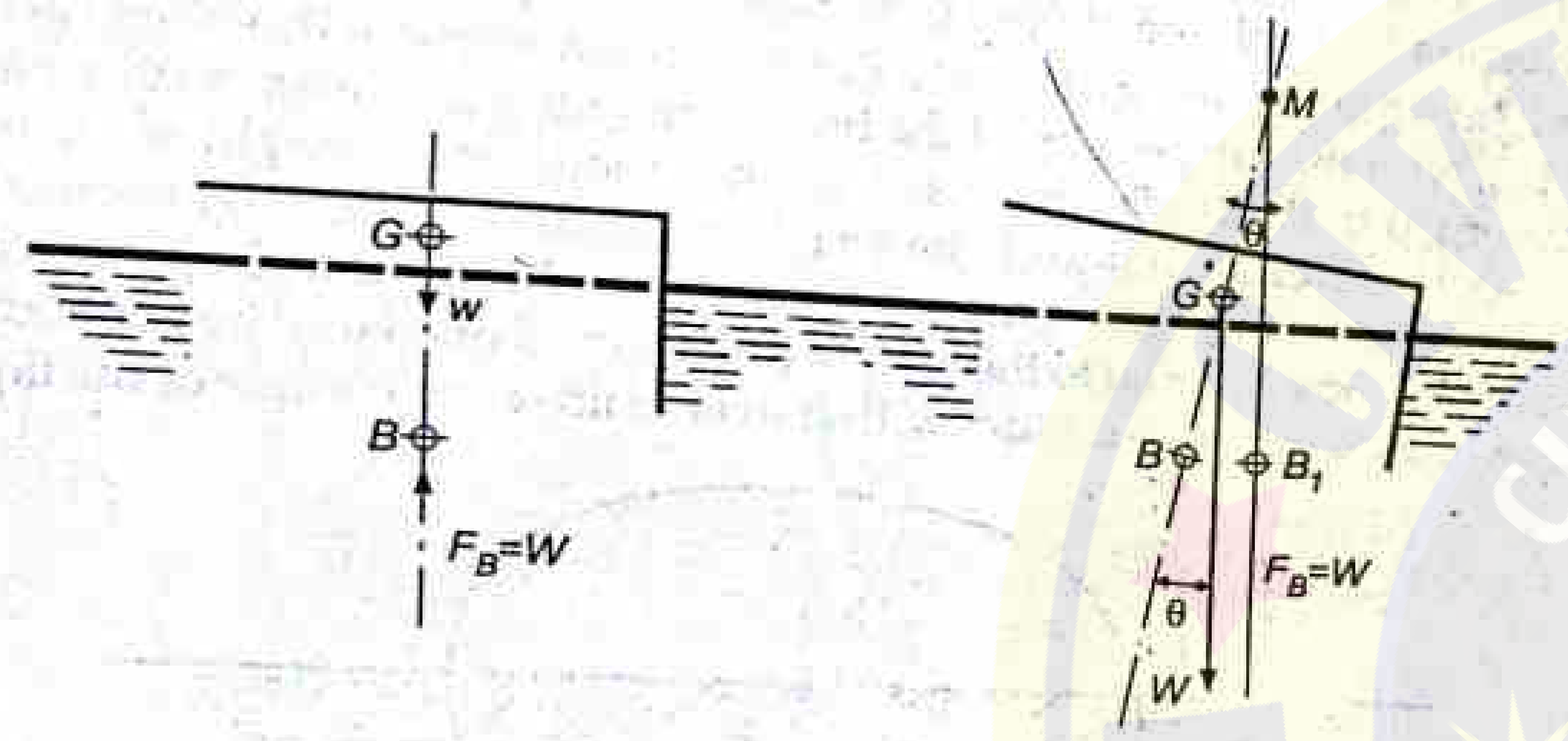


Figure 4.4 *Metacentre for a floating body*

may be defined as the point of intersection between the axis of the floating body passing through the points B and G and a vertical line passing through the new centre of buoyancy B_1. For small values of the angle of heel θ, the position of the metacentre M is practically constant. Further the distance between the centre of gravity G and the metacentre M of a floating body (i.e., GM) as $\theta \to 0$, is known as *metacentric height*. As indicated in the next section the position of the metacentre relative to the position of the centre of gravity of a floating body determines the stability condition of the floating body.

4.3 STABILITY OF SUBMERGEDAND FLOATING BODIES

By stability of a submerged or a floating body is meant the tendency for the body to return to the original upright position after it has been displaced slightly. A submerged or a floating body may be given a small linear displacement in horizontal or vertical direction or it may be given small angular displacement under the action of external forces. If a submerged or a floating body which is in equilibrium, is slightly displaced in the vertical or horizontal direction, then after the removal of the outside force causing such displacement, certain unbalanced force is developed on the body which tends to return the body to its original position. For example if a small upward displacement is given to a floating body, it results in the reduction of the buoyant force acting on the body thereby developing an unbalanced downward force which tends to return the body to its original position. However a slight horizontal displacement does not change either the magnitude or the location of the buoyant force, the body is still in equilibrium.

When a submerged or a floating body is given a slight angular displacement, it may have either of the following three conditions of equilibrium developed viz.,

(i) Stable equilibrium.
(ii) Unstable equilibrium.
(iii) Neutral equilibrium.

(i) Stable equilibrium. A body is said to be in a state of stable equilibrium if a small angular displacement of the body sets up a couple that tends to oppose the angular displacement of the body, thereby tending to bring the body back to its original position.

(ii) Unstable equilibrium. A body is said to be in a state of unstable equilibrium if a small angular displacement of the body sets up a couple that tends to further increase in the angular displacement of the body, thereby not allowing the body to restore its original position.

***(iii)* Neutral equilibrium.** A body is said to be in a state of neutral equilibrium if a small angular displacement of the body does not set up couple of any kind, and therefore the body adopts the new position given to it by the angular displacement, without either returning to its original position, or increasing the angular displacement.

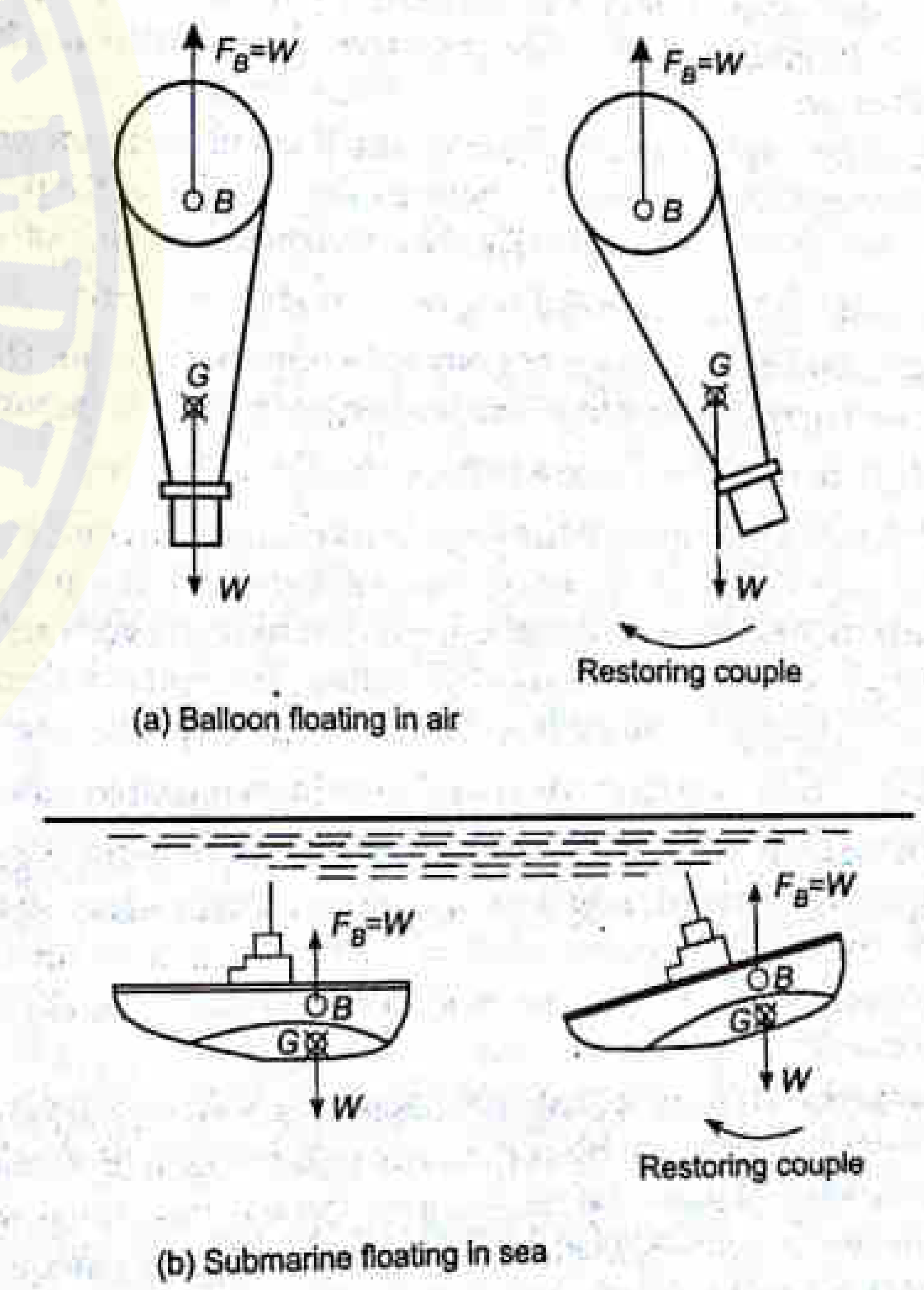


Figure 4.5 *Stability of a wholly submerged body*

For a body such as a balloon or a submarine, which is wholly submerged in a fluid, the positions of its centre of gravity and centre of buoyancy remain fixed. Therefore for a body wholly submerged in a fluid the conditions for stability of equilibrium are simple. In general a wholly submerged body is considered to be in a stable state of equilibrium if its centre of gravity is below the centre of buoyancy. This is so because as shown in Fig. 4.5, if such a submerged body is tilted slightly in a counter clockwise direction, the buoyant force and the weight produce a couple in the clockwise direction which restores the body to its original position. On the other hand a wholly submerged body will be in an unstable state of equilibrium if its centre of buoyancy is below its centre of gravity, because the couple produced by a slight angular displacement of the body tends to rotate the body away from its original position. However, if the centre of gravity and centre of buoyancy of a wholly submerged body coincide with each other, it is rendered in a neutral state of equilibrium.

A body floating in a liquid (or a partially immersed body) which is initially in equilibrium (i.e., $FB = W$) when undergoes a small angular displacement, the shape of the immersed volume in general changes and hence centre of buoyancy moves relative to the body. It is on account of this shifting of the centre of buoyancy in the case of a partially immersed body that stable equilibrium can be achieved even when the centre of buoyancy is below the centre of gravity. Moreover, for a floating body the stability is not determined simply by the relative positions of the centre of gravity and the centre of buoyancy, but it depends on the position of the metacentre relative to the position of the centre of gravity as described below.

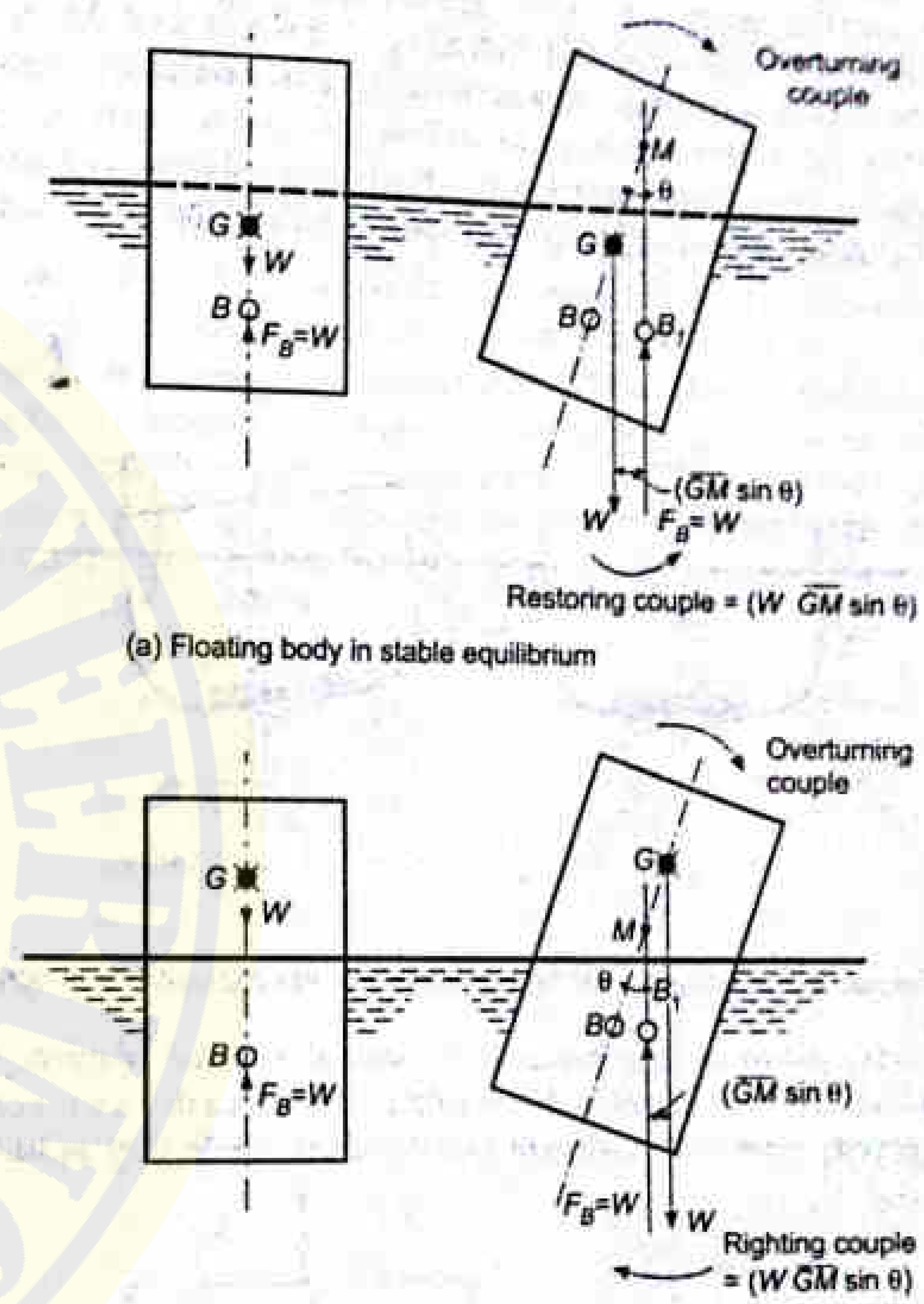
Consider a floating body which has undergone a small angular displacement in the clockwise direction as shown in Fig. 4.6. If the new centre of buoyancy B_1 is such that the metacentre M lies above the centre of gravity G of the body, as shown in Fig. 4.6 (a), then the buoyant force F_B and the weight W produce a couple of magnitude ($W\overline{GM}\sin\theta$) acting on the body in the anticlockwise direction, which is thus a restoring couple, tending to restore the body to its original position. The body is therefore in stable equilibrium. Hence it may be stated that for a floating body if the metacentre lies above its centre of gravity, i.e., $\overline{BM} > \overline{BG}$, then the body is in a stable state of equilibrium.

As shown in Fig. 4.6 (b), if for a floating body slightly tilted in clockwise direction, the metacentre M lies below the centre of gravity G of the body, then the buoyant force and the weight produce a couple acting on the body in the clockwise direction, which is thus an overturning couple, tending to increase the angular displacement of the body still further. The body is then considered to be in unstable equilibrium. Thus it may be stated that for a floating body if the metacentre lies below its centre of gravity, i.e., $\overline{BM} < \overline{BG}$, then the body is said to be in an unstable state or equilibrium.

However, if for a floating body the metacentre coincides with the centre of gravity of the body i.e., $BM = BG$, then the body will be in a neutral state of equilibrium. This is so because there will be neither a restoring couple nor an overturning couple developed when the body is tilted slightly. As such the body will neither return to its original position nor increase its angular displacement, but it will simply adopt its new position.

Stability of floating objects such as boats, ships etc., is required to be studied on account of the fact that these objects are always acted upon by certain external forces which may temporarily cause angular displacement or heeling of these objects. Amongst these forces which are common are : wind and wave action, pressure due to tidal or river currents, pressure due to manoeuvring a boat or ship in a curved path by rudder or propeller action, anchor line pulls etc. The heeling may also be caused by any probable shifting of the cargo or of the passengers, or by any probable variation in the weight and

distribution of the cargo (for example due to consumption of fuel and other supplies) etc. Therefore in the design of the floating objects such as boats, ships etc., care has to be taken to keep the metacentre well above the centre of gravity of the object so that even under the worst conditions the floating object



(a) Floating body in stable equilibrium

(b) Floating body in unstable equilibrium

Figure 4.6 *Stability of a partially immersed (or floating) body*

shall be in a stable state of equilibrium. This may however be achieved by lowering the position of the centre of gravity of the body by loading it permanently by adding some heavier material known as ballast. Furthermore the location of the metacentre for the objects may be known by determining the metacentric height as discussed in Section 4.4.

4.4 DETERMINATION OF METACENTRIC HEIGHT

There are two methods which may be used to determine the metacentric height of a floating body. These are

(i) Experimental Method.

(ii) Theoretical Method.

(i) Experimental Method for the Determination of the Metacentric Height. As shown in Fig. 4.7, consider a ship (or a boat) floating in water. Let w be a movable weight placed centrally on the deck of the ship and W be the total weight of the ship including w. It is assumed that the ship is initially in equilibrium, so that the deck is horizontal. Now the weight w is moved transversely through a distance x across the deck, so that the ship tilts through a small angle θ, and comes to rest in a new position of equilibrium. The angle θ may be measured by noting the horizontal distance moved by a long plumb line or a pendulum, hanging inside the ship from the centre point of the deck, on a horizontal scale, as shown in Fig. 4.7.

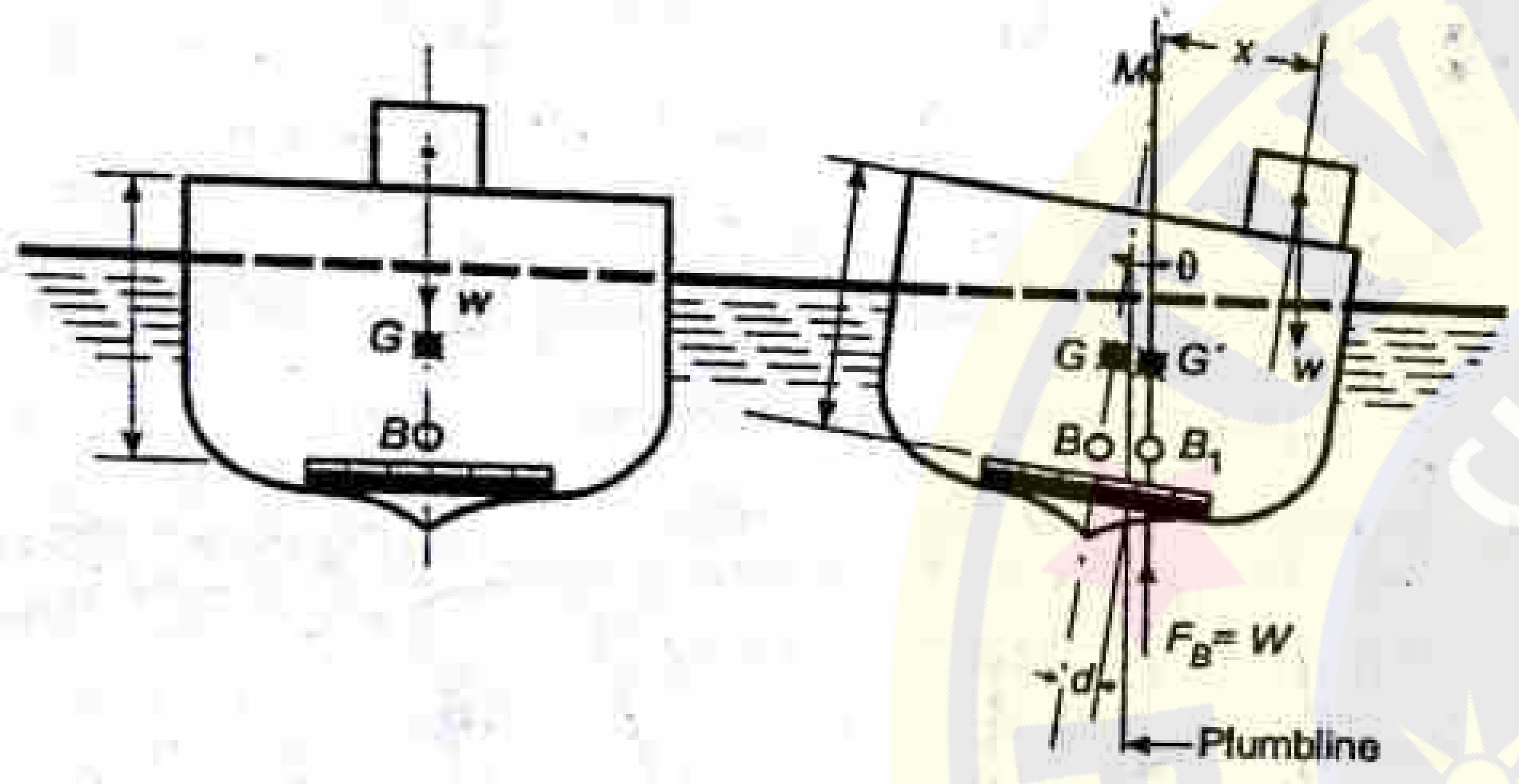


Figure 4.7 *Experimental determination of the metacentric height*

In the new (i.e., tilted) position of equilibrium of the ship, the centres of gravity and buoyancy will again be vertically in line. The movement of the weight w through a distance x causes a parallel shift of the total centre of gravity (that is the centre of gravity of the whole ship including w) from G to G' such that

$$wx = W\left(\overline{GG'}\right)$$

But $$\overline{GG'} = (\overline{GM})\tan\theta \text{ and so}$$

$$W(\overline{GM})\tan\theta = wx$$

$$\therefore \quad (\overline{GM}) = \frac{wx}{W\tan\theta} \qquad \text{...(4.3)}$$

Now if l is the length of the plumb line or pendulum and d is the distance moved by it on the horizontal scale then

$$\tan\theta = \frac{d}{l}$$

and therefore $\overline{GM} = \dfrac{wx}{W} \times \dfrac{l}{d}$ ...(4.3 a)

Since the point M corresponds to the metacentre for small angles of heel only, the true metacentric height is the limiting value of $\overline{GM}$ as $\theta \to 0$. This may be determined from a graph plotted between the nominal values of calculated from Eq. 4.3 for various values of θ (both positive and negative) and the angle θ.

The metacentric height for a ship or a boat may be determined by the method discussed above only after it has been constructed. But in many cases it is, however, desirable to be able to determine the metacentric height before a ship or a boat has been constructed. In such cases the theoretical method for the determination of the metacentric height, as discussed below, may be adopted.

(ii) Theoretical Method for the Determination of the Metacentric Height. By this method the metacentric height for a ship or a boat may be determined simply by considering its shape Fig. 4.8 shows the different views of a ship. At (a) is shown the initial equilibrium position of the ship and at (b) is shown the position of the ship after it has been tilted about the longitudinal axis through a small angle θ. In the tilted position of the ship the portion on the left AOA' has emerged from the liquid,

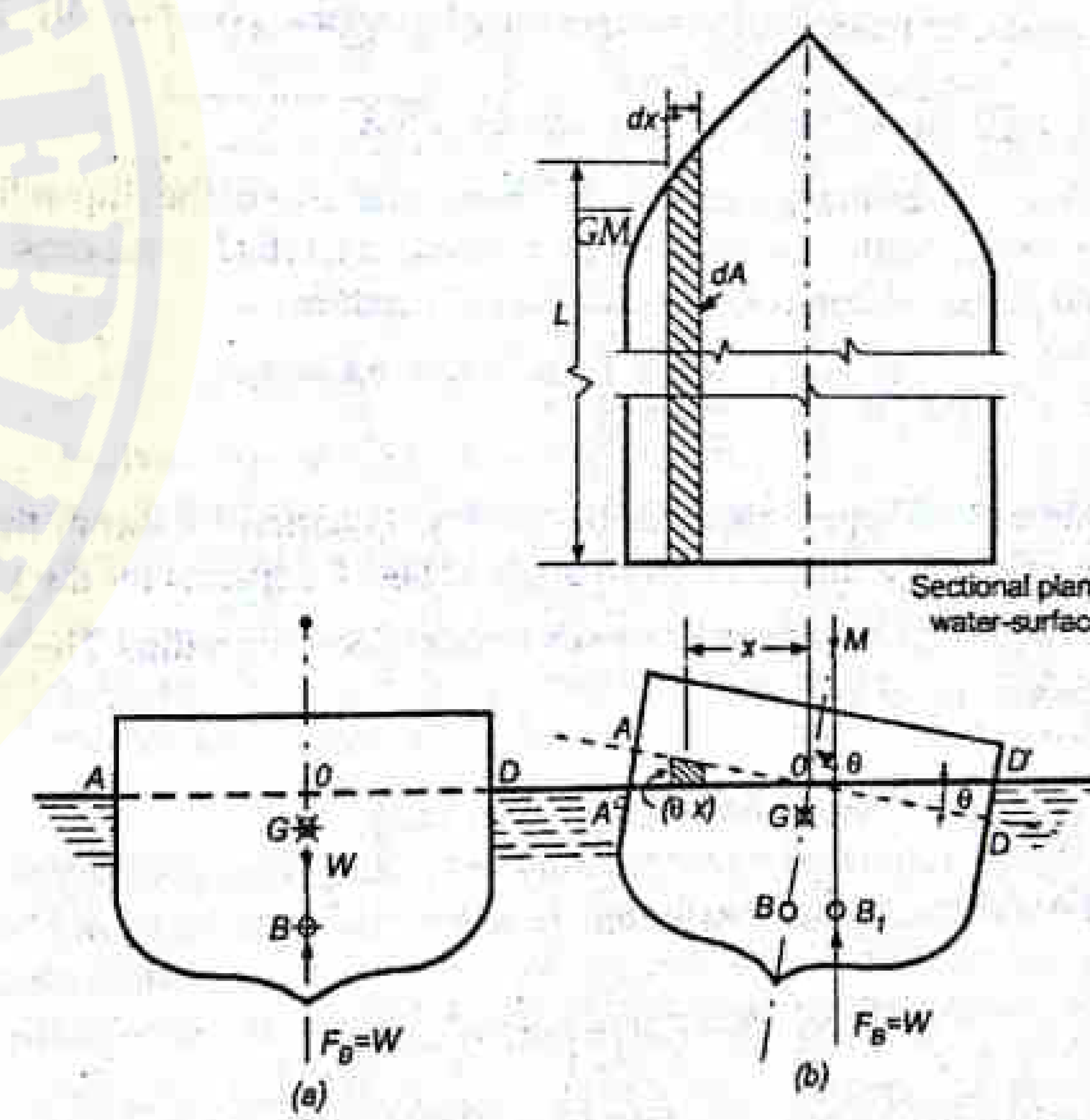


Figure 4.8 *Determination of metacentric height by the theoretical method*

whereas the portion DOD' on the right has moved down into the liquid. It is assumed that there is no overall vertical movement, thus the vertical equilibrium is not disturbed. Further as the total weight of the body is not altered, the volume immersed remains unaltered, and therefore the volumes corresponding to AOA' and DOD' wedges on either sides are equal. This is so if the planes of floatation for the equilibrium and the tilted positions intersect along the centroidal axes of the planes.

As the ship is tilted its centre of buoyancy shifts from B to B_1 which causes a change in the moment of the buoyant force amounting to $(F_B \times \overline{BM} \times \theta)$. This shift of the centre of buoyancy is entirely due to change in the geometrical form of the displaced volume by the submergence of the right wedge DOD' into the liquid and the emergence of the left wedge AOA' from the liquid. These two wedges may represent respectively a positive and negative buoyant force which form a couple. The effect of this couple must be the same as the change in moment of the buoyant force that is $(F_B \times \overline{BM} \times \theta)$ produced by the shift of the total buoyant force F_B.

To evaluate the couple due to the wedges. consider two small prisms of the wedges at a distance x from O on either side. The volume of each prism is $(L\theta x dx)$, where L is the length of the ship. If w is the specific weight of the liquid, then the weight of the volume of the liquid equal to that of the prism is $(wL\theta x dx)$. The moment of the couple due to this pair of prisms is $(2x \times wL\theta x dx)$. The summation of the moments of such prisms considered in the wedges must be equal to $(F_B \times \overline{BM} \times \theta)$. Thus

$$F_B \times \overline{BM} \times \theta = 2w\,\theta + \int x^2 L dx = 2w\,\theta + \int x^2 dA$$

in which dA $(= Ldx)$ is an elementary area of the cross-sectional area of the ship at the liquid surface, as shown in Fig. 4.8. The quantity $(2 + \int x^2 dA)$ is the moment of inertia I of the cross-sectional area of the ship at the liquid surface about its longitudinal axis. Therefore

$$\overline{BM} = \frac{wI}{F_B} = \frac{wI}{wV} = \frac{I}{V} \qquad \text{...(4.4)}$$

where V is the volume of the liquid displaced by the ship. Equation 4.4 shows that the location of metacentre for any ship tilting through a small angle of heel θ depends on the geometry and the weight. Further the length $\overline{BM}$ is sometimes known as the metacentric radius. The metacentric height GM may then be determined as

$$\overline{GM} = (\overline{BM} - \overline{BG}) = \left(\frac{I}{V} - \overline{BG}\right)$$

if the metacentre M lies above the centre of gravity G, and

$$\overline{GM} = (\overline{BG} - \overline{BM}) = \left(\overline{BG} - \frac{I}{V}\right)$$

if the metacentre M lies below the centre of gravity G. Both the above expressions for $\overline{GM}$ may be combined as follows:

$$\overline{GM} = \pm(\overline{BM} - \overline{BG}) = \pm\left(\frac{I}{V} - \overline{BG}\right). \qquad \text{...(4.5)}$$

4.5 METACENTRIC HEIGHT FOR FLOATING BODIES CONTAINING LIQUID

If a floating body contains liquid with a free surface then as shown below, the metacentric height of the body is reduced and consequently the stability of the body is reduced.

Consider a rectangular body of width b and length l floating in a liquid of specific weight w and containing liquid of specific weight w_1 as shown in Fig. 4.9 (a). When the body floats with its axis vertical, let CD be the free surface of the liquid contained by it, G be the centre of gravity of the floating body including the inside liquid and B be the centre of buoyancy. Now if the body is tilted through an angle θ, the centre of buoyancy would shift from B to B_1 and since the surface of the liquid in the body will remain horizontal, the new surface of this liquid is represented by HK as shown in Fig. 4.9 (b).

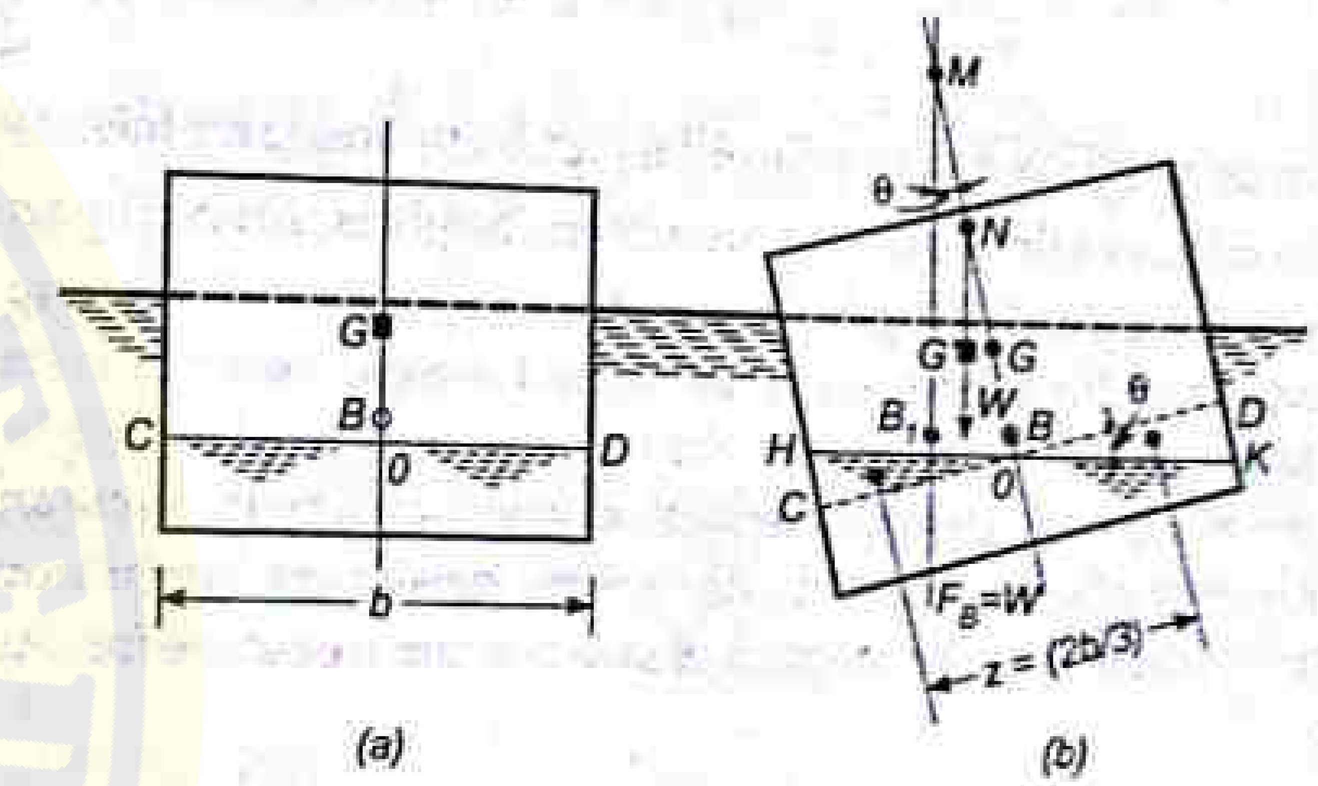


Figure 4.9 *Metacentric height for a floating*

Thus the tilting of the body has the effect of moving the wedge of liquid DOK to COH, which would result in shifting of the centre of gravity of the inside liquid and this in turn would cause the centre of gravity G to shift to new position G'. The moment of the restoring couple acting on the body is then equal to $W\ \overline{MN}\ \sin\theta$ (or $W\ \overline{MN}\ \theta$ for θ being small), where W is the weight of the floating body including the liquid contained by it, M is the metacentre and N is the point of intersection between the vertical through G' and the vertical axis of the body. As such for this case the metacentric height is $\overline{MN}$.

The movement of the wedge of the liquid DOK to COH develops a turning couple acting on the body in the same direction in which the body tilts, due to which the restoring couple acting on the body is reduced. The moment of this turning couple due to the movement of the liquid is $(w_1 V_1 z)$ where V_1 is the volume of the either wedge and z is the distance between the centre of gravity of these wedges. If θ is small,

$$CH = DK = (b/2)\,\theta \text{ and hence}$$

$$V_1 = (b^2/8)\,l\,\theta \text{ and } z = (2b/3)$$

$$\therefore \quad (w_1 V_1 z) = w_1 (lb^3/12)\theta = (w_1 I_1 \theta)$$

where $I_1 = (lb^3/12)$ is the moment of inertia of the area of the free surface of the liquid contained by the floating body about its longitudinal axis.

Thus the restoring couple acting on the body is

$$(W\overline{MN}\ \theta) = (W\overline{GM}\ \theta) - (w_1 I_1 \theta)$$

from which

$$\overline{MN} = \overline{GM} - \left(\frac{w_1 I_1}{W}\right)$$

Further since $W = wV$, where V is the volume of the liquid displaced by the floating body,

$$\overline{MN} = \overline{GM} - \left(\frac{w_1 I_1}{wV}\right)$$

In the above expression $\overline{GM}$ is given by Eq. 4.5 which is applicable for this case also. It is thus seen that when a floating body contains liquid the metacentric height is reduced by an amount $(w_1 I_1/wV)$.

If the liquid contained by a floating body is put into a number of separate compartments, the movement of the liquid within the body is minimized and consequently there would be less reduction in the metacentric height of the floating body. For this reason when a liquid is to be carried by a ship it is put into a number of separate compartments. It may be shown by the same analysis that when the liquid is put into a number of separate compartments the metacentric height would be reduced by an amount $[\Sigma(w_1 I'_1)/wV]$ where I'_1 is the moment of inertia of the area of the free surface of the liquid in each compartment.

4.6 TIME PERIOD OF TRANSVERSE OSCILLATION OF A FLOATING BODY

A floating body may be set in a state of oscillation as if suspended at the metacentre M in the same manner as a simple pendulum. This may happen when an overturning couple by which a floating body is tilted through an angle θ, is suddenly removed. The body is then acted upon by a torque equal to the moment of the restoring couple only, which is equal to $(W\ \overline{GM}\sin\theta)$ and sets it in a state of oscillation. But the torque exerted on an oscillating body is equal to the rate of change of angular momentum, which is equal to the moment of inertia (i.e., second moment of mass) multiplied by angular acceleration. Thus we may write

$$W\,\overline{GM}\sin\theta = -(Mk_G^2)\frac{d^2\theta}{dt^2} \qquad \text{...(4.6)}$$

where M is the mass of floating body ; k_G is radius of gyration of the body about its centre of gravity so that $(Mk_G{}^2)$ represents the moment of inertia of the body about its axis of rotation ; and $\left(\frac{d^2\theta}{dt^2}\right)$ is angular acceleration. Here the minus sign has been introduced because the torque acts so as to decrease θ, that is the angular acceleration $\left(\frac{d^2\theta}{dt^2}\right)$ is negative. Thus for smaller angular movements $(\sin\theta)$ is proportional to $\left(-\frac{d^2\theta}{dt^2}\right)$ as for a simple pendulum. If θ is small so that $\sin\theta \simeq \theta$ (radians), Eq. 4.6 may be expressed as

$$W\overline{GM}\,\theta = -(Mk_G^2)\frac{d^2\theta}{dt^2}$$

or

$$\frac{d^2\theta}{dt^2} = -\left(\frac{W\overline{GM}}{Mk_G^2}\right)\theta \qquad \text{...(4.6 a)}$$

The solution of the differential Eq. 4.6 (a) is as noted below

$$\theta = A\sin\left(t\sqrt{\frac{W\overline{GM}}{Mk_G^2}}\right) + B\cos\left(t\sqrt{\frac{W\overline{GM}}{Mk_G^2}}\right)$$ where A and B are constants.

Let T be the time period of one complete oscillation of the floating body from one side to the other and back again. Then if at $t = 0, \theta = 0$; again at $t = (T/2), \theta = 0$. Introducing these values in the above expression, we get

$$B = 0 \text{ and } A\sin\left(\frac{T}{2}\sqrt{\frac{W\overline{GM}}{Mk_G^2}}\right) = 0$$

$$\text{Since } A \neq 0; \ \therefore\ \sin\left(\frac{T}{2}\sqrt{\frac{W\ \overline{GM}}{Mk_G^2}}\right) = 0$$

from which

$$T = 2\pi\left\{\frac{Mk_G^2}{W\overline{GM}}\right\}^{1/2} = 2\pi\left\{\frac{k_G^2}{g\overline{GM}}\right\}^{1/2} \qquad \text{...(4.7)}$$

where g is acceleration due to gravity.

It may however be indicated that the oscillating motion of a floating body ultimately dampens by the frictional forces between the floating body and the liquid.

A ship or a boat may have two types of oscillatory motions viz., *rolling* and *pitching*. The oscillatory motion of a ship or a boat about its longitudinal axis is designated as rolling. On the other hand the pitching movement or simply pitching may be defined as the oscillatory motion of a ship or a boat about its transverse axis. Although the above noted expressions for the metacentric height (Eq. 4.5) and the time period of oscillation (Eq. 4.7) are derived for the rolling motion, the same may be adopted for the pitching motion also. It may, however, be noted that since the moment of inertia of the cross-sectional area of the ship or boat at the liquid surface about its transverse axis is much more than the same about its longitudinal axis, the metacentric height for the pitching motion is invariably greater than that for the rolling motion. As such if a ship or a boat has a safe metacentric height for rolling motion then it will be safe in pitching motion also. As regards the time period of oscillation it has been found that for rolling motion of ships or boats the theoretical values of T given by Eq. 4.7 agree reasonably with the experimental values, but the agreement is less good for pitching movements.

In the case of ships the shifting of cargo may cause the ships to roll. As such along with the considerations of the stability of a ship, its period of roll is also required to be determined. This is so because, as indicated earlier, increasing the metacentric height gives greater stability to a floating body, but as indicated by Eq. 4.7 an increase in the metacentric height reduces the time period of rolling of the body. A smaller value of time period of rolling of a passenger ship is quite uncomfortable for the passengers. Further a ship with a smaller time period of rolling is subjected to undue strains which may damage its structure. In the case of cargo ships the metacentric height varies with the loading and hence some control on the value of the metacentric height as well as the time period of rolling is possible by adjusting the position of the cargo. However, in the case of warships and racing yachts, the stability is more important than comfort, and hence such vessels have larger metacentric height.

The metacentric height of ocean-going vessels is usually of the order of 0.3 m to 1.2 m. On the other hand for warships it may range from 1 m to 1.5 m and for some of the river crafts it may be as large as about 3.6 m.

ILLUSTRATIVE EXAMPLES

Examples 4.1. *A wooden block of rectangular section 1.25 m wide, 2 m deep, 4 m long floats horizontally in sea water. If the specific gravity of wood is 0.64 and water weighs 1 025 kg(f)/m³, find the volume of liquid displaced and the position of the centre of buoyancy.*

Solution

According to Archimedes' principle weight of the liquid displaced by the body = weight of the body

Weight of the block= $(1.25 \times 2 \times 4) \times 0.64 \times 1000$

$= 6\,400$ kg(f)

Thus

Volume of sea water displaced by the block $= \dfrac{6400}{1025} = 6.24\ \text{m}^3$

Let h be the depth of the block under water, then

$$(4 \times 1.25 \times h) = 6.24$$

$$\therefore \qquad h = \frac{6.24}{4 \times 1.25} = 1.248\ \text{m}$$

Centre of buoyancy B is at a height of $\dfrac{1.248}{2} = 0.624$ m above the base.

Example 4.2. *A wooden cylinder of circular section and uniform density, specific gravity 0.6 is required to float in oil of specific gravity 0.8. If the diameter of the cylinder is d and its length is l, show that l cannot exceed about 0.817d for cylinder to float with its longitudinal axis vertical.*

Solution

As shown in the Fig. Ex. 4.2 if the depth of immersion is x then

$$\left(\frac{\pi d^2 x}{4}\right) \times 0.8 = \left(\frac{\pi d^2}{4} \times l\right) \times 0.6$$

or
$$x = \left(\frac{0.6}{0.8}\right) l = \frac{3}{4} l$$

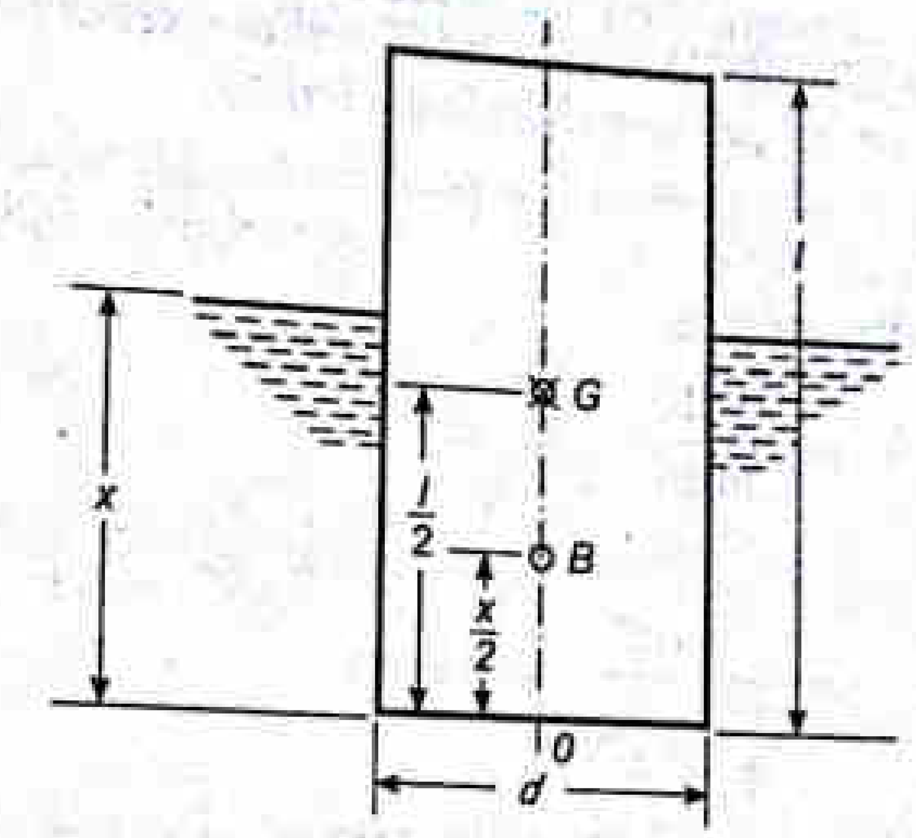


Figure. Ex. 4.2

So
$$\overline{OB} = \frac{x}{2} = \frac{3}{8} l$$

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

$$I = \frac{\pi d^4}{64}$$

$$V = \left(\frac{\pi d^2}{4} \times \frac{3}{4} l\right) = \frac{3\pi d^2}{16} l$$

$$\therefore \qquad \overline{BM} = \left(\frac{\pi d^4}{64} \times \frac{16}{3\pi d^2 l}\right) = \frac{d^2}{12 l}$$

Further $\overline{BG}$

$$= (\overline{OG} - \overline{OB})$$

$$= \left(\frac{l}{2} - \frac{3}{8} l\right) = \frac{l}{8}$$

For stable equilibrium

$$\overline{BM} > \overline{BG}$$

or
$$\frac{d^2}{12 l} > \frac{l}{8}$$

or
$$l^2 < \frac{2}{3} d^2$$

or
$$l < \left(\sqrt{2/3}\right) d$$

or
$$l < 0.817\, d$$

Example 4.3. *Show that a cylindrical buoy 1.25 m diameter and 3.25 m high weighing 11 127 N will not float vertically in sea water weighing 10 055 N/m³. Find the tension necessary in a vertical chain attached to the centre of the base of the buoy that will just keep the cylinder vertical.*

Solution

Weight of the buoy $= 11\,127$ N

$\therefore$ Volume of sea water displaced is

$$V = \frac{11127}{10055} = 1.107 \text{ m}^3$$

Depth of immersion

$$= \frac{1.107}{\frac{\pi}{4}\times(1.25)^2} = 0.90 \text{ m}$$

$\therefore$ Height of the centre of buoyancy B above the base

$$= \frac{0.90}{2} = 0.45 \text{ m}$$

Since $\overline{BM} = \dfrac{I}{V}$

$$I = \frac{\pi d^4}{64} = \frac{\pi\times(1.25)^4}{64} = 0.119\,8 \text{ m}^4$$

$$V = 1.107 \text{ m}^3$$

$$\therefore \quad \overline{BM} = \frac{0.1198}{1.107} = 0.1082 \text{ m}$$

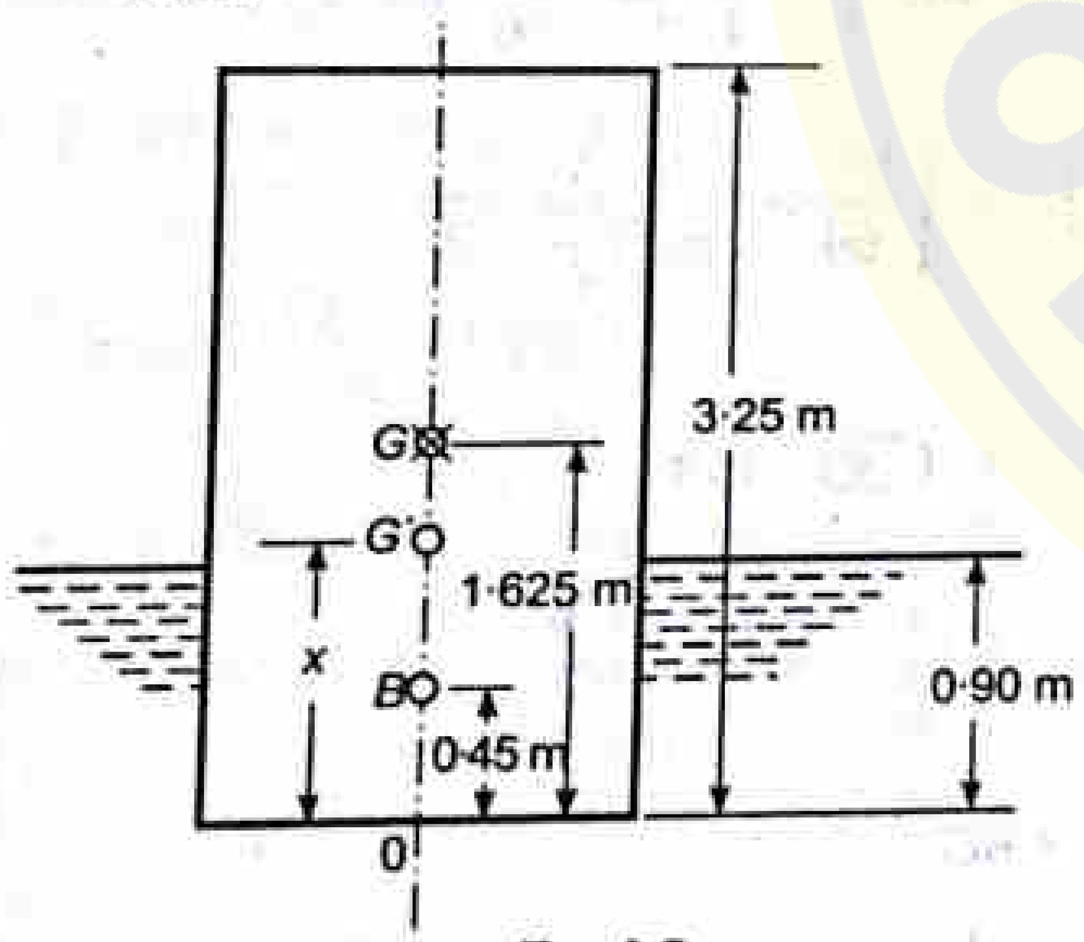


Figure. Ex. 4.3

Further $\overline{BG} = (\overline{OG} - \overline{OB})$

$$= \left[\left(\frac{1}{2}\times 3.25\right) - 0.45\right]$$

$$= 1.175 \text{ m}$$

Since $\overline{BM} < \overline{BG}$, metacentre M lies below the centre of gravity G, the cylinder will be in an unstable state of equilibrium.

Let T be the tension required to be applied vertically in the chain attached to the centre of the base. Hence total effective weight of the body becomes

$$(11\,127 + T)\ N$$

$\therefore$ Volume of sea water displaced is

$$V = \left(\frac{11\,127 + T}{10055}\right) \text{m}^3$$

Depth of immersion

$$\left(\frac{11127+T}{10055}\right) = \times \frac{1}{\frac{\pi}{4}\times(1.25)^2}$$

$$= \frac{(11127+T)\times 64}{(10055\times 78.54)} \text{m}$$

If B' is new centre of buoyancy, then the height of B' above the base of the buoy is

$$OB' = \frac{1}{2}\left[\frac{(11\,127+T)\times 64}{(10\,055\times 78.54)}\right] \text{m}$$

Also $\overline{B'M'} = \dfrac{I}{V}$

$$I = \frac{\pi d^4}{64} = \frac{\pi\times(1.25)^4}{64}$$

$$V = \left(\frac{11\,127+T}{10\,055}\right)$$

$$\therefore \quad \overline{B'M'} = \frac{\pi\times(1.25)^4\times 10055}{64\times(11127+T)} \text{m}$$

The effect of applying a tension T is to lower the centre of gravity of the buoy. Thus if G' is new centre of gravity then since,

Weight acting at $G = 11\,127$ N and weight acting at $G' = (11\,127 + T)$ N taking moments about O, we get

$$(11\,127 + T)\,x = 11\,127\times\frac{3.25}{2}$$

$$\therefore \quad x = \frac{(11127\times 3.25)}{2(11127+T)} \text{m}$$

For stable equilibrium the metacentre M' must be above G', i.e.,

$$\overline{B'M'} > \overline{B'G'}$$

Since

$$\overline{B'G'} = \left(\overline{OG} - \overline{OB'}\right)$$

$$= \left[\frac{(11127 \times 3.25)}{2(11127+T)} - \frac{(11127+T)\times 64}{2(10055 \times 78.54)}\right]$$

$$\therefore \quad \frac{\pi \times (1.25)^4 \times 10055}{64 \times (11127+T)} > \left[\frac{(11127 \times 3.25)}{2(11127+T)} - \frac{(11127+T)\times 64}{2(10055 \times 78.54)}\right]$$

or

$$\frac{(11127+T)\times 64}{2(10055 \times 78.54)} > \frac{(11127 \times 3.25 \times 32) - (\pi \times 1.25^4 \times 10055)}{(11127+T)\times 64}$$

or $\quad (11127 + T)^2 > 4.165 \times 10^8$

or $\quad (11127 + T) > 20\,408$

or $\quad T > 9281$

$\therefore$ Minimum tension in the chain is

$$T = 9281 \text{ N}$$

Example 4.4. *A cylindrical buoy, diameter 1.5 m and 1.1 m high weighing 4.450 kN is floating in sea water with its axis vertical. Find the maximum permissible height above the top of the buoy, of the centre of gravity of a 450 N load which is placed centrally on top of the buoy. Take specific gravity of sea water as 1.025.*

Solution

The weight of sea water displaced

$$= (4\,450 + 450) = 4\,900 \text{ N}$$

$\therefore$ Volume of sea water displaced is

$$V = \frac{4900}{1.025 \times 9810} = 0.487 \text{ m}^3$$

Depth of immersion

$$= \frac{0.487}{\frac{\pi}{4}\times(1.5)^2} = 0.276 \text{ m}$$

$\therefore$ The height of the centre of buoyancy above the base

$$= \frac{0.276}{2} = 0.138 \text{ m}$$

The combined centre of gravity of the buoy and the load may be determined by taking the moments above the base of the buoy. Thus if the distance of the centre of gravity of the load from the base is x and that of the combined centre of gravity G' is $\bar{x}$, then by taking the moments about the mid-point of base O, we get

$$4\,900\ \bar{x} = 4\,450 \times \frac{1.1}{2} + 450 \times x$$

$$\therefore \quad \bar{x} = \left(\frac{2447 \times 5 + 450x}{4900}\right)$$

$$= (0.499 + 0.092\,x)$$

Thus

$$\overline{BG'} = \overline{OG'} - \overline{OB'}$$

$$= \bar{x} - 0.138$$

$$= (0.499 + 0.092x) - 0.138$$

$$= (0.361 + 0.092x)$$

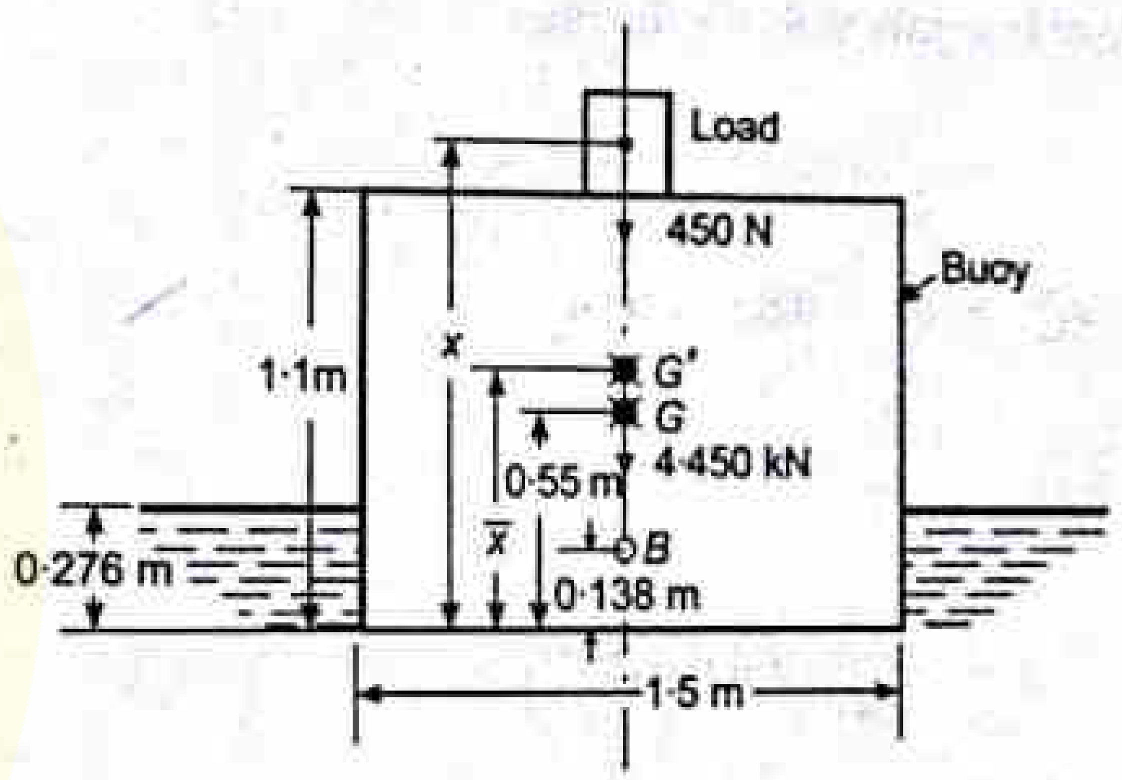


Figure Ex. 4.4

Since

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

$$I = \frac{\pi}{64}(1.5)^4 \text{ m}^4 ; V = 0.487 \text{ m}^3$$

$$\therefore \quad \overline{BM} = \frac{\pi}{64} \times \frac{(1.5)^4}{0.487} = 0.510 \text{ m}$$

For stable equilibrium of the floating buoy

$$\overline{BM} > \overline{BG'}$$

or $\quad 0.510 > (0.361 + 0.092\,x)$

or $\quad 0.092x < 0.149$

$\therefore \quad x < 1.62$ m

i.e., the centre of gravity of the load must not be more than $(1.62 - 1.1) = 0.51$ m above the top of the buoy.

Example 4.5. *A wooden cylinder of diameter d and length $2d$ floats in water with its axis vertical. Is the equilibrium stable? Locate the metacentre with reference to water surface. Specific gravity of wood is 0.6.*

Solution

Weight of the cylinder $= \left(\frac{\pi}{4}d^2 \times 2d\right) \times 0.6 \times 9\,810\text{ N} = 9\,245.7\,d^3\text{ N}$

$\therefore$ Volume of water displaced by the cylinder

$$= \frac{9245.7d^3}{9810} = 0.942\,d^3$$

Depth of immersion $= \dfrac{0.942d^3}{\frac{\pi}{4}d^2} = 1.2\,d$

$\therefore$ Height of the centre of buoyancy above the base

$$= \frac{1}{2}(1.2d) = 0.6\,d$$

$$\therefore \quad \overline{BG} = (d - 0.6d) = 0.4d$$

Since $\overline{BM} = \dfrac{I}{V}$

$$I = \frac{\pi d^4}{64};\ V = 0.942\,d^3$$

$$\therefore \quad \overline{BM} = \frac{\pi d^4}{64} \times \frac{1}{0.942d^3} = 0.052\,d$$

In this case since, $\overline{BM} > \overline{BG}$ the metacentre lies below the centre of gravity. As such the cylinder is in unstable equilibrium.

The metacentre is at a depth of $(0.6d - 0.052d) = 0.548d$ below the water surface.

Example 4.6. *A hollow cylinder of outside diameter 1.25 m, length 3.5 m and specific weight 75 537 N/m³, floats just in stable equilibrium in sea water. Find the minimum permissible thickness of the cylinder. Sea water weighs 10 055 N/m³.*

Solution

Let t be the thickness of the cylinder and x be the depth of the portion immersed in sea water. Then by the principle of floatation, weight of the liquid displaced by the body is equal to the weight of the body.

Weight of sea water displaced

$$= \left[\frac{\pi}{4} \times (1.25)^2 \times x \times 10055\right]\text{N}$$

$$= 12\,339.34\,x\text{ N}$$

Weight of the body

$$= [\pi \times (1.25) \times t \times 3.5 + 2 \times \frac{\pi}{4} \times (1.25)^2 \times t\,]\,75\,537$$

$$= 1\,223\,611.40\,t\text{ N}$$

In calculating the weight of the body it has been assumed that the thickness t is very small and hence the weight of the two end sections and the circular portion have been separately taken. Thus

$$12\,339.34\,x = 1\,223\,611.40\,t$$

$$\therefore \quad x = 99.16\,t$$

As $\overline{BM} = \dfrac{I}{V}$

$$I = \frac{\pi}{64}(1.25)^4 = 0.12\text{ m}^4$$

and $V = \dfrac{1223611.40\,t}{10055} = 121.69\,t\text{ m}^3$

$$\therefore \quad \overline{BM} = \frac{0.12}{121.69t} = \frac{0.986 \times 10^{-3}}{t}$$

$$\overline{BG} = \left[\frac{3.5}{2} - \frac{99.16t}{2}\right]$$

$$= (1.75 - 49.58\,t)$$

For the cylinder to be just in stable equilibrium

$$\overline{BM} = \overline{BG}$$

or $\dfrac{0.985 \times 10^{-3}}{t} = (1.75 - 49.58\,t)$

or $49.58\,t^2 - 1.75\,t + 0.986 \times 10^{-3} = 0$

$$t = 0.000\,57\text{ m or } 0.035\text{ m}$$

Thus minimum permissible thickness of the cylinder = 0.000 57 m = 0.57 mm.

Example 4.7. *A cone floating in water with its apex downwards has a diameter d and vertical height h. If the specific gravity of the cone is S, prove that for stable equilibrium*

$$h^2 < \frac{1}{4}\left[\frac{d^2 S^{1/3}}{1 - S^{1/3}}\right]$$

Solution

Let the height of the portion of the cone under water be x. Then, we have

$$\frac{1}{3} \times \frac{\pi d^2}{4} \times h \times wS = \frac{1}{3}\frac{\pi d_1^2}{4} \times x \times w$$

where d_1 is the diameter of the cone at the water surface level.

or
$$(d^2 h) \times S = (d_1^2 \times x)$$

Also
$$\left(\frac{d_1}{d}\right) = \frac{x}{h} \quad \text{...(i)}$$

or
$$d_1 = \frac{xd}{h} \quad \text{...(ii)}$$

Substituting the value of d_1 from Eq. (ii) in Eq. (i), we get

$$(d^2 h) \times S = x \times \left(\frac{xd}{h}\right)^2$$

or
$$x = h(S)^{1/3}$$

The centre of gravity of the cone is at a depth of $\frac{1}{4}$ th of the height of the cone below the base

$$\therefore \quad \overline{BG} = \left(\frac{3h}{4} - \frac{3x}{4}\right) = \frac{3}{4}(h - hS^{1/3})$$

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

$$I = \frac{\pi d_1^4}{64}; \text{and} \quad V = \frac{1}{3}\frac{\pi d_1^2}{4} \times x$$

$$\therefore \quad \overline{BM} = \frac{\frac{\pi d_1^4}{64}}{\frac{1}{3}\frac{\pi d_1^2}{4} \times x} = \frac{3d_1^2}{16x}$$

By substituting for d_1 and x, we get

$$\overline{BM} = \frac{3S^{1/3}d^2}{16h}$$

For stable equilibrium

$$\overline{BM} > \overline{BG}$$

or
$$\frac{3S^{1/3}d^2}{16h} > \frac{3}{4}(h - hS^{1/3})$$

or
$$h^2 < \frac{1}{4}\left[\frac{d^2 S^{1/3}}{1 - hS^{1/3}}\right]$$

Example 4.8. *An object which has a volume of 0.18 m³ requires a force of 265 N [27 kg(f)] to keep it immersed in water. If a force of 157 N [16 kg(f)] is required to keep it immersed in another liquid, find the specific gravity of the liquid.*

Solution

SI Units

Let S_1 and S_2 be the specific gravities of the object and the liquid respectively.

When the object is immersed in water, the total force acting in the downward direction

$$= (265 + 0.18 \times 9\,810 \times S_1)\text{ N}$$

Buoyant force $= (0.18 \times 9\,810)$ N

Thus equating the two, we get

$$(265 + 0.18 \times 9\,810 \times S_1) = (0.18 \times 9\,810)$$
$$S_1 = 0.85$$

When the object is immersed in another liquid, the total force acting in the downward direction

$$= (157 + 0.18 \times 9\,810 \times 0.85)\text{ N}$$

Buoyant force $= (0.18 \times 9\,810 \times S_2)$ N

Thus equating the two, we get

$$(157 + 0.18 \times 9\,810 \times 0.85) = (0.18 \times 9\,810 \times S_2)$$
$$S_2 = 0.94$$

Metric Units

Let S_1 and S_2 be the specific gravities of the object and the liquid respectively.

When the object is immersed in water, the total force acting in the downward direction

$$= (27 + 0.18 \times 1000 \times S_1)\text{ kg (f)}$$

Buoyant force $= (0.18 \times 1000)$ kg (f)

Thus equating the two, we get

$$(27 + 0.18 \times 1000\,S_1) = (0.18 \times 1000)$$
$$S_1 = 0.85$$

When the object is immersed in another liquid, the total force acting in the downward direction

$$= (16 + 0.18 \times 1000 \times 0.85)\text{ kg (f)}$$

Buoyant force $= (0.18 \times 1000 \times S_2)$ kg(f)

Thus equating the two, we get

$$(16 + 0.18 \times 1000 \times 0.85) = (0.18 \times 1000 \times S_2)$$
$$S_2 = 0.94$$

Example 4.9. *A closed pontoon 4 m square by 1m deep is divided into 4 equal compartments by vertical diaphragms. It is constructed throughout from 6 mm thick metal weighing 78 480 N/m³. Calculate the metacentric height when floating in sea water.*

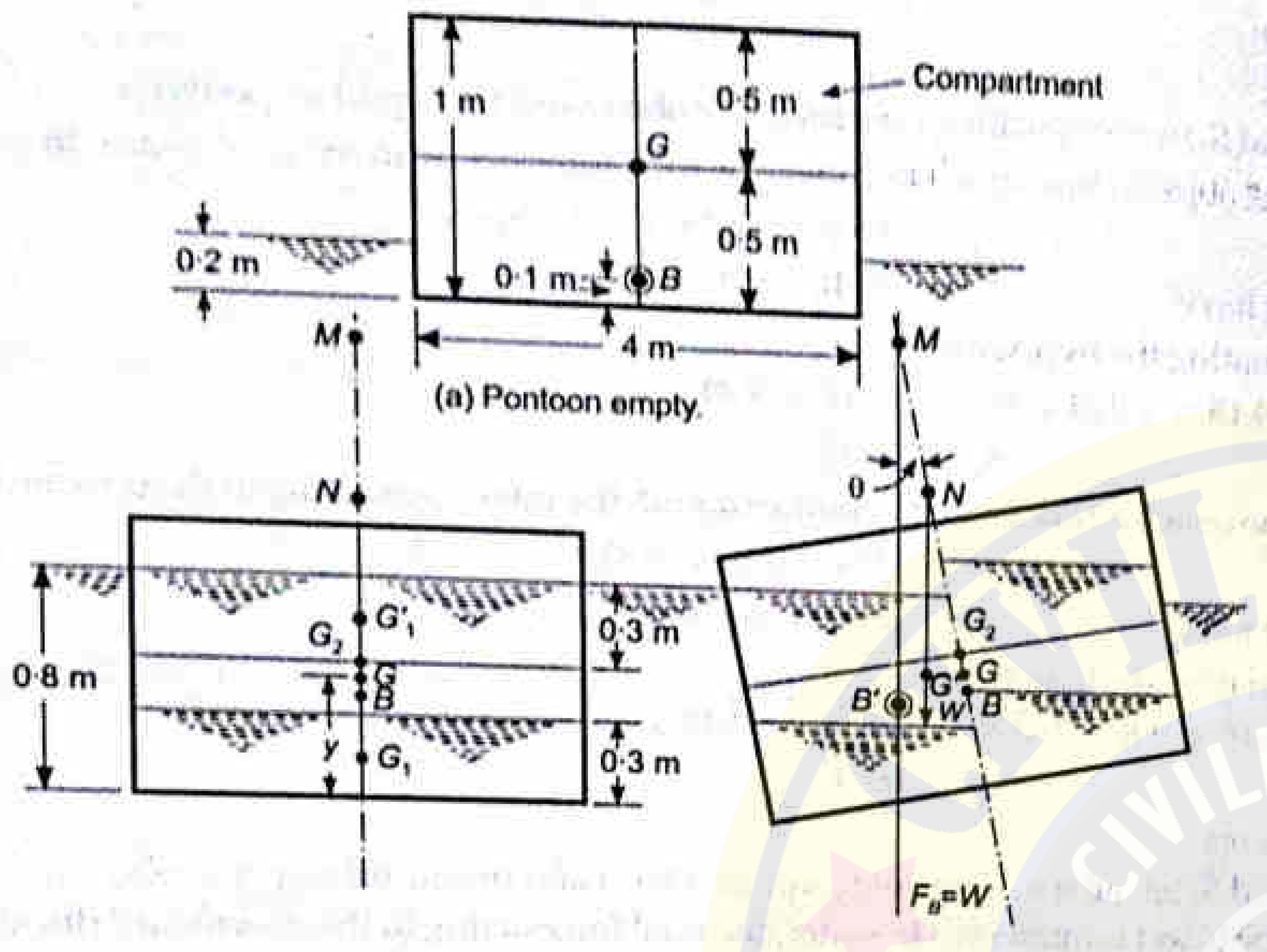
(i) With no water inside.

(ii) When sea water is filled in all the compartments to a depth of 0.3 m. Specific weight of sea water is 10 006 N/m³.

Solution

(i) The weight of the pontoon along with the diaphragm is

$$W = \left[5\left\{4 \times 1 \times \frac{6}{1000}\right\} + 3\left\{4 \times 4 \times \frac{6}{1000}\right\}\right] 78\,480$$
$$= 32\,019.84\text{ N}$$



(b) Pontoon with compartments filled with sea water

Figure Ex. 4.9

$\therefore$ Volume of sea water displaced is

$$V = \frac{32019.84}{10006} = 3.2\ \text{m}^3$$

Depth of immersion is

$$x = \frac{3.2}{4.4} = 0.2\ \text{m}$$

So the height of the centre of buoyancy above the bottom of the pontoon

$$= \frac{1}{2} \times (0.2) = 0.1\ \text{m}$$

$$\therefore \qquad \overline{BG} = (0.5 - 0.1) = 0.4\ \text{m}$$

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

$$I = \left[\frac{1}{12} \times 4 \times (4)^3\right] = 21.33\ \text{m}^4,$$

and
$$V = 3.2\ \text{m}^3$$

$$\therefore \qquad \overline{BM} = \frac{21.33}{3.2} = 6.67\ \text{m}$$

$$\therefore \qquad \overline{GM} = \overline{BM} - \overline{BG} = (6.67 - 0.4) = 6.27\ \text{m}$$

(ii) When sea water is filled in all the inner compartments the total weight of the pontoon is

$$W = 32\,019.84 + 4\,[4 \times 2 \times 0.3] \times 10{,}006$$
$$= 128\,077.44\ \text{N}$$

$\therefore$ Volume of sea water displaced is

$$V = \frac{128077.44}{10006} = 12.8\ \text{m}$$

$$\therefore \qquad \text{Depth of immersion} = \frac{12.8}{4 \times 4} = 0.8\ \text{m}$$

$\therefore$ Height of the centre of buoyancy above the bottom of the pontoon

$$= \frac{1}{2} \times (0.8) = 0.4\ \text{m}$$

As shown in Fig. Ex. 4.9 (b) let G_1 be the centre of gravity of lower tanks and G'_1 be the centre of gravity of the top tanks, and G_2 be the centre of gravity of the unloaded pontoon. Now if G is centre of gravity of the loaded pontoon and its height above the bottom of the pontoon is y, then the value of y can be determined by taking moments about the base:

$$[\,32\,019.84 \times 0.5 + 2(4 \times 2 \times 0.3) \times 10\,006 \times \frac{0.3}{2} + 2\,(4 \times 2 \times 0.3) \times 10\,006 \times (0.5 + \frac{0.3}{2})\,]$$
$$= 128\,077.44\ y$$
$$\therefore \qquad y = 0.425\ \text{m}$$

As shown in Fig. Ex. 4.9 (b) when the loaded pontoon heels through a small angle θ, the centre of buoyancy B moves to B', i.e., the centre of gravity of the displaced volume V moves such that

$$\overline{BM'} = \overline{BM} \times \theta = \frac{I}{V} \times \theta$$

$\therefore$ The centre of gravities of each of the compartments will also be displaced by

$$\frac{I_1}{V_1} \times \theta$$

where the subscript 1 refers to the compartments.

These movements will cause the point G to move to G' and if w is the specific weight of sea water then GG' is given by

$$V \times w \times \overline{GG'} = 4\left(V_1 \times w \times \frac{I_1}{V_1} \times \theta\right)$$

$$\therefore \qquad \overline{GG'} = \frac{4I_1}{V} \times \theta$$

Also
$$\overline{GG'} = \overline{NG} \times \theta$$

$$\therefore \qquad \overline{NG} = \frac{4I_1}{V}$$

where N is the point of intersection between the new vertical through G′ and the original vertical through G.

The unbalanced righting couple is now ($W\ \overline{MN} \sin\theta$)

$\therefore$ Metacentric height = $\overline{MN}$

From Fig. Ex. 4.9 (b)

$$\overline{MN} = \overline{GM} - \overline{NG}$$
$$= \overline{BM} - \overline{BG} - \overline{NG}$$

$$\overline{BM} = \frac{I}{V} = \frac{\frac{1}{12}\times 4\times(4)^3}{12.8} = 1.67$$

$$\overline{BG} = (0.425 - 0.4) = 0.025 \text{ m}$$

$$\overline{NG} = \frac{4I_1}{V} = \frac{4\left(\frac{1}{12}\times 4\times 2^3\right)}{12.8} = 0.83 \text{ m}$$

$$\therefore \quad \overline{MN} = (1.67 - 0.025 - 0.83)$$
$$= 0.815 \text{ m}$$

Example 4.10. *A battle ship weighs 127.53 MN (13 000 tonnes). On filling the ship's boats on one side with water weighing 588.6 kN (60 tonnes) and its mean distance from the centre of the ship being 10 m, the angle of displacemnt of the plumb line is 2° 16′. Determine the metacentric height.*

Solution

SI Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W\tan\theta}$$

w = 588.6 kN = 0.588 6 MN

W = (127.53 + 0.588 6) = 128.1186 MN

x = 10 m

and θ = 2°16′, so tan (2° 16′) = 0.039 6

$\therefore$ By substitution, we get

$$\overline{GM} = \frac{0.5886\times 10}{128.11860\times 0.0396} = 1.16 \text{ m}$$

Metric Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W\tan\theta}$$

w = 60 tonnes; x = 10 m

W = (13 000 + 60) = 13 060 tonnes

and θ = 2° 16′ so tan (2°16′) = 0.039 6

$\therefore$ By substitution, we get

$$\overline{GM} = \frac{60\times 10}{13\,060\times 0.0396} = 1.16 \text{ m}$$

Example 4.11. *In finding the metacentre of a ship of 98.1 MN (10 000 tonnes) displacement, a weight of 490.5 kN (50 tonnes) at a distance of 6 m from the longitudinal centre plane causes the ship to heel through an angle of 3°. What is the metacentric height? Hence find the angle of heel and its direction when the ship is going ahead and 2.833 6 MW (3 850 metric h.p.) is being transmitted to a single propeller shaft which is rotated at a speed of 3π rad/s (90 r.p.m).*

Solution

SI Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W\tan\theta}$$

w = 490.5 kN = 0.490 5 MN

W = 98.1 MN ; x = 6 m

θ = 3°, so tan 3° = 0.052 4

$\therefore$ By substitution, we get

$$\overline{GM} = \frac{0.4905\times 6}{98.1\times 0.0524} = 0.573 \text{ m}$$

If T is torque in N.m and ω is angular velocity in rad/s, then power transmitted in watts is

$$P = T\omega$$

or $$2.8336\times 10^6 = T\times 3\pi$$

$\therefore$ $$T = 300\,654.3 \text{ N.m}$$

If θ is the angle of heel, then we have

$$W\cdot\overline{GM}\tan\theta = T$$

or $$98.1\times 10^6\times 0.573\times\tan\theta = 300\,654.3$$

or $$\tan\theta = 0.005\,35$$

$\therefore$ $$\theta = 0°\,18.4'$$

Metric Units

The metacentric height is given by Eq. 4.3 as

$$\overline{GM} = \frac{wx}{W\tan\theta}$$

w = 50 tonnes ; x = 6 m

W = 10 000 tonnes

θ = 3°, so tan 3° = 0.0524

∴ By substitution, we get

$$\overline{GM} = \frac{50\times6}{10000\times0.0524} = 0.573 \text{ m}$$

Horse power transmitted

$$= \frac{2\pi NT}{75\times60}$$

where N is in r.p.m., and T is torque in kg(f)-m.

Thus by substitution, we get

$$3\,850 = \frac{2\pi\times90\times T}{75\times60}$$

$$\therefore \quad T = 30\,637 \text{ kg-m}$$

If θ is the angle of heel, then we have

$$W\;\overline{GM}\;\tan\theta = T$$

or $10\,000\times1000\times0.573\times\tan\theta = 30\,637$

$$\therefore \quad \tan\theta = \frac{30\,637}{10\,000\times1000\times0.573}$$

$$= 0.005\,35$$

$$\therefore \quad \theta = 0°\,18.4'$$

Example 4.12. *A merchant ship coming into port has a draught of 1.25 m. After unloading its cargo it has a draught of 1 m. Find the ratio of the periodic times before and after leaving the cargo if the breadth of the ship is 7 m and it is assumed that the centre of gravity remains at the water line.*

Solution

The periodic time is given by Eq. 4.7 as

$$T = 2\pi\left\{\frac{k_G^2}{gGM}\right\}^{1/2}$$

Let T_1 and T_2 be the periodic times for the ship before and after leaving the cargo respectively.

In the first case since the centre of gravity lies in the water line,

$$\overline{BG} = \frac{1.25}{2} = 0.625 \text{ m}$$

Now if l is the length of the ship at the water level then

$$I = \frac{1}{12}\times l\times(7)^3 \text{ m}^4$$

and

$$V = (l\times7\times1.25) \text{ m}^3$$

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

$$= \frac{\frac{1}{12}\times l\times(7)^3}{(l\times7\times1.25)} = 3.27 \text{ m}$$

Thus

$$\overline{GM} = \overline{BM}-\overline{GM}$$

$$= (3.27-0.625) = 2.645 \text{ m}$$

∴ By substitution, we get

$$T_1 = 2\pi\sqrt{\frac{k_G^2}{g\times2.645}}$$

Similarly after leaving the cargo

$$\overline{BG} = \frac{1}{2} = 0.5 \text{ m}$$

$$I = \frac{1}{12}\times l\times(7)^3 \text{ m}^4$$

$$V = (l\times7\times1) \text{ m}^3$$

$$\therefore \quad \overline{BM} = \frac{\frac{1}{12}\times l\times(7)^3}{(l\times7\times1)} = 4.08 \text{ m}$$

Thus

$$\overline{GM} = \overline{BM}-\overline{BG}$$

$$= (4.08-0.5) = 3.58 \text{ m}$$

∴ By substitution, we get

$$T_2 = 2\pi\sqrt{\frac{k_G^2}{g\times3.58}}$$

$$\therefore \quad \frac{T_1}{T_2} = \sqrt{\frac{3.58}{2.645}} = 1.16$$

Example 4.13. *A log of wood of square section 0.36 m × 0.36 m and specific gravity 0.8 floats in water. One edge is depressed and released causing the log to roll. Estimate the period of a roll.*

Solution

Let x be the depth of immersion and l be the length of the log, then

$$(l\times0.36\times0.36)\times9\,810\times0.8 = (x\times0.36\times l)\times9\,810$$

$$x = 0.288 \text{ m}$$

Thus height of the centre of buoyancy above the bottom of the log

$$= \left(\frac{0.288}{2}\right) = 0.144 \text{ m}$$

$$\therefore \quad \overline{BG} = (0.18 - 0.144) = 0.036 \text{ m}$$

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

$$I = \frac{1}{12} \times l \times (0.36)^3$$

$$V = (0.36 \times 0.288 \times l)$$

$$\therefore \quad \overline{BM} = \frac{\frac{1}{12} \times l \times (0.36)^3}{(0.36 \times 0.288 \times l)} = 0.037\,5 \text{ m}$$

$$\therefore \quad \overline{GM} = \overline{BM} - \overline{BG}$$

$$\therefore \quad = (0.0375 - 0.036) = 0.001\,5 \text{ m}$$

Time period of rolling is given by

$$T = 2\pi \sqrt{\frac{k_G^2}{g\,GM}}$$

$$k_G^2 = \frac{(0.36)^2}{12} = 0.010\,8 \text{ m}^2$$

$$\therefore \quad T = 2\pi \sqrt{\frac{0.0108}{9.81 \times 0.0015}} = 5.38 \text{ s.}$$

Example 4.14. *A wooden cone weighing 88 N floats with its apex downwards in a liquid of specific gravity 0.8. If the specific gravity of wood is 0.5, find what weight of a steel piece of specific gravity 7.8 suspended from the apex of the cone by a cord will just suffice to submerge the cone and what would then be the tension in the cord.*

Solution

Let W be the weight of the steel piece and V_1 and V_2 be the volumes of the cone and the steel piece respectively. Then we have

$$(W + 88) = (V_1 + V_2) \times 9\,810 \times 0.8 \quad \text{...(i)}$$

$$88 = 9\,810 \times 0.5 \times V_1 \quad \text{...(ii)}$$

and
$$W = 9\,810 \times 7.8 \times V_2 \quad \text{...(iii)}$$

By substituting the values of V1 and V2 from Eq. (ii) and (iii) in Eq. (i), we get

$$(W + 88) = \left(\frac{88}{9810 \times 0.5} + \frac{W}{9810 \times 7.8}\right) \times 9810 \times 0.8$$

or
$$W = 58.83 \text{ N}$$

The tension in the cord is equal to the weight of the steel piece in the liquid.

The weight of the steel piece in the liquid is equal to the weight of the steel piece in air minus the loss in the weight of the steel piece in the liquid.

Also the loss in the weight of the steel piece in the liquid is equal to the weight of the liquid displaced by it.

$$\text{The weight of the liquid displaced by the steel piece} = \left(\frac{W \times 0.8}{7.8}\right)$$

$$\therefore \text{ Tension in the cord} = \left[W - \left(\frac{W \times 0.8}{7.8}\right)\right]$$

$$= 58.83 \left(1 - \frac{0.8}{7.8}\right) = 52.8 \text{ N}$$

Example 4.15. *In Fig. Ex. 4.15 (a) is shown a float valve regulating the flow of oil of specific gravity 0.8 into a cistern. The spherical float is 150 mm in diameter. AOB is a weightless link carrying the float at one end and a valve at the other end which closes the pipe through which oil flows into the cistern. The link is mounted on a frictionless hinge at O and the angle AOB is 135°. The length OA is 0.2 m and the distance between the centre of the float and the hinge is 0.5 m. When the flow is stopped AO will be vertical. The valve is to be pressed on the seat with a force of 10 N to completely stop the flow into the cistern. It was observed that the flow of oil is stopped when the free surface of oil in the cistern is 0.35 m below the hinge. Determine the weight of the float.*

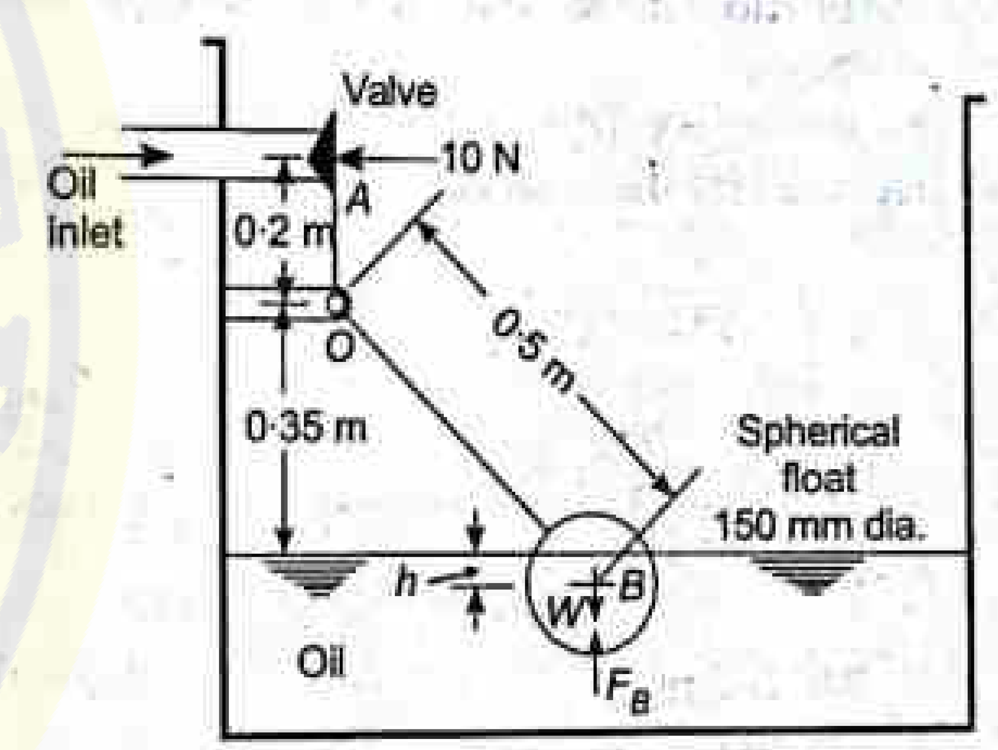


Figure Ex. 4.15 (a)

Solution

The depth of the centre of the spherical float below the oil surface in the cistern is

$$h = (0.5 \cos 45° - 0.35)$$

$$= 3.55 \times 10^{-3} \text{ m} = 3.55 \text{ mm}$$

The forces acting on the float are the weight of the float W acting vertically downwards at the centre of the float and the force of buoyancy F_B acting vertically upwards at the centre of buoyancy. Both these forces are acting along the same vertical line.

According to the principle of floatation the force of buoyancy F_B is equal to the weight of the oil displaced by the float.

Thus

$$F_B = (0.8 \times 9\,810)V = 7\,848V \text{ N}$$

where V is the volume of the oil displaced by the float in m³.

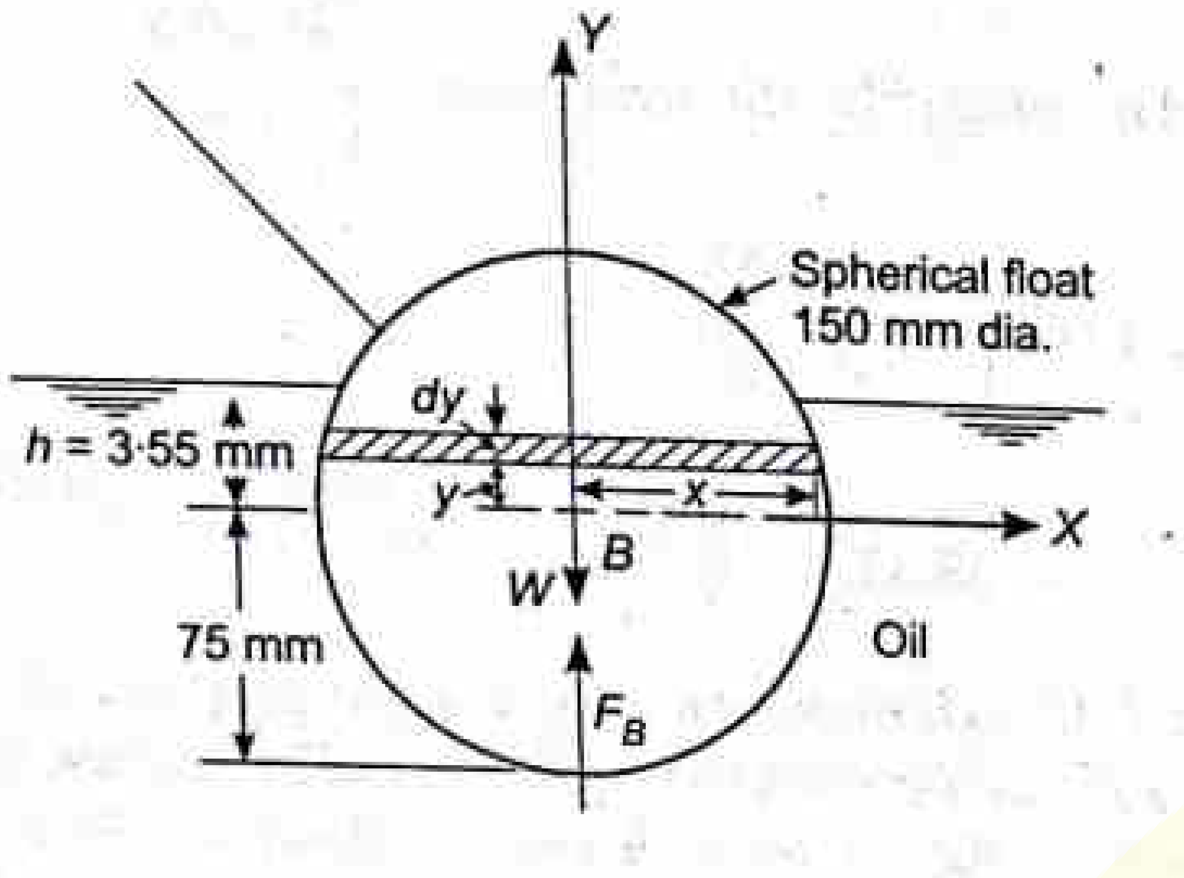


Figure Ex. 4.15 (b)

For determining the volume of oil displaced by the float consider on the surface of the float an element of radius x and thickness dy at a distance y from x-axis as shown in the Fig. Ex. 4.15 (b). The volume of the oil displaced by the portion of the float corresponding to the small element is

$$dV = \pi x^2 dy$$
$$= \pi [(75)^2 - y^2]\, dy$$

Thus the volume of the oil displaced by the float is

$$V = \pi \int_{-75}^{3.55} \left[(75)^2 - y^2\right] dy$$

$$= \pi \left[(75)^2 y - \frac{y^3}{3}\right]_{-75}^{3.55}$$

$$= 946\ 260 \text{ mm}^3$$
$$= 946.260 \times 10^{-6} \text{ m}^3$$

Note. In this case since the portion of the float lying between its centre and the oil surface is very small, it may approximately be considered to be of cylindrical shape with radius 75 mm and height 3.55 mm. The volume of the oil displaced by the float may then be obtained as

$$V = \left[\frac{2}{3}\pi(75)^3 + \pi(75)^2 \times 3.55\right]$$
$$= 946\ 307 \text{ mm}^3$$
$$= 946.307 \times 10^{-6} \text{ m}^3$$

which is very close to the value of V obtained above by the exact method.

The force of buoyancy is obtained as

$$F_B = 0.8 \times 9\ 810 \times 946.26 \times 10^{-6}$$
$$= 7.426 \text{ N}$$

Since the hinge is frictionless, the moment of the net vertical force acting on the float about the hinge O is transmitted to the valve A without any loss. The net vertical force acting on the float is

$$(F_B - W) = (7.426 - W)$$

Thus taking moments about hinge O, we get

$$(7.426 - W)\, 0.5 \cos 45° = 10 \times 0.2$$
$$\therefore \quad W = 1.769 \text{ N}$$

SUMMARY OF MAIN POINTS

1. When a body is immersed in a fluid, either wholly or partially, an upward force is exerted by the fluid on the body which tends to lift (or buoy) up the body. This tendency for an immersed body to be lifted up in the fluid, due to an upward force opposite to the action of gravity is known as 'buoyancy'.
2. The force tending to lift up a body immersed in a fluid either wholly or partially is known as 'buoyant force' or 'force of buoyancy' or 'upthrust'.
3. The point of application of the force of buoyancy on the body is known as 'centre of buoyancy'.
4. According to Archimedes' principle the buoyant force is equal to the weight of the fluid displaced by the body.
5. The principle of floatation states that the weight of a body floating in a fluid is equal to the buoyant force which in turn is equal to the weight of the fluid displaced by the body.
6. For a body floating in a fluid the centre of buoyancy coincides with the centroid of the volume of the fluid displaced by the body.
7. 'Metacentre' is defined as the point of intersection between the axis of a floating body and a vertical line passing through the new centre of buoyancy obtained when the floating body is slightly tilted or it undergoes a small angular displacement (or angle of heel) θ
8. The distance between the centre of gravity G and metacentre M of a floating body (i.e. GM) as $\theta \to 0$, is known as 'metacentric height'.
9. Conditions of equilibrium of a floating body and a wholly submerged body are as indicated below:

Conditions of equilibrium	*Floating body*	*Wholly submerged body*
(i) Stable equilibrium	M is above G	B is above G
(ii) Unstable equilibrium	M is below G	B is below G
(iii) Neutral equilibrium	M and G coincide	B and G coincide

10. The value of the metacentric height determined experimentally is given as

$$GM = \frac{wx}{W \tan\theta}$$

where
w = movable weight;
x = distance through which w is moved;
W = weight of the ship or floating body including w; and
θ = angle through which the ship or floating body is tilted.

Further

$$\tan\theta = \frac{d}{l}$$

where
l = length of the plumbline or pendulum; and
d = distnace moved by the plumbline or pendulum on the horizontal scale.

11. The metacentric height determined by analytical method is given as

$$GM = \frac{I}{V} - BG$$

where
I = moment of inertia of the cross-sectional area of the ship or floating body about the longitudinal axis;
V = volume of liquid displaced by the ship or floating body; and
BG = distance between centre of buoyancy and centre of gravity of the ship or floating body.

12. A ship or a floating body may have two types of oscillatory motions, viz., 'rolling' and 'pitching.' The oscillatory motion of a ship or a floating body about its longitudinal axis is designated as rolling, and that about its transverse axis is designated as pitching.

13. The time period of transverse oscillation (i.e, oscillation about the longitudinal axis) or rolling of a ship or a floating body is given by

$$T = 2\pi\left[\frac{k_G^2}{g\overline{GM}}\right]^{\frac{1}{2}}$$

where

T = time period of transverse oscillation;
k_G = radius of gyration of the floating body about its centre of gravity;
$\overline{GM}$ = metacentric height; and
g = acceleration due to gravity.

PROBLEMS

4.1 Explain the terms: Force of buoyancy and centre of buoyancy.

4.2 Explain the terms: Metacentre and metacentric height.

4.3 Explain the three states of equilibrium of a floating body in terms of the magnitude and direction of righting couple acting on the body as a result of slight angular displacement.

4.4 What is meant by stability of a floating body? Explain the stability of a floating body with reference to its metacentric height. Give neat sketches.

4.5 Define the terms: Stable, unstable and neutral equilibrium.

4.6 Describe briefly the experimental method of determination of the metacentric height of a floating object.

4.7 An iceberg weighing 8 976 N/m³ [915 kg (f)/m³] floats in the ocean with a volume of 600 m³ above the surface. Determine the total volume of the iceberg if specific weight of ocean water is 10 055 N/m³ [1 025 kg (f)/m³].
[*Ans.* 5 591 m³]

4.8 An empty balloon and its equipment weighs 441.45 N [45 kg (f)]. When inflated with gas weighing 5.415 N/m³ [0.552 kg(f)/m³], the balloon is spherical and 7 m in diameter. What is the maximum weight of cargo that the balloon can lift, assuming air to weigh 12.066 N/m³ [1.23 kg (f)/m³].
[*Ans.* 753.03 N [76.76 kg (f)]]

4.9 A cylinder has a diameter 0.3 m and a specific gravity of 0.75. What is the maximum permissible length in order that it may float in water with its axis vertical? [*Ans.* 0.245 m]

4.10 A buoy carrying a beacon light has the upper portion cylindrical 2.5 m diameter and 2 m deep. The lower portion which is curved displaces a volume of 0.4 m³ and its centre of buoyancy is situated 2.5 m below the top of the cylinder. The centre of gravity of the whole buoy and beacon is situated 1.5 m below the top of the cylinder and the total displacement is 19.62 kN (2 tonnes). Find the metacentric height if the specific weight of sea water is 10 055 N/m³ [1 025 kg (f)/m³] [*Ans.* 0.505 m]

4.11 A rectangular pontoon weighing 1 716.75 kN (175 tonnes) has a length of 20 m. The centre of gravity is 0.3 m above the centre of the cross-section and the metacentric height is to be 1.25 m when the angle of heel is 9°. The free board must not be less than 0.6 m when the pontoon is vertical. Find the breadth and the height of the pontoon, if it is floating in fresh water.
[*Ans.* 5.792 m, 2.11 m]

4.12 A ship is 80 m long and 12 m broad and has a displacement of 14 715 kN (1500 tonnes). When a weight of 490.5 kN is moved 5 m across the deck it inclines the ship 6°. The moment of inertia of the section of the ship at water plane about its fore-and-aft axis is 60 per cent of the moment of inertia of the circumscribing rectangle. The centre of buoyancy is 2.5 m below the water line. Find the position of the metacentre and the centre of gravity. Specific weight of sea water is 10 055 N/m³ [1025 kg (f)/m³].
[*Ans.* 2.224 m and 0.638 m above water line]

4.13 A solid cylinder is to be 0.25 m in diameter. The base of axial length 25 mm is to be of metal which has a specific gravity 7, and the remainder of material which has a specific gravity 0.5. Find the maximum overall length of the cylinder in order that it may float in water in the stable equilibrium with its axis vertical.
[*Ans.* 0.801 m]

4.14 A ship has a displacement of 49 050 kN. The second moment of area of the water line section about a fore- and-aft axis is 12 × 104 m⁴ and the centre of buoyancy is 2.8 m below the centre of gravity. The radius of gyration is 4 m. Calculate the time period of oscillation. Take the weight of sea water as 10 055 N/m³
[*Ans.* 1025 kg (f)/m³]. [1.72 s]

4.15 Find the value of K_G for a ship which has a period of rolling of 20s. The displacement is 98 100 kN (10000 tonnes), the second moment of the load-water-plane above its fore-and-aft axis is 30 × 102 m⁴ and the centre of buoyancy is 3.25 m below the centre of gravity. Sea water weighs 10 055 N/m³ [1 025 kg(f)/m³].
[*Ans.* 4.174 m]

4.16 A solid conical float of wood weighing 7 553.7 N/m³ [770 kg(f)/m³] is to float in a liquid weighing 9417.6 N/m³ [960 kg(f)/m³]. Find the least apex angle in order that it may float with apex down and its axis vertical. [*Ans.* 30° 50′]

4.17 A hollow cylindrical of diameter 2 m and length 3 m weighs 29.43 kN [3 tonnes] and has its centre of gravity at the mid point of the longitudinal axis. Show that it will not float in sea water with its longitudinal axis vertical.

Neglecting the thickness of the plates, find to what depth the inside of the buoy should be filled with concrete, of specific weight 22 563 N/m³ [2 300 kg/m³], to give a depth of immersion of 2.5 m and what is then the metacentric height? Sea water weighs 10 055 N/m³ [1 025 kg(f)/m³].
[*Ans.* 0.6984 m; 0.572 m]

4.18 A conical buoy floating with its apex pointing downward is 3.5 m high and 2 m diameter. Calculate its weight if it is just stable when floating in sea water weighing 10 055 N/m³ [1025 kg(f)/m³].
[*Ans.* 29.127 kN [2 969 kg(f)]]

4.19 A floating buoy in the Bombay harbour is to be assisted in floating upright by a submerged weight of concrete attached to the bottom of the buoy. How many m³ of concrete weighing 22 759 N/m³ [2 320 kg (f)/m³] must be provided to get a net downward pull of 2217 N [226 kg(f)] from the weight? Explain the principle involved.
[*Ans.* 0.1745 m³]

4.20 A rectangular pontoon 6 m by 3 m in plan, floating in water has a depth of immersion of 0.9 m and is subjected to a torque of 7848 N.m [800kg(f)-m] about the longitudinal axis. If the centre of gravity is 0.7 m up from the bottom, estimate the angle of heel. [*Ans.* 4°50′]

4.21 A solid cylinder 1 m diameter and 0.75 m high is of uniform specific gravity 0.85. Calculate the periodic time of small oscillations when the cylinder floats with its axis vertical in still water.

[Hint. Take $k_G^2 = \left(\frac{l^2}{12}+\frac{r^2}{4}\right)$; where l is the length and r is the radius of the cylinder.]
[*Ans.* 3.25 s]

Fundamentals of Fluid Flow

Chapter 6

6.1 INTRODUCTION

In the preceding chapters the fluids at rest have been considered, in which case the only fluid property of significance is the specific weight of the fluids. On the other hand when a fluid is in motion various other fluid properties become significant. As such the nature of flow of a real fluid is complex and not always subject to exact mathematical analysis, and often recourse to experimentation is required. However, in some cases the mathematical analysis of problems of fluid flow is possible if some simplifying assumptions are made.

The science which deals with the geometry of motion of fluids without reference to the forces causing the motion is known as *hydrokinematics* or simply *kinematics*. Thus kinematics involves merely the description of the motion of fluids in terms of space-time relationship. On the other hand the science which deals with the action of the forces in producing or changing motion of fluids is known as *hydrokinetics* or simply *kinetics*. Obviously the study of fluids in motion involves the consideration of both the kinematics and the kinetics.

A fluid unlike solid, is composed of different particles, which move at different velocities and may be subject to different accelerations. Moreover, the velocity and acceleration of a fluid particle may change both with respect to time and space. Therefore in the study of fluid flow it is necessary to observe the motion of fluid particles at various points in space and at successive instants of time.

There are in general two methods by which the motion of a fluid may be described. These are the *Lagrangian* method and the *Eulerian* method.

In the Lagrangian method any individual fluid particle is selected, which is pursued throughout its course of motion and the observation is made about the behaviour of this particle during its course of motion through space. In the Eulerian method any point in the space occupied by the fluid is selected and observation is made of whatever changes of velocity, density and pressure which take place at that point. Out of these two methods the Eulerian method is commonly adopted in fluid mechanics and therefore the same is used in the following analysis.

6.2 VELOCITY OF FLUID PARTICLES

The motion of a fluid like that of solid is described quantitatively in terms of the characteristic known as velocity. However, in case of solids it is generally sufficient to measure the velocity of the body as a whole, but in the case of fluids the motion of fluid may be quite different at different points of observation.

Therefore the velocity V at any point of fluid mass is expressed as the ratio between the displacement of a fluid element along its path and the corresponding increment of time as the later approaches zero. As shown in Fig. 6.1, a particular point in the space occupied by a fluid in motion is selected, which may be denoted by the coordinates (x, y, z). Since this point is fixed in space, the coordinates x, y, z and the time t are independent variables. At this point if ds is the distance travelled by a fluid particle in time dt then the velocity V of the fluid particle at this point may be expressed as

$$V = \lim_{dt \to 0} \frac{ds}{dt}$$

The velocity is a vector quantity and hence it has magnitude as well as direction. Therefore the velocity V at any point in the fluid can be resolved into three components u, v and w along three mutually perpendicular directions x, y and z respectively. Each of these components can also be expressed as the limiting rate of displacement in the corresponding direction. Thus if dx, dy and dz are the components of the displacement ds in x, y and z directions respectively, then

$$u = \lim_{dt \to 0} \frac{dx}{dt}, \quad v = \lim_{dt \to 0} \frac{dy}{dt} \quad \text{and} \quad w = \lim_{dt \to 0} \frac{dz}{dt}$$

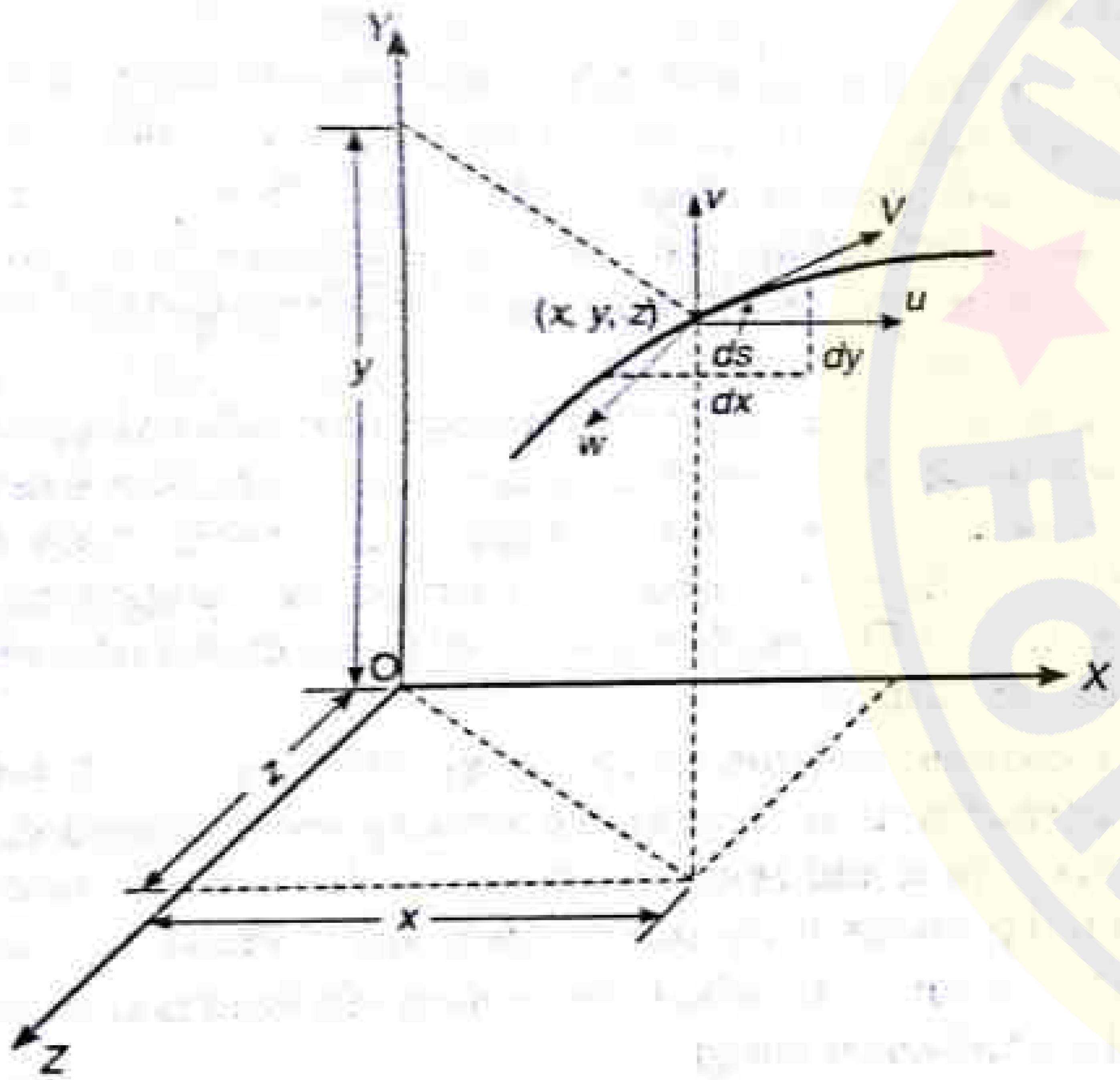


Figure 6.1 *Velocity at a point in fluid in motion*

Since the velocity V at any point in a flowing mass of fluid in general depends on x, y and z, i.e., the coordinate position of the point under consideration and the time t. Therefore the components u, v and w will also in general depend on x, y, z and t. Hence the velocity V and its components u, v and w may be expressed in terms of the following functional relationships

$$\left.\begin{aligned} V &= f_1(x,y,z,t) \\ u &= f_2(x,y,z,t) \\ v &= f_3(x,y,z,t) \\ w &= f_4(x,y,z,t) \end{aligned}\right\} \qquad \text{...(6.1)}$$

In vector notation velocity V may be expressed in terms of its components as

$$V = iu + jv + kw$$

where i, j, k are the unit vectors parallel to the x, y, z axis respectively.

For a particular point since the coordinates x, y and z may be regarded constant, the above expressions will indicate how u, v and w vary as the time t varies. On the other hand, if time t is regarded constant then since (x, y, z) may represent any point in the flowing mass of fluid the above expression will indicate how u, v and w vary at different points at a particular instant of time under consideration.

6.3 TYPES OF FLUID FLOW

According to different considerations fluid flows may be classified in several ways as indicated below:

(i) Steady flow and Unsteady flow.
(ii) Uniform flow and Non-uniform flow.
(iii) One-dimensional flow, Two dimensional flow and Three dimensional flow.
(iv) Rotational flow and Irrotational flow
(v) Laminar flow and Turbulent flow.

Steady Flow. Fluid flow is said to be steady if at any point in the flowing fluid various characteristics such as velocity, pressure, density, temperature etc., which describe the behaviour of the fluid in motion, do not change with time. In other words a steady flow may be defined as that in which at any point in the flowing fluid various characteristics which describe the behaviour of the fluid in motion are independent of time. However, these characteristics may be different at different points in the flowing fluid. Thus the steady flow may be expressed mathematically by the following expression at any point in the flowing fluid.

$$\left(\frac{\partial u}{\partial t}\right) = 0; \left(\frac{\partial v}{\partial t}\right) = 0; \left(\frac{\partial w}{\partial t}\right) = 0; \left(\frac{\partial p}{\partial t}\right) = 0; \left(\frac{\partial \rho}{\partial t}\right) = 0$$

Unsteady Flow. Fluid flow is said to be unsteady if at any point in the flowing fluid any one or all the characteristics which describe the behaviour of the fluid in motion change with time. Thus a flow of fluid is unsteady, if at any point in the flowing fluid

$$\left(\frac{\partial V}{\partial t}\right) \neq 0 \text{ ; and or} \left(\frac{\partial p}{\partial t}\right) \neq 0 \text{ etc.}$$

It may however be indicated that in the case of vector quantities such as velocity of flow etc., even the change in the direction of such quantities with respect to time at any point in the flowing fluid may also make the flow unsteady.

Obviously a steady flow is simpler to analyse than unsteady flow. Moreover, most of the practical problems of engineering involve only steady flow conditions. Therefore in this book the discussion has been mainly confined to the steady flow conditions only.

Uniform Flow. When the velocity of flow of fluid does not change, both in magnitude and direction, from point to point in the flowing fluid, for any given instant of time, the flow is said to be uniform. In the mathematical form a uniform flow may therefore be expressed as

$$\left(\frac{\partial V}{\partial s}\right) = 0$$

where time is held constant and s represents any direction of displacement of the fluid elements. The above expression states that there is no change in the velocity vector in any direction throughout the flowing fluid at any instant of time. For example flow of liquids under pressure through long pipe lines of constant diameter is uniform flow.

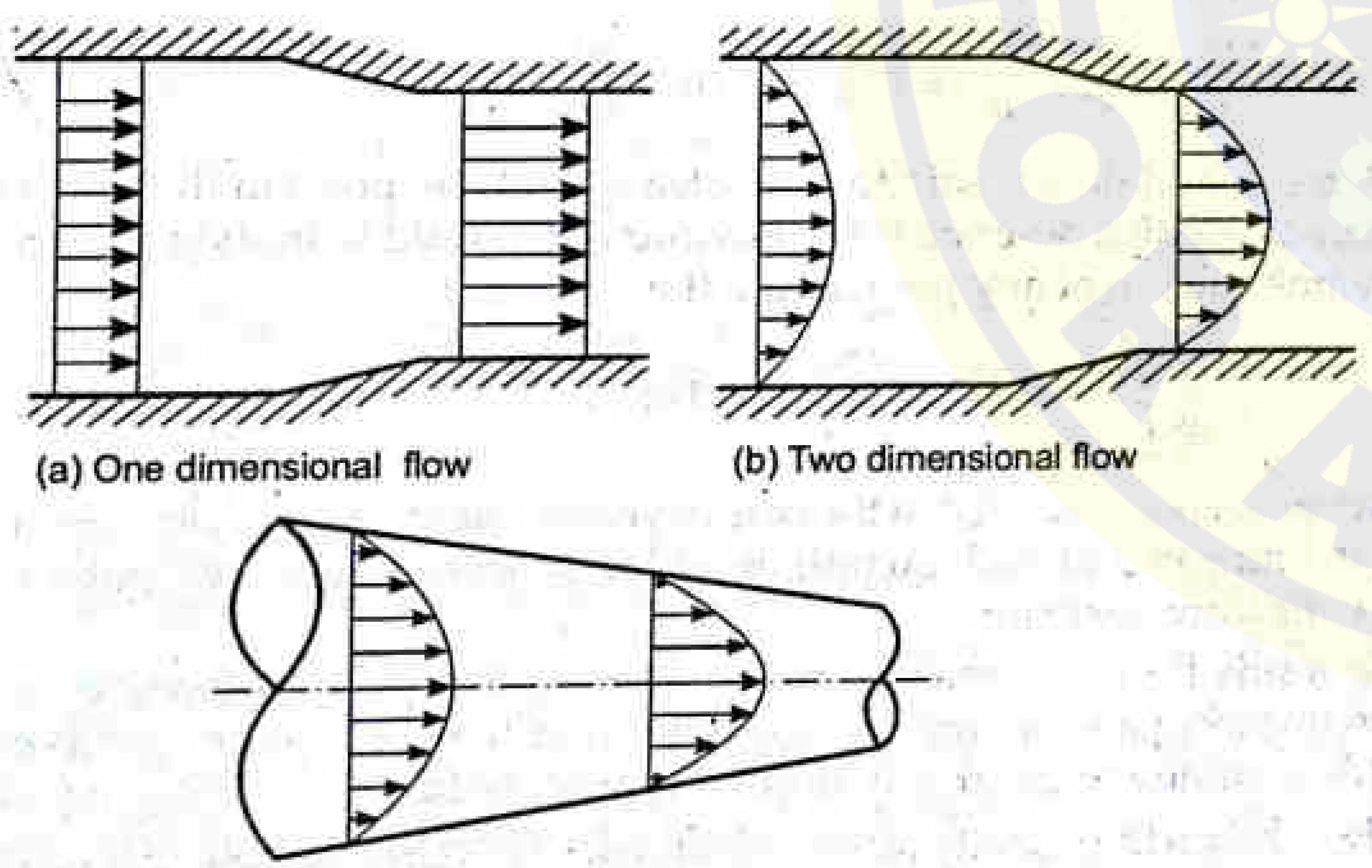
Non-uniform Flow. If the velocity of flow of fluid changes from point to point in the flowing fluid at any instant, the flow is said to be non-uniform. In the mathematical form a non-uniform flow may be expressed as

$$\frac{V}{s} \neq 0$$

For example flow of liquids under pressure through long pipelines of varying diameters is non-uniform flow.

All these types of flows can exist independent of each other so that any of the four types of combinations of flows is possible, viz., (a) steady-uniform flow; (b) steady-non-uniform flow; (c) unsteady uniform flow; and (d) unsteady-non-uniform flow. Some of the common examples of these combinations of flows are: flow of liquid through a long pipe of constant diameter at a constant rate is *steady uniform flow*; flow of liquid through a long pipe line of constant diameter, at either increasing or decreasing rate is *unsteady-uniform flow*; flow of liquid through a tapering pipe at a constant rate is *steady-non-uniform flow* and flow through a tapering pipe at either increasing or decreasing rate is *unsteady-non-uniform flow*.

One-dimensional, Two-dimensional and Three-dimensional Flows. The various characteristics of flowing fluid such as velocity, pressure, density, temperature etc., are in general the functions of space and time i.e., these may vary with the coordinates of any point x, y and z and time t. Such a flow is known as a *three-dimensional flow*. If any of these characteristics of flowing fluid does not vary with respect to time, then it will be a steady three-dimensional flow.



Figuree 6.2 *One, Two and Three dimensional flows*

When the various characteristics of flowing fluid are the functions of only any two of the three coordinate directions, and time t, i.e., these may not vary in any one of the directions, then the flow is known as *two-dimensional flow*. For example, if the characteristics of flowing fluid do not vary in the coordinate direction Z, then it will be a two-dimensional flow having flow conditions indentical in the various planes perpendicular to the Z-axis. Again it will be a steady two-dimensional flow if the characteristics of flowing fluid do not vary with respect to time.

When the various characteristics of flowing fluid are the functions of only one of the three coordinate directions and time t, i.e., these may vary only in one direction, then the flow is known as *one-dimensional flow*. Similarly, it will be a steady one dimensional flow if the characteristics of flowing fluid do not vary with respect to time. Considering one of the characteristics of flowing mass of fluid, say velocity of flow V, the following expressions may be written which clearly exhibit the difference between these three types of flows:

Types of Flow	*Unsteady*	*Steady*
Three-dimensional	$V = f(x, y, z, t)$	$V = f(x, y, z)$
Two-dimensional	$V = f(x, y, t)$	$V = f(x, y)$
One-dimensional	$V = f(x, t)$	$V = f(x)$

Similar expressions can also be written for various other characteristics of flowing fluid for the three different types of flows. Figure 6.2 shows the examples of one-dimensional, two-dimensional and three-dimensional flows.

Obviously the problems involving three-dimensional flow are quite complicated because of the variation of the characteristics of flowing fluid with x, y and z, and t. Even in steady three-dimensional flow the variation of the characteristics of flowing fluid with x, y and z render the problems fairly complicated to solve. In this respect the assumption of either two-dimensional flow or one-dimensional flow, considerably simplifies the problem making it amenable to analytical solution.

Rotational Flow. A flow is said to be rotational if the fluid particles while moving in the direction of flow rotate about their mass centres. In Chapter 5, the liquid in the rotating tanks illustrates rotational flow where the velocity of each particle varies directly as the distance from the centre of rotation.

Irrotational Flow. A flow is said to be irrotational if the fluid particles while moving in the direction of flow do not rotate about their mass centres. It may however be stated that a true irrotational flow exists only in the case of flow of an ideal fluid for which no tangential or shear stresses occur. But the flow of practical fluids, may also be assumed to be irrotational if the viscosity of the fluid has little significance.

Laminar Flow. A flow is said to be laminar when the various fluid particles move in layers (or laminae) with one layer of fluid sliding smoothly over an adjacent layer. Thus in the development of a laminar flow, the viscosity of the flowing fluid plays a significant role. As such the flow of a very viscous fluid may in general be treated as laminar flow.

Turbulent Flow. A fluid motion is said to be turbulent when the fluid particles move in an entirely haphazard or disorderly manner, that results in a rapid and continuous mixing of the fluid leading to momentum transfer as flow occurs. In such a flow eddies or vortices of different sizes and shapes are present which move over large distances. These eddies and their random movement give rise to fluctuations in the velocity and pressure at any point in the flow field, which are necessarily the functions of time. Thus at any point in a turbulent flow the velocity and pressure are the functions of time thereby rendering such a flow as unsteady. However, temporal mean values of the velocity and pressure considered over sufficiently long time do not change with time. Hence a turbulent flow may be considered to be steady flow in terms of temporal mean values of the velocity and pressure.

The occurrence of turbulent flow is more frequent than that of laminar flow. Flow in natural streams, artificial channels, water supply pipes, sewers etc., are a few examples of turbulent flow.

6.4 DESCRIPTION OF THE FLOW PATTERN

The flow pattern may be described by means of streamlines, stream-tubes, path lines and streak-lines.

Streamline. A *streamline* is an imaginary curve drawn through a flowing fluid in such a way that the tangent to it at any point gives the direction of the velocity of flow at that point. Since a fluid is composed of fluid particles, the pattern of flow of fluid may be represented by a series of stream-lines, obtained by drawing a series of curves through the flowing fluid such that the velocity vector at any point is tangential to the curves. Figure 6.3 shows some of the streamlines for a flow pattern in the xy plane in which a streamline passing through a point $P(x, y)$ is tangential to the velocity vector V at P. If u and v are the components of V along x and y directions, then

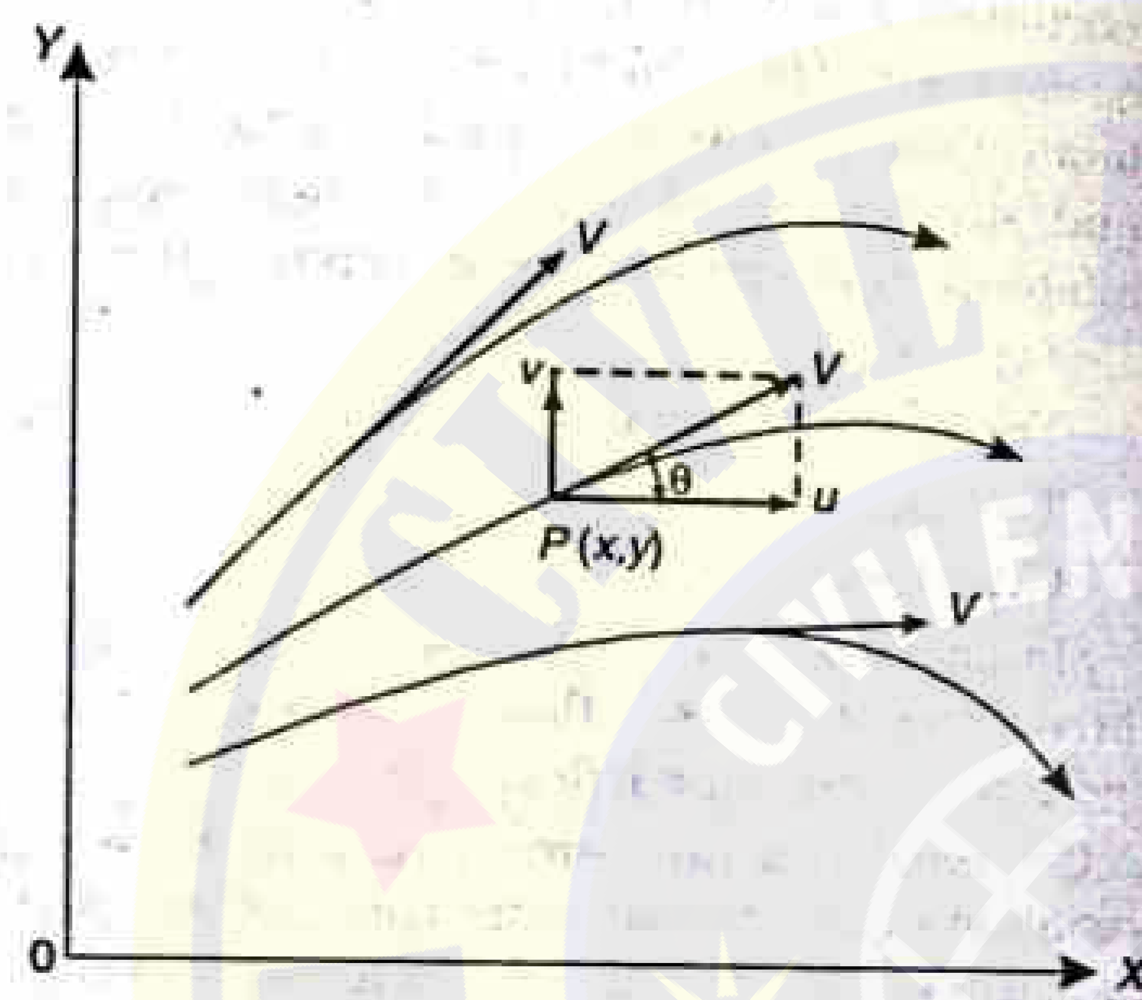


Figure 6.3 *Streamlines for a flow pattern in plan*

$$\frac{v}{u} = \tan\theta = \frac{dy}{dx}$$

where dy and dx are the y and x components of the differential displacement ds along the streamline in the immediate vicinity of P. Therefore the differential equation for streamlines in the xy plane may be written as

$$\frac{dx}{u} = \frac{dy}{v}\ ;\text{ or } (u\,dy - v\,dx) = 0 \qquad \text{...(6.2)}$$

A general differential equation for streamlines in space for three-dimensional flow may however be obtained in the same manner as

$$\frac{dx}{u} = \frac{dy}{v} = \frac{dz}{w} \qquad \text{...(6.3)}$$

Since a streamline is everywhere tangent to the velocity vector, there can be no component of the velocity at right angles to the streamline and hence there can be no flow of fluid across any streamline. Further in steady flow, since there is no change in direction of the velocity vector at any point, the flow pattern is not changing. But in unsteady flow, since the direction of the velocity vector at any point may change with time, the flow pattern also changes with time. Therefore for a steady flow the streamline pattern remains the same at different times, but for an unsteady flow the streamline pattern may change from time to time, and hence in the case of unsteady flow the streamlines are to be regarded as instantaneous streamlines representing an instantaneous flow pattern only.

Stream-tube. A *stream-tube* is a tube imagined to be formed by a group of streamlines passing through a small closed curve, which may or may not be circular, as shown in Fig. 6.4. Since the stream-tube is bounded on all sides by streamlines and since the velocity has no component normal to a streamline, there can be no flow across the bounding surface of a stream-tube. Therefore a fluid may enter or leave a stream-tube only at its ends. A stream-tube with a cross-sectional area small enough for the variation of velocity over it to be negligible is sometimes termed as a *stream filament*. The concept of stream-tubes is quite useful in analysing several fluid flow problems, since the entire flow field may be divided into a large number of stream-tubes, thus yielding a clear picture of the actual pattern of flow. However, only in steady flow a stream-tube is fixed in space.

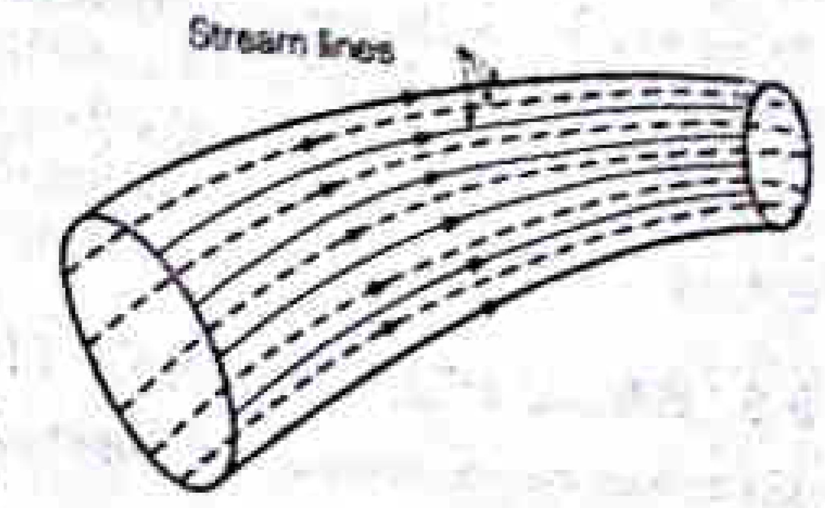


Figure 6.4 *Stream tube*

Path-line. A *path-line* may be defined as the line traced by a single fluid particle as it moves over a period of time. Thus a path-line will show the direction of velocity of the same fluid particle at successive instants of time. As indicated earlier a streamline on the other hand shows the direction of velocity of a number of fluid particles at the same instant of time. A fluid particle always moves tangent to the streamline, and in the case of steady flow since there is no change in direction of the velocity vector at any point with time, the streamline is fixed in space. Therefore in steady flow the path-lines and streamlines are identical. However, in unsteady flow since the direction of velocity vector at any point may change with time, streamline may shift in space from instant to instant. A particle then follows one streamline at one instant and another at the next instant and so on, so that the path of the particle may have no resemblance to any given instantaneous streamline. In other words, in unsteady flow path-lines and streamlines are different.

Streak-line. In experimental work often a colour or dye or some other substance (such as smoke in case of gases) is injected into the flowing fluid, in order to trace the motion of the fluid particles. The resulting trail of colour is known as a *streak-line* or *filament line*. Thus a streak-line may be defined as a line that is traced by a fluid particle passing through a fixed point in a flow field.

In steady flow, since there is no change in the flow pattern, a streak-line is same as a streamline and the path-line of a particle. In other words, in steady flow a streak-line, a streamline and a path-line are all indentical. In unsteady flow a streak-line at any instant is the locus of end points of particle paths

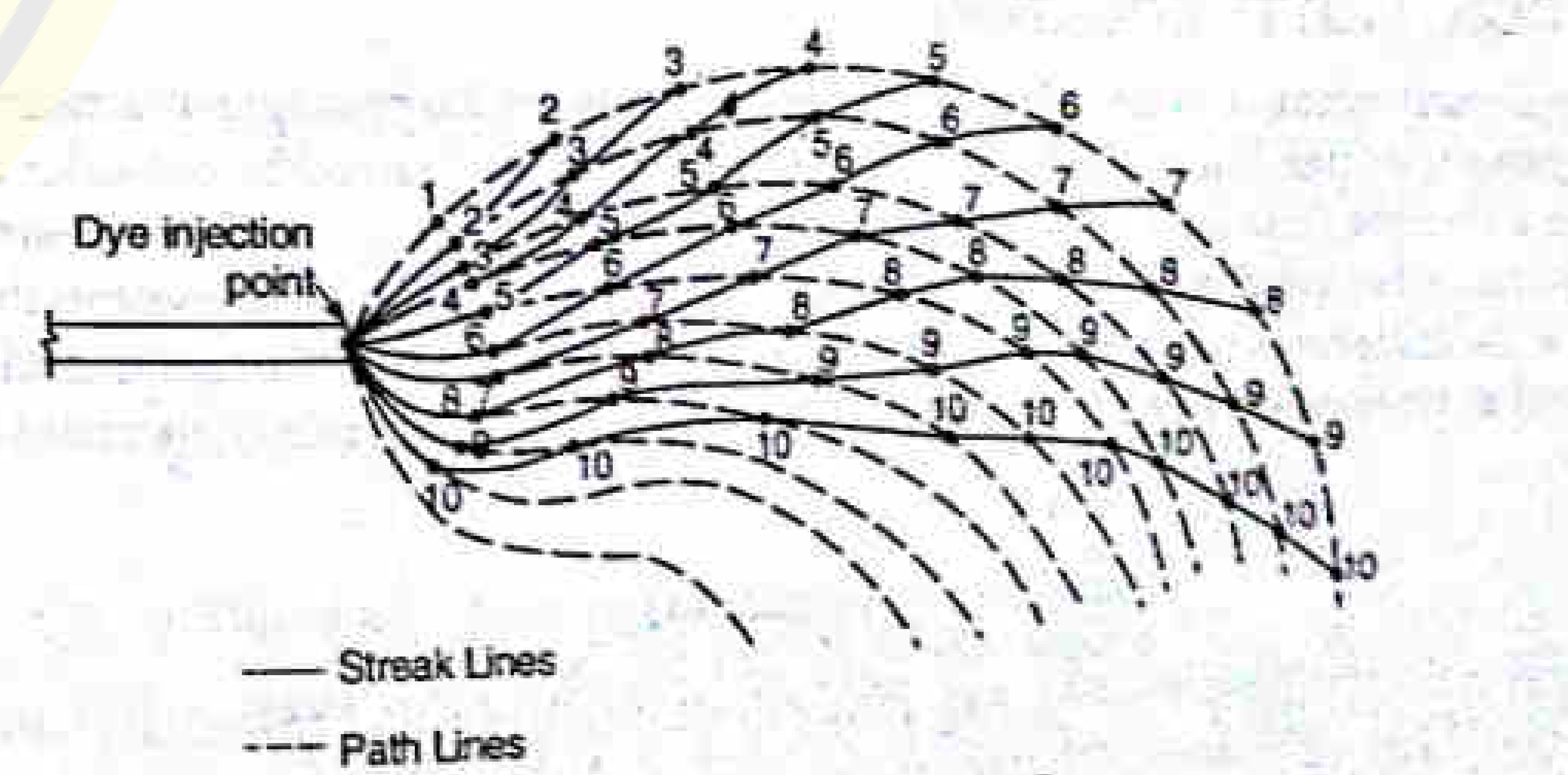


Figure 6.5 *Streak–lines in unsteady flow*

(or path-lines) that started at the instant the particle passed through the injection point. Figure 6.5 indicates streak-lines in a possible unsteady flow situation. It is assumed that a dye is injected into the fluid for a 10-second period. The position of the particle that was at the dye jet for each second $t = 0, 1, 2, 3, 4......10$ is plotted for each subsequent second in the 10-second period. The dots represent the position of each of these particles at the time indicated, the dotted lines represent the paths of particles (i.e., path-lines) and the solid lines represent the position of the streak-line at each second of time. At $t = 1$, it is practically the path of the particle; at $t = 2$ it is the line through the points marked 2 and moves so that it takes up the new positions marked by the time at the end of each one second interval, However, only one streak-line is visible at a given instant.

6.5 BASIC PRINCIPLES OF FLUID FLOW

Alike solid mechanics there are three basic principles used in the analysis of the problems of fluid in motion as noted below :

(i) Principle of conservation of mass.
(ii) Principle of conservation of energy.
(iii) Principle of conservation of momentum.

The *principle of conservation of mass* states that mass can neither be created nor destroyed. On the basis of this principle the continuity equation is derived.

The *principle of conservation of energy* states that energy can neither be created nor destroyed. On the basis of this principle the energy equation is derived.

The *principle of conservation of momentum* or *impulse momentum principle* states that the impulse of the resultant force, or the product of the force and time increment during which it acts, is equal to the change in the momentum of the body. On the basis of this principle the momentum equation is derived.

In applying these principles usually control volume approach is adopted, in which a definite volume with fixed boundary shape is chosen in space along the fluid flow passage. This definite volume is called the *control volume* and the boundary of this volume is known as the *control surface.* The boundaries of the control volume may be extended upto such an extent that it includes the portion of the flow passage which is to be studied. In fact the use of the Eulerian method for describing fluid motion, implies the use of the control volume approach without specifying it.

In this chapter only the continuity equation is derived. The derivation of energy and momentum equations and their application are dealt with in the subsequent chapters.

6.6 CONTINUTTY EQUATION

The continuity equation is actually mathematical statement of the principle of conservation of mass. A most general expression on the basis of this principle may be obtained by considering a fixed region within a flowing fluid as shown in Fig. 6.6. Since fluid is neither created nor destroyed within this region it may be stated that the rate of increase of the fluid mass contained within the region must be equal to the difference between the rate at which the fluid mass enters the region and the rate at which the fluid mass leaves the region. However if the flow is steady, the rate of increase of the fluid mass

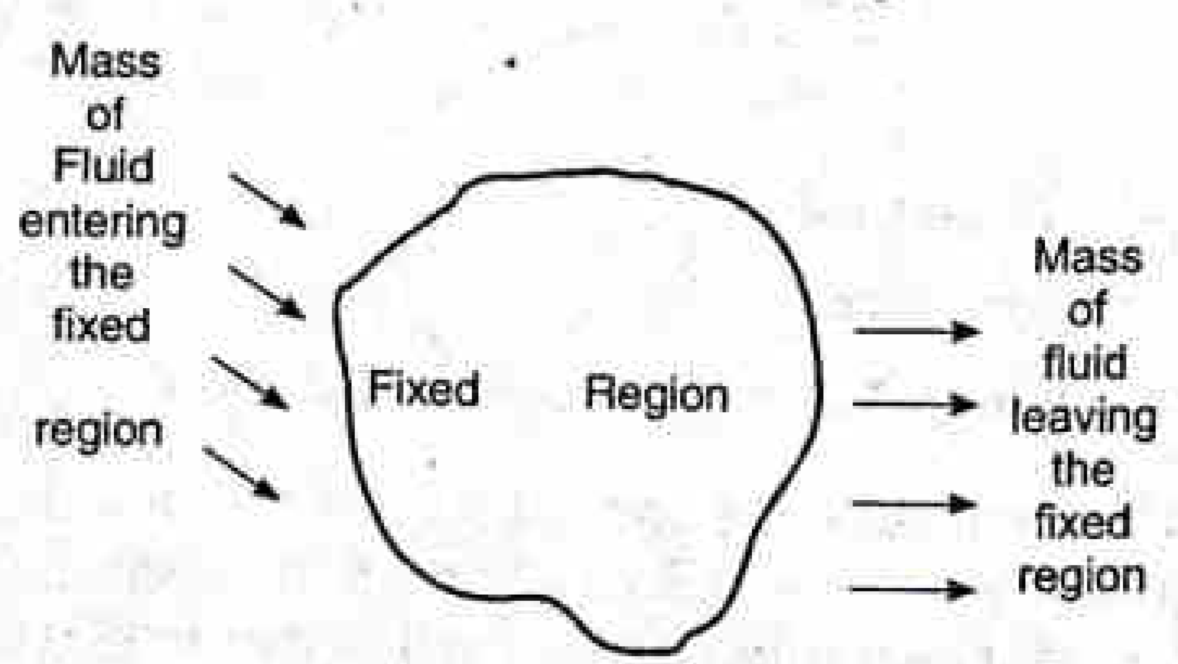


Figure 6.6 *Diagrammatic representation of the principle of conservation of mass*

within the region is equal to zero, then the rate at which the fluid mass enters the region is equal to the rate at which the fluid mass leaves the region. This relation is used to derive a general equation of continuity for a three dimensional steady or unsteady flow, as indicated below.

Continuity Equation in Cartesian Coordinates. Consider an elementary rectangular parallelopiped with sides of length δx, δy and δz as shown in Fig. 6.7. Let the centre of the parallelopiped be at a point $P(x, y, z)$ where the velocity components in the x, y, z direction are u, v and w respectively and ρ be the mass density of the fluid.

The mass of fluid passing per unit time through the face of area $\delta y\ \delta z$ normal to the x-axis through point P, is

$$(\rho u\ \delta y\ \delta z)$$

Then the mass of fluid flowing per unit time into the parallelopiped through the face $ABCD$ is

$$\left[(\rho u\ \delta y\ \delta z)+\frac{\partial}{\partial x}(\rho u\ \delta y\ \delta z)\left(-\frac{\delta x}{2}\right)\right]$$

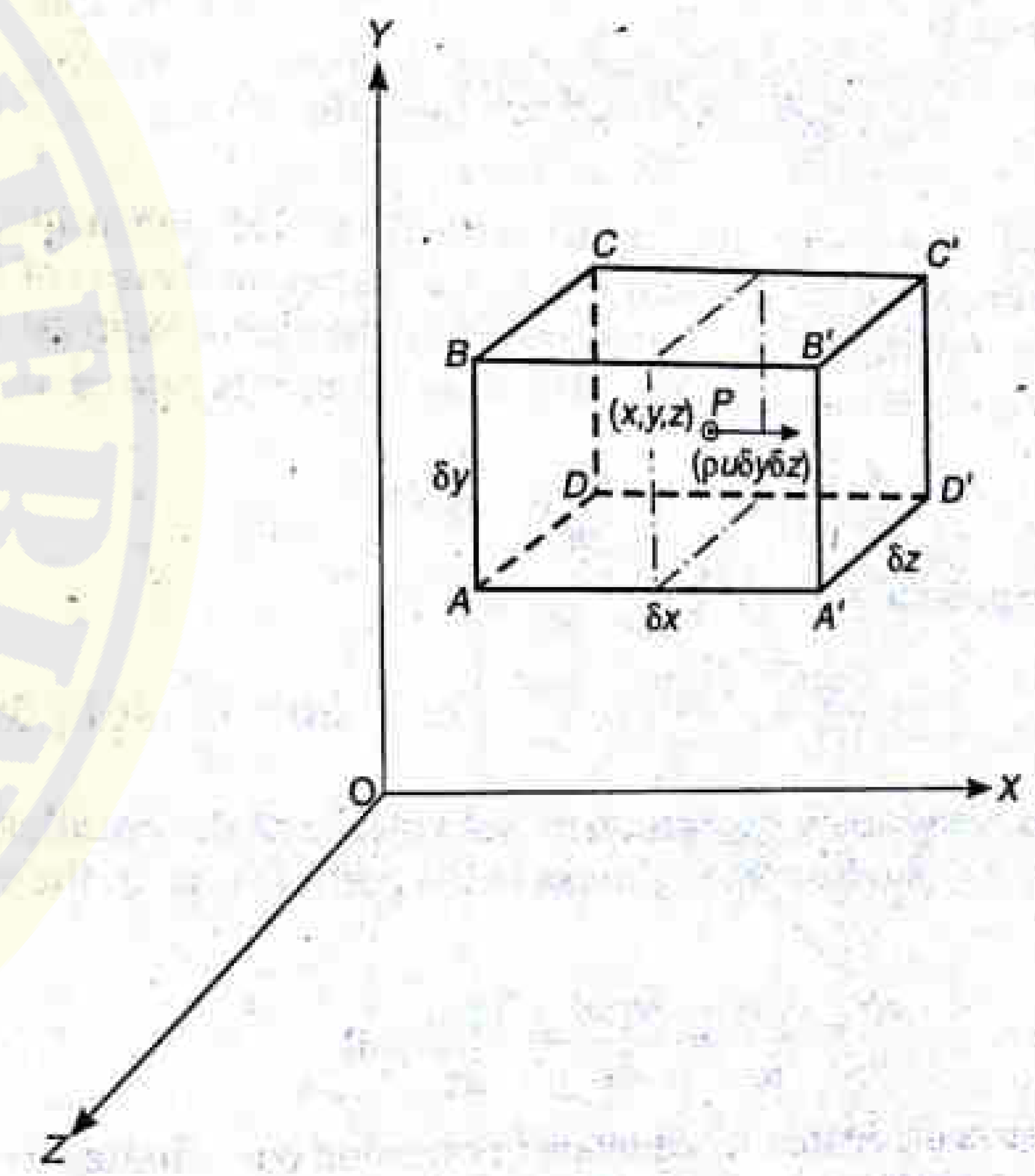


Figure 6.7 *Elementary rectangular parallelopiped*

In the above expression negative sign has been used because face $ABCD$ is on the left of point P. Similarly the mass of fluid flowing per unit time out of the parallelopiped through the face $A'B'C'D'$ is

$$\left[(\rho u\ \delta y\ \delta z)+\frac{\partial}{\partial x}(\rho u\ \delta y\ \delta z)\left(\frac{\delta x}{2}\right)\right]$$

Therefore, the net mass of fluid that has remained in the parallelopiped per unit time through the pair of faces $ABCD$ and $A'B'C'D'$ is obtained as

$$\left[\rho u\,\delta y\,\delta z-\frac{\partial}{\partial x}(\rho u\,\delta y\,\delta z)\frac{\delta x}{2}\right]-\left[\rho u\,\delta y\,\delta z+\frac{\partial}{\partial z}(\rho u\,\delta y\,\delta z)\frac{\delta x}{2}\right]=-\frac{\partial}{\partial x}(\rho u\,\delta y\,\delta z)$$

$$=-\frac{\partial}{\partial x}(\rho u)\,\delta x\,\delta y\,\delta z$$

The area ($\delta y\,\delta z$) has been taken out of the parentheses since it is not a function of x.

By applying the same procedure the net mass of fluid that remains in the parallelopiped per unit time through the other two pairs of faces of the parallelopiped may also be obtained as

$$=-\frac{\partial}{\partial y}(\rho v)\,\delta x\,\delta y\,\delta z, \text{ through pair of faces } AA'D'D \text{ and } BB'C'C$$

$$=-\frac{\partial}{\partial z}(\rho w)\,\delta x\,\delta y\,\delta z, \text{ through pair of faces } DD'C'C \text{ and } AA'B'B$$

By adding all these expressions the net total mass of fluid that has remained in the parallelopiped per unit time is obtained as

$$-\left[\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}\right]\delta x\,\delta y\,\delta z$$

Since the fluid is neither created nor destroyed in the parallelopiped, any increase in the mass of the fluid contained in this space per unit time is equal to the net total mass of fluid that remained in the parallelopiped per unit time, which is represented by the above expression.

The mass of the fluid in the parallelopiped is ($\rho\,\delta x\,\delta y\,\delta z$) and its rate of increase with time is

$$\frac{\partial}{\partial t}(\rho\,\delta x\,\delta y\,\delta z)=\frac{\partial\rho}{\partial t}(\delta x\,\delta y\,\delta z)$$

Equating the two expressions

$$-\left[\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}\right](\delta x\,\delta y\,\delta z)=\frac{\partial\rho}{\partial t}(\delta x\,\delta y\,\delta z)$$

Dividing both sides of the above expression by the volume of the parallelopiped ($\delta x\,\delta y\,\delta z$) and taking the limit so that the parallelopiped shrinks to the point $P(x, y, z)$, the continuity equation is obtained as

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad ...(6.4)$$

Equation 6.4 represents the continuity equation in cartesian coordinates in its most general form which is applicable for steady as well as unsteady flow, uniform and non-uniform flow, and compressible as well as incompressible fluids.

For steady flow since, $\frac{\partial\rho}{\partial t}=0$, Eq. 6.4 reduces to

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad ...(6.5)$$

Further for an incompressible fluid the mass density ρ does not change with x, y, z and t and hence Eq. 6.4 simplifies to

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad ...(6.6)$$

In vector notation the continuity equation may be expressed as

$$\frac{D\rho}{Dt}+\rho\nabla.V=0$$

in which

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

and

$$\nabla.V=\text{div } V=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}$$

Continuity Equation in Cylindrical Polar Coordinates. Often a continuity equation is required to be used in terms of the cylindrical polar coordinates which may also be derived by adopting the procedure as indicated below.

Consider any point $P(r, \theta, z)$ in space. Let δr, $\delta\theta$ and δz be the small increments in the direcions r, θ and z respectively, so that $PS = \delta r$, $PQ = r\delta\theta$ and $PT = \delta z$. Construct an elementary parallelopiped as shown in Fig. 6.8. Let V_r, V_θ and V_z be the components of the velocity V in the directions of r, θ and z at point P. Further let ρ represent the mass density of fluid at point P.

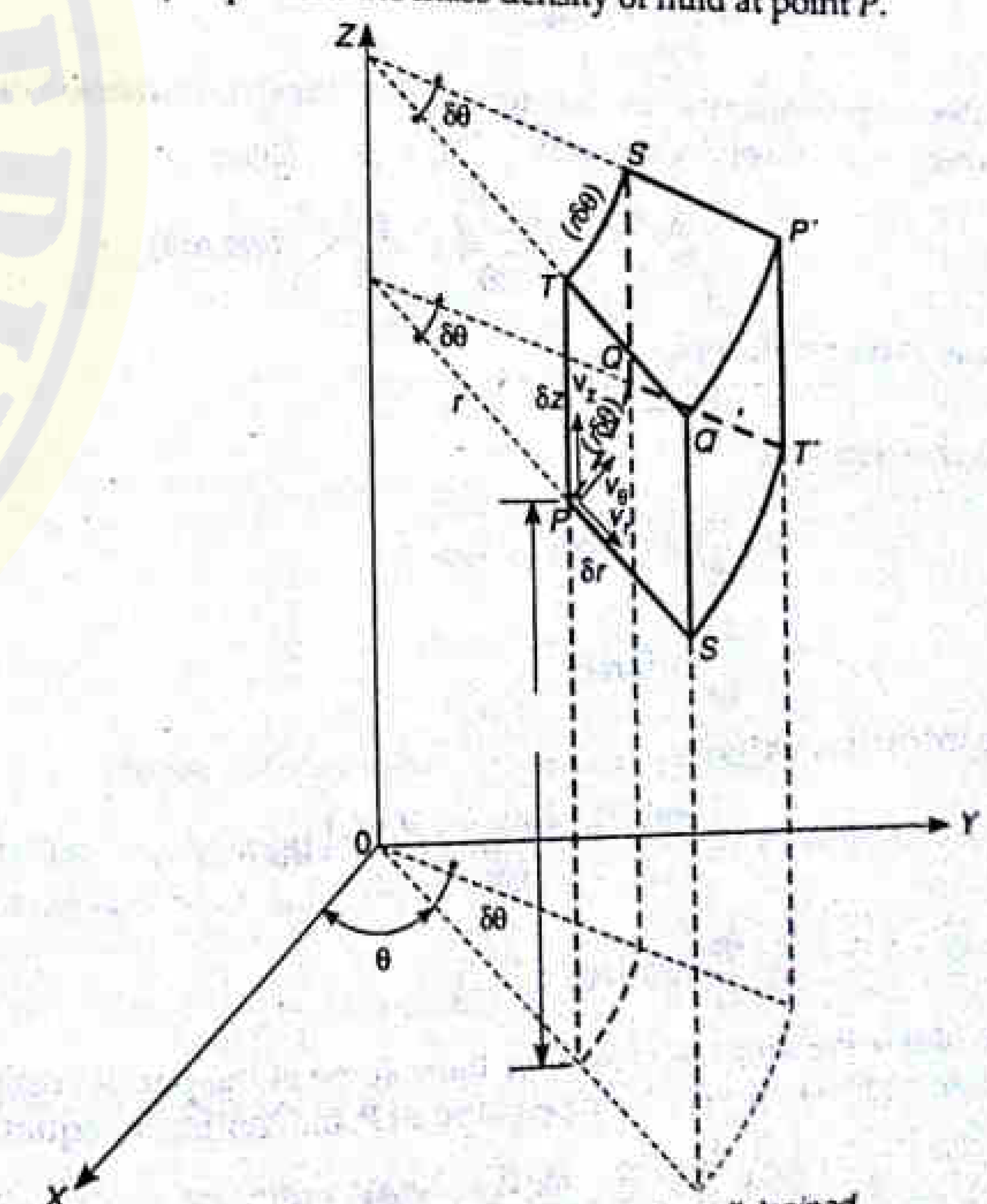


Figure 6.8 *Elementary cylindrical parallelopiped*

Considering the pair of faces $PST'Q$ and $P'S'TQ'$, the mass of fluid entering the parallelopiped per unit time through the face $PST'Q$

$$= \rho V_z\ (\delta r \times r\delta\theta)$$

Mass of fluid leaving the parallelopiped per unit time through the face $P'S'TQ'$

$$= \rho V_z\ (\delta r \times r\delta\theta) +$$

Therefore the net mass of fluid that has remained in the parallelopiped per unit time through this pair of faces

$$= -\frac{\partial}{\partial z}(\rho V_z \delta r \times r\delta\theta)\delta z$$

Similarly the net mass of fluid that remains in the parallelopiped per unit time through the pair of faces $PTQ'S$ and $P'T'QS'$

$$= -\frac{\partial}{\partial\theta}(\rho V_\theta \delta r\ \delta z)\delta\theta$$

and that through the pair of faces $PQS'T'$ and $P'Q'ST$

$$= -\frac{\partial}{\partial r}(\rho V_r r\delta\theta\delta z)\delta r$$

By adding all these expressions the net total mass of fluid that has remained in the parallelopiped per unit time through all the three pairs of faces $\frac{\partial}{\partial z}(\rho V_z \delta r \times r\delta\theta)\delta z$

$$= -\left[\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(\rho V_z)}{\partial z}\right](\delta r\delta z r\delta\theta)$$

The mass of fluid in the parallelopiped

$$= \rho\,(\delta r\,\delta z\, r\delta\theta)$$

and its rate of increase with time

$$= \frac{\partial}{\partial t}(\rho\delta r\delta z r\delta\theta)$$

$$= \frac{\partial\rho}{\partial t}(\delta r\delta z r\delta\theta)$$

Thus equating the two expressions

$$-\left[\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(\rho V_z)}{\partial z}\right](\delta r\delta z r\delta\theta)$$

$$= \frac{\partial\rho}{\partial t}(\delta r\delta z r\delta\theta)$$

Dividing both sides of the above expression by the volume of the parallelopiped ($\delta r\ \delta z\ r\delta\theta$) and taking the limit so as to reduce the parallelopiped to point P, the continuity equation is obtained as

$$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(\rho V_z)}{\partial z} = 0 \quad ...(6.7)$$

Equation 6.7 represents the continuity equation in cylindrical polar coordinates in its most general form which is applicable for steady or unsteady flow, uniform or non-uniform flow and compressible or incompressible fluids.

Again for steady flow since, $\frac{\partial\rho}{\partial t} = 0$, Eq. 6.7 reduces to

$$\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} + \frac{\partial(V_z)}{\partial z} = 0 \quad ...(6.8)$$

Further for an incompressible fluid the mass density ρ does not change with r, θ, z and t and hence Eq. 6.7 simplifies to

$$\frac{\partial(V_r r)}{r\partial r} + \frac{\partial(V_\theta)}{r\partial\theta} + \frac{\partial(V_z)}{\partial z} = 0 \quad ...(6.9)$$

Continuity Equation in Spherical Polar Coordinates. By considering an elementary parallelopiped as shown in Fig. 6.9 and adopting the same procedure, the equation of continuity in terms of

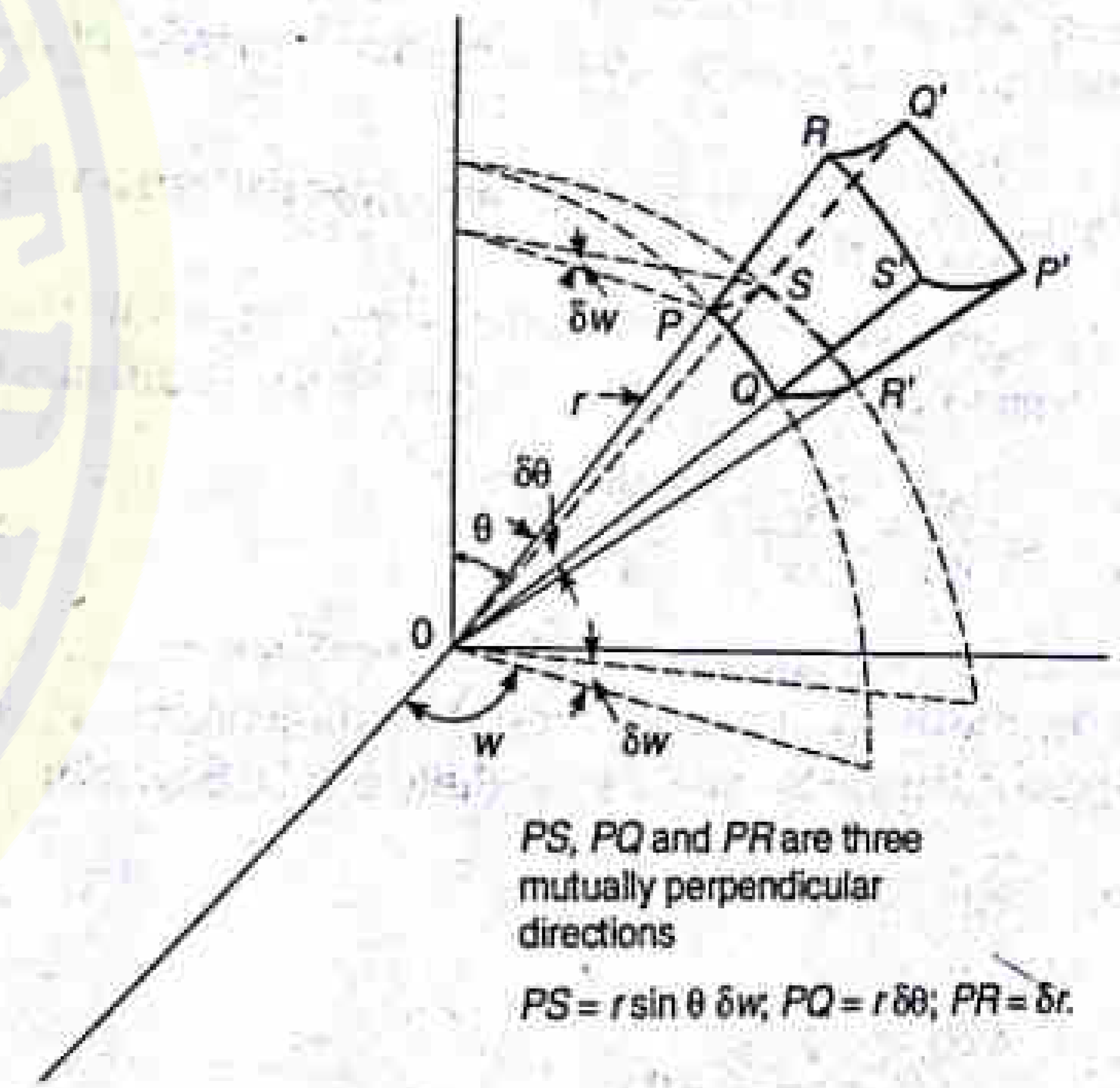


Figure 6.9 *Elementary spherical parallelopiped*

spherical polar coordinates can also be derived. However, it is not derived here, but simply mentioned in its most general form as noted below:

$$\frac{\partial\rho}{\partial t} + \frac{1}{r^2}\frac{\partial}{\partial r}(\rho V_r r^2) + \frac{1}{r\sin\theta}\frac{\partial}{\partial\theta}(\rho V_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial w}(\rho V_w) = 0 \quad ...(6.10)$$

in which the coordinates of any point are represented as (r, θ, w).

Fot steady flow since $\frac{\partial\rho}{\partial t} = 0$, Eq. 6.10 reduces to

$$\frac{1}{r^2}\frac{\partial}{\partial r}(\rho V_r r^2) + \frac{1}{r\sin\theta}\frac{\partial}{\partial\theta}(\rho V_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial w}(\rho V_w) = 0 \quad \text{...(6.11)}$$

Further for an incompressible fluid the mass density ρ does not change with r, θ and w and hence Eq. 6.10 simplifies to

$$\frac{1}{r^2}\frac{\partial}{\partial r}(V_r r^2) + \frac{1}{r\sin\theta}\frac{\partial}{\partial\theta}(V_\theta \sin\theta) + \frac{1}{r\sin\theta}\frac{\partial}{\partial w}(V_w) = 0 \quad \text{... (6.12)}$$

Continuity Equations for Two-Dimensional and One-Dimensional Flows. In several practical problems the flow may be considered as either two-dimensional or one-dimensional only. The consideration leads to a considerable simplification of the continuity equation as indicated below.

In two-dimensional flow the various characteristics of flowing fluid are the functions of only any two of the three coordinate directions. Thus in this case same flow pattern is displayed in all the planes perpendicular to the coordinate direction in which the characteristics of flowing fluid do not vary. The velocity vector has therefore components in only two coordinate directions and in the third coordinate direction in which the characteristics of flowing fluid do not vary, the component of the velocity is every where equal to zero. The continuity equation may then be modified accordingly for the case of a two-dimensional flow. Thus assuming that the characteristics of flowing fluid do not vary in the coordinate direction z $\left(\text{i.e., } \frac{\partial(\rho u)}{\partial z} = 0\right)$ then the same flow pattern is displayed in all the planes parallel to x–y plane and the component of velocity in the z-direction, $w = 0$. The equation of continuity for two-dimensional flow may therefore be obtained in cartesian coordinates from Eq. 6.4 as

$$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad \text{...(6.13)}$$

Equation 6.13 represents the continuity equation in the most general form for a two-dimensional flow which may also be derived directly by considering a two-dimensional elementary parallelopiped and adopting the same procedure as in the case of three-dimensional flow. For steady two-dimensional flow since $\frac{\partial\rho}{\partial t} = 0$, Eq. 6.13 reduces to

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad \text{...(6.14)}$$

Further for an incompressible fluid the mass density ρ does not change with x, y and t and hence Eq. 6.13 simplifies to

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad \text{...(6.15)}$$

Similarly for a two-dimensional flow continuity equation in cylindrical polar coordinates may be obtained from Eq. 6.7.

Thus if it is assumed that the component $V_z = 0$ then the continuity equation for a two-dimensional flow in cylindrical polar coordinates may be obtained from Eq. 6.7 as

$$\frac{\partial\rho}{\partial t} + \frac{1}{r}\frac{\partial(\rho V_r r)}{\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} = 0 \quad \text{... (6.16)}$$

For steady flow Eq. 6.16 simplifies to

$$\frac{\partial(\rho V_r r)}{r\partial r} + \frac{\partial(\rho V_\theta)}{r\partial\theta} = 0 \quad \text{... (6.17)}$$

and for an incompressible fluid Eq. 6.16 further simplifies to

$$\frac{\partial(V_r r)}{r\partial r} + \frac{\partial(V_\theta)}{r\partial\theta} = 0 \quad \text{... (6.18)}$$

In one-dimensional flow the various characteristics of flowing fluid are the functions of only one of the three coordinate directions which is the general direction of flow. One-dimensional consideration of flow leads to a simple form of continuity equation which may also be derived from Eq. 6.4. Thus if it is assumed that the characteristics of flowing fluid do not vary in the coordinate directions y and z $\left(\text{i.e., } \frac{\partial(\rho u)}{\partial y} = 0\ \frac{\partial(\rho u)}{\partial z} = 0\right)$ then for one-dimensional flow the equation of continuity Eq. in Cartesian coordinates may be obtained from equation 6.4 as

$$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} = 0$$

The continuity equation in the above noted form does not involve the cross-sectional area of the flow passage and hence it is applicable to the case of one-dimensional flow in which the flow passage has uniform cross-sectional area. In fact a true one-dimensional flow would occur only in a straight flow passage of uniform cross-sectional area. However, one-dimensional flow may also be assumed to occur in the case of straight or curved flow passage with varying cross-sectional area, if the velocity of flow is uniform at each section of the flow passage. The continuity equation involving the variation of the cross-sectional area of the flow passage may be derived for one-dimensional flow as explained below.

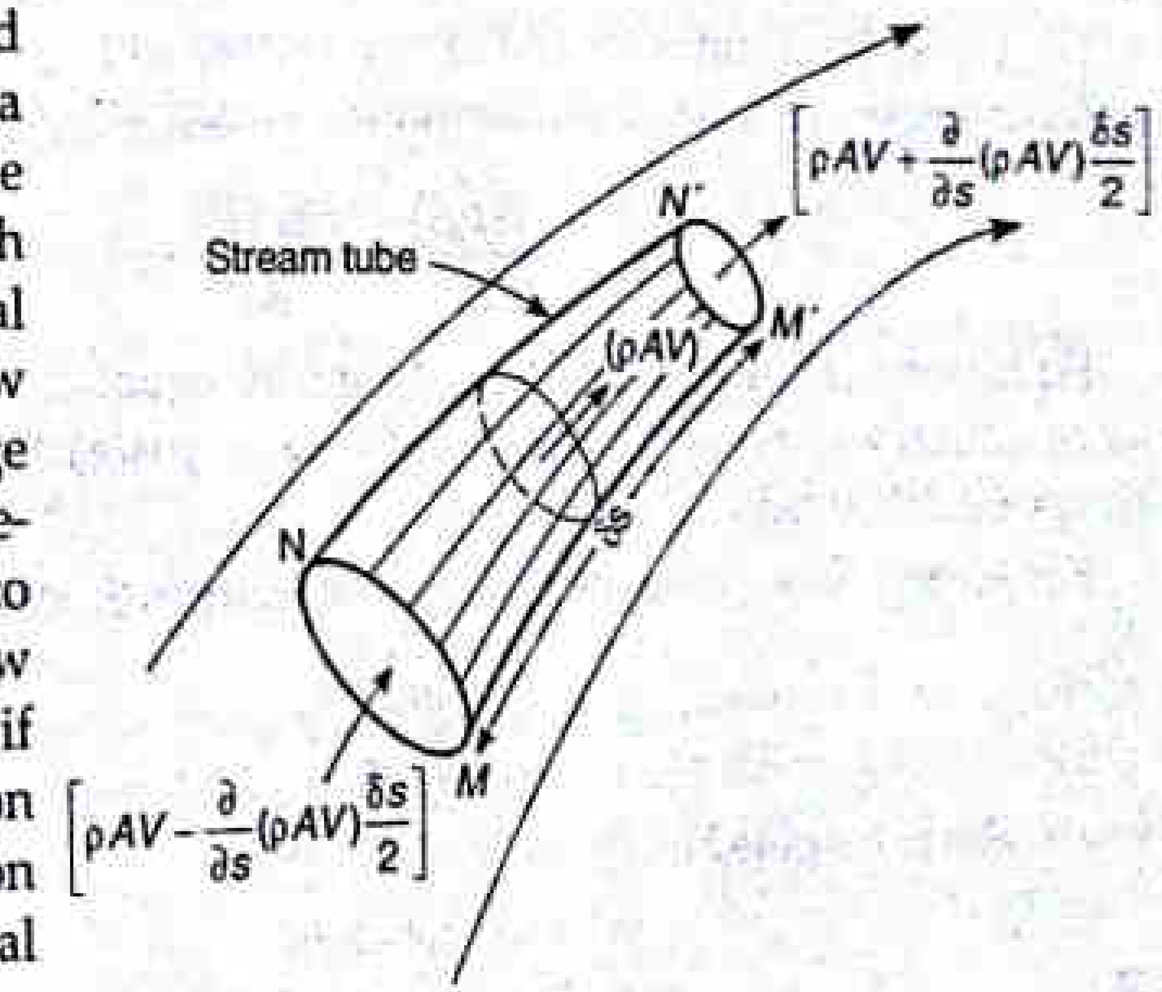


Figure 6.10 *Flow through a stream tube*

Consider a tube-shaped elementary parallelopiped along a stream-tube of length δs as shown in Fig. 6.10. Since the flow through a stream-tube is always along the tangential direction, there is no component of velocity in the normal direction. If at the central section of the elementary stream-tube, A is the cross-sectional area, V is the mean velocity of flow and ρ is the mass density of the fluid, then the mass of fluid passing through this section per unit time is equal to (ρAV).

The mass of fluid entering the parallelopiped per unit time at section NM

$$= \left[\rho AV - \frac{\partial}{\partial s}(\rho AV)\frac{\delta s}{2}\right]$$

Similarly the mass of fluid leaving the parallelopiped per unit time at section $N'M'$

$$= \left[\rho AV + \frac{\partial}{\partial s}(\rho AV)\frac{\delta s}{2}\right]$$

Therefore the net mass of fluid that has remained in the parallelopiped per unit time

$$= -\frac{\partial}{\partial s}(\rho AV)\,\delta s$$

The mass of fluid in the parallelopiped

$$= (\rho A\delta s)$$

and its rate of increase with time

$$= \frac{\partial}{\partial t}(\rho A\delta s) = \frac{\partial}{\partial t}(\rho A)\,\delta s$$

Since the net mass of fluid that has remained in the parallelopiped per unit time is equal to the rate of increase of mass with time,

$$-\frac{\partial}{\partial s}(\rho AV)\,\delta s = \frac{\partial}{\partial t}(\rho A)\,\delta s$$

Dividing both sides of the above expression by δs and taking the limit so as to reduce the parallelopiped to a point the continuity equation is obtained as

$$\frac{\partial(\rho A)}{\partial t} + \frac{\partial(\rho AV)}{\partial s} = 0 \qquad \text{...(6.19)}$$

Equation 6.19 represents the continuity equation for one-dimensional flow in a most general form which will be applicable for steady or unsteady flow, uniform or non-uniform flow, and for compressible fluids.

For steady flow since there is no variation with respect to time Eq. 6.19 reduces to

$$\frac{\partial(\rho AV)}{\partial s} = 0 \qquad \text{...(6.20)}$$

from which we obtain

$$\rho AV = \text{constant} \qquad \text{...(6.21)}$$

or

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 = \rho_3 A_3 V_3 = \text{constant} \qquad \text{...(6.21 a)}$$

where ρ, A and V with subscripts 1, 2 and 3 correspond to mass density, cross-sectional area and velocity of flow respectively at any three sections of stream-tube under consideration. Equation 6.21 (a) thus represents equation of continuity which is applicable to a steady one-dimensional flow of compressible as well as incompressible fluids. However, for an incompressible fluid since the mass density ρ is constant and does not vary from point to point, Eq. 6.21 may be further simplified as

$$\frac{\partial(AV)}{\partial s} = 0 \qquad \text{...(6.22)}$$

from which we obtain

$$AV = \text{constant} \qquad \text{...(6.23)}$$

or

$$A_1V_1 = A_2V_2 = A_3V_3 = \text{constant} \qquad \text{...(6.23 a)}$$

Further $AV = q$, where q is the volume of fluid flowing through any section per unit time or the volume rate of flow of fluid which is known as *discharge*. Discharge is expressed in terms of cubic metre per second (m^3/s) or litres per second. One cubic metre per second is usually written as one cumec. Equation 6.23 (a) may then be expressed as

$$q = A_1V_1 = A_2V_2 = A_3V_3 \qquad \text{...(6.23 b)}$$

Equation 6.23 thus represents equation of continuity which is applicable to a steady one-dimensional flow of incompressible fluid. Equations 6.21 (a) and 6.23 (b) have been derived for a stream-tube having small cross-sectional areas A_1, A_2, A_3, etc., so that the velocity of flow at each section V_1, V_2, V_3 etc., may be assumed to be uniform for the entire cross-sectional area. However, these equations can also be applied to flow passages of larger areas, even if the velocity of flow at any cross-section of the flow passage is not uniform, i.e., it varies from point to point. In such cases the flow passage may be assumed to be divided into a number of small stream-tubes (or stream filaments) as shown in Fig. 6.11. If at any section MN of the flow passage the small stream-tubes have cross-sectional areas dA_1, dA_2, dA_3 etc., and the velocities of flow through these areas being v_1, v_2, v_3, etc., then according to Eq. 6.23 the discharge through each of these small areas is obtained as

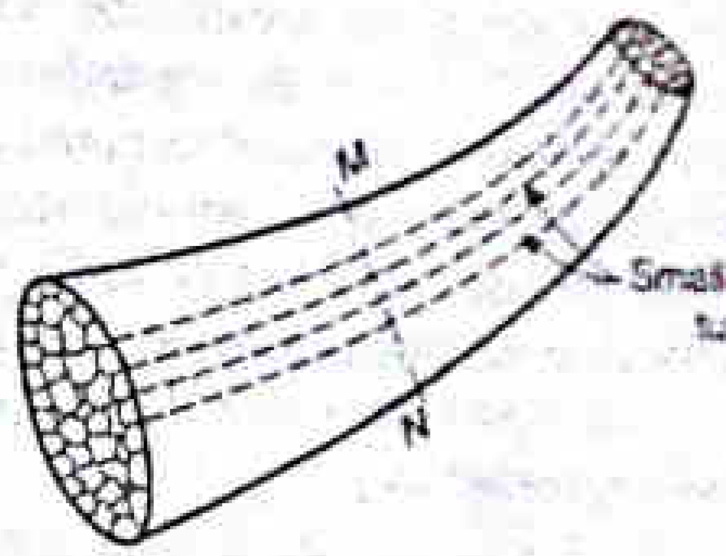


Figure 6.11 *Flow passage divided into a number of small stream-tubes*

$$dQ_1 = v_1 dA_1,\ dQ_2 = v_2 dA_2,\ dQ_3 = v_3 dA_3; \text{etc.}$$

The total discharge Q passing through the entire section of the flow passage of cross-sectional area A is then obtained as

$$\begin{aligned} Q &= dQ_1 + dQ_2 + dQ_3 + \ldots \\ &= v_1 dA_1 + v_2 dA_2 + v_3 dA_3 + \ldots \\ &= \Sigma v dA \end{aligned}$$

Moreover the discharge Q may also be expressed in terms of the average or mean velocity of flow through the section. Thus if V is the mean velocity of flow at any section of the flow passage of area A then

$$Q = AV, \text{ where } V = \frac{1}{A}\int v\,dA$$

Further if the flow passage has cross-sectional areas A_1, A_2, A_3, etc., at different sections taken normal to the direction of flow, where the mean velocities of flow are V_1, V_2, V_3 etc., then from Eq. 6.23 (b) the discharge Q flowing through the passage may be expressed as

$$Q = A_1V_1 = A_2V_2 = A_3V_3 \qquad \text{...(6.24)}$$

It is observed from Eq. 6.24 that the velocity of flow at any section in inversely proportional to the area of flow section, that is, as the area of flow section increases the velocity of flow decreases and as the area of flow section decreases the velocity of flow increases.

Although Eq. 6.24 is derived for one-dimensional flow conditions and accordingly it is usually referred to as the one-dimensional equation of continuity, yet in most engineering problems of fluid flow in pipes and open channels, the one-dimensional approach is used by considering the mean velocities of flow at the flow sections.

Moreover the one-dimensional approach may also be used to derive a simplified continuity equation for the problems of fluid flow involving two-dimensional flow. In such a case as shown in Fig. 6.12 the space between two adjacent streamlines on the two planes parallel to the xy plane at a unit distance (in the z direction) apart can be considered as a stream-tube. Then the area along the stream-tube is numerically equal to the spacing dn of the streamlines. Now if v_1 and v_2 are the mean velocities at sections 1 and 2 where the spacings between the two adjacent streamlines are dn_1 and dn_2 respectively, then according to Eq. 6.23 for steady flow of incompressible fluid through a stream-tube in a two-dimensional flow, the discharge dq may be expressed as

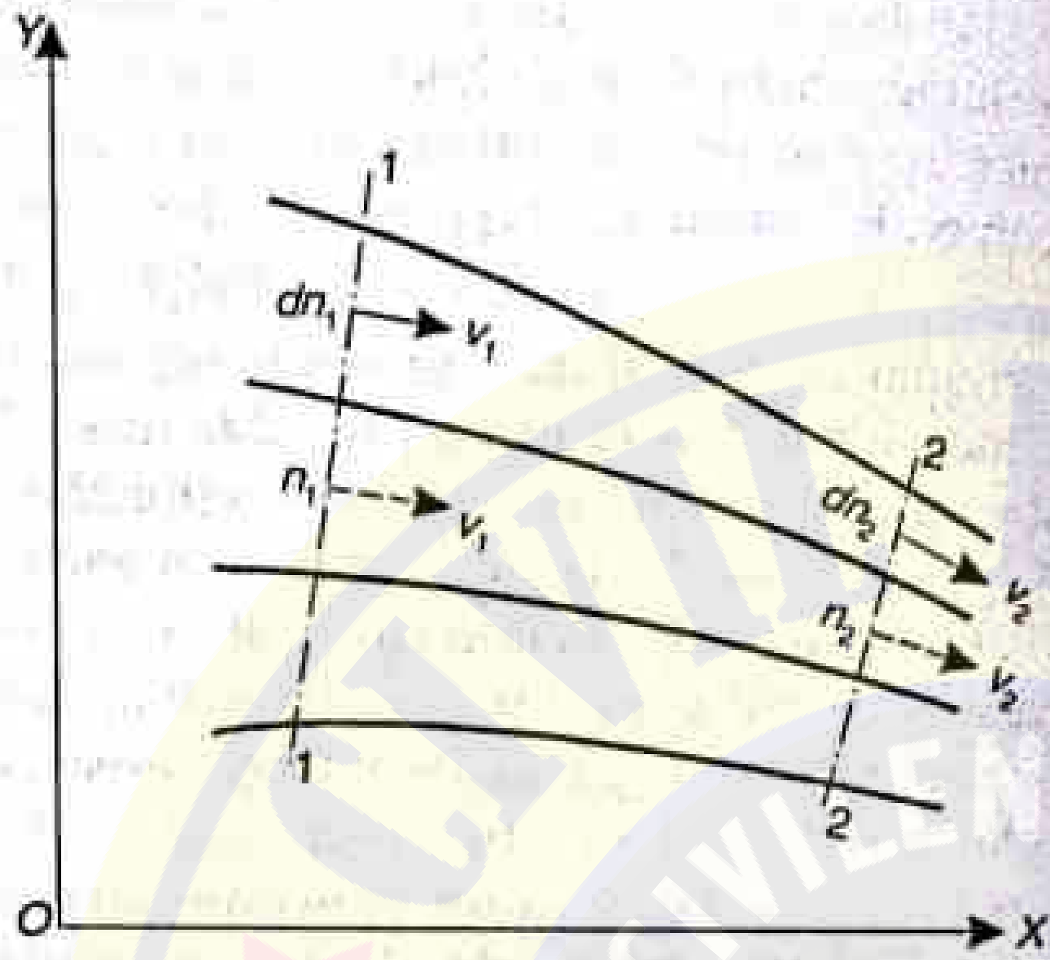


Figure 6.12 *Two-dimensional streamlines*

$$dq = v_1 dn_1 = v_2 dn_2$$

which also shows that the velocity of flow between two adjacent streamlines is inversely proportional to the spacing of the streamlines. The continuity equation for the total volume rate of flow of fluid per unit width of the entire section of the flow passage may be obtained by the integration of the above expression. Therefore

$$q = \int v_1 dn_1 = \int v_2 dn_2$$

or

$$q = V_1 n_1 = V_2 n_2 \quad \text{...(6.25)}$$

where V_1 and V_2 are the mean velocities of flow for the entire flow passage at sections 1 and 2, and n_1 and n_2 are the total depths of flow of the entire flow passage in the directions normal to the mean velocities of flow at the sections 1 and 2 respectively.

6.7 ACCELERATION OF A FLUID PARTICLE

Acceleration is defined as the rate of change of velocity with respect to time. As stated earlier, the velocity of a fluid particle is a function of both the positions of the point and time. Thus if a certain fluid particle has a velocity V which has u, v and w as the three components along three mutually perpendicular directions x, y and z, then the acceleration in the x, y and z directions are given by

$$\left.\begin{aligned} a_x &= \lim_{dt\to 0}\frac{du}{dt} \\ a_y &= \lim_{dt\to 0}\frac{dv}{dt} \\ a_z &= \lim_{dt\to 0}\frac{dw}{dt} \end{aligned}\right\} \quad \text{...(6.26)}$$

But Eq. 6.1 indicates that $u = f(x, y, z, t)$, and hence the total or substantial derivative of u with respect to time is

$$\frac{du}{dt} = \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\frac{dz}{dt} + \frac{\partial u}{\partial t}\frac{dt}{dt}$$

Further it has been shown that

$$\lim_{dt\to 0}\frac{dx}{dt} = u,\ \lim_{dt\to 0}\frac{dy}{dt} = v, \text{ and } \lim_{dt\to 0}\frac{dz}{dt} = w$$

Therefore substantial derivative of u becomes

$$\lim_{dt\to 0}\frac{du}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}$$

or

$$a_x = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t} \quad \text{...(6.27)}$$

By adopting the same procedure as above the following expressions for the other two components of acceleration a_y and a_z may also be obtained.

$$a_y = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t} \quad \text{...(6.28)}$$

$$a_z = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t} \quad \text{...(6.29)}$$

In vector notation acceleration may be represented as

$$a = \frac{DV}{Dt} = \frac{\partial V}{\partial t} + V.\nabla\, V$$

where

$$a = ia_x + ja_y + ka_z$$

Equations (6.27), (6.28) and (6.29) represent the expressions for the components of acceleration in the three mutually perpendicular directions. In these expressions $\left(\frac{\partial u}{\partial t}\right)$ or $\left(\frac{\partial v}{\partial t}\right)$ or $\left(\frac{\partial w}{\partial t}\right)$ represents the rate of increase of velocity with respect to time at a particular point in the flow and hence it is known as *local acceleration* or *temporal acceleration*. The remaining terms in these expressions represent the rate of increase of velocity due to the particle's change of position and hence it is known as *convective acceleration*. In steady flow the local acceleration in zero, but the convective acceleration is not necessarily zero. However, not necessarily zero and hence the total or substantial acceleration is not necessarily zero. However, in the case of uniform flow the convective acceleration is also zero.

Alike velocity, acceleration is also a vector quantity. However, unlike velocity vector the acceleration vector has no specific orientation with respect to the streamline i.e., it need not be always tangential to streamline. In other words the acceleration vector may have any direction so that at any point it has components both tangential and normal to the streamline. The tangential acceleration is developed for a fluid particle when the magnitude of the velocity changes with respect to space and time. On the other hand a normal acceleration is developed when a fluid particle moves in a curved path along which the direction of the velocity changes. Hence a normal acceleration may be developed simply due to the change in the direction of the velocity of the fluid particle, regardless of whether the magnitude of the velocity is changing. The expressions for the normal and the tangential accelerations may be developed as indicated below.

Tangential and Normal Accelerations. Let s and n represent the tangential and normal directions respectively at any point on the streamline as shown in Fig. 6.13. Let V_s and V_n be the components of velocity along tangential and normal directions respectively.

Then
$$V_s = f_1(s, n, t) \text{ and } V_n = f_2(s, n, t)$$

The accelerations a_s and a_n in the tangential and the normal directions may be expressed as

$$a_s = \lim_{dt \to 0} \frac{dV_s}{dt} \text{ and } a_n = \lim_{dt \to 0} \frac{dV_n}{dt}$$

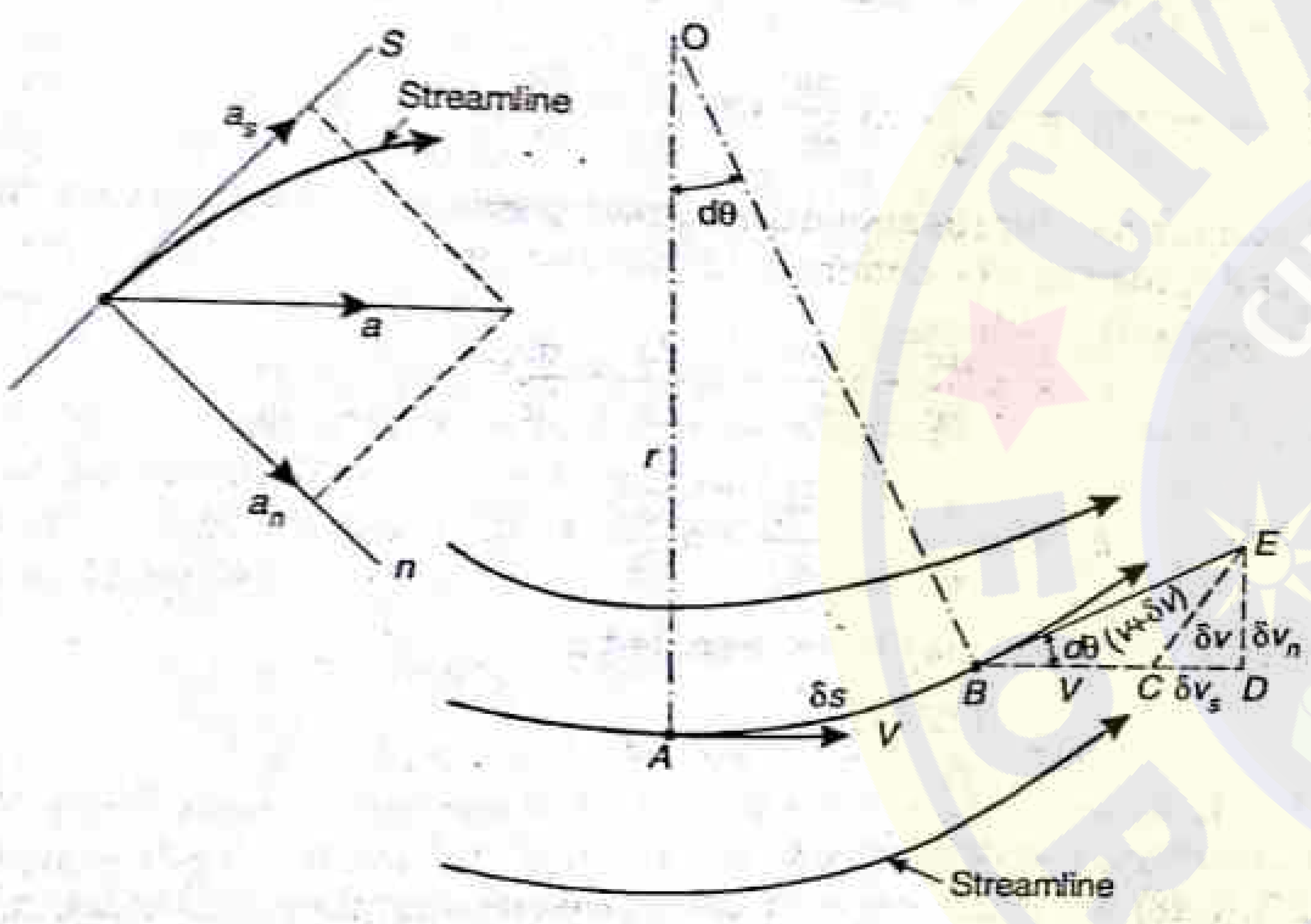


Figure 6.13 *Tangential and normal accelerations*

The tangential component of the acceleration is due to the change in the magnitude of velocity along the streamline and the normal component of the acceleration is due to the change in the direction of velocity vector.

Using the partial differentiation the total or substantial derivatives of V_s and V_n with respect to time are obtained as

$$\frac{dV_s}{dt} = \frac{\partial V_s}{\partial s}\frac{ds}{dt} + \frac{\partial V_s}{\partial n}\frac{dn}{dt} + \frac{\partial V_s}{\partial t}\frac{dt}{dt}$$

and
$$\frac{dV_n}{dt} = \frac{\partial V_n}{\partial s}\frac{ds}{dt} + \frac{\partial V_n}{\partial n}\frac{dn}{dt} + \frac{\partial V_n}{\partial t}\frac{dt}{dt}$$

Further
$$\lim_{dt \to 0}\frac{ds}{dt} = V_s \text{ and } \lim_{dt \to 0}\frac{dn}{dt} = V_n$$

Hence
$$\lim_{dt \to 0}\frac{dV_s}{dt} = a_s = V_s\frac{\partial V_s}{\partial s} + V_n\frac{\partial V_s}{\partial n} + \frac{\partial V_s}{\partial t}$$

and
$$\lim_{dt \to 0}\frac{dV_n}{dt} = a_n = V_s\frac{\partial V_n}{\partial s} + V_n\frac{\partial V_n}{\partial n} + \frac{\partial V_n}{\partial t}$$

Since for any streamline $V_n = 0$, the above expressions for the tangential and the normal accelerations become

$$a_s = V_s\frac{\partial V_s}{\partial s} + \frac{\partial V_s}{\partial t} \qquad ...(6.30)$$

and
$$a_n = V_s\frac{\partial V_n}{\partial s} + \frac{\partial V_n}{\partial t} \qquad ...(6.31)$$

In the above expressions it is to be noted that even though $V_n = 0$, $\left(\frac{\partial V_n}{\partial_s}\right)$ need not be zero. This is so because $V_n = 0$ at any point on the streamline but at any other point on the streamline the component of the velocity in the direction parallel to that of V_n need not always be zero.

Consider a series of curved streamlines which are not equidistant as shown in Fig. 6.13. Let at any point A on the streamline the fluid particle has a velocity V. In small time dt the fluid particle moves through a distance δs and occupies new position at point B where the velocity becomes $(V + \delta V)$. The total change in velocity δV can be resolved into two components δV_s and δV_n along the tangential and the normal directions respectively. As shown in Fig. 6.13 from triangle AOB, $\delta s = r\delta\theta$ and from triangle BDE, $\delta V_n = V\delta\theta$.

Thus

$$d\theta = \frac{\delta s}{r} = \frac{\delta V_n}{V}$$

or
$$\frac{\delta V_n}{\delta s} = \frac{\partial V_n}{\partial s} = \frac{V}{r} = \frac{V_s}{r} \text{ (since } V = V_s)$$

in which r is the radius of curvature of the streamlines.

Substituting the value of $\left(\frac{\partial V_n}{\partial s}\right)$ in Eq. 6.31 the acceleration in the normal direction becomes

$$a_n = \frac{V_s^2}{r} + \frac{\partial V_n}{\partial t} \qquad ...(6.32)$$

Again in Eqs 6.30 and 6.31 or 6.32, $\left(\frac{\partial V_s}{\partial t}\right)$ and $\left(\frac{\partial V_n}{\partial t}\right)$ represent the rate of increase of velocities V_s and V_n with respect to time at a particular point in the flow and hence these are known as local tangential acceleration and local normal acceleration respectively. Similarly in these expressions

$\left(V_s \frac{\partial V_s}{\partial s}\right)$ and $\left(V_s \frac{\partial V_n}{\partial s}\right)\left[\text{or } \frac{V_s^2}{r}\right]$ represent the rate of increase of velocities V_s and V_n due to the change in the position of the fluid particle and hence these are known as *convective tangential acceleration* and *convective normal acceleration* respectively.

For steady flow there is no variation of velocity with respect to time and hence there will be no local acceleration i.e.,

$$\frac{\partial V_s}{\partial t} = 0 \text{ and } \frac{\partial V_n}{\partial t} = 0$$

Therefore for steady flow there exists only convective acceleration and hence in such cases

$$a_s = V_s \frac{\partial V_s}{\partial s}$$

and

$$a_n = \frac{V_s^2}{r}$$

Further if the streamlines are straight lines then since $r = \infty$ the normal convective acceleration is zero. In other words, the normal convective acceleration is developed only if the fluid flows along a curved path, so that the streamlines are curved.

If the streamlines are straight and parallel to each other then even the tangential convective acceleration is also equal to zero. In other words, in such cases there is no acceleration. However if the streamlines are straight and converging then there will be tangential convective acceleration developed.

TABLE 6.1 *Types of Accelerations for Different Types of Streamline Patterns for Steady Flow*

S.No.	*Streamline pattern*	*Type of acceleration*
1	Straight parallel Streamlines	No acceleration
2	Straight converging Streamlines	Convective tangential acceleration
3	Concentric Streamlines	Convective normal acceleration
4	Curved converging Streamlines	Both tangential and normal convective acceleration

When the streamlines are curved and equidistant then the tangential convective acceleration is zero and there will be only normal convective acceleration developed. However if the streamlines are curved and converging then both normal and tangential convective accelerations will be developed. It may however be stated that if instead of converging streamlines there are diverging streamlines then instead of acceleration there will be corresponding retardation developed in the direction of flow.

Table 6.1 shows different streamline patterns and the types of accelerations for each of these patterns for the case of steady flow.

6.8 ROTATIONAL AND IRROTATIONAL MOTIONS

As shown in Fig. 6.14, a fluid particle as it moves can undergo the following four types of displacements viz.,

(i) Linear translation or pure translation.
(ii) Linear deformation.
(iii) Angular deformation.
(iv) Rotation.

A fluid particle is said to have undergone linear or pure translation if it moves bodily in such a way that the two axes *ab* and *cd* have their new positions represented by *a'b'* and *c'd'* which are parallel to the previous ones, as shown in Fig. 6.14 (a).

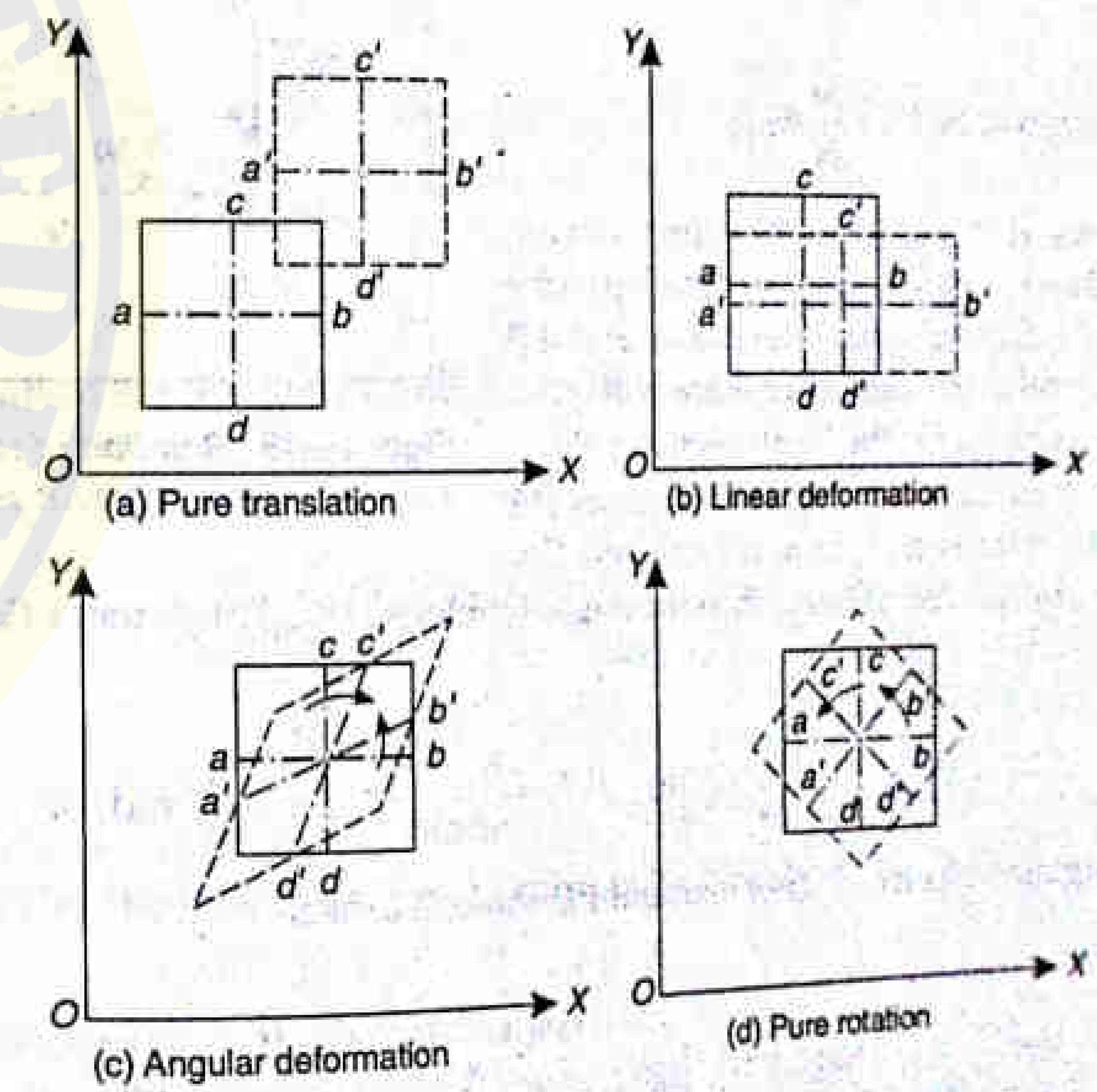


Figure 6.14 *Displacements of fluid particles*

A fluid particle is said to have undergone linear deformation if as it moves it gets deformed in the linear direction, so that two axes *a'b'* and *c'd'* of the deformed particle are parallel to the two axes *ab* and *cd* of the undeformed particle, as shown in Fig. 6.14 (b). It is thus observed that in the case of both linear translation and linear deformation the fluid particles are displaced parallel to their original position.

A fluid particle is said to have undergone angular deformation if it deforms in such a way that the two axes ab and cd rotate by the same amount but in opposite directions with respect to their previous positions i.e., the clockwise rotation of the vertical axis cd is at the same rate as the counter-clockwise rotation of the horizontal axis ab as shown in Fig. 6.14 (c).

A fluid particle is said to have undergone rotation if its both the horizontal and the vertical axes rotate in the same direction as shown in Fig. 6.14 (d). When the fluid particles while moving in the direction of flow rotate about their mass centres then the flow is said to be rotational flow.

The rotation of a fluid element may be defined in terms of the components of rotation about three mutually perpendicular axes. Referring to Fig. 6.15 the mathematical expression for the rotation component about an axis parallel to z-axis is developed.

Let a fluid element at any point $P(x, y, z)$ has the velocity components u and v in the x and y directions respectively. Consider any two line segments PA and PB of lengths δx and δy, taken parallel to the x and y axes respectively for the sake of convenience. The velocity at A in the y direction will be $\left(v+\frac{\partial v}{\partial x}dx\right)$ and the velocity at B in the x direction will be $\left(u+\frac{\partial u}{\partial y}dy\right)$.

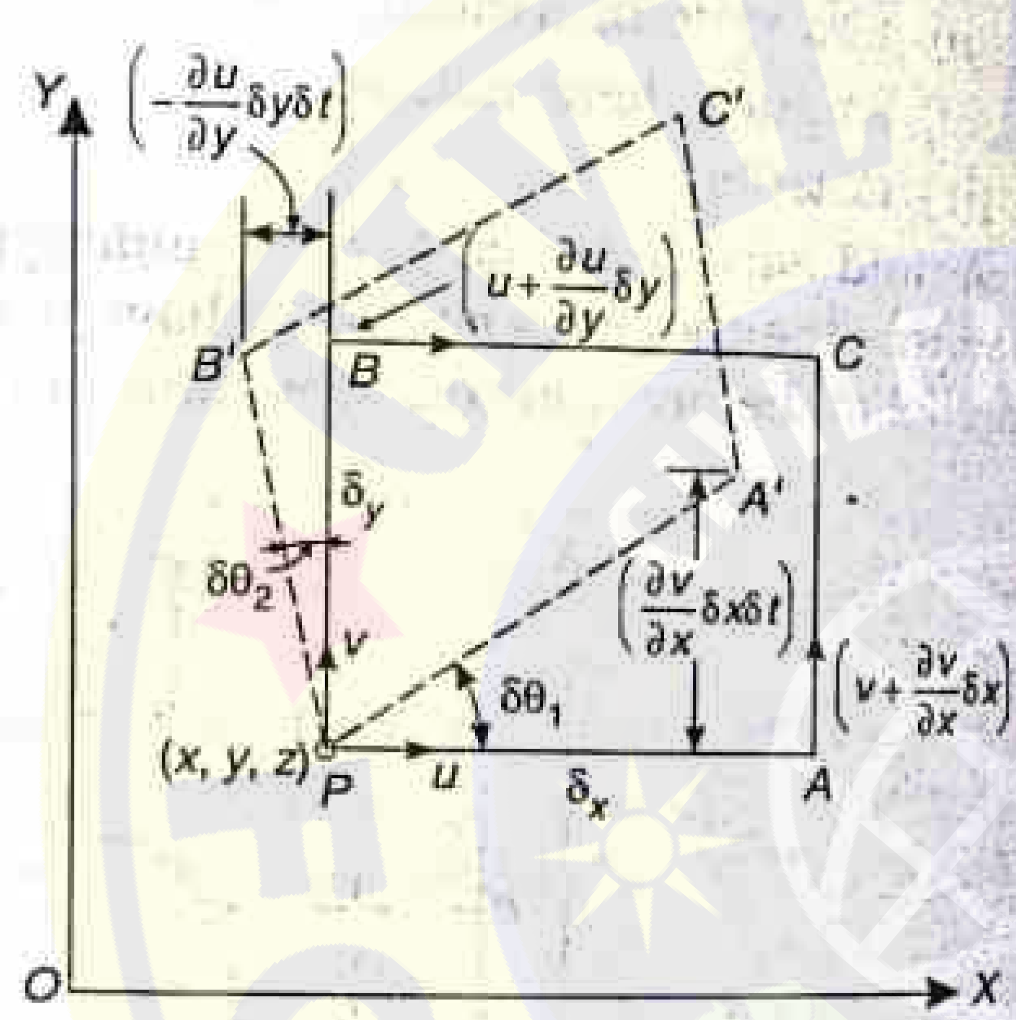


Figure 6.15 *Rotation of rectangular fluid element about z-axis.*

Since the velocities at P and A in the y direction are different, there will be an angular velocity developed for the linear element PA. Similarly the velocities at P and B in the x direction are different and hence there will be an angular velocity developed for the linear element PB.

Now if during a time interval of dt the elements PA and PB have moved, relative to P, to new positions PA' and PB' as indicated by the dotted lines, then the angular velocity (w_{PA}) of element PA about Z axis is

$$\omega_{P_A} = \lim_{\delta t\to 0}\frac{\delta\theta_1}{\delta t} = \lim_{\delta t\to 0}\frac{\left[\left(v+\frac{\partial v}{\partial x}\delta x\right)-v\right]\delta t}{\delta x\delta t} = \frac{\partial v}{\partial x}\ \text{rad/s.}$$

Similarly, the angular velocity (ω_{PB}) of element PB about Z axis is

$$\omega_{P_B} = \lim_{\delta t\to 0}\frac{\delta\theta_2}{\delta t} = \lim_{\delta t\to 0}\frac{-\left[\left(u+\frac{\partial u}{\partial y}\delta y\right)-u\right]\delta t}{\delta x\delta t} = -\frac{\partial u}{\partial x}\ \text{rad/s.}$$

The negative sign has been introduced because the motion in the anticlockwise direction has been considered as positive.

The rotation component about any axis may be defined as the average angular velocity of any two infinitesimal linear elements in the particle that are perpendicular to each other and to the axis of rotation (in this case it is Z axis).

Thus by the above definition the rotation component ω_z of a particle situated at point P is

$$\omega_z = \frac{1}{2}(\omega_{PA}+\omega_{PB}) = \frac{1}{2}\left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right) \quad ...(6.33\ a)$$

By adopting the same procedure the rotation components about the x and y axes will be obtained as

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y}-\frac{\partial v}{\partial z}\right) \quad ...(6.33\ b)$$

and

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z}-\frac{\partial w}{\partial x}\right) \quad ...(6.33\ c)$$

If at every point in the flowing fluid the rotation components ω_x, ω_y and ω_z are equal to zero, then the flow is known as irrotational flow. Thus for a flow to be irrotational the following conditions must be satisfied throughout the flow field :

For $\quad \omega_x = 0;\ \frac{\partial w}{\partial y}=\frac{\partial v}{\partial z}$ ...(6.34 a)

For $\quad \omega_y = 0;\ \frac{\partial u}{\partial z}=\frac{\partial w}{\partial x}$ ...(6.34 b)

For $\quad \omega_z = 0;\ \frac{\partial v}{\partial x}=\frac{\partial u}{\partial y}$ ...(6.34 c)

In vector notation the rotation of a fluid at a point is expressed as

$$\omega = \frac{1}{2}\text{curl } V = \frac{1}{2}\nabla\times V$$

where $\quad \omega = i\omega_x + j\omega_y + k\omega_z$

Thus $\quad \nabla\times V = i2\omega_x + j\,2\omega_y + k\,2\omega_z$

or $\quad \nabla\times V = i\left(\frac{\partial w}{\partial y}-\frac{\partial v}{\partial z}\right)+j\left(\frac{\partial u}{\partial z}-\frac{\partial w}{\partial x}\right)+k\left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right)$

The condition for the flow to be irrotational may be expressed as

$$\text{curl } V = \nabla\times V = 0$$

which is equivalent to the condition expressed by Eq. 6.34.

The rotation of a fluid particle is always associated with shear stress, because the rotation can be caused only by a torque exerted on the fluid particle and this will be produced by the shear forces. As such in the case of flow of fluids having larger viscosity or in the regions of flow field where the viscosity of the fluid has predominance the flow is invariably rotational flow. However, in the case of fluids such as air or water having small viscosity the flow in the region away from the boundary may for all practical purposes be treated as irrotational. Moreover, in the case of rapidly coverging or accelerating flows the flow may be treated as irrotational. The consideration of an irrotational flow in general leads to a simplified analysis of fluid flow problems.

6.9 CIRCULATION AND VORTICITY

The flow along a closed curve is called *circulation* (i.e., the flow in eddies and vortices). The mathematical concept of circulation is the line integral, taken completely around a closed curve, of the tangential component of the velocity vector. Consider a closed curve C as shown in Fig. 6.16 (a), and let at any point on the curve the velocity of flow of fluid be V. If α is the angle between a small element ds along the curve in the tangential direction and the velocity V, then the component of the velocity in the direction tangential to the curve is $V\cos\alpha$. By the definition the circulation Γ (Greek, capital 'gamma') around a closed curve C is

$$\Gamma = \int_C V\cos\alpha\, ds \qquad \text{...(6.35)}$$

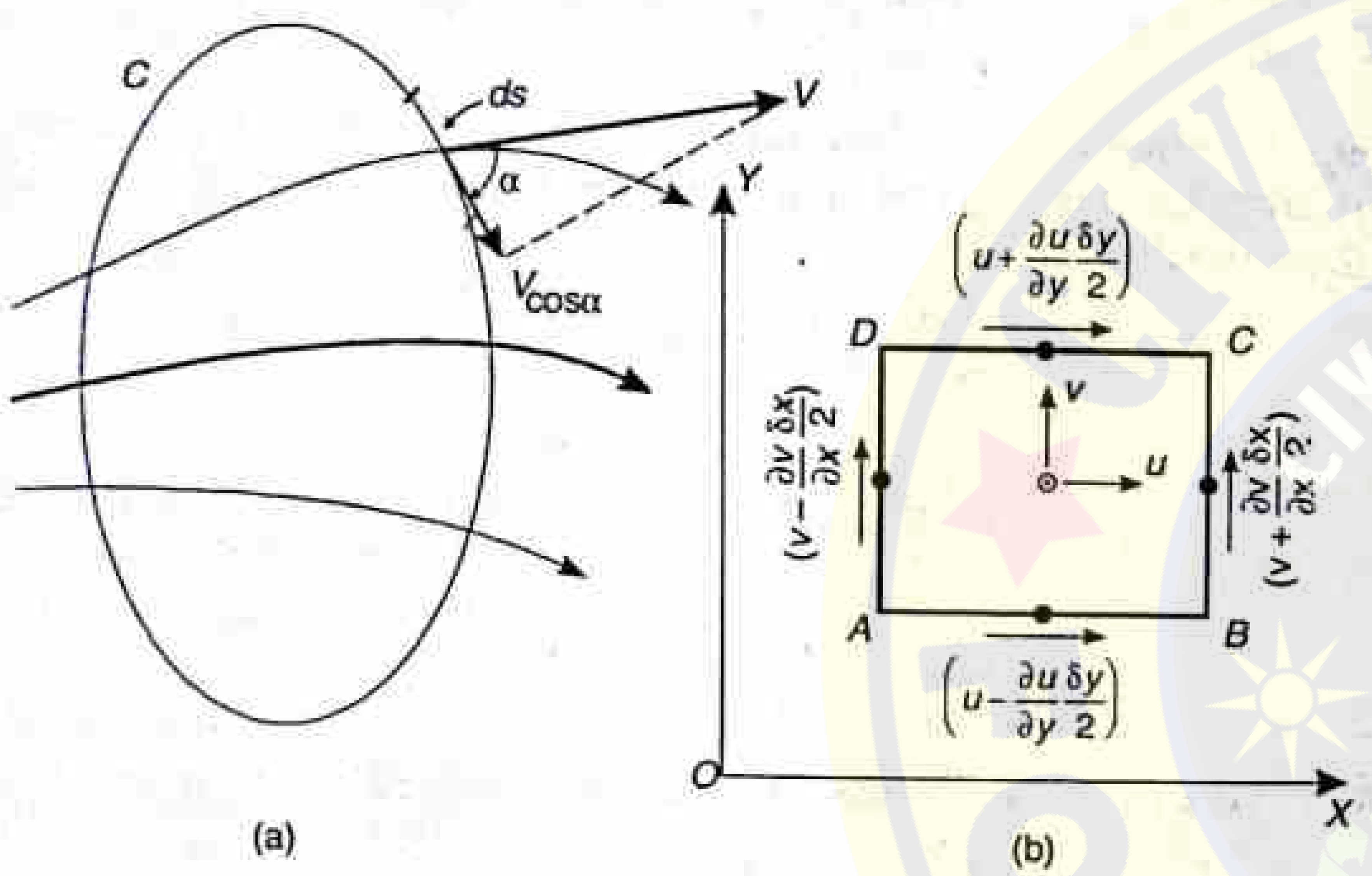


Figure 6.16 *Circulation around (a) a closed curve; (b) around an elementary rectangle, in the plane of a two-dimensional steady flow field*

Further if u, v and w are the components of velocity V, and dx, dy and dz are the components of the displacement ds, then the circulation can also be written as

$$\Gamma = \int_C (u\,dx + v\,dy + w\,dz) \qquad \text{...(6.35 a)}$$

The circulation around an elementary rectangle with sides parallel to the axes x and y as shown in Fig. 6.16 (b) may be written as follows:

$$\text{Circulation along } AB = \left(u-\frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x$$

$$\text{Circulation along } BC = \left(v+\frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y$$

$$\text{Circulation along } CD = -\left(u+\frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x$$

$$\text{Circulation along } DA = -\left(v-\frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y$$

The positive sense of integration is such that the enclosed surface is on left when viewed from the side of the outward normal.

Further whatever be the shape of the curve, the circulation around the periphery of the curve, must equal the sum of the circulation around the elementary surfaces of which it consists, provided the boundary of the curve is wholly in the fluid. Thus in this case the total circulation will be given by

$$\Gamma = \Gamma_{AB}+\Gamma_{BC}+\Gamma_{CD}+\Gamma_{DA}$$

$$= \left(u-\frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x + \left(v+\frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y - \left(u+\frac{\partial u}{\partial y}\frac{\delta y}{2}\right)\delta x - \left(v-\frac{\partial v}{\partial x}\frac{\delta x}{2}\right)\delta y$$

$$= \left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right)\delta x\,\delta y \qquad \text{...(6.36)}$$

The *vorticity* at any point is defined as the ratio of the circulation around an infinitesimal closed curve at that point to the area of the curve, i.e., it is defined as circulation per unit area. Thus from Eq. 6.36, the vorticity ζ 'Greek 'zeta') may be expressed as

$$\zeta = \frac{\text{Circulation}}{\text{Area}}$$

$$= \left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right) \qquad \text{...(6.37)}$$

By comparing Eqs 6.37 and 6.33 (a), we get

$$\zeta = 2\omega_z \qquad \text{...(6.38)}$$

that is vorticity (or circulation per unit area) is equal to twice the rotation component about an axis perpendicular to the plane in which the area is lying. Vorticity is a vector quantity whose direction is perpendicular to the plane of the small curve round which the circulation is measured. Thus in a general case of three-dimensional flow, Eq. 6.38 represents only a component of vorticity in the z direction, i.e., $\zeta_z = 2\omega_z$. Similarly the other two components of vorticity may also be obtained as $\zeta_x = 2\omega_x$ and $\zeta_y = 2\omega_y$. If the vorticity is zero at all points in a region then the flow in that region is said to be *irrotational*. On the other hand flow in regions where the vorticity is other than zero is said to be *rotational*.

The above results have been derived by considering a rectangular curve. But it may be stated that the results obtained in Eqs 6.36 and 6.38 are independent of the shape of the closed curve considered, and the rectangular curve has been chosen only for the sake of simplicity.

In vector notation circulation Γ may be expressed as

$$\Gamma = \iint_A \text{curl } V.\,dA$$

Similarly vorticity ζ may also be expressed in vector notation as

$$\zeta = \text{curl } V = \nabla\times V$$

6.10 VELOCITY POTENTIAL

The velocity potential ϕ (Greek 'phi') is defined as a scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction. Thus mathematically the velocity potential is defined as $\phi = f(x, y, z, t)$ for unsteady flow and $\phi = f(x, y, z)$ for steady flow such that

$$\left.\begin{aligned} u &= -\frac{\partial \phi}{\partial x} \\ v &= -\frac{\partial \phi}{\partial y} \\ w &= -\frac{\partial \phi}{\partial z} \end{aligned}\right\} \quad \text{...(6.39)}$$

where u, v and w are the components of velocity in the x, y and z directions respectively. The negative sign signifies that ϕ decreases with an increase in the values of x, y and z. In other words it indicates that the flow is always in the direction of decreasing ϕ.

For an incompressible fluid if the flow is steady then equation of continuity is given by Eq. 6.6 as

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

By substituting the values of u, v and w in terms of ϕ from Eq. 6.39, we get

$$\frac{\partial}{\partial x}\left(-\frac{\partial \phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(-\frac{\partial \phi}{\partial y}\right) + \frac{\partial}{\partial z}\left(-\frac{\partial \phi}{\partial z}\right) = 0$$

or

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0 \quad \text{...(6.40)}$$

This equation is known as *Laplace Equation*, which may be expressed in vector notation as

$$\nabla^2 \phi = 0$$

It is evident that any function ϕ that satisfies the Laplace equation will correspond to some case of fluid flow.

Further for a rotational flow the rotaion components are given by Eq. 6.33 as

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)$$

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

and

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

By substituting the values of u, v and w in terms of ϕ from Eq. 6.39, we get

$$\omega_x = \frac{1}{2}\left[-\frac{\partial^2 \phi}{\partial y \partial z} + \frac{\partial^2 \phi}{\partial z \partial y}\right]$$

$$\omega_y = \frac{1}{2}\left[-\frac{\partial^2 \phi}{\partial z \partial x} + \frac{\partial^2 \phi}{\partial x \partial z}\right]$$

and

$$\omega_z = \frac{1}{2}\left[-\frac{\partial^2 \phi}{\partial x \partial y} + \frac{\partial^2 \phi}{\partial y \partial x}\right]$$

However, if ϕ is a continuous function then

$$\frac{\partial^2 \phi}{\partial y \partial z} = \frac{\partial^2 \phi}{\partial z \partial y};\ \frac{\partial^2 \phi}{\partial z \partial x} = \frac{\partial^2 \phi}{\partial x \partial z} \text{ and } \frac{\partial^2 \phi}{\partial x \partial y} = \frac{\partial^2 \phi}{\partial y \partial x}$$

according to which $\omega_x = \omega_y = \omega_z = 0$ i.e., the flow is irrotational.

Therefore it may be stated that any function ϕ that satisfies Laplace equation is a possible irrotational flow case since the continuity is satisfied. In other words velocity potential exists only for irrotational flows of fluids. Hence often an irrotational flow is known as potential flow.

For two-dimensional flows the cylindrical polar coordinate system (with $z = 0$) is useful in several problems. As such the expressions for the velocity components in terms of the velocity potential can also be obtained in the cylindrical polar coordinates. Thus if V_r and V_θ are the velocity compoents in r and θ directions respectively then these can be expressed as

$$V_r = -\frac{\partial \phi}{\partial r};\ \text{and } V_\theta = -\frac{\partial \phi}{r \partial \theta}$$

By substituting the values of V_r and V_θ in the continuity Eq. 6.18, we get

$$\frac{\partial\left(-\frac{\partial \phi}{\partial r} r\right)}{r \partial r} + \frac{\partial\left(-\frac{\partial \phi}{r \partial \theta}\right)}{r \partial \theta} = 0$$

or

$$\frac{1}{r}\frac{\partial \phi}{\partial r} + \frac{\partial^2 \phi}{\partial r^2} + \frac{1}{r^2}\frac{\partial^2 \phi}{\partial \theta^2} = 0$$

which is Laplace equation in cylindrical polar coordinates for two-dimensional flows.

6.11 STREAM FUNCTION

The stream function ψ (Greek 'psi') is defined as a scalar function of space and time, such that its partial derivative with respect to any direction gives the velocity component at right angles (in the counter-clockwise direction) to this direction. For two-dimensional flow and three-dimensional flow with axial symmetry (i.e., axially symmetric flow) separate stream functions are considered. The stream function for the case of two-dimensional flow only is considered here. Thus mathematically stream function may be defined as $\psi = f(x, y, t)$ for unsteady flow and $\psi = f(x, y)$ for steady flow, such that

$$\left.\begin{aligned}\frac{\partial \psi}{\partial x} &= v\\ \frac{\partial \psi}{\partial y} &= -u\end{aligned}\right\} \qquad ...(6.41)$$

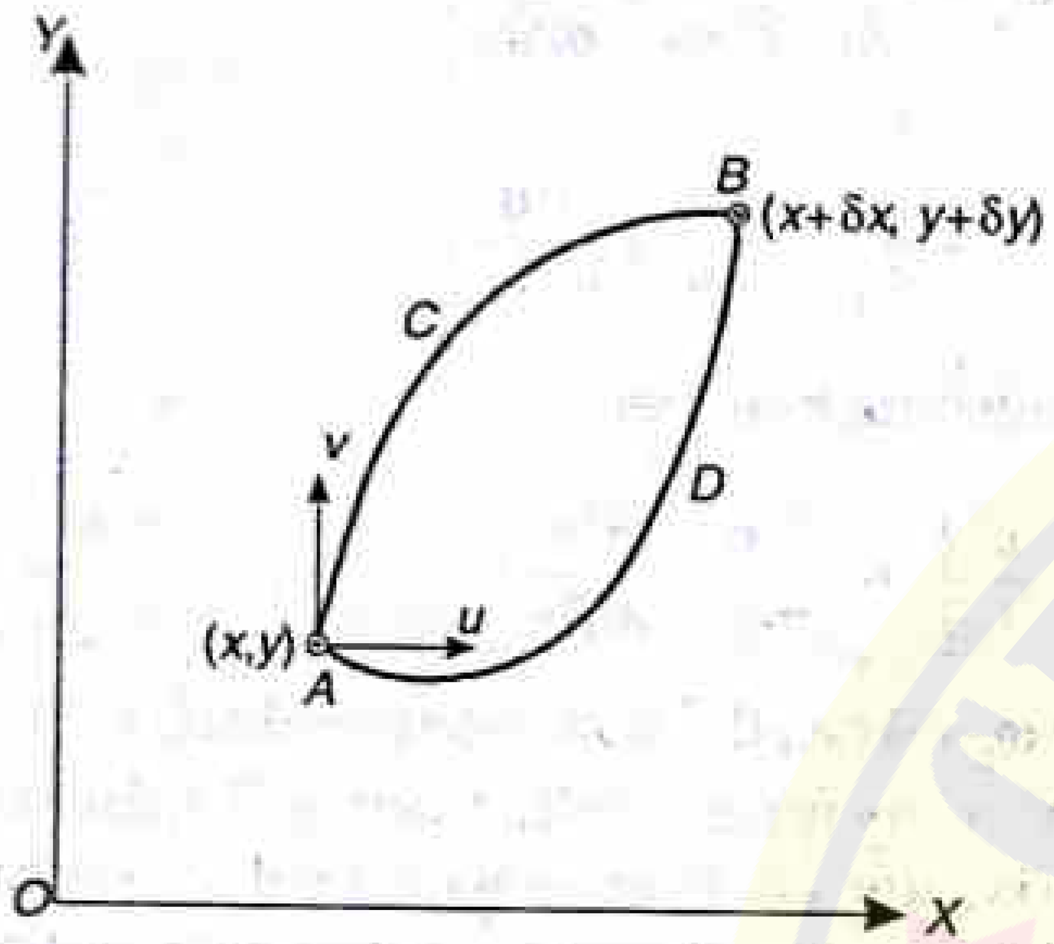


Figure 6.17 *Flow across curves between any two points in the fluid region*

The values of the velocity components u and v as represented by Eq. 6.41 may be derived as explained below.

Consider two points A and B with coordinates (x, y) and $(x + \delta x, y + \delta y)$ respectively, which are joined by any two arbitrary curves ACB and ADB as shown in Fig. 6.17. Let u and v be the velocity components at point A in the x and y directions respectively. Then the flow across the curve ACB in the x direction is equal to $(-u\delta y)$ and in the y direction it is equal to $(v\delta x)$. In the computation of the flows across the curve ACB the sign convention that is assumed is that as the observer views from A to B then the flow from left to right (or in the clockwise direction) is considered as negative and the flow from right to left (or in the counter-clockwise direction) is considered as positive. Now if $d\psi$ represents the total flow across the curve ACB then

$$d\psi = -u\delta y + v\delta x \qquad ...(6.42)$$

It may however be stated that if the fluid is homogeneous and incompressible, then the flow across ADB or any other curve must be the same as that across the curve ACB.

Further as stated earlier for steady flow, $\psi = f(x, y)$, the total derivative of ψ may be expressed as

$$d\psi = \frac{\partial \psi}{\partial x}\delta x + \frac{\partial \psi}{\partial y}\delta y \qquad ...(6.43)$$

Comparing the Eqs 6.42 and 6.43, we get

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

which is same as Eq. 6.41. Similarly in terms of cylindrical polar coordinates if V_r and V_θ are the components of velocity in the r and θ directions respectively, then by adopting the same procedure the following expressions may be obtained

$$\frac{\partial \psi}{r\partial \theta} = -V_r \text{ and } \frac{\partial \psi}{\partial r} = V_\theta$$

By comparing Eqs 6.39 and 6.41 the relationship between u, v and ϕ and ψ are obtained as

$$-\frac{\partial \phi}{\partial x} = u = -\frac{\partial \psi}{\partial y} \text{ and } -\frac{\partial \phi}{\partial y} = v = \frac{\partial \psi}{dx}$$

That is

$$\left.\begin{aligned}\frac{\partial \phi}{\partial x} &= \frac{\partial \psi}{\partial y}\\ -\frac{\partial \phi}{\partial y} &= \frac{\partial \psi}{\partial x}\end{aligned}\right\} \qquad ...(6.44)$$

These equations are known as Cauchy–Rieman equations and they enable the computation of stream function if the velocity potential is known and vice-versa in a potential flow.

By substituting the values of the velocity, components u and v from Eqs 6.41 in Eq. 6.33(a) the rotation component ω_z becomes

$$\omega_z = \frac{1}{2}\left[\frac{\partial}{\partial x}\left(\frac{\partial \psi}{\partial x}\right) - \frac{\partial}{\partial y}\left(-\frac{\partial \psi}{\partial y}\right)\right]$$

or

$$\omega_z = \frac{1}{2}\left[\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2}\right] \qquad ...(6.45)$$

Equation 6.45 is known as Poisson's equation. For an irrotational flow since $\omega_z = 0$, Eq. 6.45 becomes

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

which is Laplace equation for ψ.

Further by substituting the values of u and v from Eq. 6.41 in Eq. 6.15 which is the equation of continuity for two dimensional steady flow of an incompressible fluid, we get

$$\frac{\partial}{\partial x}\left(-\frac{\partial \psi}{\partial y}\right) + \frac{\partial}{\partial y}\left(\frac{\partial \psi}{\partial x}\right) = 0$$

or

$$\frac{\partial^2 \psi}{\partial x \partial y} = \frac{\partial^2 \psi}{\partial y \partial x}$$

This will be true if ψ is a continuous function and its second derivative exists. Therefore it may be stated that any function ψ which is continuous is a possible case of fluid flow (which may be rotational or irrotational) since the equation of continuity is satisfied.

However, if the function ψ is such that it satisfies Laplace equation then it is a possible case of an irrotational flow, as indicated above.

Further the expressions for the velocity components in terms of stream function can also be obtained in the cylindrical polar coordinates. Thus, if V_r and V_θ are the velocity components in r and θ directions respectively then these can be expressed as

$$V_r = -\frac{\partial \psi}{r\partial \theta};$$

and
$$V_\theta = \frac{\partial \psi}{\partial r}$$

The Cauchy–Rieman equations in cylindrical polar coordinates may thus be expressed as

$$\frac{\partial \phi}{\partial r} = \frac{\partial \psi}{r\partial \theta}$$

and
$$\frac{\partial \phi}{r\partial \theta} = -\frac{\partial \psi}{\partial r}$$

6.12 STREAMLINES, EQUIPOTENTIAL LINES AND FLOW NET

A property of the stream function is that the difference of its values at two points represents the flow across any line joining the points. Therefore, when two points lie on the same streamline, then since there is no flow across a streamline, the difference between the stream functions ψ_1 and ψ_2 at these two points is equal to zero, i.e., $(\psi_1 - \psi_2) = 0$. In other words, it means that *streamline* is given by ψ = constant.

Similarly ϕ = constant, represents a curve for which the velocity potential is same at every point, and hence it represents an *equipotential line.*

Consider two curves, viz., ϕ = constant and ψ = constant, intersecting each other at any point. The slopes of these curves at the point of intersection may be determined as indicated below.

For the curve ϕ = constant,

$$\text{Slope} = \frac{\partial y}{\partial x} = \frac{\left(\frac{\partial \phi}{\partial x}\right)}{\left(\frac{\partial \phi}{\partial y}\right)} = \frac{-u}{-v} = \frac{u}{v}$$

Similarly for the curve ψ = constant,

$$\text{Slope} = \frac{\partial y}{\partial x} = \frac{\left(\frac{\partial \psi}{\partial x}\right)}{\left(\frac{\partial \psi}{\partial y}\right)} = \frac{v}{-u} = -\frac{v}{u}$$

Therefore the product of the slopes of these two curves at the point of intersection = –1, which indicates that these two sets of curves, viz., streamlines and equipotential lines intersect each other orthogonally at all points of intersection.

A grid obtained by drawing a series of streamlines and equipotential lines is known as a *flow net*. A flow net may be drawn for a two-dimensional irrotational flow and it provides a simple, yet valuable

indication of the flow pattern. Figure 6.18 shows the elements of a flow net which has been obtained by drawing a set of curves corresponding to $\psi = C_1, C_2, C_3$ etc., and $\phi = C_1, C_2, C_3$ etc. Let at one of the intersections V_s and V_n be the velocity components in the directions tangential to the curves ψ = constant, and ϕ = constant, respectively.

At any point in the direction n along the equipotential line since ϕ = constant, C_2

$$\frac{\partial \phi}{\partial n} = 0$$

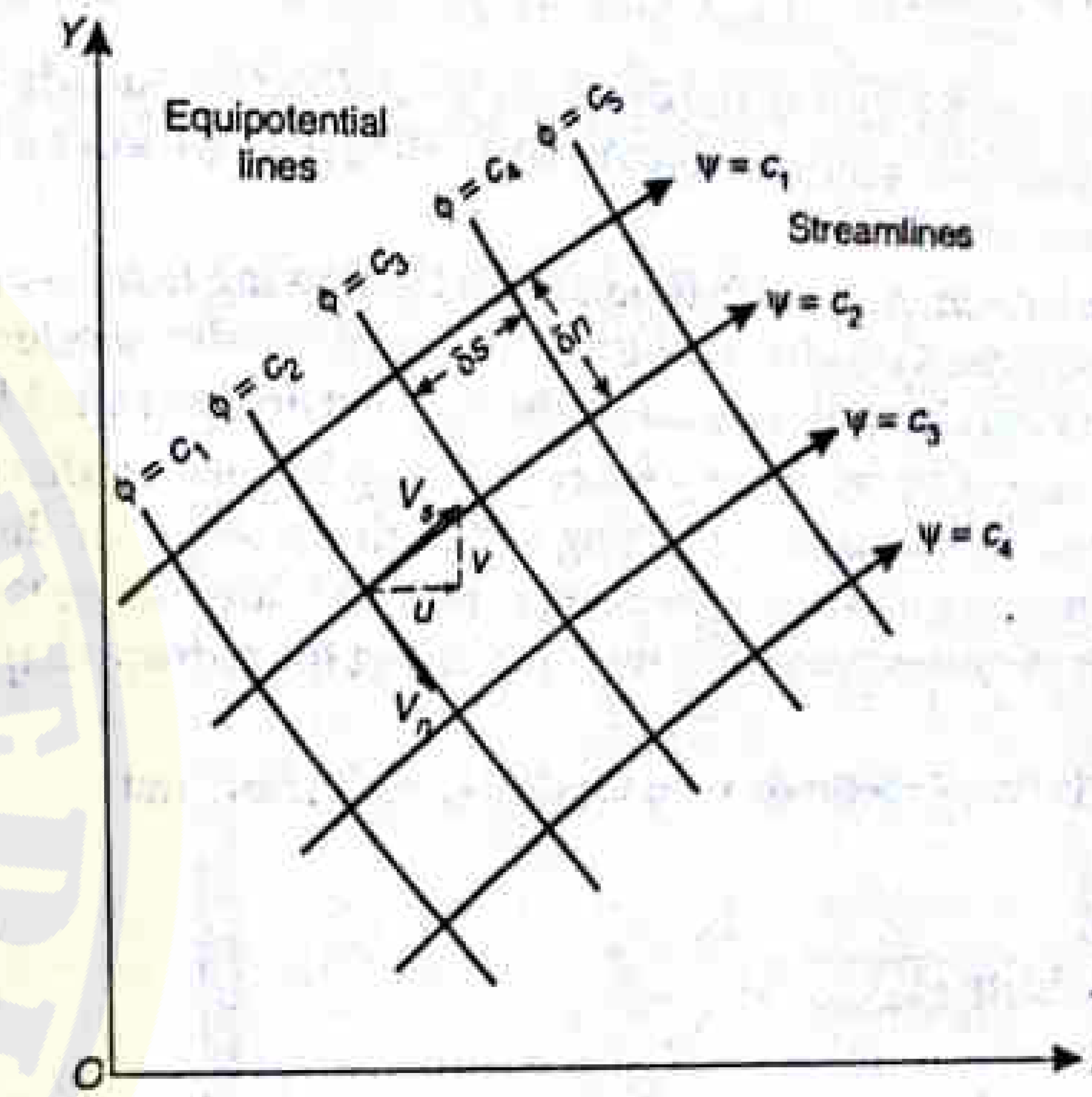


Figure 6.18 *Elements of a flow net*

But according to the definition of the velocity potential

$$-\frac{\partial \phi}{\partial n} = V_n$$

$$\therefore \quad V_n = 0$$

Further
$$-\frac{\partial \phi}{\partial s} = V_s$$

The relations therefore indicate that there is no flow along the direction tangential to the equipotential lines, but the flow always takes place in the direction at right angles to the equipotential lines.

Similarly at the point shown in Fig. 6.18, by definition

$$\frac{\partial \psi}{\partial s} = V_n = 0$$

and
$$\frac{\partial \psi}{\partial n} = -V_s$$

These relations also indicate that there is no flow in the direction normal to the streamlines but the flow is always along the direction tangential to the streamlines.

Moreover, since the flow is along the direction tangential to the streamlines, the space between two adjacent streamlines may be considered as a flow channel and the discharge flowing through it is proportional to $(\psi_2 - \psi_1)$, where ψ_1 and ψ_2 are the values of the stream functions for these two adjacent streamlines.

6.13 METHODS OF DRAWING FLOW NETS

The construction of the flow net for the flows is restricted by certain conditions which are as indicated below :

(i) The flow should be steady. This is so because it is only for the steady flow the streamline pattern will remain constant. For unsteady flow the streamline pattern will be instantaneous and it may change from instant to instant.

(ii) The flow should be irrotational, which is possible if the flowing fluid is an ideal fluid (having no viscosity) or it has negligible viscosity. However, in case of rapidly accelerating or converging flow of fluids, even if the fluids have low viscosity, the flow net analysis may be adopted.

(iii) The flow is not governed by the gravity force. This is so because under the action of gravity the shape of the free surface is constantly changing and with the shape of the extreme boundary surface (free surface in this case) undergoing a change, no fixed flow net pattern can be obtained. However, in such cases the flow nets can be drawn after fixing the correct shape of the free surface boundary.

The following are the different methods used for drawing the flow nets :

(1) Analytical Method.

(2) Graphical Method.

(3) Electrical Analogy Method.

(4) Relaxation Method.

(5) Hele Shaw or Viscous flow Analogy Method.

Out of these five methods, the first three methods are briefly described below.

Analytical Method. In this method the equations corresponding to the curves ϕ and ψ are first obtained and the same are plotted to give the flow net pattern for the flow of fluid between the given boundary shapes. In other words, this method involves a solution of Laplace equation for ϕ and ψ, which gives the corresponding equations for ϕ and ψ. But this method cannot be applied in various cases on account of the boundary shapes being such that it may not be possible to obtain the solution of the Laplace equation for ϕ and ψ. In such cases, other methods may be adopted to obtain the flow net pattern for the corresponding flow.

Graphical Method. The graphical method may be used to draw a flow net for the flow of fluid between the boundaries of any shape. The fixed solid boundaries correspond to streamlines, since they have no flow across them. In between these extreme streamlines (i.e., fixed boundaries) a number of other streamlines are suitably sketched by guess work. A set of smooth equipotential lines is then drawn so as to intersect the streamlines (including the fixed boundaries) perpendicularly and so spaced that streamlines and equipotential lines form approximate squares throughout the entire network. Successive adjustments of both the streamlines and the equipotential lines are then made to fulfil the above requirements. When the complete flow net or any part of it appears to consist of 'squares', a check can be made by drawing both diagonals of each 'square'. These diagonals should

also form continuous curves and should themselves intersect to form 'squares'. Another check which may also be used to test the accuracy of the flow net is that in each 'square' the sides are tangent to a common circle inscribed in it.

Electrical Analogy Method. It is a practical method of drawing a flow net for a particular set of boundaries, which is based on the fact that the flow of fluids and the flow of electricity through a conductor are analogous. These two systems are similar in this respect that the electric potential is analogous to the velocity potential, the electric current is analogous to the velocity of flow, and the homogeneous conductor is analogous to the homogeneous fluid.

As shown in Fig. 6.19, the fixed boundaries of the model are formed out of strips of non-conducting material mounted on a flat non-conducting surface. The end equipotential lines are formed out of conducting strips e.g. copper or brass. An electrolyte (conducting liquid) is placed at uniform depth

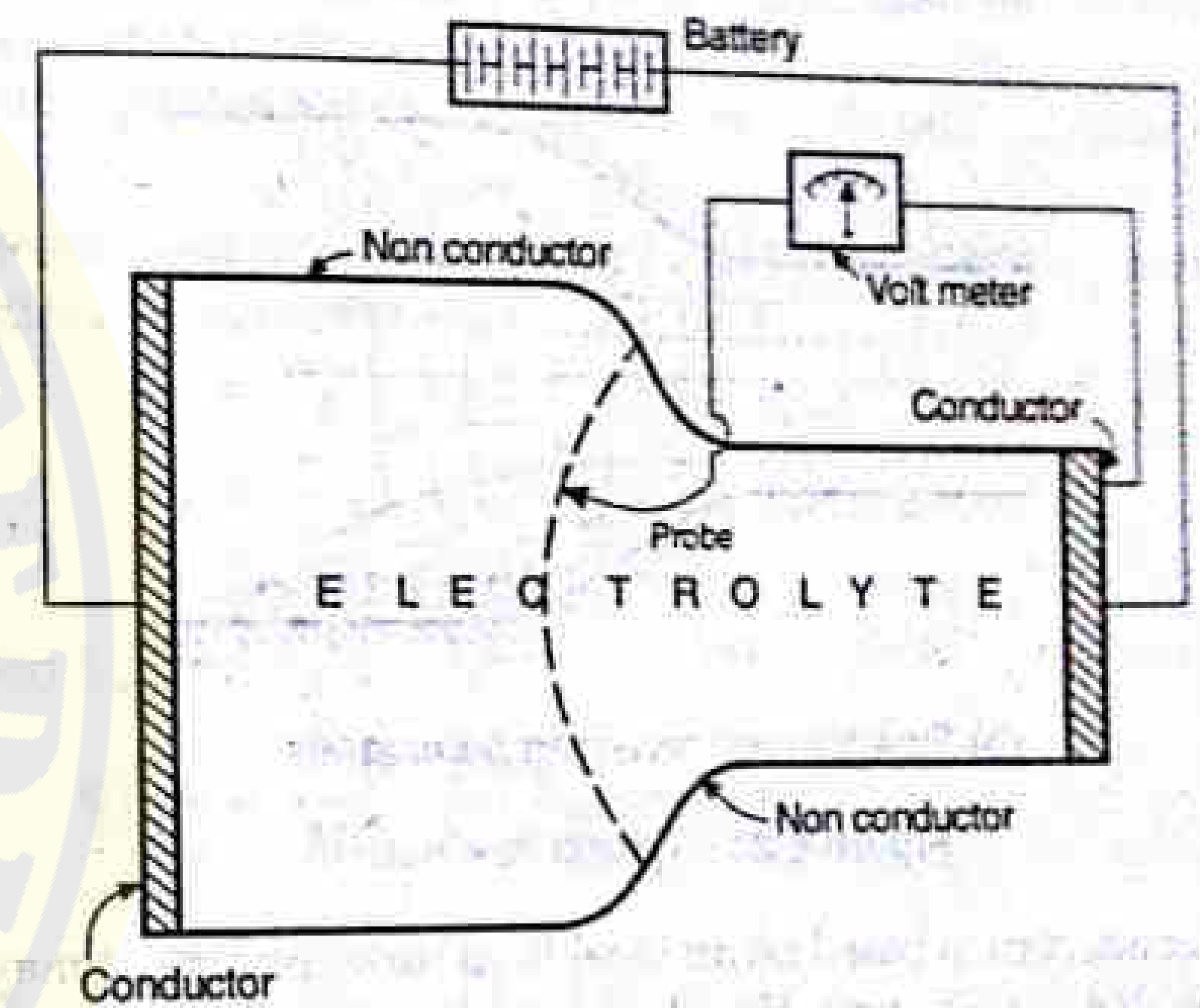


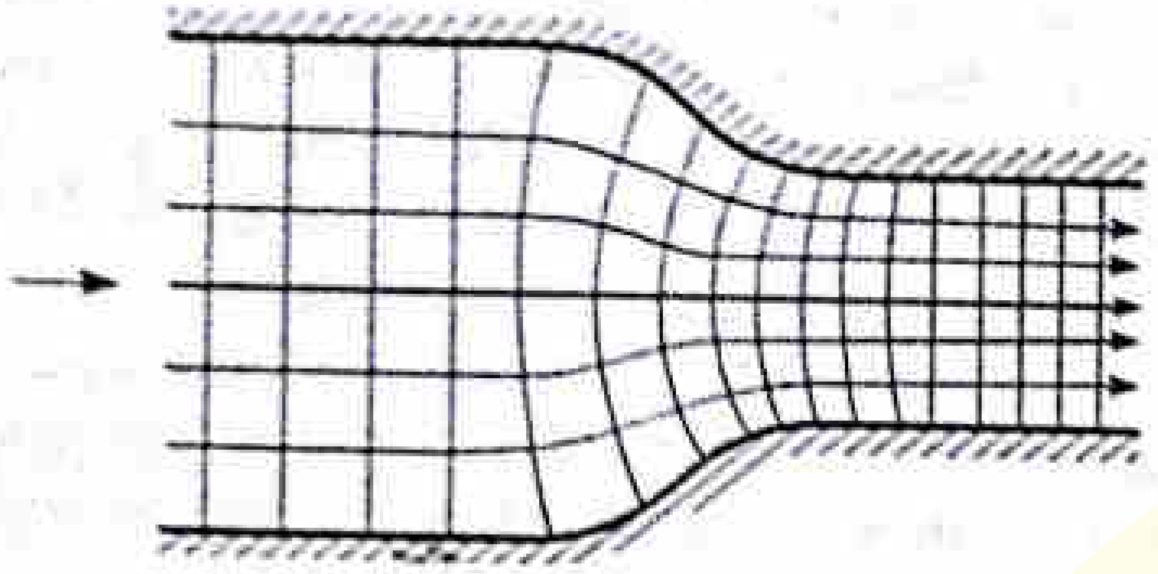
Figure 6.19 *Electrical analogy circuit*

in the flow space. A voltage potential is applied to the two ends of the conducting strips by means of a battery system or by connecting them to the electric mains. By means of a probe and a voltmeter, lines with constant drop in voltage from one end are located which may be plotted. These are equipotential lines. Once the exact pattern of equipotential lines are obtained, the streamlines can be drawn by graphical method. However, by reversing the process and making the flow boundaries out of the conducting material and the end equipotential lines from non-conducting material, the streamlines can also be sketched by this method.

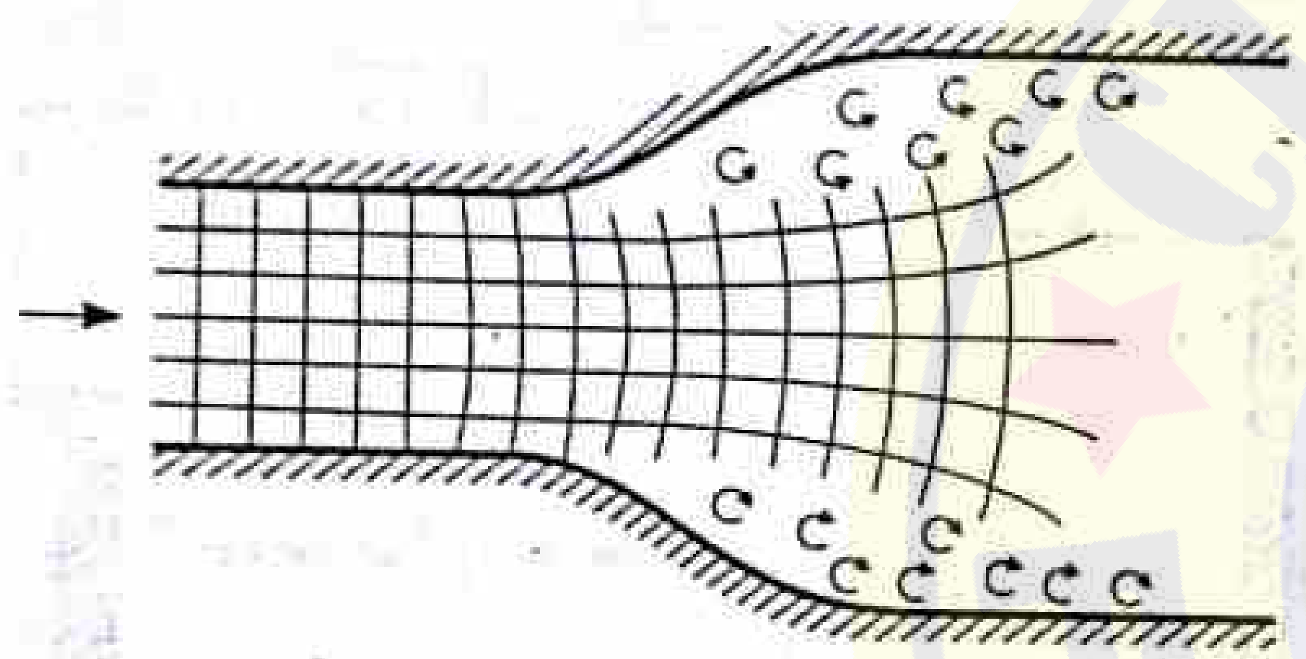
6.14 USE OF THE FLOW NET

For a given set of boundary configuration there is only one possible pattern of the flow of an ideal fluid, and a correctly drawn flow net will represent this pattern. As such after a flow net for a given boundary configuration has been obtained, it may be used for all irrotational flows with geometrically similar boundaries. Once the flow net is drawn, the spacing between the adjacent streamlines is determined and the application of the continuity equation gives the velocity of flow at any point, if the velocity of flow at any reference point is known.

Further the flow net analysis assists in the determination of the efficient boundary shapes, for which the flow does not separate from the boundary surface.



(a) Flow through converging boundaries



(b) Flow through diverging boundaries

Figure 6.20 *Typical flow nets*

Although the flow net analysis is based on an ideal fluid flow concept, it may also be applied to the flow of a real fluid within certain limit. The ideal fluid theory neglects the effect of fluid friction or viscosity, which is however possessed by a real fluid. But the viscosity effects of a real fluid are most pronounced at or near a solid boundary and diminish rapidly with distance from the boundary. As such in the regions where viscosity effects are not predominant, the real fluid behaves more or less like an ideal fluid, and in these regions the flow net analysis may be applied to the real fluids with sufficient accuracy.

Figure 6.20 shows a few examples of the flow nets. It is observed in Fig. 6.20 (a) that since the boundaries are converging the streamlines also converge rapidly. In this case the accelerating flow is developed and the actual flow pattern approximates closely to that represented by the flow net. On the other hand as observed in Fig. 6.20 (b) the boundaries in the direction of flow are diverging and therefore the streamlines also tend to diverge. In the region where the streamlines diverge, a phenomenon known as separation of flow generally occurs. That is, in such cases the flowing fluid does not remain in contact with the boundary surface or it separates from the boundary, thereby developing regions of flow separation in which eddies are developed. Therefore, in such cases where the flow separation takes place, the flow net which is constructed with streamlines conforming to the boundaries does not describe the actual pattern of flow field.

6.15 LIMITATIONS OF FLOW NET

A flow net always indicates some velocity at the boundary, but a real fluid must have zero velocity adjacent to the boundary on account of the fluid friction or viscosity. As such the flow net analysis cannot be applied in the region close to the boundary where the effects of viscosity are predominant. As stated earlier the flow net analysis can also not be applied to a sharply diverging flow, since the actual flow pattern is not represented by the flow net. Further in the case of flow of a fluid past a solid body, while the flow net gives a fairly accurate picture of the flow pattern for the upstream part of the solid body, it can give little information concerning the flow conditions at the rear because of separation and eddies. The disturbed flow in the rear of the solid body is known as wake, the formation of which is not indicated by a flow net.

ILLUSTRATIVE EXAMPLES

Example 6.1. *An airplane is observed to travel due north at a speed of 240 km per hour in a 80 km per hour wind from the north-west. What is the apparent wind velocity observed by the pilot ?*

Solution

Let V_R be the apparent wind velocity observed by the pilot. Now if V_W and V_P are the velocities of the wind and the plane respectively, then

$$V_W = V_R + \rightarrow V_P$$

$$\therefore \quad V_R = \sqrt{(240+80\sin 45°)^2+(80\cos 45°)^2}$$

$$= 301.9 \text{ km/hour}$$

If the angle between V_R and V_P is θ, then

$$\tan\theta = \frac{80\cos 45°}{240+80\sin 45°} = 0.191$$

$$\therefore \quad \theta = 10.8°$$

Thus the apparent wind direction is N 10.8° W.

Example 6.2. *When 2500 litres of water flows per minute through a 0.3 m diameter pipe which later reduces to a 0.15 diameter pipe, calculate the velocities of flow in the two pipes.*

Solution

From Eq. 6.24 discharge

$$Q = A_1V_1 = A_2V_2$$

$$Q = \frac{2500}{60\times 10^3} = 0.042 \text{ m}^3\text{/s}$$

$$A_1 = \frac{\pi}{4}\times(0.3)^2 = 0.0707 \text{ m}^2$$

$$A_2 = \frac{\pi}{4}\times(0.15)^2 = 0.0177 \text{ m}^2$$

Thus

$$V_1 = \frac{Q}{A_1} = \frac{0.042}{0.0707} = 0.59 \text{ m/s}$$

and $$V_2 = \frac{Q}{V_2}\frac{0.042}{0.0177} = 2.37 \text{ m/s}$$

Example 6.3. *Two velocity components are given in the following cases, find the third component such that they satisfy the continuity equation.*

(a) $u = x^3 + y^2 + 2z^2;\ v = -x^2y - yz - xy;$

(b) $u = \log(y^2 + z^2)\ ;\ v = \log(x^2 + z^2);$

(c) $u = \dfrac{-2xyz}{(x^2+y^2)^2}\ ;\ w = \dfrac{y}{(x^2+y^2)}$

Solution

For an incompressible fluid the equation of continuity is

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

(a) In this case

$$u = x^3 + y^2 + 2z^2$$

$$\therefore \quad \frac{\partial u}{\partial x} = 3x^2$$

and $$v = -x^2y - yz - xy$$

$$\frac{\partial v}{\partial y} = -x^2 - z - x$$

$\therefore$ By substitution in the equation of continuity, we get

$$3x^2 - x^2 - z - x + \frac{\partial w}{\partial z} = 0$$

or $$\frac{\partial w}{\partial z} = x + z - 2x^2$$

or $$\partial w = (x + z - 2x^2)\partial z$$

By integrating both sides, we get

$$w = \left(xz + \frac{z^2}{2} - 2x^2z\right) + \text{constant of integration}$$

The constant of integration could be a function of x and y, that is $f(x, y)$. Hence the third component is

$$w = \left(xz + \frac{z^2}{2} - 2x^2z\right) + f(x, y).$$

(b) In this case

$$u = \log(y^2 + z^2)$$

$$\therefore \quad \frac{\partial u}{\partial x} = 0$$

Also $$v = \log(x^2 + z^2)$$

$$\therefore \quad \frac{\partial v}{\partial y} = 0$$

By substitution in the equation of continuity, we get

$$\frac{\partial w}{\partial z} = 0$$

By integration, we get

$$w = f(x, y)$$

By symmetry one of the values of the third component can be

$$w = \log(x^2 + y^2)$$

(c) In this case

$$u = \frac{-2xyz}{(x^2+y^2)^2}$$

$$\therefore \quad \frac{\partial u}{\partial x} = \frac{(x^2+y^2)^2(-2yz) - (-2xyz)\times 2(x^2+y^2)\times 2x}{(x^2+y^2)^4} = \frac{6x^2yz - 2y^3z}{(x^2+y^2)^3}$$

Also $$w = \frac{y}{(x^2+y^2)}$$

$$\therefore \quad \frac{\partial w}{\partial z} = 0$$

By substituting in the continuity equation, we get

$$\frac{6x^2yz - 2y^3z}{(x^2+y^2)^3} + \frac{\partial v}{\partial y} + 0 = 0$$

or $$\frac{\partial v}{\partial y} = \frac{2y^3z - 6x^2yz}{(x^2+y^2)^3}$$

or $$\partial v = \frac{2y^3z - 6x^2yz}{(x^2+y^2)^3}\partial y$$

By integrating both sides, we get

$$v = \frac{z(x^2 - y^2)}{(x^2+y^2)^2} + f(x, z).$$

Example 6.4. *The velocity components in a two-dimensional flow field for an incompressible fluid are expressed as* $u = \frac{y^3}{3} + 2x - x^2y$; $v = xy^2 - 2y - \frac{x^3}{3}$

(*a*) *Show that these functions represent a possible case of an irrotational flow.*

(*b*) *Obtain an expression for stream function* ψ.

(*c*) *Obtain an expression for velocity potential* ϕ.

Solution

(a) The partial derivatives of the given functions are

$$\frac{\partial u}{\partial x} = 2 - 2xy \ ; \ \frac{\partial v}{\partial y} = 2xy - 2$$

For a two-dimensional flow of incompressible fluid the continuity equation may be expressed as

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

Thus by substitution, we get

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + = 2 - 2xy + 2xy - 2 = 0$$

Therefore the functions represent a possible case of fluid flow.

The rotation component ω_z of any fluid element in the flow field is

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

Now $$\frac{\partial v}{\partial x} = (y^2 - x^2)$$

and $$\frac{\partial u}{\partial y} = (y^2 - x^2)$$

Hence by substitution, we get

$$\omega_z = [(y^2 - x^2) - (y^2 - x^2)] = 0$$

which shows that the given functions represent an irrotational flow.

(b) From Eq. 6.41

$$\frac{\partial \psi}{\partial x} = v = xy^2 - 2y - \frac{x^3}{3} \quad ...(i)$$

and $$\frac{\partial \psi}{\partial y} = -u = -\left(\frac{y^3}{3} + 2x - x^2y\right) \quad ...(ii)$$

Integrating Eq. (i), we get

$$\psi = \frac{x^2y^2}{2} - 2xy - \frac{x^4}{12} + f(y) \quad ...(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \psi}{\partial y} = x^2y - 2x + f'(y) \quad ...(iv)$$

Equating the values of $\left(\frac{\partial \psi}{\partial y}\right)$ from Eqs (ii) and (iv), we get

$$-\left(\frac{y^3}{3} + 2x - x^2y\right) = x^2y - 2x + f'(y)$$

or $$f'(y) = -\frac{y^3}{3}$$

By integrating both sides of the above expression, we get

$$f(y) = -\frac{y^4}{12} + C$$

where C is a constant of integration which is a numerical constant only. Therefore

$$\psi = \frac{x^2y^2}{2} - 2xy - \frac{x^4}{12} - \frac{y^4}{12} + C$$

Since C is numerical constant, it may also be considered as zero, in which case

$$\psi = \frac{x^2y^2}{2} - 2xy - \frac{x^4}{12} - \frac{y^4}{12}$$

(c) From Eq. (6.39)

$$-\frac{\partial \phi}{\partial x} = u = +2x - x^2y \quad ...(i)$$

$$-\frac{\partial \phi}{\partial y} = v = xy^2 - 2y - \frac{x^3}{3} \quad ...(ii)$$

Integrating Eq. (i), we get

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3y}{3} + f(y) \quad ...(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \phi}{\partial y} = -xy^2 + \frac{x^3}{3} + f'(y) \quad ...(iv)$$

Equating the values of $\left(\frac{\partial\phi}{\partial y}\right)$ from Eqs (ii) and (iv), we get

$$-xy^2 + 2y + \frac{x^3}{3} = -xy^2 + \frac{x^3}{3} + f'(y)$$

or

$$f'(y) = 2y$$

By integrating both sides of the above expression, we get

$$f(y) = y^2 + C$$

where C is a numerical constant of integration.

Therefore

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3y}{3} + y^2 + C$$

Again since C is a numerical constant, it may be considered as zero and hence

$$\phi = -\frac{xy^3}{3} - x^2 + \frac{x^3y}{3} + y^2.$$

Example 6.5. *For a three-dimensional flow field described by*

$$V = (y^2 + z^2)\,\mathbf{i} + (x^2 + z^2)\,\mathbf{j} + (x^2 + y^2)\,\mathbf{k}$$

find at (1, 2, 3) (i) the components of acceleration, (ii) the components of rotation.

Solution

The components of velocity u, v, and w are given by

$$u = y^2 + z^2; \qquad v = x^2 + z^2; \qquad w = x^2 + y^2$$

$$\frac{\partial u}{\partial x} = 0 \qquad \frac{\partial v}{\partial x} = 2x \qquad \frac{\partial w}{\partial x} = 2x$$

$$\frac{du}{\partial y} = 2y \qquad \frac{\partial v}{\partial y} = 0 \qquad \frac{\partial w}{\partial y} = 2y$$

$$\frac{\partial u}{\partial z} = 2z \qquad \frac{\partial v}{\partial z} = 2x \qquad \frac{\partial w}{\partial z} = 0$$

(i) Introducting the above expressions in Eqs 6.27, 6.28 and 6.29 the components of acceleration are obtained as follows.

$$a_x = 2y\,(x^2 + z^2) + 2z\,(x^2 + y^2) = 70$$
$$a_y = 2x\,(y^2 + z^2) + 2z\,(x^2 + y^2) = 56$$
$$a_z = 2x\,(y^2 + z^2) + 2y\,(x^2 + x^2) = 66$$

(ii) Introducting the above expression in Eq. 6.33 the components of rotation are obtained as follows.

$$\omega_x = \frac{1}{2}(2y - 2z) = -1 \text{ rad/s}$$

$$\omega_y = \frac{1}{2}(2z - 2x) = 2 \text{ rad/s}$$

$$\omega_z = \frac{1}{2}(2x - 2y) = -1 \text{ rad/s}$$

Example 6.6. *For a two dimensional flow $\phi = 3xy$ and $\psi = \frac{3}{2}(y^2 - x^2)$. Determine the velocity components at the points (1, 3) and (3, 3). Also find the discharge passing between the streamlines passing through the points given above.*

Solution

From Eq. 6.39, we have

$$u = -\frac{\partial\phi}{\partial x}; \text{ and } v = -\frac{\partial\phi}{\partial y}$$

Since $\phi = 3xy$, we obtain

$$u = -3y; \text{ and } v = -3x$$

Thus at point (1, 3) velocity components are

$$u = -9; \text{ and } v = -3$$

and at point (3, 3), velocity components are

$$u = -9; \text{ and } v = -9$$

Alternatively from Eq. 6.41, we have

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

Since $\psi = \frac{3}{2}(y^2 - x^2)$, we obtain

$$u = -3y; \text{ and } v = -3x$$

which are same as obtained above.

The value of ψ for streamline passing through point (1, 3) is

$$\psi_1 = 12$$

Similarly the value of ψ for streamline passing through point (3, 3) is

$$\psi_2 = 0$$

$\therefore$ Discharge passing between these two streamlines is

$$(\psi_1 - \psi_2) = (12 - 0) = 12$$

Example 6.7. *The stream function $\psi = 4xy$ in which y is in cm^2 per second and x and y are in metres describe the incompressible flow between the boundary shown below.*

Calculate

(i) Velocity at B.

(ii) Convective acceleration at B.

(iii) Flow per unit width across AB.

Solution

The stream function is given as

$$\psi = 4xy$$

in which ψ is in cm^2/s and x and y are in m. Thus 4 is a dimensional constant having its dimensions as $\left(\frac{cm^2}{s \times m^2}\right)$.

From Eq. 6.41, we have

$$\frac{\partial \psi}{\partial x} = v = 4y$$

and

$$\frac{\partial \psi}{\partial y} = -u = 4x$$

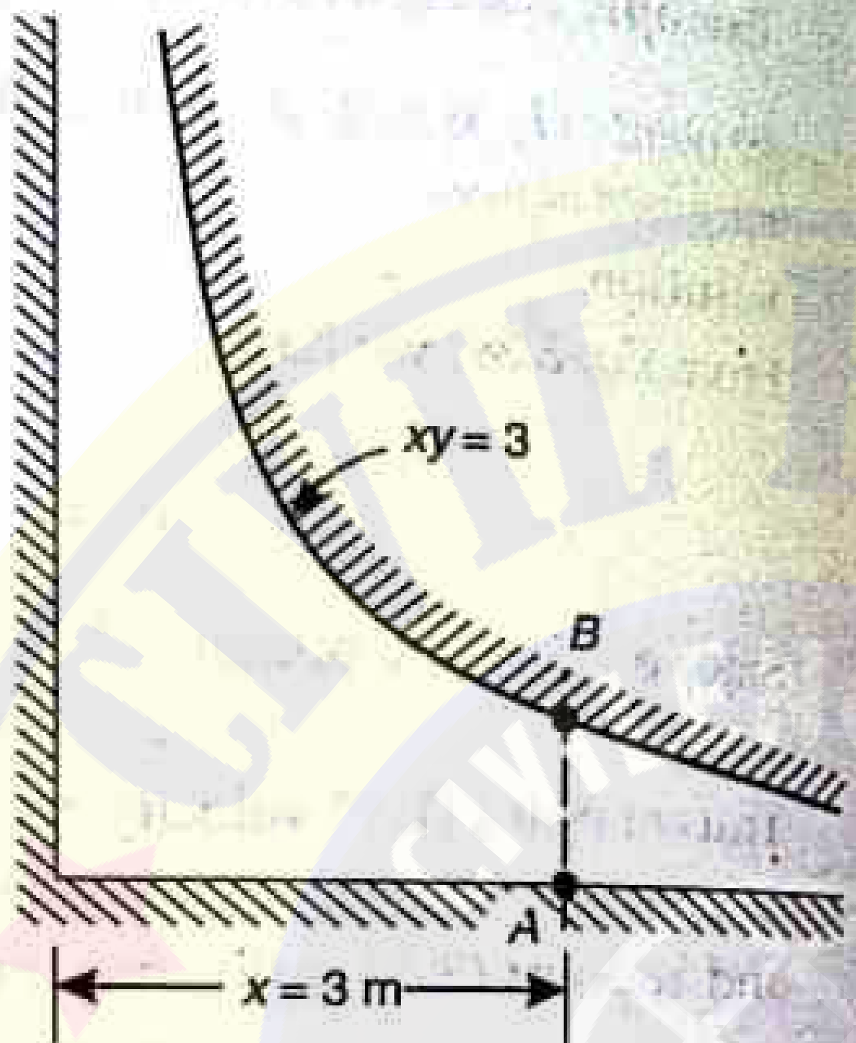


Figure Ex. 6.7

(i) For point B, $x = 3$ m and $y = 1$ m, and hence at this point

$$u = -\frac{4\times 3}{100} = -0.12 \text{ cm/s.}$$

and

$$v = \frac{4\times 1}{100} = 0.04 \text{ cm/s}$$

$\therefore$ Velocity at point B is

$$V = \sqrt{u^2+v^2}$$

$$= \sqrt{(-0.12)^2+(0.04)^2} = 0.126 \text{ cm/s}$$

(ii) For two dimensional flow the components of convective acceleration are

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

and

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

$$\frac{\partial u}{\partial x} = -4\,;\ \frac{\partial u}{\partial y} = 0\,;\ \frac{\partial v}{\partial x} = 0 \text{ and } \frac{\partial v}{\partial x} = 4$$

Thus at point B, we have

$$a_x = -0.12\times(-4)\times 10^{-4} \text{ cm/s}^2$$
$$= 48 \times 10^{-6} \text{ cm/s}^2$$

and

$$a_y = 0.04\times 4\times 10^{-4} \text{ cm/s}^2$$
$$= 16\times 10^{-6} \text{ cm/s}^2$$

$\therefore$ Convective acceleration at B is

$$a_x = \sqrt{a_x^2 + a_y^2}$$

$$= \sqrt{(48\times 10^{-6})^2 + (16\times 10^{-6})^2}$$

$$= 50.6\times 10^{-6} \text{ cm/s}^2$$

(iii) The flow rate per unit width across AB is given by

$$Q = u\times AB\times 1$$

$$u = -0.12 \text{ cm/s} = -0.12\times 10^{-2} \text{m/s; and } AB = 1\text{m}$$

$$\therefore \quad Q = -0.12\times 10^{-2}\times 1\times 1 \text{ m}^3/\text{s}$$

$$= -0.12\times 10^{-2} \text{m}^3/\text{s}$$

$$= -1200 \text{ cm}^3/\text{s}$$

Note: The negative sign simply indicates the flow in the negative x direction. However, the flow rate will be the same if it is in the positive x direction.

Alternatively the flow rate per unit width across AB is also given by

$$Q = (\psi_2 - \psi_1)\times 1$$

where ψ_2 is the value of the stream function for the curved boundary through point B; and ψ_1 is the value of the stream function for the straight horizontal boundary through point A

$$\psi_2 = 12 \text{ cm}^2\text{/s; and } \psi_1 = 0$$

$$\therefore \quad Q = 12\times 1\times 100 \text{ cm}^3\text{/s}$$

$$= 1200 \text{ cm}^3/\text{s}$$

Example 6.8. *The velocity components of the two-dimensional plane motion of a fluid are*

$$u = \frac{y^2 - x^2}{(x^2+y^2)^2} \text{ and } v = -\frac{2xy}{(x^2+y^2)^2}$$

(a) *Show that the fluid is incompressible and flow is irrotational.*

(b) *Show that the points (2, 2) and $(1, 2-\sqrt{3})$ are located on the same streamline.*

(c) *Determine the discharge across a line joining point (1, 1) and (2, 2) given that the thickness of the fluid stream normal to the $x-y$ plane is t.*

Solution

(a) For two dimensional flow of incompressible fluid the continuity equation may be expressed as

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

$$u = \frac{y^2-x^2}{(x^2+y^2)^2}$$

$$\therefore \quad \frac{\partial u}{\partial x} = \frac{(-2x)(x^2+y^2)^2-(y^2-x^2)\times 2(x^2+y^2)\times 2x}{(x^2+y^2)^4}$$

$$= \frac{-2x(x^2+y^2)\ (x^2+y^2+2y^2-2x^2)}{(x^2+y^2)^4} = \frac{-2x(3y^2-x^2)}{(x^2+y^2)^3}$$

$$v = -\frac{2xy}{(x^2+y^2)^2}$$

$$\therefore \quad \frac{\partial v}{\partial y} = \frac{(-2x)(x^2+y^2)^2-(-2xy)\times 2(x^2+y^2)\times 2y}{(x^2+y^2)^4}$$

$$= \frac{-2x(x^2+y^2)\ (x^2+y^2-4y^2)}{(x^2+y^2)^4} = \frac{-2x(x^2-3y^2)}{(x^2+y^2)^3}$$

Introducing these values in the continuity equation, we get

$$\frac{-2x(3y^2-x^2+x^2-3y^2)}{(x^2+y^2)^3} = 0$$

which indicates that the fluid is incompressible

For two-dimensional flow in $x-y$ plane to be irrotational the condition to be satisfied is

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

$$\frac{\partial v}{\partial x} = \frac{(-2y)\ (x^2+y^2)^2-(-2xy)\times 2(x^2+y^2)\times 2x}{(x^2+y^2)^4}$$

$$= \frac{-2y(x^2+y^2)\ (x^2+y^2-4x^2)}{(x^2+y^2)^4}$$

$$= \frac{-2y(y^2-3x^2)}{(x^2+y^2)^3}$$

$$= \frac{2y(3x^2-y^2)}{(x^2+y^2)^3}$$

$$\frac{\partial u}{\partial y} = \frac{2y(x^2+y^2)^2-(y^2-x^2)\times 2(x^2+y^2)\times 2y}{(x^2+y^2)^4}$$

$$= \frac{2y(x^2+y^2)\ (x^2+y^2-2y^2+2x^2)}{(x^2+y^2)^4}$$

$$= \frac{2y(3x^2-y^2)}{(x^2+y^2)^3}$$

Since in this case $\frac{\partial v}{\partial x}=\frac{\partial u}{\partial y}$, the flow is irrotational.

(b) The velocity components may be expressed in terms of stream function ψ as indicated below.

$$\frac{\partial \psi}{\partial x} = v = -\frac{2xy}{(x^2+y^2)^2} \quad \text{...(i)}$$

and

$$\frac{\partial \psi}{\partial y} = -u = -\frac{y^2-x^2}{(x^2+y^2)^2} \quad \text{...(ii)}$$

Integrating Eq. (i), we get

$$\psi = \frac{y}{(x^2+y^2)}+f(y) \quad \text{...(iii)}$$

Differentating Eq. (iii) with respect to y, we get

$$\frac{\partial \psi}{\partial y} = \frac{(x^2+y^2)-y\times 2y}{(x^2+y^2)^2}+f'(y)$$

or

$$\frac{\partial \psi}{\partial y} = \frac{x^2-y^2}{(x^2+y^2)^2}+f'(y) \quad \text{...(iv)}$$

Equating the values of $(\partial\psi/\partial y)$ given by Eqs (ii) and (iv), we get

$$f'(y) = 0$$

or $f(y)$ = constant, which may be taken as 0

$$\therefore \quad \psi = \frac{y}{(x^2+y^2)}$$

At point (2, 2) the value of ψ is obtained as

$$\psi = \frac{2}{(2)^2+(2)^2}=\frac{1}{4}$$

and at point $(1,\ 2-\sqrt{3})$ the value of ψ is obtained as

$$\psi = \frac{(2-\sqrt{3})}{(1)^2+(2-\sqrt{3})^2} = \frac{1}{4}$$

Since the value of ψ is same at the points (2, 2) and $(1,\ 2-\sqrt{3})$, both the points are located on the same streamline.

(c) For streamline passing through point (1, 1), the value of ψ is obtained as

$$\psi_1 = \frac{1}{2}$$

Similarly for streamline passing through point (2, 2), the value of ψ is obtained as

$$\psi_2 = \frac{1}{4}$$

$\therefore$ Discharge across a line joining points (1, 1) and (2, 2) is

$$Q = \psi_1 - \psi_2 \times t = \left(\frac{1}{2}-\frac{1}{4}\right)\times t = \frac{t}{4}$$

Example 6.9. *A two-dimensional incompressible flow field is described by equation (a) $V = (C/r)$; (b) $V = Cr$, in which V is the tangential velocity at a radius r and C is a constant. Determine for each case the circulation (i) around a circle of radius R, (ii) around a closed path formed by the arcs of two circles of radii R_1 and R_2 and the two radius vectors with an angle θ between them.*

Also calculate the vorticities of the flows described by these equations.

Solution

(i) From Eq. 6.35 circulation around a circle of radius R is

$$\Gamma = \int_0^{2\pi} VR\,d\theta = (2\pi R)V$$

(a) For $r = R$; $\quad V = (C/R)$

$\therefore \quad \Gamma = 2\pi C$

(b) For $r = R$; $\quad V = CR$

$\therefore \quad \Gamma = 2\pi CR^2$

(ii) Let $ABCD$ be the closed path formed by the arcs of two circles of radii R_1 and R_2 and the two radius vectors with an angle θ between them, then $AB = R_1\theta$ and $CD = R_2\theta$. The total circulation around the path $ABCD$ is given by

$\therefore \quad \Gamma = \Gamma_{AB} + \Gamma_{BC} + \Gamma_{CD} + \Gamma_{DA}$

(a) $\Gamma_{AB} = 2\pi C$; $\quad \Gamma_{CD} = -2\pi C$; $\Gamma_{BC} = \Gamma_{DA} = 0$

$\therefore \quad \Gamma = 0$

(b) $\Gamma_{AB} = 2\pi CR_1^2$; $\quad \Gamma_{CD} = -2\pi CR_2^2$; $\Gamma_{BC} = \Gamma_{DA} = 0$

$$\Gamma = 2\pi C\left(R_1^2 - R_2^2\right)$$

In cylindrical polar coordinates the vorticity may be expressed as

$$\zeta = \frac{\partial(rV_\theta)}{r\partial r} - \frac{\partial V_r}{r\partial\theta}$$

in which V_θ and V_r are the velocity components in the tangential and radial directions respectively.

(a) $\quad V_\theta = V = (C/r); V_r = 0$

$\therefore \quad \zeta = 0$

(b) $\quad V_\theta = V = Cr; V_r = 0$

$\therefore \quad \zeta = 2C$

Note: The flow fields represented by case (a) and (b) correspond to free vortex motion and forced vortex motion respectively, both of which have been described in the next chapter. It may be noted that in the case of free vortex motion although the circulation is not equal to zero, the vorticity is equal to zero. This fact has been explained in the next chapter.

Example 6.10. *Derive the equation of stream function and velocity potential for a uniform stream of velocity V in a two-dimensional field, the velocity V being inclined to the x-axis at a positive angle α.*

Solution

The components of the velocity V in the x and y directions are

$$u = V\cos\alpha \text{ and } v = V \sin\alpha$$

From Eq. (6.41)

$$\frac{\partial\psi}{\partial x} = v = V\sin\alpha \quad \text{...(i)}$$

and

$$\frac{\partial\psi}{\partial y} = -u = -V\cos\alpha \quad \text{...(ii)}$$

Integrating Eq. (i), we get

$$\psi = (V\sin\alpha)\,x + f(y) \quad \text{...(iii)}$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial\psi}{\partial y} = f'(y) \quad \text{...(iv)}$$

Equating the values of $\frac{\partial\psi}{\partial y}$ from Eqs (ii) and (iv), we get

$$f'(y) = -V\cos\alpha$$

By integrating both sides of the above expression, we get

$$f(y) = (-V\cos\alpha)\,y + C$$

where C is a constant of integration.

Therefore

$$\psi = (V\sin\alpha)\,x - (V\cos\alpha)\,y + C \quad \text{...(i}$$

From Eq. (6.39)

$$-\frac{\partial\phi}{\partial x} = u = V\cos\alpha$$

and $$-\frac{\partial \phi}{\partial y} = v = V \sin\alpha \quad ...(ii)$$

Integrating Eq. (i), we get

$$f = (-V\cos\alpha)\,x + f(y) \quad ...(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \phi}{\partial y} = f'(y) \quad ...(iv)$$

Equating the values of $\left(\frac{\partial \phi}{\partial y}\right)$ from Eqs (ii) and (iv), we get $f'(y) = -V\sin\alpha$.

By integrating both sides of the above expression, we get

$$f(y) = (-V\sin\alpha)\,y + C$$

where C is a constant of integration.

Therefore

$$\phi = (-V\cos\alpha)\,x + (-V\sin\alpha)\,y + C.$$

Example 6.11. *A stream function is given by $\psi = 3x^2 - y^3$. Determine the magnitude of velocity components at the point (2, 1).*

Solution

From Eq. (6.41)

$$v = \frac{\partial \psi}{\partial x} = \frac{\partial}{\partial x}(3x^2 - y^3) = 6x$$

and $$u = -\frac{\partial \psi}{\partial y} = -\frac{\partial}{\partial y}(3x^2 - y^3) = 3y^2$$

At any point (2,1)

$$u = 3 \text{ and } v = 12$$

and the total velocity is the vector sum of two components. Thus

$$V = \sqrt{u^2 + v^2}$$

$$= \sqrt{(3)^2 + (12)^2} = 12.37$$

Example 6.12. *A stream function in a two-dimensional flow is $\psi = 2xy$. Show that the flow is irrotational and determine the corresponding velocity potential ϕ.*

Solution

From Eq. (6.45) a stream function ψ represents an irrotational flow if

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

By substitution

$$\left[\frac{\partial^2}{\partial x^2}(2xy) + \frac{\partial^2}{\partial y^2}(2xy)\right] = 0$$

Hence it represents an irrotational flow.

From Eq. 6.44

$$\frac{\partial \phi}{\partial x} = \frac{\partial \psi}{\partial y} = \frac{\partial}{\partial y}(2xy) = 2x \quad ...(i)$$

$$\frac{\partial \phi}{\partial y} = -\frac{\partial \psi}{\partial x} = -\frac{\partial}{\partial x}(2xy) = -2y \quad ...(ii)$$

Integrating Eq. (i), we get

$$\phi = x^2 + f(y) \quad ...(iii)$$

Differentiating Eq. (iii) with respect to y, we get

$$\frac{\partial \phi}{\partial y} = f'(y) \quad ...(iv)$$

Equating the values of $\left(\frac{\partial \phi}{\partial y}\right)$ from Eqs (ii) and (iv), we get

$$f'(y) = -2y$$

Integrating both the sides of the above expression, we get

$$f(y) = -y^2 + C$$

where C is the constant of integration.

Therefore

$$\phi = x^2 - y^2 + C.$$

Example 6.13. *A nozzle is so shaped that the velocity of flow along the centre line changes linearly from 1.5 m/s to 15 m/s in a distance of 0.375 m. Determine the magnitude of the convective acceleration at the beginning and end of this distance.*

Solution

The rate of change of V with respect to s is

$$\frac{\partial V}{\partial s} = \frac{15 - 1.5}{0.375} = 36 \text{ m/s per m}$$

Convective acceleration

$$a_s = V\frac{\partial V}{\partial s}$$

Therefore

$$a_{s_1} = (1.5 \times 36) = 54 \text{ m/s}^2$$

and $$a_{s_2} = (15 \times 36) = 540 \text{ m/s}^2$$

Example 6.14. *The bucket of a spillway has a radius of 6 m. When the spillway is discharging 5 cubic metre of water per second per metre length of crest, the average thickness of the sheet of water over the bucket is 0.4 m. Compare the resulting normal acceleration (or centripetal acceleration) with the acceleration due to gravity.*

Solution

$$V = \frac{q}{y} = \frac{5}{0.40} = 12.5 \text{ m/s}$$

$$a_n = \frac{V^2}{r} = \frac{(12.5)^2}{6} = 26.04 \text{ m/s}^2$$

$$\therefore \quad \frac{a_n}{g} = \frac{26.04}{9.81} = 2.65.$$

Example 6.15. *If for two-dimensional flow the stream function is given by $\psi = 2xy$, calculate the velocity at a point (3, 6). Show that the velocity potential ϕ exists for this case and deduce it. Also draw the streamlines corresponding to $\psi = 100$ and $\psi = 300$ and equipotential lines correspond to $\phi = 100$ and $\phi = 300$.*

Solution

$$\psi = 2xy$$

Since
$$\frac{\partial \psi}{\partial x} = v \text{ and } \frac{\partial \psi}{\partial y} = -u$$

$$\therefore \quad v = 2y \text{ and } u = -2x$$

Thus at point (3, 6)

$$v = 12 \text{ and } u = -6$$

$$\therefore \quad V = \sqrt{u^2 + v^2}$$

$$= \sqrt{(6)^2 + (12)^2} = 13.42$$

As shown in Ex. 6.8 the velocity potential exists for this case and its value is

$$\phi = x^2 - y^2.$$

Thus considering the following equations corresponding to the given values of ψ and ϕ viz.,

$$2xy = 100$$

or
$$xy = 50 \quad ...(i)$$

$$2xy = 300$$

or
$$xy = 150 \quad ...(ii)$$

$$x^2 - y^2 = 100 \quad ...(iii)$$

$$x^2 - y^2 = 300 \quad ...(iv)$$

the streamlines and equipotential lines may be plotted as shown in Fig. Ex. 6.15 by means of firm lines. In the same figure the streamlines and equipotential lines shown by means of dotted lines have been drawn for symmetry by considering the negative values of the given constants.

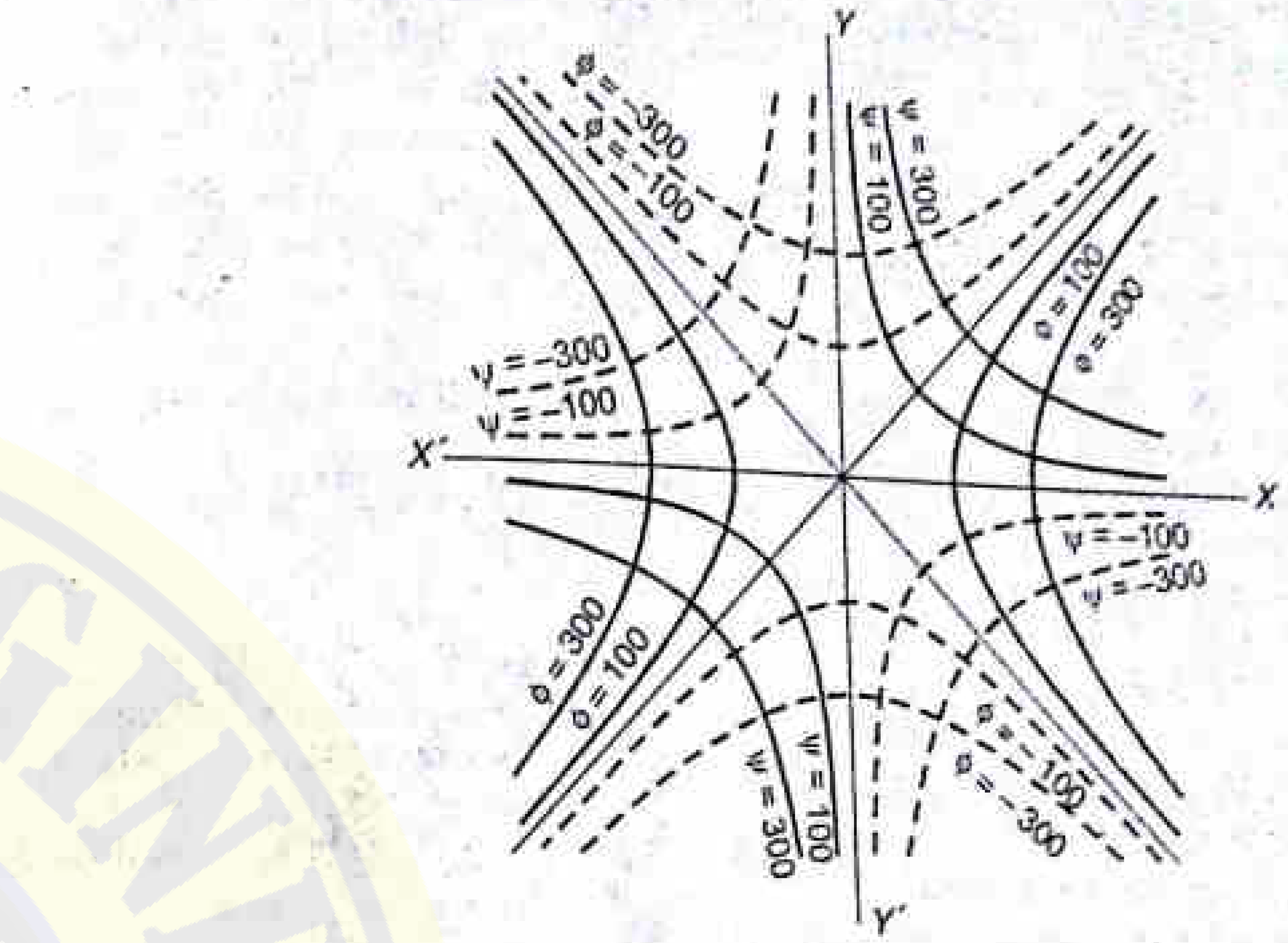


Figure Ex. 6.15

It may however be indicated that the flow-net obtained in the present case represents the flow pattern for flow around a 90° corner (or bend).

SUMMARY OF MAIN POINTS

1. The science which deals with the geometry of motion of fluids without reference to the forces causing the motion is known as 'kinematics'. Thus kinematics involves merely the description of the motion of fluids in terms of space-time relationship.
2. The science which deals with the action of the forces in producing or changing motion of fluids is known as 'kinetics'.
3. Fluid flow is said to be 'steady' if at any point in the flowing fluid various characteristics such as velocity, pressure, density, temperature, etc., which describe the behaviour of the fluid in motion, do not change with time. Mathematically it may be expressed as
$$\frac{\partial V}{\partial t} = 0; \frac{\partial p}{\partial t} = 0; \frac{\partial \rho}{\partial t} = 0; \text{etc.}$$
4. Fluid flow is said to be 'unsteady' if at any point in the flowing fluid, any one or all the characteristics which describe the behaviour of the fluid in motion change with time. Mathematically it may be expressed as
$$\frac{\partial V}{\partial t} \neq 0; \text{and or } \frac{\partial p}{\partial t} \neq 0; \text{etc.}$$
5. When the velocity of flow of fluid does not change, both in magnitude and direction, from point to point in the flowing fluid, for any given instant of time, the flow is said to be 'uniform'. In the mathematical form a uniform flow may be expressed as
$$\frac{\partial V}{\partial s} = 0$$
6. If the velocity of flow of fluid changes from point to point in the flowing fluid at any instant, the flow is said to be non-uniform. In the mathematical form a non-uniform flow may be expressed as
$$\frac{\partial V}{\partial s} \neq 0$$

7. The flow is said to be 'one dimensional' if the various characteristics of the flowing fluid are the functions of only one of the three coordinate directions, and time t.
8. The flow is said to be two dimensional if the various characteristics of the flowing fluid are the function of only any two of the three coordinate directions, and time t.
9. The flow is said to be 'three dimensional' if the various characteristics of the flowing fluid are the functions of all the three coordinate directions and time t.
10. A flow is said to be 'rotational' if the fluid particles while moving in the direction of flow rotate about their mass centres.
11. A flow is said to be 'irrotational' if the fluid particles while moving in the direction of flow do not rotate about their mass centres.
12. A flow is said to be laminar when the various fluid particles move in layers (or laminae) with one layer of fluid sliding smoothly over an adjacent layer. In the development of laminar flow the viscosity of the flowing fluid plays a significant role, and hence the flow of a very viscous fluid may in general be treated as laminar flow.
13. A fluid motion is said to be turbulent when the fluid particles move in an entirely haphazard or disorderly manner, that results in a rapid and continuous mixing of the fluid leading to momentum transfer as flow occurs.
14. A 'streamline' is an imaginary curve drawn through a flowing fluid in such a way that the tangent to it at any point gives the direction of the velocity of flow at that point.

   If u and v are the components of the velocity V along x and y directions then the equation for streamlines in the xy plane may be written as

   $$u\,dy - v\,dx = 0$$
15. A 'stream tube' is a tube imagined to be formed by a group of streamlines passing through a small closed curve, which may or may not be circular.
16. A 'path-line' may be defined as the line traced by a single fluid particle as it moves over a period of time.

   In steady flow path-lines and streamlines are identical.
17. A streak-line may be defined as a line that is traced by a fluid particle passing through a fixed point in a flow field.

   In steady flow a streak-line a streamline and a path line are identical. In unsteady flow a streak-line at any instant is the locus of end points of particle paths (or path-lines) that started at the instant the particle passed through the injection point.
18. The continuity equation in the differential form in cartesian coordinates in the most general form which is applicable for steady as well as unsteady flow, uniform as well as non-uniform flow, and compressible as well as incompressible fluids, is

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

where

ρ = mass density of fluid ; and u, v, w = components of velocity V along x, y and z directions respectively. For steady flow

since $\frac{\partial \rho}{\partial t} = 0$; the above equation reduces

to

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

For an incompressible fluid the mass density does not change with x, y, z, and t and hence the equation of coutinuity simplifies to

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

19. The continuity equation for two-dimensional flow in the most general form, is

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

For steady two dimensional flow the continuity equation is

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

For two dimensional flow of incompressible fluid the continuity equation is

$$\frac{\partial \rho}{\partial t} + \frac{\partial v}{\partial y} = 0$$

20. The continuity equation for one dimensional flow in the most general from is

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} = 0$$

For steady one dimensional flow the continuity equation is

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

For one dimensional flow of incompressible fluid the continuity equation is

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

21. If velocity of flow is uniform at each section of the flow passage, the continuity equation may be written as

$$Q = AV = \text{constant}$$

or

$$Q = A_1V_1 = A_2V_2 = A_3V_3 = \text{constant}$$

where Q is known as 'discharge' which is defined as the volume rate of flow of fluid or the volume of fluid flowing through any section per unit time. Discharge is usually measured in terms of cubic metre per second (or m^3/s) or liters per second. One cubic metre per second is usually written as one cumec.
22. Acceleration is defined as the rate of change of velocity with respect to time. The components of acceleration a_x, a_y, and a_z along x, y, and z, directions are as follows.

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

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

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

In the expressions for the components of acceleration each of the terms $\left(\frac{\partial u}{\partial t}\right), \left(\frac{\partial v}{\partial t}\right)$ and $\left(\frac{\partial w}{\partial t}\right)$ represents the 'local acceleration', and the remaining terms in these expressions represent the 'convective acceleration'.
23. When a fluid flows along a curved path, normal and tangential accelerations are developed. The expressions for the normal acceleration a_n and the tangential accelerations a_s are

$$a_n = \frac{V_s^2}{r} + \frac{\partial V_n}{\partial t}$$

$$a_s = V_s\frac{\partial V_s}{\partial s} + \frac{\partial V_s}{\partial t}$$

For steady flow

$$a_n = \frac{V_s^2}{r}$$

$$a_s = V_s\frac{\partial V_s}{\partial s}$$

where

V_s = tangential component of velocity ;
V_n = normal component of velocity ;

and

r = radius of curvature of the streamlines.
24. The rotation components ω_x, ω_y and ω_z about x, y and z axes respectively are given as

$$\omega_x = \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)$$

$$\omega_y = \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)$$

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

25. The flow along a closed curve is called 'circulation'. The mathematical concept of circulation is that it is the line integral, taken completely around a closed curve, of the velocity vector.
26. The vorticity is defined as circulation per unit area. Mathmatically vorticity is equal to two times the rotation component.
27. The 'velocity potential' ϕ is defined as a scalar function of space and time such that its negative derivative with respect to any direction gives the fluid velocity in that direction.

   Thus

   $$u = -\frac{\partial \phi}{\partial x};\ v = -\frac{\partial \phi}{\partial y};\ \text{and } w = -\frac{\partial \phi}{\partial z}$$
28. The 'stream function' ψ is defined as a scalar function of space and time such that its partial derivative with respect to any direction gives the

velocity component at right angles (in the counter-clock wise direction) to this direction. Thus for the case of two dimensional flow

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

29. The 'streamlines' are represented by ψ = constant and ϕ = constant represents 'equipotential lines'. The streamlines and equipotential lines intersect each other orthogonally at all points of intersection.

29. A grid obtained by drawing a series of streamlines and equipotential lines is known as a 'flow net'.

PROBLEMS

6.1 Define and distinguish between (a) steady and unsteady flow: (b) uniform and non-uniform flow (c) rotational and irrotational flow.

6.2 Define and distinguish between streamline, path line and streak line.

6.3 What is meant by one-dimensional, two-dimensional and three dimensional flows?

6.4 An incompressible fluid flows steadily through two pipes of diameter 0.15 m and 0.2 m which combine to discharge in a pipe of 0.3 m diameter. If the average velocities in the 0.15 m and 0.2 m diameter pipes are 2 m/s and 3 m/s respectively, then find the average velocity in the 0.3 m diameter pipe. [*Ans.* 1.83 m/s]

6.5 Determine which of the following pairs of velocity componets u and v satisfy the continuity equation for a two-dimensional flow of an incompressible fluid.

(a) $u = Cx$; $v = -Cy$

(b) $u = (3x - y)$; $v = (2x + 3y)$

(c) $u = (x + y)$; $v = (x^2 - y)$

(d) $u = A \sin xy$; $v = -A \sin xy$

(e) $u = 2x^2 + 3y^2$; $v = -3xy$.

[*Ans.* (a) Satisfy ; (b) Does not satisfy (c) Satisfy; (d) Does not satisfy ; (e) Does not satisfy]

6.6 Calculate the unknown velocity components so that they satisfy continuity equation:

(a) $u = 2x^2$; $v = xyz$; $w = ?$

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

[*Ans.* (a) $-4xz - xz^2 + f(x,y)$; (b) $-3yz^2 + f(x, z)$]

6.7 Determine which of the following velocity fields represent possible example of irrotational flow:

(a) $u = Cx$; $v = -Cy$

(b) $u = -Cxy$; $v = C \log xy$

(c) $u = (Ax^2 - Bxy)$; $v = \left(-2Axy + \frac{1}{2}By^2\right)$.

[*Ans.* (a) Irrotational flow ; (b) Rotational flow; (c) Rotational flow]

6.8 Which of the following stream function ψ are possible irrotational flow fields?

(a) $\psi = Ax + By^2$; (b) $\psi = Ax^2 y^2$

(c) $\psi = A \sin xy$; (d) $\psi = A \log (x/y)$

(e) $\psi = Ay\left(1 - \frac{C}{x^2 + y^2}\right)$ (f) $\psi = (y^2 - x^2)$.

[*Ans.* (a) to (d) None ; (e) and (f) Possible cases of Irrotational Flow Fields]

6.9 In a steady flow two points A and B are 0.5 m apart on a straight streamline. If the velocity of flow varies linearly between A and B what is the acceleration at each point if the velocity at A is 2 m/s and the velocity at B is 6 m/s?

[*Ans.* at A 16 m/s²; at B 48 m/s²]

6.10 Calculate the velocity components u and v for the following velocity potential functions ϕ:

(a) $\phi = x + y$; (b) $\phi = x^2 + y^2$

(c) $\phi = \tan^{-1}(y/x)$; (d) $\phi = \frac{Ax}{x^2 + y^2}$;

(e) $\phi = \sin x \sin y$; (f) $\phi = \log (x + y)$.

Which of these velocity potential functions satisfy the continuity equation?

[*Ans.* (a) $-1, -1$; (b) $-2x, -2y$; (c) $\frac{y}{(x^2+y^2)}$; $\frac{-x}{(x^2+y^2)}$; (d) $\frac{A(x^2-y^2)}{(x^2+y^2)^2}$; $\frac{2Axy}{(x^2+y^2)^2}$;

(e) $-\cos x, -\cos y$; (f) $\frac{-1}{(x+y)}$; $\frac{-1}{(x+y)}$. (a), (c) (d) Satisfy ; (b) (e) (f) Do not satisfy]

6.11 Calculate the total velocity at points (–2, 1), (2, 3) and (4, 0) for the velocity potential functions in Problem 6.10.

[*Ans.* At Point	(a)	(b)	(c)	(d)	(e)	(f)
(–2,1)	$\sqrt{2}$;	$\sqrt{20}$;	$1/\sqrt{5}$;	$A/\sqrt{25}$	$\left[\sqrt{\cos^2(2) + \cos^2(1)};\right]$	$\sqrt{2}$
(2,3)	$\sqrt{2}$;	$\sqrt{52}$;	$1/\sqrt{13}$;	$A/\sqrt{169}$;	$\left[\sqrt{\cos^2(2) + \cos^2(3)};\right]$	$\sqrt{2}/5$
(4,0)	$\sqrt{2}$;	–8;	1/4;	$A/16$;	$[\sqrt{1 + \cos^2(4)}]$	$\sqrt{2}/4$

6.12 Determine the corresponding stream function for each of the velocity potential functions in Problem 6.10

[*Ans.* (a) $y - x$ + constant ; (c) $-\frac{1}{2}\log(x^2 + y^2)$ + constant ; (d) $-\frac{Ay}{(x^2+y^2)}$ + constant]

Note: The velocity potential functions (b), (e), (f) do not represent the flow field and therefore no explicit stream functions corresponding to these can be obtained. The velocities of (b), (e), (f) determined in Problem 6.10 above have therefore no physical significance.

6.13 Describe the use and limitations of flow nets.

6.14 Discuss briefly the different methods of drawing the flow nets.

6.15 For a two dimensional flow of an incompressible fluid the velocity components V_r and V_θ are given as

$V_r = akr^n e^{[-k(n+1)\theta]}$, and $V_\theta = ar^n e^{[-k(n+1)\theta]}$.

Determine the stream function ψ and show that the fluid velocity at any point is given by

$$V = \left[(n+1)\psi\sqrt{(1+k^2)}/r\right]$$

$$\left[\text{Ans. } \psi = \frac{1}{n+1}\{ar^{(n+1)}e^{-k(n+1)\theta}\}\right]$$

6.16 What is the irrotational velocity field associated with the potential

$\phi = 3x^2 - 3x + 3y^2 + 16t^2 + 12zt$

Does the flow field satisfy the incompressible continuity equation.

[*Ans.* $u = -3(2x - 1)$; $v = -6y$, $w = -12t$; Does not satisfy]

6.17 In two-dimensional incompressible flow show that the flow rate per unit width between two streamlines is equal to the difference between the values of the stream function corresponding to these stream lines.

6.18 The velocity components in the x– and y–directions are given as $u = (2xy^2 - (2yx^2/3)$ and $v = xy^2 - (2yx^3/3)$. Indicate whether the given velocity distribution is a possible field of flow or not.

[*Ans.* Not a possible field of flow]

6.19 If the expression for the stream function is described by $\psi = x^3 - 3xy^2$ indicate whether the flow is rotational or irrotational. If the flow is irrotational determine the value of the velocity potential.

[*Ans.* Flow is irrotational; $\phi = y^3 - 3x^2y$]

of the approaching flow is 13.734 kN/m^2 $\left[0.14 \text{ kg(f)/cm}^2\right]$.

$\left[Ans.\ 117.72 \quad N\{12 \quad kg(f)\}\right]$

8.6 If a 0.552 kW (0.75 h.p.) motor is required by a ventilating fan to produce a 0.6 m stream of air having a velocity of 12 m/s, what is the efficiency of the fan? Take r = 1.207 kg/m^3 (0.123 msl/m^3).

[*Ans.* 53.4%]

8.7 A small ship is fitted with jets of total area 0.65 m^2. The velocity through the jets relative to ship is 9 m/s and ship's speed is 18.5 km/hour. The engine efficiency is 85%; the pump efficiency is 65% and pipe losses are equal to 10% of the kinetic energy of the jets. Determine the propelling force and the overall efficiency. Water is drawn in amid ship (i.e., at the middle of the ship from sides) and sea water weighs 10.055 kN/m^3 [1025 kg(f)/m^3].

$\left[Ans.\ 23.145 \ \ kN\{2359.4 \ kg(f)\};24.6\%\right]$

8.8 A jet propelled boat has two jets each of 100 mm diameter. The boat travels at 30 km/hour. If the resistance of the motion of the boat is 88.29 u^2 N [9u^2 kg(f)] where u is the velocity of the boat in m/s, calculate:

(a) the velocity of the jet relative to boat;

(b) the power required to drive the pump if the pump efficiency is 80%; and

(c) the overall efficiency.

Assume that the pump intake is facing the direction of motion of the boat.

[*Ans.* (a) 24.36 m/s; (b) 125.30 kW (170.33 h.p.); (c) 50.96%]

8.9 To propel a light aircraft at an absolute velocity of 250 km per hour against a head wind of 48 km per hour a thrust of 10.3 kN [1050 kg(f)] is required. Assume a theoretical efficiency of 90% and a constant air density of 1.207 kg/m^3 (0.123 msl/m^3), determine the diameter of ideal propeller required and the power needed to drive it.

[*Ans.* 2.54 m; 947.45 kW (1288 h.p.)]

8.10 A pipeline 0.6 m diameter conveying oil (sp. gr. 0.85) at the flow rate of 1800 litres per second has a 90° bend in the horizontal plane. The pressure at the entrance to the bend is 147.15 kN/m^2 [1.5 kg(f)/cm^2] and loss of head in the bend is 2 m of oil. Find the magnitude and direction of the force exerted by the oil on the bend.

$\left[Ans.\ 69.368 \text{ kN}\{7071 \text{kg(f)}\} \text{ to the right at } 42°15' \text{ with horizontal}\right]$

8.11 A nozzle at the end of a 80 mm hose produces a jet 40 mm in diameter. Determine the longitudinal stress in the joint at the base of the nozzle when it is discharging 1200 litres of water per minute. $\left[Ans.\ 38.322 \text{ kN}\{3906.5 \text{ kg(f)}\}\right]$

8.12 A motor boat is driven at 4 m/s by means of a jet of water issuing from an opening 100 mm square directly behind the boat and having discharge equal to 0.5 m^3/s. Find the driving force. The coefficient of contraction of the jet is 0.62.

[*Ans.* 38.322 kN {3906.5 kg (f)}]

8.13 Oil of sp. gr. 0.8 flows through a horizontal pipe of diameter 100 mm which is provided with a nozzle 25 mm diameter. If the pressure at the base of the nozzle is 784.8 kN/m^2 [8 kg(f)/cm^2), find the force exerted on the nozzle.

$\left[Ans.\ 5.439 \text{ kN}\{554.39 \text{ kg(f)}\}\right]$

Flow Through Orifices and Mouthpieces

Chapter 9

9.1 DEFINATION

An *orifice* is an opening having a closed perimeter, made in the walls or the bottom of a tank or a vessel containing fluid, through which the fluid may be discharged.

A *mouthpiece* is a short tube of length not more than two to three times its diameter, which is fitted to a circular opening or orifice of the same diameter, provided in a tank or a vessel containing fluid, such that it is an extension of the orifice and through which also the fluid may be discharged.

Both the orifices and the mouthpieces are usually used for measuring the rate of flow of fluid.

9.2 CLASSIFICATIONS OF ORIFICES AND MOUTHPIECES

The orifices may be classified on the basis of their size, shape, shape of the upstream edges and the discharge conditions. According to the size, the orifices may be classified as *small and large orifices*. According to the shape, the orifices may be classified as *circular, rectangular, square and triangular*. Out of these, however, circular and rectangular orifices are most commonly used. The upstream edge of the orifice may be made either sharp or it may be rounded. Thus according to the shape of the upstream edge the orifices may be classified as *sharp-edged orifices* and *bell-mouthed orifices* or orifices with the round corners. As shown in Fig. 9.1, a sharp-edged orifice has the bevelled side facing the downstream so that there is a minimum contact with the fluid flowing through the orifice and consequently the minimum frictional effects. A sharp-edged orifice is considered as a standard orifice and it is mostly used for the purposes of discharge measurement. The orifices may be made to discharge fluid from the tank freely in the atmosphere or it may be made to discharge fluid into the same fluid contained on the downstream side of the orifice such that the whole or the part of the outlet side of the orifice is submerged in the same fluid. Thus according to these discharge conditions, the orifices may be classified as *orifices discharging free* and *drowned* or *submerged orifices*. The drowned or submerged orifices may be further classified as *fully submerged orifices* (or totally submerged or totally drowned orifices) and *partially submerged orifices*.

The mouthpieces may be classified on the basis of their shape, position and the discharge conditions. According to the shape the mouthpieces may be classified as *cylindrical, convergent, divergent* and *convergent-divergent*. According to the position the mouthpieces may be classified as *external* and *internal mouthpieces*. An external mouthpiece is the one which is fitted to tank or reservoir such that it is projecting outside the tank or reservoir and it may be of any of the shapes noted above. An internal mouthpiece is also called *re-entrant or Borda's mouthpiece* which is fitted to a tank or a reservoir such that it is projecting inside the tank or reservoir and it is generally of cylindrical shape only. According to the discharge conditions the mouthpieces may be classified as *running full* and *running free* mouthpieces. Generally these conditions of discharge may be developed only in the case of internal mouthpieces.

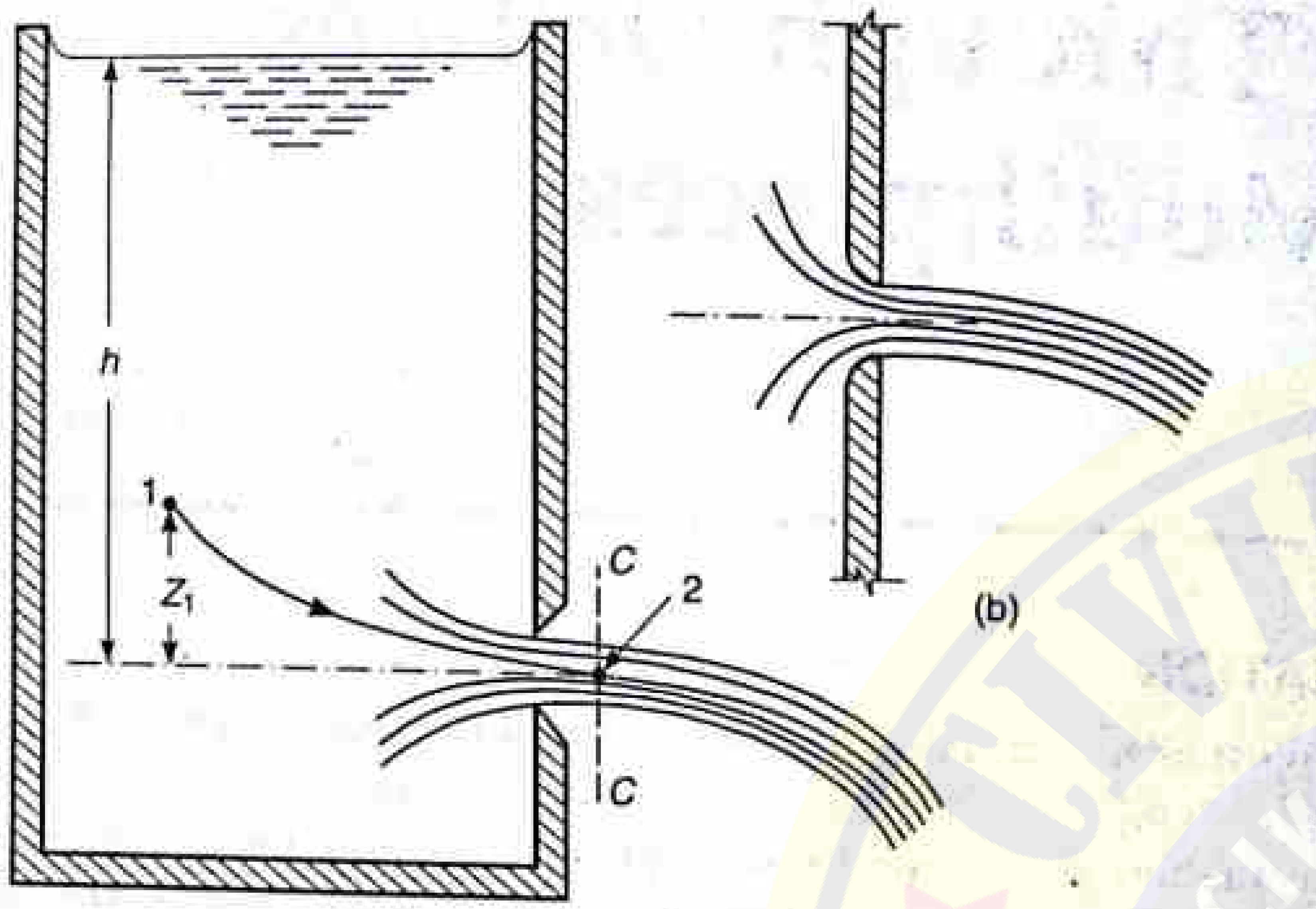


Figure 9.1 *(a) Sharp-edged orifice discharging free (b) Bell-mouthed orifice*

9.3 SHARP-EDGED ORIFICE DISCHARGING FREE

Figure 9.1 shows a sharp-edged small orifice in one side of a reservoir containing liquid. The liquid will emerge from the orifice as a free jet, that is, a jet discharged in the atmosphere and will therefore be under the influence of gravity only. It may be observed that the liquid approaching the orifice gradually converges towards it, to form a jet whose cross-sectional area is less than that of the orifice. This is so because the liquid particles lying close to the inner wall, have a motion, along the wall towards the orifice, which cannot be abruptly changed in direction at the orifice edge. Since an abrupt change of direction of motion is impossible, the streamlines continue to converge beyond the orifice upto a certain distance until they become parallel at the section C–C. Strict parallelism of the streamlines should be approached asymptotically in an ideal fluid; but in practice however, frictional effects produce the section C–C with parallel flow at only a short distance (about half the orifice diameter) from the orifice. The section C–C of the jet, at which the streamlines are straight and parallel to each other and perpendicular to the plane of the orifice, and the jet has the minimum cross-sectional area, is known as *vena-contracta* (Latin word meaning contracted vein or jet). Beyond the section C–C the jet may, however, diverge again and it undergoes a downward deflection due to gravity.

Since at vena-contracta the streamlines are parallel and straight, the pressure in the jet at this section is uniform and it is equal to the pressure of the fluid surrounding the jet, which in the case of a free jet, is thus equal to the atmospheric pressure. Moreover, at vena-contracta since the jet has the least cross-sectional area, the velocity of flow of liquid at this section will be maximum by the principle of continuity.

Let it be assumed that the flow through the orifice is steady under a constant head h measured above the centre of the orifice. Then considering two points 1 and 2 along a particular streamline the point 1 being inside the reservoir and the point 2 being at the centre of the jet at vena-contracta, as shown in Fig. 9.1, and applying the Bernoulli's equation between the points 1 and 2, we have, neglecting the loss of energy between these points,

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + 0$$

where p_1 and V_1 are respectively the pressure intensity and velocity of flow at point 1; z_1 is the height of point 1 above the datum which is taken as a horizontal plane passing through the centre of the orifice; p_2 and V_2 are respectively the pressure intensity and velocity of flow at point 2, for which the datum head is zero, and w is the specific weight of the liquid. Since V_1 is the velocity of the liquid approaching the orifice, it is often termed as *velocity of approach*.

Since the hydrostatic conditions prevail, the pressure p_1 may be expressed as

$$p_1 = p_a + w(h - z_1)$$

where p_a is the intensity of atmospheric pressure.

Also $$p_2 = p_a$$

Thus by substituting for p_1 and p_2 in the above expression, we obtain

$$\frac{V_2^2}{2g} = \frac{V_1^2}{2g} + h \qquad \text{...(9.1)}$$

Now if the orifice is small in comparison with the head h, the velocity of jet V_2 may be considered to be constant across the vena-contracta. Further if the velocity of approach V_1 is assumed to be constant across the cross-section of the reservoir at point 1, then by the equation of continuity the discharge Q may be expressed as

$$Q = a_1V_1 = a_cV_2$$

where a_1 is the cross-sectional area of the reservoir and a_c is the cross-sectional area of the jet at vena-contracta. By substituting for V_1 in Eq. 9.1 the expression for the velocity of flow of jet V_2 may be obtained as

$$V_2 = \sqrt{\frac{2gh}{1-\left(\frac{a_c}{a_1}\right)^2}} \qquad \text{...(9.2)}$$

However, if the reservoir is assumed to be sufficiently large and the point 1 is considered to be sufficiently far from the orifice then the velocity of approach V_1 will be very small in comparison with V_2 and hence it may be neglected in Eq. 9.1, in which case

$$\frac{V_2^2}{2g} = h \qquad \text{...(9.3)}$$

or $$V_2 = \sqrt{2gh}$$

Equation 9.3 is known as *Torricelli's formula* in honour of Evangelista Torricelli (1608–47) who in 1643 demonstrated, experimentally that the velocity with which a jet of liquid emerges from a small orifice is proportional to the square root of the head above the orifice. It may however be noted that Eq. 9.3 gives a velocity of jet at vena-contracta which is always more than the average velocity of the jet at a section in the plane of the orifice itself where neither the pressure nor the velocity is constant.

In the above analysis the loss of energy that takes place as the liquid flows through the orifice has been neglected. But in actual practice as the real fluid flows through an orifice there is always some loss of energy due to friction and surface tension. As such Eq. 9.3 represents only the ideal (or theoretical) velocity of the jet and the actual velocity of the jet at vena-contracta is slightly less than the ideal velocity given by Eq. 9.3. The actual velocity of jet at vena-contracta may therefore be determined by multiplying the ideal velocity by a factor called coefficient of velocity.

The *coefficient of velocity* C_v is defined as the ratio between the actual velocity of jet at vena-contracta and the ideal (or the theoretical) velocity of the jet. Thus if V is the actual velocity of the jet at vena-contracta then

$$C_v = \frac{V}{V_{th}} = \frac{V}{\sqrt{2gh}}$$

or

$$V = C_v\sqrt{2gh} \qquad \ldots(9.4)$$

The difference between the theoretical and the actual velocities of the jet at vena-contracta is mainly due to friction at the orifice. For sharp-edged orifices since the liquid passing through these touches only a line, the friction is considerably less and hence the difference between the theoretical and the actual velocities is very small. Therefore, for sharp-edged orifices discharging water and other liquids of similar viscosity the value of C_v is slightly less than unity. Experimentally it has been observed that the value of C_v varies from 0.95 to 0.99 for different orifices, depending on the shape and size of the orifice and on the head of liquid under which the flow takes place. In general an average value of 0.97 or 0.98 may be assumed for C_v, for sharp-edged orifices discharging water and other liquids of similar viscosity. For orifices which are not sharp-edged the value of C_v may be markedly lower.

As indicated earlier a jet of liquid issuing from an orifice has its cross-sectional area at vena-contracta less than the area of the orifice. In other words the jet of liquid issuing from an orifice undergoes a contraction. The actual area of the jet at vena-contracta may therefore be determined by multiplying the area of the orifice by a factor called coefficient of contraction.

The *coefficient of contraction* C_c is defined as the ratio between the area of the jet at vena-contracta and the area of the orifice. Thus if a_c is area of the jet at vena-contracta and a is the area of the orifice, then

$$C_v = \frac{a_c}{a}$$

or

$$a_c = C_c\, a \qquad \ldots(9.5)$$

The theoretical value of C_c for a sharp-edged orifice is equal to $\left(\frac{\pi}{\pi+2}\right)$ which equals 0.611, but in practice its value varies from 0.61 to 0.69 depending on the size and shape of the orifice and the head of liquid under which the flow takes place. However, for general purposes for small sharp-edged orifices an average value of 0.64 or 0.65 may be assumed for C_c.

In the case of a bell-mouthed orifice as shown in Fig. 9.1 (b) the contraction after the orifice is almost completely eliminated. The coefficient of contraction is therefore equal to unity for a bell-mouthed orifice. But on account of friction the coefficient of velocity for a bell-mouthed orifice is considerably reduced. Moreover, the velocity at the edge of the jet is reduced due to friction and so the velocity over the cross-section of jet emerging from a bell-mouthed orifice is non-uniform.

Theoretically the cross-sectional area of the jet of liquid issuing from an orifice will be equal to the area of the orifice, which may be considered as an ideal (or theoretical) cross-sectional area of the jet. The product of the ideal cross-sectional area of the jet and the ideal velocity of jet given by Eq. 9.2 will therefore give ideal (or theoretical) discharge. However, on account of the effect of friction due to which the actual velocity of the jet is reduced and due to the contraction of the jet, the actual discharge of liquid through an orifice is always less than the ideal (or theoretical) discharge. The actual discharge of liquid through an orifice may therefore be determined by multiplying the ideal discharge by a factor called coefficient of discharge.

The *coefficient of discharge* C_d is defined as the ratio between the actual discharge issued from an orifice and its theoretical (or ideal) discharge.

Thus if Q_{th} is the theoretical discharge and Q is the actual discharge emerging from an orifice then

$$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} = \frac{Q}{Q_{th}} \qquad \ldots(9.6)$$

But actual discharge Q = (actual area of the jet at vena-contracta × actual velocity of jet there)

i.e.,

$$Q = (a_c \times V) = (C_c a) \times (C_v\sqrt{2gh})$$

Similarly, theoretical discharge Q_{th} = (theoretical area of jet × theoretical velocity of jet)

i.e.,

$$Q_{th} = (a \times \sqrt{2gh})$$

By substituting the values in Eq. 9.6

$$C_d = \frac{Q}{Q_{th}} = \frac{(C_c a)\times\left(C_v\sqrt{2gh}\right)}{a\times\sqrt{2gh}}$$

Therefore

$$C_d = C_c \times C_v \qquad \ldots(9.7)$$

Equation 9.7 indicates that the coefficient of discharge C_d of an orifice is equal to the product of the coefficient of contraction C_c and the coefficient of velocity C_v and it may therefore be found by first determining C_c and C_v of the orifice and by multiplying these together. The coefficient of discharge varies from 0.61 to 0.65 depending on the size and the shape of the orifice, and the head of liquid under which the flow takes place. For general purposes for small sharp-edged orifices discharging water or liquids of similar viscosity, an average value of 0.62 or 0.63 may be adopted for C_d.

In addition to the three coefficients for an orifice defined above, one more coefficient may be introduced which is called the coefficient of resistance.

The *coefficient of resistance* C_r is defined as the ratio of the loss of kinetic energy as the liquid flows through an orifice and the actual kinetic energy possessed by the flowing fluid.

The loss of kinetic energy as the liquid flows through an orifice is equal to the difference between the theoretical kinetic energy and the actual kinetic energy.

Theoretical kinetic energy per unit weight of liquid

$$\frac{V_{th}^2}{2g} = \frac{(\sqrt{2gh})^2}{2g} = h$$

Actual kinetic energy per unit weight of liquid

$$= \frac{V^2}{2g} = \frac{C_v^2\left(\sqrt{2gh}\right)^2}{2g} = hC_v^2$$

∴ Loss of kinetic energy per unit weight of liquid

$$= (h - C_v^2 h) = h\,(1 - C_v^2)$$

$$\therefore \qquad C_r = \frac{h(1-C_v^2)}{hC_v^2} = \left(\frac{1}{C_v^2} - 1\right) \qquad \text{...(9.8)}$$

The coefficient of resistance thus accounts for the loss of energy that occurs as any liquid flows through an orifice and its value may be determined by knowing the value the coefficient of C_v of the orifice.

Often the values of the various coefficients for an orifice, defined above are required to be determined in the laboratory in order to calibrate the orifice for being used as a measuring device. The various methods commonly adopted for determining these coefficients in the laboratory are discussed in next section.

9.4 EXPERIMENTAL DETERMINATION OF THE COEFFICIENTS FOR AN ORIFICE

There are several methods by which the value of each of these coefficients of an orifice may be determined, some of which are commonly adopted, are described below.

(a) Determination of the Coefficient of Velocity C_v. The different methods which are adopted for the determination of C_v are as follows:

(i) Jet Distance Measurement Method. Consider a tank provided with a small orifice on one of the sides and an arrangement for the supply of liquid to the tank as shown in Fig. 9.2. By adjusting the inflow of the liquid in the tank equal to the outflow (or efflux) through the orifice a constant head h of the liquid above the centre of the orifice is maintained under which a jet of liquid emerges through the orifice. Let V be the velocity of the jet at vena-contracta. Now consider a liquid particle in the jet of the liquid, which is at vena-contracta at any instant of time and in time t as it is carried along the jet it occupies a new position represented by a point in the jet as shown in Fig. 9.2. If x and y are respectively the horizontal and the vertical distances of the point from the vena-contracta, then it may be considered that as the fluid particle moves from its original position at vena-contracta to its new position at the point in time t, it is displaced horizontally through a distance x from the vena-contracta and at the same time it drops down under the action of gravity through a vertical distance y from the vena-contracta. If the air resistance is negligible the horizontal component V of jet velocity remains unchanged (as there is no acceleration acting on the jet in the horizontal direction), and hence the horizontal distance travelled by a particle in time t after it has left the vena-contracta is

$$x = Vt \qquad \text{...(i)}$$

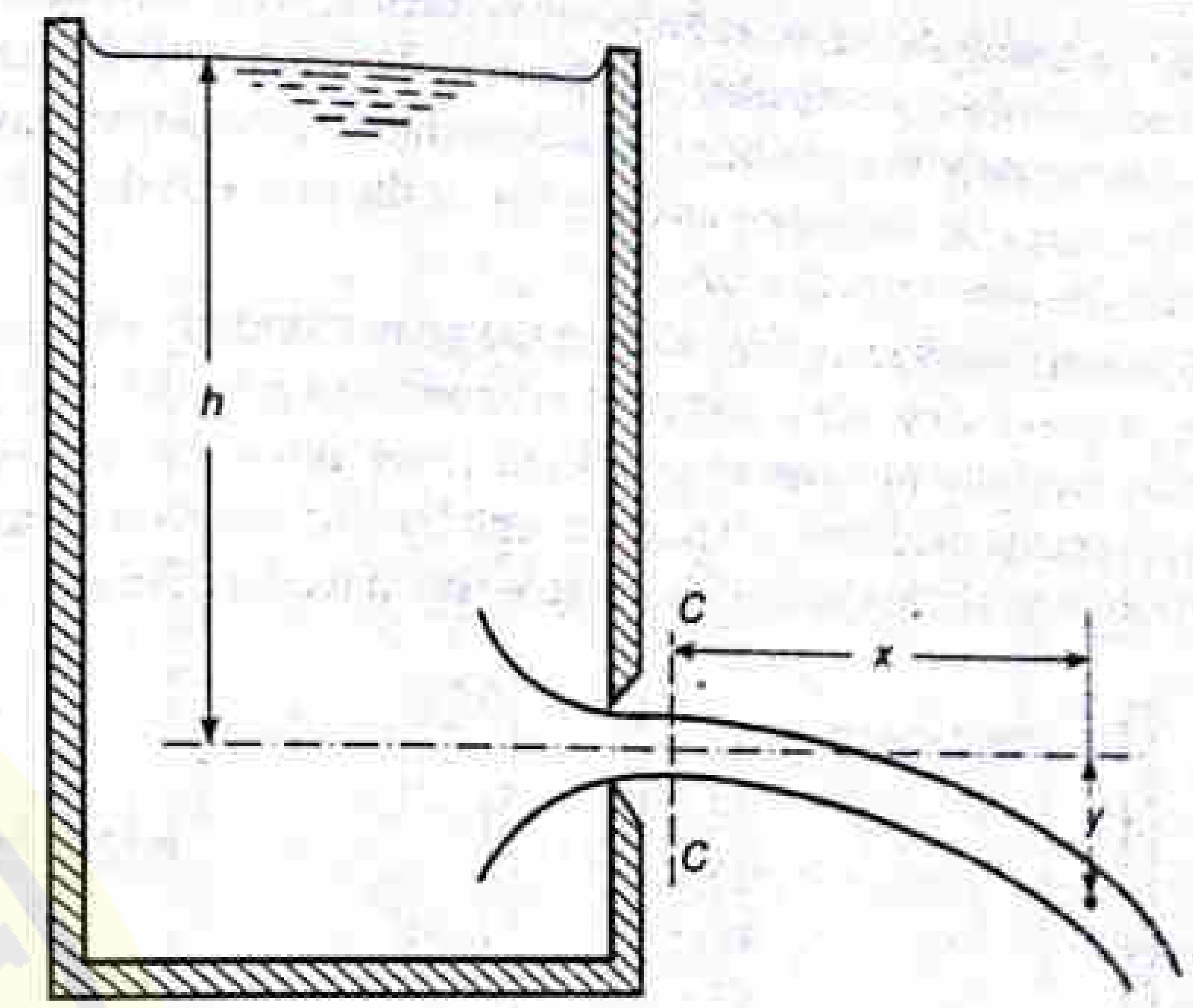


Figure 9.2 *Determination of C_v by jet distance measurement method*

In the vertical direction since there is a uniform downward acceleration g, and the vertical component of velocity is initially zero for horizontal discharge, the vertical distance travelled by the particle in the same time t is

$$y = \frac{1}{2}gt^2 \qquad \text{...(ii)}$$

By eliminating t between the Eqs. (i) and (ii), we have

$$x^2 = \frac{2V^2}{g}\,y\,;\text{ or } V = \sqrt{\frac{gx^2}{2y}} \qquad \text{...(9.9)}$$

which shows that the path of a liquid particle moving along a free-jet of liquid is parabolic, with the vertex of the parabola at the vena-contracta.

Equation 9.9 represents the actual velocity of flow of jet at vena-contracta. The theoretical velocity of jet emerging from an orifice under a constant head h is given by Eq. 9.3 as

$$V_{th} = \sqrt{2gh}$$

Therefore coefficient of velocity

$$C_v = \frac{V}{V_{th}} = \frac{\sqrt{\dfrac{gx^2}{2y}}}{\sqrt{2gh}} = \sqrt{\frac{x^2}{4hy}} \qquad \text{...(9.10)}$$

If the coordinates, x and y, of a point in the jet and the head h of the liquid are measured, then Eq. 9.10 may be used to obtain the value of the coefficient of velocity C_v of the orifice. For the measurement of the coordinates of any point in the jet generally a scale and sliding apparatus is fitted to the orifice tank. It consists of a simple triangular frame at the base of which a graduated scale is provided which facilitates the measurement of the x coordinate of any point in the jet. A vertical graduated scale which can slide on the horizontal scale is also provided, on which the vertical depth y of the point in the jet can be determined. In order to facilitate the exact location of the point in the jet a hooked pointer is provided which is attached to the vertical scale.

(ii) Velocity Measurement Method. In this method the actual velocity of jet at vena-contracta is measured by inserting a previously calibrated pitot tube as shown in the Fig. 9.3. The theoretical velocity may however be computed by measuring the head h over the centre of the orifice. Knowing the actual and the theoretical velocities of flow of liquid through the orifices, its coefficient of velocity may be computed by Eq. 9.4. However, this method is suitable only if the jet diameter is not too small.

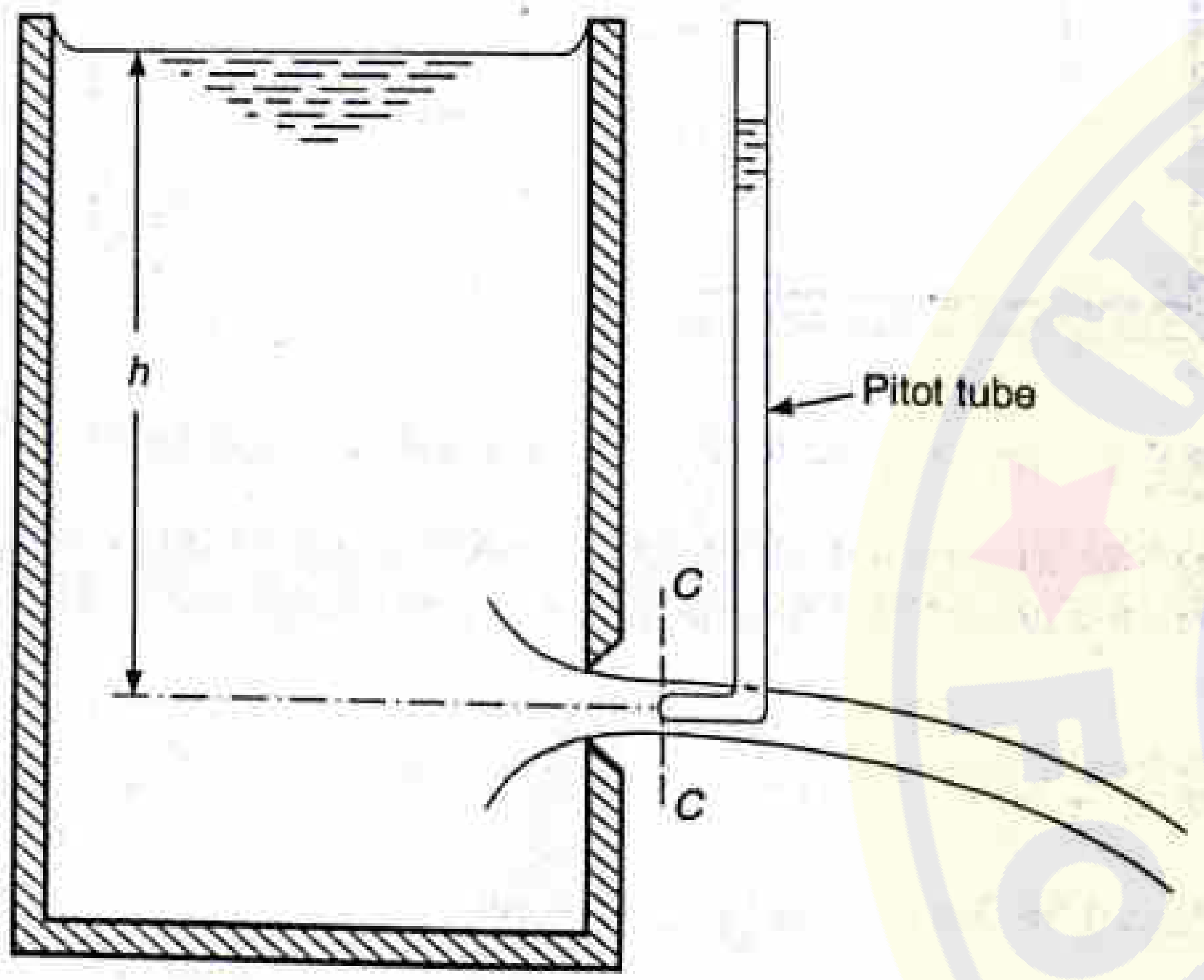


Figure 9.3 *Determination of C_v by the velocity measurement method*

The actual velocity of the jet at vena-contracta may also be determined by measuring the cross-sectional area of the jet at vena-contracta and the actual discharge of liquid flowing through the orifice under a constant head h. Thus if Q is the discharge actually flowing through the orifice and a_c is the cross-sectional area of the jet at vena-contracta then the actual velocity of the jet at vena-contracta is

$$V = \frac{Q}{a_c}$$

Again $$V_{th} = \sqrt{2gh}$$

∴ Coefficient of velocity

$$C_v = \frac{V}{V_{th}} = \frac{Q}{a_c\sqrt{2gh}}$$

(iii) Momentum Method. This method involves the impulse momentum equation for the determination of the actual velocity of jet at vena-contracta. As shown in Fig. 9.4 a tank provided with an orifice in one of its sides and containing liquid is supported in a suspended position on knife edges on either

side of the tank. On the vertical side of the tank opposite to that having the orifice a short projecting platform is provided on which the desired weights may be placed. By adjusting the inflow equal to the outflow through the orifice a constant head h of the liquid is maintained above the centre of the orifice.

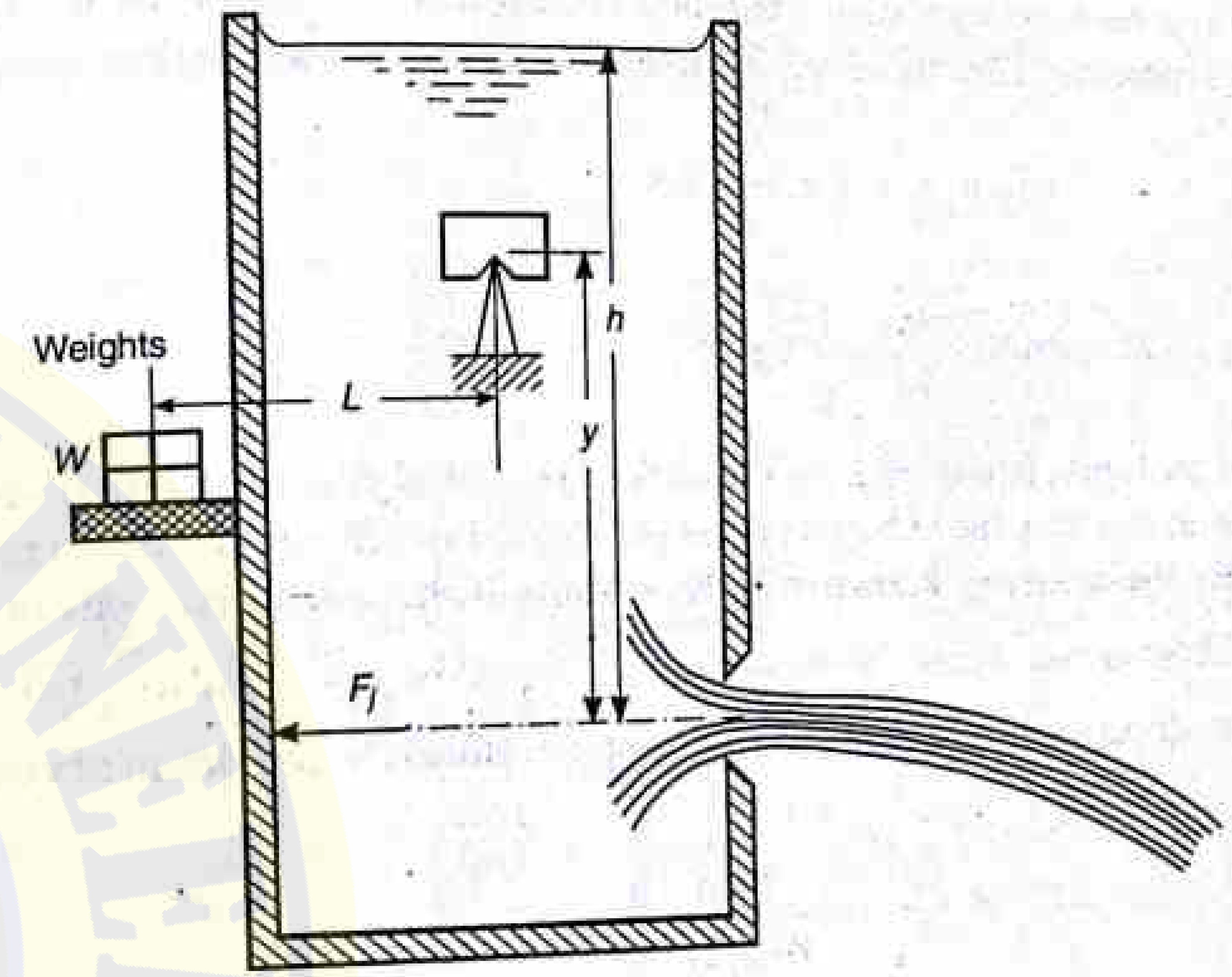


Figure 9.4 *Determination of C_v by the momentum method*

Originally when no liquid is discharged through the orifice the tank will remain in a vertical position. But as the jet of liquid is issued from the orifice some change of momentum takes place for the liquid flowing through the orifice, which results in exerting a force F_j on the tank in the direction opposite to that of the flowing jet. The force exerted on the tank causes the tank to be tilted towards the issuing jet from its vertical position. The vertical position of the tank is however restored by placing the necessary weights on the platform.

The force exerted by the issuing jet on the tank may be computed by the application of the impulse-momentum principle. Thus if W_l is the weight of the liquid discharged through the orifice per second and V is the velocity of the issuing jet of liquid at vena-contracta, then the momentum per second of liquid leaving the tank in the horizontal direction is $\left[\frac{(W_l V)}{g}\right]$. Neglecting the velocity of approach the initial momentum possessed by this liquid is equal to zero. Therefore the rate of change of momentum of liquid leaving the tank is $\left[\frac{(W_l V)}{g}\right]$ which is according to impulse-momentum principle equal to the force F_j exerted on the tank by the issuing jet. That is

$$F_j = \frac{W_l V}{g}$$

Due to this force exerted on the tank, it will get tilted. Now let W be the weight required to be placed on the platform in order to counter balance the effect of the force of the issuing jet and to bring back the tank to its vertical position. Then by taking the moments of the forces about the knife edge, we have

$$W \times L = F_j \times y = \frac{W_l V}{g} \times y$$

or

$$V = \frac{W}{W_l} \times \frac{L}{y} \times g \quad \text{...(9.11)}$$

The weight of the liquid emerging from the orifice per second W_l, can be determined by actually collecting the liquid in a tank for a known period of time and weighing the same. Thus all the terms except the velocity V of the jet are known in Eq. 9.11, from which the actual velocity of the jet at vena-contracta can be determined. Again the theoretical velocity of the jet is equal to $\sqrt{2gh}$ which may be computed by measuring the head h above the centre of the orifice. The coefficient of velocity C_v is then given by

$$C_v = \frac{V}{V_{th}} = \frac{WLg}{W_l y \sqrt{2gh}} \quad \text{...(9.12)}$$

(b) Determination of the Coefficient of Contraction C_c. The coefficient of contraction for an orifice may be determined either by a direct method or by an indirect method. In the direct method of the determination of C_c the radius of the jet at vena-contracta is determined with the help of a micrometer contraction gage. As shown in Fig. 9.5 (a) the gage consists of a circular collar or a ring which is provided with four micrometer screws fitted at the diametrically opposite points at right angles to each other. The pointed ends of the screws are so adjusted that in closed position they all meet at the centre of the ring and the reading on the scale of each screw is zero and when opened these may be made to touch the outer periphery of the jet as shown in Fig. 9.5 (b). However there may be some zero error which may be noted beforehand. In order to determine the radius of the jet of liquid issuing from an orifice under a constant head, the gage with its screws open is held in front of the orifice at vena-contracta such that the jet of liquid passes through the ring. The screws are then moved and so adjusted that their pointed ends are just touching the outer periphery of the jet. The reading of each of the micrometer screws (corrected for the zero error if any) then gives the radius of the jet. The mean of the readings of all the four screws will give a fairly accurate value of the radius of the jet from which the actual area of the jet at vena-contracta may be computed. The area of the orifice being already known the coefficient of contraction C_c of the orifice may then be computed from Eq. 9.5.

The above described method is, however, applicable only for the determination of C_c of a circular orifice. Moreover even for a circular orifice the jet of liquid issuing from the orifice is quite irregular and hence it is quite difficult to measure the correct radius of the jet at vena-contracta. Therefore, generally the indirect method for the determination of the coefficient of contraction for an orifice is adopted, which in addition to being applicable for an orifice of any shape, avoids the cumbersome process of

measurement of the actual area of the jet at vena-contracta. In this indirect method the value of C_c is obtained by using the relation derived from Eq. 9.7 as

$$C_c = \frac{C_d}{C_v}$$

in which the values of C_d and C_v for an orifice are determined previously.

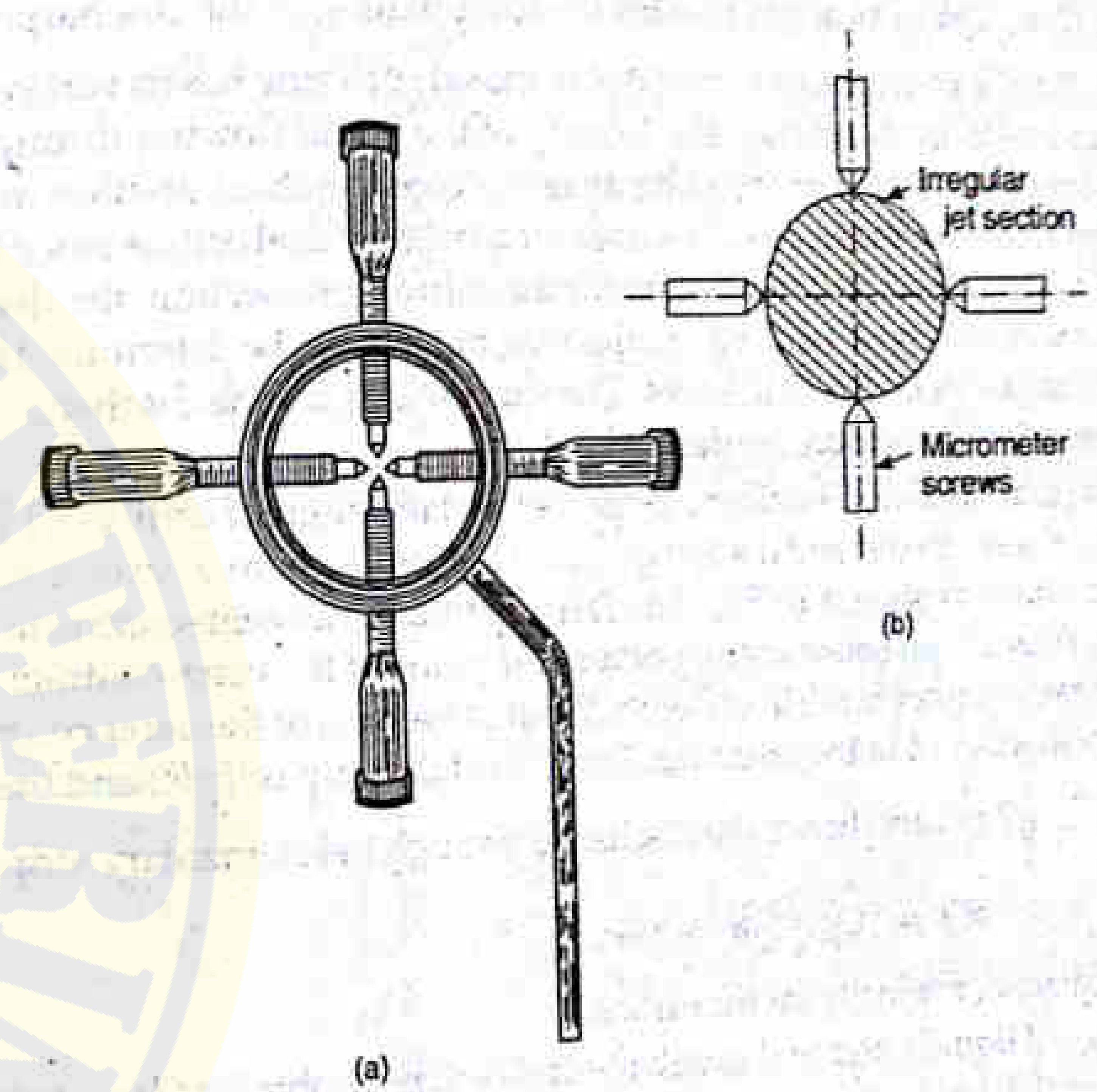


Figure 9.5 *(a) Micrometer contraction gage (b) Screws touching the jet periphery*

(c) Determination of the Coefficient of Discharge C_d. The coefficient of discharge for an orifice may be more easily determined than the other coefficients. To determine C_d for an orifice, it is allowed to discharge under a constant head and for a known period of time the liquid passing through the orifice is collected in a tank and it is measured either by weight or by volume. The actual discharge Q passing through the orifice may then be computed by dividing the volume of the liquid collected by the time during which it is collected. The theoretical discharge of the orifice being equal to $(a \times \sqrt{2gh})$ which may be computed from the known area of the orifice a and the constant head h under which it is discharging. The value of the coefficient of discharge for the orifice may then be determined as

$$C_d = \frac{Q}{a \times \sqrt{2gh}} \quad \text{...(9.13)}$$

The value of C_d for an orifice may also be determined by first determining its C_c and C_v and then multiplying the two in accordance with Eq. 9.7 to obtain the value of C_d. But generally this method is not preferred because as stated earlier the value of C_c cannot be determined accurately by the direct experimental method, which will result in giving an inaccurate value of C_d for the orifice.

The coefficient of resistance C_r for an orifice may be computed by using Eq. 9.8 if the value of the coefficient of velocity C_v for the orifice is known.

9.5 FLOW THROUGH LARGE VERTICAL ORIFICE

So far we have dealt with an orifice whose vertical dimension has been small in comparison to the head, in which case the velocity in the entire cross-section of the jet could be considered to be constant and equation $Q = C_d\, a \sqrt{2gh}$ was applicable for the computation of the discharge actually passing through the orifice. But if a vertical orifice provided in the side of a tank has its vertical dimension large enough as compared with the head then the velocity of the liquid flowing through such an orifice varies over the entire cross-section of the jet because of a considerable variation in head at different points in the vertical section of the orifice. Therefore for a large vertical orifice, since the velocity of the flowing liquid cannot be considered as constant for the enitre cross-section, the discharge cannot be computed by the same equation as that for a small orifice, but it has to be determined by integrating the discharges through small elements of the area. The expressions for the discharge flowing through large rectangular and circular orifices are derived below.

(a) Large Rectangular Orifice. Consider a large, vertical rectangular orifice of breadth b, depth d, provided in the side of a reservoir, and discharging into the atmosphere. A vena-contracta of breadth b_c and depth d_c is formed as shown in Fig. 9.6. The streamlines at vena-contracta are parallel and practically straight, thus the pressure at any point in the plane of the vena-contracta is atmospheric. Consider an elementary horizontal strip of depth dh out of the area of the vena-contracta at a depth h below the free surface of liquid in the reservoir. The area of the strip is $(b_c dh)$ and the velocity of flow through the strip is $C_v \sqrt{2gh}$, and hence the discharge through the elementary strip equals

$$dQ = C_v \sqrt{2gh} \times b_c dh$$

where C_v is the coefficient of velocity for the orifice.

The total discharge of liquid passing through the orifice may be obtained by integrating the above expression across the plane of the vena-contracta. Thus if H_c represents the depth of the centre of the vena-contracta below the free surface of the liquid then h varies from $(H_c - d_c/2)$ to $(H_c + d_c/2)$ and the total discharge is

$$Q = C_v b_c \sqrt{2g} \int_{(H_c - d_c/2)}^{(H_c + d_c/2)} h^{1/2} dh$$

$$= \frac{2}{3} C_v b_c \sqrt{2g} \left\{ \left(H_c + \frac{d_c}{2}\right)^{3/2} - \left(H_c - \frac{d_c}{2}\right)^{3/2} \right\}$$

However the above derived expression for the discharge through a large rectangular orifice involves the quantities such as b_c, d_c and H_c which are rather difficult to be determined. This difficulty may be overcome by introducing b, d and H respectively in place of b_c, d_c and H_c and a coefficient of contraction C_c, where H is the depth of the centre of the orifice below the free surface in the reservoir. Then

$$Q = \frac{2}{3} C_c C_d b \sqrt{2g} \left\{ \left(H + \frac{d}{2}\right)^{3/2} - \left(H - \frac{d}{2}\right)^{3/2} \right\}$$

or

$$Q = \frac{2}{3} C_d b \sqrt{2g} \left\{ \left(H + \frac{d}{2}\right)^{3/2} - \left(H - \frac{d}{2}\right)^{3/2} \right\} \quad ...(9.14)$$

where C_d is the coefficient of discharge for the orifice.

Often Eq. 9.14 is expressed in a slightly different form. Thus if the height of the free surface above the top edge of the orifice is H_1 and that above the bottom edge of the orifice is H_2, then since

$$H_1 = \left(H - \frac{d}{2}\right)$$

and

$$H_2 = \left(H + \frac{d}{2}\right)$$

$$Q = \frac{2}{3} C_d b \sqrt{2g} \{H_2^{3/2} - H_1^{3/2}\} \quad ...(9.15)$$

However, if the same rectangular orifice is treated as a small orifice the discharge through it may be expressed as

$$Q = C_d (b \times d) \sqrt{2gH} \quad ...(9.16)$$

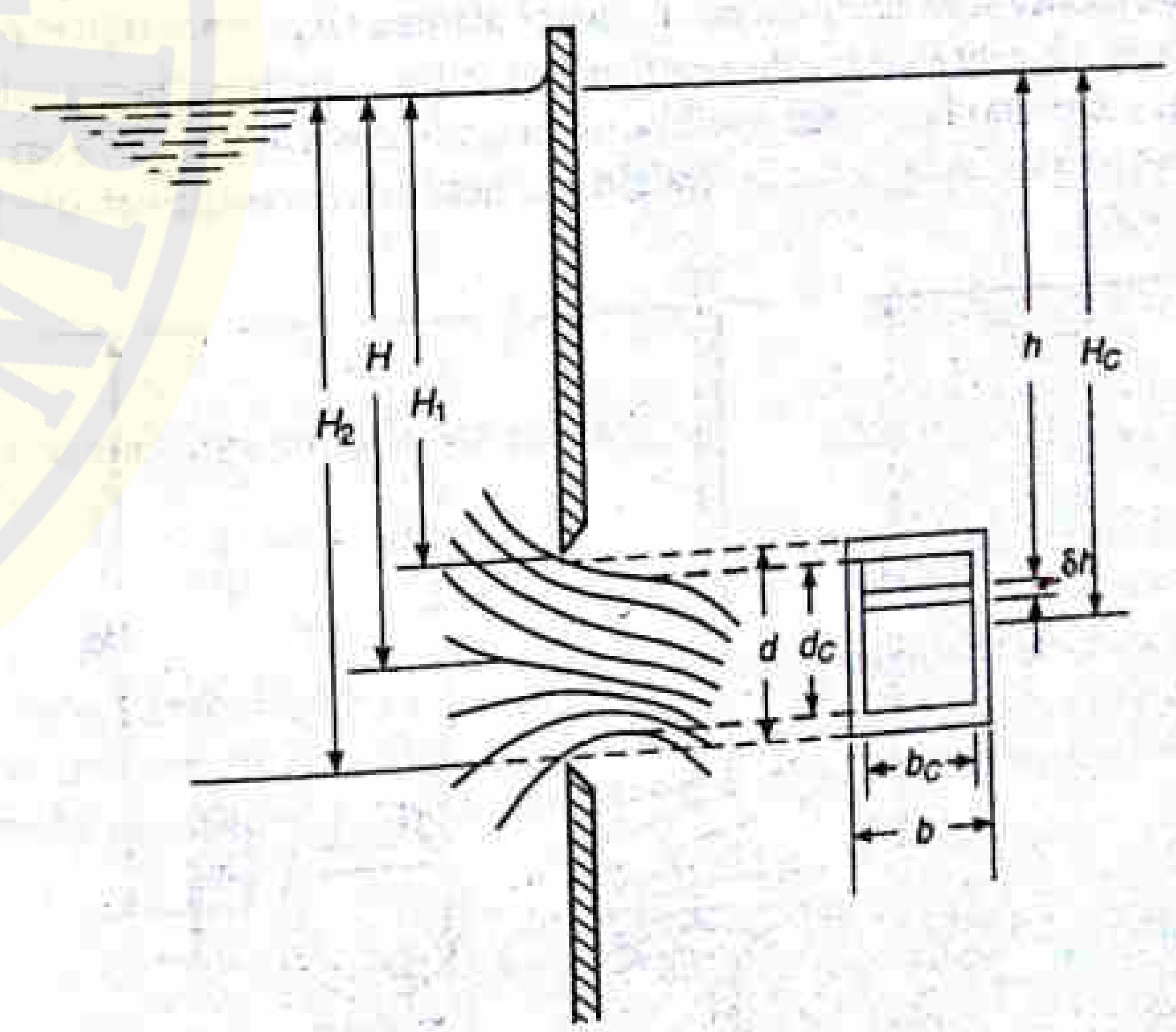


Figure 9.6 *Large rectangular orifice*

By comparing the values of the discharge computed by Eqs 9.14 and 9.16 it can be shown that there is only a slight difference in the two values which may however be neglected provided the head H has certain minimum value. Thus if $H = d$ then from Eq. 9.14

$$Q = 0.99[C_d (b \times d) \sqrt{2gH}]$$

which shows that for the head above the centre of the orifice equal to its depth there is an error of only 1% in the actual discharge computed by Eq. 9.14 and the one computed by treating it as a small orifice and using Eq. 9.16. Moreover for $H > d$ the use of Eq. 9.16 in place of Eq. 9.14 will give an error of even less than 1%. *Thus as long as $H > d$, i.e., head above the centre of the orifice is greater than its vertical dimension, it may be treated as a small orifice.*

In deriving the above expression for the discharge through a large rectangular orifice the velocity of approach has been neglected. But if the velocity of approach is not negligible and if V_1 is the velocity of approach then the discharge through the elementary strip equals

$$dQ = C_v \sqrt{2g\left(h+\frac{V_1^2}{2g}\right)} \times b_c dh$$

By integrating this, the expression for the total discharge through the orifice is obtained as

$$Q = \frac{2}{3} C_d b \sqrt{2g}\left\{\left(H+\frac{d}{2}+\frac{V_1^2}{2g}\right)^{3/2} - \left(H-\frac{d}{2}+\frac{V_1^2}{2g}\right)^{3/2}\right\}$$

or

$$Q = \frac{2}{3} C_d b \sqrt{2g}\left\{\left(H_2+\frac{V_1^2}{2g}\right)^{3/2} - \left(H_1+\frac{V_1^2}{2g}\right)^{3/2}\right\} \quad ...(9.17)$$

(b) Large Circular Orifice. Consider a circular orifice of diameter d provided in the side of a reservoir containing liquid upto a height H above the centre of the orifice which is discharging liquid into the atmosphere. A vena-contracta of diameter d_c will be formed as shown in Fig. 9.7. Again the streamlines at vena-contracta are parallel and practically straight and hence the pressure at any point in the plane

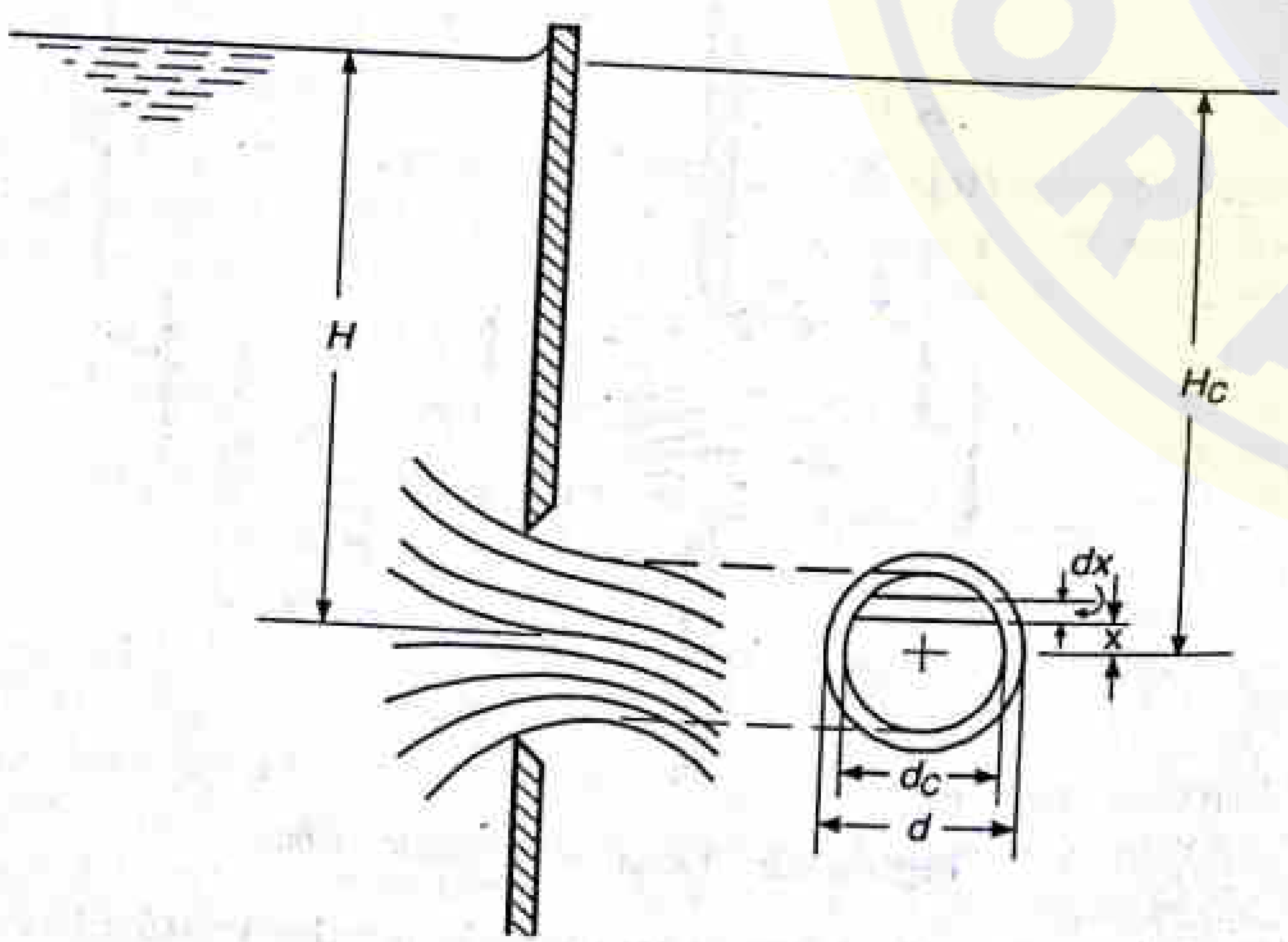


Figure 9.7 *Large circular orifice*

of the vena-contracta is atmospheric. Consider an elementary horizontal strip of depth dx out of the area of the vena-contracta at a distance x from the centre of the vena-contracta as shown in Fig. 9.7. The

area of the strip is $\left\{2\sqrt{(d_c/2)^2 x^2}\right\} \times dx$ and the velocity of flow through the strip is $C_v\sqrt{2g(H_c - x)}$ where C_v represents the coefficient of velocity for the orifice and H_c is the depth of the centre of the vena-contracta below the free surface of the liquid. The discharge through the elementary strip is

$$dQ = C_v\sqrt{2g(H_c - x)}\ \left\{2\sqrt{(d_c/2)^2 - x^2}\right\}dx$$

By making x to vary between the values $(-d_c/2)$ and $(+d_c/2)$, the integration of this expression will give the discharge from the entire orifice. For the integration it will be necessary to expand the term $(H_c-x)^{1/2}$ by the binomial theorem.

$$(H_c - x)^{1/2} = H_c^{\ 1/2} - \frac{H_c^{-1/2}x}{2} - \frac{H_c^{-3/2}x^2}{8} - \frac{H_c^{-5/2}x^3}{16} - ....$$

$$\therefore \quad dQ = 2C_v\sqrt{2g}\left[H_c^{\ 1/2}\left\{\left(\frac{d_c}{2}\right)^2 x^2\right\}^{1/2} - \frac{\left\{\left(\frac{d_c}{2}\right)^2 - x^2\right\}^{1/2} x}{2H_{c\,1/2}} - \frac{\left\{\left(\frac{d_c}{2}\right)^2 x^2\right\}^{1/2} x^2}{8H_c^{3/2}} - \frac{\left\{\left(\frac{d_c}{2}\right)^2 - x^2\right\}^{1/2} x^3}{16H_c^{5/2}} ....\right]dx$$

By integrating the above expression we obtain

$$Q = C_v \frac{\pi d_c^2}{4}\sqrt{2gH_c}\left[1 - \frac{d_c^2}{128H_c^2} - \frac{5d_c^4}{16384H_c^4} - ....\right]$$

The above derived expression for the discharge through a circular orifice involves the quantities such as d_c and H_c which are difficult to be determined. This difficulty may be overcome by introducing d and H respectively in place of d_c and H_c and a coefficient of contraction C_c. Then

$$Q = C_c C_v \frac{\pi d^2}{4}\sqrt{2gH}\left[1 - \frac{d^2}{128H^2} - \frac{5d^4}{16384H^4} - ....\right] \quad ...(9.18)$$

or

$$Q = C_d \frac{\pi d^2}{4}\sqrt{2gH}\left[1 - \frac{d^2}{128H^2} - \frac{5d^4}{16384H^4} - ....\right]$$

where C_d is the coefficient of discharge for the orifice. This is an exact expression for the discharge through a large circular orifice. Since the quantity in the brackets in Eq. 9.18 has a value less than unity, and hence the actual discharge flowing through a large circular orifice is always less than that given by the expression

$$Q = C_d \frac{\pi d^2}{4} \sqrt{2gH} \quad ...(9.19)$$

which represents the discharge through the same orifice if it is treated as a small orifice. Now if $H = d$ then from Eq. 9.18

$$Q = 0.992 \left[C_d \frac{\pi d^2}{4} \sqrt{2gH} \right] \quad ...(9.20)$$

which shows that for the head above the centre of the orifice equal to its diameter there is an error of less than even 1% in the discharge computed by the exact expression of Eq. 9.18 and the one computed by treating it as a small orifice and using Eq. 9.19. As such it may be stated that *as long as H > d i.e., the head above the centre of the orifice is greater than its diameter, it may be treated as a small orifice.*

Exact information regarding the value of C_d for large orifices is not available but a value of 0.60 may be used for approximate computations for both rectangular as well as circular orifice when the width of the rectangular orifice or the diameter of the circular orifice is about 0.3 m or more.

9.6 FLOW UNDER PRESSURE THROUGH ORIFICES

In the above analysis for the flow through the small and the large orifices it has been assumed that the liquid in the reservoir or the tank containing the orifice is at atmospheric pressure so that the pressures on the liquid surface in the reservoir and on the issuing jet are same, being equal to the atmospheric pressure. However, there may be a case when the liquid contained in the reservoir may be under pressure and it is then discharged through an orifice freely in atmosphere. Obviously in this case the pressures on the liquid surface in the reservoir and on the emerging jet are not equal. But even in this case also the same equations as derived earlier may be applied with the modification that the head on the orifice is computed as the sum of the static head of the liquid on the orifice and the head of the liquid equivalent to the pressure (gage) intensity on the liquid surface in the reservoir.

9.7 FLOW THROUGH SUBMERGED (OR DROWNED) ORIFICE

1. Totally Submerged Orifice. If an orifice has its whole of the outlet side submerged under liquid so that it discharges a jet of liquid into the liquid of the same kind then it is known as totally submerged or totally drowned orifice. Figure 9.8 shows a totally submerged orifice for which let H_1 be the height of the liquid on the upstream side above the centre of the orifice and H_2 be the height of the liquid on the downstream side above the centre of the orifice. A vena-contracta will be formed in this case also and the pressure there corresponds to the head H_2. Now considering two points 1 and 2; point 1 being in the reservoir on the upstream side of the orifice and point 2 being at vena-contracta as shown in Fig. 9.8, and applying Bernoulli's equation between the points 1 and 2, we have, neglecting the loss of energy between these points,

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2$$

But
$$\frac{p_1}{w} = (H_1 + z_2 - z_1)$$

and if the velocity V_1 at point 1 is negligible then by substitution the velocity at vena-contracta is

$$V_2 = \sqrt{2g(H_1 - H_2)} \quad ...(9.21)$$

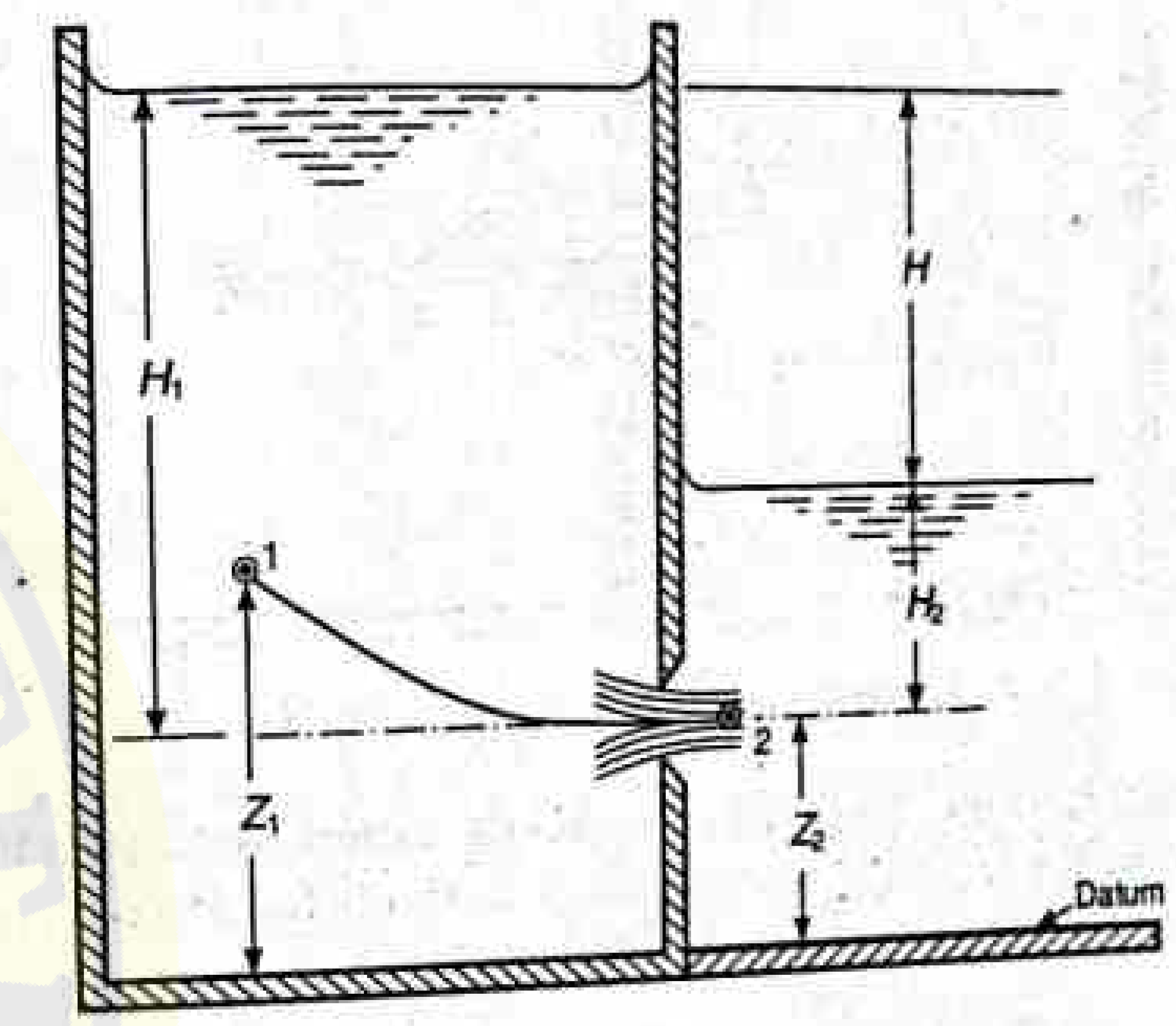


Figure 9.8 *Totally submerged orifice*

Equation 9.21 indicates that the Torricelli's formula is still applicable in this case provided that the head refers to the difference of head $(H_1 - H_2)$ on the two sides of the orifice. Since the loss of energy has been neglected, Eq. 9.21 gives only an ideal (or theoretical) velocity of flow through the orifice. Now if a is the cross-sectional area of the orifice, then the ideal (or theoretical) discharge through the orifice is $\left[a \times \sqrt{2g(H_1 - H_2)}\right]$ and introducing the coefficient of discharge C_d, the actual discharge through a totally submerged orifice is given by

$$Q = C_d\, a \sqrt{2g(H_1 - H_2)} \quad ...(9.22)$$

In a totally submerged orifice the issuing jet is interfered with the liquid present on the outlet side of the orifice. This results in slightly reducing the coefficient of discharge. As such the coefficient of discharge for a totally submerged orifice is slightly less than that for an orifice discharging free.

2. Partially Submerged Orifice. If the outlet side of an orifice is only partly submerged under liquid then it is known as partially submerged or partially drowned orifice. As such in a partially submerged orifice its upper portion behaves as an orifice discharging free, while the lower portion behaves as a submerged orifice. Obviously, only a large orifice having its vertical dimension sufficiently large can behave as a partially submerged orifice. The discharge through a partially submerged orifice may be determined by computing separately the discharge through the free and the submerged portions and then adding together the two discharges thus computed.

Figure 9.9 shows a partially submerged orifice. Let H_1 be the height of the liquid on the upstream side above the bottom edge of the orifice, H_2 be the height of the liquid on the upstream side above the

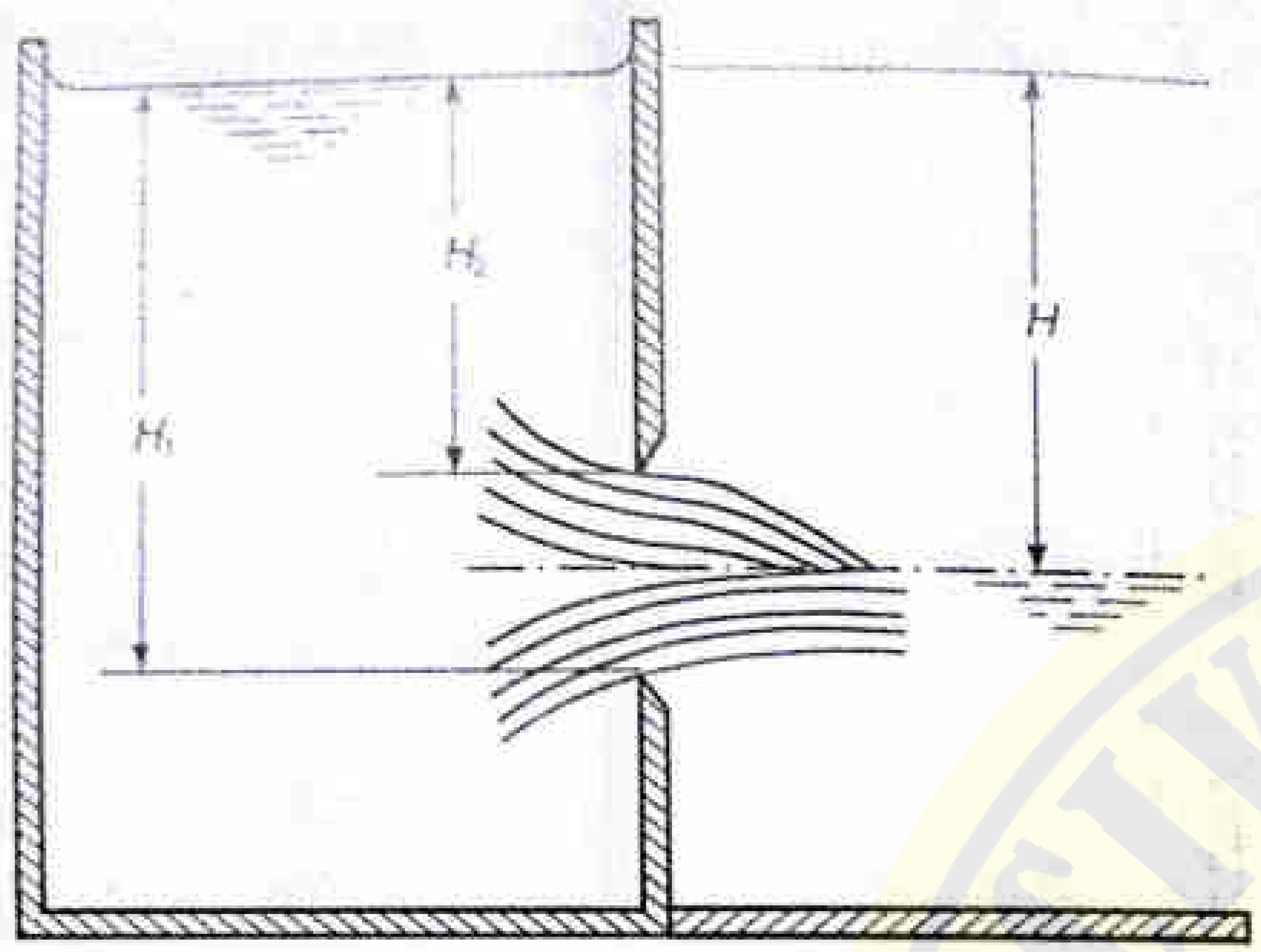


Figure 9.9 *Partially submerged orifice*

top edge of the orifice; and H be the difference between the liquid surfaces on the upstream and the downstream sides of the orifice. Now if Q_1 and Q_2 are respectively the discharges through the free and the submerged portions, then the total discharge Q through the orifice is

$$Q = Q_1 + Q_2 \qquad ...(9.23)$$

Again if the breadth of the orifice is b then from Eq. 9.15

$$Q_1 = \frac{2}{3} C_{d_1} b \sqrt{2g} \left\{H^{3/2} - H_2^{3/2}\right\}$$

where C_{d_1} represents the coefficient of discharge for the free portion of the orifice. Similarly for the submerged portion, area a is equal to $\{b \times (H_1 - H)\}$ and from Eq. 9.22

$$Q_2 = C_{d_2} b (H_1 - H) \sqrt{2gH}$$

where C_{d_2} represents the coefficient of discharge for the submerged portion of the orifice. Then

$$Q = \frac{2}{3} C_{d_1} b \sqrt{2g} \left\{H^{3/2} - H_2^{3/2}\right\} + C_{d_2} b (H_1 - H) \sqrt{2gH} \qquad ...(9.24)$$

9.8 ENERGY OR HEAD LOSSES OF FLOWING LIQUID DUE TO SUDDEN CHANGE IN VELOCITY

When the velocity of the flowing liquid changes, either in magnitude or direction, there is a large-scale turbulence generated due to the formation of the eddies in which a portion of the energy possessed by the flowing liquid is utilized which is ultimately dissipated as heat, and hence it is to be considered as lost. The change in the magnitude of the velocity of flow of liquid is due to the change in the cross-sectional area of the flow passage, and the change in the direction of the velocity of flow of liquid is due to the change in the direction of the flow passage. The change in the velocity of flow of liquid may be

either gradual or sudden and in both the cases the energy is lost. But as compared with the gradual change of velocity, the loss of energy is much more when the sudden change of velocity takes place. Since the losses of energy have been found to vary as the square of the mean velocity of flow, they are frequently expressed in terms of the velocity head of the flowing liquid. Some of the losses of energy which may be caused due to the change of velocity are as indicated below:

(a) Loss of energy due to sudden enlargement.

(b) Loss of energy due to sudden contraction.

(c) Loss of energy at the entrance to a pipe from the large vessel.

(d) Loss of energy at the exit from a pipe.

(e) Loss of energy due to an obstruction in the flow passage.

(f) Loss of energy due to gradual contraction or enlargement.

(g) Loss of energy in bends.

(h) Loss of energy in various pipe fittings.

All the above noted losses of energy are termed as 'minor' losses, because the magnitude of these losses is generally quite small as compared with the loss of energy due to friction in long pipes which is distinguished as 'major' loss. Although the turbulence produced due to the change of velocity may persist for a considerable distance downstream and on account of which the flow after the sudden change of velocity is rendered extremely complicated, yet in general all the above noted minor losses are confined to a very short length of the passage of the flowing liquid. Therefore, it is possible to derive analytically the expressions representing the approximate losses of energy in some of the above cases as indicated below. However, the actual losses of energy in all the cases may be determined only experimentally.

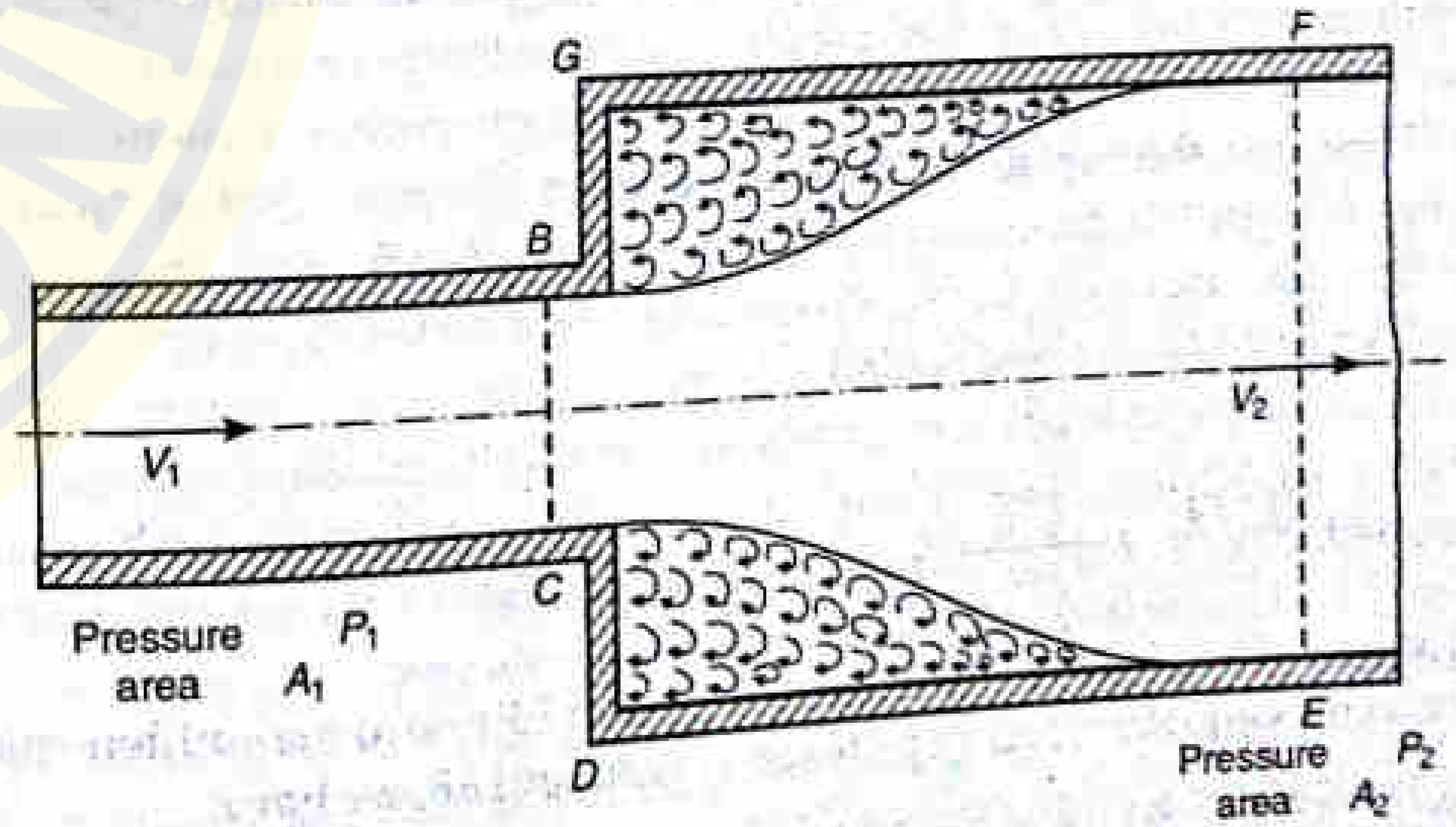


Figure 9.10 *Flow through a sudden expansion in a pipe*

(*a*) *Loss of energy due to sudden enlargement.* The loss of energy that occurs when a pipe of certain diameter suddenly expands to a large diameter, can be determined by applying the impulse momentum equation in addition to the Bernoulli's equation and the continuity equation. As shown in Fig. 9.10, consider a pipe of cross-sectional area A_1 and carrying a liquid of specific weight w, connected to another pipe of larger cross-sectional area A_2, at a certain section. Since there is sudden change in the cross-sectional area of the flow passage, the liquid emerging from the smaller pipe is not able to follow the abrupt change of the boundary. Consequently at this section the flow separates from the boundary, forming regions of separation in which turbulent eddies are formed which result in the loss of energy which is ultimately dissipated as heat. The region of separation however extends upto some distance on the downstream where the flowing liquid again follows the boundary of the pipe of larger cross-sectional area.

Consider a control volume $BCDEFG$, as shown in Fig. 9.10, which includes the non-uniform region of separation and its end sections BC and EF are located in the narrower and the wider pipes respectively, where the velocities may be assumed to be practically uniform. Let V_1 and V_2 be the velocities of flow of liquid in the narrower and the wider pipes respectively. From continuity, since $Q = A_1V_1 = A_2V_2$, the velocity V_2 is smaller than V_1 and hence a change of momentum takes place as the liquid flows from the narrower pipe to the wider pipe. This change of momentum per second will be equal to the net force acting in the direction of flow on the liquid in the control volume $BCDEFG$. Now if p_1 and p_2 are the respective pressure intensities at the end sections BC and FE of the control volume and p' is the mean pressure of the eddying fluid over the annular face GD joining the two pipes, then the force acting on the liquid in the control volume in the direction of flow is

$$[p_1A_1 + p'(A_2 - A_1) - p_2A_2]$$

It is experimentally found that the pressure p' is equal to pressure p_1. The net force is thus equal to $(p_1 - p_2)A_2$. The momentum of liquid passing through section BC is equal to $\left(\frac{wQV_1}{g}\right)$ and that passing through section FE is equal to $\left(\frac{wQV_2}{g}\right)$ and hence the change of momentum per second is equal to $\left[\frac{wQ(V_2 - V_1)}{g}\right]$. Then from the impulse momentum equation

$$(p_1 - p_2)A_2 = \frac{wQ}{g}(V_2 - V_1)$$

or

$$\frac{p_1 - p_2}{w} = \frac{1}{g}\frac{Q}{A_2}(V_2 - V_1) = \frac{1}{g}V_2(V_1 - V_2) \quad \text{...(i)}$$

Now if h_L is the loss of head between the sections BC and EF due to the sudden enlargement, then applying the Bernoulli's equation between the sections BC and FE, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_L$$

For the sake of convenience the pipe has been assumed to be horizontal, so that $z_1 = z_2$ and hence

$$h_L = \left(\frac{p_1}{w} - \frac{p_2}{w}\right) + \frac{V_1^2}{2g} - \frac{V_2^2}{2g} \quad \text{...(ii)}$$

By substituting for $\frac{(p_1 - p_2)}{w}$ from Eq. (i) in Eq. (ii) we get

$$h_L = \frac{1}{g}V_2(V_2 - V_1) + \frac{V_1^2}{2g} - \frac{V_2^2}{2g}$$

or

$$h_L = \frac{(V_1 - V_2)^2}{2g} \quad \text{...(9.25)}$$

Equation 9.25 gives the loss of head due to sudden enlargement. Since by continuity, $A_1V_1 = A_2V_2$, Eq. 9.25 may also be alternatively expressed as

$$h_L = \frac{V_1^2}{2g}\left(1 - \frac{A_1}{A_2}\right)^2 = \frac{V_2^2}{2g}\left(\frac{A_2}{A_1} - 1\right)^2 \quad \text{...(9.26)}$$

The expression for the loss of head due to sudden enlargement was first obtained by J.C. Borda (1753–99) and L. Carnot (1738–1823) and is therefore sometimes known as the *Borda-Carnot equation* for head loss.

(*b*) *Loss of energy due to sudden contraction*. Consider a pipe carrying some liquid of specific weight w, whose cross-sectional area at a certain section reduces abruptly from A_1 to A_2 as shown in Fig. 9.11. Geometrically a sudden contraction is reverse of the sudden enlargement but it is not possible to apply the inpulse momentum equation to a control volume between sections 1–1 and 2–2 in the case of sudden contraction. This is because just upstream of the junction between the two pipes, in the wider pipe the streamlines are curved and the liquid is accelerated, due to which the pressure at the annular face varies in an unknown manner which cannot be determined easily. Moreover, in the region just upstream of the junction there being converging or accelerating flow no major loss of energy occurs. However, immediately downstream of the junction as the liquid flows from the wider pipe to the narrower pipe a vena-contracta is formed, after which the stream of liquid widens again to fill completely the narrower pipe. In between the vena-contracta and the wall of the pipe a lot of eddies are formed which cause a considerable dissipation of energy. It is due to the formation of these eddies and the consequent dissipation of energy that the most of the loss of the energy is caused in the case of sudden contraction. Between the vena-contracta and section 2–2 at a certain distance away from it where the velocity has again become almost uniform, the flow pattern is similar to that after a sudden enlargement. As such Eq. 9.25 may be applied between the sections at vena-contracta and 2–2 to obtain an approximate value of the loss of head due to sudden contraction. Thus if A_c is the cross-sectional area of the stream at vena-contracta, and V_c and V_2 are the velocities of the flow of liquid at vena-contracta and in the narrower pipe at section 2–2 respectively, then applying Eq. 9.25 the loss of head due to sudden contraction is obtained as

$$h_L = \frac{(V_c - V_2)^2}{2g}$$

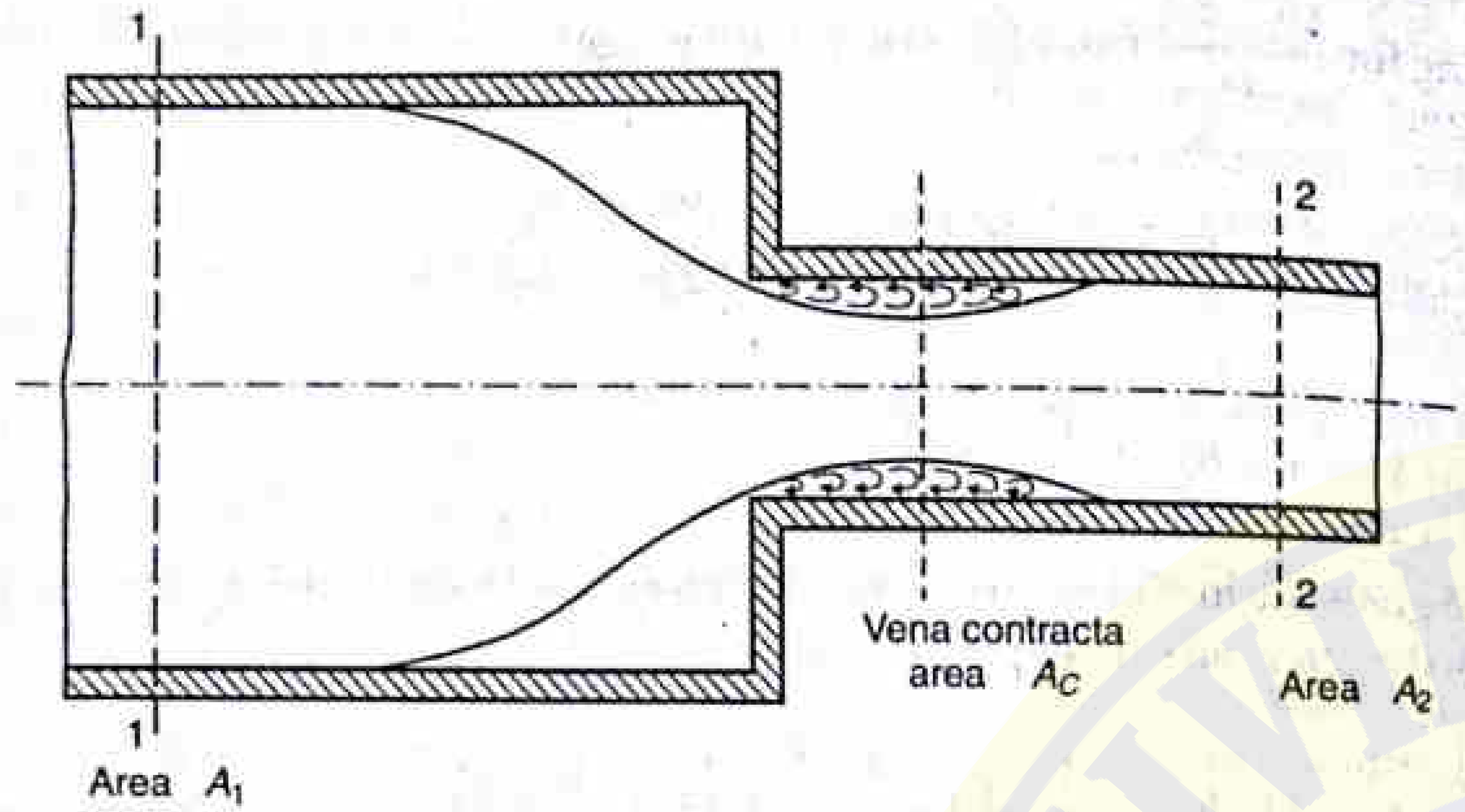


Figure 9.11 *Flow through a sudden contraction in a pipe*

Again by continuity, $A_cV_c = A_2V_2$ and hence by substitution

$$h_L = \frac{V_2^2}{2g}\left(\frac{A_2}{A_c}-1\right)^2 \qquad ...(9.27)$$

Further since the ratio of the area of the jet at a vena-contracta and the area of the opening is defined as a coefficient of contraction C_c. Thus in this case

$$C_c = \left(\frac{A_c}{A_2}\right)$$

and hence Eq. 9.27 becomes

$$h_L = \frac{V_2^2}{2g}\left(\frac{1}{C_c}-1\right)^2 \qquad ...(9.28)$$

Often the loss of head due to sudden contraction is expressed as

$$h_L = k\frac{V_2^2}{2g} \qquad ...(9.29)$$

The value of C_c or k is however not constant but depends on the ratio $\left(\frac{A_2}{A_1}\right)$. Table 9.1 gives some of the values of the coefficient k for the different values of $\left(\frac{D_2}{D_1}\right)$, the ratio of the diameters of the narrower and the wider pipes, which may be adopted for the computation of the loss of head due to sudden contraction.

TABLE 9.1

$\frac{D_2}{D_1}$	0	0.2	0.4	0.6	0.8	1.0
k	0.5	0.45	0.38	0.28	0.14	0

If C_c is assumed to be equal to 0.62 then by substituting in Eq. 9.28

$$h_L = \frac{V_2^2}{2g}\left(\frac{1}{0.62}-1\right)^2 = 0.375\frac{V_2^2}{2g}$$

i.e., $$k = 0.375.$$

However, in general the loss of head due to sudden contraction is taken to be equal to $0.5\frac{V_2^2}{2g}$ i.e., the value of k is adopted as 0.5.

(c) *Loss of energy at the entrance to a pipe from the large vessel.* When a liquid enters a pipe from a large vessel (or tank or reservoir) some loss of energy occurs at the entrance to the pipe which is sometimes known as inlet loss of energy. The flow pattern at the entrance to the pipe is similar to that in the case of a sudden contraction and hence the loss of energy at the entrance to the pipe is assumed to be the same as in the case of sudden contraction. The loss of energy actually depends on the form of the entrance. Thus for a rounded or bell-mouthed entrance the loss of energy is relatively much less as compared with a sharp-cornered entrance. In general, for a sharp-cornered entrance the loss of head at the entrance is taken to be equal to $0.5\frac{V^2}{2g}$, where V is the mean velocity of flow of liquid in the pipe.

(d) *Loss of energy at the exit from a pipe.* The outlet end of a pipe carrying a liquid may be either left free so that the liquid is discharged freely in the atmosphere or it may be connected to a large reservoir so that the pipe outlet becomes submerged and the liquid is discharged into a large body of static liquid. The liquid leaving the pipe at its outlet end still-possesses a kinetic energy corresponding to the velocity of flow of liquid in the pipe, which is ultimately dissipated either in the form of a free jet or it is lost in turbulence in the reservoir, depending on the condition of the outlet. Therefore, the loss of head at the exit from a pipe is equal to $\frac{V^2}{2g}$, when V is the mean velocity of flow in the pipe.

When the outlet end of a pipe is connected to a large reservoir, the flow pattern is similar to that of the sudden enlargement and hence the loss of head may also be determined by using Eq. 9.26 with the condition that in this case $A_2 \to \infty$. The loss of head at the exit of the pipe is then equal to $\frac{V^2}{2g}$, where V is the velocity of flow of liquid in the pipe.

(e) *Loss of energy due to an obstruction in the flow passage.* The loss of energy due to an obstruction in a pipe takes place on account of the reduction in the cross-sectional area of the pipe by the presence of the obstruction which is followed by an abrupt enlargement of the stream beyond the obstruction. As such the loss of head due to an obstruction may be computed by applying Eq. 9.25.

Consider a pipe of cross-sectional area A, and let an obstruction of maximum cross-sectional area a be placed in the pipe, as shown in Fig. 9.12. Thus at section 1–1 the area of the flow passage is reduced to $(A-a)$. As the liquid flows through the pipe a vena-contracta is formed just beyond section 1–1, after which the stream of liquid widens again. Between the vena-contracta and section 2-2 at a certain distance away from the obstruction on the downstream where the velocity has again become almost uniform, the flow pattern is similar to that after a sudden enlargement. Thus if V_c and V are respectively the velocities at vena-contracta and section 2–2, then applying Eq. 9.25 the loss of head due to the obstruction is obtained as

$$h_L = \frac{(V_c - V)^2}{2g}$$

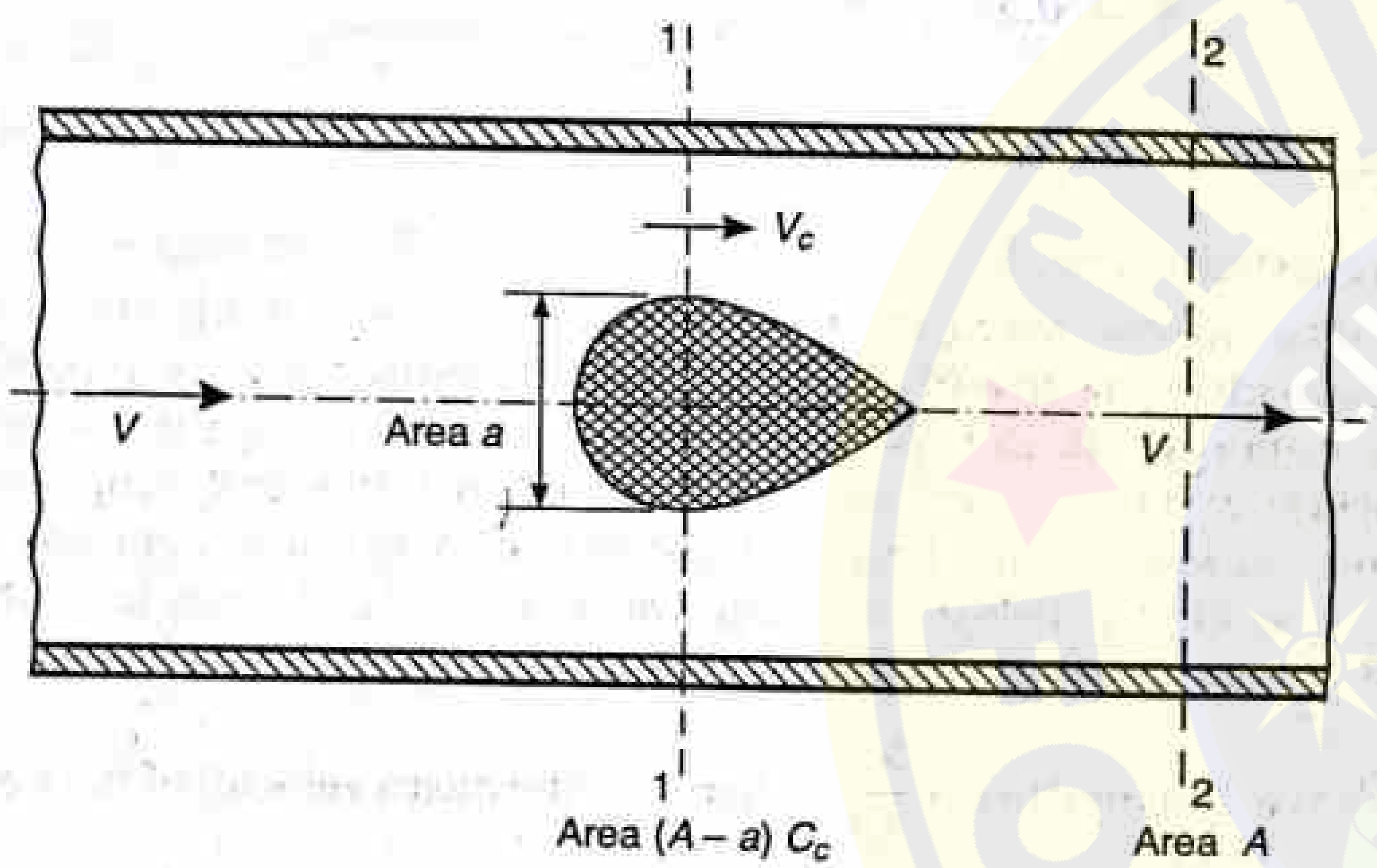


Figure 9.12 *Flow through a pipe with an obstruction*

Further, if C_c is the coefficient of contraction then the area of the stream at vena-contracta is $C_c(A-a)$ and by continuity equation

$$C_c(A-a)\,V_c = AV\,;\ \text{or}\ V_c = \frac{A}{C_c(A-a)}\,V$$

By substituting V_c, we have

$$h_L = \left[\frac{A}{C_c(A-a)} - 1\right]^2 \frac{V^2}{2g} \qquad \text{...(9.30)}$$

Equation 9.30 thus represents the loss of head due to an obstruction in the flow passage. The value of C_c is however not constant but it depends on the type of obstruction. Therefore it has to be suitably assumed. For general purposes it may be assumed to vary between 0.60 and 0.66.

(*f*) *Loss of energy due to gradual contraction or enlargement.* The loss of energy can be considerably reduced if in place of a sudden contraction or a sudden enlargement a gradual contraction or gradual enlargement is provided. This is so because in a gradual contraction or a gradual enlargement the velocity of the fluid is gradually increased or reduced, and thus, as far as possible, the formation of the

eddies responsible for the dissipation of energy is eliminated. Generally the loss of head in a gradual contraction or a gradual enlargement may be expressed as

$$h_L = k\left(\frac{V_1 - V_2}{2g}\right)^2 \qquad \text{...(9.31)}$$

where k is a coefficient and V_1 and V_2 are the mean velocities at the inlet and the outlet. In general, the value of k depends on the angle of convergence or divergence and on the ratio of the upstream and the downstream cross-sectional areas. However, for a gradual contraction the value of k is usually very small even for larger values of the angle of convergence. Therefore, for a gradual contraction without sharp corners the loss of energy caused is so small that it may usually be neglected. On the other hand, for a gradual enlargement the value of k considerably depends on the angle of divergence. The value of k increases as the angle of divergence increases for a given ratio of the cross-sectional areas at the inlet and the outlet. Moreover in the case of a gradual enlargement except for very small angles of divergence, the flow of fluid is always subjet to separation from the boundaries, and the consequent formation of the eddies result in the loss of energy. Therefore in the case of gradual enlargement the loss of energy cannot be completely eliminated.

(*g*) *Loss of energy in bends.* The bends are provided in a flow passage to change the direction of flow. The change in the direction of flow also results in causing loss of energy. The loss of energy in the bends is due to the separation of flow from the boundary and the consequent formation of the eddies resulting in the dissipation of energy in turbulence. In general the loss of head in bends provided in pipes may be expressed as

$$h_L = k\frac{V^2}{2g} \qquad \text{...(9.32)}$$

where k is a coefficient and V is the mean velocity of flow of fluid. The value of k however depends on the total angle of the bend and on the relative radius of curvature R/d, where R is the radius of curvature of the pipe axis and d is the diameter of the pipe.

(*h*) *Loss of energy in various pipe fittings.* All pipe fittings such as valves, couplings etc., cause a loss of energy. The loss of head in the various pipe fittings may also be represented as

$$h_L = k\frac{V^2}{2g} \qquad \text{...(9.33)}$$

where k is a coefficient and V is the mean velocity of flow in the pipe. The value of the coefficient k actually depends on the type of the pipe fitting.

9.9 FLOW THROUGH AN EXTERNAL CYLINDRICAL MOUTHPIECE

A mouthpiece is a short tube fitted to a circular orifice provided in a tank or a reservoir. As indicated below by fitting a mouthpiece the discharge through an orifice may be increased. The increase in the discharge would, however, result only if the mouthpiece is running full so that the jet of liquid emerging from the mouthpiece is of the same diameter as that of the mouthpiece. A mouthpiece will be running full if its length is equal to about two to three times its diameter and also the head on the mouthpiece does not exceed certain value as indicated below. However, if the length of the mouthpiece is less and nearly equal to its diameter, then even at low heads it will be running free in which case the jet of liquid will emerge from the mouthpiece without touching its sides and it will behave as an orifice only. Since the external cylindrical mouthpieces generally run full, only this condition of discharge has been considered for these mouthpieces.

Consider a tank having a cylindrical mouthpiece of cross-sectional area a, attached externally to one of its sides, as shown in Fig. 9.13, and filled with a liquid of specific weight w upto a constant height H above the centre of the mouthpiece. The liquid will emerge through the mouthpiece in the form of a free jet. Since the entrance section of the mouthpiece is exactly same as a sharp-edged orifice, the jet of liquid entering the mouthpiece contracts to form a vena-contracta at a section cc only a short distance from the entrance-section. Beyond the section cc the jet again expands to fill the mouthpiece completely, so that the jet of liquid that emerges from the mouthpiece has its cross-sectional area equal to that of the mouthpiece.

Now if a_c is the cross-sectional area of the jet at vena-contracta and C_c is the coefficient of contraction for the orifice (at the entrance to the mouthpiece) then

$$a_c = C_c a$$

Assuming a value of the coefficient of contraction C_c to be 0.62 then

$$a_c = 0.62a$$

Further, if V_c is the velocity of the jet at vena-contracta and V is the velocity of the jet at the outlet of the mouthpiece then by continuity

$$a_c V_c = aV$$

or

$$V_c = \frac{a}{a_c} V = \frac{V}{0.62}$$

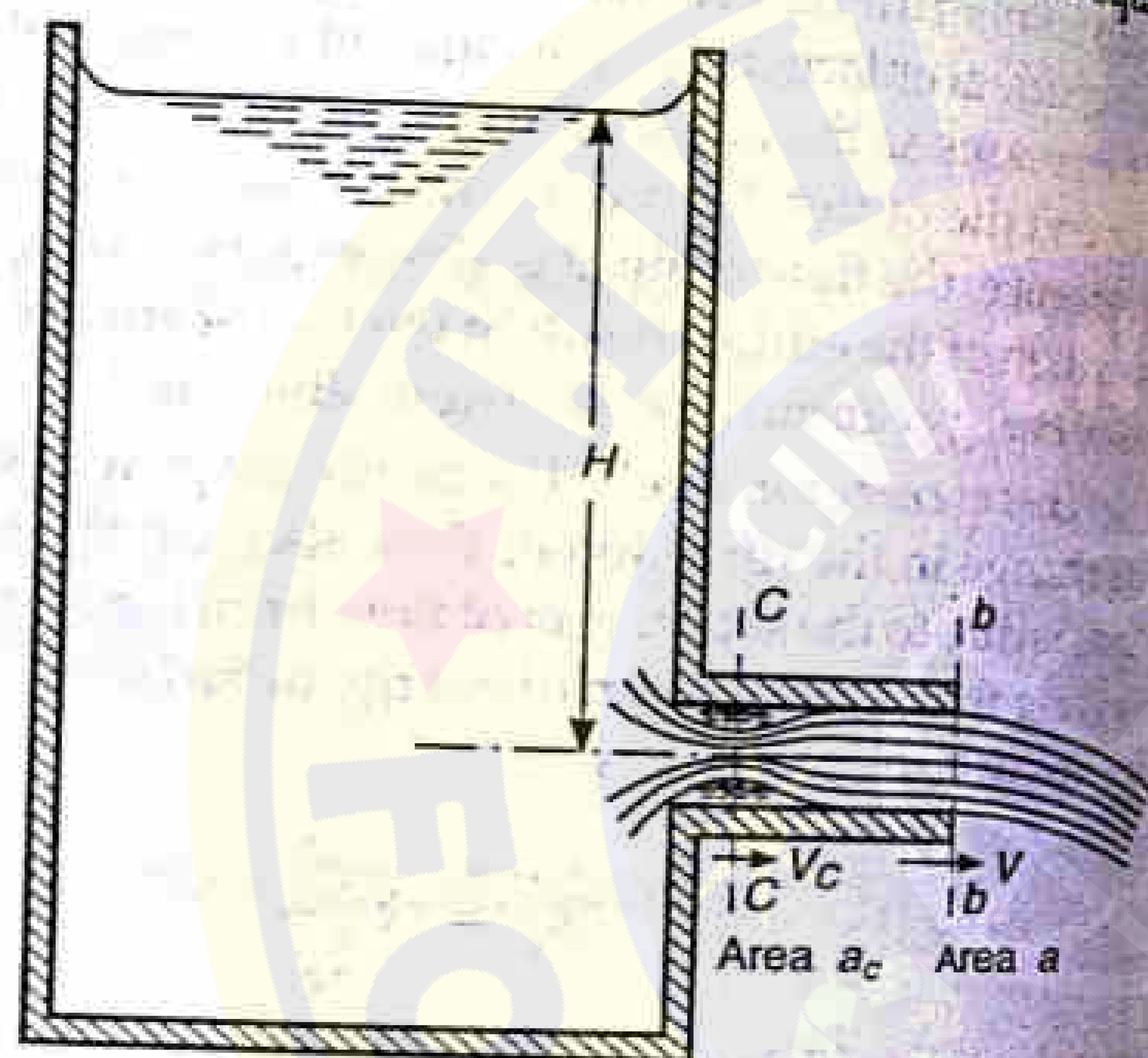


Figure 9.13 *Flow through an external cylindrical mouthpiece*

If H_a represents the atmospheric pressure head then applying Bernoulli's equation between the free surface of the liquid in the tank and a section bb just at the outlet of the mouthpiece, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

where h_L represents the loss of head between the sections cc and bb. Between the sections cc and bb the flow pattern is similar to that after a sudden enlargement and hence the loss of head h_L between these sections is that due to sudden enlargement which may be computed by Eq. 9.25. Thus

$$h_L = \frac{(V_c - V)^2}{2g} = \frac{\left(\frac{V}{0.62} - V\right)^2}{2g} = 0.375\frac{V^2}{2g}$$

By substituting the value of h_L, we have

$$H_a + H = H_a + \frac{V^2}{2g} + 0.375\frac{V^2}{2g}$$

or

$$H = 1.375\frac{V^2}{2g}$$

$$\therefore \quad V = \sqrt{\frac{2gH}{1.375}} = 0.855\sqrt{2gH}$$

Again the theoretical velocity of flow of liquid through the mouthpiece is

$$V_{th} = \sqrt{2gH}$$

Therefore the coefficient of velocity for the mouthpiece is

$$C_v = \frac{V}{V_{th}} = \frac{0.855\sqrt{2gH}}{\sqrt{2gH}} = 0.855$$

Further the coefficient of contraction C_c for the mouthpiece is equal to 1, because the cross-sectional area of the jet emerging from the mouthpiece is equal to the cross-sectional area of the mouthpiece. Therefore the coefficient of discharge for the mouthpiece is

$$C_d = C_c \times C_v = 1 \times 0.855 = 0.855$$

However, in actual practice the frictional resistance reduces the value of the coefficient of velocity for the mouthpiece and the actual value of C_v for an external cylindrical mouthpiece is approximately 0.82. Accordingly the actual value of C_d for a mouthpiece is approximately 0.82. The discharge through the mouthpiece is

$$Q = C_d a\sqrt{2gH} = 0.82\, a\sqrt{2gH}$$

Since the coefficient of discharge for an external cylindrical mouthpiece is more than that for a standard orifice, the discharge through such a mouthpiece is greater than that through a standard orifice of the same diameter under the same head.

In order to determine the pressure at the vena-contracta apply Bernoulli's equation between the free surface of the liquid in the tank and the section cc. Thus if H_c represents the absolute pressure head at vena-contracta, then

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

Since the flow of liquid from the tank upto section cc is a converging type of flow, the loss of energy may be neglected. Then

$$H_c = H_a + H - \frac{V_c^2}{2g}$$

But $V_c = \frac{V}{0.62}$ and $H = 1.375\frac{V^2}{2g}$, so by substitution, we get

$$H_c = H_a + 1.375\frac{V^2}{2g} - \frac{1}{2g}\left(\frac{V}{0.62}\right)^2$$

or
$$H_c = H_a - 1.225\frac{V^2}{2g} \quad ...(9.34)$$

or
$$H_c = H_a - \frac{1.225}{1.375}H$$

or
$$H_c = H_a - 0.89\,H \quad ...(9.35)$$

Equation 9.35 is however only theoretical because as stated earlier in actual practice the value of C_v is 0.82 which is slightly less than the theoretical value of 0.855. Thus considering the actual value of C_v

$$V = 0.82\sqrt{2gH}$$

or
$$\frac{V^2}{2g} = (0.82)^2 H$$

By substituting in Eq. 9.34

$$H_c = H_a - 1.225\,(0.82)^2 H$$

or
$$H_c = H_a - 0.82\,H \quad ...(9.36)$$

From Eqs 9.34 and 9.36, it is observed that the pressure head at vena-contracta is less than atmospheric pressure by $1.225\frac{V^2}{2g}$ or $0.82\,H$. In other words, the pressure at vena-contracta is vacuum or suction pressure. In fact, it is this reduction in the pressure at the vena-contracta in the case of a mouthpiece that results in increasing the effective head causing the flow, thereby increasing the discharge through the mouthpiece.

The minimum possible value upto which the pressure at vena-contracta may be reduced is the absolute zero pressure. Then from Eq. 9.36, the maximum or the limiting value of the head corresponding to which this minimum pressure would occur may be obtained as

$$H_c = 0 = H_a - 0.82\,H$$

or
$$H = \frac{H_a}{0.82}$$

If the discharging liquid is water, then $H_a = 10.3$ m of water

and
$$H = \frac{10.3}{0.82} = 12.56 \text{ m}$$

Thus, if the head H is equal to 12.56 m of water then the pressure at vena-contracta is reduced to absolute zero. However in actual practice the pressure at vena-contracta cannot be reduced below the vapour pressure of the flowing liquid. As such in actual practice when the head H becomes equal to only about 12.2 m of water the pressure at vena-contracta becomes equal to the vapour pressure of the flowing water, which then starts vapourizing and due to the bubbles of the vapour so released, the flow becomes unsteady. A further increase in the head H causes the velocity of the central core of the emerging jet of liquid to become so great that it sweeps aside the annulus of eddying liquid that surrounds it. The jet then comes out clear of the wall of the mouthpiece, so that it behaves as if the mouthpiece has been removed altogether and the flow is taking place only through the orifice.

Consequently the coefficient of discharge is also reduced to about 0.6 which is same as that for an orifice. Under such condition of discharge, the external cylindrical mouthpiece is said to be running free.

9.10 FLOW THROUGH A CONVERGENT DIVERGENT MOUTHPIECE

In the case of a cylindrical mouthpiece, on account of the formation of the vena-contracta and the subsequent enlargement of the jet there occurs a loss of energy, which results in reducing the coefficient of discharge of the mouthpiece. However, if the mouth-piece is made to conform to the shape of the jet upto vena-contracta then the loss of energy may be eliminated. Such a mouthpiece is known as *convergent mouthpiece*. Theoretically, the coefficient of discharge for a convergent mouthpiece is equal to unity, but in actual practice on account of frictional resistance some energy is lost due to which the coefficient of discharge for this mouthpiece is about 0.975, Fig. 9.14 (a).

The loss of energy owing to the sudden enlargement of a jet in a mouthpiece can however be eliminated by making the mouthpiece gradually diverging. Such a mouthpiece is known as *divergent mouthpiece*. The value of the coefficient of discharge for this mouth-piece however depends on the angle of divergence and the length of the mouthpiece, Fig. 9.14 (b)

Often a convergent-divergent mouthpiece is also used which is made convergent upto vena-contracta and then diverges as shown in Fig. 9.14 (c). In such a mouthpiece as the divergence increases the velocity at the section *cc* increases, which in turn causes an increase in the vacuum or the suction pressure at vena-contracta. Since the pressure at vena-contracta cannot be reduced below a certain value (which is theoretically equal to the absolute zero but practically equal to the vapour pressure of the flowing liquid), there is a limit to the amount of divergence for a steady flow to be maintained. Thus for a convergent-divergent mouthpiece the value of the maximum divergence which may be provided for a convergent-divergent mouthpiece may be determined as indicated below.

Applying Bernoulli's equation between the free liquid surface in the reservoir and the sections *cc* and *bb* as shown in Fig. 9.14 (c) we have neglecting the losses,

$$H_a + H = H_c + \frac{V_c^2}{2g} = H_a + \frac{V^2}{2g}$$

where H_a is the atmospheric pressure head in terms of the flowing liquid, H is the height of the free liquid surface above the centre of the mouthpiece, H_c is the absolute pressure head at section *cc*, and V_c and V are the velocities of flow at the sections *cc* and *bb* respectively.

From the above equation

$$\frac{V^2}{2g} = H$$

and
$$\frac{V_c^2}{2g} = H + H_a - H_c$$

Now if a_c and a are the cross-sectional areas at the vena-contracta and the outlet end of the mouthpiece, then by continuity

$$a_c V_c = aV$$

or
$$\frac{a}{a_c} = \frac{V_c}{V} = \sqrt{\frac{H+H_a-H_c}{H}}$$

or
$$\frac{a}{a_c} = \sqrt{1+\frac{H_a-H_c}{H}} \quad ...(9.37)$$

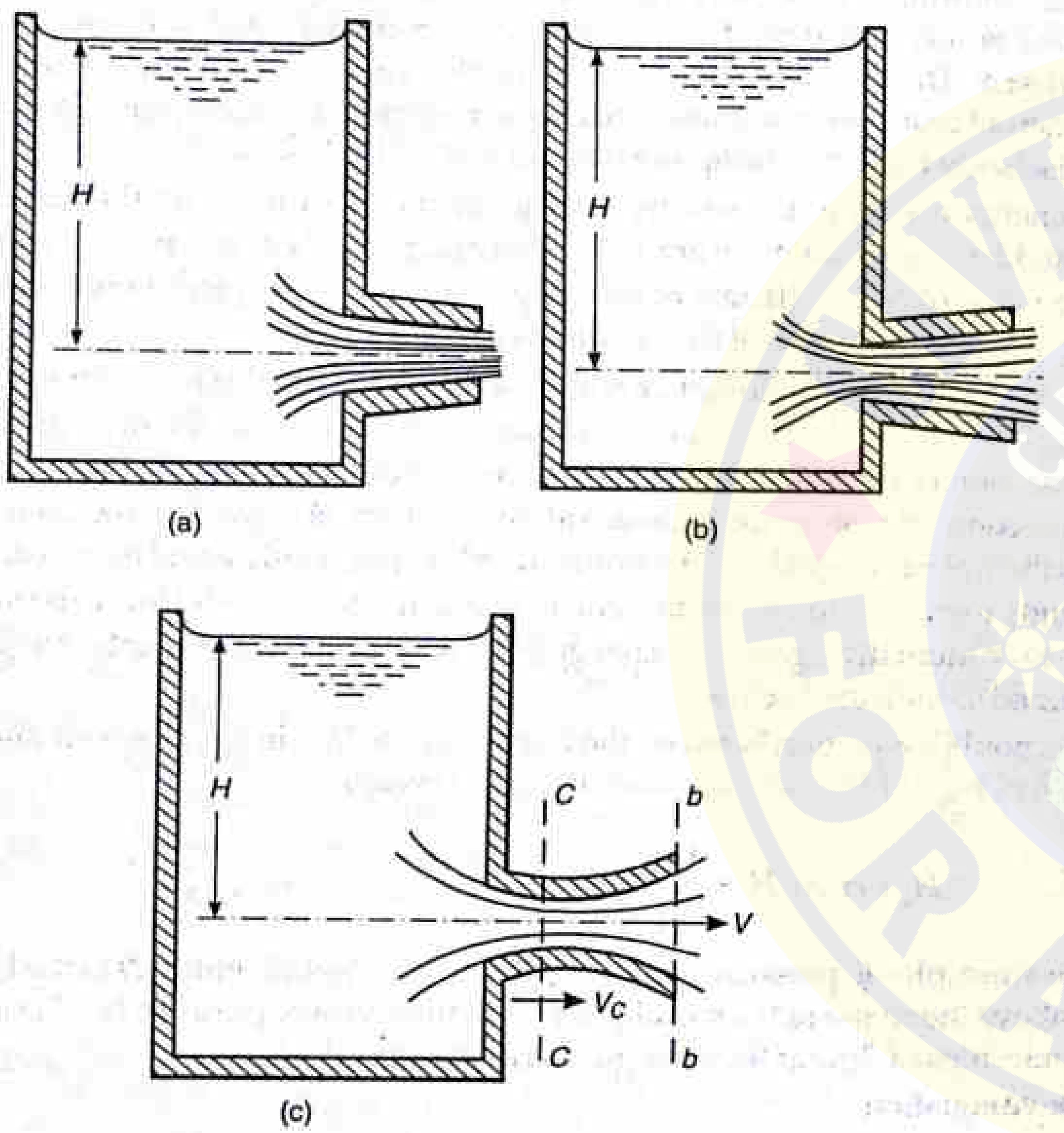


Figure 9.14 (a) *Convergent mouthpiece,* (b) *Divergent mouthpiece,* (c) *Convergent-Divergent mouthpiece*

If the flowing liquid is water then the limiting value of the suction pressure at vena-contracta $(H_a - H_c) = 7.8$ m, hence the maximum value of the ratio

$$\frac{a}{a_c} = \sqrt{1+\frac{7.8}{H}}$$

9.11 FLOW THROUGH INTERNAL OR RE-ENTRANT OR BORDA'S MOUTHPIECE

An internal mouthpiece is a short cylindrical tube attached to a circular orifice in the side of a reservoir or tank such that it projects inwardly as shown in Fig. 9.15. Since in the case of internal mouthpiece both the conditions of discharge viz., running free and running full may be developed, the same have been discussed below.

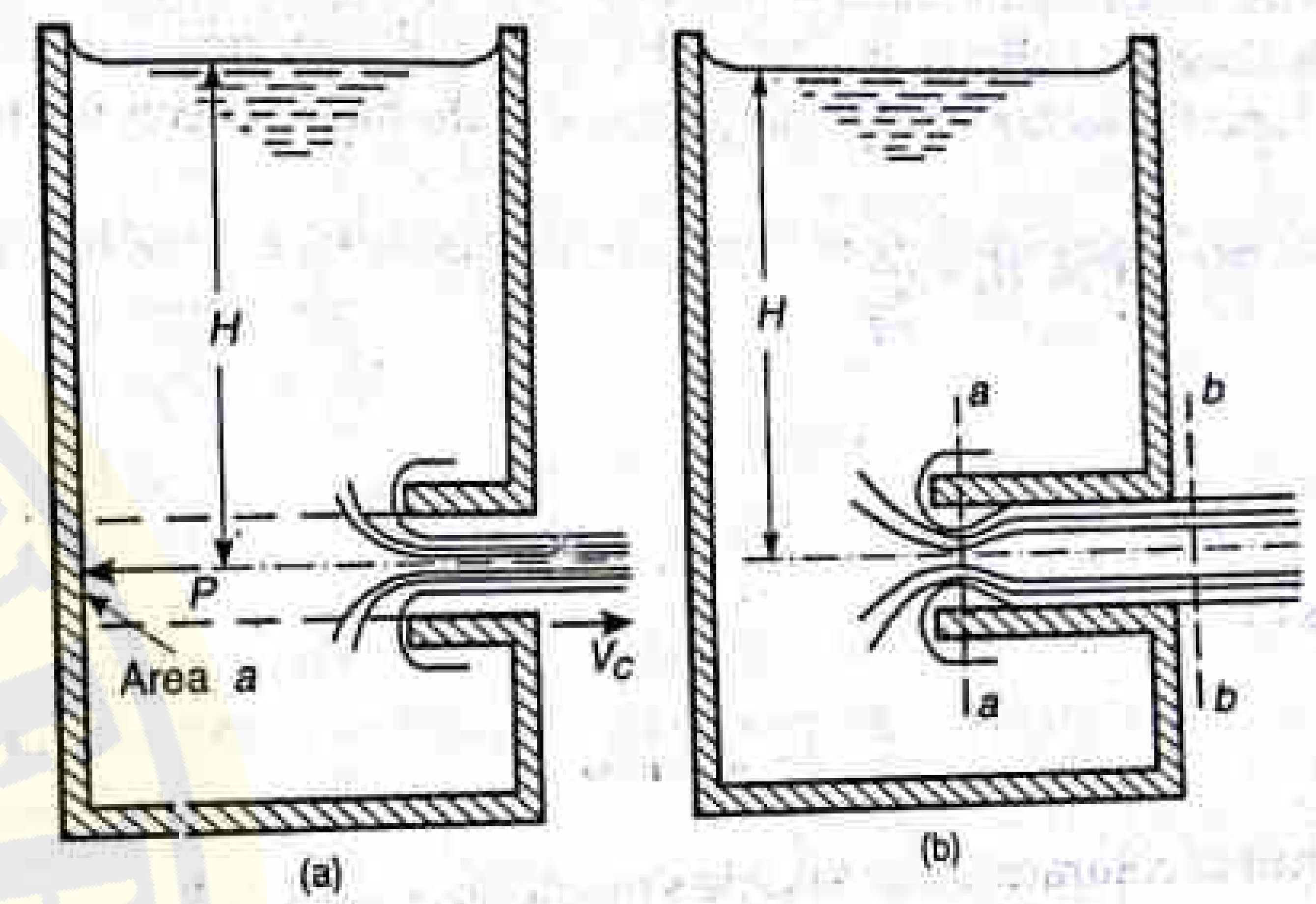


Figure 9.15 *Borda's mouthpiece (a) running free; (b) running full*

1. Borda's Mouthpiece Running Free. Consider a Borda's mouthpiece of cross-sectional area a, discharging liquid under a constant head H above the centre of the mouthpiece. As the jet of liquid enters the mouthpiece it contracts and vena-contracta is formed. Assuming that the length of the mouthpiece is small the jet cannot expand to fill the mouthpiece before it emerges from the mouthpiece, and hence it runs free as shown in Fig.9.15 (a). Let a_c be the cross-sectional area of the emerging jet and V_c be its velocity. The value of the coefficient of contraction C_c for such a mouthpiece can be analytically determined by the application of the impulse-momentum equation and the energy equation.

If we consider a circle of area a on the left hand wall of the tank, exactly opposite the orifice, so that if the mouthpiece were prolonged it would meet the wall in this circle, then the total pressure (or static thrust) on the area a is $P = waH$. But on the right hand wall there can be no counter balancing thrust because of the opening in the wall. Therefore it is the unbalanced reaction (waH) which may be considered to be acting at the entrance section of the mouthpiece and causing the liquid to flow through the mouthpiece. The force thus acting is equal to the rate of change of momentum of the flowing liquid. The mass of liquid flowing through the mouthpiece per second is $\left(\frac{wa_cV_c}{g}\right)$ and the change in the velocity of flow of liquid is from 0 in the tank to V_c as the liquid flows through the mouthpiece. Hence the rate of change of momentum imposed by the force (waH) is $\left(\frac{wa_cV_c}{g}\right)V_c$. Thus applying the impulse-momentum equation, we have

$$waH = \left(\frac{wa_c V_c}{g}\right) V_c$$

$$\frac{a_c}{a} = \frac{gH}{V_c^2} \quad ...(i)$$

Since the jet of liquid is not expanding and also the jet emerges clear of the walls of the mouthpiece, the loss of energy is altogether eliminated. As such applying the energy equation between the free surface of the liquid and the section aa of the jet just outside the mouthpiece, we have

$$H_a + H = H_a + \frac{V_c^2}{2g}$$

or

$$\frac{V_c^2}{2g} = H \quad ...(ii)$$

By combining Eqs (i) and (ii), we get

$$C_c = \frac{a_c}{a} = \frac{1}{2} \text{ or } a_c = 0.5a$$

That is the coefficient of contraction for a Borda's mouthpiece = 0.5.

However, if some loss of energy is considered and the flowing liquid is water, then

$$V_c = 0.98\sqrt{2gH} \text{ ; or } \frac{V_c^2}{2g} = 0.96\,H$$

and

$$C_c = \frac{0.5}{0.96} = 0.52$$

This method cannot, however, be used for finding the value of C_c for a plain sharp-edged orifice. This is so because as the liquid approaches the orifice, the streamlines converge, thereby resulting in reducing the pressure below the full static head H in a zone immediately surrounding the orifice. As such the unbalanced force producing the change of momentum for the liquid flowing through the orifice is the static thrust waH, plus a reaction equivalent to the reduction in pressure in the zone surrounding the orifice, which is unknown and cannot be readily determined. On the other hand there is no such zone in the side of the tank containing the Borda's mouthpiece, since in this case still liquid is in contact with the tank walls, and pressure changes take place well towards the middle of the tank.

The value of C_v for a Borda's mouthpiece running free is equal to 1 and therefore the coefficient of discharge C_d equals C_c, that is

$$C_d = 0.5 \text{ (or } 0.52)$$

2. Borda's Mouthpiece Running Full. Consider a Borda's mouthpiece of cross-sectional area a discharging liquid under a constant head H above the centre of the mouthpiece. As shown in Fig. 9.15 (b), the jet of liquid as it enters the mouthpiece contracts and vena-contracta is formed, but since the mouthpiece is long enough the jet of liquid again expands to fill the mouthpiece completely, so that the mouthpiece is running full and the jet of liquid emerging from the mouthpiece has the same diameter

as that of the mouthpiece. The flow pattern in this case is thus same as in the case of an external cylindrical mouthpiece. Again since the issuing jet is of the same cross-sectional area as that of the mouthpiece the coefficient of contraction C_c at the outlet of the mouthpiece is equal to 1. The value of the coefficient of discharge for a Borda's mouthpiece running full may be determined as indicated below.

If a_c is the cross-sectional area of the jet at vena-contracta and C_c is the coefficient of contraction at the entrance to the mouthpiece then $a_c = C_c a$. At the entrace section of the mouthpiece since the flow pattern is same as that for the running free condition and hence

$$C_c = 0.5$$

or

$$a_c = 0.5\,a$$

Further if V_c is the velocity of the jet at vena-contracta and V is the velocity of the jet at the outlet of the mouthpiece, then by continuity

$$a_c V_c = aV$$

or

$$V_c = \frac{a}{a_c} V = \frac{V}{0.5}$$

Consider section aa at the vena-contracta and section bb just outside the mouthpiece. If H_a is the atmospheric pressure head, then applying Bernoulli's equation between the free surface of the liquid in the tank and the section bb, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

where h_L represents the loss of head between the sections aa and bb. Between the section aa and bb the flow pattern is similar to that after a sudden enlargement and hence the loss of head h_L may be computed by Eq. 9.25. Thus

$$h_L = \frac{(V_c - V)^2}{2g} = \frac{\left(\frac{V}{0.5} - V\right)^2}{2g} = \frac{V^2}{2g}$$

By substituting the value of h_L, we have

$$H_a + H = H_a + \frac{V^2}{2g} + \frac{V^2}{2g}$$

or

$$\frac{V^2}{g} = H ; V = \sqrt{gH}$$

But theoretical velocity

$$V_{th} = \sqrt{2gH}$$

$\therefore$ Coefficient of velocity

$$C_v = \frac{\sqrt{gH}}{\sqrt{2gH}} = \frac{1}{\sqrt{2}} = 0.707$$

Since C_c at the outlet of the mouthpiece running full is equal to 1, the coefficient of discharge for the Borda's mouthpiece running full is

$$C_d = C_c \times C_v = (1 \times 0.707)$$
$$= 0.707$$

However, in practice the coefficient of discharge for a Borda's mouthpiece running full is found to be slightly more than this value and it is about 0.75. This is probably on account of the fact that in actual practice the loss of energy is less than that due to sudden enlargement as considered above.

The pressure at vena-contracta may be found by applying Bernoulli's equation between the free surface of the liquid in the tank and the section *cc*. Thus if H_c represents the absolute pressure head at vena-contracta, then

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

or

$$H_c = H_a + H - \frac{V_c^2}{2g}$$

But $V_c = 2V$

and $V^2 = gH$, so by substitution

$$H_c = H_a - H \qquad ...(9.38)$$

From Eq. 9.38 it is observed that at vena-contracta formed in a Borda's mouthpiece running full, the pressure is less than the atmospheric pressure by an amout equal to the head H of the liquid above the centre of the mouthpiece.

If it is assumed that the flowing liquid is water and the limiting value of the absolute pressure at vena-contracta is 2.5 m of water, then the maximum value of H upto which the flow through the Borda's mouthpiece will remain steady may be obtained from Eq. 9.38 as

$$2.5 = 10.3 - H$$

or

$$H = 7.8 \text{ m}$$

A further increase in the head H would cause the jet to come out clear of the walls of the mouthpiece thereby resulting in running free condition of discharge for the mouthpiece.

9.12 FLOW THROUGH AN ORIFICE OR A MOUTHPIECE UNDER VARIABLE HEADS

In the preceding analysis of flow through orifice and mouth-pieces it has been assumed that the head above the centre of the orifice or the mouthpiece is constant with time, which can be achieved by adjusting the inflow of liquid into the tank equal to the outflow of liquid through the orifice or the mouthpiece. The flow through an orifice or a mouthpiece under such conditions is steady. However, if the head on an orifice or a mouthpiece is not constant with time, the flow becomes unsteady. The variation of head on an orifice or a mouthpiece may be either due to the inflow of the liquid being completely cut off or the inflow of the liquid being different from the outflow through the orifice (or mouthpiece). Under both these conditions of flow the main problem is to determine the time required for the free liquid surface to change from its initial position to some other position. Both these conditions of flow are discussed below.

(a) Time of emptying a tank with no inflow. Consider a tank containing liquid upto a height H_1 above the centre of the orifice (or mouthpiece) provided in the side or bottom of the tank. It is required to find the time t for the liquid surface to fall from the height H_1 to a height H_2 above the centre of the opening. Since the discharge through an orifice or a mouthpiece depends on the head, which is changing in this case with time, the discharge also changes with the head. Therefore the time t can be determined only by integration as indicated below.

Let at any instant the liquid surface be at a height h above the orifice (or mouthpiece) and let the liquid surface fall by a small amount dh in time dt. Now if A is the horizontal cross-sectional area of the tank then the volume of liquid leaving the tank in time dt is Adh. Again if at this instant the discharge through the orifice (or mouthpiece) is Q, then the volume of liquid discharged during the interval of time dt is Qdt. As the volume of the liquid leaving the tank is equal to the volume of the liquid flowing through the orifice (or mouthpiece) during the same interval of time, we have

$$A(-dh) = Qdt$$

The negative sign is introduced becuase as the time increases the head decreases. Further if a is the cross-sectional area of the orifice (or mouthpiece) and C_d is its coefficient of discharge, then

$$Q = C_d a\sqrt{2gh}$$

and by substitution

$$-Adh = C_d a\left(\sqrt{2gh}\right)dt$$

or

$$dt = -\frac{Adh}{C_d a\sqrt{2gh}}$$

By integrating both the sides of the above expression, we get

$$\int_0^t dt = -\int_{H_1}^{H_2} \frac{Adh}{C_d a\sqrt{2gh}}$$

or

$$t = \int_{H_1}^{H_2} -\frac{Adh}{C_d a\sqrt{2gh}} \qquad ...(9.39)$$

Equation 9.39 may be evaluated if the shape of the tank is known. The tanks of the following shapes are commonly found in practice.

(i) Cylindrical (or rectangular or prismatic, with constant horizontal cross-sectional area).

(ii) Conical.

(iii) Hemispherical.

(i) Cylindrical Tank. As shown in Fig. 9.16 (a) a cylindrical tank placed with its axis vertical has a constant horizontal cross-sectional area A, then from Eq. 9.39

$$t = -\frac{A}{C_d a\sqrt{2g}}\int_{H_1}^{H_2} h^{-1/2}\,dh$$

or

$$t = \frac{2A}{C_d a\sqrt{2g}}\left(H_1^{1/2} - H_2^{1/2}\right) \qquad ...(9.40)$$

If the tank is completely emptied then $H_2 = 0$ and Eq. 9.40 becomes

$$t = \frac{2A(H_1^{1/2})}{C_d a\sqrt{2g}} \qquad ...(9.41)$$

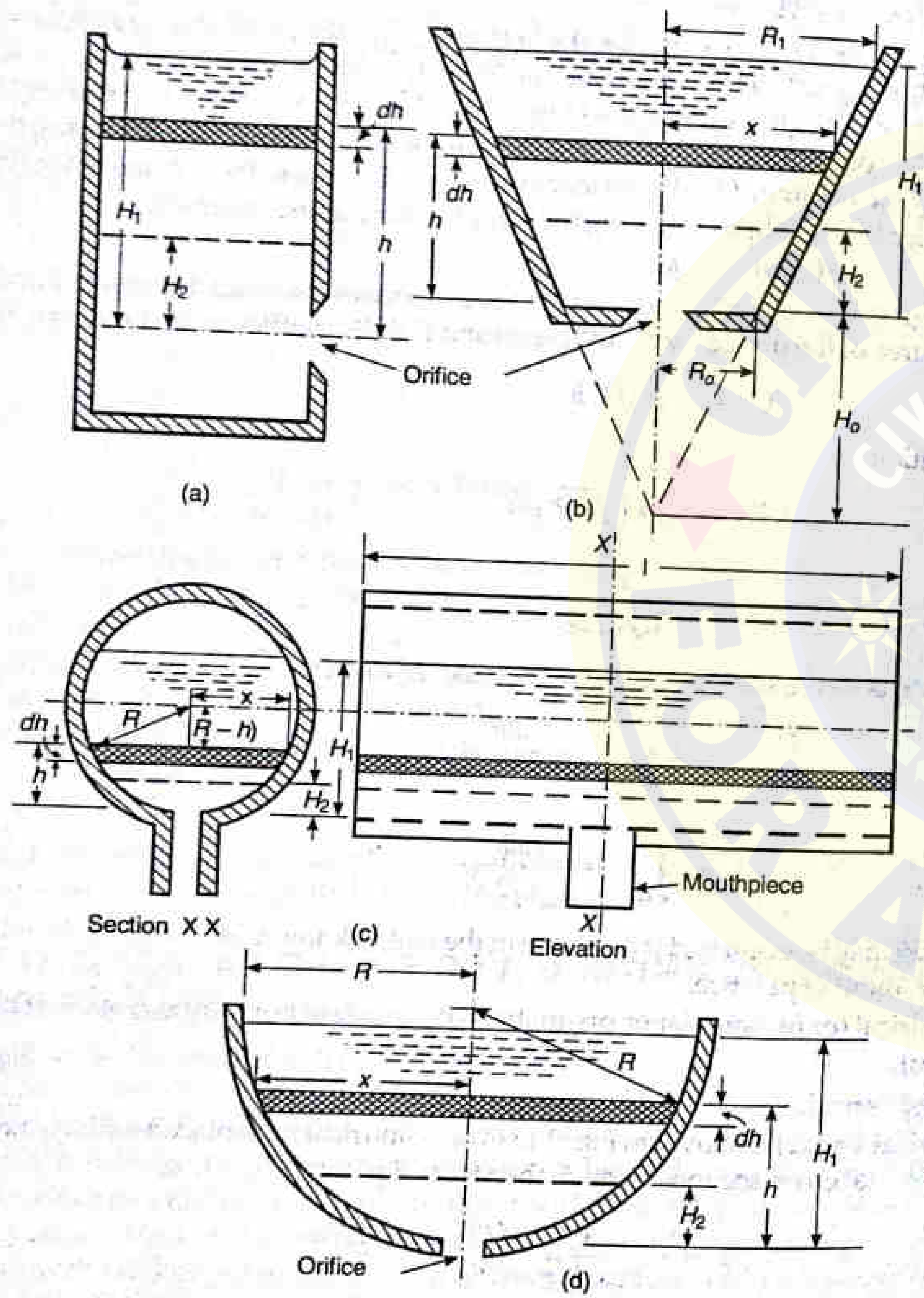


Figure 9.16 *Emptying of tanks (a) vertical cylindrical; (b) conical; (c) horizontal cylindrical; (d) hemispherical*

However, if a cylindrical tank is placed with its axis horizontal then the horizontal cross-sectional area A of the tank will also vary with the head. As shown in Fig. 9.16 (c), in this case

$$A = 2l\sqrt{2Rh - h^2}$$

where l is the length and R is the radius of the tank. Then from Eq. 9.39

$$t = -\frac{2l}{C_d a\sqrt{2g}}\int_{H_1}^{H_2}\frac{\sqrt{2Rh-h^2}}{\sqrt{h}}\,dh$$

or

$$t = \frac{4l}{3C_d a\sqrt{2g}}\left[(2R-H_2)^{3/2}-(2R-H_1)^{3/2}\right] \qquad ...(9.42)$$

(ii) Conical Tank. Generally a conical tank has a shape of a frustrum of a cone. In this case the horizontal cross-sectional area A varies. Thus as shown in Fig. 9.16 (b)

$$A = \pi x^2$$

where x is the radius of the cone at a height h above the bottom. From the similar triangles, we have

$$\frac{R_1}{(H_1+H_0)} = \frac{x}{(h+H_0)}$$

or

$$x = \frac{R_1(h+H_0)}{(H_1+H_0)}$$

Then from Eq. 9.39

$$t = -\frac{\pi R_1^2}{C_d a\sqrt{2g}(H_1+H_0)^2}\int_{H_1}^{H_2}\frac{(H_0+h)^2}{\sqrt{h}}\,dh$$

or

$$t = -\frac{\pi R_1^2}{C_d a\sqrt{2g}(H_1+H_0)^2} \times \left[\frac{2}{5}h^{5/2}+2H_0^2h^{1/2}+\frac{4}{3}H_0h^{3/2}\right]_{H_1}^{H_2} \qquad ...(9.43)$$

In the above expression the value of H_0 may be obtained if the radius R_0 at the bottom of the vessel is known. Thus again by similar triangles

$$\frac{R_1}{H_1+H_0} = \frac{R_0}{H_0}$$

or

$$H_0 = \frac{R_0 H_1}{(R_1 - R_0)}$$

(iii) Hemispherical Tank. In this case too the horizontal cross-sectional area is varying as shown in Fig. 9.16 (d)

$$A = \pi x^2$$

and

$$x = \sqrt{2Rh - h^2}$$

where R is the radius of tank. Then from Eq. 9.39

$$t = -\frac{\pi}{C_d a\sqrt{2g}} \int_{H_1}^{H_2} \frac{(2Rh - h^2)}{\sqrt{h}}\, dh$$

or

$$t = \frac{2\pi}{C_d a\sqrt{2g}} \left[\frac{2}{3}R\left(H_1^{3/2} - H_2^{3/2}\right) - \frac{1}{5}\left(H_1^{5/2} - H_2^{5/2}\right)\right] \quad ...(9.44)$$

Now if the tank (or vessel) was full at the beginning and it is completely emptied, then

$$H_1 = R$$

and

$$H_2 = 0$$

Equation 9.44 then becomes

$$t = \frac{14\pi R^{5/2}}{15 C_d a\sqrt{2g}} \quad ...(9.45)$$

(b) *Time of emptying (or filling) a tank with inflow.* Consider a tank of constant horizontal cross-sectional area A, which is provided with an orifice (or mouthpiece) of cross-sectional area a. Let there be a constant inflow of liquid of Q m³ per second and at the same time the liquid is discharging through the orifice (or mouthpiece). It is required to find the time t in which the height of the liquid surface changes from H_1 to H_2 above the centre of the orifice (or mouthpiece). As shown in Fig. 9.17 let at any instant liquid surface be at a height h above the centre of the orifice (or mouthpiece) and in time dt the level is increased by dh. Then volume of liquid added to the tank is Adh.

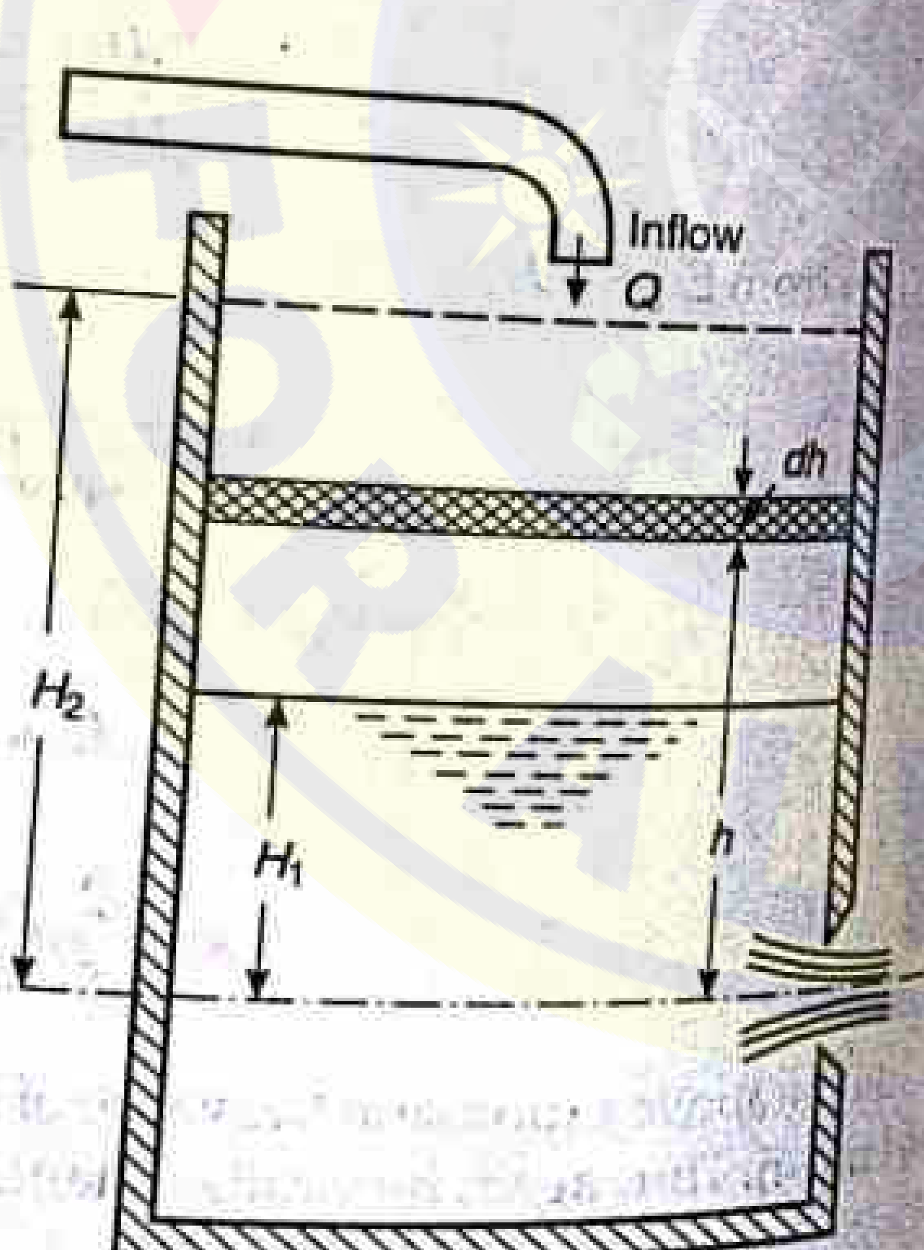


Figure 9.17 *Flow from a vessel with inflow*

Further in time dt the volume of the inflow of the liquid into the tank is Qdt, and during the same time the volume of the liquid discharged through the orifice (or mouthpiece) is qdt where q represents the discharge through the orifice (or mouthpiece) at that instant.

Since

$$q = C_d a\sqrt{2gh}$$

$$= K\sqrt{h}$$

where $K = C_d a\sqrt{2g}$

$$qdt = K\sqrt{h}\, dt$$

Thus net volume of liquid added to the tank during time dt is

$$(Qdt - qdt) = \left(Q - K\sqrt{h}\right) dt$$

Thus equating the two, we have

$$Adh = \left(Q - K\sqrt{h}\right) dt$$

or

$$dt = \frac{Adh}{Q - K\sqrt{h}}$$

By integrating this equation the time required to raise the liquid surface from the height H_1 to H_2 may be obtained. Thus

$$\int_0^t dt = \int_{H_1}^{H_2} \frac{Adh}{Q - K\sqrt{h}}$$

or

$$t = \int_{H_1}^{H_2} \frac{Adh}{Q - K\sqrt{h}} \quad ...(i)$$

Let

$$Q - K\sqrt{h} = z$$

or

$$h = \frac{(Q-z)^2}{K^2}$$

Differentiating with respect to z

$$dh = -\frac{2(Q-z)}{K^2}\, dz$$

Substituting this value of dh and h in Eq. (i), we have

$$t = -\frac{2A}{K^2} \int \left(\frac{Q-z}{z}\right) dz$$

or

$$t = -\frac{2A}{K^2} [Q \log_e z - z]$$

or

$$t = -\frac{2A}{K^2} \left[Q\log_e\left(Q - K\sqrt{h}\right) - \left(Q - K\sqrt{h}\right)\right]_{H_1}^{H_2}$$

or

$$t = -\frac{2A}{K^2} \left[Q\log_e\left(\frac{Q - K\sqrt{H_2}}{Q - K\sqrt{H_1}}\right) + K\left(\sqrt{H_2} - \sqrt{H_1}\right)\right] \quad ...(9.46)$$

Equation 9.46 can also be used to compute the time required to lower the liquid surface from the initial height H_1 to another height H_2, in which case Eq. 9.46 gives negative result.

9.13 FLOW OF LIQUID FROM ONE VESSEL TO ANOTHER

As shown in Fig. 9.18 consider a liquid flowing from one vessel to another, so that as the liquid surface falls in one vessel, it will rise by a corresponding amount in the other. In this case, the orifice will be drowned and therefore the head causing the flow at any instant will be the difference between the liquid surfaces in the two tanks at that instant. The problem in this case is to determine the time required to reduce the difference between the liquid surfaces in the two tanks from H_1 to H_2.

Let the liquid flow from a vessel of cross-sectional area A_1 to a vessel of cross-sectional area A_2 and let a be the cross-sectional area of the orifice. Let the difference between the liquid surfaces in the two vessels be H_1 at the beginning and it is required to determine the time t, taken for the difference to be reduced to H_2.

At any instant let the difference between the liquid surfaces in the two vessels be h and let a small quantity Q flow through the orifice in time dt. This will cause the liquid surface to fall by dH in the upstream tank and the corresponding rise in the liquid surface in the downstream tanks will be $\frac{A_1}{A_2}dH$.

$\therefore$ Difference of liquid levels after time dt

$$= h - \left(dH + dH\frac{A_1}{A_2}\right) = h - dH\left(1+\frac{A_1}{A_2}\right)$$

Hence the change of liquid levels in time dt

$$= dH\left(1+\frac{A_1}{A_2}\right)$$

It is this change of the liquid levels in time dt that equals the head which causes flow. Let this head be dh, then

$$dh = \left(1+\frac{A_1}{A_2}\right); \text{ or } dH = \frac{dh}{\left(1+\frac{A_1}{A_2}\right)}$$

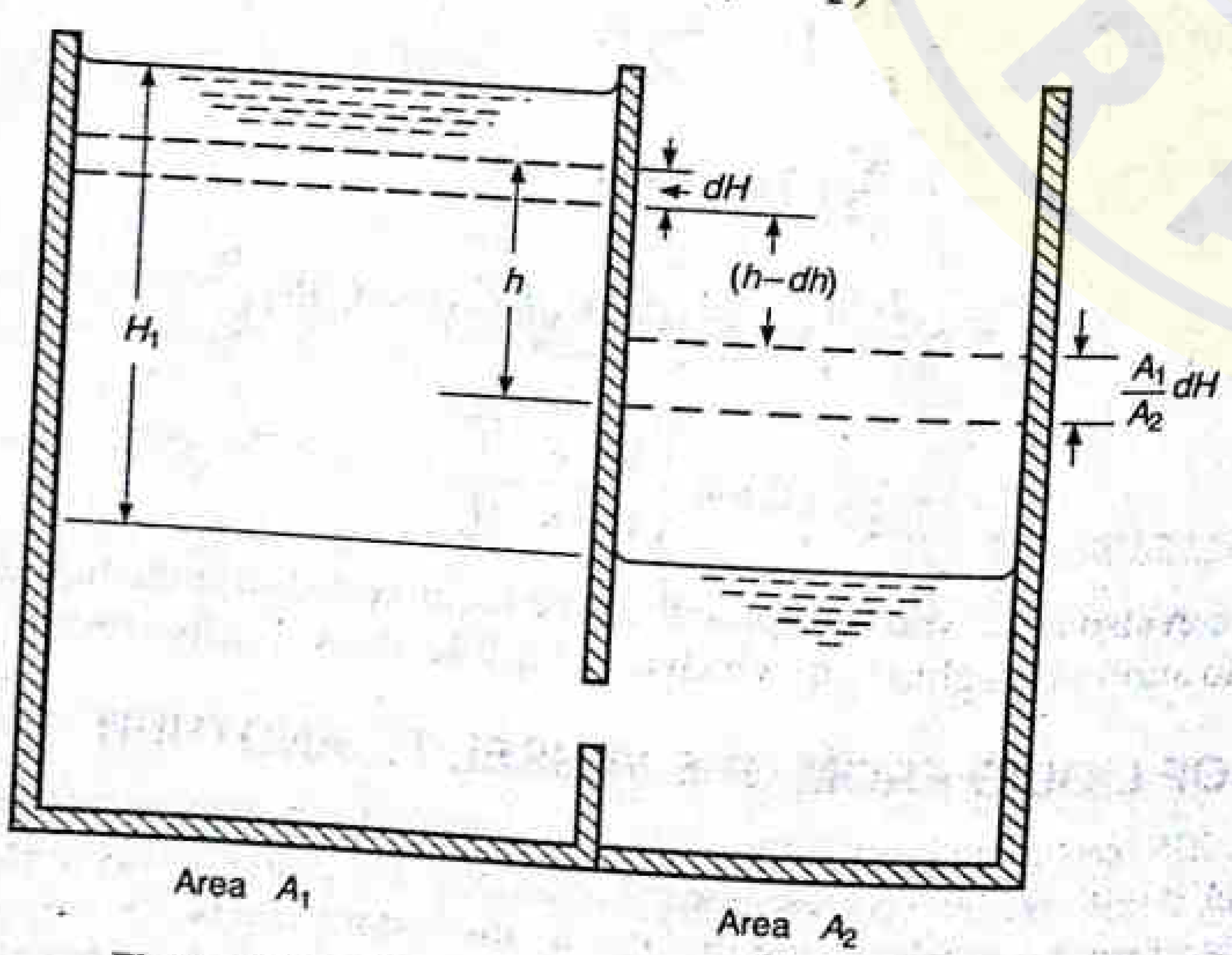


Figure 9.18 *Flow of liquid from one vessel to another*

Again if Q is the discharge through the orifice at this instant then the total volume of liquid passing through the orifice in time dt is Qdt.

But
$$Q = C_d a\sqrt{2gh}$$

$\therefore$
$$Qdt = C_d a\sqrt{2gh}\ dt$$

Further the quantity of liquid flowing through the orifice in time dt is also equal to the volume of liquid leaving the upstream tank which is equal to $A_1 dH$. Thus equating the two, we have

$$-A_1 dH = C_d a\sqrt{2gh}\ dt$$

The negative sign has been introduced because as the time dt increases the liquid surface in the upstream tank lowers down.

$\therefore$
$$dt = -\frac{A_1 dH}{C_d a\sqrt{2gh}}$$

Substituting the value of dH as derived above

$$dt = -\frac{A_1 dh}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2gh}}$$

By integrating both the sides of the above expression, we get

$$\int_0^t dt = -\int_{H_1}^{H_2}\frac{A_1 dh}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2gh}}$$

or
$$t = \frac{2A_1(H_1^{1/2}H_1^{1/2})}{C_d a\left(1+\frac{A_1}{A_2}\right)\sqrt{2g}} \quad ...(9.47)$$

If both the vessels have the same horizontal cross-sectional area i.e., $A_1 = A_2$, then

$$t = \frac{A_1(H_1^{1/2}-H_1^{1/2})}{C_d a\sqrt{2g}} \quad ...(9.48)$$

It may however be noted that the time taken to reduce the difference of liquid level between two vessels of different cross-sectional areas is the same whether the liquid flows from a larger to smaller vessel or from a smaller to larger vessel, provided the reduction in liquid level is the same in each case.

9.14 TIME OF EMPTYING AND FILLING OF A CANAL LOCK

As discussed in Chapter 3 a canal lock is a rectangular chamber constructed at the junction of two canal reaches which are at different levels. It is constructed to facilitate the transfer of boats from a higher to a lower level and vice versa. The difference of level of the water surfaces in the two canal reaches is termed as the *lift* of the lock. The lock chamber is closed at each end by a pair of gates called lock gates. When the lock is full it can be emptied by means of orifices (or sluices) placed in the lower gates below the water surface in the lower reach. When the lock is empty it can be filled by means of orifices (or sluices) placed in the upper gates. The orifices may be closed or opened as per requirements by means of sliding shutters. In designing locks, it is necessary to estimate the time required to empty or fill the lock which may be determined by using some of the equations derived earlier as indicated below.

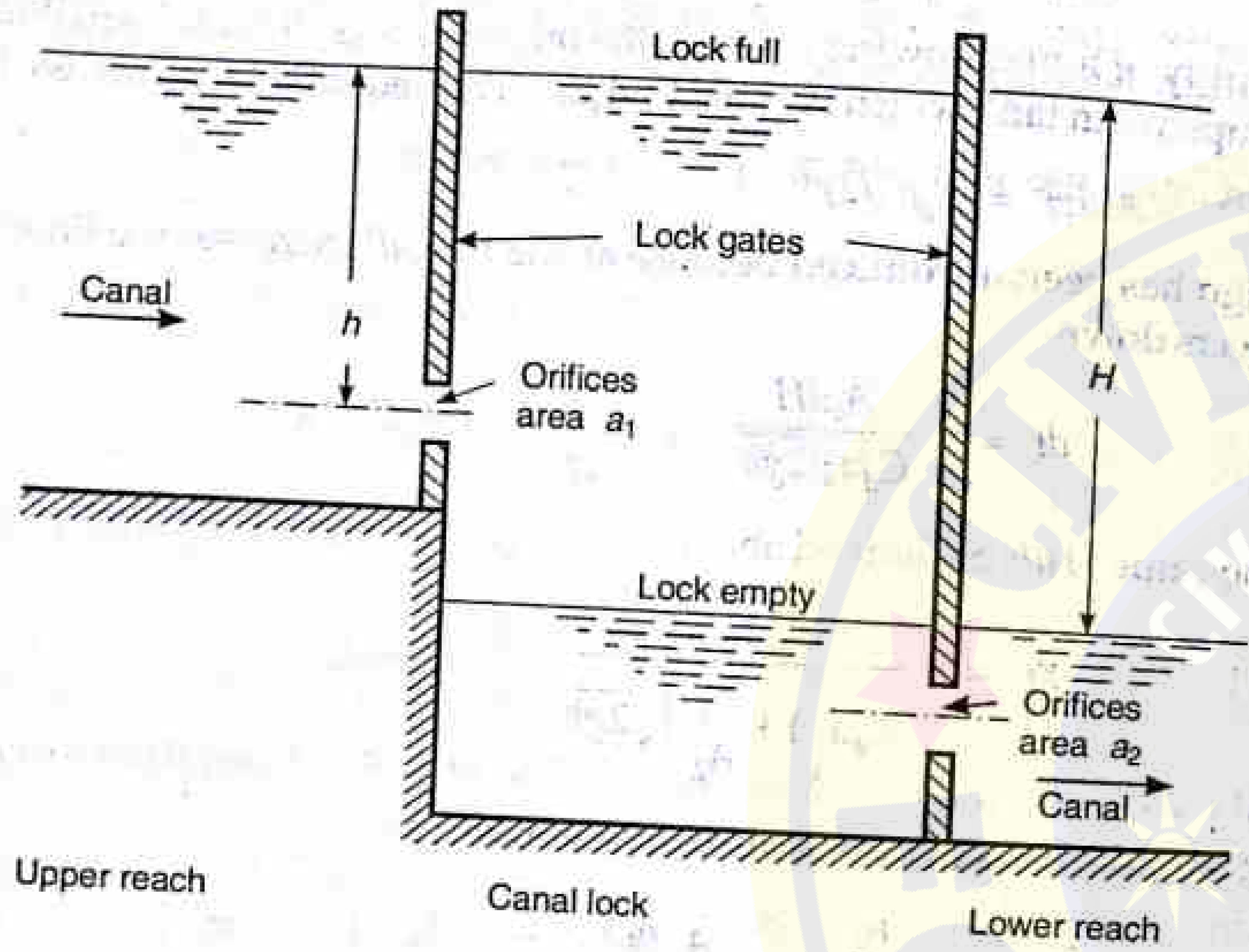


Figure 9.19 *Canal lock*

Let A be cross-sectional area of the water surface in the lock; H be the lift; and h be the depth from the water surface of the upper reach to the centre of the upper orifices. Further let a_1 and a_2 be the total areas of the upper and the lower orifices respectively.

(*i*) *To empty the lock*. The orifices being submerged, the head varies from H to 0. The time required to empty the lock may be obtained by Eq. 9.40 as

$$t = \frac{2A\sqrt{H}}{C_d a_2 \sqrt{2g}} = \frac{2AH}{C_d a_2 \sqrt{2gH}} \quad ...(9.49)$$

(*ii*) *To fill the lock*. From the level of the lower reach to the centre of the upper orifices the head is constant, equal to h. Thus the time taken to fill the lock upto the centre of the upper orifices is

$$t_1 = \frac{A(H-h)}{C_d a_1 \sqrt{2gh}}$$

From the centre of the upper orifices to the level of the upper reach, the head varies from h to 0. Thus the time for filling this portion is obtained from Eq. 9.40 as

$$t_2 = \frac{2A\sqrt{h}}{C_d a_1 \sqrt{2g}} = \frac{2Ah}{C_d a_1 \sqrt{2gh}}$$

Hence the total time t taken to fill the lock is

$$t = t_1 + t_2 = \frac{A(H+h)}{C_d a_1 \sqrt{2gh}} \quad ...(9.50)$$

ILLUSTRATIVE EXAMPLES

Example 9.1. *A jet of water issues from a sharp edged vertical orifice under a constant head of 0.51 m. At a certain point of issuing jet, the horizontal and vertical coordinates measured from the vena-contracta are 0.406 m and 0.085 m respectively. Determine C_v. If $C_d = 0.62$, find C_c.*

Solution

From Eq. 9.10, we have

$$C_v = \sqrt{\frac{x^2}{4hy}}$$

$$x = 0.406 \text{ m}; y = 0.085 \text{ m}; \text{and } h = 0.51 \text{ m}$$

$$\therefore \quad C_v = \sqrt{\frac{(0.406)^2}{4\times 0.51\times 0.085}} = 0.975$$

$$C_d = C_c \times C_v$$

$$\therefore \quad C_c = \frac{C_d}{C_v} = \frac{0.62}{0.975} = 0.636$$

Example 9.2. *Water discharges at the rate of 98 litres per second through a 0.12 m diameter vertical sharp edged orifice placed under a constant head of 18 m. A point on the jet measured from the vena-contracta of the jet has coordinates 4.5 m horizontal and 0.54 m vertical. Find (a) the coefficients C_v, C_c, C_d and C_r for the orifice and (b) the power lost at the orifice.*

Solution

(a)
$$C_v = \sqrt{\frac{x^2}{4hy}}$$

$$x = 4.5; h = 10 \text{ m}; \text{and } y = 0.54 \text{ m}$$

$$\therefore \quad C_v = \sqrt{\frac{(4.5)^2}{4\times 10\times 0.54}} = 0.968$$

$$Q = 98\ l/s = 0.098 \text{ m}^3/\text{s}$$

Theoretical discharge

$$Q_{th} = a\sqrt{2gh}$$

$$= \frac{\pi}{4}(0.12)^2 \times \sqrt{2\times 9.81\times 10}$$

$$= 0.158 \text{ m}^3/\text{s}$$

$$\therefore \quad C_d = \frac{0.098}{0.158} = 0.62$$

But

$$C_d = C_c \times C_v$$

$$\therefore \quad C_c = \frac{C_d}{C_v} = \frac{0.62}{0.968} = 0.64$$

$$C_r = \left(\frac{1}{C_v^2} - 1\right) = \left[\frac{1}{(0.968)^2} - 1\right] = 0.067$$

(b) *SI Units*

$$\text{Loss of power} = wQH\,(1 - C_v^2)$$
$$= 9810 \times 0.098 \times 10\,[1 - (0.968)^2]$$
$$= 0.605 \text{ kW}$$

Metric Units

$$\text{Loss of power} = \frac{wQH(1 - C_V^2)}{75}$$

$$= \frac{1000 \times 0.098 \times 10[1 - (0.968)^2]}{75}$$

$$= 0.823 \text{ h.p.}$$

Example 9.3. *A closed tank partially filled with water discharges through an orifice of 12.5 mm diameter d has a coefficient of discharge of 0.65. If air is pumped into the upper part of the tank, determine the pressure quired to produce a discharge or 36.6 litres/minute when the water surface is 1m above the outlet.*

Solution

$$Q = C_d\, a\sqrt{2gh}$$

$$Q = \frac{36.6 \times 10^{-3}}{60} = 6.1 \times 10^{-4} \text{ m}^3/\text{s}$$

$$a = \frac{\pi}{4} \times (12.5 \times 10^{-3}) = 1.23 \times 10^{-4} \text{ m}^2$$

Thus by substitution, we get

$$6.1 \times 10^{-4} = 0.65 \times 1.23 \times 10^{-4} \times \sqrt{2 \times 9.81 \times h}$$
$$h = 2.97 \text{ m}$$

Pressure head of the air to be pumped

$$= (2.97 - 1) = 1.97 \text{ m of water}$$

I Units

ressure intensity of air

$$= (9\,810 \times 1.97)$$
$$= 19\,326 \text{ N/m}^2 = 19.326 \text{ kN/m}^2$$

Metric Units

Pressure intensity of air

$$= (1000 \times 1.97)$$
$$= 1970 \text{ kg(f)/m}^2 = 0.197 \text{ kg(f)/cm}^2$$

Example 9.4. *A large tank having a circular orifice 6.45×10^{-4} m² in area in its vertical side rests on a smooth horizontal surface. When the depth of water in the tank is 1.22 m the discharge through the orifice is 1118.34 N per minute [114 kg(f) per minute] and a horizontal force of 9.123 N in line with the centre of the orifice is required to keep the tank at rest. From these data determine the coefficients C_v, C_c and C_d.*

Solution

Actual discharge $$Q = \frac{1118.34}{9810 \times 60} = 1.9 \times 10^{-3} \text{ m}^3/\text{s}$$

Theoretical discharge

$$Q_{th} = a\sqrt{2gh}$$
$$= (6.45 \times 10^{-4}) \times \sqrt{2 \times 9.81 \times 1.22}$$
$$= 3.16\ \ 10^{-3} \text{ m}^3/\text{s}$$

$$\therefore \quad C_d = \frac{1.9 \times 10^{-3}}{3.16 \times 10^{-3}} = 0.601$$

According to impulse-momentum principle

Force $$F = \frac{WV}{g}$$

or $$9.123 = \frac{1118.34}{60} \times \frac{V}{9.81}$$

$$\therefore \quad V = \frac{9.123 \times 60 \times 9.81}{1\,118.34} = 4.80 \text{ m/s}$$

Theoretical velocity

$$V_{th} = \sqrt{2gh}$$
$$= \sqrt{2 \times 9.81 \times 1.22} = 4.89 \text{ m/s}$$

$$\therefore \quad C_v = \frac{4.80}{4.89} = 0.982$$

Since $$C_d = C_c \times C_v$$

$$\therefore \quad C_c = \frac{C_d}{C_v} = \frac{0.601}{0.982} = 0.612$$

Example 9.5. *A circular sharp-edged orifice of 25 mm diameter is situated in a vertical side of a large oil tank and discharges to a region where the pressure is 82.404 kN/m² [0.84 kg(f)/cm²] absolute. The oil in the tank is maintained at a constant level and subject to an external pressure of 48.069 kg(f)/m² [0.49 kg(f)/cm²] gage.*

The values of C_c and C_v for the orifice are 0.62 and 0.98 respectively, the specific gravity of the oil is 0.85 and the barometric pressure head is 760 mm of mercury absolutely.

If the jet drops 0.05 m in a horizontal distance of 1.375 m, calculate (a) the height of the oil surface in the tank above the axis of the orifice, (b) the volume rate of flow through the orifice.

Solution

(a) From Eq. 9.9, we have

$$V = \sqrt{\frac{gx^2}{2y}}$$

or
$$\frac{V^2}{2g} = \frac{x^2}{4y}$$

$$x = 1.375 \text{ m; and } y = 0.05 \text{ m}$$

$\therefore$
$$\frac{V^2}{2g} = \frac{(1.375)^2}{4\times 0.05}$$

$$= 9.453 \text{ m of oil}$$

But
$$V = C_v\sqrt{2gh}$$

or
$$h = \frac{1}{C_v^2}\times\frac{V^2}{2g}$$

$\therefore$
$$h = \frac{9.453}{(0.98)^2} = 9.843 \text{ m of oil}$$

Now if h' is the height of the oil surface in the tank above the axis of the orifice, then

In SI Units

$$h = 9.843 = \frac{760\times 13.6}{1000\times 0.85} + \frac{48.069\times 10^3}{9810\times 0.85} + h' - \frac{82.404\times 10^3}{9810\times 0.85}$$

$\therefore$
$$h' = 1.80 \text{ m}$$

In Metric Units

$$h = 9.843 = \frac{760\times 13.6}{1000\times 0.85} + \frac{0.49\times 10^4}{1000\times 0.85} + h' - \frac{0.84\times 10^4}{1000\times 0.85}$$

$\therefore$
$$h' = 1.80 \text{ m}$$

(b)
$$Q = C_c C_v a\sqrt{2gh}$$

$$= (0.62\times 0.98)\ \frac{\pi}{4}(25\times 10^{-3})^2\times\sqrt{2\times 9.81\times 9.843}$$

$$= 0.004\ 14 \text{ m}^3/\text{s} = 4.14\ l/\text{s}.$$

Example 9.6. *A circular orifice of area 6.45×10^{-4} m^2 is provided in the vertical side of a large tank. The tank is suspended from a knife edge 1.53 m above the level of the orifice. When the head of water is 1.22 m the discharge is 1161.5 N/min [118.4 kg(f)/min] and a turning moment of 14.421 N-m [1.47 kg(f)-m] has to be applied to the knife edges to keep the tank vertical. Determine C_v, C_d and C_c of the orifice.*

Solution

From Eq. 9.12, we have

$$C_v = \frac{Wl g}{W_t y\sqrt{2gH}}$$

$$(Wl) = 14.421 \text{ N-m}$$

$$W_t = \frac{1161.5}{60} = 19.358 \text{ N/s}$$

$$y = 1.53 \text{ m}; h = 1.22 \text{ m}$$

Thus by substitution, we get

$$C_v = \frac{14.421\times 9.81}{19.358\times 1.53\times\sqrt{2\times 9.81\times 1.22}}$$

$$= 0.976$$

Actual discharge
$$Q = \frac{1161.5}{9810\times 60} = 1.973\times 10^{-3} \text{ m}^3/\text{s}$$

Theoretical discharge

$$Q_{th} = a\sqrt{2gh}$$

$$= 6.45\times 10^{-4}\times\sqrt{2\times 9.81\times 1.22}$$

$$= 3.156\ 10^{-3} \text{ m}^3/\text{s}$$

$\therefore$
$$C_d = \frac{1.973\times 10^{-3}}{3.156\times 10^{-3}} = 0.625$$

Since
$$C_d = C_c\times C_v$$

$$C_c = \frac{C_d}{C_v} = \frac{0.625}{0.976} = 0.640$$

Example 9.7. *A reservoir discharges through a sluice 0.915 m wide by 1.22 m deep. The top of the opening is 0.61 m below the water level in the reservoir and the downstream water level is below the bottom of the opening. Calculate (a) the discharge through the opening if $C_d = 0.60$; and (b) percentage error if the opening is treated as a small orifice.*

Solution

(a) From Eq. 9.15, we have

$$Q = \frac{2}{3}C_d B\sqrt{2g}\left[H_2^{3/2} - H_1^{3/2}\right]$$

$$C_d = 0.60; B = 0.915 \text{ m};$$

$$H_2 = (1.22 + 0.61) = 1.83 \text{ m}$$
$$H_1 = 0.61 \text{ m}$$

Thus by substitution, we get

$$Q = \frac{2}{3} \times 0.60 \times 0.915 \times \sqrt{2\times 9.81}\left[(1.83)^{3/2} - (0.61)^{3/2}\right]$$
$$= 3.241 \text{ m}^3/\text{s}$$

(b) For a small orifice

$$Q = C_d\, a\sqrt{2gh}$$
$$a = (0.915 \times 1.22) = 1.116 \text{ m}^2$$
$$h = \left(0.61 + \frac{1.22}{2}\right) = 1.22 \text{ m}$$

Thus
$$Q = 0.60 \times 1.116 \times \sqrt{2\times 9.81\times 1.22}$$
$$= 3.276 \text{ m}^3/\text{s}$$

$\therefore$
$$\text{Error} = \frac{3.276 - 3.241}{3.241}$$
$$= 0.010\,8 \text{ or } 1.08\%$$

Example 9.8. *Water is supplied from a tank into a canal through a rectangular sluice 1 m wide and 1.75 m high. The water level in the tank is 2 m above the top edge of the opening and the canal water level is 0.3m below the top edge. If the coefficient of discharge is 0.62 for both the free and the submerged portions, calculate the discharge.*

Solution

In this case the orifice is partially submerged, thus the discharge through the free portion is

$$Q_1 = \frac{2}{3} C_d B\sqrt{2g}\left(H_2^{3/2} - H_1^{3/2}\right)$$
$$C_d = 0.62;\ B = 1 \text{ m}$$
$$H_2 = (2 + 0.3) = 2.3 \text{ m}$$

and
$$H_1 = 2 \text{ m}$$

Thus by substitution, we get

$$Q_1 = \frac{2}{3} \times 0.62 \times 1 \times \sqrt{2\times 9.81}\ [(2.3)^{3/2} - (2)^{3/2}]$$
$$= 1.208 \text{ m}^3/\text{s}$$

Discharge through the submerged portion is

$$Q_2 = C_d\, a\sqrt{2gh}$$
$$C_d = 0.62$$
$$a = 1 \times (1.75 \times 0.3) = 1.45 \text{ m}^2$$
$$H = (2 + 0.3) = 2.3 \text{ m}$$

Thus by substitution, we get

$$Q^2 = 0.62 \times 1.45 \times \sqrt{2\times 9.81\times 2.3}$$
$$= 6.039 \text{ m}^3/\text{s}$$

$\therefore$ Total discharge through the orifice

$$Q = (Q_1 + Q_2) = (1.208 + 6.039) = 7.247 \text{ m}^3/\text{s}$$

Example 9.9. *Water flows vertically downwards through a 0.15 m diameter pipe at the rate of 70 litres per second. The pipe suddenly enlarges to 0.3 m diameter. A is a point 0.5 m distant from and above the section of enlargement, and B is another point 0.5 m below the same. It was found that pressure gage connected to A gives a reading of 210.915 kN/m². Determine the pressure at point B. Neglect the friction loss between A and B.*

Recalculate the pressure at B, if the same discharge flows up the pipe, considering the pressure at A as unchanged. Assume that $C_c = 0.64$.

Solution

Area of pipe at
$$A = \frac{\pi}{4}(0.15)^2 = 0.0177 \text{ m}^2$$

Area of pipe at
$$B = \frac{\pi}{4}(0.30)^2 = 0.0708 \text{ m}^2$$
$$Q = 70 \text{ l/s} = 0.07 \text{ m}^3/\text{s}$$
$$V_A = \frac{0.07}{0.0177} = 3.96 \text{ m/s}$$
$$V_B = \frac{0.07}{0.0708} = 0.99 \text{ m m/s}$$

When the flow is vertically downwards, there is sudden enlargement and hence there is a loss of head due to sudden enlargement which equals

$$h_L = \frac{(V_A - V_B)^2}{2g}$$
$$= \frac{(3.96 - 0.99)^2}{2\times 9.81} = 0.45 \text{ m}$$

Applying Bernoulli's equation between the points A and B, we have

$$\frac{p_A}{w} + \frac{V_A^2}{2g} + z_A = \frac{p_B}{w} + \frac{V_B^2}{2g} + z_B + h_L$$
$$p_A = 210.915 \text{ kN/m}^2 = 210.915 \times 10^3 \text{ N/m}^3$$
$$z_A = (0.50 + 0.50) = 1.0 \text{ m}$$
$$z_B = 0, \text{ assuming datum level at } B.$$

Thus by substitution, we have

$$\frac{210.915\times 10^3}{9810} + \frac{(3.96)^2}{2\times 9.81} + 1.0 = \frac{p_B}{w} + \frac{(0.99)^2}{2\times 9.81} + 0 + 0.45$$

$\therefore$
$$p_B = 223.661 \text{ kN/m}^2$$

When the flow is vertically upwards there is a sudden contraction and hence the loss of head is h_L' due to sudden contraction which equals

$$h_L' = \frac{V_A^2}{2g}\left(\frac{1}{C_c}-1\right)^2$$

$$= \frac{(3.96)^2}{2\times 9.81}\left(\frac{1}{0.64}-1\right)^2 = 0.253 \text{ m}$$

Again applying Bernoulli's equation between the points B and A, we have

$$\frac{p_B}{w}+\frac{V_B^2}{2g}+Z_B = \frac{p_A}{w}+\frac{V_A^2}{2g}+Z_A+h_L'$$

or
$$\frac{p_B}{w}+\frac{(0.99)^2}{2\times 9.81}+0 = \frac{210.915\times 10^3}{9810}+\frac{(3.96)^2}{2\times 9.81}+1.0+0.253$$

$\therefore$
$$p_B = 230.558 \text{ kN/m}^2$$

Example 9.10. *(a) For sudden expansion in pipe flow what is the optimum ratio between the diameter of the pipe before expansion and the diameter of the pipe after expansion so that the pressure rise may be maximum. What will be the corresponding pressure rise?*

(b) Oil of specific gravity 0.8 flows in a 0.08 m diameter pipeline of such diameter that the maximum pressure rise is obtained. If the rate of flow through the pipeline is 12.5 liters per second find (i) the loss of energy in the sudden expansion in m of oil, and (ii) the differential gage length h indicated on an oil mercury manometer connected between the two pipes.

Solution

(a) As indicated in Section 9.8, by applying the momentum equation the pressure rise between two sections on either side of the sudden expansion may be expressed as

$$\frac{p_2-p_1}{w} = \frac{V_2(V_1-V_2)}{g}$$

If Q is the discharge and d_1 and d_2 are the diameters of the pipe before and after the enlargement respectively, then

$$V_1 = \frac{Q}{(\pi/4)d_1^2}\ ;\text{and}\ V_2 = \frac{Q}{(\pi/4)d_2^2}$$

and by substituting these values of V_1 and V_2, we get

$$\frac{p_2-p_1}{w} = \frac{Q^2}{g(\pi/4)^2 d_1^4}\left[\left(\frac{d_1}{d_2}\right)^2-\left(\frac{d_1}{d_2}\right)^4\right]$$

Let $\left(\frac{p_2-p_1}{w}\right)$ be equal to H and $\frac{d_1}{d_2}$ be equal to x, then

$$H = \frac{Q^2}{g(\pi/4)^2 d_1^4}(x^2-x^4)$$

For a given discharge Q flowing through a pipe of given diameter d_1 the condition for maximum pressure rise at a sudden expansion may be obtained by equating (dH/dx) to zero. Thus

$$\frac{dH}{dx} = \frac{Q^2}{g(\pi/4)^2 d_1^4}(2x-4x^3) = 0$$

from which the optimum value of the ratio (d_1/d_2) is obtained as

$$x = \frac{d_1}{d_2} = \frac{1}{\sqrt{2}} = 0.707$$

The corresponding pressure rise

$$H_{max} = \frac{Q^2}{4g(\pi/4)^2 d_1^4} = \frac{V_1^2}{4g}$$

(b) (i) From Eq. 9.25 the head loss due to sudden expansion is given by

$$h_L = \frac{(V_1-V_2)^2}{2g}$$

$$d_1 = 0.08 \text{ m; and } d_2 = \left(0.08\times\sqrt{2}\right) = 0.113 \text{ m}$$

$$V_1 = \frac{12.5\times 10^{-3}}{(\pi/4)\times(0.08)^2} = 2.487 \text{ m/s}$$

$$V_2 = \frac{12.5\times 10^{-3}}{(\pi\times 4)\times(0.113)^2} = 1.246 \text{ m/s}$$

$\therefore$
$$h_L = \frac{(2.487-1.246)^2}{2\times 9.81} = 0.0785 \text{ m of oil}$$

(ii) By substituting the given values in the above noted expression for the pressure rise at the sudden expansion and considering the optimum value of (d_1/d_2) as $(1/\sqrt{2})$, we get

$$\frac{p_2-p_1}{w} = \frac{(12.5\times 10^{-3})^2}{9.81\times(\pi/4)^2(0.08)^4}\left(\frac{1}{2}-\frac{1}{4}\right)$$

$$= 0.1576 \text{ m of oil}$$

When an oil mercury manometer is connected between the two pipes, then

$$\frac{p_2-p_1}{w} = h\left(\frac{S_m}{S_{oil}}-1\right)$$

or
$$0.1576 = h\left(\frac{13.6}{0.8} - 1\right)$$

∴
$$h = 9.85 \times 10^{-3}\ \text{m} = 9.85\ \text{mm}$$

Example 9.11. *At a sudden enlargement of a pipeline from a diameter of 0.3 m to 0.6 m, the hydraulic gradient line rises 0.125 m. Estimate the discharge if the pipe is horizontal.*

Solution

Let V_1 and V_2 be the velocities of flow in 0.3 m and 0.6 m pipes respectively. Then by continuity equation

$$\frac{\pi}{4}(0.3)^2 V_1 = \frac{\pi}{4}(0.6)^2 \times V_2$$

or
$$V_1 = 4V_2$$

Applying Bernoulli's equation between the points 1 and 2 lying on either side of the enlargement, we have

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 + h_L$$

From the given data

$$z_1 = z_2\text{; and } \left(\frac{p_2}{w} - \frac{p_1}{w}\right) = 0.125\ \text{m}$$

and
$$h_L = \frac{(V_1 - V_2)^2}{2g} = \frac{(4V_2 - V_2)^2}{2g} = \frac{9V_2^2}{2g}$$

Thus by substitution, we have

$$\frac{(4V_2)^2}{2g} = 0.125 + \frac{V_2^2}{2g} + \frac{9V_2^2}{2g}$$

or
$$V_2 = 0.639\ \text{m/s}$$

∴
$$Q = a_2 V_2 = \frac{\pi}{4}(0.6)^2 \times 0.639$$

$$= \frac{\pi}{4}(0.6)^2 \times 0.639$$

$$= 0.181\ \text{m}^3/\text{s} = 181\ l/\text{s}$$

Example 9.12. *Water under a constant head of 4.5 m discharges through an external cylindrical mouthpiece 50 mm diameter and 150 mm long. If C_c for the orifice is 0.60, find (a) the discharge in litres per second; (b) the coefficient of discharge; and (c) the absolute pressure at vena-contracta. Assume atmospheric pressure to be 10.3 m of water.*

Solution

Refer Fig. 9.13. Applying Bernoulli's equation between the free surface and the outlet section of the mouthpiece, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

The loss of head is due to sudden enlargement between the vena-contracta and the end sections and hence

$$h_L = \frac{(V_c - V)^2}{2g}$$

By continuity
$$a_c V_c = aV\text{; or } V_c = \frac{V}{C_c}$$

∴
$$h_L = \frac{V^2}{2g}\left(\frac{1}{C_c} - 1\right)^2$$

$$= \frac{V^2}{2g}\left(\frac{1}{0.6} - 1\right)^2 = 0.444\frac{V^2}{2g}$$

Thus by substitution, we get

$$4.5 = \frac{V^2}{2g} + 0.444\frac{V^2}{2g}$$

∴
$$V = 7.819\ \text{m/s}$$
$$Q = aV$$
$$= \left[\frac{\pi}{4} \times (0.05)^2 \times 7.819\right]$$
$$= 0.0154\ \text{m}^3/\text{s} = 15.4\ l/\text{s}$$

$$C_v = \frac{V}{\sqrt{2gH}}$$

$$= \frac{7.819}{\sqrt{2 \times 9.81 \times 4.5}} = 0.832$$

Since for the outlet end of the mouthpiece $C_v = 1$

∴
$$C_d = C_c \times C_v = (1 \times 0.832) = 0.832$$

For finding the absolute pressure at vena-contracta, apply Bernoulli's equation between free surface of liquid and the vena-contracta. Thus

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

$$V_c = \frac{V}{C_c} = \frac{7.819}{0.6} = 13.03\ \text{m/s}$$

By substitution, we get

$$10.3 + 4.5 = h_c + \frac{(13.03)^2}{2 \times 9.81}$$

$$\therefore \quad h_c = 6.147 \text{ m of water absolute.}$$

Example 9.13. *Water under a constant head of 3 m discharges through an external cylindrical mouthpiece 50 mm diameter, for which $C_v = 0.82$, determine (a) the discharge in cumec; (b) absolute pressure at vena-contracta; and the maximum head for the mouthpiece to flow full.*

Solution

For the mouthpiece at the outlet end $C_c = 1$.

Thus
$$C_d = C_c \times C_v = (1 \times 0.82) = 0.82$$

$$Q = C_d a \sqrt{2gh}$$

$$= 0.82 \times \frac{\pi}{4}(0.05)^2 \times \sqrt{2 \times 9.81 \times 3}$$

$$= 0.0124 \text{ m}^3/\text{s}$$

If the head loss in the mouthpiece is expressed as $k\dfrac{V^2}{2g}$, then applying Bernoulli's equation between the free surface and the outlet section of the mouthpiece, we have

$$H_a + 3 = H_a + \frac{V^2}{2g} + k\frac{V^2}{2g}$$

or
$$V = \sqrt{\frac{2g \times 3}{(1+k)}}$$

But
$$V = C_v\sqrt{2g \times 3} = 0.82\sqrt{2g \times 3}$$

$$\therefore \quad C_v = \sqrt{\frac{1}{1+k}} = 0.82$$

or
$$k = 0.487$$

Further the loss of head in the mouthpiece is due to sudden enlargement, which equals

$$h_L = \frac{(V_c - V)^2}{2g} = \frac{V^2}{2g}\left(\frac{1}{C_c} - 1\right)^2$$

From which
$$k = \left(\frac{1}{C_c} - 1\right)^2$$

or
$$0.487 = \left(\frac{1}{C_c} - 1\right)^2$$

$$C_c = 0.589$$

$\therefore$

By continuity
$$V_c = V\frac{a}{a_c} = \frac{V}{C_c}$$

$$\therefore \quad V_c = \frac{0.82\sqrt{2 \times 9.81 \times 3}}{0.589} = 10.68 \text{ m/s}$$

Again applying Bernoulli's equation between the free surface and the vena-contracta, we have

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

or
$$10.3 + 3 = H_c + \frac{(10.68)^2}{2 \times 9.81}$$

$$\therefore \quad H_c = 7.486 \text{ m of water absolute}$$

Let H be the maximum head upto which the mouthpiece flows full and let the corresponding minimum pressure at vena-contracta be 2.5 m of water absolute. Then

$$10.3 + H = 2.5 + \frac{V_c^2}{2g}$$

But
$$V_c = \frac{V}{C_c}$$

$$= \frac{C_v\sqrt{2gH}}{C_c} = \frac{0.82\sqrt{2gH}}{0.589}$$

or
$$10.3 + H = 2.5 + \left(\frac{0.82}{0.589}\right)^2 H$$

$$\therefore \quad H = \frac{7.8}{0.938} = 8.316 \text{ m}$$

However, if the minimum pressure at vena-contracta is taken as zero, then

$$H = \frac{10.3}{0.938} = 10.98 \text{ m}$$

Example 9.14. *A Borda's mouthpiece 40 mm diameter discharges under a constant head of 1.5 m. If the coefficient of velocity for the entrance section of the mouthpiece is 0.95, find (a) the coefficient of contraction and the discharge when the mouth piece is running free, and (b) the discharge when the mouth piece is running full, assuming that vena-contracta is formed in the mouthpiece and the coefficient of contraction being the same as in (a) and allowing for the loss of energy between the vena-contracta and the outlet.*

Solution

(a) For the Borda's mouthpiece running free as indicated in Section 9.11.

$$\frac{a_c}{a} = \frac{gH}{V_c^2}$$

But
$$V_c = C_v\sqrt{2gH}$$

$$\therefore \quad \frac{a_c}{a} = C_c = \frac{1}{2C_v^2} = \frac{1}{2\times(0.95)^2} = 0.554$$

Discharge
$$Q = a_c V_c = C_c a \times C_v\sqrt{2gH}$$

$$\therefore \quad Q = 0.554 \times \frac{\pi}{4}(0.04)^2 \times 0.95 \times \sqrt{2\times9.81\times1.5}$$

$$= 3.588\times10^{-3}\ \text{m}^3/\text{s} = 3.588\ l/\text{s}.$$

(b) For the Borda's mouthpiece running full, the loss of head between the vena-contracta and the outlet is

$$h_L = \frac{(V_c^2 - V)^2}{2g}$$

$$= \frac{V^2}{2g}\left(\frac{1}{C_c}-1\right)^2$$

$$= \frac{V^2}{2g}\left(\frac{1}{0.554}-1\right)^2 = 0.648\frac{V^2}{2g}$$

Applying Bernoulli's equation between the free surface and the outlet, we have

$$H_a + H = H_a + \frac{V^2}{2g} + h_L$$

or
$$H = \frac{V^2}{2g} + 0.648\frac{V^2}{2g}$$

or
$$1.5 = 1.648\frac{V^2}{2g}$$

$$\therefore \quad V = \sqrt{\frac{2\times9.81\times1.5}{1.648}} = 4.226\ \text{m/s}$$

$$\therefore \quad \text{Discharge } Q = aV$$

$$= \frac{\pi}{4}(0.04)^2 \times 4.226$$

$$= 5.311\times10^{-3}\ \text{m}^3/\text{s} = 5.311\ l/\text{s}$$

Example 9.15. *A convergent-divergent nozzle is fitted into the side of a tank containing water. Assuming that there are no losses in the convergent part of the nozzle, that the losses in the divergent part are equivalent to 0.18 times the velocity head at exit, and that the minimum absolute pressure at the throat is 2.5 m of water for a barometric pressure of 10.3 m of water, determine the throat and exit diameters of the nozzle to discharge 0.0042 m³/s for a head of 1.5 m above the centre-line of the nozzle.*

Solution

Refer Fig. 9.14 (c). Applying Bernoulli's equation between the free surface and the throat, we have

$$H_a + H = H_c + \frac{V_c^2}{2g}$$

$$H_a = 10.3;\ H = 1.5\ \text{m; and } H_c = 2.5\ \text{m}$$

Thus by substitution, we have

$$10.3 + 1.5 = 2.5 + \frac{V_c^2}{2g}$$

or
$$V_c = 13.508\ \text{m/s}$$

$$Q = a_c V_c$$

and
$$Q = 0.0042\ \text{m}^3/\text{s}$$

Thus
$$0.0042 = \frac{\pi}{4}(d_c)^2 \times 13.508$$

$$\therefore \quad d_c = \sqrt{\frac{4\times0.0042}{\pi\times13.508}}$$

$$= 0.0199\ \text{m or } 19.9\ \text{mm}$$

i.e., throat diameter $= 19.9$ mm

Applying Bernoulli's equation between the free surface and exit, allowing for the loss of energy in the divergent portion, we have

$$H_a + H = H_a + \frac{V^2}{2g} + 0.18\frac{V^2}{2g}$$

or
$$V = \sqrt{\frac{2\times9.81\times1.5}{1.18}} = 4.994\ \text{m/s}$$

Now if d is the diameter at the exit, then

$$Q = 0.0042 = \frac{\pi}{4}(d)^2 \times 4.994$$

$$\therefore \quad d = \sqrt{\frac{4\times0.0042}{\pi\times4.994}} = 0.0327\ \text{m} = 32.7\ \text{mm}$$

Example 9.16. *In a laboratory experiment, on finding the coefficient of discharge of an orifice 25 mm* *diameter, provided in a tank of constant cross-sectional area of 0.305 m by 0.305 m, it was found that it took $1\frac{1}{4}$ minutes for the head over the centre of the orifice to fall from 1.2 m to 0.3 m. Find the coefficient of discharge of the orifice.*

Solution

From Eq. 9.40, we have

$$t = \frac{2A}{C_d a\sqrt{2g}}\left(H_1^{1/2} - H_2^{1/2}\right)$$

$$t = 1\frac{1}{4}\ \text{min} = 75\ \text{s}$$

$$A = (0.305 \times 0.305)\ \text{m}^2;$$

$$a = \frac{\pi}{4}(0.025)^2$$

$$H_1 = 1.2\ \text{m; and } H_2 = 0.3\ \text{m}$$

Thus by substitution, we have

$$75 = \frac{2\times(0.305\times0.305)}{C_d\times\frac{\pi}{4}(0.025)^2\times\sqrt{2\times9.81}}[(1.2)^{1/2} - (0.3)^{1/2}]$$

$$\therefore \quad C_d = 0.625$$

Example 9.17. *A tank of constant cross-sectional area of 2.8 m² has two orifices each 9.3×10^{-4} m² in area in one of its vertical sides at heights of 6 m and 1.5 m respectively above the bottom of the tank. Calculate the time taken to lower the water level from 9 m to 3.6 m above the bottom of the tank. Assume $C_d = 0.625$.*

Solution

In the beginning both the orifices will be discharging till the water level falls to 6 m, then the top orifice will be ineffective and the discharge will take place only through the lower orifice. Thus the total time taken to lower the water level may be divided into two parts viz., time taken to lower the water level from 9 m to 6 m when both the orifices are discharging, and the time taken to lower the water level from 6 m to 3.6 m when only the lower orifice is discharging.

Let t_1 and t_2 be these times, then the total time t is equal to $(t_1 + t_2)$. Let at any instant water level be at a height h above the lower orifice and let the water level be lowered by dh in time dt, then

$$Q\,dt = -A\,dh$$

in which $Q = (Q_1 + Q_2)$, where Q_1 and Q_2 are the discharges through the upper and lower orifices respectively.

Then

$$Q_1 = C_d \times (9.3 \times 10^{-4})\sqrt{2g(h-4.5)}$$

and

$$Q_2 = C_d \times (9.3\ \ 10^{-4})\ \sqrt{2gh}$$

Since the vertical distance between the two orifices

$$= (6 - 1.5) = 4.5\ \text{m}$$

By substitution, we have

$$\left[C_d\times(9.3\times10^{-4})\times\sqrt{2g}\left(\sqrt{h-4.5}+\sqrt{h}\right)\right]dt = -A\,dh$$

$$C_d = 0.625;\ A = 2.8\ \text{m}^2$$

$$H_1 = (9-1.5) = 7.5\ \text{m}$$

and

$$H_2 = (6-1.5) = 4.5\ \text{m}$$

$$\therefore \quad dt = -\frac{2.8}{0.625\times(9.3\times10^{-4})\times\sqrt{2g}}\times\frac{dh}{(\sqrt{h-4.5}+\sqrt{h})}$$

By integrating both sides of the above equation, we get

$$\int_0^1 dt = -\frac{2.8}{0.625\times(9.3\times10^{-4})\times\sqrt{2g}}\times\int_{7.5}^{4.5}\frac{dh}{\left(\sqrt{h-4.5}+\sqrt{h}\right)}$$

or

$$t_1 = -1087.54\int_{7.5}^{4.5}\left[\frac{(\sqrt{h-4.5}-\sqrt{h})}{-4.5}\right]dh$$

$$= 241.68\left[\frac{2}{3}(h-4.5)^{3/2} - \frac{2}{3}h^{3/2}\right]_{7.5}^{4.5}$$

$$= \frac{241.68\times2}{3}\left[0-(4.5)^{3/2}-(3)^{3/2}+(7.5)^{3/2}\right]$$

$$= 934\ \text{s}$$

The value of t_2 may be obtained directly by Eq. 9.40 as

$$t_2 = \frac{2A}{C_d a\sqrt{2g}}(H_1^{1/2} - H_2^{1/2})$$

$$H_1 = (6-1.5) = 4.5\ \text{m}$$

and

$$H_2 = (3.6-1.5) = 2.1\ \text{m}$$

$$\therefore \quad t_2 = \frac{2\times2.8\left[(4.5)^{1/2}-(2.1)^{1/2}\right]}{0.625\times(9.3\times10^{-4})\times\sqrt{(2\times9.81)}}$$

$$= 1462\ \text{s}$$

$\therefore$ Total time

$$t = (934 + 1462)$$

$$= 2\,396\ \text{s} = 39.93\ \text{min.}$$

Example 9.18. *A cylindrical boiler, diameter 2 m and length 10 m is lying horizontally and is half full of water. It is to be emptied by an external cylindrical mouthpiece of diameter 75 mm fixed at the bottom of the boiler. How long will the mouthpiece take to empty the boiler? Take the coefficient of discharge for the mouthpiece as 0.82.*

Solution

From Eq. 9.42, we have

$$t = \frac{4l}{3C_d a\sqrt{2g}}[(2R-H_2)^{3/2}-(2R-H_1)^{3/2}]$$

$$l = 10 \text{ m}; C_d = 0.82; a = \frac{\pi}{4}(0.075)^2$$

$$= 4.42\times10^{-3} \text{ m}^2; (2R) = 2 \text{ m}; H_1 = 1 \text{ m}; \text{and } H_2 = 0$$

Thus by substitution, we get

$$t = \frac{4\times10}{3\times0.82\times4.42\times10^{-3}\times\sqrt{2\times9.81}}[(2)^{3/2}-1]$$

$$= 1518.56 \text{ s} = 25.31 \text{ min.}$$

Example 9.19. *A swimming bath 36 m long and 12 m wide has vertical sides, and the bottom slopes uniformly from a depth of 1.2 m at the shallow end to 2.1 m at the deep end. There are two outlets at the bottom of the deeper end, one is an orifice 0.225 m diameter and the other a small pipe 0.75 m long fitted on the outside of a similar opening of 0.225 m diameter. Find the time taken to empty the tank if the values of C_d for the orifice and the mouthpiece are 0.62 and 0.85 respectively.*

Solution

As shown in Fig. Ex. 9.19 upto the shallow end the tank has a constant cross-sectional area but below the shallow end the cross-sectional area varies. Thus the total time t taken to empty may be divided into two parts t_1 and t_2 required to empty these two portions, which may be computed as indicated below.

Let at any instant water level be at a height h above the centre of the openings and in time dt, let the water level be lowered by dh.

Then $\quad Qdt = -A\,dh$

$$Q = \left[0.62\times\frac{\pi}{4}(0.225)^2\times\sqrt{2gh}\right]+\left[0.85\times\frac{\pi}{4}(0.225)^2\times\sqrt{2gh}\right]$$

$$= 0.259\sqrt{h}$$

$$A = (36\times12) = 432 \text{ m}^2; H_2 = 2.1 \text{ m};$$

$$H_1 = (2.1-1.2) = 0.9 \text{ m}$$

Thus by substitution, we have

$$dt = -\frac{432dh}{0.259\sqrt{h}}$$

By integrating both sides, we have

$$\int_0^{t_1} dt = -\int_{2.1}^{0.9}\frac{432dh}{0.259\sqrt{h}}$$

$$\therefore \quad t_1 = \frac{432\times2}{0.259}\left[(2.1)^{1/2}-(0.9)^{1/2}\right]$$

$$= 1669.5 \text{ s}$$

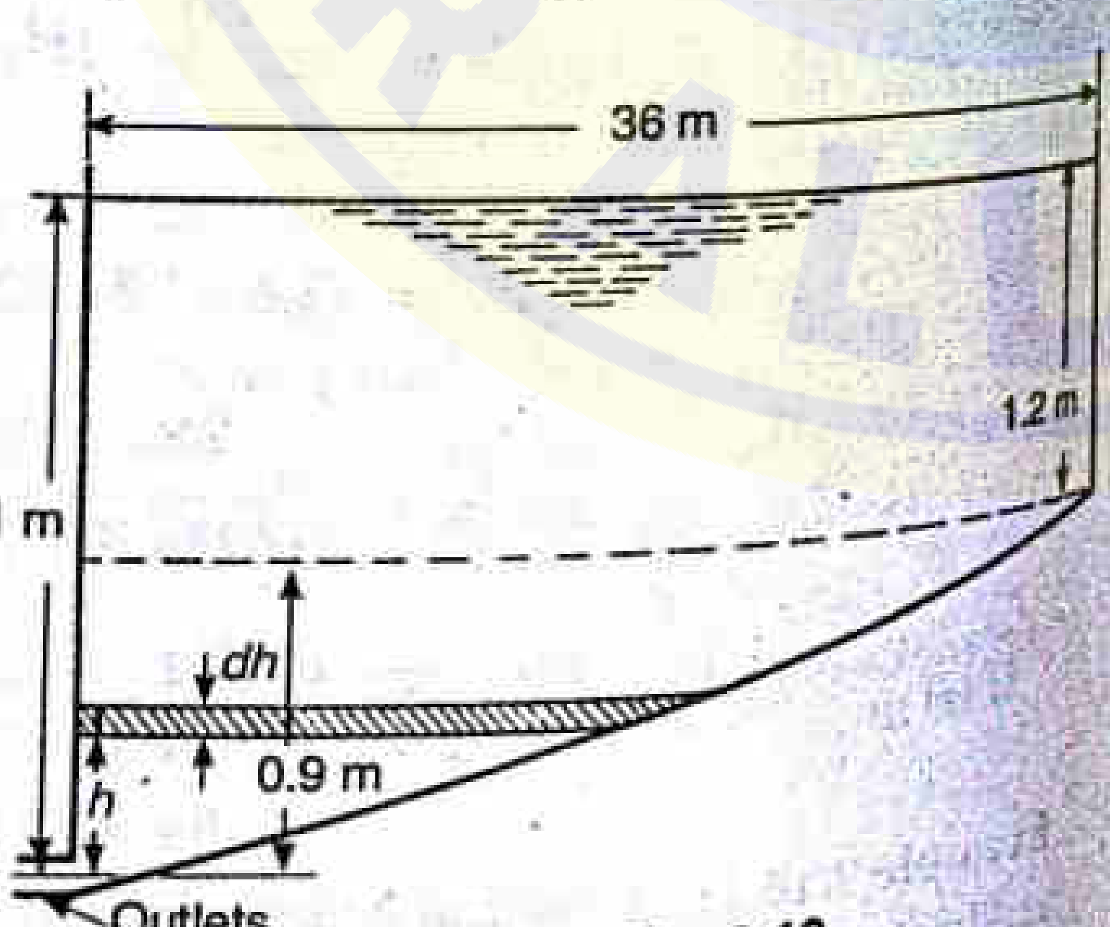


Figure Ex. 9.19

As shown in Fig. Ex. 9.19 if h is the height of the water level above the centre of the opening at any instant in the lower portion of the tank then cross-sectional area of the tank at this level is

$$A = \left(\frac{36\times h\times12}{0.9}\right) = 480\,h$$

Again if in time dt the water level is lowered by dh, then

$$Qdt = -A\,dh$$

or

$$dt = -\frac{Adh}{Q} = \frac{-480\,h\,dh}{0.259\sqrt{h}}$$

h varies from $\quad H_1 = 0.9$ m to $H_2 = 0$

Thus by integrating both sides, we have

$$\int_0^{t_2} dt = -\frac{480}{0.259}\int_{0.9}^{0} h^{1/2}dh$$

or

$$t_2 = \frac{480}{0.259}\times\frac{2}{3}[(0.9)^{3/2}-0]$$

$$= 1054.9 \text{ s}$$

Therefore total time,

$$t = (t_1+t_1)$$

$$= (1669.5+1054.9)$$

$$= 2724.4 \text{ s} = 45.41 \text{ min}$$

Example 9.20. *A tank 3 m high and filled with water has the form of frustrum of a cone with the smaller end pointing downwards. The upper end is 2.4 m diameter and lower end 1.2 m diameter. The bottom end contains a central orifice whose average coefficient of discharge is 0.6, which is to empty the tank in 6 minutes. Find the size of the orifice.*

Solution

Refer Fig. 9.16 (c)

$$R_1 = \frac{2.4}{2} = 1.2 \text{ m}; R_0 = \frac{1.2}{2} = 0.6 \text{ m}$$

$$H_1 = 3 \text{ m}$$

$$\therefore \quad H_0 = \frac{R_0H_1}{R_1-R_0} = \frac{0.6\times3}{1.2-0.6} = 3 \text{ m}$$

From Eq. 9.43, we have

$$t = -\frac{\pi R_1^2}{C_d a\sqrt{2g}(H_1+H_0)^2}$$

$$\times\left[\frac{2}{5}h^{5/2}+2H_0^2h^{1/2}+\frac{4}{3}H_0h^{3/2}\right]_{H_1}^{H_2}$$

$$t = (6\times60) = 360 \text{ s}; C_d = 0.6; H_2 = 0; \text{and } H_1 = 3 \text{ m}$$

Let d be the diameter of the orifice. Thus by substitution, we have

$$360 = \frac{\pi\times(1.2)^2}{0.6\times\frac{\pi}{4}d^2\times\sqrt{2\times9.81}\times(3+3)^2}$$

$$\times\left[\frac{2}{5}(3)^{5/2}+2\times(3)^2\times(3)^{1/2}+\frac{3}{4}\times(3)\times(3)^{3/2}\right]$$

$$\therefore \quad d = 0.098\,65\text{ m} = 98.65\text{ mm}.$$

Example 9.21. *A rectangular tank 6 m long by 1.5 m wide is divided into two parts by a partition so that one part is 4 times the other part. The water level in the large portion is 3 m above that in the smaller. Find the time required for the difference of water levels in the two portions to be reduced to 1.2 m, if the water flows through an orifice in the bottom of the partition having an area of 58×10^{-4} m² and $C_d = 0.6$.*

Solution

From Eq. 9.47, we have

$$t = \frac{2A_1A_2(H_1^{1/2}-H_2^{1/2})}{C_d a(A_1+A_2)\sqrt{2g}}$$

$$A_1 = 4A_2$$

and $\quad (A_1+A_2) = (6\times1.5) = 9\text{ m}^2$

$$\therefore \quad A_1 = 7.2\text{ m}^2;\text{ and } A_2 = 1.8\text{ m}^2$$

$$H_1 = 3\text{ m}; H_2 = 1.2\text{ m}$$

$$C_d = 0.6; a = 58\times10^{-4}\text{ m}^2$$

Thus by substitution, we have

$$t = \frac{2\times(7.2\times1.8)\left[(3)^{1/2}-(1.2)^{1/2}\right]}{0.6\times58\times10^{-4}\times(7.2+1.8)\times\sqrt{2\times9.81}}$$

$$= 119\text{ s} = 1\text{ min }59\text{ s}$$

Example 9.22. *A reservoir 180 m × 120 m at bottom has slopes 1 to 1 and a depth of 7.5 m of water. Find the time taken to reduce the water level by 2.5 m by a short tube 1m diameter at the bottom having $C_d = 0.70$.*

Solution

Let at any instant the water level be at a height h above the orifice and let in time dt the water level be reduced by dh. Then

$$Qdt = -Adh$$

$$Q = C_d a\sqrt{2gh} = 0.70\times\frac{\pi}{4}(1)^2\times\sqrt{2\times9.81\times h}$$

$$A = (180+2h)(120+2h)$$

Thus by substitution, we have

$$dt = -\frac{(180+2h)(120+2h)dh}{0.70\times\frac{\pi}{4}(1)^2\times\sqrt{2\times9.81\times h}}$$

By integrating both sides, we have

$$\int_0^t dt = -\frac{4}{0.70\times\pi\times\sqrt{2\times9.81}}\int_{H_1}^{H_2}\frac{(180+2h)(120+2h)dh}{\sqrt{h}}$$

$$H_1 = 7.5\text{ m; and } H_2 = (7.5-2.5) = 5\text{ m}$$

$$\therefore \quad t = \frac{-4\times4}{0.7\times\pi\times\sqrt{2\times9.81}}\times\int_{7.5}^{5}\frac{(90+h)(60+h)dh}{\sqrt{h}}$$

$$= 10\,517\text{ s} = 2\text{ h }55\text{ min}$$

Example 9.23. *A cylindrical tank of internal diameter 0.6 m, length 1.5 m and axis vertical has a 50 mm diameter sharp-edged orifice ($C_d = 0.6$) in the bottom opening to atmosphere. The tank is open at the top and is empty. If water is admitted into the tank from above at a constant rate of 14 litres per second how long will it take to just fill the tank? How much water will escape through the orifice during that period?*

Solution

From Eq. 9.46, we have

$$t = \frac{2A}{K^2}\left[Q\log_e\frac{Q-K\sqrt{H_2}}{Q-K\sqrt{H_1}}+K(\sqrt{H_2}-\sqrt{H_1})\right]$$

$$A = \frac{\pi}{4}\times(0.6)^2 = 0.2827$$

$$K = C_d a\sqrt{2g} = 0.6\times\frac{\pi}{4}(0.05)^2\times\sqrt{2\times9.81} = 0.00522$$

$$Q = 14\ l/\text{s} = 0.014\text{ m}^3/\text{s}$$

$$H_1 = 0;\text{ and } H_2 = 1.5\text{ m}$$

Thus by substitution, we have

$$t = \frac{2\times0.2827}{(0.005\,22)^2}$$

$$\times\left[0.014\log_e\left\{\frac{0.014-0.005\,22(1.5)^{1/2}}{0.014}\right\}+0.005\,22(1.5)^{1/2}\right]$$

$$= 44.55\text{ s}$$

Total inflow $= (0.014\times44.55) = 0.6237\text{ m}^3$

Water contained in the tank

$$= \frac{\pi}{4}(0.6)^2\times1.5 = 0.4241\text{ m}^3$$

$\therefore$ Water escaped through the orifice

$$= (0.6237-0.4241)$$

$$= 0.199\,6\text{ m}^3$$

Example 9.24. *A cylindrical tank is placed with its axis vertical and is provided with a circular orifice 40 mm in diameter in its bottom. Water flows into the tank at a uniform rate and is discharged through the orifice. It is found that it takes 107 seconds for the head in the tank to rise from 0.60 m to 0.76 m and takes 120 seconds for it to rise from 1.20 m to 1.28 m. Find the rate of inflow in cumec and the cross-sectional area of the tank. Take* $C_d = 0.62$.

Solution

In Eq. 9.46

$$K = C_d a\sqrt{2g}$$

$$= 0.62 \times \frac{\pi}{4}(0.04)^2 \times \sqrt{2\times 9.81}$$

$$= 0.00345$$

Two values of (dh/dt) are given as indicated below.

For an average head

$$h = \frac{0.60+0.76}{2} = 0.68 \text{ m}$$

$$dh = 0.16 \text{ m}$$

and $$dt = 107 \text{ s}$$

or $$\frac{dh}{dt} = \frac{0.16}{107}$$

For an average head

$$h = \frac{1.20+1.28}{2} = 1.24 \text{ m}$$

$$dh = 0.08 \text{ m}$$

and $$dt = 120\text{s}$$

or $$\frac{dh}{dt} = \frac{0.08}{120}$$

Since $$Adh = \left(Q - K\sqrt{h}\right) dt$$

$$\frac{dh}{dt} = \frac{Q-K\sqrt{h}}{A}$$

Thus by substitution, we get

$$\frac{0.16}{107} = \frac{1}{A}(Q - 0.00345\sqrt{0.68}) \quad \text{...(i)}$$

$$\frac{0.08}{120} = \frac{1}{A}(Q - 0.00345\sqrt{1.24} \quad \text{...(ii)}$$

Solving Eqs (i) and (ii), we get

$$Q = 0.0046 \text{ cumec}$$

$$A = 1.204 \text{ m}^2$$

Example 9.25. *A tank one metre square in plan area at all levels is provided with an orifice* ($C_d = 0.677$) *of area 0.01 m² in its bottom. The tank receives from the top a constant supply of water which would maintain a constant water level in the tank at a height of 4 m above the orifice. Calculate the rate of rise of the water level in the tank (a) when the tank is empty and water just enters the tank, and (b) when the head over the orifice is 2 m.*

Solution

When a constant head of 4 m is maintained, then inflow

$$= Q = \text{outflow through orifice}$$

$$= C_d a\sqrt{2gh}$$

$$= 0.677 \times (0.01) \times \sqrt{2\times 9.81\times 4}$$

$$= 0.06 \text{ m}^3/\text{s}$$

When the inflow and outflow are not balanced, then

$$Adh = (Q-q)\,dt$$

(a) When the tank is empty and the water just enters the tank, $q = 0$ as $h = 0$, and hence

$$\frac{dh}{dt} = \frac{Q}{A}$$

$$= \frac{0.06}{1\times 1} = 0.06 \text{ m/s}$$

(b) When the head over the orifice is 2 m, then

$$q = 0.677 \times 0.01 \times \sqrt{2\times 9.81\times 2}$$

$$= 0.042\,4 \text{ m}^3/\text{s}$$

$$\therefore \quad \frac{dh}{dt} = \frac{Q-q}{A}$$

$$= \frac{0.06-0.0424}{1\times 1} = 0.0176 \text{ m/s}$$

Example 9.26. *A streamlined nozzle of diameter d is supplied at a constant head, the magnitude of the head being large compared to d. The nozzle discharges directly into the atmosphere and so shaped that the issuing jet is paralled at the nozzle exit. To increase the flow rate a shroud of diameter D is firmly secured to the nozzle as shown. The jet expands to fill the shroud and the shroud is long enough to ensure that the flow leaving it is steady and parallel.*

Determine what the diameter of the shroud should be to maximize the flow rate. Neglect shear stresses at the walls of the shroud.

Also obtain the expression for the maximum discharge.

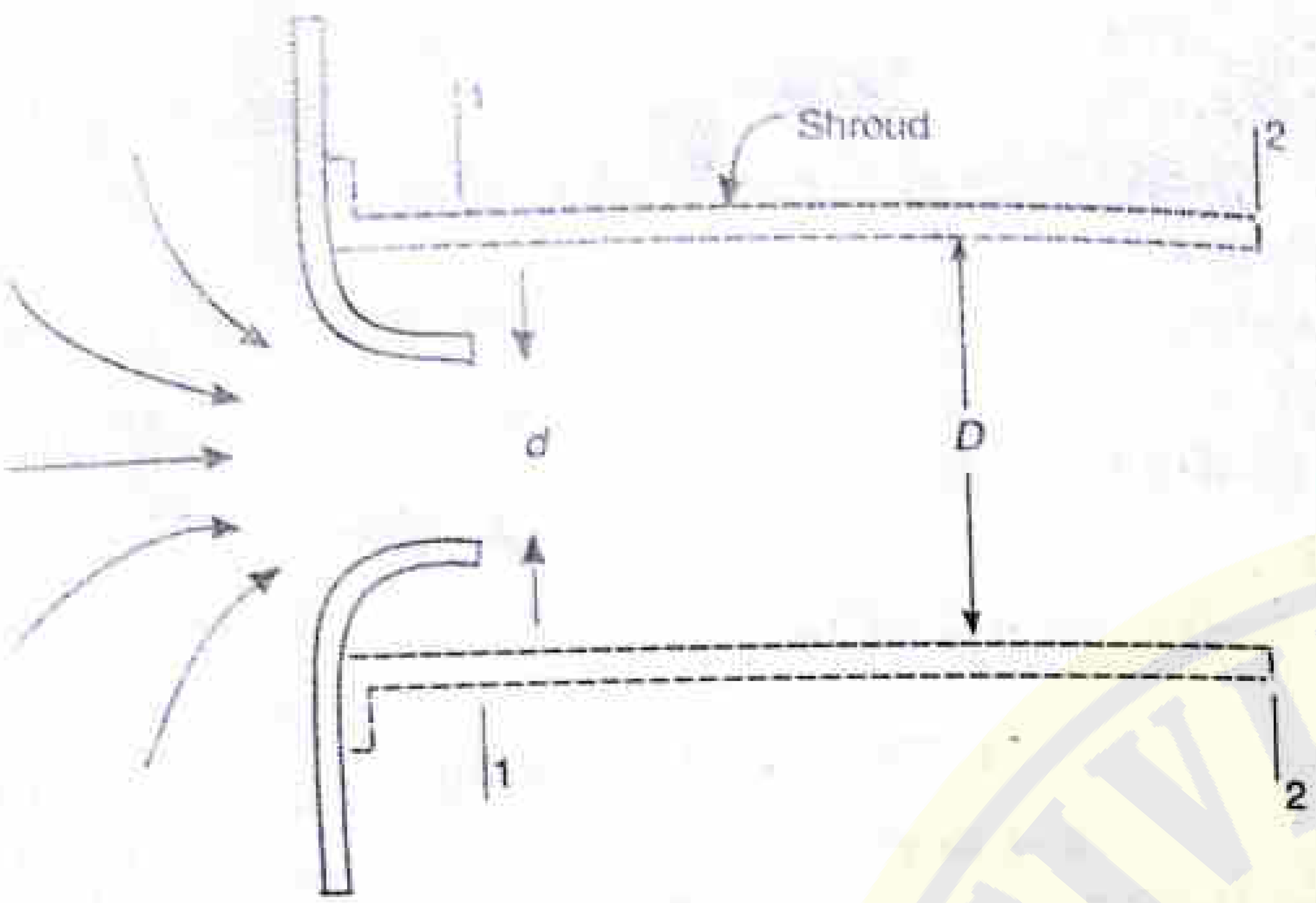


Figure Ex. 9.26

Solution

The head loss between sections 1–1 and 2–2 is due to sudden enlargement and is given by equation 9.25 as

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

If H is the constant head above the centre line of the nozzle, then applying Bernoulli's equation between the liquid surface in the supplying tank and section 2–2 at the exit end of the shroud, we get

$$H = \frac{V_2^2}{2g} + \frac{(V_1 - V_2)^2}{2g}$$

If Q is the discharge, then by continuity, we have

$$Q = \frac{\pi d^2}{4} \times V_1 = \frac{\pi D^2}{4} V_2$$

or

$$V_1 = \frac{4Q}{\pi d^2};\ \text{and } V_2 = \frac{4Q}{\pi D^2}$$

$\therefore$

$$H = \frac{1}{2g}\left[\left(\frac{4Q}{\pi D^2}\right)^2 + \left\{\left(\frac{4Q}{\pi d^2}\right) - \left(\frac{4Q}{\pi D^2}\right)\right\}^2\right]$$

$$= \frac{8Q^2}{\pi^2 g}\left[\frac{1}{D^4} + \frac{1}{d^4} + \frac{1}{D^4} - \frac{2}{D^2 d^2}\right]$$

or

$$Q^2 = \frac{\pi^2 gH}{8}\left[\frac{D^4 d^4}{2d^4 - 2d^2 D^2 + D^4}\right]$$

For Q to be maximum, $\frac{dQ}{dD} = 0$, which gives

$$D = \sqrt{2}\, d$$

Substituting this value of D in the equation for Q, the expression for maximum discharge is Q_{max} obtained as

$$Q_{max}^2 = \frac{\pi^2 gH}{8}\left[\frac{4d^8}{2d^4 - 4d^4 + 4d^4}\right]$$

or

$$Q_{max} = \frac{\pi d^2}{2\sqrt{2}}\sqrt{2gH}$$

SUMMARY OF MAIN POINTS

1. An orifice is an opening having a closed perimeter, made in the walls or the bottom of a tank or a vessel containing fluid, through which the fluid may be discharged.
   According to the size, the orifices may be classified as small and large orifices. According to the shape, the orifices may be classified as circular, rectangular, square and triangular. According to the shape of the upstream edge the orifices may be classified as sharp-edged orifices and bell-mouthed orifices. According to the discharge conditions, the orifices may be classified as orifices discharging free and drowned or submerged orifices. The drowned or submerged orifices may be further classified as fully submerged orifices and partially submerged orifices.
2. A mouthpiece is a short tube of length not more than two to three times its diameter, which is fitted to a circular opening or office of the same diameter, provided in a tank or a vessel containing fluid, such that it is an extension of the orifice and through which also the fluid may be discharged.
   According to the shape the mouthpiece may be classified as cylindrical, convergent and convergent-divergent. According to the position the mouthpieces may be classified as external and internal mouthpieces. An internal mouthpiece is also called re-entrant or Borda's mouthpiece.
   According to the discharge conditions the mouthpieces may be classified as running full and running free mouthpieces. The running free condition of discharge may be developed only in the case of internal mouthpieces.
3. Theoretical velocity V_{th} of jet of liquid emerging from an orifice under a head h is given as

   $$V_{th} = \sqrt{2gh}$$

4. The various coefficients for an orifice are defined as
   (i) Coefficient of velocity

   $$C_v = \frac{\text{Actual velocity at vena contracta}}{\text{Theoretical velocity}}$$

   $$C_v = \sqrt{\frac{x^2}{4hy}}$$

   where x and y are the coordinates of a point in the jet measured from the vena contracta.
   (ii) Cofficient of contraction

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

   (iii) Cofficient of discharge

   $$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}}$$

$$C_d = C_c \times C_v$$

(iv) Coefficient of resistance

$$C_r = \frac{\text{Loss of kinetic energy}}{\text{Actual kinetic energy}}$$

$$C_r = \left(\frac{1}{C_v^2} - 1\right)$$

Depending on the shape and size of the orifice and on the head of liquid under which the flow takes place, C_c varies from 0.61 to 0.69, C_v varies from 0.95 to 0.99, and C_d varies from 0.61 to 0.65.

For a small sharp edged circular orifice discharging water and other liquids of similar viscosity the average values of these coefficients may be taken as

C_c = 0.64 or 0.65 ; C_v = 0.97 or 0.98 ; C_d = 0.62 or 0.63

6. Discharge through a small orifice of area a under a head h measured above the centre of the orifice is given by

$$Q = C_d a \sqrt{2gh}$$

7. Discharge through a large rectangular orifice is given by

$$Q = \frac{2}{3} C_d b \sqrt{2g}\left[H_2^{3/2} - H_1^{3/2}\right]$$

where

b = width of orifice ;
C_d = coefficient of discharge for the orifice, usually taken as 0.6;
H_1 = height of the liquid surface above the top edge of the orifice;
H_2 = height of the liquid surface above the bottom edge of the orifice; and
g = acceleration due to gravity.

8. Discharge through a totally submerged orifice is given by

$$Q = C_d a \sqrt{2g(H_1 - H_2)}$$

where

a = area of the orifice;
C_d = coefficient of discharge for the orifice ;
H_1 = height of the liquid surface on the upstream side above the centre of the orifice ;
H_2 = height of the liquid surface on the downstream side above the centre of the orifice; and
g = acceleration due to gravity.

9. Discharge through a partially submerged orifice is given by

$$Q = Q_1 + Q_2$$

where Q_1 and Q_2 are respectively the discharges through the free and the submerged portions of the orifice, which are given as

$$Q_1 = \frac{2}{3} C_{d_1} b \sqrt{2g}\left[H^{3/2} - H_2^{3/2}\right]$$

$$Q_2 = C_{d_2} b (H_1 - H)\sqrt{2gH}$$

where

b = width of the orifice;
H = difference between the liquid surfaces on the upstream and the downstream sides of the orifice
H_1 = height of the liquid surface on the upstream side above the bottom edge of the orifice;
H_2 = height of the liquid surface on the upstream side above the top edge of the orifice;
C_{d_1} = coefficient of discharge for the free portion of the orifice;
C_{d_2} = coefficient of discharge for the submerged portion of the orifice; and
g = acceleration due to gravity.

10. Discharge through a mouthpiece of area a under a head h measured above the centre of the mouthpiece is given by

$$Q = C_d a \sqrt{2gh}$$

(i) For external mouthpiece, $C_d = 0.855$
(ii) For internal mouthpiece,
(a) running full, $C_d = 0.707$
(b) running free, $C_d = 0.50$
(iii) For convergent, divergent and convergent-divergent mouthpieces, C_d = 0.975 to 1.0

11. For external mouthpiece absoulte pressure head H_c at vena-contracta is given as

$$H_c = H_a - 0.89\,H$$

where

H_a = atmosphereic pressure head = 10.3 m of water; and
H = height of liquid surface above the centre of the mouthpiece.

For internal mouthpiece absolute pressure head H_c at vena-contracta is given as

$$H_c = H_a - H$$

where H_a and H are same as indicated above.

12. For convergent-divergent mouthpiece the ratio of the area a at the outlet end the area a_c at the vena-contracta is given as

$$\frac{a}{a_c} = \sqrt{1 + \frac{H_a - H_c}{H}}$$

where

H_a = atmospheric pressure head;
H_c = absoulute pressure head at vena-contracta; and
H = height of liquid surface above the centre of the mouthpiece.

13. Time of emptying a cylindrical tank with its axis vertical, through an orifice at its bottom is given by

$$t = \frac{2A\left(H_1^{1/2} - H_2^{1/2}\right)}{C_d a \sqrt{2g}}$$

where

H_1 = initial height of liquid surface above the centre of the orifice;
H_2 = final height of liquid surface above the centre of the orifice;
A = cross-sectional area of the tank;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

If the tank is to be completely emptied then time t is given by

$$t = \frac{2A\left(H_1^{1/2}\right)}{C_d a \sqrt{2g}} \qquad (\text{Since } H_2 = 0)$$

14. Time of emptying a cylindrical tank with its axis horizontal, through an orifice at its bottom is given by

$$t = \frac{4l}{3 C_d a \sqrt{2g}}\left[(2R - H_2)^{3/2} - (2R - H_1)^{3/2}\right]$$

where

l = length of the tank;
R = radius of the tank;
H_1 = initial height of liquid surface above the centre of orifice;
H_2 = final height of liquid surface above the centre of orifice;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

15. Time of emptying a hemispherical tank through an orifice at its bottom is given by

$$t = \frac{2\pi}{C_d a \sqrt{2g}}\left[\frac{2}{3}R\left(H_1^{3/2} - H_2^{3/2}\right) - \frac{1}{5}\left(H_1^{5/2} - H_2^{5/2}\right)\right]$$

where

R = radius of the tank;
H_1 = initial height of liquid surface above the centre of the orifice;
H_2 = final height of liquid surface above the centre of the orifice;
a = area of the orifice;
C_d = coefficient of discharge for the orifice; and
g = acceleration due to gravity.

If the tank was full at the beginning and it is to be completely emptied then the time t is given by

$$t = \frac{14\pi R^{5/2}}{C_d a \sqrt{2g}}$$

(Since $H_1 = R$, and $H_2 = 0$)

16. Time required to reduce the difference between the liquid surfaces in the two tanks is given by

$$t = \frac{2A_1\left(H_1^{1/2} - H_2^{1/2}\right)}{C_d a\left(1 + \frac{A_1}{A_2}\right)\sqrt{2g}}$$

where

A_1 = cross-sectional area of tank from which the liquid flows;
A_2 = cross-sectional area of tank into which the liquid flows;
H_1 = initial difference between the liquid surfaces in the two tanks;
H_2 = final difference between the liquid surfaces in the two tanks;
a = area of the orifice;
C_d = coefficient of discharge for the orifice, and
g = acceleration due to gravity.

If both the tanks are of the same cross-sectional areas, i.e., $A_1 = A_2$, then

$$t = \frac{A_1\left(H_1^{1/2} - H_2^{1/2}\right)}{C_d a \sqrt{2g}}$$

17. Time required to empty a canallock is given by

$$t = \frac{2AH}{C_d a_2 \sqrt{2gh}}$$

Time required to fill a canal lock is given by

$$t = \frac{A(H+h)}{C_d a_1 \sqrt{2gh}}$$

where

A = cross-sectional area of the lock;

H = difference of water levels on the upstream reach and the downstream reach (also known as lift);

h = height of the water surface of the upper reach above the centre of the upper orifices;

a_1 = area of the upper orifices;

a_2 = area of the lower orifices;

C_d = coefficient of discharge for the orifices and

g = acceleration due to gravity.

PROBLEMS

9.1 Define 'vena-contracta'. Explain how is it developed.

9.2 Define the various coefficients for an orifice.

9.3 Describe the different methods for the determination of the various coefficients for an orifice.

9.4 What is a mouthpiece? What is the advantage of providing a mouthpiece?

9.5 Show that discharge of water through a sharp-edged orifice shall be increased by about 38% if a short cylindrical mouthpiece of the same diameter is fitted to it on the outside of the tank. Take coefficient of contraction as 0.62 and neglect friction.

9.6 A jet of water issuing from an orifice 25 mm diameter under a constant head of 1.5 m falls 0.915 m vertically before it strikes the ground at a distance of 2.288 m measured horizontally from the vena-contracta. The discharge was found to be 102 litres per minute. Determine C_d, C_v and C_c for the orifice.

[*Ans.* 0.638; 0.976; 0.654]

9.7 A sharp-edged orifice of 50 mm diameter discharges water under a head of 4.5 m. Find the coefficient of discharge if the measured rate of flow is 11.76 litres per second. If there is an average pressure within the jet in the plane of the orifice of 39.044 kN/m² [0.398 kg(f)/cm²] above atmospheric pressure, calculate the coefficient of contraction. Neglect loss of energy due to friction. [*Ans.* 0.637; 0.677]

9.8 Oil of specific gravity 0.75 flows through a 80 mm diameter orifice whose coefficients of velocity and contraction are 0.95 and 0.65 respectively. At what pressure the air be pumped in the space above the oil surface so that the power in the jet may be 7.356 kW [10 hp]. Take height of oil above the centre of the orifice as equal to 2.7 m.

[*Ans.* 117.329 kN/m² [1.197 kg(f)/cm²]]

9.9 A vessel containing water has a vertical circular orifice 25 mm diameter which is at first plugged up. The tank is suspended in such a way that any displacing force can be accurately measured. On the removal of the plug the horizontal force required to keep it in place, applied opposite to the orifice is 16 N [1.631 kg (f)]. The discharge was found to be 141 litres per minute. The level of the water being maintained at a constant height of 2.745 m above the orifice. Determine C_c, C_v and C_d for the orifice. [*Ans.* 0.703; 0.928; 0.652]

9.10 Compensation water is to be discharged by two circular orifices under a constant head of 0.9 m measured to the centres of the orifices. What diameter will be required to give a discharge of 1510⁶ litres per day? Assume C_d = 0.66.

[*Ans.* 0.2 m]

9.11 A large tank has a rectangular sharp-edged orifice 1 m broad and 0.75 m deep, the top edge of which is 0.45 m below the level of water in the tank. Find the quantity of water flowing through the orifice in cumec, if the coefficient of discharge is 0.62. [*Ans.* 1.854 cumec]

9.12 An orifice in the side of a large tank is rectangular in shape 1.2 m wide and 0.7 m deep. The water level on one side of the orifice is 1.2 m above the top edge and the water level on the other side of the orifice is 0.25 m below the top edge. Compute the discharge if C_d = 0.62. [*Ans.* 2.734 m³/s]

9.13 A horizontal boiler 2.4 m diameter and 7.5 m long contains water to a depth of 1.5 m. Find the time of emptying the boiler through a short vertical pipe 75 mm in diameter attached to the bottom of the boiler. Take C_d = 0.8.

[*Ans.* 0.508 *h*]

9.14 A tank has two indentical orifices, one vertically above the other and 1m apart, in one of its vertical sides. The water surface is 1.22 m above the higher orifice and is maintained at a constant level. It is found that the jets intercept each other at a horizontal distance of 2.65 m from the vena-contracta. Determine the C_v for the orifices.

[*Ans.* 0.805]

9.15 Water flows vertically downward through a 0.15 m diameter pipe at a rate of 50 litres per second. The pipe suddenly enlarges to 0.3 m diameter. Find the loss of head due to this sudden enlargement. Also find the loss of head if the same discharge passes vertically upward through the pipe, the coefficient of contraction being 0.62. If A is a point 0.3 m above the enlargement and B is a point 0.3 m below it, find the excess of pressure head at B over that at A in both cases.

[*Ans.* 0.23 m; 0.153 m; 0.7524 m; 1.1354 m]

9.16 A vertical cylindrical tank is 0.6 m in diameter and water flows in it at a constant rate of 0.0042 m³/s. There is a sharp-edged orifice in the bottom having a cross-sectional area of 12.9 m². If C_d = 0.6, how long will it take for the depth in the tank to increase from 0.3 m to 0.6 m.

[*Ans.* 45 s]

9.17 A Borda's mouthpiece of 0.15 m diameter discharges water under a head of 3 m. Calculate (a) the discharge; and (b) the diameter of the jet at the vena-contracta for both conditions of flow.

[*Ans.* (a) Running free 68 *l*/s; Running full 95.85 *l*/s; (b) 0.106 1 to 0.108 2 m]

9.18 A convergent-divergent nozzle is fitted into the side of a tank containing water and, under a constant head H m above the centre line of the nozzle, discharges into the atmosphere. Obtain an expression for the ratio of the exit area of the nozzle to the area of the throat for maximum discharge, making the following assumptions: (i) the barometric pressure is 10.3 m of water, (ii) separation will occur at an absolute pressure head of 1.83 m of water, (iii) the only hydraulic losses occur in the divergent portion of the nozzle and amount to 25 per cent of the head lost at a sudden enlargement for the same change of area.

If the supply head H is 2.75 m and the throat diameter is 0.05 m, calculate the maximum discharge.

$$\left[\frac{\text{Exit area}}{\text{Throat area}} = \frac{4[\sqrt{(H+8.47)(H-2.1175)}] - (H+8.47)}{(3H-8.47)}\right]$$

[*Ans.* Q = 0.0291 m³/s]

9.19 Find the time required to empty a swimming pool through an opening provided in the bottom of the deep end. Depth of water at the deep end is 2.4 m and at the shallow end it is 0.9 m. Length of the pool is 27 m and the width of the pool is 9 m. Area of the opening is 9.2 m² and its C_d = 0.6. [*Ans.* 11.166 min]

9.20 A reservoir is circular in plan, the diameter of the top water level is 90 m. At a depth of 1.5 m the diameter is 75 m. The mouth of the outlet pipe which is 0.6 m in diameter is 3.6 m below top water level. How long will it take to lower the depth of the water level by 1.5 m. Take C_d = 0.87. [*Ans.* 72.91 min]

9.21 A cylindrical tank 12 m high, containing water upto brink is completely emptied through a hole located in the bottom in 8.4 minutes. How long would it take for the water level to drop from the top of the tank when full to 4m from the top. [*Ans.* 92.49 s]

9.22 A tank has upper cylindrical portion of 3m diameter and 4 m high with a hemispherical base. If the tank is full of water, determine the time taken to empty it through an orifice 0.1 m diameter at its bottom. Take C_d = 0.62.

[*Ans.* 18.48 min]

Flow Over Notches and Weirs

Chapter **10**

10.1 INTRODUCTION

A *notch* may be defined as an opening provided in the side of a tank (or vessel) such that the liquid surface in the tank is below the top edge of the opening. Notches made of metallic plates are also provided in narrow channels (particularly in laboratory channels) in order to measure the rate of flow of liquid. As such in general notches are used for measuring the rate of flow of liquid from a tank or in a channel.

A *weir* is the name given to a concrete or masonry structure built across a river (or stream) in order to raise the level of water on the upstream side and to allow the excess water to flow over its entire length to the downstream side. Thus a weir is similar to a small dam constructed across a river, with the difference that whereas in the case of a dam excess water flows to the downstream side, only through a small portion called spillway, the same in the case of a weir flows over its entire length. Weirs may also be used for measuring the rate of flow of water in rivers or streams.

The sheet of water flowing through a notch or a weir is known as the *nappe* (French term meaning sheet) or *vein*. The bottom edge of a notch or the top of a weir over which the water flows is known as the *sill* or *crest*, and its height above the bottom of the tank or channel is known as the *crest height*.

10.2 CLASSIFICATION OF NOTCHES AND WEIRS

The notches are usually classified according to the shape of the opening as *rectangular notch*, *triangular notch (or V-notch)*, *trapezoidal notch*, *parabolic notch* and *stepped notch*. The edges of all these notches are bevelled on the downstream side so as to have sharp-edged sides and crest, resulting in minimum contact with the flowing liquid. Since the liquid surface is always below the top edge of the notch, a notch is usually provided with only a crest and sides with no top edge.

Notches may also be classified according to the effect of the sides on the nappe emerging from a notch, as *notch with end contraction* and *notch without end contraction* or *suppressed notch*. If the sides of a notch cause the contraction of nappe, then it is said to be notch with end contraction (or contracted notch). On the other hand if there is no contraction of the nappe due to the sides (or in other words the end contractions are suppressed) then it is known as a notch without end contraction. Notches provided in the sides of tanks or vessels are essentially the notches with end contraction. However, in a channel if the crest length of the notch is less than the width of the channel then it is a notch with end contraction. But if the crest length of the notch is equal to the width of the channel then it is a notch without end contraction or a suppressed notch.

Weirs may be classified according to the shape of the opening, the shape of the crest, the effect of the sides on the issuing nappe and the discharge conditions. According to the shape of the opening, the weirs may be classified as *rectangular*, *triangular and trapezoidal weirs*. A particular type of a trapezoidal weir is known as *Cipolletti weir*. According to the shape of the crest, the weirs may be classified as *thin-plate or sharp-edged weir*, *narrow-crested weir*, *broad-crested weir* and *Ogee shaped weir*. A sharp-edged weir may be formed by means of a metallic plate which is embedded in the concrete or masonry and therefore it is same as a notch of the similar shape. A like notches weirs may also be classified according to the effect of the sides on the issuing nappe as *weir with end contraction* and *weir without end contraction*. If water is allowed to flow over a portion of the weir so that the crest length of the weir over which water flows is less than the width of the channel then the nappe emerging from the weir is affected by the sides and it gets contracted. Such a weir is known as weir with end contraction (or *contracted weir*). However, if water is allowed to flow over the entire length of the weir so that the crest length of the weir is equal to the width of the channel, then the nappe emerging from the weir is not affected by the sides and therefore it does not undergo contraction. The weir is then known as weir without end contractio (or *suppressed weir* because the end contractions are suppressed). In some cases on the weir crest, pier are constructed to support a foot-bridge, on account of which the effective crest length of the weir ove which water flows is reduced. Moreover, the construction of the piers on the weir crest results i dividing the entire weir into a number of small weirs with end contraction (due to the piers). Accordin to the discharge conditions weirs may be classified as *freely discharging weir* and *submerged (or drowne weir*. If the water level on the downstream of the weir is well below the weir crest then the napp emerges freely in the atmosphere and it is known as freely discharging weir. On the other hand if th water level on the downstream of the weir is above the crest of the weir then it is known as submerge weir.

There exists a considerable similarity between the pattern of flow over a sharp (or narrow) creste weir and a notch of the same shape. As such the same expressions as derived below may be adopt or computing the rate of flow of water over a sharp (or narrow) crested weir and a notch of the sa shape. Moreover, on account of the similarity a notch is more often termed as sharp-crested weir.

10.3 FLOW OVER A RECTANGULAR SHARP-CRESTED WEIR OR NOTCH

Consider a rectangular sharp-crested weir or notch provided in a channel carrying water as shown Fig. 10.1. Let L be the length of the crest of the weir or notch and H be the height of the water surf above its crest, which is known as the head causing the flow over the weir or notch. As water flo over weir or notch, the surface of water over the crest and immediately upstream of it becomes curv The head H above the crest is therefore measured at a certain distance upstream of the weir or no where the water surface may be assumed to be unaffected by the curvature effect. According to 9108–1979,* the head H above the crest should be measured on the upstream of the weir or notch distance of 4 to 5 times the maximum head (H_{max}) above the crest.

For computing the discharge of water flowing over the weir or notch consider an elemen horizontal strip of water of thickness dh and length L at depth h below the water surface as show Fig. 10.1.

Area of strip is ($L \times dh$) and the theoretical velocity of the water flowing through the strip wi $\sqrt{2gh}$. Thus if dQ is the discharge through the strip, then

* IS : 9108–1979 provides detailed specifications for thin plate weirs or notches.

$$dQ = C_d \times L \times dh \times \sqrt{2gh}$$

where C_d is the coefficient of discharge. The total discharge Q for the entire weir or notch may be detrmined by integrating the above expression within limits 0 to H. Thus

$$Q = \int_0^H C_d L \sqrt{2gh}\, dh$$

Assuming the coefficient of discharge C_d to be constant for the entire weir or notch we obtained

$$Q = \frac{2}{3} C_d \sqrt{2g}\, L H^{3/2} \qquad ...(10.1)$$

In the above equation for the discharge over the weir or notch the *velocity of approach* has not been taken into account. The velocity of approach may be defined as the velocity with which water approaches or reaches the weir or notch before it passes over it. Thus if V_a is the velocity of approach, then the water flowing over the weir or notch possesses an additional head h_a equal to $\left(\frac{V_a^2}{2g}\right)$ due to the velocity of approach. Thus the limits of integration for the above equation become h_a and $(H+h_a)$ instead of 0 to H. The discharge Q passing over the weir or notch is then given by

$$Q = \int_{h_a}^{(H+h_a)} C_d L \sqrt{2gh}\, dh$$

or

$$Q = \frac{2}{3} C_d \sqrt{2g} L \left[(H+h_a)^{3/2} - h_a^{3/2}\right] \qquad ...(10.2)$$

Equation 10.2 gives the discharge of water flowing over a rectangular weir or notch when the velocity of approach is taken into account. The head $(H + h_a)$ is usually known as *still water head*.

Equations 10.1 and 10.2 are applicable to a suppressed rectangular weir or notch only, for which the crest length is equal to the width of the channel. For a rectangular weir or notch with end contraction these equations are to be modified as indicated below, in order to take into account the effect of the end contraction on the discharge over such a weir or notch. It has been indicated by J.B. Francis on the basis of his experiments that the end contraction has the effect of decreasing the effective length of the crest of the weir or notch, due to the contraction of the nappe, which results in decreasing the discharge. Further it has been proposed by Francis that the amount by which the crest length is reduced depends on the head H and for each end contraction the reduction in the crest length may be taken to be equal to $(0.1\,H)$ or $(H/10)$. Thus if there are n end contractions for a weir or notch of the crest length L then the effective length of the crest of the weir or notch will be $(L - 0_1\, n\, H)$. The discharge Q of water flowing over a weir or notch with n end contractions when the velocity of approach is notconsidered, is then given by

$$Q = \frac{3}{2} C_d \sqrt{2g}\, (L - 0.1\, n\, H)^{3/2} \qquad ...(10.3)$$

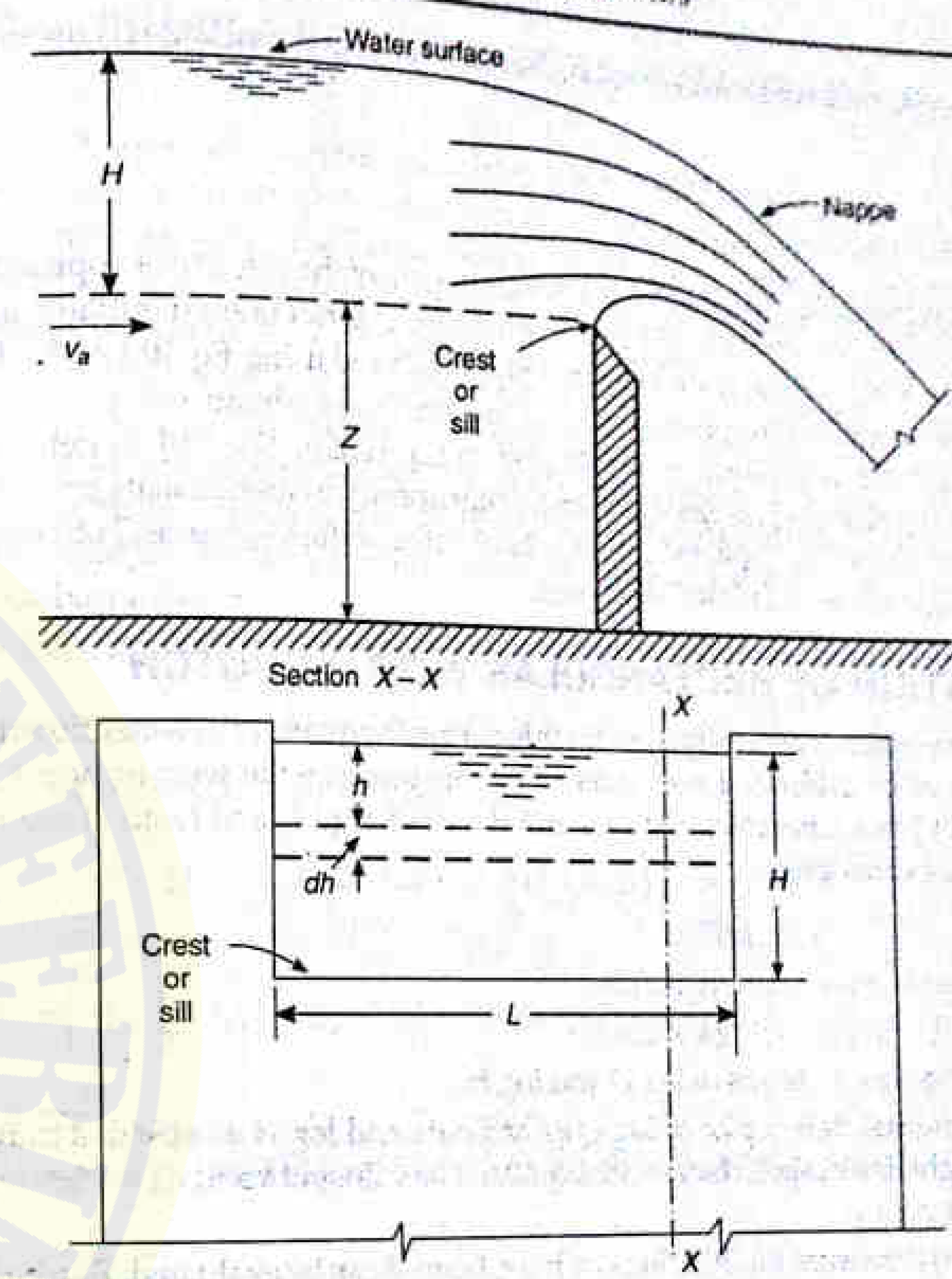


Figure10.1 *Flow over rectangular sharp crested weir or notch*

However, for such a weir or notch if the velocity of approach is taken into account, then the discharge Q is given by

$$Q = \frac{2}{3} C_d \sqrt{2g} (L - 0.1\, n\, H_1)\left(H_1^{3/2} - h_a^{3/2}\right) \qquad ...(10.4)$$

in which $\qquad H_1 = (H + h_a) = \left[H + \left(V_a^2/2g\right)\right]$

The mean value of the velocity of approach to be used in Eqs 10.2 and 10.4 may be determined by dividing the discharge Q by the area of the flow section of the channel. Thus if the height of the crest of the weir or notch above the bottom of the channel is Z, then the depth of the flow of water in the channel is $(H + Z)$. Now if the width of the channel is equal to B then the area of the flow section of the channel becomes $B(H + Z)$ and the velocity of approach is given by

$$V_a = \frac{Q}{B(H+Z)} \qquad ...(10.5)$$

However, in case of suppressed weir or notch, the width of the channel equals the crest length L, in which case

$$V_a = \frac{Q}{L(H+Z)} \qquad \text{...(10.5a)}$$

It may be observed from Eq.. 10.5 that for the determination of the velocity of approach the discharge Q of water flowing in the channel should be known. Since Q is not known initially, an approximate value of Q is computed by neglecting the velocity of approach and using Eq. 10.1 or Eq. 10.3, depending on whether the weir or notch is suppressed or not. Using this approximate value of Q, an approximate value of V_a is then computed which is substituted in Eq. 10.2 or Eq. 10.4 to get another value of discharge Q. The new value of Q obtained will be a better approximation than its previously calculated value. The process is, therefore, repeated until the final value of the discharge Q obtained is within 1% of the preceding value of the calculated discharge.

10.4 CALIBRATION OF RECTANGULAR WEIR OR NOTCH

A rectangular weir or notch is generally used for measuring the quantity of water flowing in a channel, in which case it must be calibrated previously. For this purpose the weir or notch which is to be calibrated is installed in a laboratory channel and the discharge Q and head H are measured. The discharge Q may be expressed as

$$Q = kH^n$$

By taking log of both sides of this equation

$$\log Q = \log k + n \log H \qquad \text{... (10.6)}$$

which is a straight line law in terms of log Q and log H.

From the experimental data a plot of log Q as ordinate and log H as abscissa may be prepared, which will be a straight line in accordance with Eq. 10.6. The values of k and n may be determined from the plot as described below.

For $H = 1$, $\log H = 0$ then from Eq. 10.6, $\log Q = \log k$, hence k can be evaluated. Further from Eq. 10.6,

$$n = \frac{\log Q - \log k}{\log H}$$

Therefore by choosing any convenient point on the straight line, and substituting the corresponding values of log Q and log H in the above equation the value of n can be calculated.

As alternative method is to assume $Q = kH^{3/2}$ where $k = \frac{2}{3}C_d\left(\sqrt{2g}\right)L$. From the experimental data a plot of Q and $H^{3/2}$ may be prepared which is accordance with the above expression will be a straight line, the slope of which will give the value of k. However, in this method a perfect straight line may not be obtained because the value of C_d though assumed to be constant, varies slightly with the head. From the value of k evaluated by either of the methods the coefficient of discharge C_d for the rectangular weir or notch may be determined.

The value of C_d for a rectangular weir or notch is approximately 0.6. However the value of C_d depends on head H and the ratio (H/Z) where Z is the height of the crest above the bed of the channel. Further if the head is small the value of C_d is also affected by viscosity and surface tension.

10.5 EMPIRICAL FOMULA FOR DISCHARGE OVER RECTANGULAR WEIRS

A rectangular weir is frequently used for measuring the rate of flow of water in channels. As such various investigators have conducted experiments with rectangular weirs on the basis of the experimental results they have proposed a number of empirical formulae for computing the discharge over rectangular weirs. Some of these formulae which are commonly used are described below.

1. Francis' Formula. It is one of the most commonly used formula for computing the discharge over sharp or narrow crested weirs with or without end contraction. This formula was proposed by J. B. Francis on the basis of his extensive series of experiments with rectangular weirs during the years 1848 to 1852 at Lowell, Massachusetts (USA). The length of the crest varied from 1.07 m to 5.19 m, but in most of the experiments the crest length was 3.05 m. The head H varied from 0.18 m to 0.49 m. On the basis of the results obtained from more than eighty experiments. Francis proposed the following formulae for the discharge over rectangular weir.

When the velocity of approach is not considered

$$Q = 1.84(L - 0.1\ nH)H^{3/2} \qquad \text{...(10.7)}$$

With the velocity of approach taken into account

$$Q = 1.84(L - 0.1\,nH_1)\left(H_1^{3/2} - h_a^{3/2}\right) \qquad \text{...(10.8)}$$

where $H_1 = (H + h_a) = \left(H + \frac{V_a^2}{2g}\right)$, H is the height of water surface above the crest of the weir, and n is the number of end contractions.

By comparing the above noted Francis' formulae with the Eqs. 10.3 and 10.4, we obtained

$$\frac{2}{3}C_d\sqrt{2g} = 1.84$$

from which $\quad C_d = 0.623$

i.e., according to Francis' formula the value of C_d for a rectangular weir is 0.623.

If the end contractions are suppressed i.e., if the crest length of the weir over which water flows is equal to the width of the channel, then $n = 0$ and Eq. 10.7 and 10.8 reduce to

$$Q = 1.84LH^{3/2} \qquad \text{... (10.9)}$$

and

$$Q = 1.84L\left(H_1^{3/2} - h_a^{3/2}\right) \qquad \text{... (10.10)}$$

Figure 10.2 shows the typical cases of flow over rectangular weirs for which the different values of n as indicated thereon may be adopted.

2. Bazin's Formula. In 1886 H. Bazin of France undertook a series of experiments with rectangular weirs. On the basis of the results of these experiments Bazin proposed the following formulae for the discharge over rectangular weir.

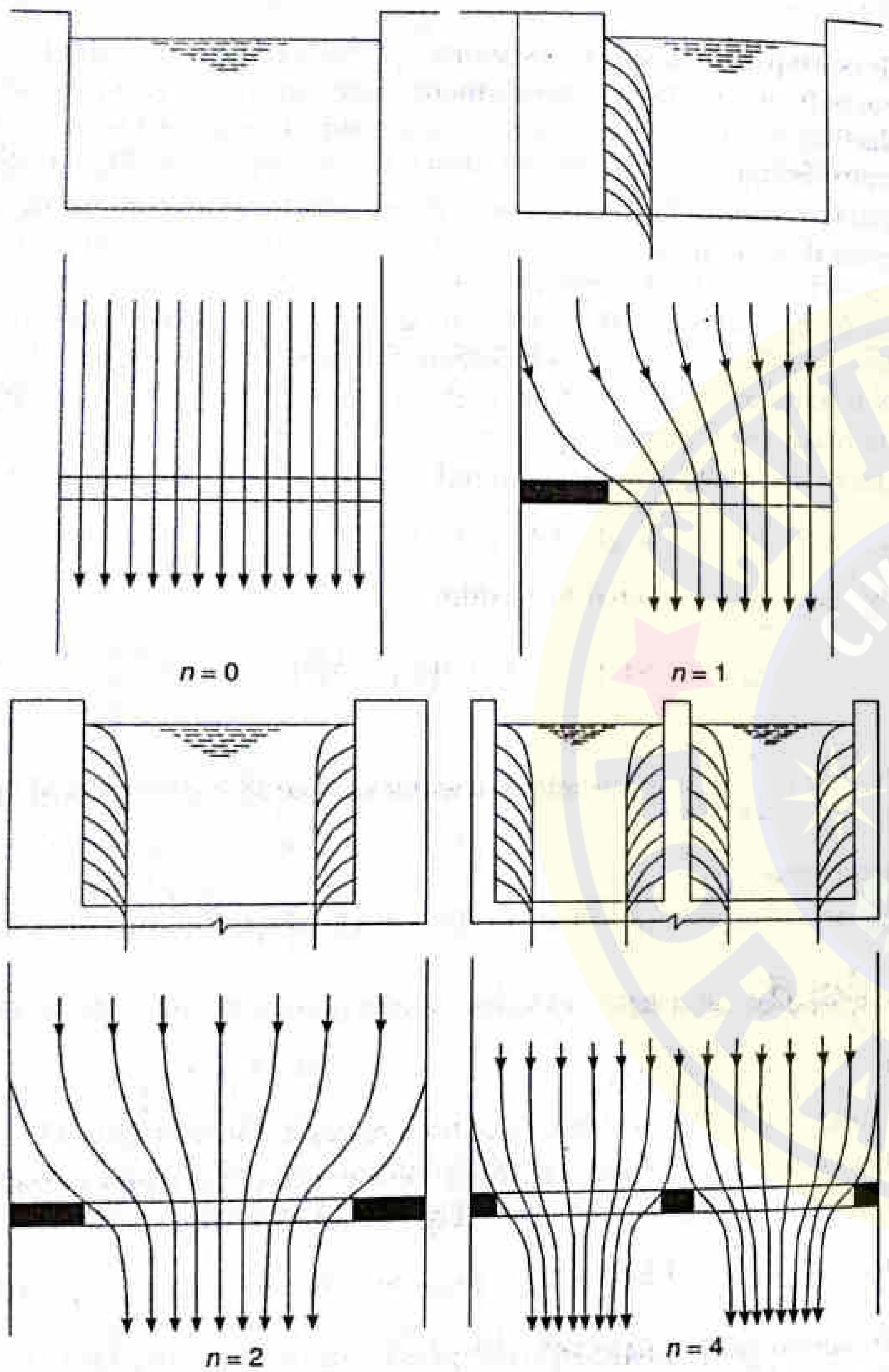


Figure 10.2 *End contractions for rectangular weirs*

For the velocity of approach not considered.

$$Q = m\sqrt{2g}\, LH^{3/2} \qquad \text{...(10.11)}$$

in which according to Bazin the value of the coefficient m is given by

$$m = \left(0.405 + \frac{0.003}{H}\right)$$

With the velocity of approach taken into account

$$Q = m_1\sqrt{2g}\, LH_1^{3/2} \qquad \text{... (10.12)}$$

in which the coefficient m_1 is given by

$$m_1 = \left(0.405 + \frac{0.003}{H_1}\right)$$

and H_1, which is called the still water head is given by

$$H_1 = \left(H + \alpha\frac{V_a^2}{2g}\right)$$

where α is a constant, the mean value of which has been given by Bazin as 1.6.

3. Rehbock's Formula. In 1929 T. Rehbock on the basis of his experiments with suppressed rectangular weirs proposed the following empirical formula for the coefficient of discharge C_d for a suppressed weir

$$C_d = \left(0.605 + 0.08\frac{H}{Z} + \frac{0.001}{H}\right) \qquad \text{... (10.13)}$$

where H is the head in metres and Z is the crest height in metres. The Rehbock's formula for the discharge over rectangular weir then becomes

$$Q = \frac{2}{3}\left(0.605 + 0.08\frac{H}{Z} + \frac{0.001}{H}\right)\sqrt{2g}\, LH^{3/2} \qquad \text{... (10.14)}$$

10.6 VENTILATION OF WEIRS

In the case of suppressed weir, since the crest-length is equal to the width of the channel, the nappe emerging from the weir touches the side walls of the channel. On account of this, air is trapped in the space between the side walls of the channel, the falling nappe, the weir and the bottom of the channel as shown in Fig. 10.3. This air is gradually carried away with the flowing water, thereby reducing the pressure in the space below the nappe, which may even become negative i.e., below atmospheric pressure or vacuum pressure. The negative pressure so developed in the space draws the lower nappe more and more towards the downstream surface of the weir as shown in Fig. 10.3 (b). Such a nappe is known as *depressed nappe*, which results in drawing more water thereby increasing the actual discharge over the weir. With a further withdrawal of the air from the space when no air is left below the nappe, the nappe may adhere to the downstream surface of the weir as shown in Fig. 10.3 (c). Such a nappe is called *adhering or clinging nappe*, which further results in drawing more water, thereby further increasing the actual discharge over the weir.

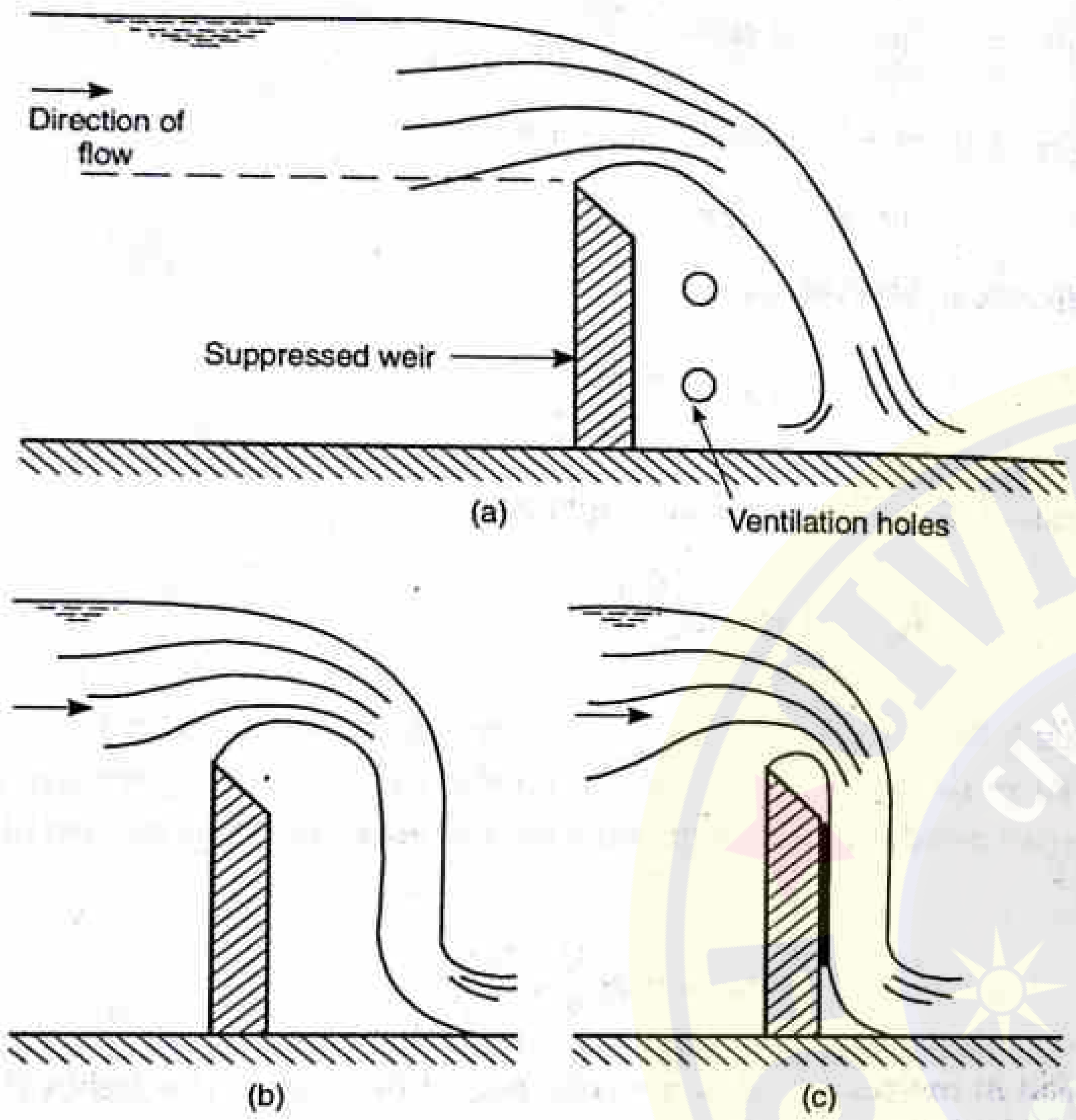


Figure 10.3 *Different types of nappe over a suppressed weir*: (*a*) *Free nappe;* (*b*) *Depressed nappe;* (*c*) *Adhering or clinging nappe*

The various formulae both analytical and empirical as indicated earlier for computing the discharge over suppressed rectangular weirs are based on the assumption that in the space below the emerging nappe there is atmospheric presure. Such a nappe is known as *free nappe* [Fig. 10.3 (a)]. A free nappe may be developed for a suppressed weir if the pressure in the space below the nappe is always atmospheric and it is not allowed to be reduced. For this purpose just downstream of the weir, holes are made in the side walls of the channel in the space below the lower nappe, so that this region is connected to the atmospheric air outside and the air which is carried away with the flowing water from this region is recouped from the atmosphere through these holes. These holes are thus called ventilation holes and the weirs are called ventilated weirs. Experimentally it has been found that the minimum cross-sectional area of the ventilation holes may be about 0.5 per cent of $(L \times Z_1)$ where L is the crest length and Z_1 is the depth of water surface on the downstream side below the crest of the weir. Generally two 25 mm diameter holes are sufficient for discharges upto 100 litres per second. However, if a suppressed weir is not properly ventilated then as stated earlier a partial withdrawal of the air takes place which may lead to the development of a depressed nappe. Moreover, if a suppressed weir is not ventilated then the air from the space below the nappe may be completely removed which may lead to the development of an adhering or clinging nappe. It has been observed in actual practice that

the discharge over a weir with depressed nappe is about 6 to 7% more than that obtained with a free nappe. Further the discharge over a weir with adhering nappe is about 25 to 39% more than that obtained with a free nappe.

When a suppressed weir is used as measuring device, then it should be properly ventilated so that a free nappe is developed and the actual discharge of water flowing over the weir is same as that obtained by using any of the formulae indicated in the preceding section. On the other hand if a weir with either depressed or adhering nappe is used for the discharge measurement then the actual discharge of water flowing over the weir being more, the use of any of the formulae for the computation of the discharge will give wrong results.

10.7 FLOW OVER A TRIANGULAR WEIR (V-NOTCH WEIR) OR TRIANGULAR NOTCH (V-NOTCH)

A triangular weir is an ordinary weir which is having a triangular or V-shaped opening or notch provided in its body, so that water is discharged through this opening only. Thus a triangular weir and a triangular notch are the same and the discharge of water flowing over a triangular weir or notch may be computed by using the same expression as described below. Generally a triangular weir or notch is preferred to a rectangular weir or notch for measuring the low discharges. This is so because with low discharges if rectangular weirs or notches are used for measuring the discharges then the head over the crest of the weir or notch may be so small that it may not be possible to measure it accurately. For such cases a triangular weir or notch may be used because the crest length for a triangular weir or notch is equal to zero and therefore even for a low discharge the head over the crest is fairly large which can be measured more accurately.

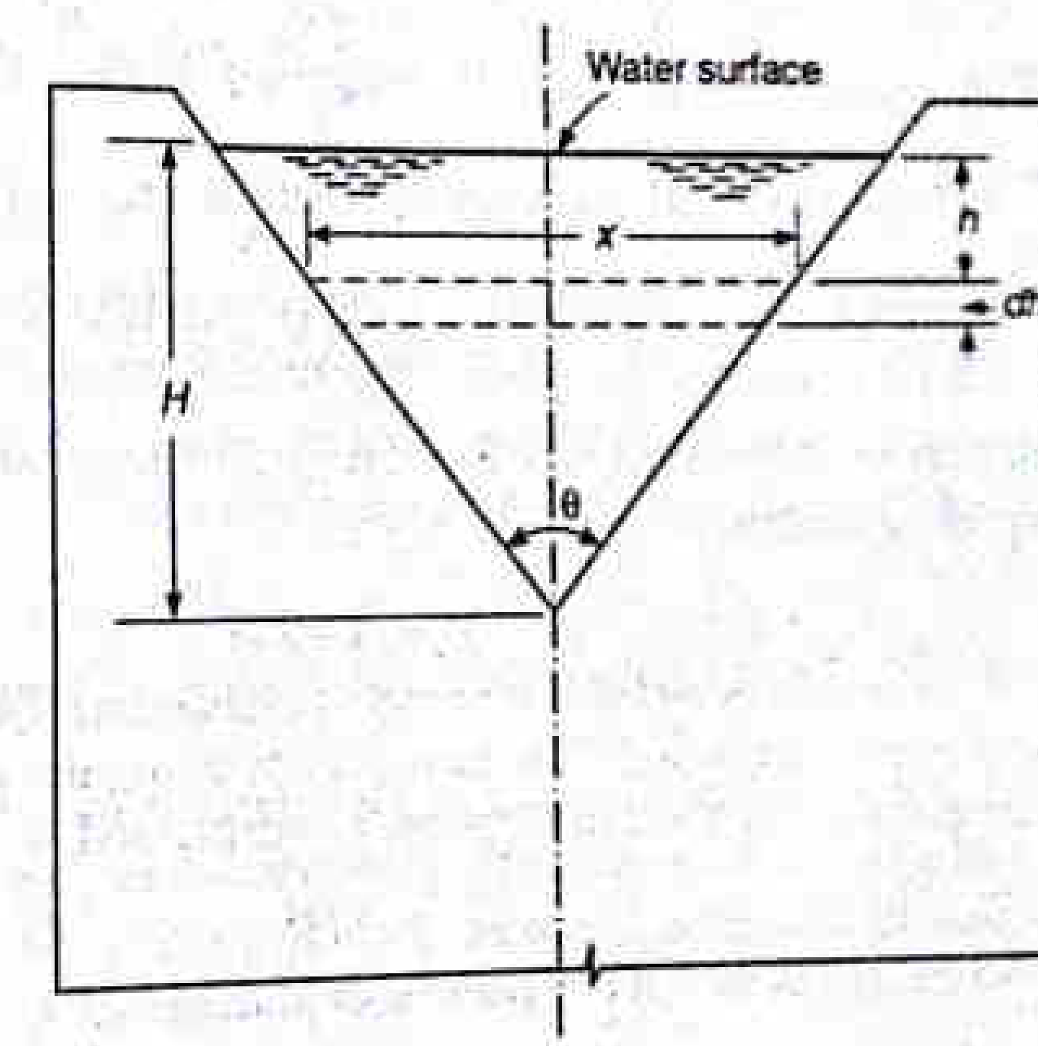


Figure 10.4 *Flow over triangular weir or notch*

Figure 10.4 shows a triangular weir or notch with a vertex angle equal to θ. Let H be the head above the crest of the weir. Consider a horizontal elementary strip of thickness dh at a depth h below the water surface. If x is the width of the strip then

$$\frac{x}{2(H-h)} = \tan\frac{\theta}{2}; \text{or } x = 2(H-h)\tan\frac{\theta}{2}$$

The area of the strip is $(x\,dh)$ or $[2(H-h)(\tan\theta/2)dh]$ and the theoretical velocity of water flowing through the strip will be $\sqrt{2gh}$.

Thus if dQ is the discharge through the strip then

$$dQ = C_d \times 2(H-h)(\tan\theta/2)dh \times \sqrt{2gh}$$

where C_d is the coefficient of discharge. The total discharge Q for the entire triangular weir or notch may be determined by integrating the above expression within limits 0 to H. Thus

$$Q = \int_0^H C_d \times 2(H-h)\tan\frac{\theta}{2}\sqrt{2gh}\; dh$$

Assuming the coefficient of discharge C_d to be constant for the entire weir or notch, we obtain

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

or

$$Q = 2C_d\sqrt{2g}\tan\frac{\theta}{2}\left[\frac{2}{3}Hh^{3/2}-\frac{2}{5}h^{5/2}\right]_0^H$$

or

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2} \quad ...(10.15)$$

If the vertex angle θ equals 90° then for a right-angled triangular weir or notch

$$Q = \frac{8}{15}C_d\sqrt{2g}\,H^{5/2} \quad ...(10.16)$$

since $\theta/2 = 45°$ and than $\theta/2$= 1. Further for a right angled weir or notch if C_d is assumed to be 0.6, then Eq. 10.16 becomes

$$Q = 1.418H^{5/2} \quad ...(10.17)$$

It is thus observed that for a right-angled triangular weir or notch the expression for the disharge is very much simplified. However, for any triangular weir or notch since the vertex angle θ is constant, the equation for the discharge may be expressed as

$$Q = KH^{5/2} \quad ...(10.18)$$

where K is a constant for the weir or notch given by

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2} \quad ...(10.19)$$

The value of K will, however, depend on the vertex angle θ and the coefficient of discharge C_d for the triangular weir or notch.

In deriving Eq. 10.15 for the discharge over a triangular weir or notch the velocity of approach has been neglected. In general for a triangular weir or notch the cross-section area of the approach channel is usually so much greater than that of the notch that the velocity of approach may be neglected. However, if the velocity of approach is to be taken into account then Eq. 10.15 for discharge Q over a triangular weir or notch may be modified as

$$Q = \frac{8}{15}C_d\sqrt{2g}\;\tan\frac{\theta}{2}\left[(H+h_a)^{5/2}-h_a^{5/2}\right] \quad ...(10.20)$$

where h_a is the head due to velocity of approach V_a and is equal to $(V_a^2/2g)$.

Advantages of Triangular Weir over Rectangular Weir. A triangular weir or notch has several advantages over a rectangular weir or notch which are as noted below:

(1) The nappe emerging from a triangular weir or notch has the same shape for all the heads. As such the value of the coefficient of discharge for a triangular weir or notch is fairly constant for all the heads. On the other hand for a rectangular weir or notch the shape of the nappe is affected by the head and therefore the coefficient of discharge varies with the head.

(2) For measuring low discharges a triangular weir or notch is very useful. This is because with a triangular weir or notch even with a low discharge the head over the crest is comparatively large which can be measured accurately. However if a rectangular weir is used to measure low discharge, the head over the crest may be very small which may not be measured accurately. Moreover, when the head over th crest is small, the flow is markedly affected by surface tension and viscosity which may also lead to inaccurate measurement of head.

(3) For a right-angled triangular weir or notch the expression for the computation of the discharge is very much simplified.

(4) In most of the cases of flow over a triangular weir or notch the velocity of approach may be neglected without introducing an appreciable error.

(5) Ventilation of a triangular weir is not necessary.

On account of the above mentioned advantages a triangular weir or notch is preferred to a rectangular weir or notch for measuring the discharges. For laboratory channels since the discharges are usually low, a right-angled triangular weir or notch is preferred to other types. The calibration of triangular weirs or notches may also be carried out in the same manner as in the case of rectangular weirs or notches as described in Section 10.4.

10.8 FLOW OVER A TRAPEZOIDAL WEIR OR NOTCH

As shown in Fig. 10.5 a trapezoidal weir or notch is a combination of a rectangular and a triangular weir or notch. As such the discharge over such a weir or notch may be determined by adding the discharges over the two different types. Thus if L is the crest length of a trapezoidal weir or notch and $\theta/2$ is the angle of inclination of its sides with the vertical then for such a weir

$$Q = \left[\frac{2}{3}C_{d_1}\sqrt{2g}\,LH^{3/2}+\frac{8}{15}C_{d_2}\sqrt{2g}\,\tan\frac{\theta}{2}H^{5/2}\right] \quad ...(10.21)$$

where C_{d_1} and C_{d_2} are respectively the coefficients of discharge for the rectangular and the triangular portions of the trapezoidal weir or notch. However, if for whole of the trapezoidal weir or notch the coefficient of discharge is assumed to be C_d then the expression for the discharge becomes

$$Q = C_d\sqrt{2g}\,H^{3/2}\left[\frac{2}{3}L+\frac{8}{15}H\tan\frac{\theta}{2}\right] \quad ...(10.22)$$

Cipolletti Weir (or Notch). A Cipolletti weir is particular type of trapezoidal weir, the sloping sides of which have an inclination of 1 horizontal to 4 vertical, $\left(\text{i.e., }\frac{\theta}{2}=14°\right)$. This weir was invented by an Italian engineer Cipolletti in 1887. As explained below the slope of 1 in 4 provided for the sides of this weir results in making the decrease in the discharge over a rectangular weir due to two end contractions just equal to the increase in the discharge trough the two triangular portions. So that

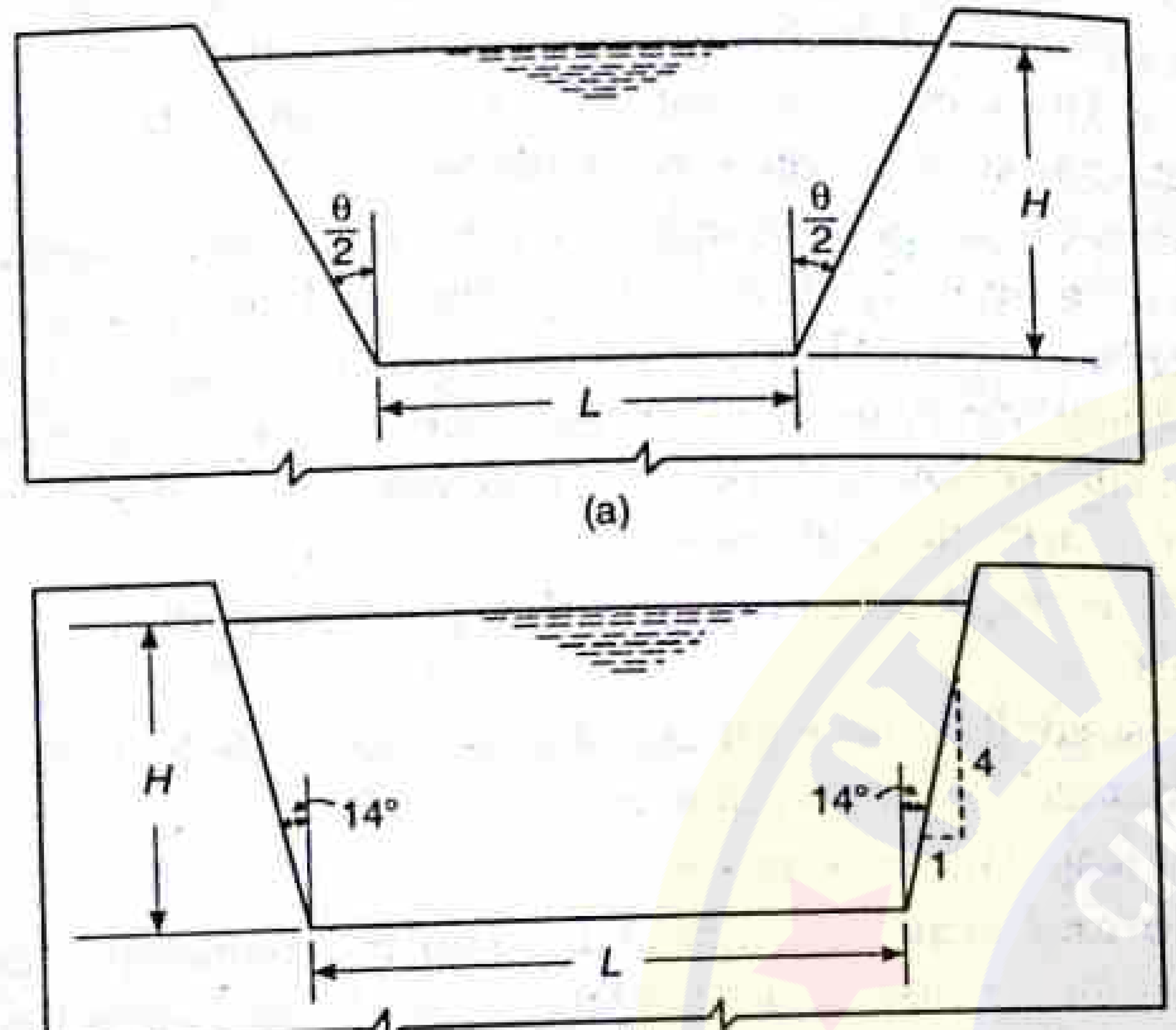


Figure 10.5 (*a*) *Trapezoidal weir;* (*b*) *Cipolletti weir*

the discharge over a Cipolletti weir may be computed by using the formula for a suppressed rectangular weir.

As in the case of a trapezoidal weir, for a Cipolletti weir also the discharge is given by

$$Q = (Q_1 + Q_2)$$

where Q_1 is the discharge through the rectangular portion and Q_2 is the discharge through the two triangular portions on either side. For a rectangular weir with two end contractions

$$Q_1 = \frac{2}{3} C_d \sqrt{2g}(L - 0.2H)H^{3/2}$$

and for a triangular weir

$$Q_2 = \frac{8}{15} C_d \sqrt{2g} \tan\frac{\theta}{2} H^{5/2}$$

For
$$\tan\frac{\theta}{2} = \frac{1}{4}$$

$$Q_2 = \frac{2}{15} C_d \sqrt{2g}\, H^{5/2}$$

Thus adding the two, the discharge for the Cipolletti weir is given by

$$Q = \frac{2}{3} C_d \sqrt{2g}(L - 0.2H)H^{3/2} + \frac{2}{15} C_d \sqrt{2g}\, H^{5/2}$$

or
$$Q = \frac{2}{3} C_d \sqrt{2g}\, H^{3/2}\left[L - \frac{H}{5} + \frac{H}{5}\right]$$

or
$$Q = \frac{2}{3} C_d \sqrt{2g}\, LH^{3/2} \quad \text{...(10.23)}$$

which is same as the Equation for the discharge over a suppressed rectangular weir. On the basis of his own experiments and those of Francis, Cipolletti proposed the following equation for the discharge over a Cipolletti weir

$$Q = 1.86LH^{3/2} \quad \text{...(10.24)}$$

By comparing Eqs. 10.23 and 10.24 the value of the coefficient of discharge C_d for a Cipolletti weir is obtained as 0.632. Again Eq. 10.24 is applicable only if the velocity of approach is neglected. However, if the velocity of approach V_a is to be considered then Eq. 10.23 may be modified as

$$Q = 1.86L\left(H_1^{3/2} - h_a^{3/2}\right) \quad \text{...(10.25)}$$

where
$$H_1 = (H + h_a) = H + \left(V_a^2/2g\right).$$

10.9 TIME REQUIRED TO EMPTY A RESERVOIR WITH RECTANGULAR WEIR

Consider a reservoir of uniform cross-sectional area A, which is provided with a rectangular weir or notch in one of its sides. Let L be the crest length of the weir or notch and C_d be its coefficient of discharge. It is required to determine the time taken to reduce the level of liquid in the reservoir from H_1 to H_2 above the crest of the weir or notch.

Let at any instant the height of the liquid surface above the crest of the weir or notch be h, and in a small time dt let the liquid surface in the reservoir be lowered by dh. Now if Q is the discharge of liquid flowing over the weir or notch during the interval or time dt, then

$$-Adh = Qdt$$

But
$$Q = \frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2}$$

Thus
$$-Adh = \frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2} dt$$

or
$$dt = -\frac{Adh}{\frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2}}$$

By integrating both sides of the above equation, the time t required to lower the liquid surface from H_1 to H_2 above the crest of the weir or notch is obtained as

$$\int_0^t dt = t = \int_{H_1}^{H_2} -\frac{Adh}{\frac{2}{3} C_d \sqrt{2g}\, Lh^{3/2}}$$

or
$$t = -\frac{A}{\frac{2}{3}C_d\sqrt{2g}\,L}\left[-2h^{-1/2}\right]_{H_1}^{H_2}$$

or
$$t = \frac{2A}{\frac{2}{3}C_d\sqrt{2g}\,L}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right] \quad ...(10.26)$$

If Bazin's formula is used for the computation of discharge then Eq. 10.26 becomes

$$t = \frac{2A}{m\sqrt{2g}}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right] \quad ...(10.27)$$

Similarly if Francis formula is used for the computation of discharge, then Eq. 10.26 becomes

$$t = \frac{2A}{1.84(L-0.1n\,H)}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right] \quad ...(10.28)$$

in which the value of H being taken as a mean of H_1 and H_2.

In order to determine the time required to lower the liquid surface to the crest of the weir or notch, H_2 must be considered to be equal to zero, in which case the time t in Eq. 10.26 becomes infinity. It therefore means that the liquid surface can never be reduced to the level of the crest of the weir or notch. This is however not true because the liquid in the reservoir is not at rest but it has some velocity with which it is approaching the weir or notch. This causes the time of emptying the reservoir upto the crest of the weir or notch to be finite, that is less than that given by Eq. 10.26. Moreover as $H_2 \to 0$ the effect of surface tension becomes significant, in which case Eq. 10.25 is not applicable.

If instead of a rectangular weir or notch, a triangular weir or notch is provided in the side of the tank in order to reduce the level of liquid in the tank from H_1 to H_2 above the crest of the weir or notch, then as in the previous case

$$-Adh = Qdt$$

But in this case

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}h^{5/2}$$

Thus
$$-Adh = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}h^{5/2}dt$$

or
$$dt = \frac{-Adh}{\frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}h^{5/2}}$$

By integrating both sides

$$t = \frac{\frac{2}{3}A}{\frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}}\times\left[\frac{1}{(H_2)^{3/2}}-\frac{1}{(H_1)^{3/2}}\right] \quad ...(10.29)$$

10.10 EFFECT ON COMPUTED DISCHARGE OVER A WEIR OR NOTCH DUE TO ERROR IN THE MEASUREMENT OF HEAD

The discharge over a rectangular weir or notch is proportional to $(H)^{3/2}$ and over a triangular weir or notch it is proportional to $(H)^{5/2}$, where H is the height of the liquid surface above the crest of the weir or notch. As such the accurate measurement of the head H is quite essential in order to obtain an accurate value of the discharge over the weir or notch. However, if an error is introduced in the measurement of the head, it will affect the computed discharge. It is, therefore, desired to determine upto what extent an error in measuring the head will affect the computed discharge over the weir or notch. For the rectangular and triangular weir or notch this may be computed as follows:

(a) *Rectangular weir or notch.* The discharge over a rectangular weir or notch is given by

$$Q = \frac{2}{3}C_d\sqrt{2g}\,LH^{3/2} = KH^{3/2}$$

where
$$K = \left(\frac{2}{3}C_d\sqrt{2g}\,L\right)$$

Differentiating the above equation

$$dQ = \frac{3}{2}KH^{1/2}dH$$

Then
$$\frac{dQ}{Q} = \frac{\frac{3}{2}KH^{1/2}dH}{KH^{3/2}} = \frac{2}{3}\frac{dH}{H}$$

That is, a percentage change in H produces 1.5 times the same percentage change in Q, or an error of 1 per cent in measuring H will produce 1.5 per cent error in the computed Q over a rectangular weir or notch.

(b) *Triangular weir or notch.* The discharge over a triangular weir or notch is given by

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2} = KH^{5/2}$$

where
$$K = \left(\frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}\right)$$

Differentiating the above equation

$$dQ = \frac{5}{2}KH^{3/2}dH$$

Then
$$\frac{dQ}{Q} = \frac{\frac{5}{2}KH^{3/2}dH}{KH^{5/2}} = \frac{5}{2}\frac{dH}{H}$$

Therefore a small change or error of 1 per cent in the head produces an error of 2.5 per cent in the computed discharge over a triangular weir or notch.

10.11 BROAD CRESTED WEIR

A weir having a wide crest is known as a broad crested weir. Such a weir differs from thin-plate and narrow-crested weirs in this respect that as water flows over it a different flow pattern is developed. Experiments have shown that if the width of the crest of the weir $B < 0.625\,H$ [i.e., $(H/B) > 1.6$] then it is known as thin-plate weir in which case the jet of water touches only the upstream edge of the weir and it flows clear of the downstream edge. If the width of the crest of the weir $B \geq 0.625\,H$ and $\leq 2.5H$ [i.e., $0.4 \leq (H/B) \leq 1.6$] it is known as narrow-crested weir in which case as the water flows over it the jet of water remains in contact with the entire crest. On the other hand when the width of the crest of the weir $B \geq 2.5H$ and $\leq 10H$ [i.e., $0.1 \leq (H/B) \leq 0.4$] it is known as *broad-crested weir* in which case as shown in Fig. 10.6 as water flows over it there occurs a drop in the water surface over the crest of the weir. There are two types of broad crested weirs namely (i) with a sharp corner at the upstream end [Fig. 10.6 (a)] and (ii) with a round corner at the upstream end [Fig. 10.6 (b)]. In the case of a broad crested weir with sharp corner at the upstream end a small air pocket is developed on the crest of the weir close to the upstream edge. This is due to the lower nappe being slightly raised by the upstream sharp edge and then it drops down and flows over the surface of the weir crest. From the air pocket so developed the air is gradually carried away by the flowing stream of water, thereby resulting in the reduction of the pressure in this portion, by which the cavitation may occur at the corner and consequently the weir may be damaged. On the other hand, in the case of a broad crested weir with the upstream corner rounded, no air pocket is formed and consequent possibility of cavitation in avoided. Therefore for most of the broad crested weirs the upstream edges are rounded in order to avoid cavitation.

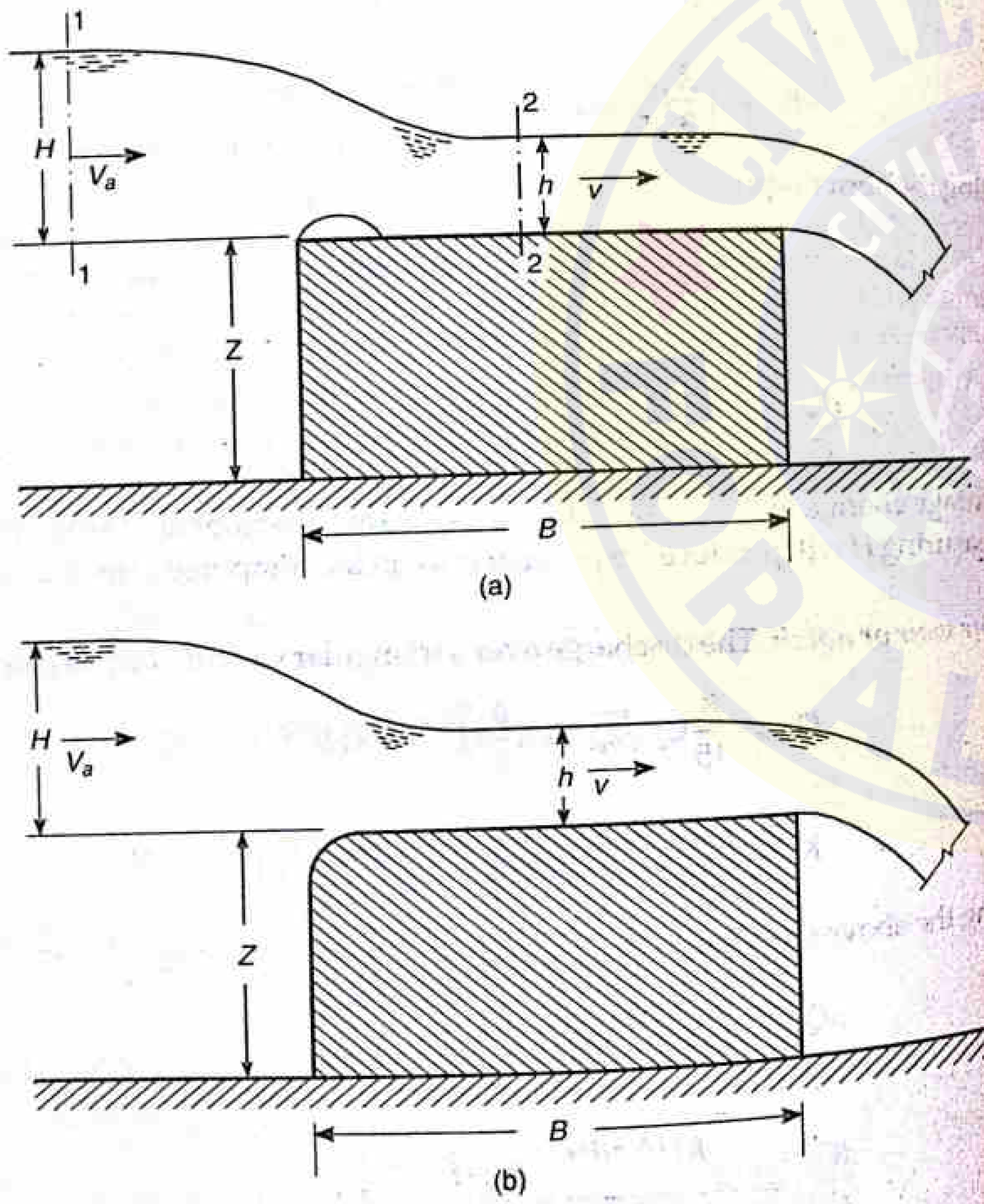


Figure 10.6 *Broad crested weir : (a) with sharp corner at the upstream end (b) with round corner at the upstream end*

Figure 10.6 shows the flow pattern over a broad crested weir. As water flows over a broad crested weir the water surface drops from H on the upstream of the weir to h over the weir crest, due to the reduction in the area of flow section and consequent increase in the velocity of flow of water. If the velocity of approach is neglected then just upstream from the crest the total head above the weir crest may be represented by H. Further over the middle portion of the crest of the weir the depth of flow h may be assumed to be uniform, where let the velocity of flow be v. Then applying Bernoulli's equation between section 1–1 just upstream of the weir and section 2-2 at the middle of the weir crest, we have

$$H = h + \frac{v^2}{2g}$$

or

$$v = \sqrt{2g(H-h)}$$

Now if L is the length of the crest of the weir then the discharge over the broad crested weir is given by

$$Q = Lh\sqrt{2g(H-h)}$$

In deriving the above expression the loss of head between section 1–1 and 2-2 has not been considered. As such the above expression gives only the theoretical discharge over a broad crested weir. The actual discharge over a broad crested weir is therefore obtained by introducing the coefficient of discharge C_d and hence

$$Q = C_d Lh\sqrt{2g(H-h)} \qquad \text{...(10.30)}$$

Equation 10.30 shows that for the computation of discharge over a broad crested weir, two heads H and h need be measured. However, experiments have shown that in the case of broad crested weirs the flow adjusts itself to have maximum discharge for the available head H. The condition for the discharge Q over a broad crested weir to be maximum for constant H may be obtained by differentiating Eq. 10.30 with respect to h, and equating $\left(\frac{dQ}{dh}\right)$ to zero.

Thus

$$\frac{dQ}{dh} = C_d L\sqrt{2g}\left[\sqrt{H-h} - \frac{h}{2\sqrt{H-h}}\right] = 0$$

or

$$h = \frac{2}{3}H.$$

This value of h is known as the *critical depth*. In other words, the discharge over a broad crested weir

IS: 6059–1971 provides detailed specifications for weirs of finite crest width (or broad crested weirs) is maximum when the critical depth of flow occurs over the surface of the weir crest. Introducing this value of h in Eq. 10.30 the discharge over a broad crested weir is given by

$$Q = C_d L\left(\frac{2}{3}H\right)\sqrt{2g}\left(\frac{1}{3}H\right)^{1/2} = 1.70\, C_d L H^{3/2} \quad \cdots (10.31)$$

Equation 10.31 gives maximum discharge over a broad crested weir, but as stated earlier for broad crested weirs the flow adjusts itself to discharge at the maximum rate. Therefore the discharge over a broad crested weir may be computed by using Eq. 10.31 in which only the head H on the upstream of the weir is required to be measured. The value of C_d for a broad crested weir may also be determined by calibrating it previously with the liquid the discharge of which it will measure. In general the value of C_d for a broad crested weir varies from 0.85 to 1.0.

In deriving the above equation for discharge over a broad crested weir the velocity of approach has been neglected. However if the velocity of approach V_a is to be considered then Eq. 10.30 may be modified as

$$Q = 1.70\, C_d L H_1^{3/2} \quad \cdots (10.32)$$

in which $$H_1 = (H + h_a) = \left[H + \left(\frac{V_a^2}{2g}\right)\right]$$

10.12 SUBMERGED WEIRS

When the water level on the downstream of the weir is above the crest of the weir then the weir is said to be a submerged weir. During floods often the weirs constructed across rivers become submerged. Submerged weirs have larger discharging capacity as compared with freely discharging weirs, thereby indicating that during floods when river carries huge quantity of water, the flow adjusts itself by setting the weir in the state of submergence, so that the discharge over the weir is increased and the flood water is quickly discharged to the downstream side.

As shown in Fig. 10.7 the discharge over a submerged weir may be obtained by dividing it into two parts. The portion between the upstream and downstream water surfaces may be treated as a free weir and the portion between the downstream water surface and the crest of the weir may be treated as a drowned orifice. Thus if Q_1 and Q_2 are the discharges through the free and the drowned portions respectively then

$$Q_1 = \frac{2}{3} C_{d_1} L\sqrt{2g}\,(H_1 - H_2)^{3/2}$$

and $$Q_2 = C_{d_2}(L \times H_2)\sqrt{2g(H_1 - H_2)}$$

where H_1 and H_2 are respectively the heads on the upstream and the downstream of the weir; L is the length of the weir; and C_{d_1} and C_{d_2} are the coefficients of discharge for the free and the drowned portions respectively.

The total discharge over a submerged weir is then

$$Q = Q_1 + Q_2$$

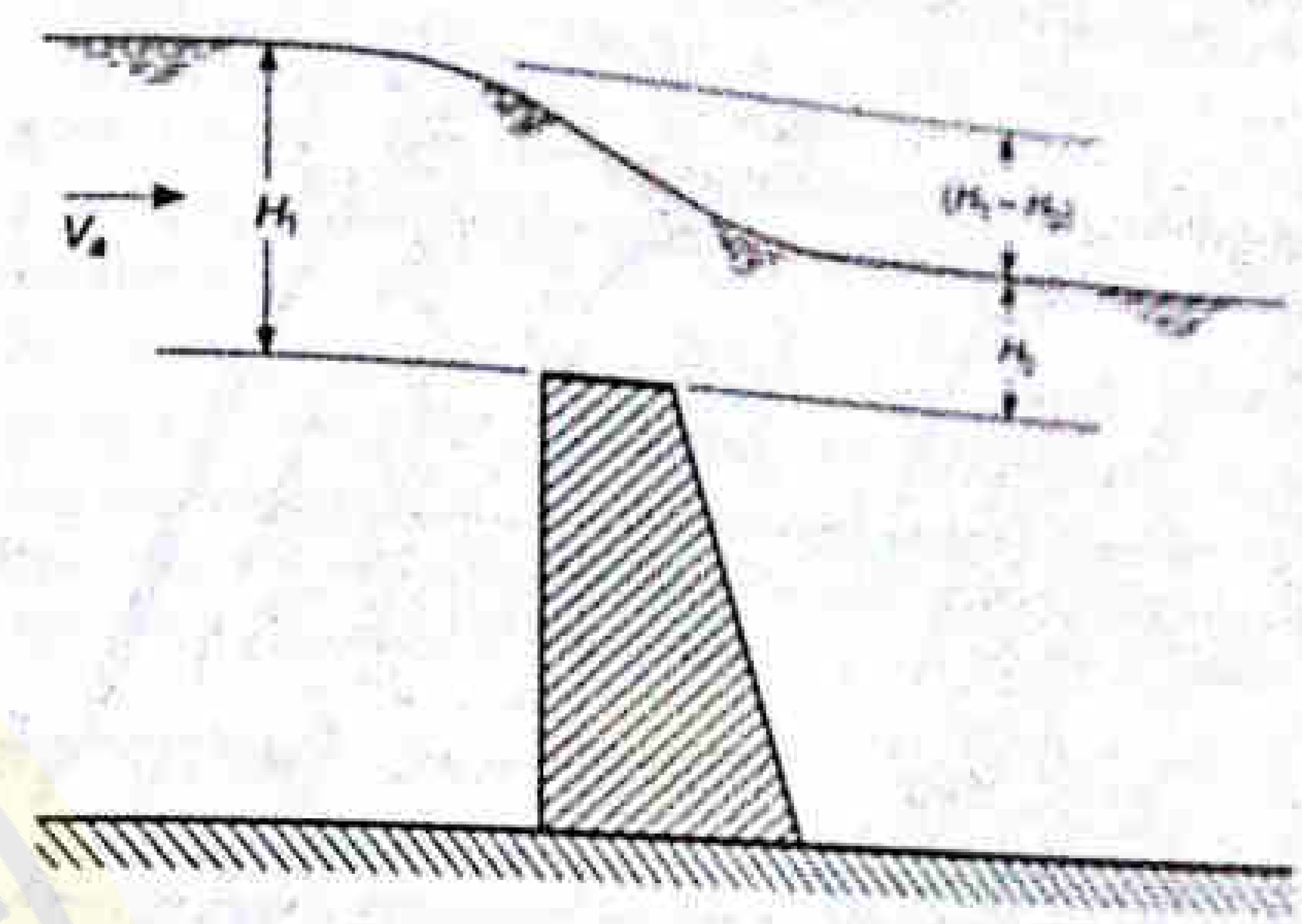


Figure 10.7 *Submerged weir*

If the velocity of approach V_a is to be considered then the values of Q_1 and Q_2 may be expressed as

$$Q_1 = \frac{2}{3} C_{d_1}\sqrt{2g}\, L\left[\left\{(H_1 - H_2) + \frac{V_a^2}{2g}\right\}^{3/2} - \left(\frac{V_a^2}{2g}\right)^{3/2}\right]$$

$$Q_2 = C_{d_2}(L \times H_2)\left[\sqrt{2g(H_1 - H_2) + V_a^2}\right]$$

A sharp crested weir is more susceptible to submergence than a broad crested weir. But even a sharp crested weir behaves as a free weir only, for a submergence ratio $\left[\dfrac{H_2}{H_1 + \left(\dfrac{V_a^2}{2g}\right)}\right]$ as high as 0.66.

The corresponding value of the submergence ratio upto which a broad crested weir behaves as a free weir is as high as 0.83 to 0.85. This is attributed to the fact that the flow conditions are such that the downstream water level is held away from the crest and hence it does not affect the flow conditions upstream. The limiting value of the submergence ratio upto which any submerged weir may behave as a free weir is known as its *modular limit*.

10.13 SPILLWAY AND SIPHON SPILLWAY

A spillway is a portion of a dam over which the excess water, which cannot be stored in the reservoir formed on the upstream of the dam, flows to the downstream side. In general the shape of the spillway profile is made to follow the profiles of a lower nappe of a well-ventilated sharp crested weir. In other words, the spillway is formed by filling the space between the sharp crested weir and the lower nappe with concrete or masonry as shown in Fig. 10.8 (a). Such a spillway is known as Ogee spillway (or Ogee weir). The main advantage of providing such a shape for the spillway is that the flowing sheet of water remains in contact with the surface of the spillway, thereby preventing the negative pressure being

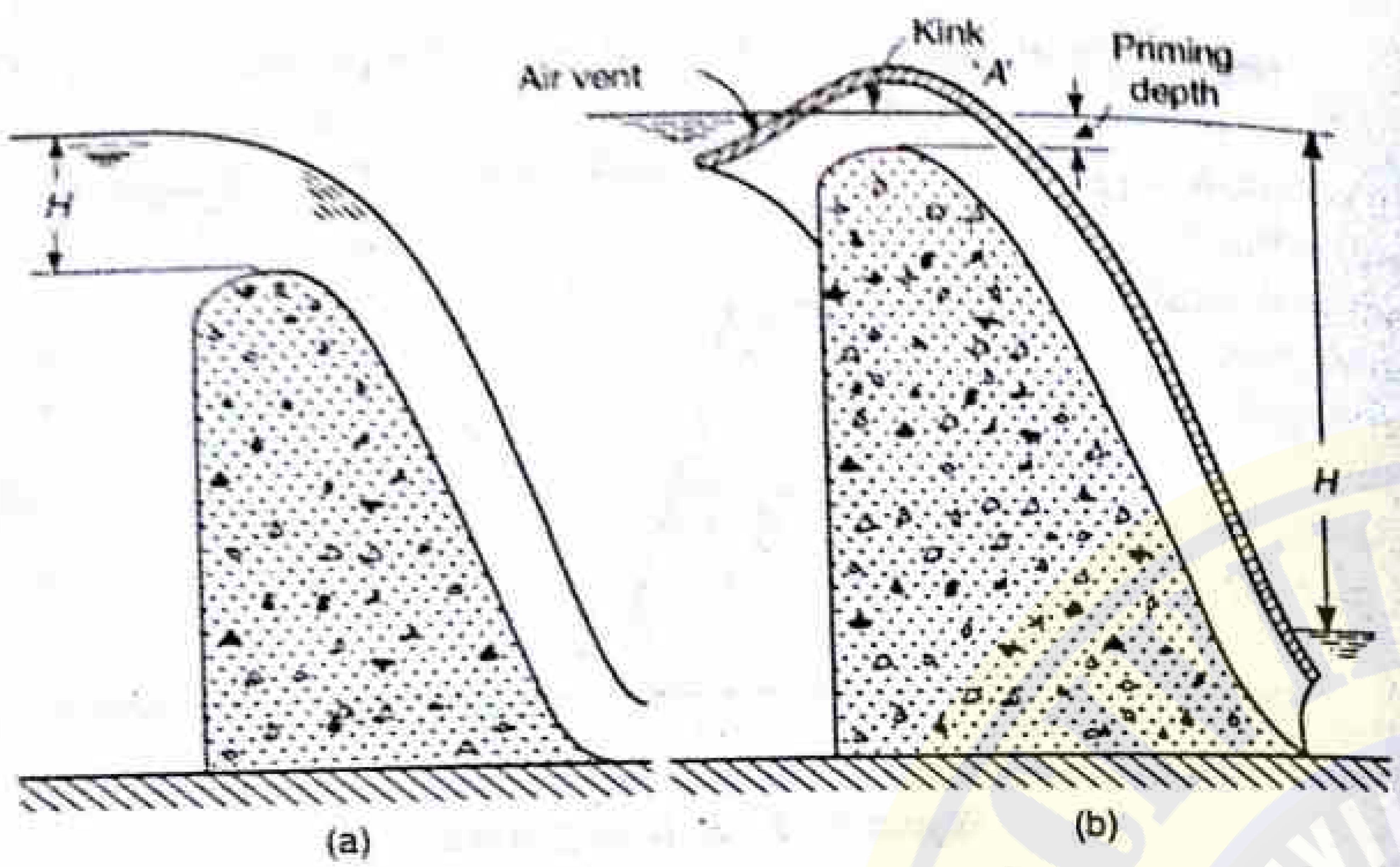


Figure 10.8 (*a*) *Ogee spillway*, (*b*) *Siphon spillway*

developed on the downstream side. However this condition will be fulfilled only as long as the head over crest of the spillway is equal to or less than the designed head. For the heads larger than the designed head the sheet of water may separate from the spillway, resulting in the development of the negative pressure on some portion of the spillway. As such, the actual shape of a spillway is usually determined by conducting the experiments on the models of the actual spillways. The discharge over an Ogee spillway may be expressed as

$$Q = CLH^{3/2} \quad ...(10.33)$$

where C is the coefficient of the spillway, L is the length of the spillway and H is the head above the crest of the spillway. The coefficient C of the spillway may be determined by calibration.

Siphon Spillway. A siphon spillway consists of an Ogee weir which is provided with an air-tight cover as shown in Fig. 10.8 (b), thus converting the discharge face of the spillway into a large rectangular sectioned pipe connecting the upstream and the downstream water surfaces. In the case of a siphon spillway the head H under which the water flows is equal to the difference between the water surfaces on the upstream and the downstream sides. As such as compared with an ordinary spillway the head in the case of a siphon spillway is more, on account of which the siphon spillway has a much greater discharge for a given length than an ordinary open spillway. The working of a siphon spillway is automatic which is as explained below.

When the water level in the upstream reservoir rises above the crest of the spillway, water begins to flow over the crest through the siphon. The jet of flowing water strikes the inside of the cover, the enclosing a small space A called kink, in the upper portion of the cover, in which air is trapped. As more and more water flows down, it sucks the trapped air from the kink, thus creating a partial vacuum in this portion which sucks up the water from the reservoir and completely fills the pipe. The siphonic action is thus started and the effective head causing the flow of water becomes equal to H, the difference between the water surfaces on the upstream and the downstream sides. When the level of

water in the reservoir reduces and becomes almost equal to the crest level of the spillway, then the siphonic action should stop, otherwise the water in the reservoir may be lowered to a level much below the crest of the spillway. For this purpose air vents are provided in the cover at a level slightly above the crest of the spillway so that when the water in the reservoir drops below this level the air vents get exposed and air enters the upper portion thereby stopping the siphonic action. By such an arrangement the working of a siphon spillway may be made automatic and hence often it is called *automatic siphon spillway*.

A siphon spillway has the following advantages over an ordinary open spillway

(i) The operating head is increased considerably due to which the discharge is increased.

(ii) The depth of water (called priming depth) required for the commencement of the siphonic action is only a few centimetres above the crest of the spillway. As such the crest of a siphon spillway can be raised, thereby allowing a greater amount of water to be stored in the reservoir.

10.14 PROPORTINAL WEIR OR SUTRO WEIR

In general for the weirs discussed earlier the discharge Q may be expressed as $Q \propto H^n$, where $n = \frac{3}{2}$ for a rectangular weir and $n = \frac{5}{2}$ for a triangular weir. However, it is possible to design a weir of such a shape for which $Q \propto H$ i.e., discharge Q varies linearly with the head H over the weir crest. Such a weir is called a proportional weir or Sutro weir in honour of its first inventor. Figure 10.9 shows a proportional weir.

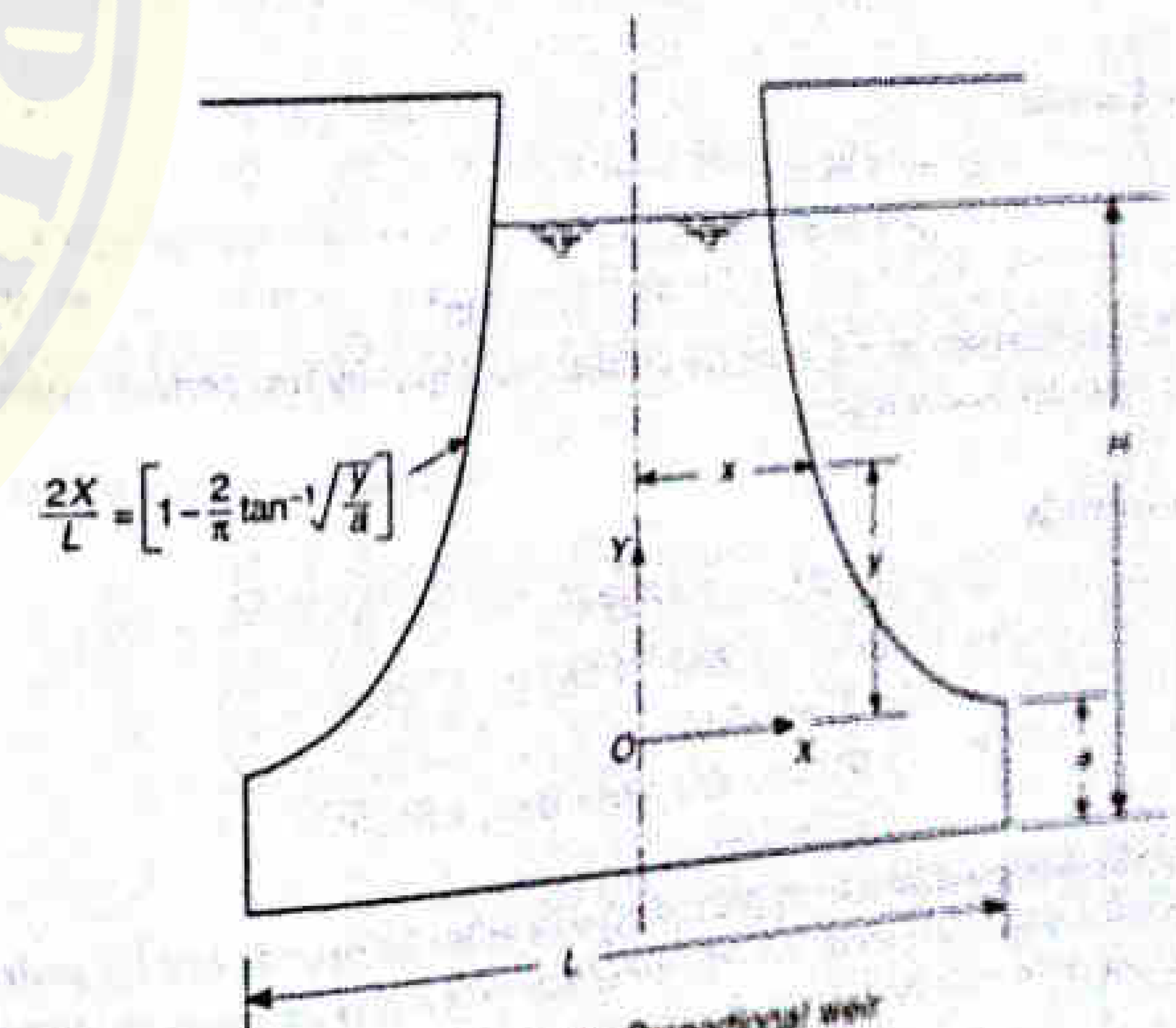


Figure 10.9 *Proportional weir*

Sutro analytically obtained the relationship for the shape of the proportional weir profile having $Q \propto H$, as

$$x \propto y^{-1/2} \quad ...(10.34)$$

However, it may be noted from the above relationship that as $y \to 0, x \to \infty$, which means that the width of the weir aperture becomes infinity at the crest. The infinity tending profile is however not practicable. In order to overcome this practical limitation Sutro modified the shape of the weir profile so that a finite width at the weir crest may be provided. The modified proportional weir profile as given by Sutro is shown in Fig. 10.9, which has its sides diverging downward in the form of hyperbolic curves having the equation

$$\frac{2x}{L} = \left[1-\frac{2}{\pi}\tan^{-1}\sqrt{\frac{y}{a}}\right] \quad ...(10.35)$$

where a and L are respectively the height and width of the small rectangular shaped aperture which forms the base of the weir. The discharge through this weir is given by

$$Q = k\left(H-\frac{a}{3}\right) \quad ...(10.36)$$

where $k = C_d L(2ga)^{1/2}$

The coefficient of discharge C_d for such weirs varies from 0.60 to 0.65. Proportional weir is very useful as a control device, especially in chemical dosing and sampling.

ILLUSTRATIVE EXAMPLES

Example 10.1. *Find the discharge over a suppressed rectangular weir 4 m long with a head over the crest as 0.35 m.*

Solution

Using Francis formula

$$Q = 1.84\, LH^{3/2}$$

$$L = 4 \text{ m and } H = 0.35 \text{ m}$$

$$Q = 1.84 \times 4 \times (0.35)^{3/2} = 1.524 \text{ m}^3/\text{s}.$$

Example 10.2. *A rectangular weir 6 m long is divided into 3 bays by two vertical posts each 0.3 m wide. Find the discharge when the head is 0.45 m.*

Solution

Using Francis formula

$$Q = 1.84(L-0.1nH)H^{3/2}$$

$$L = (6-2\times 0.30) = 5.4 \text{ m}$$

$$n = 6;\ H = 0.45 \text{ m}$$

$$\therefore \quad Q = 1.84\,[5.4-(0.1\times 6)\times 0.45]\times(0.45)^{3/2}$$

$$= 2.85 \text{ m}^3/\text{sec}.$$

Example 10.3. *The maximum flow through a rectangular flume 1.8 m wide and 1.2 m deep is 1.65 m³/sec. It is proposed to install a suppressed sharp crested rectangular weir across the flume to measure flow. Find the maximum height at which the weir crest can be placed in order that water may not overflow the sides of the flume. Assume $C_d = 0.6$.*

Solution

Neglecting the velocity of approach, the discharge over a rectangular weir is given by

$$Q = \frac{2}{3}C_d\sqrt{2g}\,LH^{3/2}$$

$$Q = 1.65 \text{ m}^3/\text{sec};\ L = 1.8 \text{ m}$$

$$1.65 = \frac{2}{3}\times 0.6\times\sqrt{2\times 9.81}\times 1.8\times H^{3/2}$$

$$H = 0.644 \text{ m}.$$

If Z is the height of weir crest above the bottom of the flume, then

$$Z + 0.644 = 1.2$$

$$\therefore \quad Z = 0.556 \text{ m}$$

If the velocity of approach is taken into account then

$$Q = \frac{2}{3}C_d\sqrt{2g}L\left[(H+h_a)^{3/2}-h_a^{3/2}\right]$$

$$V_a = \frac{1.65}{1.8\times 1.2} = 0.764 \text{ m/sec}$$

$$h_a = \frac{V_a^2}{2g} = \frac{(0.764)^2}{2\times 9.81} = 0.0298 \text{ m}$$

$$\text{Thus } 1.65 = \frac{2}{3}\times 0.6\times\sqrt{2\times 9.81}\times 1.8\times\left[(H+0.0298)^{3/2}-(0.0298)^{3/2}\right]$$

$$\text{or} \quad H = 0.619 \text{ m}$$

$$\therefore \quad Z = (1.2-0.169) = 0.581 \text{ m}.$$

Example 10.4. *A rectangular notch of crest width 0.4 m is used to measure the flow of water in a rectangular channel 0.6 m wide and 0.45 m deep. If the water level in the channel is 0.225 m above the weir crest, find the discharge in the channel. For the notch assume $C_d = 0.63$ and take velocity of approach into account.*

Solution

If the velocity of approach is neglected

$$Q = \frac{2}{3}C_d\sqrt{2g}(L-0.1n\,H)H^{3/2}$$

$$C_d = 0.63;\ L = 0.40$$

$$H = 0.225 \text{ m};\ n = 2$$

Thus

$$Q = \frac{2}{3}\times 0.63\times\sqrt{2\times 9.81}\ (0.40-0.2\times 0.225)\times(0.225)^{3/2}$$

$$= 0.0705 \text{ m}^3/\text{s}$$

For this discharge the velocity of approach

$$V_a = \frac{0.0705}{0.6\times 0.45}$$

$$= 0.261 \text{ m/sec}$$

Then

$$h_a = \frac{V_a^2}{2g} = \frac{(0.261)^2}{2\times 9.81} = 0.0035 \text{ m}$$

and

$$H_1 = (H + h_a) = (0.225 + 0.003\,5) = 0.2285 \text{ m}$$

Thus

$$Q = \frac{2}{3}\times 0.63\times\sqrt{2\times 9.81}\ (0.40 - 0.2\times 0.2285)\times[(0.2285)^{3/2} - (0.0035)^{3/2}$$

$$= 0.07186 \text{ m}^3/\text{s}$$

New velocity of approach

$$V_a = \frac{0.07186}{0.6\times 0.45} = 0.266 \text{ m/s}$$

$$h_a = \frac{V_a^2}{2g} = \frac{(0.266)^2}{2\times 9.81} = 0.036 \text{ m}$$

$$H_1 = (H + h_a) = (0.225 + 0.0036) = 0.2286 \text{ m}$$

Thus

$$Q = \frac{2}{3}\times 0.63\times\sqrt{2\times 9.81}\ (0.40 - 0.2\times 0.2286)\times[(0.2286)^{3/2} - (0.0036)^{3/2}]$$

$$= 0.0719 \text{ m}^3/\text{s}$$

The new value of discharge is quite close to the previous value and hence it may be considered as the correct value of the discharge.

Example 10.5. *Find the depth and top width of a V-notch capable of discharging a maximum of 0.7 m³/sec and such that the head shall be 75 mm for a discharge of 5.6 litres per second. Its C_d is the same as that of a similar (in material and sharpness of edges only) right-angled V-notch for which*

$$Q = 1.407^{5/2}$$

Solution

For a triangular notch

$$Q = \frac{8}{15}C_d\sqrt{2g}\ \tan\frac{\theta}{2}H^{5/2}$$

in which $\left(\frac{8}{15}C_d\sqrt{2g}\right) = 1.407$, as given.

Thus

$$Q = 1.407\tan\frac{\theta}{2}H^{5/2}$$

When

$$Q = 5.6\times 10^{-3} = 0.0056 \text{ m}^3/\text{s}$$

$$H = 75\times 10^{-3} = 0.075 \text{ m}$$

Thus by substitution, we get

$$0.0056 = 1.407(0.075)^{5/2}\tan\frac{\theta}{2}$$

or

$$\tan\frac{\theta}{2} = \frac{0.0056}{1.407\times(0.075)^{5/2}} = 2.584$$

Now if H is the required head when $Q = 0.7$ m³/s, then

$$0.7 = 1.407\times H^{5/2}\times 2.584$$

$$H = \left(\frac{0.7}{1.407\times 2.584}\right)^{2/5} = 0.517 \text{ m}$$

Top width

$$= 2H\tan\frac{\theta}{2}$$

$$= (2\times 0.517\times 2.5840) = 2.672 \text{ m}$$

Example 10.6. *The following observations of head and the corresponding discharge were made in connection with a weir 0.277 m wide.*

Head in mm	30	150	300	450	600	750	900	1050	1200
Discharge in litres / second	4.60	32.4	90.45	164.7	251.64	351.81	459.81	581.6	712.8

Assuming $Q = CLH^n$, *find C and n.*

Solution

$$Q = CLH^n$$

$$L = 0.277 \text{ m}$$

so

$$Q = 0.277\,CH^n$$

Taking log of both sides, we get

$$\log Q = \log(0.277\,C) + n\log H$$

The values of log Q and log H (taking H in metres and Q in m³/sec) for the given data are as tabulated below:

log H	$\bar{2}.477$	$\bar{1}.176$	$\bar{1}.477$	$\bar{1}.653$	$\bar{1}.778$	$\bar{1}.875$	$\bar{1}.954$	$\bar{0}.021$	$\bar{0}.079$
log Q	$\bar{3}.663$	$\bar{2}.511$	$\bar{2}.956$	$\bar{1}.217$	$\bar{1}.401$	$\bar{1}546$	$\bar{1}.663$	$\bar{1}.765$	$\bar{1}.853$

The plot of log Q v/s log H is as shown in the Fig. Ex. 10.6

Since for

$$H = 1$$

$$\log H = 0$$

$$\log Q = \log(0.277\,C)$$

From the graph, for

$$\log H = 0$$

$$\log Q = \bar{1}.70$$

Thus

$$\log(0.277\,C) = \bar{1}.70$$

$$\therefore\quad C = \frac{0.5012}{0.277} = 1.81$$

Further
$$n = \frac{\log Q - \log(0.277C)}{\log H}$$

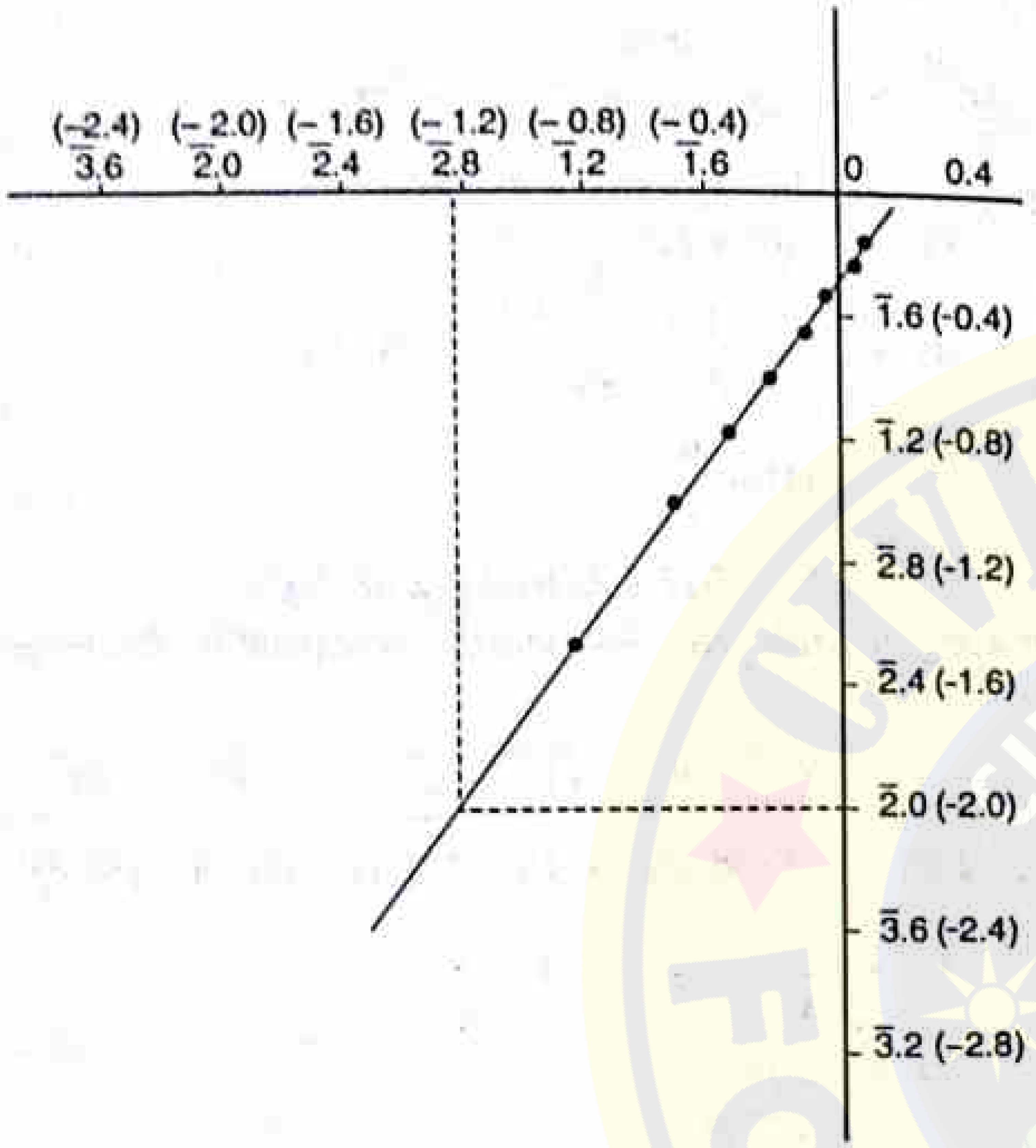


Figure Ex. 10.6

Choosing any point on the curve

$$\log H = \bar{2}.78$$
$$\log Q = \bar{2}.0$$
$$n = \frac{\bar{2}.0 - \bar{1}.70}{\bar{2}.78} = 1.39$$
$$\therefore \quad Q = 1.817H^{1.39}$$

Example 10.7. *A triangular notch is used to measure flow in a channel under a head of 0.2 m. If the discharge is to be measured within 3% accuracy, what is the maximum velocity of approach that can be neglected.*

Solution

Let Q be the discharge when the velocity of approach is neglected, and Q_1 be the discharge when the velocity of approach is taken into account.

Thus

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2} \quad \text{...(i)}$$

and
$$Q_1 = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}\left[(H+h_a)^{5/2} - h_a^{5/2}\right] \quad \text{...(ii)}$$

From Eq. (i) and (ii), we have

$$\frac{(Q_1 - Q)}{Q} = \frac{\left[\left(H + h_a^{3/2}\right) - h_a^{5/2}\right] - H^{5/2}}{H^{5/2}}$$

By substituting the given values, we get

$$0.03 = \frac{\left[(0.2+h_a)^{5/2} - h_a^{5/2}\right]}{(0.2)^{5/2}} - 1$$

or
$$1.03 = \frac{\left[(0.2+h_a)^{5/2} - h_a^{5/2}\right]}{(0.2)^{5/2}}$$

Solving for h_a by trial and error, we get

$$h_a = 2.38 \times 10^{-3}\text{ m}$$

or
$$\frac{V_a^2}{2g} = 2.38 \times 10^{-3}$$

$$\therefore \quad V_a = 0.216\text{ m/s}$$

Example 10.8. *A stream approaching a water fall having a fall of 36 m is gaged by a weir. The measured head over the weir is 0.2575 m and the length o the weir is 3 m. The velocity of approach is 1.2 m/s. Determine the power available at the water fall. Use Bazin's formula with $\alpha = 1.5$ for flow over the weir.*

Solution

According to Bazin's formula

$$Q = m_1\sqrt{2g}LH_1^{3/2}$$

$$m_1 = \left(0.405 + \frac{0.003}{H_1}\right)$$

$$H_1 = \left(H + \alpha\frac{V_a^2}{2g}\right)$$

$$= \left[0.275 + 1.5 \times \frac{(1.2)^2}{2\times 9.81}\right]$$

$$= 0.385\text{ m}$$

$$m_1 = \left(0.405 + \frac{0.003}{0.385}\right) = 0.413$$

Thus $$Q = 0.413\times\sqrt{2\times9.81}\times3\times(0.385)^{3/2}$$
$$= 1.311 \text{ m}^3/\text{sec}$$

Power available at the fall

$$= \frac{wQH}{1000}\text{kW}$$

$$= \frac{9810\times1.311\times36}{1000} = 463 \text{ kW}$$

Example 10.9. *Rain falls over a catchment area of 26 sq km at the rate of 1 mm per hour. The rain water flows over a weir with a free length of 12 m constructed in 8 bays each 1.5 m long. Using Francis formula, find the head over weir crest.*

Solution

The discharge over the weir is

$$Q = \frac{26\times10^6\times1}{1000\times60\times60} = 7.22 \text{ m}^3/\text{sec.}$$

Using Francis formula

$$Q = 1.84(L-0.1nH)H^{3/2}$$
$$L = 12 \text{ m and } n = (2\times8) = 16$$

Thus $$7.22 = 1.84(12-1.6H)H^{3/2}$$

Solving by trial and error

$$H = 0.5 \text{ m.}$$

Example 10.10. *Water flows through a rectangular channel 1 m wide and 0.5 m deep, and then over a sharp crested Cipolletti weir of crest length 0.6 m. If the water level in the channel is 0.225 m above the weir crest, calculate the discharge over the weir. Take $C_d = 0.6$ and make correction for velocity of approach.*

Solution

If the velocity of approach is neglected then for a Cipolletti weir

$$Q = \frac{2}{3}C_d\sqrt{2g}LH^{3/2}$$
$$C_d = 0.6, L = 0.60 \text{ m}, H = 0.225 \text{ m}$$

Then $$Q = \frac{2}{3}\times0.6\times\sqrt{2\times9.81}\times0.6\times(0.225)^{3/2}$$
$$= 0.1135 \text{ m}^3/\text{s}$$

Velocity of approach

$$V_a = \frac{0.1135}{1\times0.5} = 0.227 \text{ m/s}$$

Thus $$h_a = \frac{V_a^2}{2g} = \frac{(0.227)^2}{2\times9.81} = 0.0026 \text{ m}$$

and $$H_1 = (H+h_a) = (0.225+0.0026) = 0.2276 \text{ m}$$

For velocity of approach being taken in account

$$Q = \frac{2}{3}C_d\sqrt{2g}\;L\left[H_1^{3/2}-h_a^{3/2}\right]$$

$$= \frac{2}{3}\times0.6\times\sqrt{2\times9.81}\times0.6\times\left[(0.2276)^{3/2}-(0.0026)^{3/2}\right]$$

$$= 0.1153 \text{ m}^3/\text{s.}$$

Since the new value of the discharge is quite close to the previous value the correct discharge over the Cipolletti weir may be taken as 0.1153 m³/s.

Example 10.11. *A rectangular channel 6 m wide carries 2800 litres per second at a depth of 0.9 m. What height of a broad crested rectangular weir must be installed to double the depth? Assume a weir coefficient of 0.86.*

Solution

From Eq. 10.31 for a broad crested weir

$$Q = 1.70\,C_d LH^{3/2}$$
$$Q = 280\times10^{-3} = 2.8 \text{ m}^3/\text{s}$$
$$C_d = 0.86; L = 6 \text{ m}$$

Thus by substitution, we get

$$2.8 = 1.70\times0.86\times6\times H^{3/2}$$

$\therefore$ $$H = 0.467 \text{ m}$$

The depth of flow required to be developed in the channel

$$Z = (2\times0.90) = 1.80 \text{ m}$$

$\therefore$ Height of the broad crested weir

$$= (1.80-0.467) = 1.333 \text{ m}$$

If the velocity of approach is taken into account then using Eq. 10.32, we get

$$H_1 = (H+h_a) = 0.467 \text{ m}$$

The velocity of approach

$$V_a = \frac{2.8}{6\times1.8} = 0.26 \text{ m/s}$$

and $$h_a = \frac{V_a^2}{2g} = \frac{(0.26)^2}{2\times9.81} = 0.0034 \text{ m}$$

Thus $$H = (0.467-0.0034) = 0.4636 \text{ m}$$

$\therefore$ $$Z = (1.80-0.4636) = 1.3364 \text{ m}$$

Example 10.12. *A Cipolletti weir, with coefficient of discharge $C_d = 0.625$ and crest length $L = 30$ m is provided in the side of a reservoir. Area of water surface in the reservoir A, in sq. metres is given by the expression*

$$A = 9.3\times10^3(15+\alpha h^2)$$

where α is a constant whose value may be taken as 0.5 and h is the head of water over the weir crest in metres. Calculate the time taken to lower the water surface in the reservoir from a head of 1.2 m above the weir crest to a head of 0.3 m.

Solution

As described in Section 10.9

$$-Adh = Qdt$$

$$Q = \frac{2}{3}C_d\sqrt{2g}\ LH^{3/2}$$

and $$A = (9.3 \times 10^3)\ (15 + 0.5h^2)$$

On substitution,

$$-(9.3 \times 10^3)\ (15 + 0.5h^2)dh = \left(\frac{2}{3}\times 0.625\times\sqrt{2\times 9.81}\times 30\times h^{3/2}\right)dt$$

or $$dt = \frac{-(9.3\times 10^3)\ \ (15+0.5h^2)dh}{\frac{2}{3}\times 0.625\times\sqrt{2\times 9.81}\times 30\times h^{3/2}}$$

By integrating both sides, we get

$$t = \frac{-(9.3\times 10^3)}{\frac{2}{3}\times 0.625\times\sqrt{2\times 9.81}\times 30}\int_{1.2}^{0.3}\frac{(15+0.5h^2)}{h^{3/2}}dh$$

$$= -168\left[-\frac{30}{h^{1/2}}+\frac{h^{3/2}}{3}\right]_{1.2}^{0.3}$$

$$= 4665\ \text{s} = 1.296\ \text{h}$$

Example 10.13. *A reservoir 4.65×10^4 m² in area is to be controlled by a rectangular weir with its crest level at El.30. It is intended to provide such a length of weir that will lower the water level from El. 31.2 to El. 30.6 in half an hour time. Determine the length of weir. The discharge over the weir is given by the formula $Q = 1.9\ LH^{3/2}$ where Q is discharged in cumec, L is crest length in metres and H is head over the weir in metres.*

Solution

From Eq. 10.25

$$t = \frac{2A}{\frac{2}{3}C_d\sqrt{2g}\ \ L\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right]}$$

In this case $$\left(\frac{2}{3}C_d\sqrt{2g}\right) = 1.9$$

Thus $$t = \frac{2A}{1.9L}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right]$$

$$t = \left(\frac{1}{2}\times 60\times 60\right) = 1800\ \text{s}$$

$$A = 4.65 \times 10^4\ \text{m}^2$$

$$H_1 = (31.2 - 30.0) = 1.2\ \text{m}$$

$$H_2 = (30.6 - 30.0) = 0.6\ \text{m}$$

Thus by substitution, we get

$$1800 = \frac{2\times 4.65\times 10^4}{1.9\times L}\left[\frac{1}{(0.6)^{1/2}}-\frac{1}{(1.2)^{1/2}}\right]$$

$$\therefore \quad L = 10.28\ \text{m}$$

Example 10.14. *A right-angled triangular notch is provided in the vertical side of a tank having plan area of 0.93 m² uniform at al levels. When the head over the notch is 75 mm, it is found that the water surface in the tank is falling down at a rate of 2.54 mm per second. Calculate the coefficient of discharge of the notch.*

Solution

As described in Section 10.9

$$-Adh = Qdt$$

$$A = 0.93\ \text{m}^2;\ Q = \frac{8}{15}C_d\sqrt{2g}\ \ H^{5/2}$$

When $$H = 0.075\ \text{m},\ -\left(\frac{dh}{dt}\right) = 2.54\times 10^{-3}\ \text{m/s}$$

Thus by substitution

$$0.93\times 2.54\times 10^{-3} = \frac{8}{15}C_d\times\sqrt{2\times 9.81}\times(0.075)^{5/2}$$

$$C_d = 0.649.$$

Example 10.15. *A submerged sharp crested weir 0.81 m high stands clear across a channel having vertical sides and width of 3.15 m. The depth of water in the channel of approach 1.26 m, and 10.5 m downstream from the weir the depth of water 0.93 m. Determine the discharge in litres per minute. Assume $C_{d_1} = 0.58$ and $C_{d_2} = 0.80$.*

Solution

Discharge over a submerged weir is given by

$$Q = Q_1 + Q_2$$

where neglecting the velocity of approach

$$Q_1 = \frac{2}{3}C_{d_1}\sqrt{2g}\ L\ (H_1 - H_2)^{3/2}$$

and $$Q_2 = C_{d_2}\times(L\times H_2)\times\sqrt{2g(H_1-H_2)}$$

and $$L = 3.15\ \text{m},\ H_1 = (1.26 - 0.81) = 0.45\ \text{m}$$

$$H_2 = (0.93 - 0.81) = 0.12\ \text{m}$$

Thus
$$Q_1 = \frac{2}{3}\times 0.58\times\sqrt{2\times 9.81}\times 3.15(0.45-0.12)^{3/2}$$
$$= 1.023 \text{ m}^3/\text{s}$$
$$Q_2 = 0.80\times(3.15\times 0.12)\sqrt{2\times 9.81(0.45-0.12)}$$
$$= 0.769 \text{ m}^3/\text{s}$$
$\therefore$
$$Q = (1.023 + 0.769) \text{ m}^3/\text{s}$$
$$= 1.792 \text{ m}^3/\text{s}$$
$$= 1.792 \times 10^3 \, l/\text{s}$$
$$= 1.792 \times 10^3 \times 60 \, l/\text{min}$$
$$= 10.752 \times 10^4 \, l/\text{min}$$

Note. If velocity of approach is to be considered, necessary correction may be applied.

Example 10.16. *A tank with vertical sides and a horizontal cross-sectional area 1.86 m² is provided with a notch cut at the top of one of the sides. Water flowing into the tank at constant rate was discharged over the notch, the head over the crest of which was 0.225 m. The supply of water was suddenly stopped and it was observed that the head over the notch started to fall at the rate of 5.33 mm/s. When the head had fallen to 0.1125 m it was observed that the head over the notch was falling at the rate of 1.905 mm/s. Estimate the rate of inflow to the tank when there is a steady head of 0.125 m over the notch. Assume for the notch $Q = kH^n$.*

Solution

As mentioned in Section 10.9
$$-Adh = Qdt$$
or
$$-\frac{dh}{dt} = \frac{Q}{A} = \frac{KH^n}{A}$$
when
$$H = 0.225 \text{ m}\,; -\frac{dH}{dt} = 5.33\times 10^{-3} \text{ m/s}$$
and when
$$H = 0.1125 \text{ m}\,; -\frac{dH}{dt} = 1.905\times 10^{-3} \text{ m/s}$$

Thus by substitution, we get
$$5.33 \times 10^{-3} = \frac{K\times(0.225)^n}{1.86} \quad (1)$$
and
$$1.905 \times 10^{-3} = \frac{K\times(0.1125)^n}{1.86} \quad (2)$$

Dividing Eq. 1 by Eq. 2, we have
$$\frac{5.33}{1.905} = \left(\frac{0.225}{0.1125}\right)^n$$
$\therefore$
$$n = 1.484$$

Then
$$K = \frac{5.33\times 10^{-3}\times 1.86}{(0.225)^{1.484}} = 0.0907$$
Therefore
$$Q = 0.0907H^{1.484}$$

For steady head $H = 0.125$ m, the rate of inflow to the tank will be equal to discharge Q over the notch. Thus rate of inflow to the tank is
$$Q = 0.0907\,(0.125)^{1.484} = 0.004 \text{ m}^3/\text{s}$$

Example 10.17. *A river 30 m wide and 3 m deep has a mean velocity of 1.2 m per second. Find the height of an anicut (weir) to raise the water level by 1 m.*

Solution

Width of river = 30 m

Depth of flow = 3 m

$\therefore$ Area of flow section
$$= (30 \times 3) = 90 \text{ m}^2$$
Mean velocity of flow
$$= 1.2 \text{ m/sec}$$
$\therefore$ Discharge
$$Q = (90 \times 1.2) = 108 \text{ m}^3/\text{sec.}$$

Since the anicut is constructed to raise the water level by 1 m, the depth of flow on the upstream of the anicut becomes
$$(3+1) = 4 \text{ m}$$
$\therefore$ Velocity of approach
$$V_a = \frac{108}{30\times 4} = 0.9 \text{ m/s}$$
and
$$h_a = \frac{V_a^2}{2g} = \frac{(0.9)^2}{2\times 9.81} = 0.0413 \text{ m}$$

Assuming that the anicut is discharging free, then
$$Q = \frac{2}{3}C_{d_1}L\sqrt{2g}\left[(H+h_a)^{3/2} - h_a^{3/2}\right]$$
Assume
$$C_{d_1} = 0.58.$$
Thus by substitution, we get
$$108 = \frac{2}{3}\times 0.58\times 30\times\sqrt{2\times 9.81}\,[(H+0.0413)^{3/2} - (0.0413)^{3/2}]$$
or
$$H = 1.604 \text{ m}$$

The height of the anicut is then
$$Z = (4 - 1.604) = 2.396 \text{ m}$$

Since the depth of water in the channel on the downstream of the anicut will also be 3 m, the anicut will be drowned.

For a drowned anicut the discharge is given by

$$Q = \frac{2}{3}C_{d_1}L\sqrt{2g}\,[(H_1 - H_2 + h_a)^{3/2} - h_a^{3/2}] + C_{d_2} \times (L \times H_2)\left[\sqrt{2g(H_1-H_2)+V_a^2}\right]$$

$$(H_1 - H_2) = 1\text{ m}$$

Assume $C_{d_1} = 0.58$; and $C_{d_2} = 0.80$

Thus by substitution, we get

$$108 = \frac{2}{3}\times 0.58\times 30\times\sqrt{2\times 9.81}\left[(1+0.0413)^{3/2} - (0.0413)^{3/2}\right]$$
$$+0.80\times(30\times H_2)\left[\sqrt{(2\times 9.81\times 1)+(0.9)^2}\right]$$

or

$$108 = 54.17 + 108.5\,H_2$$

$\therefore$

$$H_2 = \frac{108-54.17}{108.5} = 0.496\text{ m}$$

Therefore the height of the anicut

$$= (3 - 0.496) = 2.504\text{ m}$$

Example 10.18. *A spillway 45 m long having discharge coefficient 1.8 permits a maximum discharge of 90 m^3/s from a storage reservoir. It is proposed to replace the spillway by a siphon spillway of section 0.75 m × 1.5 m with operating head 8 m and discharge coefficient 0.64. Find the number of siphons required and the amount of extra water stored, if the siphons have a priming depth of 0.15 m; the average surface area of the reservoir being $5 \times 10^5\ m^2$.*

Solution

For each siphon of the spillway operating head

$$H = 8\text{ m}$$
$$a = (0.75 \times 1.5) = 1.125\text{ m}^2$$

$\therefore$ Discharge through each siphon

$$= C_d a\sqrt{2gH}$$
$$= 0.64 \times 1.125 \times \sqrt{2\times 9.81\times 8}$$
$$= 9.02\text{ m}^3/\text{s}$$

Maximum discharge $= 90\text{ m}^3/\text{s}$

No. of siphons $= \dfrac{90}{9.02} = 10$

Discharge through open spillway is

$$Q = \frac{2}{3}C_d\sqrt{2g}\,LH^{3/2}$$

$$Q = 90\text{ m}^3/\text{s};\ \left(\frac{2}{3}C_d\sqrt{2g}\right) = 1.8$$
$$L = 45\text{ m}$$

Thus

$$90 = 1.8 \times 45 \times H^{3/2}$$

$\therefore$

$$H = \left(\frac{90}{1.8\times 45}\right)^{2/3} = 1.073\text{ m}$$

Priming depth for siphons

$$= 0.15$$

$\therefore$ Height of crest raised when siphons are installed

$$= (1.073 - 0.15) = 0.923\text{ m}$$

Area of reservoir $= 5 \times 10^5\text{ m}^2$

$\therefore$ Volume of extra water stored

$$= (5 \times 10^5 \times 0.923) = 46.15 \times 10^4\text{ m}^3$$

SUMMARY OF MAIN POINTS

1. A notch may be defined as an opening provided in the side of a tank such that the liquid surface in the tank is below the top edge of the opening. Notches made of metallic plates are also provided in narrow channels in order to measure the rate of flow of liquid. As such in general notches are used for measuring the rate of flow of liquid from a tank or in a channel.

   According to the shape of the opening the notches are classified as rectangular notch, triangular notch, (or *V*-notch), trapezoidal notch and stepped notch. According to the effect of the sides on the nappe (or the sheet of water) emerging from a notch, the notches are classified as 'notch with end contraction' and 'notch without end contraction' or 'suppressed notch'. Notches provided in the sides of tanks are essentially the notches with end contraction. Further in a channel if the crest length (length of the bottom edge) of the notch is less than the width of the channel then it is also a notch with end contraction. However, if the crest length of the notch is equal to the width of the channel then it is a notch without end contraction.

2. A weir is a concrete or masonry structure built across a river (or stream) in order to raise the level of water on the upstream side and to allow the excess water to flow over its entire length to the downstream side. Weirs may also be used for measuring the rate of flow of water in rivers or streams.

   According to the shape of the opening, the weirs are classified as rectangular and trapezoidal weirs. A particular type of trapezoidal weir is known as 'Cipolletti weir'. According to the shape of the crest, the weirs are classified as thin-plate or sharp-edged weir, narrow-crested weir, broad-crested weir and Ogee-shaped weir. According to the effect of the sides on the issuing nappe as wier with end contraction and wier without end contraction. According to the discharge conditions weirs are classified as freely discharging weir and submerged (or drowned) weir.

3. Discharge through a rectangular notch or weir without end contractions is given by:

$$Q = \frac{2}{3}C_d\sqrt{2g}\,LH^{3/2}$$

where

L = lenght of the crest of the notch or weir ;

H = height of the water surface or head of water above the crest of the notch or weir;

C_d = coefficient of discharge for the notch or weir ; and

g = acceleration due to gravity.

4. The velocity with which water approaches a notch or weir is called velocity of approach denoted by V_a and is given by

$$V_a = \frac{\text{Discharge through the notch or weir}}{\text{Cross-sectional area of channel}}$$

The head due to velocity of approach h_a is given by

$$h_a = \frac{V_a^2}{2g}$$

5. Discharge through a rectangular notch or weir, when head due to velocity of approach is considered, is given by

$$Q = \frac{2}{3}C_d\sqrt{2g}L\left[(H+h_a)^{3/2}-(h_a)^{3/2}\right]$$

6. For a rectangular notch or weir with end contractions discharge is given by :

When velocity of approach is not considered

$$Q = \frac{2}{3}C_d\sqrt{2g}(L-0.1nH)H^{3/2}$$

When velocity of approach is considered

$$Q = \frac{2}{3}C_d\sqrt{2g}(L-.1nH_1)\left(H_1^{3/2}-h_a^{3/2}\right)$$

where

n = number of end contractions ; and

$$H_1 = (H+h_a)=\left(H+\frac{V_a^2}{2g}\right)$$

7. According to Francis formula discharge through a rectangular weir is given by : when the velocity of apporach is not considered

$$Q = 1.84\,(L-0.1nH)H^{3/2}$$

When the velocity of approach is considered

$$Q = 1.84\,(L-0.1nH_1)(H_1^{3/2}-h_a^{3/2})$$

When end contractions are suppressed and velocity of approach is not considered

$$Q = 1.84\,LH^{3/2}$$

When end contractions are suppressed but velocity of approach is considered

$$Q = 1.84\,L(H_1^{3/2}-h_a^{3/2})$$

8. According to Bazin's formula discharge through a rectangular weir is given by :

When velocity of approach is not taken into account

$$Q = m\sqrt{2g}\,LH^{3/2}$$

where

$$m = \left(0.405+\frac{0.003}{H}\right)$$

When velocity of approach is taken into account

$$Q = m_1\sqrt{2g}LH_1^{3/2}$$

where

$$m_1 = \left(0.405+\frac{0.003}{H}\right);$$

$$H_1 = \left(H+\alpha\frac{V_a^2}{2g}\right);\text{ and}$$

α = a constant with a mean value equal to 1.6

9. Discharge through a triangular notch or weir is given by:

When velocity of approach is not taken into account

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2}$$

When velocity of approach is not taken into account

$$Q=\frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}\left[(H+h_a)^{5/2}-(h_a)^{5/2}\right]$$

where

θ = vertex angle of the notch or weir

10. Discharge through a trapezoidal notch or weir is given by

$$Q=\left[\frac{2}{3}C_{d_1}\sqrt{2g}LH^{3/2}+\frac{8}{15}C_{d_2}\sqrt{2g}\tan\frac{\theta}{2}H^{5/2}\right]$$

where

C_{d_1} = coefficient of discharge for the rectangular portion of the notch or weir ;

C_{d_2} = coefficient of discharge for the triangular portion of the notch or weir ;

L = crest length of the notch or weir; and

$\theta/2$ = the angle of inclination of the sides of the notch or weir with the vertical.

If C_d is the coefficeient of discharge for whole of the trapezoidal notch or weir then the discharge is given by

$$Q = C_d\sqrt{2g}H^{3/2}\left[\frac{2}{3}L+\frac{8}{15}H\tan\frac{\theta}{2}\right]$$

11. A Cipolletti weir or notch is a trapezoidal weir or notch having inclination of slopping sides as 1 horizontal to 4 vertical, $\left(\text{i.e. }\frac{\theta}{2}=14°\right)$. The discharge through a Cipolletti weir is given by :

When velocity of apporach is not taken into account

$$Q = \frac{2}{3}C_d\sqrt{2g}\,L\,H^{3/2}$$

When velocity of approach is not taken into account

$$Q = \frac{2}{3}C_d\sqrt{2g}\,L\left[(H+h_a)^{3/2}-(h_a)^{3/2}\right]$$

12. The error in discharge Q due to the error in the measurement or triangular of head H over a rectangular or triangular notch or weir is given by

$$\frac{dQ}{Q}=\frac{3}{2}\frac{dH}{H}\text{ .... For a rectangular notch or weir}$$

$$\frac{dQ}{Q}=\frac{5}{2}\frac{dH}{H}\text{ .... For a triangular notch or weir}$$

13. Time required to empty a reservoir or tank by a rectangular or a triangular weir is given by

$$t=\frac{3A}{C_d\sqrt{2g}L}\left[\frac{1}{\sqrt{H_2}}-\frac{1}{\sqrt{H_1}}\right]\text{ ....By a rectangular weir}$$

$$t=\frac{5A}{4C_d\sqrt{2g}\tan\frac{\theta}{2}}\left[\frac{1}{(H_2)^{3/2}}-\frac{1}{(H_1)^{3/2}}\right]$$

....By a triangular weir

where

A = cross-sectional area of the reservoir or tank ;

H_1 = initial height of the liquid surface above the crest or apex of the weir ; and

H_2 = final height of the liquid surface above the crest or apex of the weir.

14. Discharge over a broad crested weir is given by

$$Q = C_d Lh\sqrt{2g(H-h)}$$

where

H = height of water surface above the crest of the weir on the upstream side ;

h = height of water surface above the crest of the weir at the middle of the weir ; and

L = length of the crest of the weir

The condition for maximum discharge over a broad crested weir is $h=\frac{2}{3}H$, and the maximum discharge is given by

$$Q = 1.70C_d LH^{3/2}$$

15. Discharge through a submerged or drowned weir is given by

$$Q = Q_1+Q_2$$

where Q_1 and Q_2 are the discharges through the free and the drowned portions respectively.

The values of Q_1 and Q_2 are given by :

When velocity of approach is not taken into account

$$Q_1 = \frac{2}{3}C_{d_1}\sqrt{2g}\,L\,(H_1-H_2)^{3/2}$$

$$Q_2 = C_{d_2}LH_2\sqrt{2g(H_1-H_2)}$$

where H_1 and H_2 are respectively the heights of water surface above the crest (or heads) on the upstream and the downstream of the weir ; and C_{d_1} and C_{d_2} are the coefficient of discharge for the free and the drowned portion respectively.

When velocity of approach is taken into account

$$Q_1=\frac{2}{3}C_{d_1}\sqrt{2g}L\left[(H_1-H_2+h_a)^{3/2}-h_a^{3/2}\right]$$

$$Q_2=C_{d_2}(L\times H_2)\left[\sqrt{2g(H_1-H_2)+V_a^2}\right]$$

16. Discharge over an Ogee spillway is given by

$$Q = CLH^{3/2}$$

where C is the coefficient of the spillway which may be determined by calibration.

17. In the case of siphon spillway the head H under which the water flows is equal to the difference between the water surfaces on the upstream and the downstream sides.

18. The sides of a proportional weir (or Sutro weir) diverge downward in the form of hyperbolic curves having the equation

$$\frac{2x}{L} = \left[1 - \frac{2}{\pi}\tan^{-1}\sqrt{\frac{y}{a}}\right]$$

where a and L are respectively the height and width of the small rectangular shaped aperture which forms the base of the weir. Discharge through this weir is given by

$$Q = k\left(H - \frac{a}{3}\right)$$

where

$$K = C_d L (2ga)^{1/2}$$

PROBLEMS

10.1 What is a weir? How are the weirs classified? What is the difference between a sharp crested and a broad crested weir?

10.2 What is a notch? How are the notches classified?

10.3 Explain why ventilation of suppressed rectangular weir is necessary.

10.4 What is a Cipolletti weir? Show that its side slopes of 1 horizontal to 4 vertical are provided for a definite purpose.

10.5 What is suppressed weir and a weir with end contractions?

10.6 Define velocity of approach. How can you account for it while computing the discharge over weirs?

10.7 What is the difference between open spillway and siphon spillway? What are the advantages of a siphon spillway?

10.8 Discuss the various empirical formulae for discharge over weirs.

10.9 A rectangular weir is 2 m long and has a head of 0.675 m. Find the discharge taking into account two end contractions. [*Ans.* 1.90 m^3/s]

10.10 Find the discharge through a triangular notch under a constant head of 0.25 m if the angle of the notch is 120°. Take $C_d = 0.62$.
[*Ans.* 0.079 m^3/s]

10.11 A rectangular notch has a crest length 1 m; the head over the notch is 0.2 m and the height of the sill above the bed level is 0.15 m. If the width of the channel is 1.2 m, calculate the discharge in cumec, taking into account the velocity of approach. [*Ans.* 0.166 cumec]

10.12 Determine the discharge over a sharp crested weir 4.5 m long with no end contractions, the measured head over the crest being 0.45 m. The width of the approach channel is 7.5 m and its depth below the crest of the weir is 1 m.
[*Ans.* 2.521 m^3/s]

10.13 During a test in a laboratory the water which has passed through a venturi meter flows over a right angled V-notch, the head at the V-notch being registered. The larger diameter of the venturi meter is 0.25 m and the diameter of the throat is 0.1 m. When a steady head over the V-notch of 0.181 m is maintained, the difference of pressure head at the venturi meter is found to be 0.322 5 m of water. Determine the coefficient of venturi meter on the assumption that the V-notch results are correct, the coefficient being 0.6. [*Ans.* 0.987]

10.14 In an experiment on a 90° V-notch the flow is collected in a vertical cylindrical tank 0.9 m diameter. It is found that the depth of water in the tank increases by 0.65 m in 16.8 s when the head over the notch is 0.2 m. Determine the coefficient of discharge of the notch. [*Ans.* 0.582]

10.15 A channel is conveying 0.6 m^3/s of water. Assuming that an error of 1.5 mm may be made in measuring the head, determine the percentage error resulting (a) from the use of a right angled triangular weir $C_d = 0.6$; (b) from the use of a suppressed rectangular weir 0.6 m long. [**Hint:** (a) Triangular notch:

$$Q = 0.6 = \frac{8}{15} C_d \sqrt{2g} H^{5/2}$$

$$\therefore \quad H = 0.709 \text{ m}$$

$$\frac{dQ}{Q} = \frac{5}{2}\frac{dH}{H}$$

$$\therefore \text{ Percentage error in } Q = \frac{5}{2}\frac{1.5 \times 10^{-3}}{0.709} \times 100 = 0.53\%$$

(b) Rectangular notch:
Percentage error in $Q = 0.33\%$]

10.16 A stream approaching a water fall having a fall of 20 m is gaged by a weir. The measured head over the weir is 0.32 m and the length of the weir is 3 m. The velocity of approach V_a is 0.2 m/s and the head due to this may be supposed to be increased by 1.6 ($V_a^2/2g$). Determine the power available from the fall assuming that 60% of the energy can be used.
[*Ans.* 119.13 kW(162 h.p.)]

10.17 A rectangular notch is to be made to discharge 1.82×10^4 litres per minute with a head over the sill equal to 0.5 of the width of the notch. Neglecting the velocity of approach and allowing for two end contractions, determine the width of the notch and the head of water above the sill. [*Ans.* 0.768 m; 0.384 m]

10.18 A reservoir has an area of 8.5×10^4 m^2 and is provided with a weir 4.5 m long. Find how long will it take for the level at the sill to fall from 0.6 m to 0.3 m. [*Ans.* 3.112 h]

10.19 A maximum discharge of 56 m^3/s from a storage reservoir flows over an open spillway 36 m long. Find the head under which the water flows. If the open spillway is now replaced by a siphon type spillway having a section 0.6 m × 1.2 m with operating head of 6 m, find the number of siphons required. Take $C_d = 0.6$ in both cases, and average surface area of the reservoir as 6×10^5 m^2. If 0.15 m is the priming depth what will be the amount of the extra water stored.
[*Ans.* 0.917 m; 12 ; 46.02×10^4 m^3]

10.20 The following observations of head and the corresponding discharge were made for a V-notch.

Head (m)	Discharge cumec
0.05	8.1×10^{-4}
0.075	22.4×10^{-4}
0.10	47.6×10^{-4}
0.125	80.3×10^{-4}
0.15	126.6×10^{-4}

Assuming $Q = KH^n$; find K and n.
[*Ans.* 1.585, 2.5]

10.21 A Cipolletti weir has a crest length of 0.25 m. If the head on the crest is 0.15 m, claculate the discharge flowing over it. Take $C_d = 0.64$.
[*Ans.* 0.027 m^3/s]

10.22 A broad crested weir with flat top is constructed across the entire 2.7 m width of a rectangular channel. If the head on the weir crest is 0.415 m, find the discharge over the weir. Take $C_d = 0.97$.
[*Ans.* 1.19 m^3/s]

10.23 If the ordinary formula for a rectangular notch without end contractions is used to give the flow across a broad crested weir, what should be the value of the coefficient of discharge in this formula? [*Ans.* 1.082]

10.24 A river with vertical banks is 60 m wide, the depth of flow is 1.5 m and the velocity of flow is 1.2 m/s. A broad crested weir 2.4 m high is constructed across the entire width of the river. Find the head on the weir crest.
[*Ans.* 1.06 m if velocity of approach is neglected,
1.05 m if velocity of approach is considered]

10.25 A rapid stream has a depth of flow 1 m and the velocity of flow of 3.7 m/s. Find the height of a suppressed diversion weir which should be constructed across the stream to raise the water level on the upstream side of the weir by 2 m.
Take $C_{d_1} = 0.58$ and $C_{d_2} = 0.80$.
[*Ans.* 1.422 m (the weir is not submerged)]

10.26 (a) Derive an expression for discharge through a rectangular notch. Explain, how it is modified to take into account the effect of end contractions and velocity of approach.
(b) Water passing over a rectangular notch flows subsequently over a right angled triangular notch. The length of rectangular notch is 0.6 m and its coefficient is 1.84. If the coefficient for triangular notch is 1.42 what will be its working head, when the head on rectangular notch is 0.15 m? [*Ans.* 0.29 m]

10.27 The discharge in a channel varies from 0.16 m^3/s to 0.74 m^3/s. A contracted rectangular weir is to be constructed across this channel to measure the discharge. Determine the length of the weir such that the measured head will never be less than 0.16 m or greater than one-third of the length of the weir. [*Ans.* 1.38 m]

10.28 In one of the sides of a tank there is an isosceles triangular shaped opening with its apex at the top, height L and apex angle 2θ. If water flows through the opening under a head H above its base such that $H < L$, show that the discharge Q is given by

$$Q = \frac{4}{15} C_d \sqrt{2g} \tan\theta \; H^{3/2}(5L - 2H)$$

where C_d is coefficient of discharge

Flow Through Pipes

Chapter 11

11.1 INTRODUCTION

A pipe is a closed conduit which is used for carrying fluids under pressure. Pipes are commonly circular in section. As the pipes carry fluids under pressure, the pipes always run full. The fluid flowing in a pipe is always subjected to resistance due to shear forces between fluid particles and the boundary walls of the pipe and between the fluid particles themselves resulting from the viscosity of the fluid. The resistance to the flow of fluid is in general known as frictional resistance. Since certain amount of energy possessed by the flowing fluid will be consumed in overcoming this resistance to the flow, there will always be some loss of energy in the direction of flow, which however depends on the type of flow. The flow of fluid in a pipe may be either *laminar or turbulent*. Since different laws govern these two types of flows in pipes, the same are required to be dealt with separately. In this chapter, however, only some pipe flow problems have been dealt with and the detailed discussion of the two different types of flows has been made in the subsequent chapters.

11.2 TWO TYPES OF FLOW–REYNOLDS' EXPERIMENT

The existence of the two types of flow, viz., laminar and turbulent, was first demonstrated by Osborne Reynolds in 1883, with the help of a simple experiment as described below.

Reynolds' Experiment. Reynolds' experiment consisted essentially of a constant head tank filled with water, a small tank containing dye, a horizontal glass tube provided with a bell-mouthed entrance and a regulating valve as shown in Fig. 11.1. The water was made to flow from the tank through the glass tube into the atmosphere and the velocity of flow was varied by adjusting the regulating valve. A liquid dye having the same specific weight as that of water, was introduced into the flow at the bell-mouth through a small tube.

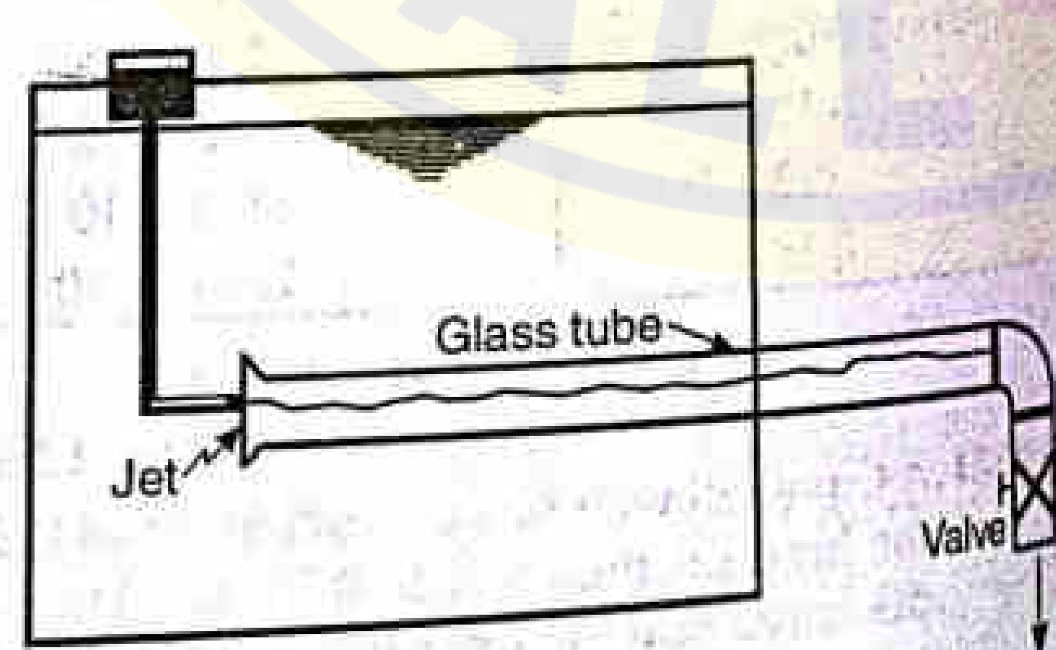


Figure 11.1 *Reynolds' apparatus for demonstrating the type of flow*

From the experiments it was disclosed that when the velocity of flow was low, the dye remained in the form of a straight and stable filament passing through the glass tube so steadily that it scarcely seemed to be in motion. With increase in the velocity of flow a critical state was reached at which the filament of dye showed irregularities and began of waver. With a further increase in the velocity of flow the fluctuations in the filament of dye became more intense and ultimately the dye diffused over the entire cross-section of the tube, due to the intermingling of the particles of the flowing fluid. Figure 11.2 shows the different states of the dye filament.

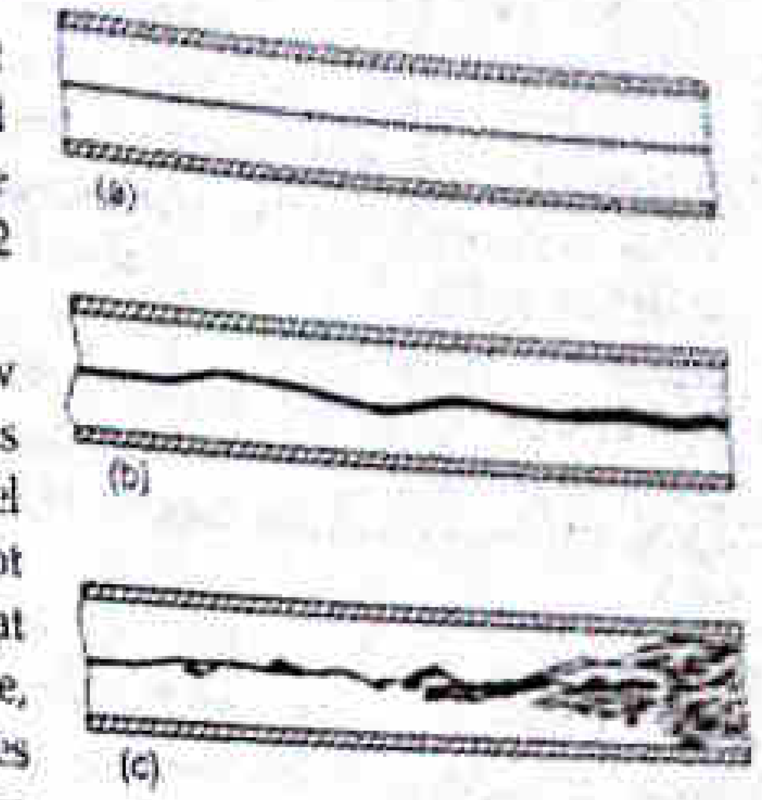


Figure 11.2 *Appearance of dye filament in : (a) laminar flow, (b) transition, and (c) turbulent flow*

Reynolds deduced from his experiments that at low velocities the intermingling of the fluid particles was altogether absent and that the fluid particles moved in parallel layers or laminae, sliding past adjacent laminae but not mixing with them. This is the regime of laminar flow. Since at higher velocities the dye filament diffused through the tube, it was apparent that the intermingling of the fluid particles was occurring, or in other words the flow was turbulent. The velocity at which the flow changes from the laminar to turbulent for the case of a given fluid at a given temperature and in a given pipe is known as *critical velocity*. The state of flow in between these two types of flow is known as 'transitional state' (or flow in transition).

On the basis of his experiments Reynolds discovered that the occurrence of a laminar and turbulent flow was governed by the relative magnitudes of the inertia and the viscous forces. It was indicated by Reynolds that at low velocities of flow, even for the fluids having very small viscosity, the viscous forces become predominant and therefore, the flow is largely viscous in character. However, at higher velocities of flow the inertial forces have predominance over the viscous forces. Reynolds related the inertia to viscous forces and arrived at a dimensionless parameter.

$$\text{Re or } N_R = \frac{\text{Inertia force}}{\text{Viscous force}} = \frac{F_i}{F_v}$$

According to Newton's second law of motion the inertia force F_i is given by

$$F_i = \text{mass} \times \text{acceleration}$$
$$= \rho \times \text{volume} \times \text{acceleration}$$
$$= \rho \times L^3 \times (L/T^2) = (\rho L^2 V^2)$$

Similarly viscous force F_v is given by Newton's law of viscosity as

$$F_v = \tau \times \text{area}$$
$$= \mu \frac{\partial v}{\partial y} \times L^2 = (\mu V L)$$

$$\therefore \quad \text{Re or } N_R = \frac{(\rho L^2 V^2)}{\mu V L} = \frac{\rho V L}{\mu}$$

This dimensionless parameter is called *Reynolds number*, in which ρ and μ are respectively the mass density and viscosity of the flowing fluid, V is the characteristic (or representative) velocity of flow and L is the characteristic linear dimension. In the case of flow through pipes the characteristic linear dimension L is taken as the diameter D of the pipe and the characteristic velocity is taken as the average velocity V of flow of fluid. Thus Reynolds number becomes $(\rho DV/\mu)$ or (VD/υ) where $(\mu/\rho) = \upsilon$, is kinematic viscosity of the flowing fluid. The Reynolds number is therefore a very useful parameter in predicting whether the flow is laminar or turbulent. The limiting values of Reynolds number corresponding to which the flow of fluid in a pipe is either laminar or turbulent are given below.

The existence of two flow regimes may also be indicated with the help of another simple experiment as shown in Fig. 11.3. The apparatus required consists of a uniform horizontal pipe of a known diameter, to which a manometer is connected for measuring the loss of head h_f, occurring in a length L, of the pipe. The head loss can be obtained from the manometer reading for a particular discharge and the mean velocity V of flow through pipe can be determined from the measured discharge. Several values of the head loss can thus be obtained for the corresponding values of the velocities of flow of fluid.

Now if a logarithmic plot of (h_f/L) as ordinate and the velocity V as abscissa is prepared, it will be as shown in Fig. 11.4. From this it will be found that for small values of V, the plot is a straight line with its slope equal to unity. This continues upto certain value of V, represented by point B on the figure, which thus indicates that as long as the velocity is less than the value corresponding to point B, the head loss due to friction will be directly proportional to the velocity of flow of fluid (i.e., $h_f/L \propto V$). Beyond the point B with increasing velocity it will be found that there exists certain transition region extending upto point C, during which there is an abrupt increase in the rate at which the loss of head varies. After the region of transition has passed, again the curve obtained is in the form of straight lines with slopes ranging from 1.72 to 2.00.

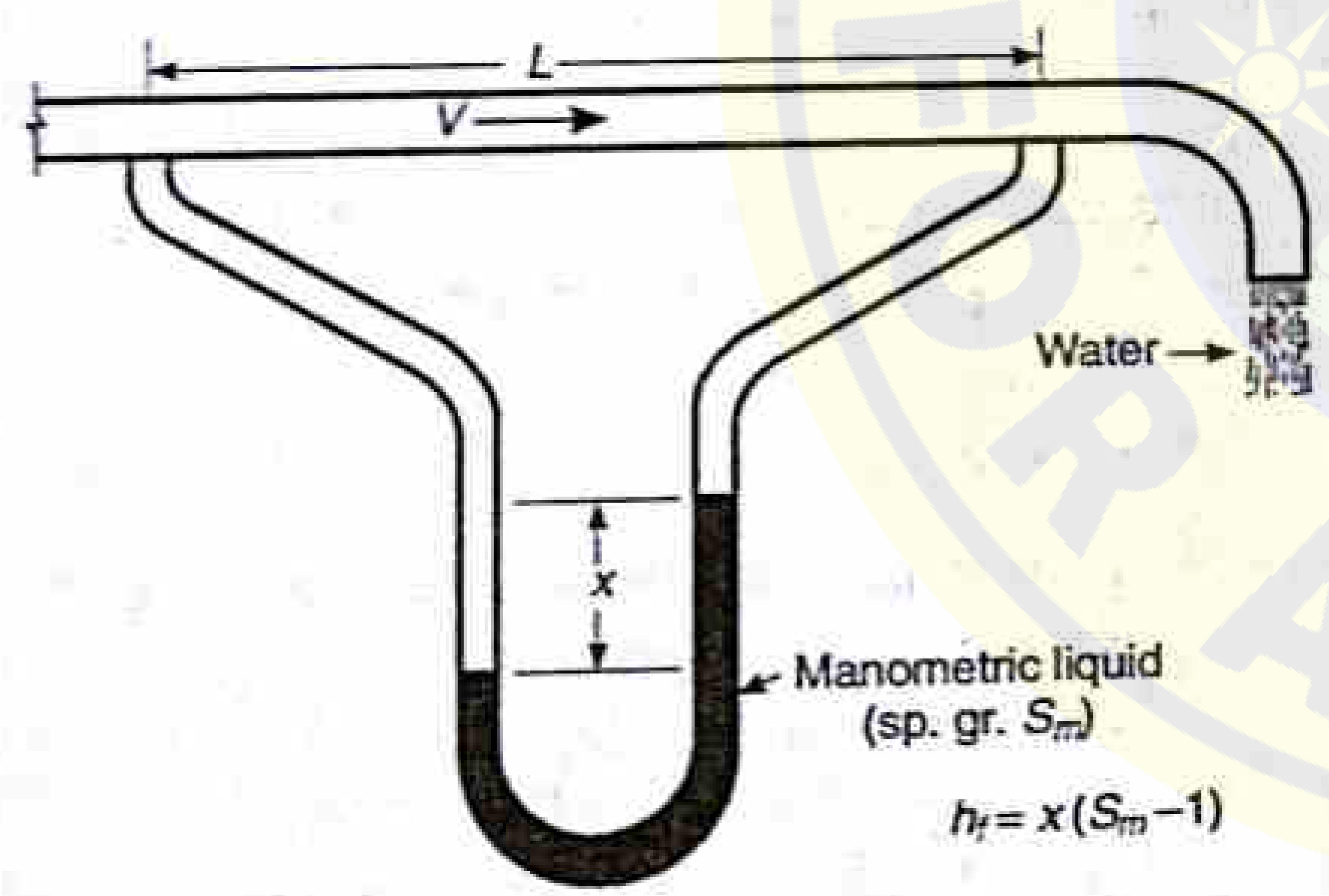


Figure 11.3 *Apparatus for measuring the loss of head in a pipe*

However, if the velocity is gradually reduced from a higher value, the line BC will not be retraced. Instead the points may be along curve CA, as indicated by arrows in Fig. 11.4. The point B is known as the *upper critical point* and the point A is known as the *lower critical point*, and the corresponding velocities are known as *upper critical velocity* and *lower critical velocity* respectively.

It is thus seen that upto point A, the drop in pressure head due to frictional resistance is directly proportional to the mean velocity of flow V, which is the range of laminar flow. Beyond point C, the drop in pressure head due to frictional resistance varies as V^n, where n ranges from 1.72 to 2.0, which is the zone of turbulent flow. Between the points A and C (i.e., in between the regimes of laminar and turbulent flow) lies the transition region as shown in Fig. 11.4.

The upper critical Reynolds number corresponding to point B (i.e., upper limit of laminar flow in pipes) was found by Reynolds to lie between 12000 to 14000. But the upper critical Reynolds number is indefinite, being dependent upon initial disturbance affecting the flow, shape of entry of pipe, roughness of pipe wall etc. Thus the practical value of upper critical Reynolds number may be considered to lie between 2700 to 4000.

The lower critical Reynolds number for flow of fluid in pipes corresponding to point A is of greater engineering importance as it indicates a condition below which all turbulence entering the flow from any source will be damped out by viscosity and thus sets a limit below which laminar flow will always occur. Experimentally the value of the lower critical Reynolds number has been found to be approximately 2000.

Between Reynolds numbers 2000 and 4000 the transition region exists.

The concept of critical Reynolds number which distinguishes the regimes of laminar and turbulent flow is indeed quite useful in the study of various fluid flow phenomena. Applying this concept to the flow of any fluid in circular pipes, one may predict that the flow will be laminar if Reynolds number is less than 2000 and turbulent if it is greater than 4000. It may however be pointed out that critical Reynolds number is very much a function of boundary geometry. Thus for example in the case of flow between parallel plates, the critical Reynolds number (computed by using mean velocity of flow and the spacing between the plates) is approximately 1000; in the case of flow in a wide open channel the critical Reynolds number (computed by using velocity of flow and depth of flow) is approximately 500; and for flow around a sphere the critical Reynolds number (computed by using the approach velocity and diameter of the sphere) is approximately 1.

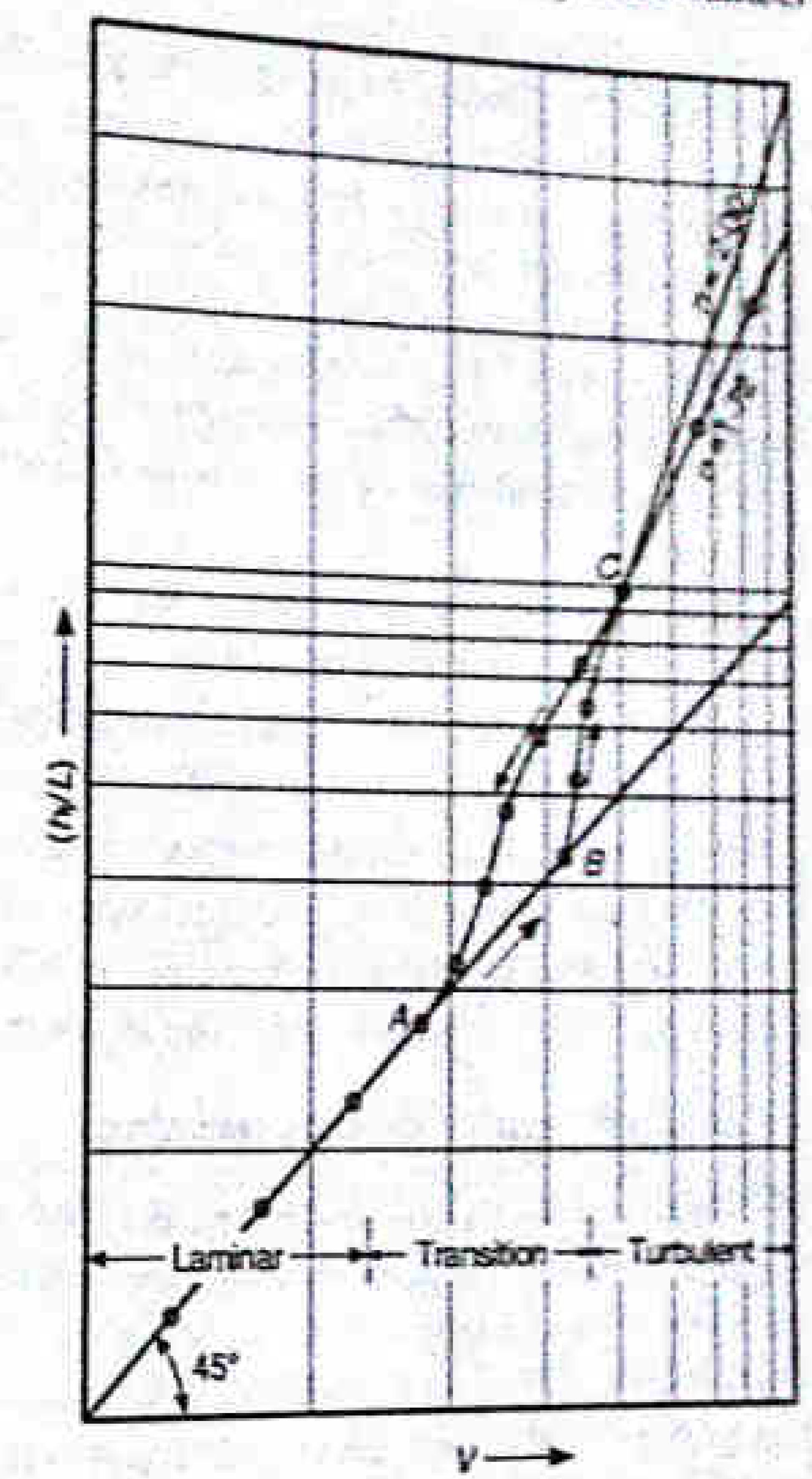


Figure 11.4 *Plot of (h_f/L) v/s V showing upper and lower critical points and velocities*

11.3 LAWS OF FLUID FRICTION

As stated earlier the frictional resistance offered to the flow depends on the type of flow. As such different laws are obeyed by the frictional resistance in the laminar and the turbulent flows. On the basis of the experimental observations the laws of fluid friction for the two types of flows may be narrated as follows.

1. Laws of Fluid Friction for Laminar Flow. The frictional resistance in the laminar flow is as follows

(i) proportional to the velocity of flow,

(ii) independent of the pressure,
(iii) proportional to the area of surface in contact,
(iv) independent of the nature of the surface in contact,
(v) greatly affected by the variation of the temperature of the flowing fluid.

The first law enunciated above can also be proved analytically as indicated in the next chapter. Further the reason for the frictional resistance in the case of laminar flow being independent of the nature of the surface in contact, is that when a fluid flows past a surface with velocity less than critical velocity, a film of almost stationary fluid is formed over the surface, which prevents the flowing fluid to came in contact with the boundary surface. Similarly in the case of laminar flow the resistance is due to viscosity only and the viscosity of a fluid depends on its temperature.

2. Laws of Fluid Friction for Turbulent Flow. The frictional resistance in the case of turbulent flow is as follow

(i) proportional to (velocity)^n, where the index n varies form 1.72 to 2.0,
(ii) independent of the pressure,
(iii) proportional to the density of the flowing fluid,
(iv) slightly affected by the variation of the temperature of the flowing fluid,
(v) proportional to area of surface in contact,
(vi) dependent on the nature of the surface in contact.

Since mostly the flow of fluids in pipes is turbulent, in the various pipe flow problems dealt with in this chapter the flow is assumed to be turbulent.

11.4 FROUDE'S EXPERIMENTS

W. Froude conducted a series of experiments to investigate frictional resistance offered to the flowing water by different surfaces. The experiments were conducted in a tank about 100 m (300 ft) long, 11 m (36 ft) broad and 3 m (10 ft) deep and containing water. Thin wooden boards about 5 mm (3/16 in) thick, 0.475 m (19 in) wide and lengths varying from 0.6 m (2 ft) to 15 m (50 ft) were towed end wise in this tank by connecting them to a carriage running on rails provided on the sides of the tank. The carriage was hauled along at speeds varying from 30 m (100 ft) to 300 m (1000 ft) per minute, by means of a wire rope passing around a drum. The boards were towed in a completely submerged position such that the upper edge was about 0.45 m $\left(1\frac{1}{2}\text{ ft}\right)$ below the water surface in the tank and the force required to tow the board being measured. In order to develop the surfaces of different types the surfaces of the boards were covered with varnish, tinfoil, calico and sand in turn.

From the results of these experiments Froude derived the following conclusions:

(i) The frictional resistance varies approximately with the square of the velocity.

(ii) The frictional resistance varies with the nature of the surface.

(iii) The frictional resistance per unit area of surface decreases as the length of the board increases but is constant for long lengths.

Thus if f' is the frictional resistance per unit area of given surface at unit velocity, A is the area of wetted surface and V is the velocity, then the total friction resistance F is given by

$$F = f'AV^n$$

Assuming the index

$$n = 2$$
$$F = f'AV^2$$

In the above expression except f' all the terms are known and hence the value of f' may be computed.

11.5 EQUATION FOR HEAD LOSS IN PIPES DUE TO FRICTION–DARCY-WEISBACH EQUATION

Consider a horizontal pipe of cross-sectional area A carrying a fluid with a mean velocity V. Let 1 and 2 be the two sections of the pipe L distance apart, where let the intensities of pressure be p_1 and p_2 respectively. By applying Bernoulli's equation between the sections 1 and 2, we obtain

$$\frac{p_1}{w}+\frac{V_1^2}{2g}+Z_1 = \frac{p_2}{w}+\frac{V_2^2}{2g}+Z_2+h_f$$

Since $\quad V_1 = V_2 = V$ and $Z_1 = Z_2$

$$\text{Loss of head} = h_f = \frac{p_1}{w}-\frac{p_2}{w}$$

i.e., the pressure intensity will be reduced by the frictional resistance in the direction of flow and the difference of pressure heads between any two sections is equal to the loss of head due to friction between these sections.

Further let f' be the frictional resistance per unit area at unit velocity, then frictional resistance

$$= f' \times \text{area} \times V^n$$
$$= f' \times PL \times V^n$$

where P is the wetted perimeter of the pipe.

The pressure forces at the sections 1 and 2 are (p_1A) and (p_2A) respectively. Thus resolving all the forces horizontally, we have

$$p_1A = p_2A + \text{frictional resistance}$$

or
$$(p_1-p_2)A = f' \times PL \times V^n$$

or
$$(p_1-p_2) = f' \times \frac{P}{A} \times LV^n$$

Dividing both sides by the specific weight w of the flowing fluid

$$\frac{p_1-p_2}{w} = \frac{f'}{w}\times\frac{P}{A}LV^n$$

But
$$h_f = \frac{p_1-p_2}{w}, \text{ then}$$

$$h_f = \frac{f'}{w}\times\frac{P}{A}\times LV^n$$

The ratio of the cross-sectional area of the flow (wetted area) to the perimeter in contact with the fluid (wetted perimeter) i.e., $\left(\frac{A}{P}\right)$ is called *hydraulic mean depth* (H.M.D.) or *hydraulic radius* and it is represented by m or R.

Then $$h_f = \frac{f'}{w} \times \frac{LV^n}{m}$$ ...(11.1)

For pipes running full

$$m = \frac{A}{P} = \frac{\left(\frac{\pi D^2}{4}\right)}{(\pi D)} = \frac{D}{4}$$

Substituting this in the equation for h_f and assuming

$$n = 2$$

$$h_f = \frac{4f'}{w}\frac{LV^2}{D}$$

Putting $$\frac{4f'}{w} = \frac{f}{2g}$$

$$h_f = \frac{fLV^2}{2gD}$$ ...(11.2)

where f is known as *friction factor*, which is a dimensionless quantity.

Equation 11.2 is known as *Darcy–Weisbach equation* which is commonly used for computing the loss of head due to friction in pipes. It may be noted that the head loss due to friction is also expressed in terms of the velocity head ($V^2/2g$) corresponding to the mean velocity. Further the observations show that the coefficient f is not a constant but its value depends on the roughness condition of the pipe surface and the Reynolds number of the flow. As such in order to determine the loss of head due to friction correctly, it is essential to estimate the value of the factor f correctly. For this purpose on the basis of experimental observations certain relationships have been developed and the same have been represented diagrammatically as shown in Chapter 14, which indicate the manner in which f varies and also facilitate the correct estimation of the value of the friction factor f.

11.6 OTHER FORMULAE FOR HEAD LOSS DUE TO FRICTION IN PIPES

In addition to Darcy–Weisbach equation there are a few more formulae, as indicated below, which are also adopted in the analysis of the pipe flow problems.

1. Chezy's Formula. One of the formula which may be developed to represent the loss of head due to friction in pipe is the Chezy's formula.

The ratio $\left(\frac{h_f}{L}\right)$ represents the slope of the hydraulic grade line or energy grade line and it is usually represented by i or S, i.e., $\left(\frac{h_f}{L}\right) = i$ or S.

Then from Eq. 11.1

$$V^n = \frac{w}{f'} \times m \times \left(\frac{h_f}{L}\right) = \frac{w}{f'}(mi)$$

Assuming $$n = 2 \text{ and } C = \sqrt{\frac{w}{f'}}$$

$$V = C\sqrt{mi} \text{ or } V = C\sqrt{RS}$$ ...(11.3)

Equation 11.3 is known as *Chezy's formula* in which C is known as Chezy's coefficient which is also determined experimentally. However, Chezy's formula is not commonly used in the case of flow through pipes.

In addition to the above derived formulae there are certain empirical formulae as indicated below which are also used for pipes.

2. Manning's Formula. It is one of the most common formula, which is mostly used for the analysis of the problems of flow through channels, but often used for the analysis of the pipe flow problems too. According to this formula the mean velocity of flow V is given by

$$V = \frac{1}{n}R^{2/3}S^{1/2}$$ ...(11.4)

where n is the Manning's roughness (or rugosity) coefficient, R is hydraulic radius and S is the slope of the hydraulic grade line or energy grade line. The value of n depends on the nature of the boundary and some of the typical values of n for pipe flow are as given below:

Type of pipe	Manning's n
(1) Brass and glass pipe	0.009 to 0.013
(2) Asbestos-cement pipe	0.010 to 0.012
(3) Wrought iron, Welded steel, Wooden stave	0.010 to 0.014
(4) Concrete pipe very smooth	0.011 to 0.012
(5) Concrete pipe with rough joints	0.016 to 0.017
(6) Vitrified sewer pipe	0.013 to 0.015
(7) Rivetted steel pipe	0.013 to 0.017
(8) Corrugated iron pipe	0.020 to 0.022

By comparing the Manning's formula with the Chezy's formula, we obtain

$$C = \frac{1}{n}R^{1/6}$$ ...(11.5)

Further for circular pipes, since $R = \left(\frac{D}{4}\right)$, where D is the diameter of the pipe the Manning's formula for pipes may also be written as

$$V = \frac{0.3968}{n}D^{2/3}S^{1/2}$$ ...(11.6)

3. Hazen–Williams Formula. It is yet another formula which is widely used for designing water supply systems. According to this formula the mean velocity of flow is given by

$$V = 0.85\, C_1\, R^{0.63}\, S^{0.54} \quad ...(11.7)$$

where C_1 is a coefficient, the value of which depends on the type of the boundary. Some of the values of C_1 are as given below:

Type of pipe	*Value of C_1*
(1) Extremely smooth and straight	140
(2) Very smooth	130
(3) Smooth wood stave, smooth masonry	120
(4) New rivetted steel, vitrified clay	110
(5) Old rivetted steel	95

11.7 OTHER ENERGY LOSSES IN PIPES

When a fluid flows through a pipe, certain resistance is offered to the flowing fluid, which results in causing a loss of energy. The various energy losses in pipes may be classified as:

(i) Major losses.

(ii) Minor losses.

The major loss of energy, as a fluid flows through a pipe, is caused by friction. It may be computed by Darcy–Weisbach equation as indicated earlier. The loss of energy due to friction is classified as a major loss because in the case of long pipelines it is usually much more than the loss of energy incurred by other causes.

The minor losses of energy are those which are caused on account of the change in the velocity of flowing fluid (either in magnitude or direction). In case of long pipes these losses are usually quite small as compared with the loss of energy due to friction and hence these are termed 'minor losses' which may even be neglected without serious error. However, in short pipes these losses may sometimes outweigh the friction loss. Some of the losses of energy which may be caused due to the change of velocity are indicated below:

(a) Loss of energy due to sudden enlargement,

$$h_L = \frac{(V_1 - V_2)^2}{2g}$$

(b) Loss of energy due to sudden contraction,

$$h_L = 0.5\frac{V^2}{2g}$$

(c) Loss of energy at the entrance to a pipe,

$$h_L = 0.5\frac{V^2}{2g}$$

(d) Loss of energy at the exit from a pipe,

$$h_L = \frac{V^2}{2g}$$

(e) Loss of energy due to gradual contraction or enlargement,

$$h_L = k\frac{(V_1 - V_2)^2}{2g}$$

(f) Loss of energy in bends,

$$h_L = k\frac{V^2}{2g}$$

(g) Loss of energy in various pipe fittings,

$$h_L = k\frac{V^2}{2g}$$

The expressions for the various minor losses of energy noted above have been derived in Chapter 9 (Section 9.8).

11.8 HYDRAULIC GRADE LINE AND ENERGY GRADE LINE

In the study of flow of fluid in pipes the concept of *hydraulic grade line (H.G.L.)* and *energy grade line* are quit useful. The *energy grade line* is also known as *total energy line (T.E.L.) or total head line*. The *hydraulic* and *energy grade lines* may be obtained as indicated below.

Consider a long pipeline carrying liquid from a reservoir *A* to a reservoir *B* as shown in Fig. 11.5. At several points along the pipeline let piezometers be installed. The liquid will rise in the piezometers to certain heights corresponding to the pressure intensity at each section. The height of the liquid surface above the axis of pipe in the piezometer at any section will be equal to the pressure head (p/w) at that section. On account of loss of energy due to friction, the pressure head will decrease gradually from section to section of the pipe in the direction of flow. If the pressure heads at the different sections of the pipe are plotted to scale as vertical ordinates above the axis of the pipe and all these points are joined by a straight line then as shown in Fig. 11.5, a straight slopping line will be obtained, which is known as *hydraulic grade line (H.G.L)*. Since at any section of the pipe the vertical distance between the pipe axis and the hydraulic grade line is equal to the pressure head at that section, it is also known as *pressure line*. Moreover if Z is the height of the pipe axis at any section above an arbitrary datum as shown in Fig.11.5, then the vertical height of the hydraulic grade line above the datum at that section of the pipe represents the piezometric head equal to $\left(\frac{p}{w}+Z\right)$. As such sometimes the hydraulic grade line is also known as *piezometric head line*.

As shown in Fig. 11.5, for some distance from the entrance section of the pipe the hydraulic grade line is not very well defined. This is so because as the liquid from the reservoir enters the pipe, vena-contracta is formed and a sudden drop in pressure head takes place in this portion of the pipe. The hydraulic grade line is this portion of the pipe is therefore shown by dotted curved line. Further the exit section of the pipe being submerged, the pressure head at this section is equal to the height of liquid surface in the reservoir *B* and hence the hydraulic grade line at the exit section of the pipe will meet the liquid surface in the reservoir *B*.

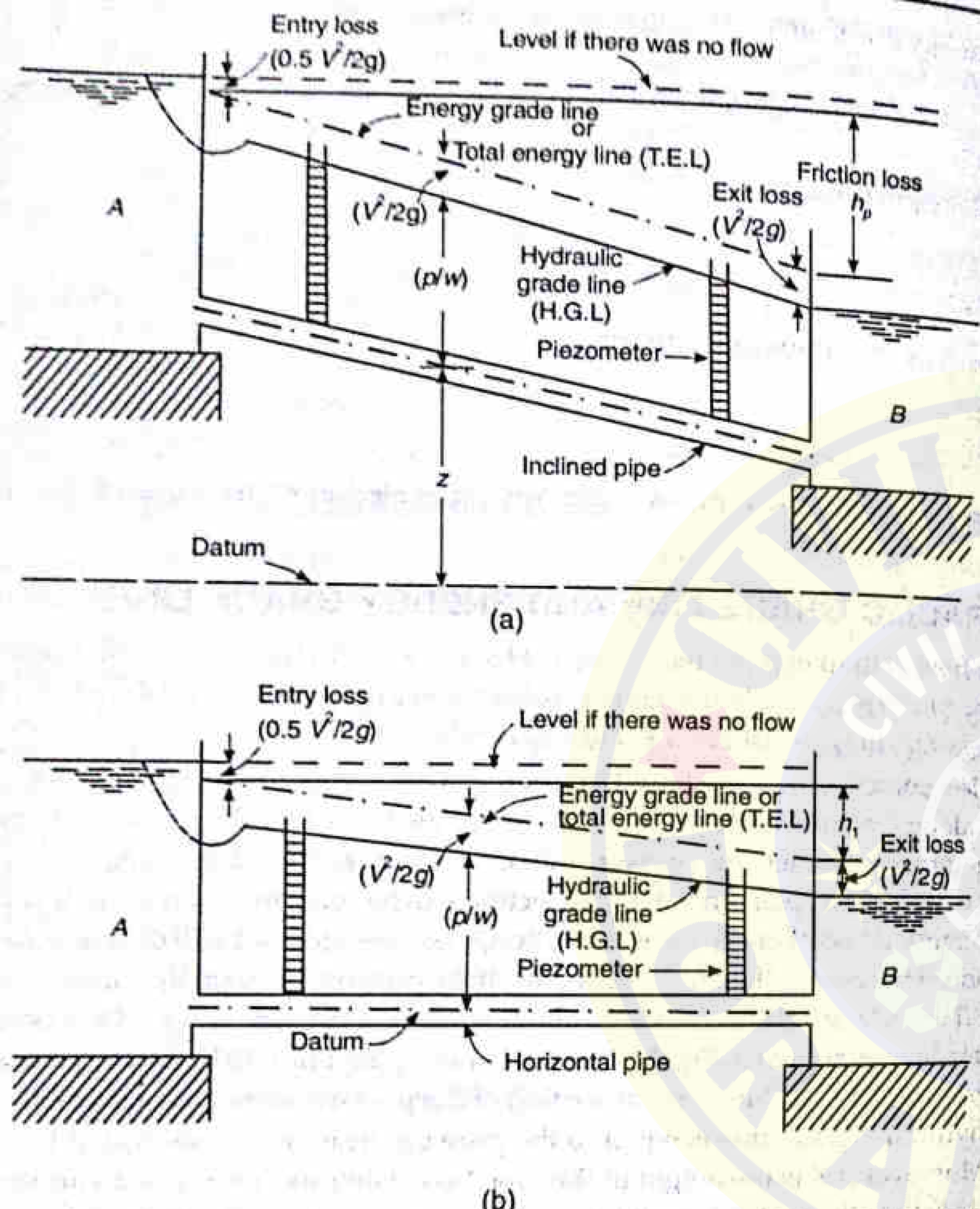


Figure 11.5 *Hydraulic grade line add energy grade line for : (a) an inclined pipe; (b) horizontal pipe, connecting two reservoirs*

If at different sections of the pipe the total energy (in terms of head) is plotted to scale as vertical ordinate above the assumed datum and all these points are joined then a straight slopping line will be obtained which is known as *energy grade line* or *total energy line (T.E.L.)*. Since total energy at any section of the pipe is equal to the sum of the pressure head (p/w), the datum head Z, and the velocity head ($V^2/2g$), and the vertical distance between the datum and the hydraulic grade line is equal to the piezometric head $\left(\frac{p}{w}+Z\right)$, the energy grade line (or total energy line) will be parallel to the hydraulic grade line with a vertical distance between them equal to ($V^2/2g$). At the entrance section of the pipe there occurs some loss of energy called 'entrance loss' equal to $h_L = 0.5\,(V^2/2g)$ and hence the energy grade line (or total energy line) at this section will lie at a vertical depth equal to 0.5 ($V^2/2g$) below the liquid surface in the reservoir A. Similarly at the exit section of the pipe since there occurs an exit loss equal to $h_L = (V^2/2g)$, the energy grade line (or total energy line) at this section will lie at a vertical distance equal to ($V^2/2g$) above the liquid surface in the reservoir B. Since at any section of the pipe the vertical distance between the datum and the energy grade line (or total energy line) represents the total energy possessed by the flowing liquid at the section, the vertical distance between the energy grade line (or total energy line) and the horizontal line drawn through the liquid surface in the reservoir A will represent the total loss of energy incurred up to that section.

If the pipe line connecting the two reservoirs is horizontal, as shown in Fig. 11.5 (b), then the datum may be assumed to be along the pipe axis only. The piezometric head and the pressure head will then become the same. The hydraulic and energy grade lines for this case may also be obtained in the same manner as shown Fig. 11.5 (b).

If pipeline carrying liquid from the reservoir A discharges freely in the atmosphere at its exit end, then as shown in Fig. 11.6, the hydraulic grade line at the exit end of the pipe will pass through the

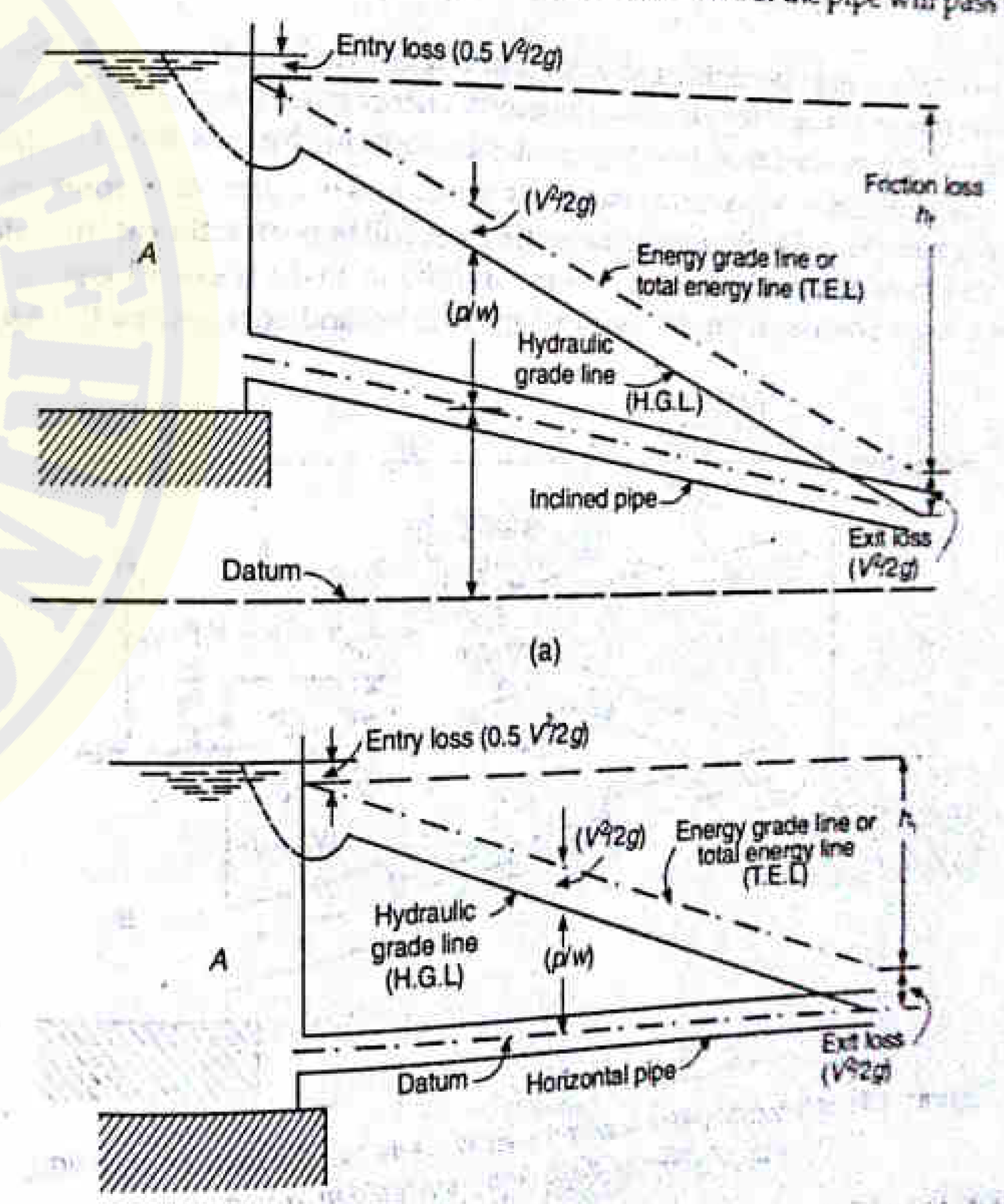


Figure 11.6 *Hydraulic grade line and energy grade line for (a) an inclined pipe; (b) horizontal pipe, discharging freely in atmosphere*

centre line of the pipe since the pressure head at the exit end of the pipe will be zero (being atmospheric). The energy grade line (or total energy line) will again be parallel to the hydraulic grade line and it will be at a vertical distance of $(V^2/2g)$ above the hydraulic grade line.

As the hydraulic and energy grade lines are parallel to each other their slopes are equal. If the pipe is horizontal the slope i of the hydraulic and energy grade lines is equal to the loss of head due to friction h_f divided by the length of pipe L; that is $i = (h_f/L) = [d(p/w)dL]$. If the pipe is inclined the slope i of the hydraulic and energy grade lines is equal to the loss of head due to friction h_f divided by the length of the horizontal projection of the pipe L'; that is $i = (h_f/L')$. However, if the inclination of the pipe is small the slope I of the hydraulic and energy grade lines may be approximately taken equal to (h_f/L) as in the case of a horizontal pipe.

Hydraulic Gradient and Energy Gradient

The change in piezometric head per unit length of the pipe is known as *hydraulic gradient*. Thus if the pipe is horizontal (for which $(dZ/dL) = 0$) the hydraulic gradient is equal to the slope of the hydraulic grade line [i.e., $d(p/w)/dL$]. However, if the pipe is inclined then the hydraulic gradient is equal to $[d(p/w + Z)/dL]$.

The change in total energy per unit length of the pipe is known as *energy gradient*. Thus if the pipe is horizontal the energy gradient is equal to the slope of the energy grade line [i.e., $(h_f/L) = d(p/w)/dL$]. However, if the pipe is inclined then the energy gradient is equal to $[d(p/w + V^2/2g + Z)/dL]$.

Figure 11.7 illustrates two reservoirs connected by three pipes of different diameters. Due to the introduction of a pipe of smaller diameter in the centre, there will be contraction at the entrance of the smaller pipe and an enlargement at its exit. As such in addition to the friction losses in each of the pipes, there will be entrance loss, contraction loss, enlargement loss and exit loss, and the corresponding

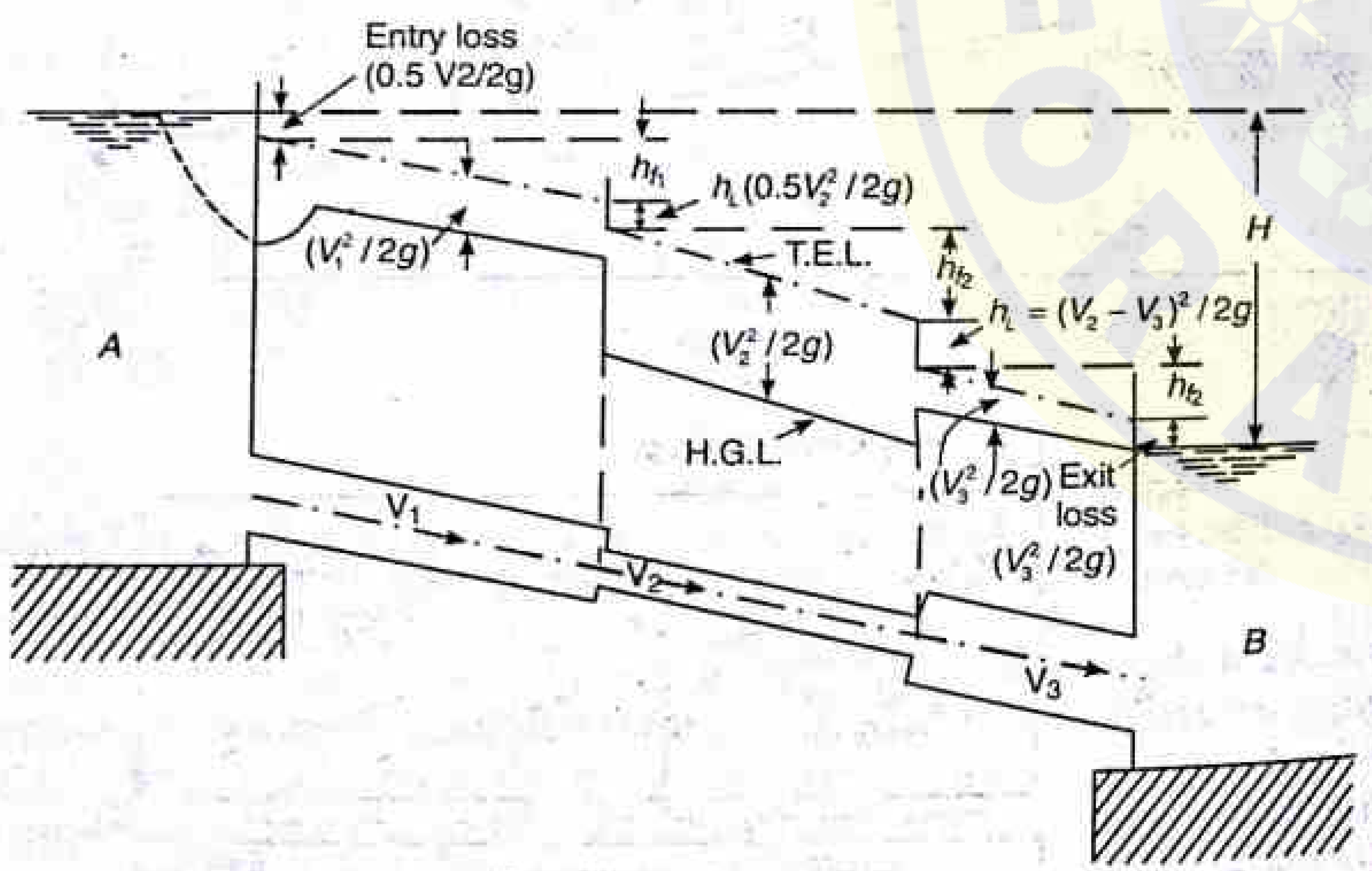


Figure 11.7 *Hydraulic grade line and energy grade line (or total energy line) for pipes of different diameters connected in series.*

hydraulic grade line and the energy grade line (or total energy line) will be as shown in Fig. 11.7. It may however be observed that the hydraulic grade line may even rise in the direction of flow when the

flow passes from a narrower pipe to a larger pipe, since the velocity in the larger pipe is smaller than that in the smaller pipe and consequently the pressure in the larger pipe is more than that in the smaller pipe. It should, however, be noted that the energy grade line (or total energy line) will not rise in the direction of flow, since there is always a loss of energy in the direction of flow, unless there exists some device, such as pump, in the pipe system which will add energy to the flowing liquid, thereby causing an abrupt rise in the hydraulic grade line as well as energy grade line (or total energy line).

11.9 FLOW THROUGH LONG PIPES

Consider a long pipeline of diameter D and length L carrying liquid from a reservoir A to another reservoir B, as shown in Fig. 11.8. Let H_A and H_B be the constant heights of the liquid surfaces in the reservoirs A and B respectively above the centre of the pipe. Further let Z_A and Z_B be the heights of the centres of the pipe ends connected to the reservoirs A and B respectively. Now if V is the mean velocity of flow through the pipe then the head loss due to friction

$$h_f = \frac{fLV^2}{2gD}$$

and head loss at the entrance of pipe

$$= 0.5\frac{V^2}{2g}$$

and head loss at the exit of pipe

$$= \frac{V^2}{2g}$$

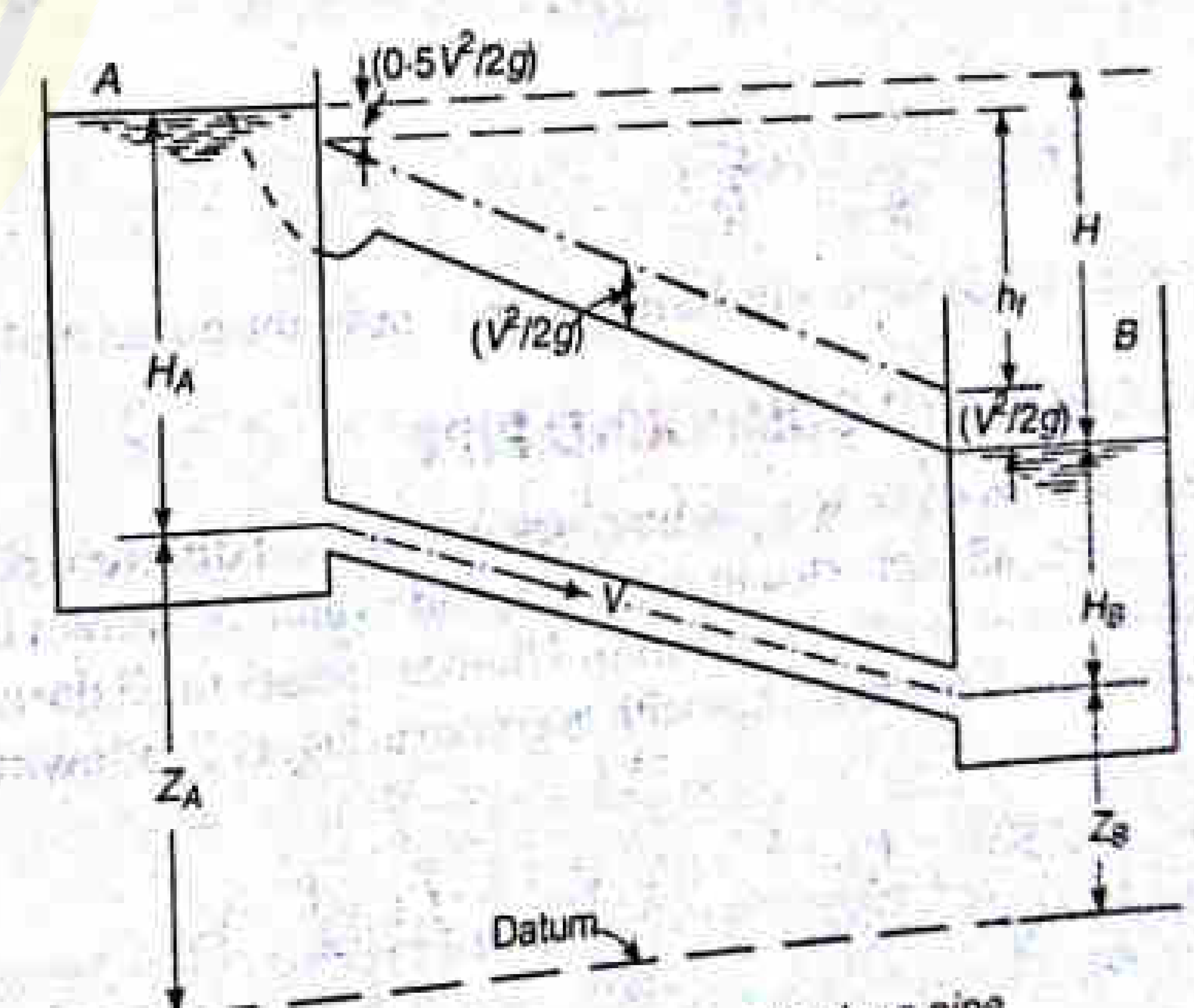


Figure 11.8 *Flow through a long pipe*

Applying Bernoulli's equation between points (1) and (2) in the reservoirs A and B respectively, we obtain

$$H_A+Z_A = H_B+Z_B+0.5\frac{V^2}{2g}+\frac{fLV^2}{2gD}+\frac{V^2}{2g}$$

or $$(H_A+Z_A)-(H_B+Z_B) = \frac{V^2}{2g}\left(1.5+\frac{fL}{D}\right)$$

But $(H_A+Z_A)-(H_B+Z_B) = H$

where H is the difference in the liquid surfaces in the reservoirs A and B. Thus

$$H = \frac{V^2}{2g}\left(1.5+\frac{fL}{D}\right) \quad ...(11.8)$$

Equation 11.8 indicates that the difference in the liquid surfaces in the two reservoirs at the two ends of the pipe is equal to the sum of the various head losses. From this equation the unknown velocity may be computed.

If the pipe is long (say, more than 1000 times the diameter) the loss of head due to friction will be very large as compared with the minor losses which may then be neglected, thereby simplifying the expression as

$$H = \frac{fLV^2}{2gD}; \text{ or } V=\sqrt{\frac{2gHD}{fL}} \quad ...(11.9)$$

If the pipe in Fig. 11.8, instead of discharging into the reservoir B, discharges into the atmosphere the equation would then be

$$(H_A+Z_A) = Z_B+0.5\frac{V^2}{2g}+\frac{fLV^2}{2gD}+\frac{V^2}{2g}$$

or $$H = \frac{V^2}{2g}\left(1.5+\frac{fL}{D}\right)$$

where H is the height of the liquid surface in the reservoir A above the outlet end of the pipe.

11.10 PIPES IN SERIES OR COMPOUND PIPE

If a pipeline connecting two reservoirs is made up of several pipes of different diameters, D_1, D_2, D_3 etc., and lengths L_1, L_2, L_3 etc., all connected in series (i.e., end to end) as shown in Fig. 11.7, then the difference in liquid surface levels is equal to the sum of the head losses in all the sections. Further the discharge through each pipe will be same. Thus with reference to Fig. 11.7 following equations may be obtained

$$H=0.5\frac{V_1^2}{2g}+\frac{f_1L_1V_1^2}{2gD_1}+\frac{0.5V_2^2}{2g}+\frac{f_2L_2V_2^2}{2gD_2}+\frac{(V_2-V_3)^2}{2g}+\frac{f_3L_3V_3^2}{2gD_3}+\frac{V_3^2}{2g} \quad ...(11.10)$$

Also

$$Q = \frac{\pi D_1^2}{4}V_1=\frac{\pi D_2^2}{4}V_2=\frac{\pi D_3^2}{4}V_3 \quad ...(11.11)$$

However, if the minor losses are neglected as compared with the loss of head due to friction in each pipe, then Eq. 11.10 becomes

$$H = \frac{f_1L_1V_1^2}{2gD_1}+\frac{f_2L_2V_2^2}{2gD_2}+\frac{f_3L_3V_3^2}{2gD_3} \quad ...(11.12)$$

The above noted equations may be used to solve the problems of pipelines in series. There are two types of problems which may arise for the pipelines in series, viz., (a) given discharge Q, to determine the head H, and (b) given H, to determine discharge Q.

In the case of (a) the solution is simple since the discharge and diameters and lengths of the pipes are known, Reynolds numbers can easily be computed and the values of the friction factors f_1, f_2 and f_3 readily found from Moody's diagram (shown in Chapter 14). These can then be substituted in the above equations to determine the value of H.

In the case of (b) first of all the values of f_1, f_2 and f_3 are assumed. For simplicity, these may even be assumed to be equal. Then by substituting in equation for H, and solving by trial the value of discharge Q may be obtained. Using this value of Q the value of Reynolds number for each pipe may be computed and from Moody's diagram the values of f_1, f_2 and f_3 are found. With these new values of f_1, f_2 and f_3 a new value of Q is computed by substitution in the equation for H. Since in actual practice rough turbulent flow* occurs for which the value of f varies very little with Reynolds number and therefore only one or two trials will be necessary to get the correct value of Q.

11.11 EQUIVALENT PIPE

Often a compound pipe consisting of several pipes of varying diameters and lengths is to be replaced by a pipe of uniform diameter, which is known as *equivalent pipe*. The uniform diameter of the equivalent pipe is known as the equivalent diameter of the compound pipe. The size of the equivalent pipe may be determined as follows.

If L_1, L_2, L_3 etc., are the lengths and D_1, D_2, D_3 etc., are the diameters respectively of the different pipes of a compound pipeline, then the total head loss in the compound pipe, neglecting the minor losses, is

$$h_L = \frac{f_1L_1V_1^2}{2gD_1}+\frac{f_2L_2V_2^2}{2gD_2}+\frac{f_3L_3V_3^2}{2gD_3}+......$$

Again by continuity

$$Q = a_1V_1=a_2V_2=a_3V_3=.......$$

$$= \frac{\pi}{4}D_1^2V_1=\frac{\pi}{4}D_2^2V_2=\frac{\pi}{4}D_3^2V_3=.....$$

Assuming $$f_1 = f_2=f_3=....=f$$

$$h_L = \frac{f}{2g}\frac{Q^2}{(\pi/4)^2}\left[\frac{L_1}{D_1^5}+\frac{L_2}{D_2^5}+\frac{L_3}{D_3^5}+.........\right]$$

If D is the diameter and L is the length of the equivalent pipe then it would carry the same discharge Q if the head loss due to friction in the equivalent pipe is same as that in the compound pipe. The loss of head due to friction in the equivalent pipe is

* See Chapter 14.

$$h_L = \frac{fLV^2}{2gD} = \frac{f}{2g}\frac{Q^2}{(\pi/4)^2}\frac{L}{D^5}$$

Thus equating the two head losses, we get

$$\frac{L}{D^5} = \left[\frac{L_1}{D_1^5}+\frac{L_2}{D_2^5}+\frac{L_3}{D_3^5}+......\right] \quad ...(11.13)$$

Equation 11.13 is known as *Dupuit's equation*, which may be used to determine the size of the equivalent pipe. Thus if the length of the equivalent pipe is equal to the total length of the compound pipe i.e., $L = (L_1 + L_2 + L_3 + ...)$, then the diameter D of the equivalent pipe may be determined by using equation 11.13. Sometimes a pipe of given diameter D which is available may be required to be used as equivalent pipe to replace a compound pipe, in which case the length of the equivalent pipe may be required to be determined and the same may also be determined by using Eq. 11.13.

11.12 PIPES IN PARALLEL

When a main pipeline divides into two or more parallel pipes which again join together downstream and continue as a main line as shown in Fig. 11.9, the pipes are said to be in parallel. The pipes are connected in parallel in order to increase the discharge passing through the main. Such a system is analogous to a parallel electric circuit in which the drop in potential and flow of electric current can be compared to head loss and discharge (or flow rate) in a fluid flow respectively.

Referring Fig. 11.9, the discharge in the main line is equal to the sum of the discharges in each of the parallel pipes.

Thus $\quad Q = Q_1 + Q_2 \quad ...(11.14)$

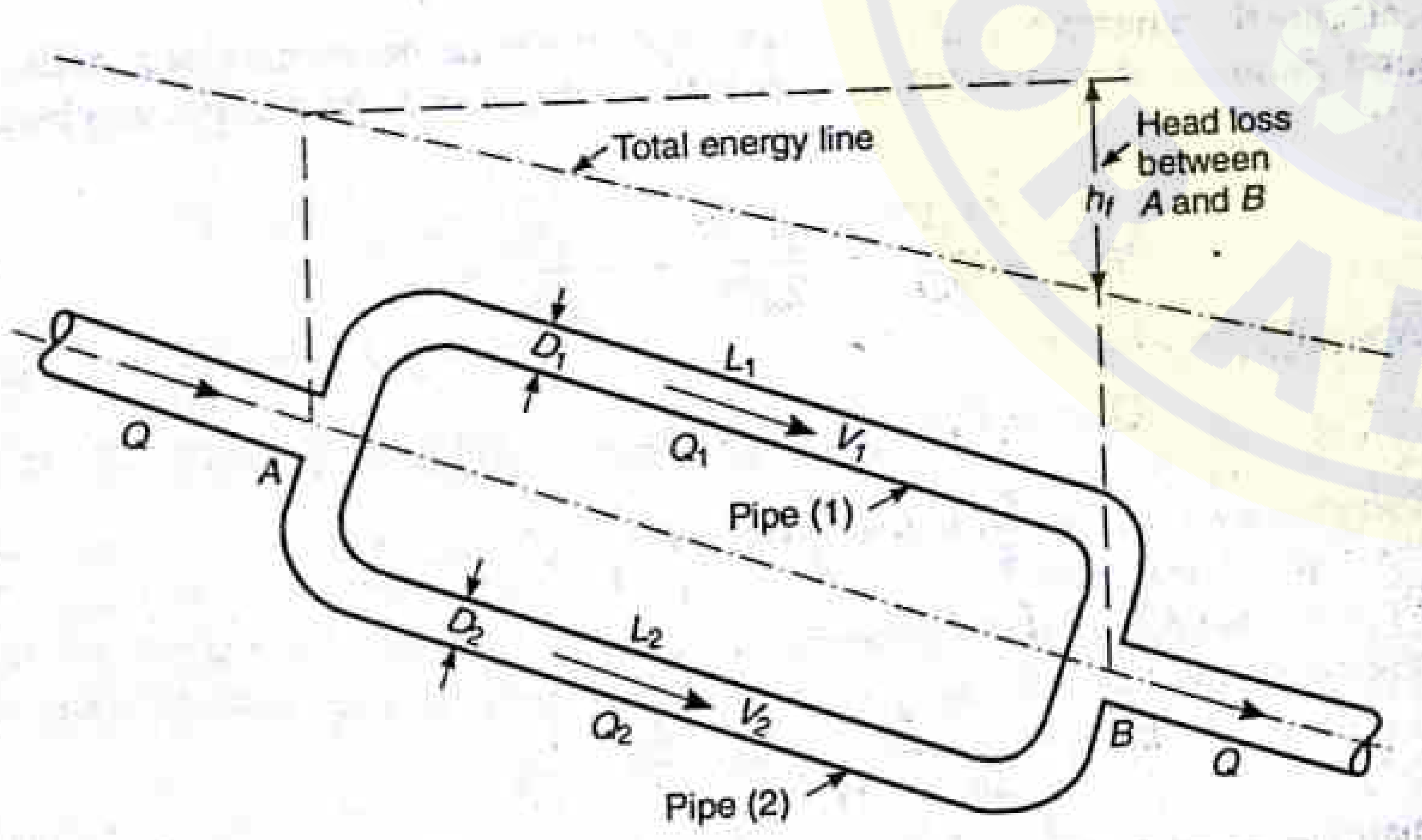


Figure 11.9 *Pipes in parallel*

The flow of liquid in pipes (1) and (2) takes place under the difference of head between the sections A and B and hence the loss of head between the sections A and B will be the same whether the liquid flows through pipe (1) or pipe (2). Thus if D_1, D_2 and L_1, L_2 are the diameters and lengths of the pipes (1) and (2) respectively; then the velocities of flow V_1 and V_2 in the two pipes must be such as to give

$$h_f = \frac{fL_1V_1^2}{2gD_1} = \frac{fL_2V_2^2}{2gD_2} \quad ...(11.15)$$

assuming the same value of f for each parallel pipe. However, if the values of f are different the same may be introduced in Eq. 11.15.

The various problems which may arise for the pipes in parallel may be solved using these equations.

11.13 FLOW THROUGH A BYE-PASS

A bye-pass is the name given to a diversion provided for a main pipeline in which the main line is tapped at some point by a smaller pipe which later returns to the main at another point as shown in Fig. 11.10. Such diversions are usually required to be provided in city water supply mains for purpose of distribution.

Referring Fig. 11.10, if sections 1 and 2 represent the inlet and the outlet of the diversion then the difference of head between sections 1 and 2 will be

$$H_1 - H_2 = \frac{fLV^2}{2gD} = \frac{flv^2}{2gd} + k'\frac{v^2}{2g}$$

where D, d and L, l are the diameters and the lengths of the main and the diversion respectively, V and v are the velocities of flow in the main and the diversion respectively, $k'\left(\frac{v^2}{2g}\right)$ represents the loss of head at the inlet and outlet sections of the diversion; and f is the friction factor which is assumed to be same for both the main and the diversion. Thus

$$\frac{LV^2}{D} = \frac{lv^2}{d} + kv^2 \left(\text{where } k = \frac{k'}{f}\right)$$

or

$$\frac{V^2}{v^2} = \frac{D}{L}\left(\frac{l}{d}+k\right)$$

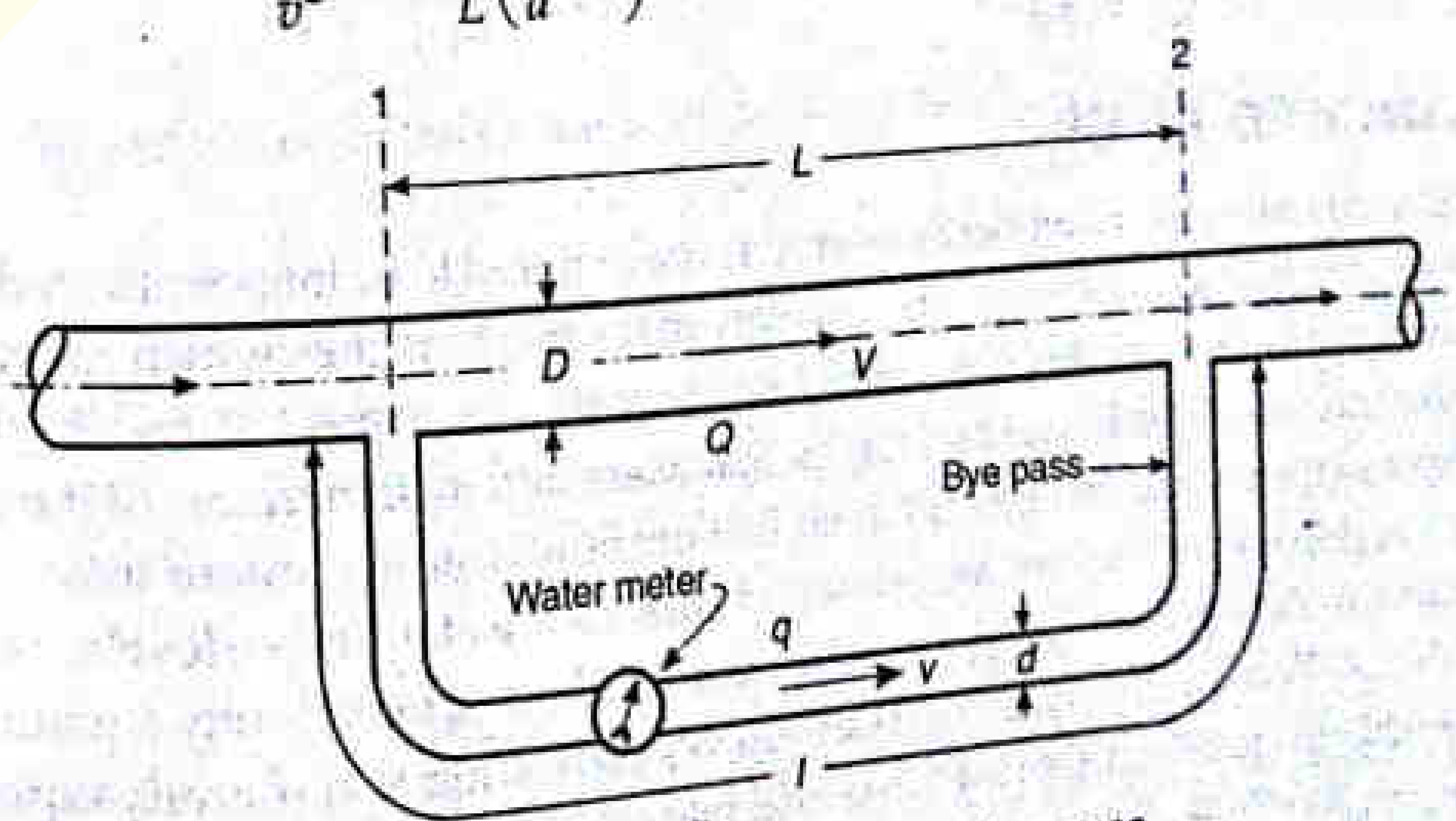


Figure 11.10 *Flow through a bye-pass*

Now if Q and q are the discharges through the main and the diversion respectively, then

$$Q = \frac{\pi D^2}{4} V \text{ and } q = \frac{\pi d^2}{4} v$$

$$\therefore \quad \frac{Q}{q} = \frac{D^2 V}{d^2 v} = \frac{D^2}{d^2}\sqrt{\frac{D}{L}\left(\frac{l}{d}+k\right)}$$

or

$$\frac{Q}{q} = \sqrt{\left(\frac{D}{d}\right)^5 \frac{l+kd}{L}} \quad \text{...(11.16)}$$

and

$$q = \frac{Q+q}{1+\sqrt{\left(\frac{D}{d}\right)^5 \frac{l+kd}{L}}} \quad \text{...(11.17)}$$

In the above analysis it is however assumed that no water is drawn from the main as well as diversion pipes between the sections 1 and 2. Then $(Q + q)$ represents the total discharge in the main before section 1 or after section 2.

A bye-pass may also be used for measuring the discharge passing through city water supply mains which are usually of very big size and for these other discharge measuring devices such as water meters etc., are not available. In such cases a bye-pass or diversion pipe containing a water meter is connected between suitable points on the main pipe, and the discharge passing through the bye-pass is determined from the water meter. The bye-pass is previously calibrated and its discharge coefficient determined earlier. The discharge coefficient K of a bye-pass is defined as the ratio between the total discharge Q_t passing through the main (before a part of it is diverted through the bye-pass) and the discharge q passing through the bye-pass. Thus

$$K = \frac{Q_t}{q} = \frac{Q+q}{q}$$

or

$$Q_t = Kq.$$

11.14 BRANCHED PIPES

In a water supply system often a number of reservoirs are required to be interconnected by means of a pipe system consisting of a number of pipes namely main and branches which meet at a junction. Figure 11.11 shows three reservoirs A, B and C interconnected by pipes 1, 2 and 3 which meet at a junction D. In the problems of this type the lengths, diameters, and friction factors of the different pipes are supposed to be known. Further it is assumed that the flow is steady, minor losses are small and hence neglected and the reservoirs are large enough so that their water surface levels are constant. The three basic equations that are used in order to solve these problems are continuity equation, Bernoulli's equation and Darcy–Weisbach equation. At any junction according to continuity equation the total rate of flow towards the junction is equal to the total rate of flow away from it.

Let D_1, D_2, D_3 be the diameters; L_1, L_2, L_3 be the lengths; Q_1, Q_2, Q_3 be the discharges; V_1, V_2, V_3 be the velocities of flow in the pipes 1, 2 and 3 respectively. Now if (p_d/w) is the pressure head at the junction D; Z_1, Z_2, Z_3 are respectively the heights of the water surfaces in the reservoirs A, B, C; Z_d is the height

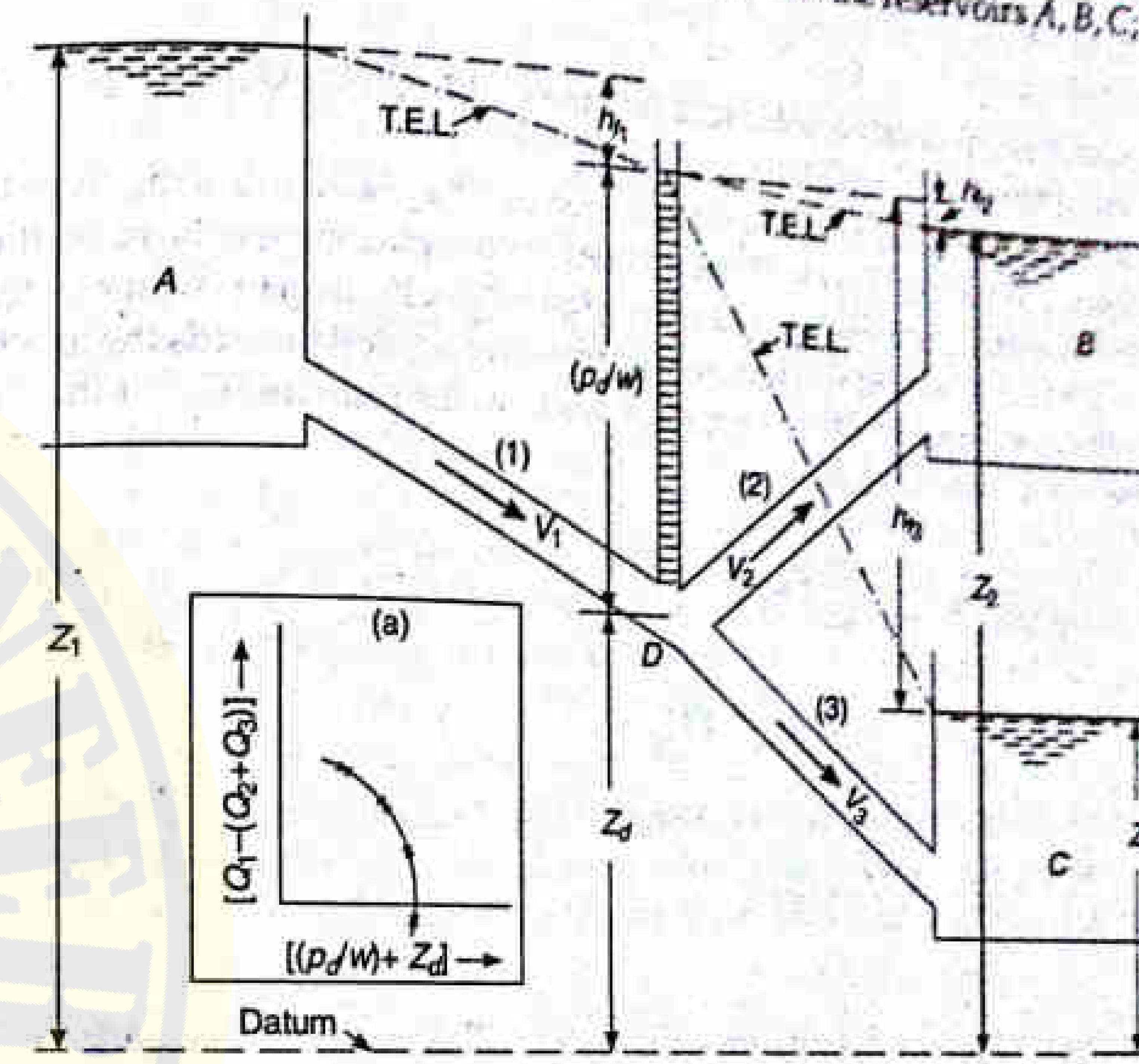


Figure 11.11 *Branched pipes—three interconnected reservoirs*

of the junction D above the assumed datum and h_{f1}, h_{f2}, h_{f3} are the head losses due to friction in the pipes 1, 2 and 3 respectively, then applying Bernoulli's equation between the reservoir A and the junction D, we get

$$Z_1 = \left(\frac{p_d}{w}+Z_d\right)+h_{f1} \quad \text{...(i)}$$

applying Bernoulli's equation between the junction D and the reservoir B, we get

$$\left(\frac{p_d}{w}+Z_d\right) = Z_2+h_{f2} \quad \text{...(ii)}$$

applying Bernoulli's equation between the junction D and the reservoir C, we get

$$\left(\frac{p_d}{w}+Z_d\right) = Z_3+h_{f3} \quad \text{...(iii)}$$

In the above equations $h_{f1} = \frac{fL_1V_1^2}{2gD_1}, h_{f2} = \frac{fL_2V_2^2}{2gD_2}, h_{f3} = \frac{fL_3V_3^2}{2gD_3}$, assuming friction factor f to be same for all the three pipes, and minor losses being neglected.

Further applying continuity equation, we get

$$Q_1 = Q_2 + Q_3$$

$$\frac{\pi}{4}D_1^2V_1 = \frac{\pi}{4}D_2^2V_2 + \frac{\pi}{4}D_3^2V_3 \qquad ...(iv)$$

The above noted four equations involve four unknowns *viz.*, Q_1, Q_2, Q_3 and (p_d/w) which may be calculated by solving these four equations.

In deriving the above equations it has been considered that the water is flowing from the junction D to both the reservoirs B and C. This will happen only if the piezometric head at the junction D, $[(p_d/w + Z_d]$ is greater then Z_2 and Z_3. However, if $[(p_d/w) + Z_d] < Z_2$, then the water instead of flowing from the junction D to the reservoir B, it will flow from the reservoir B towards the junction D. In that case both the reservoirs A and B will be supplying water to the reservoir C and the corresponding equations will then be as noted below:

$$Z_1 = \left(\frac{p_d}{w} + Z_d\right) + h_{f1} \qquad ...(i)$$

$$Z_2 = \left(\frac{p_d}{w} + Z_d\right) + h_{f2} \qquad ...(ii)$$

$$\left(\frac{p_d}{w} + Z_d\right) = Z_3 + h_{f3} \qquad ...(iii)$$

$$Q_1 + Q_2 = Q_3 \qquad ...(iv)$$

On the other hand if $[(p_d/w) + Z_d] < Z_3$, then the water will flow from the reservoir C towards the junction D. In that case both the reservoirs A and C will be supplying water to the reservoir B and equations may then be modified accordingly. Moreover, in the above noted Eqs (i) to (iii), the minor losses have been neglected, but if these losses are not negligible then these may also be considered and the equations (i) to (iii) may be modified accordingly.

There are three types of problems which may arise in such systems of interconnected reservoirs as indicated below:

(a) Given diameters, friction factors and lengths of all the pipes, water surface elevations in the reservoirs A and B and the discharge Q_1 in pipe 1; to determine the elevation of water surface in reservoir C and the discharge Q_2 and Q_3.

This is a simple and direct problem. With the discharge Q_1 known the head loss h_{f1} can be computed and using Eq. (i) the piezometric head at the junction can be determined. Knowing the piezometric head at the junction, Q_2 can be determined by using Eq. (ii), Q_3 can then be determined by using equation (iv). Knowing Q_3 the elevation of water surface in reservoir C can be determined by using equation (iii).

(b) Given diameters, friction factors and lengths of all the pipes, water surface elevations in the reservoirs A and C and the discharge Q_2 in pipe 2; to determine the elevation of water surface in reservoir B and the discharge Q_1 and Q_3.

From Eqs (i) and (iii), $(h_{f1} + h_{f3}) = (Z_1 - Z_3)$. Thus in this case $(h_{f1} + h_{f3})$ is known and also $(Q_1 - Q_3)$ or $(Q_3 - Q_1)$ is known, but a trial solution is needed for this type of problem. There are two methods of solution, viz., (1) assume trial values of Q_1 and Q_3 and compute h_{f1} and h_{f3} and compare these with the

known value of $(h_{f1} + h_{f3})$. Repeat the procedure until the computed value of the sum $(h_{f1} + h_{f3})$ is equal to the known value; (2) assume trial values of h_{f1} and h_{f3} and from these compute the discharge Q_1 and Q_3 and compare these with the known value of $(Q_1 - Q_3)$ or $(Q_3 - Q_1)$. Repeat the procedure until the two values are equal. From either of these two methods the value of the piezometric head at the junction D can be determined. From the given discharge Q_2, h_{f2} can be determined. From these two values the water surface elevation in reservoir B can be found by using Eq. (ii).

(c) Given diameters, friction factors and lengths of all the pipes and the water surface elevations of all the three reservoirs; to determine the discharge Q_1, Q_2 and Q_3 in each of the pipes.

For this case also a trial solution is necessary. Assume a trial value of the piezometric head at the junction and compute h_{f1}, h_{f2} and h_{f3} from the above noted equations and then determine Q_1, Q_2 and Q_3. Now depending on the assumed value of the piezometric head at the junction the direction of flow in pipe 2 may be considered. Thus if the flow in pipe 2 is from the junction to the reservoir B then compute the value of $[Q_1 - (Q_2 + Q_3)]$. By assuming some more trial values of the piezometric head and repeating the same procedure, more values of $[(Q_1 - (Q_2 + Q_3)]$ may be obtained. These values of $[Q_1 - (Q_2 + Q_3)]$ may be plotted against the piezometric head at the junction as shown in Fig. 11.11 (a) and the value of the piezometric head for which $[Q_1 - (Q_2 + Q_3)] = 0$ may be obtained which will be the actual value of the piezometric head at the junction. Knowing the piezometric head at the junction, the values of the discharge Q_1, Q_2 and Q_3 may be determined.

It may, however, be stated that if the direction of flow in pipe 2 is incorrectly assumed, then no solution will be obtained. As such in order to obtain the correct direction of flow in pipe 2 a preliminary trial value of the piezometric head at junction $[(p_d/w) + Z_d]$ equal to Z_2 may be assumed, in which case no flow will occur in pipe 2. The values of Q_1 and Q_3 may then be computed corresponding to this value of the piezometric head at the junction. Now if the computed value of Q_1 is greater than that of Q_3, that is, if the flow towards the junction exceeds that leaving the junction, then the actual piezometric head at the junction is greater than Z_2, and the flow in the pipe 2 will be from the junction to the reservoir B. On the other hand, if Q_1 is less than Q_3 then the actual piezometric head at the junction is less than Z_2, and the flow in pipe 2 will be from the reservoir B towards the junction. Obviously for the later case a plot of $[(Q_1 + Q_2) - Q_3]$ versus the piezometric head at the junction will have to be prepared, and the actual value of the piezometric head at the junction will correspond to $[(Q_1 + Q_2) - Q_3] = 0$, as indicated above.

For solving the problems of categories (b) and (c) indicated above there is another method, which eliminates the trial solution. In this method either discharge or velocity of flow in one of the pipes is assumed to be equal to m times the discharge or velocity of flow in the other pipe, where m is a proportionality factor. Then by substituting these values in the above noted (i) to (iv) equations and solving them simultaneously the value of the factor m is determined, from which the values of the other unknown quantities may be determined. This method has been used in the Illustrative Example 11.12.

11.15 SIPHON

A siphon is a long bent pipe which is used to carry water from a reservoir at a higher elevation to another reservoir at a lower elevation when the two reservoirs are separated by a hill or high level ground in between as shown in Fig.11.12. Since the siphon is laid over the hill or the high level ground, for some length from the entrance section it will rise above the water surface in the upper (or supply) reservoir, and then for the remaining length it will drop down to be connected to the lower reservoir. The rising portion of the siphon is known as the '*inlet leg* (or *inlet limb*), the highest point is known as *summit* and the portion between the summit and the lower reservoir is known as *outlet leg* (or outlet limb). As may be seen in Fig. 11.12 the inlet leg (or inlet limb) of a siphon is usually smaller than the outlet leg (or outlet limb). As the siphon is also a long pipe, the loss of head due to friction will be very large and hence the other minor losses may be neglected. Further the length of the siphon may be taken as the length of its horizontal projection. Hence the hydraulic grade line and the energy grade line (or total energy line) for a siphon, as shown in Fig. 11.12, may also be obtained in the same manner

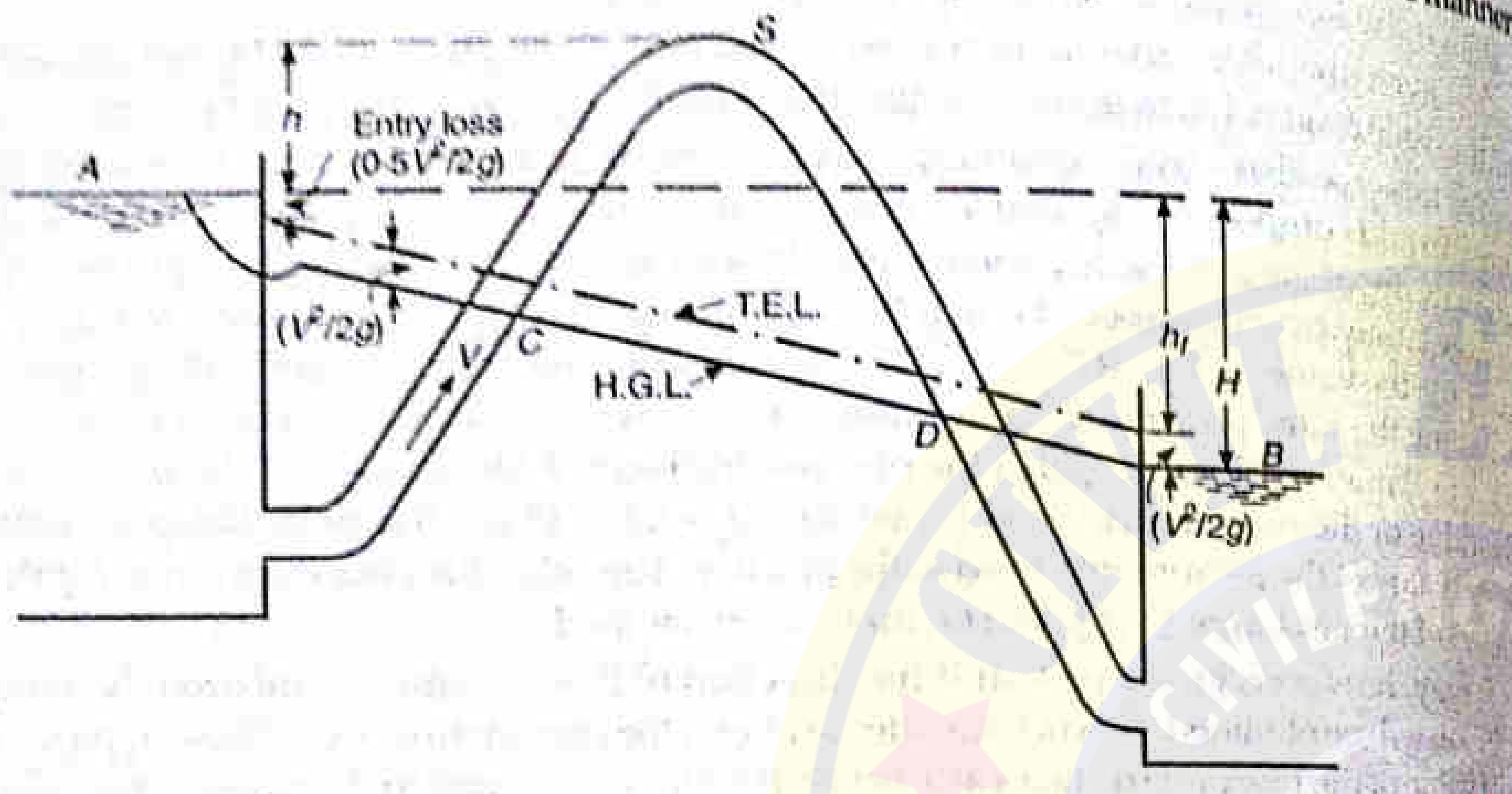


Figure 11.12 *Siphon*

as in the case of an ordinary long pipe. It will be seen from Fig. 11.12 that the hydraulic grade line cuts the siphon at points *C* and *D*, so that some portion of the siphon is above the hydraulic grade line. The vertical distance between the hydraulic grade line and the pipe centre line represents the pressure head at any section. If the hydraulic grade line is above the centre line of the pipe then the pressure is above atmospheric; and if the hydraulic grade line is below the centre line of the pipe, the pressure is negative or below atmospheric. Thus for the portion of the siphon below points *C* and *D* the pressure will be above atmospheric and at points *C* and *D* the pressure of the water flowing in the siphon is equal to atmospheric pressure. For the portion of the siphon between *C* and *D* the pressure will be below atmospheric. As the highest point of the siphon above the hydraulic grade line is the summit *S*, the water pressure at this point is the least. Further as the vertical distance between the summit of the siphon and the hydraulic grade line increases, the water pressure at this point reduces. Theoretically this pressure may be reduced to – 10.3 m of water (if the atmospheric pressure is 10.3 m of water) or absolute vacuum, because this limit would correspond to a perfect vacuum and the flow would stop. However, in practice if the pressure is reduced to about 2.5 m of water absolute or 7.8 m of water vacuum the dissolved air or other gases would come out of the solution and collect at the summit of the siphon in sufficient quantity to form an air-lock, which will obstruct the continuity of the flow, (or the flow will completely stop). A similar trouble may also be caused by the formation of the water vapour in the region of low pressure. Therefore the siphon should be laid so that no section of the pipe will be more than 7.8 m above the hydraulic grade line at the section. Moreover, in order to limit the reduction of the pressure at the summit the length of the inlet leg of the siphon is also required to be limited. This is so because as indicated below, if the inlet leg is very long a considerable loss of head due to friction is caused, resulting in further reduction of the pressure at the summit.

Assuming the siphon to run full, then applying Bernoulli's equation between point *A* and *B*, we get

$$H = 0.5\frac{V^2}{2g} + \frac{V^2}{2g} + \frac{fLV^2}{2gD}$$

or

$$H = \frac{V^2}{2g}\left[1.5 + \frac{fL}{D}\right] \quad ...(11.18)$$

where *V* is the velocity of flow, *L* and *D* are the total length and the diameter of the siphon.

Equation 11.18 may be used to determine *H* if the discharge *Q* is known; or it may be used to determine *Q* if *H* is given. In the first case the Reynolds number of the flow being known the value of *f* may be determined from Moody's diagram. But in the second case the value of *f* will be required to be assumed first and then checked for the computed discharge.

In order to determine the maximum value of *h*, the height of the summit above the water surface in the upper reservoir, Bernoulli's equation is applied between the points *A* and *S*, so that

$$\frac{p_a}{w} = \frac{p_s}{w} + \frac{V^2}{2g} + \frac{0.5V^2}{2g} + \frac{flV^2}{2gD} + h$$

where p_a is the atmospheric pressure, p_s is the absolute pressure at the summit, and *l* is the length of the inlet leg of the siphon.

or

$$h = \frac{p_a}{w} - \frac{p_s}{w} - \frac{V^2}{2g}\left[1.5 + \frac{fl}{D}\right] \quad ...(11.19)$$

As far as possible, the value of *h* lower than that given by Eq. 11.19 should be used. Further for a given value of *h*, the length of the inlet leg *l* should be so adjusted that pressure at the summit is not reduced below the permissible limit as indicated above.

11.16 LOSS OF HEAD DUE TO FRICTION IN TAPERING PIPE

A tapering pipe is the one which has a gradually varying cross-sectional area as shown in Fig. 11.13. Since the loss of head due to friction depends on the velocity of flow, which in the case of tapering pipe varies from section to section, the expression for the loss of head due to friction in such a pipe may be obtained by integration, as indicated below.

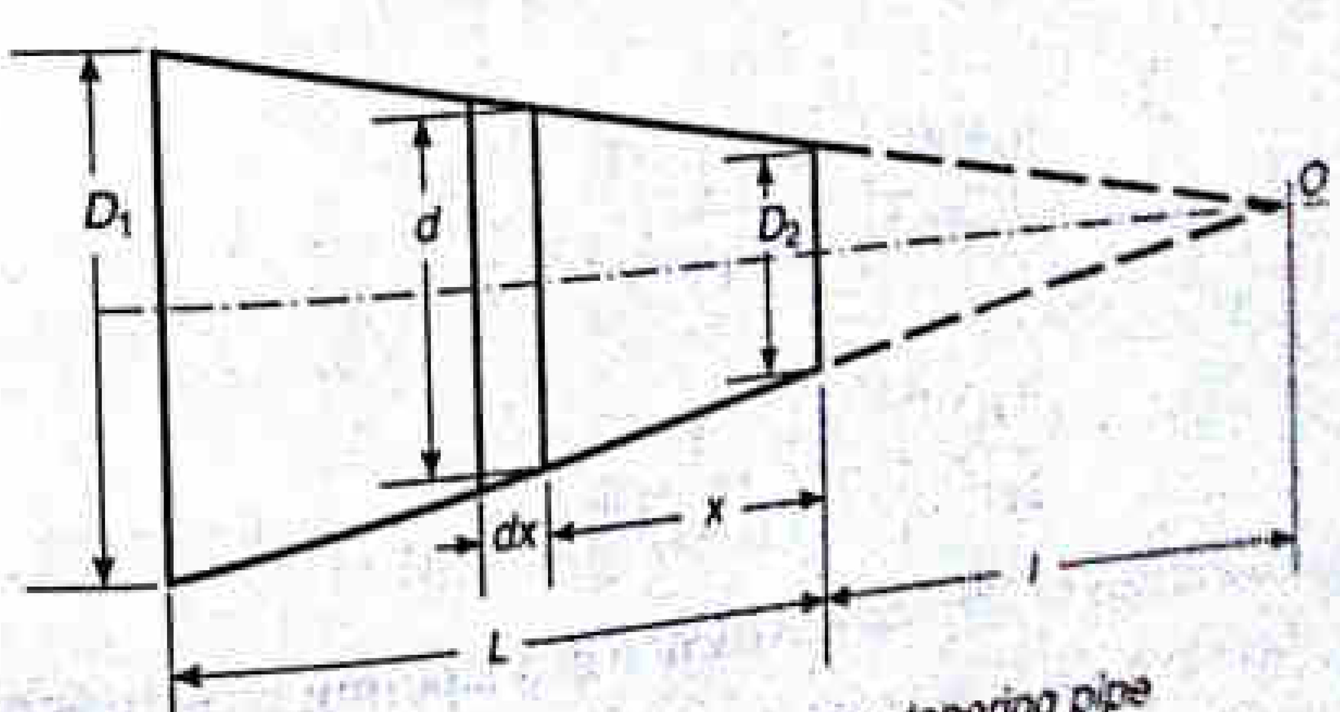


Figure 11.13 *Flow through a tapering pipe*

Let D_1 and D_2 be the diameters at the ends of a tapering pipe of length L, and Q be the discharge through the pipe. Now if O is the apex as shown in Fig. 11.13 and l is the distance of the apex from the end of the pipe with smaller diameter, then from similar triangles, we obtain

$$\frac{D_1}{D_2} = \frac{L+l}{l}, \text{ or } \frac{D_1}{D_2} = 1+\frac{L}{l}$$

$$\therefore \qquad l = \frac{D_2 L}{D_1 - D_2} \qquad \text{...(11.20)}$$

Consider a small portion of the pipe of length dx, at a distance x from the smaller end of the pipe. Let the diameter of the pipe at this section be d, then again from the similar triangles, we have

$$\frac{D_2}{d} = \frac{l}{l+x}; \text{ or } d = D_2\left(\frac{l+x}{l}\right)$$

Now the loss of head due to friction for the small length dx of the pipe is given by

$$dh_f = \frac{fdxv^2}{2gd}$$

where v is the velocity of flow at this section given by

$$v = \frac{Q}{(\pi d^2/4)}$$

$$\therefore \qquad dh_f = \frac{16fQ^2dx}{2g\pi^2d^5}$$

$$= \frac{16fQ^2}{2g\pi^2}\frac{l^5dx}{D_2^5(l+x)^5}$$

By integrating both sides of the above equation, we get

$$h_f = \frac{16fQ^2l^5}{2g\pi^2D_2^5}\int_0^L\frac{dx}{(l+x)^5}$$

$$= \frac{16fQ^2l^5}{2g\pi^2D_2^5}\left[-\frac{1}{4}\frac{1}{(l+x)^4}\right]_0^L$$

$$= \frac{4fQ^2l^5}{2g\pi^2D_2^5}\left[\frac{1}{l^4}-\frac{1}{(L+l)^4}\right] \qquad \text{...(11.21)}$$

Equation 11.21 gives the loss of head due to friction in a tapering pipe in which the value of l is given by Eq. 11.20.

11.17 LOSS OF HEAD DUE TO FRICTION IN A PIPE WITH SIDE TAPPINGS

If a pipe of uniform diameter is provided with side tappings along its length, through which the liquid may be withdrawn from the pipe, then the velocity of flow will vary from section to section along the length of the pipe. The loss of head due to friction, which depends on the velocity of flow, will also then vary and it may be obtained by integration as described below.

If the side tappings are uniformly and closely spaced and are assumed to remove the liquid at a uniform rate of q per unit length of the pipe then the rate of flow at a distance x from the inlet of the pipe is $(Q-qx)$, when Q is the initial rate of flow at the inlet end. Now for a uniform cross-sectional area A of the pipe

$$\frac{Q}{V} = A = \frac{(Q-qx)}{v}$$

or

$$v = V\left(1-\frac{qx}{Q}\right)$$

where V and v are respectively the velocities of flow at the inlet and at section x distance from the inlet. The above equation indicates that in this case the velocity of flow varies linearly along the length of the pipe.

Considering a small portion of the pipe of length dx at a distance x from the inlet, the loss of head due to friction for the small length dx of the pipe is given by

$$dh_f = \frac{fdxv^2}{2gD}$$

where D is the diameter of the pipe. Substituting the value of v

$$dh_f = \frac{f}{D}\frac{V^2}{2g}\left(1-\frac{qx}{Q}\right)^2 dx$$

By integrating the above expression from $x=0$ to $x=L$, the loss of head due to friction over the entire length L of the pipe is obtained as

$$h_f = \frac{f}{D}\frac{V^2}{2g}\int_0^L\left(1-\frac{qx}{Q}\right)^2 dx$$

or

$$h_f = \frac{f}{D}\frac{V^2}{2g}\left[-\frac{Q}{3q}\left(1-\frac{qx}{Q}\right)^3\right]_0^L$$

$$\therefore \qquad h_f = \frac{f}{D}\frac{V^2}{2g}\frac{Q}{3q}\left[1-\left(1-\frac{qL}{Q}\right)^3\right] \qquad \text{...(11.22)}$$

where f is assumed constant.

If the end of the pipe is closed so that $(Q-qL)=0$, then Eq. 11.22 becomes

$$h_f = \frac{f}{D}\frac{V^2}{2g}\frac{Q}{3q}$$

$$= \frac{f}{D}\frac{V^2}{2g}\frac{L}{3} \qquad \text{...(11.23)}$$

Equation 11.23 shows that the total loss of head in this case is one-third of what it would be if the inlet discharge Q remains constant and no liquid is drawn off along the length of the pipe.

11.18 TIME OF EMPTYING A RESERVOIR THROUGH PIPE

Let a reservoir or tank be emptied by means of a long pipe of length L and diameter D, which is assumed to be discharging freely in the atmosphere. Let the area of water surface in the reservoir be constant equal to A, and the height of the water level above the outlet of the pipe be H_1. It is required to find the time taken to lower the water level in the reservoir from H_1 to H_2 above the pipe outlet.

Let at any instant the water level be h above the outlet of the pipe, when the velocity of flow in the pipe be v. Let the water level fall by a small amount dh in time dt. Then the quantity of water flowing from the reservoir equals the quantity passing along the pipe in time dt; thus

$$-A\,dh = \frac{\pi D^2}{4} v\,dt \qquad \text{...(i)}$$

The negative sign is being introduced because as time increases the water level in the reservoir is decreasing.

Again $$h = \frac{0.5v^2}{2g} + \frac{v^2}{2g} + \frac{fLv^2}{2gD}$$

from which $$v = \sqrt{\frac{2gh}{1.5 + \frac{fL}{D}}}$$

Substituting this value of v in Eq. (i)

$$-A\,dh = \frac{\pi D^2}{4}\sqrt{\frac{2gh}{\left(1.5 + \frac{fL}{D}\right)}}\,dt$$

$$\therefore \quad dt = \frac{-4A\sqrt{1.5 + \frac{fL}{D}}}{\pi D^2\sqrt{2g}}\frac{dh}{\sqrt{h}}$$

By integrating the above equation, the total time

$$t = -\frac{4A\sqrt{1.5 + \frac{fL}{D}}}{\pi D^2\sqrt{2g}}\int_{H_1}^{H_2}\frac{dh}{\sqrt{h}}$$

$$t = \frac{8A\sqrt{1.5 + \frac{fL}{D}}}{\pi D^2\sqrt{2g}}\left(H_1^{1/2} - H_2^{1/2}\right) \qquad \text{...(11.24)}$$

If the pipe, instead of discharging freely in the atmosphere, is connected to a reservoir, then the time required to reduce the difference between the water levels in the two reservoirs from H_1 to H_2 may be determined as follows:

Let A_1 and A_2 be the cross-sectional areas of the two tanks A and B, connected by a pipe of length L and diameter D. Let at any instant the difference in the water levels in the two reservoirs be h and in time dt the water level in the tank A fall by an amount dH, Fig. 11.14.

Then, the water level will rise in tank B by an amount $\left(dH\frac{A_1}{A_2}\right)$. Now if dh is the change in the difference in head causing the flow, then

$$dh = dH + dH\frac{A_1}{A_2}$$

or $$dH = \frac{dh}{\left(1 + \frac{A_1}{A_2}\right)} \qquad \text{...(i)}$$

Further the quantity of water flowing from tank A equals the quantity flowing along the pipe in time dt, thus

$$-A_1 dH = \frac{\pi D^2}{4} v\,dt \qquad \text{...(ii)}$$

Further $$h = 0.5\frac{v^2}{2g} + \frac{v^2}{2g} + \frac{fLv^2}{2gD}$$

$$v = \sqrt{\frac{2gh}{1.5 + \frac{fL}{D}}} \qquad \text{...(iii)}$$

Introducing Eqs. (i) and (iii) in (ii), we have

$$-A_1\frac{dh}{\left(1 + \frac{A_1}{A_2}\right)} = \frac{\pi D^2}{4}\sqrt{\frac{2gh}{1.5 + \frac{fL}{D}}}\,dt$$

or $$dt = -\frac{4A_1A_2\left(\sqrt{1.5 + \frac{fL}{D}}\right)dh}{\pi D^2(A_1 + A_2)\sqrt{2g}\,\sqrt{h}}$$

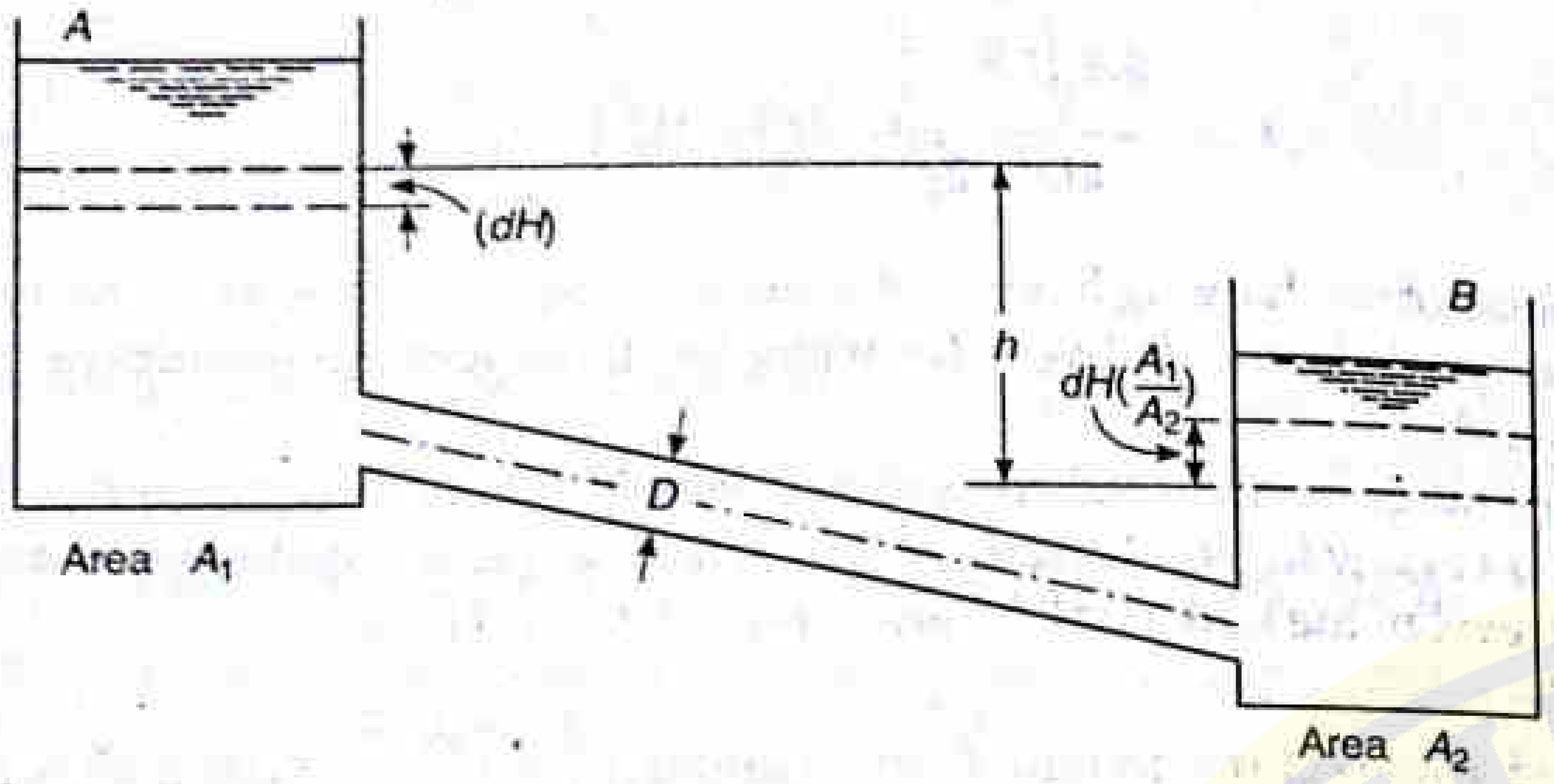


Figure 11.14 *Flow from one reservoir to other through a long pipe*

By integrating the above equation the total time

$$t = -\frac{4A_1A_2\sqrt{1.5+\frac{fL}{D}}}{\pi D^2(A_1+A_2)\sqrt{2g}}\int_{H_1}^{H_2}\frac{dh}{\sqrt{h}}$$

or

$$t = \frac{8A_1A_2\sqrt{1.5+\frac{fL}{D}}}{\pi D^2(A_1+A_2)\sqrt{2g}}\left(H_1^{1/2}-H_2^{1/2}\right) \quad ...(11.25)$$

11.19 TRANSMISSION OF POWER THROUGH PIPES

The pipes carrying water under pressure from one point to other may be utilised to transmit hydraulic power. Such transmission of power through pipes carrying water or other liquids is commonly used for the working of several hydraulic machines. The hydraulic power transmitted by a pipe however depends on the discharge passing through the pipe and the total head of the water. As the water flows along the pipe it will be subjected to frictional resistance causing a loss of head due to friction. Thus if H is the total head supplied at the entrance to the pipe and h_f is the loss of head due to friction then the head available at the outlet of the pipe (neglecting the minor losses)

$$= (H - h_f)$$

Further if Q is the discharge through the pipe, V is the velocity of flow and L and D are the length and the diameter of the pipe respectively, then

$$h_f = \frac{fLV^2}{2gD}$$

and

$$Q = \frac{\pi D^2}{4}V$$

The power (or energy per second) available at the outlet of the pipe is

$$P = \text{weight of water per second} \times \text{head available}$$

or

$$P = wQ(H-h_f)$$

or

$$P = w\left(\frac{\pi D^2}{4}\times V\right)\left(H-\frac{fLV^2}{2gD}\right) \quad ...(11.26)$$

For a given pipe supplied with a given total head H at the entrance, the condition for the maximum power transmitted through the pipe may be obtained by differentiating Eq. 11.26 with respect to V and equating it to zero; thus

$$\frac{dP}{dV} = w\left(\frac{\pi D^2}{4}\right)\left(H-\frac{3fLV^2}{2gD}\right) = 0$$

or

$$H-\frac{3fLV^2}{2gD} = 0$$

or

$$H - 3h_f = 0$$

$$\therefore \quad H = 3h_f \quad ...(11.27)$$

That is, the power transmitted through a pipe is maximum when the loss of head due to friction is one-third of the total head supplied.

The efficiency of power transmission through pipe may be expressed as

$$\eta = \frac{H-h_f}{H} \quad ...(11.28)$$

Thus corresponding to the maximum power transmitted the efficiency of power transmission is η = (2/3) or 66.7%. It may however be observed that the efficiency corresponding to maximum power transmitted is not maximum. This is so because the efficiency of power transmission would be maximum (equal to 100%), if $h_f = 0$ and as h_f increases the efficiency of power transmission decreases.

11.20 FLOW THROUGH NOZZLE AT THE END OF PIPE

A nozzle is a gradually converging short tube which is fitted at the outlet end of a pipe for the purpose of converting the total energy of the flowing water into velocity energy. As such nozzles are used where higher velocities of flow are required to be developed. For example in practice higher velocity of flow is required for extinguishing fire, which is obtained by fitting a nozzle at the end of hose. Further for the impulse type of turbines such as Pelton wheel turbines, it is required to convert whole of the hydraulic energy into kinetic (or velocity) energy and the same is obtained by fitting a nozzle at the end of the pipe (or penstock as is generally called).

Let a nozzle be fitted at the end of a pipe connected to a reservoir with its water level at a height H above the centre line of the nozzle. Let D and L be the diameter and the length of the pipe (upto the base of the nozzle) respectively, and V be the velocity of flow in the pipe. Let d be the diameter of the nozzle at the outlet end and v be the velocity of the issuing jet from nozzle. Thus if Q is the discharge passing through the pipe, then by continuity equation

$$Q = AV = av$$

or
$$\frac{V}{v} = \frac{a}{A} \quad ...(i)$$

or
$$Q = \frac{\pi D^2}{4} V = \frac{\pi d^2}{4} v$$

Further neglecting the minor losses and the loss of energy in the nozzle, we have

$$H = \frac{fLV^2}{2gD} + \frac{v^2}{2g} \quad ...(11.29)$$

Substituting the value of V from (i)

$$H = \frac{fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g} + \frac{v^2}{2g}$$

or
$$v = \sqrt{\frac{2gH}{1+\frac{fL}{D}\frac{a^2}{A^2}}} \quad ...(11.30)$$

The kinetic energy of the jet issuing from the nozzle

$$= \frac{1}{2}\frac{wQ}{g}v^2 = \frac{wav^3}{2g}$$

Efficiency of power transmission

$$\eta = \frac{\text{head transmitted}}{\text{head supplied}}$$

$$= \frac{\frac{v^2}{2g}}{H} = \frac{v^2}{2gH} = \left[\frac{1}{1+\frac{fL}{D}\frac{a^2}{A^2}}\right]$$

(a) Maximum Power Available from a Nozzle. Power transmitted by the jet issuing from a nozzle is

$$P = w(av)\frac{v^2}{2g}$$

Substituting the value of $\left(\frac{v^2}{2g}\right)$ from Eq. 11.29, we get

$$P = w(av)\left[H - \frac{fLV^2}{2gD}\right]$$

Substituting the value of V from (i), we get

$$P = w(av)\left[H - \frac{fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g}\right]$$

or
$$P = wa\left[Hv - \frac{fL}{D}\frac{a^2}{A^2}\frac{v^3}{2g}\right]$$

Again the condition for the maximum power transmitted by the nozzle may be obtained by differentiating the above expression with respect to v and equating it to zero; thus

$$\frac{d(P)}{dv} = wa\left[H - \frac{3fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g}\right] = 0$$

or
$$H - \frac{3fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g} = 0$$

or
$$H = \frac{3fLV^2}{2gD} = 3h_f \quad ...(11.31)$$

That is, the power transmitted by a nozzle is maximum when the head supplied is equal to three times the loss of head due to friction in the pipe.

(b) Diameter of the Nozzle for Transmitting Maximum Power. For maximum transmission of power, from Eq. 11.31

$$H = 3h_f = \frac{3fLV^2}{2gD}$$

Substituting for H from Eq. 11.29, we get

$$\frac{fLV^2}{2gD} + \frac{v^2}{2g} = \frac{3fLV^2}{2gD}$$

or
$$\frac{fLV^2}{2gD} = \frac{1}{2}\times\frac{v^2}{2g}$$

Substituting the value of V from (i), we get

$$\frac{fL}{2gD}\frac{a^2}{A^2}v^2 = \frac{1}{2}\times\frac{v^2}{2g}$$

or
$$\frac{2fL}{D} = \frac{A^2}{a^2}$$

$$\therefore \quad \frac{A}{a} = \sqrt{\frac{2fL}{D}} \quad \text{...(11.32)}$$

Equation 11.32 gives the ratio between the areas of the supply pipe and the nozzle for maximum power transmission.

11.21 WATER HAMMER IN PIPES

When the water flowing in a long pipe is suddenly brought to rest by closing the valve or by any similar cause, there will be a sudden rise in pressure due to the momentum of the moving water being destroyed. This causes a wave of high pressure to be transmitted along the pipe which creates noise known as *knocking*. This phenomenon of sudden rise in pressure in the pipe is known as *water hammer* or *hammer blow*. The rise in pressure in some cases may be so large that the pipe may even burst and therefore it is essential to take into account this pressure rise in the design of the pipes. The magnitude of the pressure rise depends on the speed at which the valve is closed, the velocity of flow, the length of the pipe, and the elastic properties of the pipe material as well as that of the flowing fluid.

The valve may be closed either gradually or instantaneously and according to the nature of the closure of the valve the expressions for calculating the pressure head due to water hammer may be developed as indicated below.

1. Gradual Closure of Valve. Consider a long pipe of length L and of cross-sectional area a, carrying liquid flowing with a velocity V. Due to the closing of the valve, let the liquid be brought to rest in time t seconds. The total mass of liquid contained in the pipe is $\frac{w}{g}(a \times L)$.

It is however assumed that the rate of closure of the valve is so adjusted that the liquid column in the pipe is brought to rest with a uniform retardation; from an initial velocity V to O in time t seconds.

Thus rate of retardation $= \frac{V}{t}$.

The axial force available for producing retardation

$$= \text{inertia pressure} \propto \text{cross-sectional area}$$
$$= (p_i \times a)$$

But

$$\text{force} = \text{mass} \times \text{acceleration (or retardation)}$$

$$\therefore \quad p_i \times a = \frac{waL}{g} \times \frac{V}{t} \quad \text{...(11.33)}$$

$$p_i = \frac{wLV}{gt} \quad \text{...(11.34)}$$

$$\text{Inertia head} \quad = \frac{p_i}{w} = \frac{LV}{gt}$$

2. Instantaneous Closure of Valve. Equation 11.34 shows that if the time of closure t is zero, the inertia head should rise to infinity. But the experimental observations indicate that even for a very rapid closure of the valve, the pressure rise is quite finite and measurable. It may therefore be concluded that in the derivation of Eq. 11.34 certain essential factors have been omitted. These factors are (i) the compressibility of the liquid; and (ii) the elasticity of the pipe material; and both these factors make it impossible for the liquid column to be stopped instantaneously even if it was possible to close the valve instantaneously.

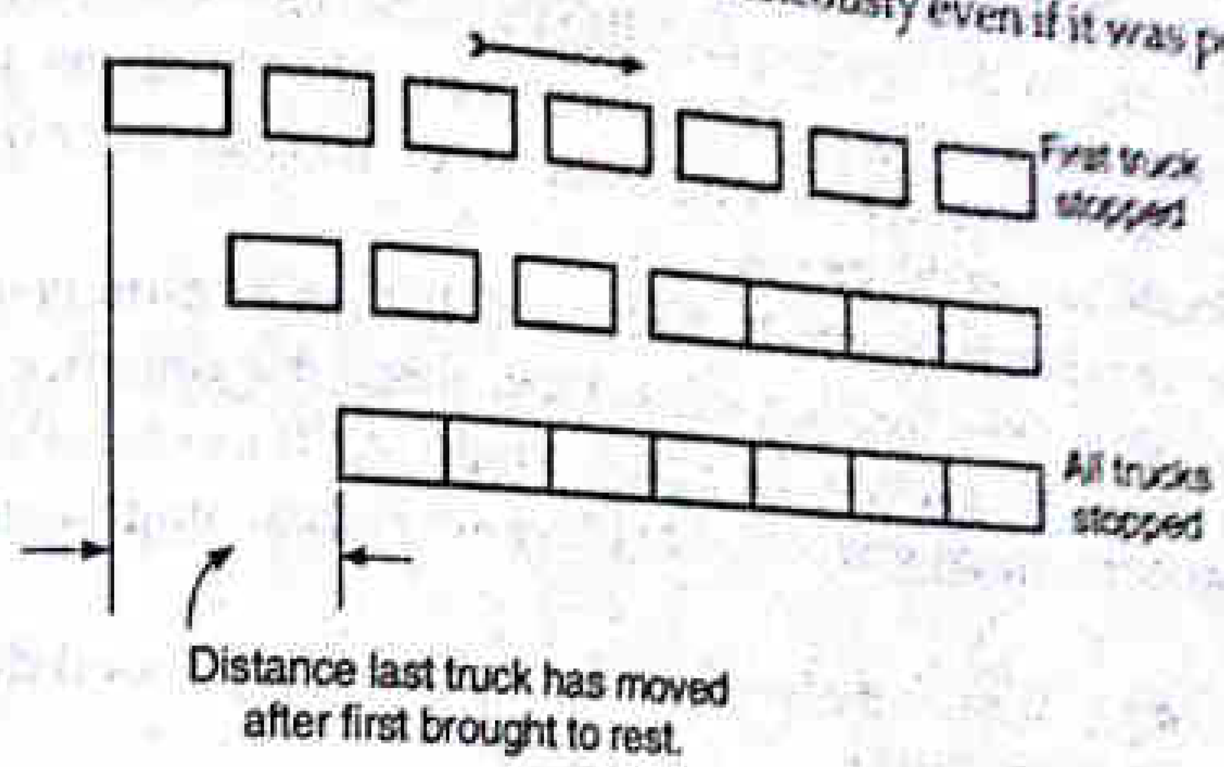


Figure 11.15 *Stopping of a loosely coupled train*

This may however be explained by means of a simple example. As shown in Fig. 11.15, if the head of a loosely coupled trucks is suddenly checked the following trucks still keep on moving until all the gap between them has been covered and only then the whole train will come to rest. That means the last truck will move through a certain distance even after the first truck has stopped, before it also finally comes to rest.

With this analogy, the actual sequence of what happens in a pipe, when the valve at its outlet end is instantaneously closed, may be studied. Fig. 11.16 (a) shows the normal flow condition of the liquid in the pipe with normal velocity V, under normal pressure p, at the moment just before the valve at end B is instantaneously shut. Since the friction head will be quite small in comparison with the inertia head, it has been neglected. Fig. 11.16 (b) shows the condition immediately after the valve at B is

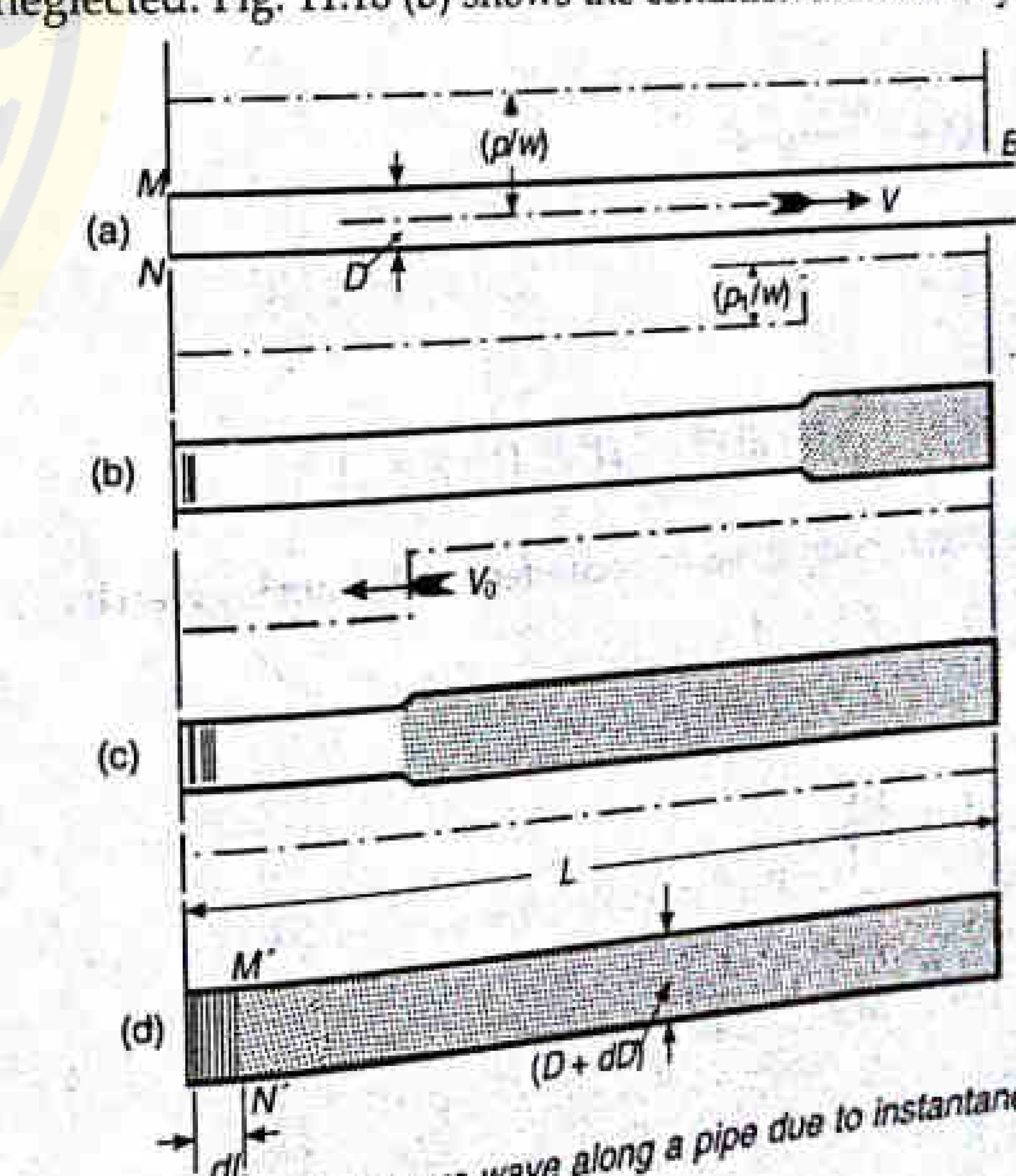


Figure 11.16 *Transmission of pressure wave along a pipe due to instantaneous closure of valve*

closed. Thus a wave of inertia pressure, as shown by the step in the hydraulic grade line, has begun to travel with velocity V_0 in the upstream direction, which results in compressing the water and also expanding the pipe. Meanwhile the liquid on the left of the advancing wave continues to move on as though nothing has happened to it. Figure 11.16 (c) shows that the wave has advanced further; and Fig. 11.16 (d) shows that the liquid in the entire pipe is at rest under a pressure $(p+p_i)$; the whole of the pipe having been distended under this pressure.

Thus if dt is the time required to stop the liquid column, then in this interval of time a transverse plane MN at the end of the column has been able to move through a distance dL to a new position $M'N'$; in other words, the compression of the liquid column together with the increase in the pipe diameter have made space for an amount of liquid dQ (shown by hatching) to enter the pipe after the valve has been shut.

Let dq_c = the volume by which the liquid is compressed due to pressure p_i

dq_e = the additional volume provided by the stretching of the pipe walls under pressure p_i

dD = the resulting increase in pipe diameter

dt = time for pressure wave to traverse the pipe

T = the wall thickness of pipe

E = Young's modulus for the material of the pipe

K = Bulk modulus for the liquid

f_t = tensile hoop stress in pipe walls due to pressure P_i

$\frac{1}{m}$ = Poisson's ratio $\quad \frac{p_i}{K}\frac{\pi}{4}$

Now
$$dQ = \frac{\pi D^2}{4} \times dL$$

$$dq_c = \quad D^2 L$$

and
$$dq_e = \left[\frac{\pi}{4}(D+dD)^2 - \frac{\pi}{4}D^2\right] L\,\Omega\,\pi D \frac{dD}{2} L$$

The strain in the direction of hoop stress or circumferential strain is given by

$$\frac{dD}{D} = \frac{f_t}{E}\left(1-\frac{1}{2m}\right)$$

as hoop stress
$$f_t = \frac{p_i D}{2T}$$

$$\frac{dD}{D} = \frac{p_i D}{2TE}\left(1-\frac{1}{2m}\right)$$

Therefore
$$dD = \frac{p_i D^2}{2TE}\left(1-\frac{1}{2m}\right)$$

and
$$dq_e = \frac{p_i D^2}{2TE}\left(1-\frac{1}{2m}\right) \times \frac{\pi DL}{2}$$

But
$$dQ = dq_c + dq_e$$

Thus by substituting the values noted above, we get

$$\frac{\pi D^2}{4} \times dL = \frac{p_i}{K}\frac{\pi}{4}D^2 L + \frac{p_i D^2}{2TE}\left(1-\frac{1}{2m}\right) \times \frac{\pi DL}{2}$$

or
$$dL = p_i L\left[\frac{1}{K} + \frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]$$

But
$$dL = V\,dt$$

$\therefore$
$$p_i = \frac{V dt}{L\left[\frac{1}{K} + \frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]} \quad ...(i)$$

This expression still contains one unknown, dt, but it can be eliminated by using Eq. 11.33 which is now applicable with time dt representing the time to bring the liquid column to rest. Thus

$$p_i = \frac{wLV}{g dt}$$

or
$$dt = \frac{wLV}{g p_i} \quad ...(ii)$$

Substituting this value in Eq. (i), we get

$$p_i = \frac{wV^2}{g p_i\left[\frac{1}{K} + \frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}$$

or inertia pressure,
$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K} + \frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}} \quad ...(11.35)$$

If elasticity of the pipe material is not considered, then the term $\frac{D}{TE}\left(1-\frac{1}{2m}\right)$ may be neglected. The equation then becomes

$$p_i = V\sqrt{\frac{wK}{g}} \quad ...(11.36)$$

Equation 11.36 therefore represents the maximum pressure rise for instantaneous closure of valve. It may, however, be noted that maximum pressure rise is independent of the dimensions of the pipe, but it depends only on the initial velocity of flow of liquid and its physical characteristics.

The velocity of the compression wave V_0 is represented by

$$V_0 = \frac{L}{dt} = \sqrt{\frac{g}{w\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}} \quad ...(iii)$$

Again if the term $\frac{D}{TE}\left(1-\frac{1}{2m}\right)$ is neglected then $V_0 = \sqrt{(gK/w)} = \sqrt{K/\rho}$, which is identical with the velocity of the propagation of sound wave in the liquid.

By introducing Eq. (iii) in Eq. 11.35 the following simple expression for the inertia head may be obtained

$$h_i = \frac{p_i}{w} = \frac{VV_0}{g} \quad ...(11.37)$$

This is often known as the 'Allievi formula'.

Further as an approximation if the Poisson's ratio is not considered then the circumferential strain is given by

$$\frac{dD}{D} = \frac{f_t}{E}$$

or

$$\frac{dD}{D} = \frac{p_i D}{2TE}$$

or

$$dD = \frac{p_i D^2}{2TE}$$

and

$$dq_e = \frac{p_i D^2}{2TE} \times \frac{\pi DL}{2}$$

Thus by substituting this value of dq_e in the above equation, we get

$$\frac{\pi D^2}{2} \times dL = \frac{p_i}{K}\frac{\pi}{4}D^2L + \frac{p_i D^2}{2TE} \times \frac{\pi DL}{2}$$

or

$$dL = p_i L\left[\frac{1}{K}+\frac{D}{TE}\right]$$

By adopting the same procedure as indicated earlier the expression for the inertia pressure is obtained as

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K}+\frac{D}{TE}\right)}} \quad ...(11.38)$$

Again if the elasticity of the pipe material is not considered, then the term (D/TE) may be neglected. Equation 11.38 then becomes same as Eq. 11.36.

11.22 PIPE NETWORKS

A group of interconnected pipes forming several loops or circuits as shown in Fig. 11.17 is called a network of pipes. Such networks of pipes are commonly used for municipal water distribution systems in cities. The main problem in a pipe network is to determine the distribution of flow through the various pipes of the network such that all the conditions of flow are satisfied and all the circuits are then balanced. The conditions to be satisfied in any network of pipes are as follows:

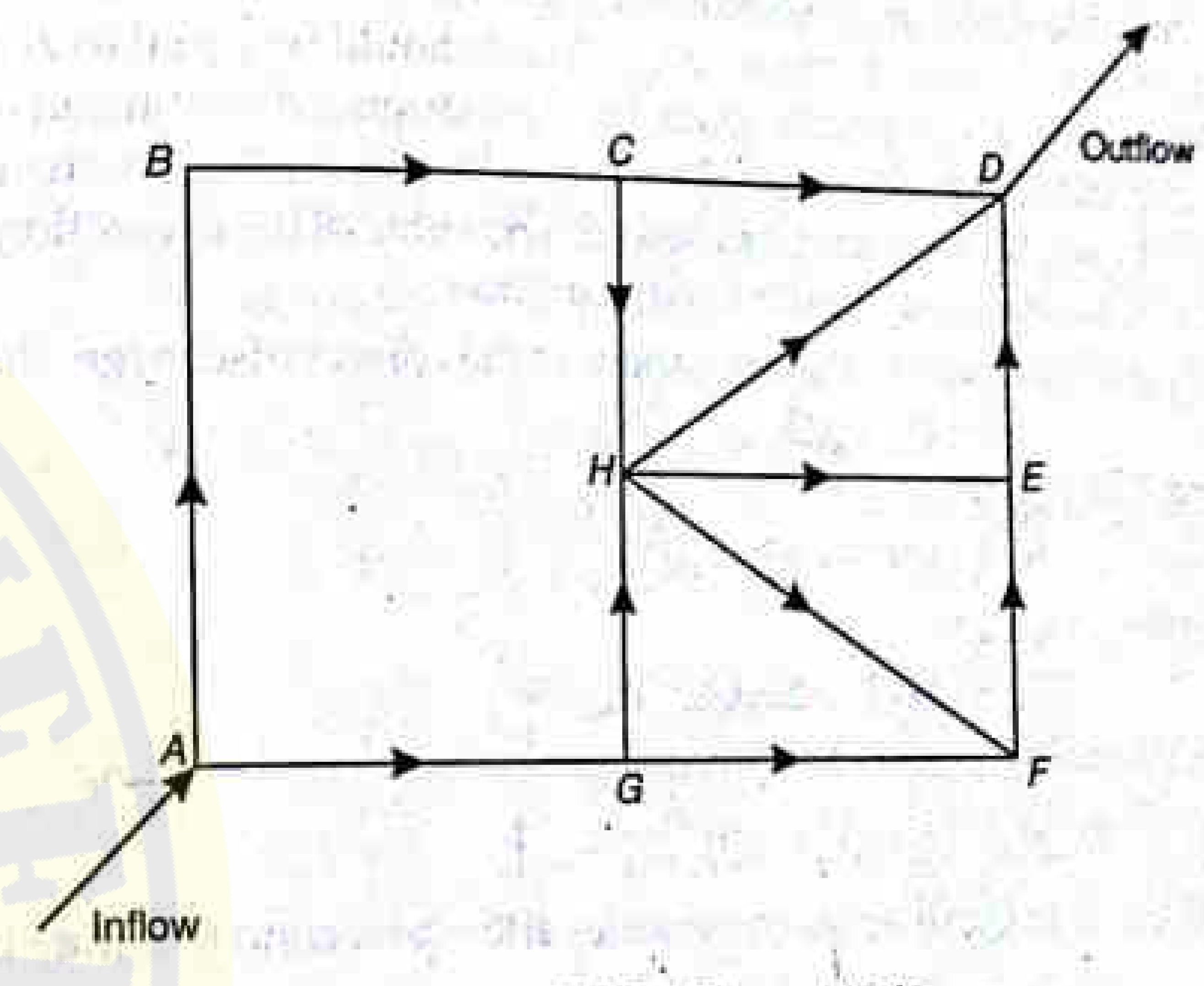


Figure 11.17 *Pipe network*

(1) According to the principle of continuity the flow into each junction must be equal to the flow out of the junction. For example at junction A, the inflow must be equal to the flow through AB and AG.

(2) In each loop, the loss of head due to flow in clockwise direction must be equal to the loss of head due to flow in anticlockwise direction. For example in the loop $ABCHG$ the sum of the head losses due to flow in AB, BC and CH (clockwise flow) must be equal to the sum of the head losses due to flow in AG and GH (anticlockwise flow).

(3) The Darcy–Weisbach equation must be satisfied for flow in each pipe. Minor losses may be neglected if the pipe lengths are large. However if the minor losses are large, they may be taken into account by considering them in terms of the head loss due to friction in equivalent pipe lengths. According to Darcy–Weisbach equation the loss of head h_f through any pipe discharging at the rate of Q can be expressed as

$$h_f = rQ^n \quad ...(11.39)$$

where r is a proportionality factor which can be determined for each pipe knowing the friction factors f, the length L and the diameter D of the pipe $\left(r = \frac{fL}{2g(\pi/4)^2 D^5} = \frac{fL}{12.10 D^5}\right)$; and n is an exponent having a numerical value ranging from 1.72 to 2.00.

The pipe network problems are in general complicated and cannot be solved analytically. As such methods of successive approximations are utilised. One such method which is commonly used is 'Hardy Cross Method', named after its original investigator. The procedure for the solution of pipe network problems by the Hardy Cross Method is as follows:

(1) Assume a most suitable distribution of flow that satisfies continuity at each junction.

(2) With the assumed values of Q, compute the head losses for each pipe using Eq. 11.39.

(3) Consider different loops or circuits and compute the net head loss around each circuit considering the head loss in clockwise flows as positive and in anti-clockwise flows as negative. For a correct distribution of flow the net head loss around each circuit should be equal to zero, so that the circuit will be balanced. However, in most of the cases, for the assumed distribution of flow the head loss around the circuit will not be equal to zero. The assumed flows are then corrected by introducing a correction ΔQ for the flows, till the circuit is balanced. The value of the correction ΔQ to be applied to the assumed flows of the circuit may be obtained as follows:

For any pipe if Q_0 is the assumed discharge and Q is the correct discharge, then

$$Q = Q_0 + \Delta Q$$

and the head loss for the pipe is

$$h_f = rQ^n = r(Q_0 + \Delta Q)^n$$

Thus for the complete circuit

$$\Sigma h_f = \Sigma rQ^n = \Sigma r(Q0 + \Delta Q)^n$$

By expanding the terms in the brackets by binomial theorem

$$\Sigma rQ^n = \Sigma r[Q_0^n + nQ_0^{n-1}\Delta Q + ...]$$

If ΔQ is small compared with Q_0 all terms of the series after the second one may be dropped. Thus

$$\Sigma rQ^n = \Sigma rQ_0^n + \Sigma rnQ_0^{n-1}\Delta Q$$

For the correct distribution the circuit is balanced and hence $\Sigma rQ^n = 0$.

Therefore, $\Sigma rQ_0^n + \Delta Q \Sigma rnQ_0^{n-1} = 0$

In the above expression ΔQ has been taken out of the summations as it is same for all the pipes in the circuit. Solving for ΔQ

$$\Delta Q = -\frac{\Sigma rQ_0^n}{\Sigma rn\, Q_0^{n-1}} \quad ...(11.40)$$

In the above expression for the correction the denominator is the sum of absolute terms and hence it has no sign. Further if the head losses due to flow in the clockwise direction are more than losses due to flow in the anti-clockwise direction, then according to the sign convention adopted, ΔQ will be negative and hence it should be added to the flow in the anti-clockwise direction and subtracted from the flow in the clockwise direction. On the other hand if the head losses due to flow in the clockwise direction are less than the head losses due to flow in the anti-clockwise direction, then ΔQ will be positive and hence it should be added to the flow in the clockwise direction and subtracted from the flow in the anti-clockwise direction. Moreover, for the pipes common to two circuits or loops (such as CH, GH, HF etc.) a correction from both the loops will be required to be applied.

(4) With the corrected flows in all the pipes, a second trial calculation is made for all the loops and the process is repeated till the corrections become negligible.

The method described above for the solution of pipe network problems is indicated in Illustrative Example 11.18.

ILLUSTRATIVE EXAMPLES

Example 11.1. *A pipeline ABC 180 m long, is laid on an upward slope of 1 in 60. The length of the portion AB is 90 m and its diameter is 0.15 m. At B the pipe section suddenly enlarges to 0.30 m diameter and remains so for the remainder of its length BC, 90 m. A flow of 50 litres per second is pumped into the pipe at its lower end A and is discharged at the upper end C into a closed tank. The pressure at the supply end A is 137.34 kN/m². Sketch (a) the total energy line, (b) the hydraulic grade line and also find the pressure at the discharge end C. Take $f = 0.02$.*

Solution

$$Q = 50\ \text{l/s} = 50 \times 10^{-3}\ \text{m}^3/\text{s}$$

$$D_1 = 0.15\ \text{m};\ D_2 = 0.30\ \text{m}$$

$$V_1 = \frac{Q}{\frac{\pi D_1^2}{4}} = \frac{50\times 10^{-3}}{\frac{\pi}{4}\times(0.15)^2} = 2.83\ \text{m/s}$$

$$V_2 = \frac{Q}{\frac{\pi D_2^2}{4}} = \frac{50\times 10^{-3}}{\frac{\pi}{4}\times(0.30)^2} = 0.71\ \text{m/s}$$

Loss of head in section AB

$$hf_1 = \frac{fL_1}{D_1}\frac{V_1^2}{2g}$$

$$= \frac{0.02\times 90}{0.15}\times\frac{(2.83)^2}{2\times 9.81} = 4.90\ \text{m}$$

Loss of head in section BC

$$hf_2 = \frac{fL_2}{D_2}\frac{V_2^2}{2g}$$

$$= \frac{0.02\times 90}{0.30}\times\frac{(0.71)^2}{2\times 9.81} = 0.153\ \text{m}$$

Loss of head at B due to sudden enlargement

$$= \frac{(V_1 - V_2)^2}{2g} = \frac{(2.83 - 0.71)^2}{2\times 9.81} = 0.229\ \text{m}$$

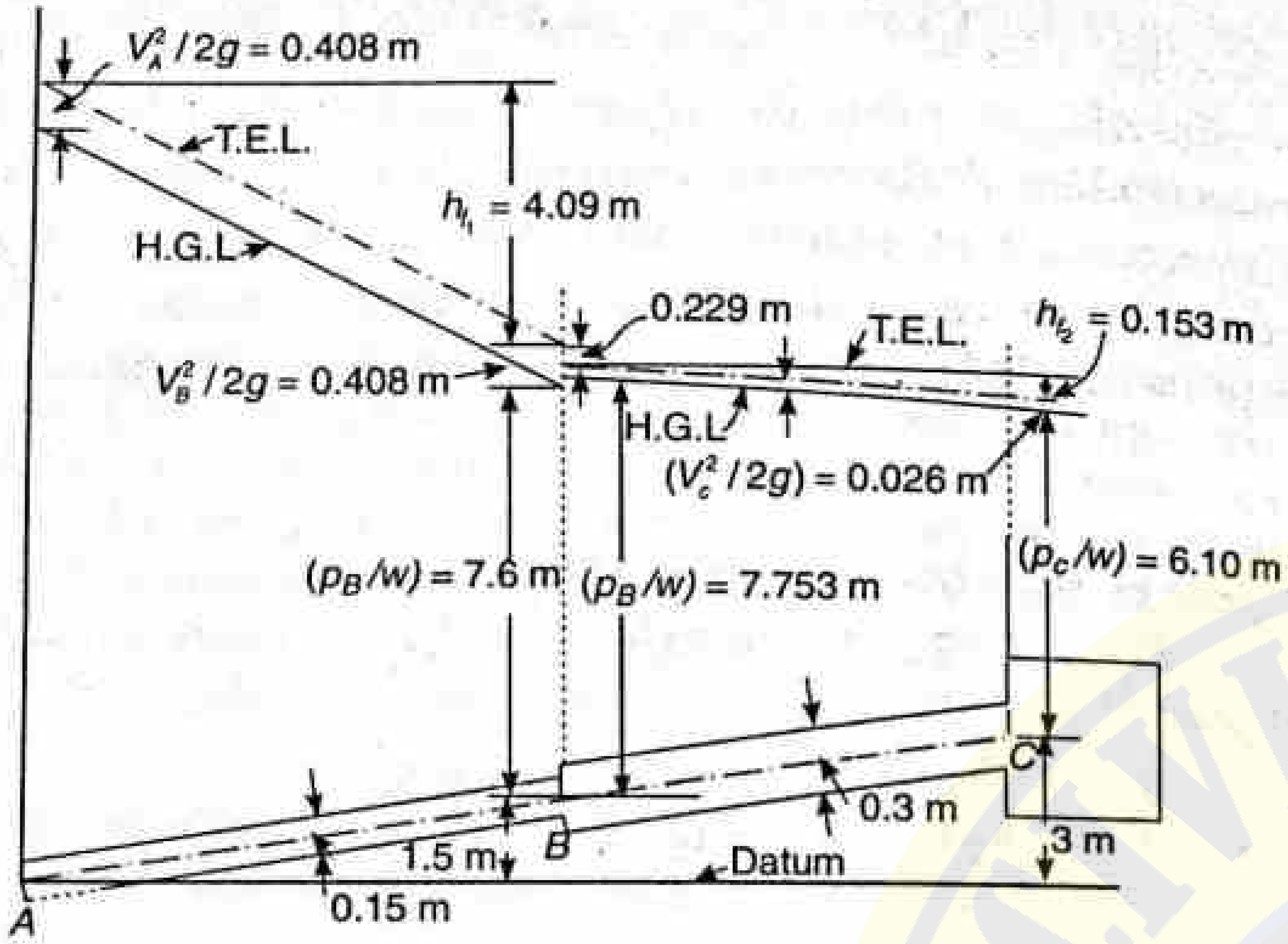


Figure Ex. 11.1

Section AB

Assuming datum to be passing through A, different heads at A are

$$Z_A = 0; \ \frac{p_A}{w} = \frac{137.34 \times 10^3}{9810} = 14 \text{ m}$$

$$\frac{V_A^2}{2g} = \frac{(2.83)^2}{2 \times 9.81} = 0.408 \text{ m}$$

Different heads at B

$$Z_B = 90 \times \frac{1}{60} = 1.5 \text{ m}; \ \frac{V_B^2}{2g} = \frac{(2.83)^2}{2 \times 9.81} = 0.408 \text{ m}$$

By applying Bernoulli's equation between points A and B

$$\frac{p_A}{w} + \frac{V_A^2}{2g} + Z_A = \frac{p_B}{w} + \frac{V_B^2}{2g} + Z_B + h_{f1}$$

$$14 + 0.408 + 0 = \frac{p_B}{w} + 0.408 + 1.5 + 4.90$$

$$\therefore \quad \frac{p_B}{w} = 7.6 \text{ m}$$

Thus hydraulic grade line of section AB will be a line joining $\frac{p_A}{w} = 14$ m and $\frac{p_B}{w} = 7.6$ m as shown

in Fig. Ex. 11.2. Total energy line is obtained by adding $\left(\frac{V_A^2}{2g}\right) = \left(\frac{V_B^2}{2g}\right) = 0.408$ m to the hydraulic grade line. Thus both lines will be running parallel to each other.

Section BC

Different heads at B

$$\frac{p'_B}{w} = \frac{p_B}{w} + \frac{V_B^2}{2g} - \frac{(V'_B)^2}{2g} - \text{loss due to sudden enlargement}$$

$$\frac{(V'_B)^2}{2g} = \frac{(0.71)^2}{2 \times 9.81} = 0.026 \text{ m (as } V'_B = V_2)$$

$$\therefore \quad \frac{p'_B}{w} = 7.6 + 0.408 - 0.026 - 0.229 = 7.753 \text{ m}$$

$$Z_B = 1.5 \text{ m}$$

Different heads at C

$$Z_C = 180 \times \frac{1}{60} = 3 \text{ m}$$

$$\frac{V_C^2}{2g} = \frac{(0.71)^2}{2 \times 9.81} = 0.026 \text{ m}$$

$$\frac{p_C}{w} = \frac{p_B}{w} - (Z_C - Z_B) - h_{f2}$$

$$= 7.753 - (3.0 - 1.5) - 0.153 = 6.10 \text{ m}$$

Thus hydraulic grade line of section BC will be a line joining $\frac{p'_B}{w} = 7.753$ m and $\frac{p_C}{w} = 6.10$ m as shown in the figure. Total energy line will be at a distance $\frac{(V'_B)^2}{2g} = \frac{V_C^2}{2g} = 0.026$ m above the hydraulic grade line

The pressure at the discharge end $= \frac{p_C}{w} = 6.10$ m of water

or $$p_C = \frac{6.10 \times 9810}{10^3}$$

$$= 59.841 \text{ kN/m}^2 = 59.841 \text{ kP}_a$$

Example 11.2. *A pipeline 0.225 m in diameter and 1580 m long has a slope of 1 in 200 for the first 790 m and 1 in 100 for the next 790 m. The pressure at the upper end of the pipeline is 107.91 kP$_a$ and at the lower end is 53.955 kP$_a$. Taking f = 0.032 determine the discharge through the pipe.*

Solution

Assuming the datum to be passing through the lower end of the pipeline, the datum head for the upper end of the pipeline is

$$Z_1 = \frac{790}{200} + \frac{790}{100} = (3.95 + 7.90) = 11.85 \text{ m}$$

Applying Bernoulli's equation between the upper and the lower ends of the pipeline

$$\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1 = \frac{p_2}{w} + \frac{V_2^2}{2g} + Z_2 + h_f$$

But $V_1 = V_2 = V$

Then the loss of head due to friction

$$h_f = \frac{fLV^2}{2gD} = \frac{0.032 \times 1580 \times V^2}{2 \times 9.81 \times 0.225} = 11.45\ V^2$$

Thus by substitution, we get

$$\frac{107.91 \times 10^3}{9810} + \frac{V^2}{2g} + 11.85 = \frac{53.955 \times 10^3}{9810} + \frac{V^2}{2g} + 0 + 11.45\ V^2$$

r $$11.45V^2 = 17.35$$

r $$V = \sqrt{\frac{17.35}{11.45}} = 1.23 \text{ m/s}$$

$\therefore$ Discharge $$Q = \frac{\pi D^2}{4} V$$

$$= \frac{\pi (0.225)^2}{4} \times 1.23 = 0.048\ 9 \text{ m}^3/\text{s}$$

Example 11.3. *The population of a city is 800 000 and it is to be supplied with water from a reservoir 6.4 n away. Water is to be supplied at the rate of 140 litres per head per day and half the supply is to be delivered 8 hours. The full supply level of the reservoir is R.L 180.00 and its lowest water level is R.L. 105.00. The livery end of the main is at R.L. 22.50 and the head required there is 12 m. Find the diameter of the pipe. Take 0.04.*

Solution

Total daily supply = (800 000 × 140 × 10^{-3}) = 11.2 × 10^4 m^3. Supply to be delivered in 8 hours

$$= \frac{11.2 \times 10^4}{2} = 5.6 \times 10^4 \text{ m}^3$$

$\therefore$ Discharge $$= \frac{5.6 \times 10^4}{8 \times 60 \times 60} = 1.94 \text{ m}^3/\text{s}$$

The two extreme cases are considered as follows.

(a) When the reservoir is full, the head available

$$= (180.00 - 22.50 - 12.00)$$
$$= 145.50 \text{ m}$$

Neglecting the minor losses, we have

$$145.50 = \frac{fLV^2}{2gD} = \frac{fLQ^2}{2g(\pi/4)^2 D^5}$$

or $$D^5 = \frac{0.04 \times 6.4 \times 10^3 \times (1.94)^2}{2 \times 9.81 \times (\pi/4)^2 \times 145.50}$$

$\therefore$ $$D = 0.886 \text{ m}$$

(b) For the lowest water level in the reservoir, the head available

$$= (105.00 - 22.50 - 12.00) = 70.50 \text{ m}$$

Again neglecting the minor losses, we have

$$70.50 = \frac{fLV^2}{2gD} = \frac{fLQ^2}{2g(\pi/4)^2 D^5}$$

or $$D^5 = \frac{0.04 \times 6.4 \times 10^3 \times (1.94)^2}{2 \times 9.81 \times (\pi/4)^2 \times 70.50}$$

$\therefore$ $$D = 1.025 \text{ m}$$

Thus the pipe diameter equal to 1.025 m may be adopted. This is so because a pipe of larger diameter than required when the reservoir has water level at full supply level, will result in less loss of head. Hence the required quantity of water at the required head may be made available by regulating the flow by means of a valve. On the other hand if a pipe of smaller diameter than required when the reservoir has the lowest water level, is provided then it would result in a considerable loss of head and the required discharge at the requisite residual head will not be obtained.

Example 11.4. *A pipe 50 mm diameter is 6 m long and the velocity of flow of water in the pipe is 2.4 m/s. What loss of head and the corresponding power would be saved if the central 2 m length of pipe was replaced by 75 mm diameter pipe, the change of section being sudden? Take f = 0.04 for the pipes of both diameters.*

Solution

Velocity of flow in the larger diameter pipe

$$= \frac{2.4 \times (\pi/4) \times (0.05)^2}{(\pi/4) \times (0.075)^2} = 1.07 \text{ m/s}$$

Head loss due to friction in the central 2 m length of the pipe is

$$h_{f1} = \frac{fLV^2}{2gD} = \frac{0.04 \times 2 \times (2.4)^2}{2 \times 9.81 \times 0.05} = 0.47 \text{ m}$$

When the central 2 m length of the 50 mm diameter pipe is replaced by a 75 mm diameter pipe, then the various head losses are as follows:

Loss of head due to sudden enlargement

$$= \frac{(V_1 - V_2)^2}{2g} = \frac{(2.4-1.07)^2}{2\times 9.81} = 0.090 \text{ m}$$

Loss of head due to friction

$$h_f = \frac{fLV^2}{2gD} = \frac{0.04\times 2\times(1.07)^2}{2\times 9.81\times 0.075} = 0.062 \text{ m}$$

Loss of head due to sudden contraction

$$= \frac{0.5V^2}{2g} = \frac{0.5\times(2.4)^2}{2\times 9.81} = 0.147 \text{ m}$$

$$\therefore \quad \text{Total head loss} = (0.090 + 0.062 + 0.147)$$
$$= 0.299 \text{ m}$$
$$\therefore \quad \text{Head saved } h = (0.47 - 0.299) = 0.171 \text{ m}$$

Discharge $$Q = \frac{\pi}{4}\times(0.05)^2 \times 2.4 = 4.71\times 10^{-3} \text{ m}^3/\text{s}$$

$$\therefore \quad \text{Power saved} = (wQh)$$
$$= (9810 \times 4.71\times 10^{-3} \times 0.171)$$
$$= 7.901 \text{ W}$$

Example 11.5. *Two pipes each 300 m long are available for connecting to a reservoir from which a flow of 0.085 m^3/s is required. If the diameters of the two pipes are 0.30 m and 0.15 m respectively, determine the ratio of the head lost when the pipes are connected in series to the head lost when they are connected in parallel. Neglect minor losses.*

Solution

When the pipes are connected in series, the loss of head due to friction is

$$(h_f)_s = h_{f1} + h_{f2}$$

$$= \frac{fLV_1^2}{2gD_1} + \frac{fLV_2^2}{2gD_2}$$

$$= \frac{fLQ^2}{2g(\pi/4)^2 D_1^5} + \frac{fLQ^2}{2g(\pi/4)^2 D_2^5}$$

$$= \frac{f\times 300\times(0.085)^2}{2g(\pi/4)^2}\left[\frac{1}{(0.30)^5} + \frac{1}{(0.15)^5}\right]$$

When the pipes are connected in parallel, then let Q_1 and Q_2 be the discharge in the bigger and the smaller pipes respectively.

Thus

$$0.085 = Q_1 + Q_2 \qquad \text{...(i)}$$

Also $$(h_f)_p = \frac{fLQ_1^2}{2g(\pi/4)^2 D_1^5} = \frac{fLQ_2^2}{2g(\pi/4)^2 D_2^5} \qquad \text{...(ii)}$$

From Eq. (ii), we have

$$\frac{Q_1^2}{(0.30)^5} = \frac{Q_2^2}{(0.15)^5}$$

or $$Q_1 = 2^{5/2} Q_2 = 5.657\, Q_2 \qquad \text{...(iii)}$$

Solving Eqs (i) and (iii), we get

$$6.657\, Q_2 = 0.085$$

or $$Q_2 = \frac{0.085}{6.657}$$

$$\therefore \quad (h_f)_p = \frac{f\times 300\times(0.085)^2}{2g(\pi/4)^2\times(0.15)^5\times(6.657)^2}$$

Thus $$\frac{(h_f)_s}{(h_f)_p} = \frac{\dfrac{f\times 300\times(0.085)^2}{2g(\pi/4)^2}\left[\dfrac{1}{(0.30)^5} + \dfrac{1}{(0.15)^5}\right]}{\dfrac{f\times 300\times(0.085)^2}{2g(\pi/4)^2\times(0.15)^5\times(6.657)^2}}$$

or $$\frac{(h_f)_s}{(h_f)_p} = \frac{[(0.30)^5 + (0.15)^5]\times(6.657)^2}{(0.30)^5}$$

$$= 45.7$$

Example 11.6. *A pipe having a length of 6 km and diameter 0.70 m connects two reservoirs A and B, the difference between their water levels is 30 m. Halfway along the pipe there is a branch through which water can be supplied to a third reservoir C. Taking f = 0.024 determine the rate of flow of reservoir B when (a) no water is discharged to reservoir C; (b) the quantity of water discharged to reservoir C is 0.15 m^3/s. Neglect minor losses.*

Solution

(a) When no water is discharged to reservoir C, then we have

$$30 = h_f = \frac{fLV^2}{2gD} = \frac{0.024\times 6\times 10^3\times V^2}{2\times 9.81\times 0.70}$$

or $$V = 1.692 \text{ m/s}$$

$$\therefore \quad Q = \frac{\pi D^2}{4}\times V$$

$$= \frac{\pi\times(0.70)^2}{4}\times 1.692$$

$$= 0.65 \text{ m}^3/\text{s}$$

(b) When 0.15 m^3/s is discharged to reservoir C, then let Q be the total discharge from reservoir A. The discharge flowing into reservoir $B = (Q - 0.15)$.

Now applying Bernoulli's equation between the reservoirs A and B, we have (neglecting the minor losses)

$$30 = \frac{0.024 \times 3 \times 10^3 \times Q^2}{2 \times 9.81 \times (\pi/4)^2 \times (0.70)^5} + \frac{0.024 \times 3 \times 10^3 \times (Q - 0.15)^2}{2 \times 9.81 \times (\pi/4)^2 \times (0.70)^5}$$

or

or
$$30 = \frac{0.024 \times 3 \times 10^3}{2 \times 9.81 \times (\pi/4)^2 \times 0.70)^5}\left[Q^2 + (Q - 0.15)^2\right]$$

or
$$30 = 35.4\left[2Q^2 - 0.3Q + 0.0225\right]$$

or
$$Q^2 - 0.15\,Q - 0.401 = 0$$

$$\therefore \quad Q = 0.713 \text{ m}^3/\text{s}$$

Hence flow to reservoir $B = (0.713 - 0.15)$

$$= 0.563 \text{ m}^3/\text{s}$$

Example 11.7. *A compound piping system consists of 1800 m of 0.50 m, 1200 m of 0.40 m and 600 m of 0.30 m new cast iron pipes connected in series. Convert the system to (a) an equivalent length of 0.40 m pipe, and (b) and equivalent size pipe 3600 m long.*

Solution

From Eq. 11.13, we have

$$\frac{L}{D^5} = \left[\frac{L_1}{D_1^5} + \frac{L_2}{D_2^5} + \frac{L_3}{D_3^5}\right]$$

$L_1 = 1800$ m; $D_1 = 0.50$ m
$L_2 = 1200$ m; $D_2 = 0.40$ m
$L_3 = 600$ m; $D_3 = 0.30$ m

(a) To find L for $D = 0.40$ m

Thus by substitution, we get

$$L = (0.40)^5\left[\frac{1800}{(0.50)^5} + \frac{1200}{(0.40)^5} + \frac{600}{(0.30)^5}\right]$$

$$= 4\,318 \text{ m}$$

(b) To find D for $L = 3600$ m

Thus by substitution, we get

$$\frac{3600}{D^5} = \left[\frac{1800}{(0.50)^5} + \frac{1200}{(0.40)^5} + \frac{600}{(0.30)^5}\right]$$

$$\therefore \quad D = 0.386 \text{ m} \simeq 0.39 \text{ m}$$

Example 11.8. *A 0.3 m diameter pipe 2340 m long is connected with a reservoir whose surface is 72 m above the discharging end of the pipe. If for the last 1170 m, a second pipe of the same diameter be laid beside the first and connected to it, what would be the increase in the discharge? Take $f = 0.02$.*

Solution

In the first case, from Eq. 11.8, we have

$$H = \left(1.5 + \frac{fL}{D}\right)\frac{V^2}{2g}$$

or
$$72 = \left(1.5 + \frac{0.02 \times 2340}{0.30}\right)\frac{V^2}{2 \times 9.81}$$

or
$$V = \sqrt{\frac{72 \times 2 \times 9.81}{157.5}} = 3 \text{ m/s}$$

$\therefore$ Discharge
$$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.30)^2}{4} \times 3 = 0.212 \text{ m}^3/\text{s}$$

In the second case let Q be the total discharge. Since for the second half of the length there are two parallel pipes of the same diameter, each pipe will carry discharge equal to $(Q/2)$ and the velocity of flow in each pipe will be equal to half the velocity of flow V in the first half of the length.

Thus applying Bernoulli's equation between the water surface in the reservoir and the outlet of pipe, we get

$$72 = 0.5\frac{V^2}{2g} + \frac{0.02 \times 1170 \times V^2}{2g \times 0.30} + \frac{0.02 \times 1170 \times (V/2)^2}{2g \times 0.30} + \frac{(V/2)^2}{2g}$$

or
$$72 = \frac{V^2}{2g}[0.5 + 78.0 + 0.25]$$

or
$$V = 3.79 \text{ m/s}$$

$$\therefore \quad Q = \frac{\pi}{4}(0.30)^2 \times 3.79 = 0.268 \text{ m}^3/\text{s}$$

$\therefore$ Increase in discharge
$$= (0.268 - 0.212) = 0.056 \text{ m}^3/\text{s}$$

or percentage increase in discharge

$$= \frac{0.056}{0.212} \times 100 = 26.42\ \%$$

Example 11.9. *For the distribution main of a city water supply a 0.30 m main is required. As pipes above 0.25 m diameter are not available, it is decided to lay two parallel mains of same diameter. Find the diameter of the parallel main.*

Solution

Let D be the diameter of the single large pipe which is to be replaced by two parallel pipes of equal diameter d. Let all the three pipes be of equal length say L.

Then

$$h_f = \frac{fL}{D}\frac{V^2}{2g}$$

$\therefore$ Velocity of flow in large pipe

$$V = \sqrt{\frac{h_f \times 2g}{f}}\sqrt{\frac{D}{L}}$$

Discharge from large pipe

$$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi D^2}{4} \times \sqrt{\frac{h_f \times 2g}{f}}\sqrt{\frac{D}{L}}$$

$$= \frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{D^{5/2}}{L^{1/2}}\right)$$

Frictional head loss in each of the parallel pipes = frictional head loss in large pipe = h_f

$\therefore$ Velocity of water in each of the parallel pipes

$$= \sqrt{\frac{h_f \times 2g}{f}}\sqrt{\frac{D}{L}}$$

Discharge from each of the parallel pipes

$$= \frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{D^{5/2}}{L^{1/2}}\right)$$

From Eq. 11.14, we have

$$Q = Q_1 + Q_2$$

but $Q_1 = Q_2$ and hence

$$\frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{D^{5/2}}{L^{1/2}}\right) = 2 \times \frac{\pi}{4}\sqrt{\frac{h_f \times 2g}{f}}\left(\frac{D^{5/2}}{L^{1/2}}\right)$$

or

$$D^{5/2} = 2d^{5/2}$$

or

$$d = \frac{D}{2^{2/5}}$$

$$= \frac{0.30}{2^{2/5}} = 0.23 \text{ m}$$

Since pipe of 0.23 m diameter is not a standard one, it may not be available. As such each pipe of diameter 0.25 m may be used.

Note: If in place of a single pipe of diameter D, n pipes each of diameter d are laid in parallel, then

$$d = \frac{D}{n^{2/5}}$$

Example 11.10. *Two reservoirs are connected by a pipeline which rises above the level of the highest reservoir. What will be the highest point of the siphon above this level if the length of the pipe leading upto this point is 540 m, the diameter of the pipe being 0.35 m? The difference in level of the two reservoirs is 12.6 m and the total length of the pipe is 960 m. The siphon must run full, the separation of dissolved gases may be assumed to occur at an absolute pressure of 2.4 m of water. What will be the discharge under the above conditions in litres per minute? Assume $f = 0.04$ and allow for all losses.*

Solution

Applying Bernoulli's equation between the reservoirs A and B, we get

$$12.6 = 0.5\frac{V^2}{2g} + \frac{fLV^2}{2gD} + \frac{V^2}{2g}$$

or

$$12.6 = \frac{V^2}{2g}\left[1.5 + \frac{fL}{D}\right]$$

$$= \frac{V^2}{2g}\left[1.5 + \frac{0.04 \times 960}{0.35}\right]$$

or

$$V = 1.49 \text{ m/s}$$

Now applying Bernoulli's equation between the reservoir A and the summit S, we have, taking water surface in reservoir A as datum

$$\frac{p_a}{w} = h + 2.4 + 0.5\frac{V^2}{2g} + \frac{V^2}{2g} + \frac{fL_1V^2}{2gD}$$

where h is the height of the summit above the water surface in the reservoir A.

Also atmospheric pressure head

$$\frac{p_a}{w} = 10.3 \text{ m of water}$$

$$L_1 = 540 \text{ m}; V = 1.49 \text{ m/s}$$

Thus by substitution, we get

$$10.3 = h + 2.4 + \frac{(1.49)^2}{2 \times 9.81}\left[1.5 + \frac{0.04 \times 540}{0.35}\right]$$

$$\therefore \quad h = 0.75 \text{ m}$$

i.e., the height of the summit of the siphon above the water surface in the reservoir $A = 0.75$ m.

Discharge $$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.35)^2}{4} \times 1.49$$

$$= 0.1434 \text{ m}^3/\text{s}$$
$$= 0.1434 \times 10^3 \times 60 \; l/\text{min}$$
$$= 8.6 \times 10^3 \; l/\text{min}.$$

Example 11.11. *The difference in the water surface levels of two reservoirs which are connected by a siphon is 8 m. The length of the siphon is 600 m and its diameter 0.3 m. Assuming $f = 0.02$, determine the discharge when the siphon is running full.*

If the summit of the pipeline is 5 m above the surface level of the upper reservoir, determine the maximum length of the inlet leg for the pipe to run full. Allow for all the losses and assume the permissible minimum pressure at the summit of the siphon to be absolute zero.

Solution

Applying Bernoulli's equation between the two reservoirs, we get

$$8 = 0.5\frac{V^2}{2g} + \frac{fLV^2}{2gD} + \frac{V^2}{2g}$$

or $$8 = \frac{V^2}{2g}\left[1.5 + \frac{fL}{D}\right]$$

$$= \frac{V^2}{2g}\left[1.5 + \frac{0.02 \times 600}{0.30}\right]$$

or $$V = 1.945 \text{ m/s}$$

$\therefore$ Discharge $$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi \times (0.30)^2}{4} \times 1.945$$

$$= 0.1375 \text{ m}^3/\text{s}$$

Applying Bernoulli's equation between the water surface in the upper reservoir and the summit of the siphon, we have

$$10.3 = 0 + 5 + 0.5\frac{V^2}{2g} + \frac{fL_1V^2}{2gD} + \frac{V^2}{2g}$$

or $$10.3 = 0 + 5 + \frac{V^2}{2g}\left[1.5 + \frac{fL_1}{D}\right]$$

$$10.3 = 0 + 5 + \frac{(1.945)^2}{2 \times 9.81}\left[1.5 + \frac{0.02 \times L_1}{0.30}\right]$$

or $$L_1 = 390 \text{ m}$$

$\therefore$

Example 11.12. *Water flows from a reservoir through a pipe of 0.15 m diameter and 180 m long to a point 13.5 m below the open surface of the reservoir. Here it branches into two pipes, each of 0.1 m diameter, one of which is 48 m long discharging to atmosphere at a point 18 m below reservoir level, and the other 60 m long discharging to atmosphere 24 m below reservoir level. Assuming a constant friction coefficient of 0.032, calculate the discharge from each pipe. Neglect any losses at the junction.*

Solution

Let the pressure at the junction (2) be p and let atmospheric pressure be 10.3 m of water.

Applying Bernoulli's equation between (1) and (2), we get

$$10.3 + 13.5 = \frac{p}{w} + \frac{V_1^2}{2g} + \frac{0.032 \times 180 \times V_1^2}{2g \times 0.15}$$

$$\therefore \quad \frac{p}{w} = 23.8 - 2.008\, V_1^2 \quad \text{...(i)}$$

Applying Bernoulli's equation between (2) and (3), we get

$$\frac{p}{w} + \frac{V_2^2}{2g} + 4.5 = 10.3 + \frac{V_2^2}{2g} + \frac{0.032 \times 48 \times V_2^2}{2g \times 0.10}$$

$$\therefore \quad \frac{p}{w} = 5.8 + 0.783\, V_2^2 \quad \text{...(ii)}$$

Applying Bernoulli's equation between (2) and (4), we get

$$\frac{p}{w} + \frac{V_3^2}{2g} + 10.5 = 10.3 + \frac{V_3^2}{2g} + \frac{0.032 \times 60 \times V_3^2}{2g \times 0.10}$$

$$\therefore \quad \frac{p}{w} = 0.979 V_3^2 - 0.2 \quad \text{...(iii)}$$

By continuity, we have ...(iv)

$$9V_1 = (V_2 + V_3) \times 4$$

These four equations are to be solved, which may be accomplished as follows:

Let $$V_3 = mV_2$$

Thus from Eq. (iv), we have

$$V_1 = \frac{4}{9}(m+1)V_2$$

Equating (i) and (ii), we get

$$23.8 - 2.008V_1^2 = 5.8 + 0.783V_2^2$$

or $$18 = \left[2.008\times\frac{16}{81}(m+1)^2+0.783\right]V_2^2 \quad \text{...(v)}$$

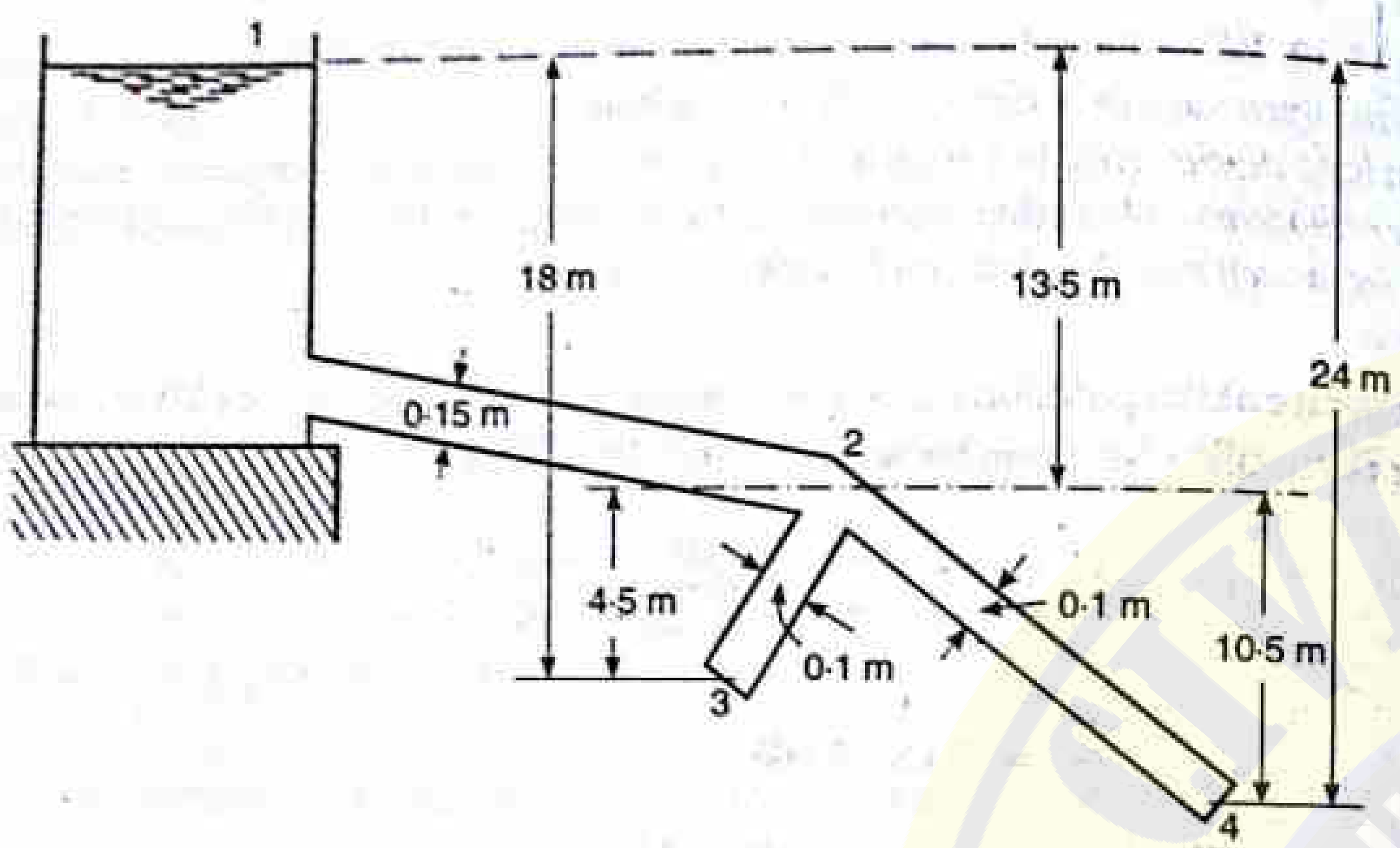


Figure Ex. 11.12

Again equating (ii) and (iii), we get

$$5.8+0.783V_2^2 = 0.979V_2^2-0.2$$

or $$6 = [0.979\,m-0.783]\,V_2^2 \quad \text{...(vi)}$$

Dividing (v) by (vi), we get

$$3 = \frac{0.397m^2+0.794\,m+1.180}{0.979\,m^2-0.783}$$

or $2.540\,m^2-0.794\,m-3.529 = 0$

or $m^2-0.313\,m-1.39 = 0$

or $m = 1.346$

Substituting this value of m in Eq. (vi), we get

$$V_2 = 2.461\text{ m/s}$$

$$V_3 = (1.346\times 2.461) = 3.313\text{ m/s}$$

$$\therefore \quad Q_2 = \left[\frac{\pi}{4}\times(0.10)^2\times 2.461\right] = 0.019\,3\text{ m}^3/\text{s}$$

$$Q_3 = \left[\frac{\pi}{4}\times(0.10)^2\times 3.313\right] = 0.026\,0\text{ m}^3/\text{s}$$

$\therefore$ Total discharge

$$Q_1 = (Q_2+Q_3)$$

$$= (0.0193+0.0260)$$

$$= 0.045\,3\text{ m}^3/\text{s}$$

Example 11.13. *Water is pumped through a 600 mm diameter pipe with a head loss of 20 m. It is intended to reduce the power consumption by laying another parallel pipe of appropriate diameter laid by the side of the existing pipeline with the same overall length and the same friction factor—by reducing the head loss to 10m, but still delivering the same discharge jointly through both pipes. What should be the diameter of this additional pipe?*

Solution

Initial head loss $h_f = 20$ m.

Thus if Q is the discharge through the pipe of diameter 600 mm, then from Darcy-Weisbach equation, we have

$$h_f = \frac{fLV^2}{2gd}$$

or $$h_f = \frac{fLQ^2}{2g\times(\pi/4)^2 d^5}$$

or $$h_f = \frac{fLQ^2}{12.1\times d^5}$$

$$d = 600\text{ mm}$$

$$= 0.6\text{ m}$$

$$\therefore \quad 20 = \frac{fLQ^2}{12.1\times(0.6)^5} \quad \text{...(i)}$$

Let d_1 be the diameter of the pipe required to be connected in parallel to the existing pipe, and Q_1 and Q_2 be the discharge through the pipes of diameters 600 m and d_1 respectively.

Since total discharge remains same, ...(ii)

$$Q = Q_1+Q_2$$

The head loss is reduced to 10 m and both the pipes have same length L and same friction f. Thus from Darcy–Weisbach Equation, we have

$$10 = \frac{fLQ_1^2}{12.1\times(0.6)^5} \quad \text{...(iii)}$$

$$10 = \frac{fLQ_2^2}{12.1\times(d_1)^5} \quad \text{...(iv)}$$

From Eqs (i) and (ii), we obtain

$$Q_1 = \frac{Q}{\sqrt{2}} = 0.0707\,Q$$

Substituting the value of Q_1 in Eq. (ii), we get

$$Q_2 = Q - Q_1 = Q - 0.707Q = 0.293\,Q$$

From Eqs. (iii) and (iv), we obtain

$$\frac{Q_1^2}{(0.6)^5} = \frac{Q_2^2}{(d_1)^5}$$

or

$$\frac{Q_1}{Q_2} = \left(\frac{0.6}{d_1}\right)^{5/2}$$

or

$$\left(\frac{0.707Q}{0.293Q}\right) = \left(\frac{0.6}{d_1}\right)^{5/2}$$

$\therefore$

$$d_1 = 0.6\left(\frac{0.293}{0.707}\right)^{2/5} = 0.422 \text{ m} = 422 \text{ mm}$$

Thus a pipe of nearest available size of diameter 425 mm may be laid by the side of the existing pipe in parallel.

Example 11.14. *A fire engine supplies water to a hose pipe, 75 m long and 75 mm in diameter at a pressure of 294.3 kN/m² [3 kg (f)/cm²] (gage). The discharge end of the hose pipe has a nozzle of diameter d fixed to it. Taking friction factor f as 0.032, determine the diameter d of the nozzle so that the momentum of the issuing jet may be maximum.*

Solution

Let D be the diameter of the hose pipe; L be the length of the hose pipe; V_1 be the velocity of flow in the hose pipe; V_2 be the velocity of the issuing jet; and Q be the discharge.

Thus momentum of the issuing jet is

$$M = \frac{w}{g}\,Q V_2$$

or

$$M = \frac{w}{g}\frac{\pi d^2}{4} V_2^2 \qquad \text{...(i)}$$

From continuity equation, we have

$$Q = \frac{\pi}{4}D^2V_1 = \frac{\pi}{4}d^2V_2 \qquad \text{...(ii)}$$

If H is the head causing the flow, then

$$H = \frac{V_2^2}{2g} + \frac{fLV_1^2}{2gD} \qquad \text{...(iii)}$$

From Eqs (ii) and (iii), we have

$$2gH = V_2^2\left[1 + \frac{fLd^4}{D^5}\right] \qquad \text{...(iv)}$$

By substituting the value of V_2^2 from Eq. (iv) in Eq. (i), we get

$$M = \frac{w}{g}\frac{\pi}{4}d^2\,\frac{2gH}{1+\dfrac{fLd^4}{D^5}}$$

or

$$M = w\,\frac{\pi}{4} \times 2H \left[\frac{D^5d^2}{D^5 + fLd^4}\right] \qquad \text{...(v)}$$

For maximum value of M,

$$\frac{dM}{dd} = 0$$

From Eq. (v), we have

$$\frac{dM}{dd} = w\,\frac{\pi}{4} \times 2H \times D^5\left[\frac{2d(D^5 + fLd^4) - d^2(4fLd^3)}{(D^5 + fLd^4)^2}\right]$$

Thus

$$2d\,(D^5 + fLd^4) = d^2\,(4fLd^3)$$

or

$$fLd^4 = D^5$$

or

$$d = \left(\frac{D^5}{fL}\right)^{1/4}$$

In this case

$$D = 75 \text{ mm} = 0.075 \text{ m}$$
$$L = 75 \text{ m};\ f = 0.032$$

Thus by substitution, we get

$$d = \left[\frac{(0.075)^5}{0.032 \times 75}\right]^{1/4}$$

or

$$d = 0.0315 \text{ m} = 31.5 \text{ mm}$$

Example 11.15. *The turbine as shown in Fig. Ex. 11.15 extrats 45 mhp (metric horse power) from the water flowing through it. The 30 cm diameter 100 m long pipe has a friction factor of 0.02. Neglecting minor losses determint the flow rate through the pipe and the turbine.*

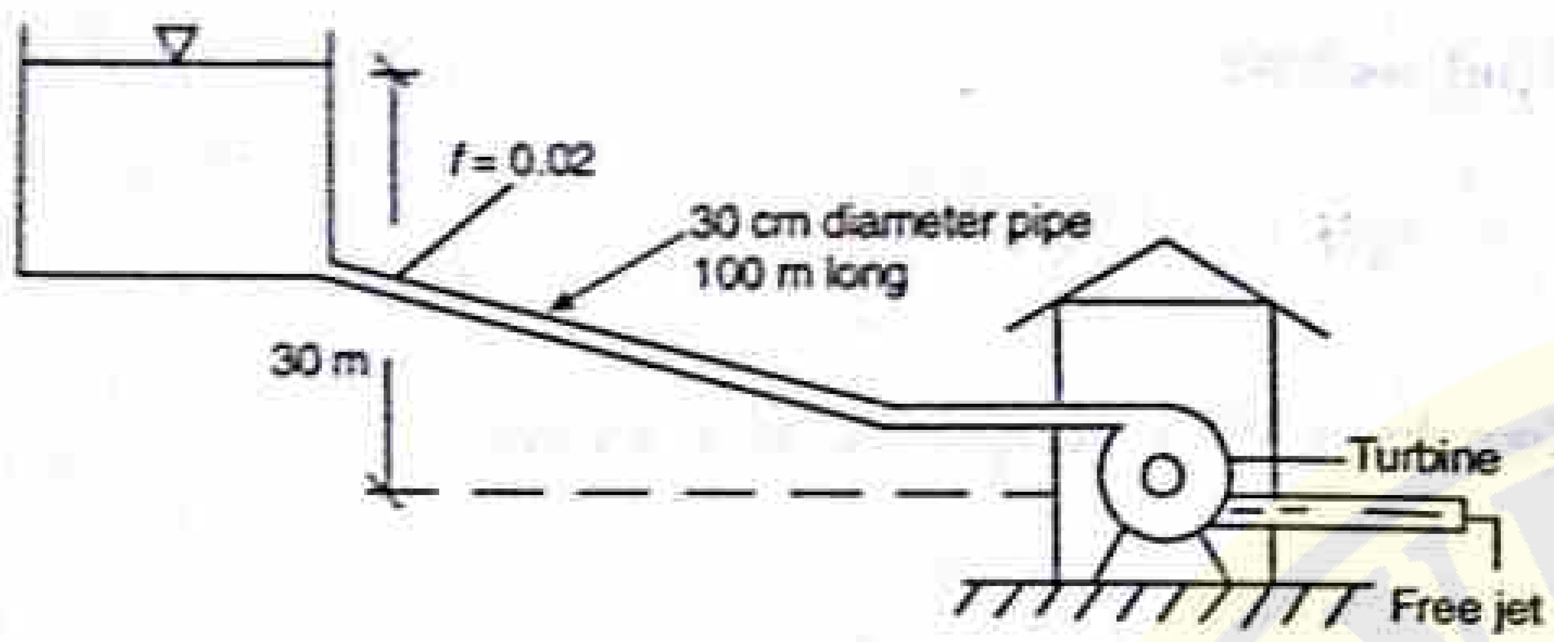


Figure Ex. 11.15

Solution

Head loss due to friction is given by

$$h_f = \frac{fLV^2}{2gd}$$

$f = 0.02$, $L = 100$ m and $d = 30$ cm $= 0.3$ m

Let Q be the discharge through the turbine. Thus

$$V = \frac{Q}{A} = \frac{Q}{(\pi/4)(0.3)^2}$$

Thus by substitution, we get,

$$h_f = \frac{0.02 \times 100 \times Q^2}{2 \times 9.81 \times (\pi/4)^2 \times (0.3)^5}$$

$$= 68.006\, Q^2$$

$\therefore$ Net head on the turbine

$$= (H - h_f)$$

$$= (30 - 68.006\, Q^2)$$

Power extracted by the turbine

$$= 45 \text{ mhp}$$

Thus

$$45 = \frac{wQ(H - h_f)}{75}$$

or

$$45 = \frac{1000 \times Q \times (30 - 68.006\, Q^2)}{75}$$

or $\quad 68.006\, Q^3 - 30Q + 3.375 = 0$

Solving by trial and error, we get

$$Q = 0.11604 \text{ cumec}$$

Example 11.16. *Power is to be transmitted hydraulically to an accumulator at a distance of 8 km by means of a number of 100 mm pipes laid horizontally, for which the coefficient of friction may be taken as 0.03. The pressure at the accumulator is maintained constant at 6524 kN/m². Determine the minimum number of pipes required to ensure an efficiency of at least 92 per cent when the power delivered is 162 kW. Also determine the maximum power that can be transmitted in this case.*

Solution

Let the pressure head at the reservoir be H.

The pressure head at the accumulator

$$= 6524 \text{ kN/m}^2 = 665.04 \text{ m of water}$$

Efficiency of transmission

$$= \frac{H - h_f}{H} \times 100$$

where h_f is head loss due to friction

$$\therefore \quad 0.92 = \frac{H - h_f}{H} = \frac{665.04}{H}$$

or

$$H = \frac{665.04}{0.92} = 722.87 \text{ m}$$

$\therefore$ Head lost due to friction

$$h_f = (722.87 - 665.04) = 57.83 \text{ m}$$

But

$$h_f = \frac{fLV^2}{2gD}$$

$$\therefore \quad 57.83 = \frac{0.03 \times 8 \times 10^3 \times V^2}{2 \times 9.81 \times 0.10}$$

or

$$V = 0.688 \text{ m/s}$$

Power transmitted in kW

$$= \frac{wQ(H - h_f)}{1000}$$

Let n be the number of pipes required. Then

$$Q = \left[\frac{\pi}{4} \times (0.10)^2 \times n \times 0.688\right]$$

$$= (5.404 \times 10^{-3})\, n \text{ m}^3/\text{s}$$

By substitution, we have

$$162 = \frac{9810 \times 5.404 \times 10^{-3} \times n \times 665.04}{1000}$$

$\therefore$ $n = 4.6$

$\therefore$ Number of pipes required = 5

For maximum power transmission

$$h_f = \frac{H}{3} = \frac{722.87}{3} = 240.96 \text{ m}$$

Thus
$$240.96 = \frac{0.030 \times 8 \times 10^3 \times V^2}{2 \times 9.81 \times 0.10}$$

or
$$V = 1.404 \text{ m/s}$$

Total discharge through 5 pipes

$$= \left[\frac{\pi}{4} \times (0.10)^2 \times 5 \times 1.404\right]$$

$$= 5.513 \times 10^{-2} \text{ m}^3/\text{s}$$

$\therefore$ Maximum power in kW that can be transmitted in this case

$$= \frac{9810 \times 5.513 \times 10^{-2} \times (722.87 - 240.96)}{1000}$$

$$= 260.63 \text{ kW}$$

Note. The efficiency corresponding to the maximum power transmission is only 66.7%

Example 11.17. *A 0.5 m diameter and 100 m long pipeline carrying 0.5 m³/s discharge is fitted with a valve at downstream end. Calculate the rise of pressure caused within the pipe due to the valve closure in (i) 1 second, (ii) instantaneously. Take the sonic velocity as 1430 m/s.*

Solution

(i) Time of value closure $t = 1$s.

The time taken by the sound wave to travel to and fro the valve is

$$t' = \frac{2L}{c} = \frac{2 \times 100}{1430} = 0.1399 \text{ s}$$

Since in this case $t' < t$, the valve closure may be considered as gradual.

Thus from Eq. 11.34, the rise of pressure

$$p_i = \frac{wLV}{gt}$$

$$w = 1000 \text{ kg/m}^3 \text{ or } 9.81 \text{ kN/m}^3; L = 100 \text{ m};$$

$$V = \frac{0.5}{(\pi/4) \times (0.5)^2} = 2.546 \text{ m/s}; g = 9.81 \text{ m/s}^2; \text{and } t = 1.0_s$$

Thus by substitution, we get

$$p_i = \frac{1000 \times 100 \times 2.546}{9.81 \times 1.0} = 2.595 \times 10^4 \text{ kg/m}^2$$

or
$$p_i = \frac{9.81 \times 100 \times 2.546}{9.81 \times 1.0} = 2.546 \times 10^2 \text{ kN/m}^2$$

(ii) For instantaneous closure of value, the rise of pressure is given by Eq. 11.37, Allievi formula, as

$$p_i = \frac{wVV_0}{g}$$

$$V_0 = 1430 \text{ m/s}$$

Thus by substitution, we get

$$p_i = \frac{1000 \times 2.546 \times 1430}{9.81} = 3.711 \times 10^5 \text{ kg/m}^2$$

or
$$p_i = \frac{9.81 \times 2.546 \times 1430}{9.81} = 3.641 \times 10^3 \text{ kN/m}^2$$

Example 11.18. *A steel penstock 0.60 m in diameter has a shell thickness of 12 mm. The modulus of elasticity of the pipe shell material is 2.1 × 10⁵ N/mm² [2.1 × 10⁶ kg(f)/cm²] with Poisson's ratio of $\frac{1}{4}$ and the volume modulus of elasticity of water is 2.1 × 10³ N/mm² [2.1 × 10⁴ kg(f)/cm²]. The pipe is designed to discharge water at a mean velocity of 2.1 m/s. Determine the water hammer pressure rise caused by sudden closure of the valve at the downstream end:*

(1) neglecting the elasticity of the pipe material; (2) considering the elasticity of the pipe meterial but neglecting the Poisson's ratio; and (3) considering the elasticity of the pipe material including the Poisson's ratio.

Solution

1. In this case water hammer pressure is given by Eq. 11.36 as

$$p_i = V\sqrt{\frac{wK}{g}}$$

SI units

$$p_i = 2.1\sqrt{\frac{9810 \times 2.1 \times 10^3 \times 10^6}{9.81}}$$

$$= 3.043 \times 10^6 \text{ N/m}^2$$

Meteric units

$$p_i = \sqrt{\frac{1000 \times 2.1 \times 10^4 \times 10^4}{9.81}}$$

$$= 3.073\ 10^5 \text{ kg (f)/m}^2$$

2. In this case water hammer pressure is given by Eq. 11.38 as

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K} + \frac{D}{TE}\right)}}$$

SI units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{9810}\left(\frac{1}{2.1 \times 10^3 \times 10^6} + \frac{0.60}{12 \times 10^{-3} \times 2.1 \times 10^5 \times 10^6}\right)}}$$

$$= 2.485 \times 10^6\ \text{N/m}^2$$

Metric units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{1000}\left(\frac{1}{2.1\times10^4\times10^4}+\frac{0.60}{12\times10^{-3}\times2.1\times10^6\times10^4}\right)}}$$

$$= 2.509 \times 10^5\ \text{kg(f)/m}^2$$

3. In this case water hammer pressure is given by Eq. 11.35 as

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}}$$

SI units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{9810}\left[\frac{1}{2.1\times10^3\times10^6}+\frac{0.60}{12\times10^{-3}\times2.1\times10^5\times10^6}\left(1-\frac{1}{2\times4}\right)\right]}}$$

$$= 2.538\ \ 106\ \text{N/m}^2$$

Metric units

$$p_i = \frac{2.1}{\sqrt{\frac{9.81}{1000}\left[\frac{1}{2.1\times10^4\times10^4}+\frac{0.60}{12\times10^{-3}\times2.1\times10^6\times10^4}\left(1-\frac{1}{2\times4}\right)\right]}}$$

$$= 2.563 \times 10^5\ \text{kg(f)/m}^2$$

Example 11.19. *What is the maximum permissible velocity in a cast iron pipeline 100 mm diameter and 15 mm thick, which can be suddenly stopped by a valve at the outlet end of the pipe without letting the rise of pressure in the pipe to exceed 1.545×10^3 kN/m² [15.75 kg (f)/cm²]? Take E for cast iron as 123.606×10^9 N/m² [12.6× 10^5 kg(f)/cm²]and K for water as 206.01×10^7 N/m² [21 × 10^3kg(f)/cm²].*

Solution

From Eq. 11.38, inertia pressure

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K}+\frac{D}{TE}\right)}}$$

SI units

$$p_i = 1.545 \times 10^3\ \text{kN/m}^2 = 1.545 \times 10^6\ \text{N/m}^2$$
$$K = 206.01 \times 10^7\ \text{N/m}^2$$
$$E = 123.606 \times 10^9\ \text{N/m}^2$$
$$D = 100\ \text{mm} = 0.10\ \text{m}$$
$$T = 15\ \text{mm} = 0.015\ \text{m}$$

Thus by substitution, we get

$$1.545 \times 10^6 = \frac{V}{\sqrt{\frac{9.81}{9810}\left(\frac{1}{206.01\times10^7}+\frac{0.10}{0.015\times123.606\times10^9}\right)}}$$

$$\therefore \quad V = 1.135\ \text{m/s}$$

Metric units

$$p_i = 15.75\ \text{kg(f)/cm}^2 = 15.75\times10^4\ \text{kg(f)/m}^2$$
$$K = 21\times10^3\ \text{kg(f)/cm}^2 = 21\times10^7\ \text{kg(f)/m}^2$$
$$E = 12.6\times10^5\ \text{kg(f)/cm}^2 = 12.6\times10^9\ \text{kg(f)/m}^2$$
$$D = 100\ \text{mm} = 0.10\ \text{m}$$
$$T = 15\ \text{mm} = 0.015\ \text{m}$$

Thus by substitution, we get

$$15.75\times10^4 = \frac{V}{\sqrt{\frac{9.81}{1000}\left(\frac{1}{21\times10^7}+\frac{0.10}{0.015\times12.6\times10^9}\right)}}$$

$$\therefore \quad V = 1.135\ \text{m/s}$$

Example 11.20. *For a pipe network shown in Fig. Ex. 11.20, determine the flow in each pipe. The value of n may be assumed as 2.0.*

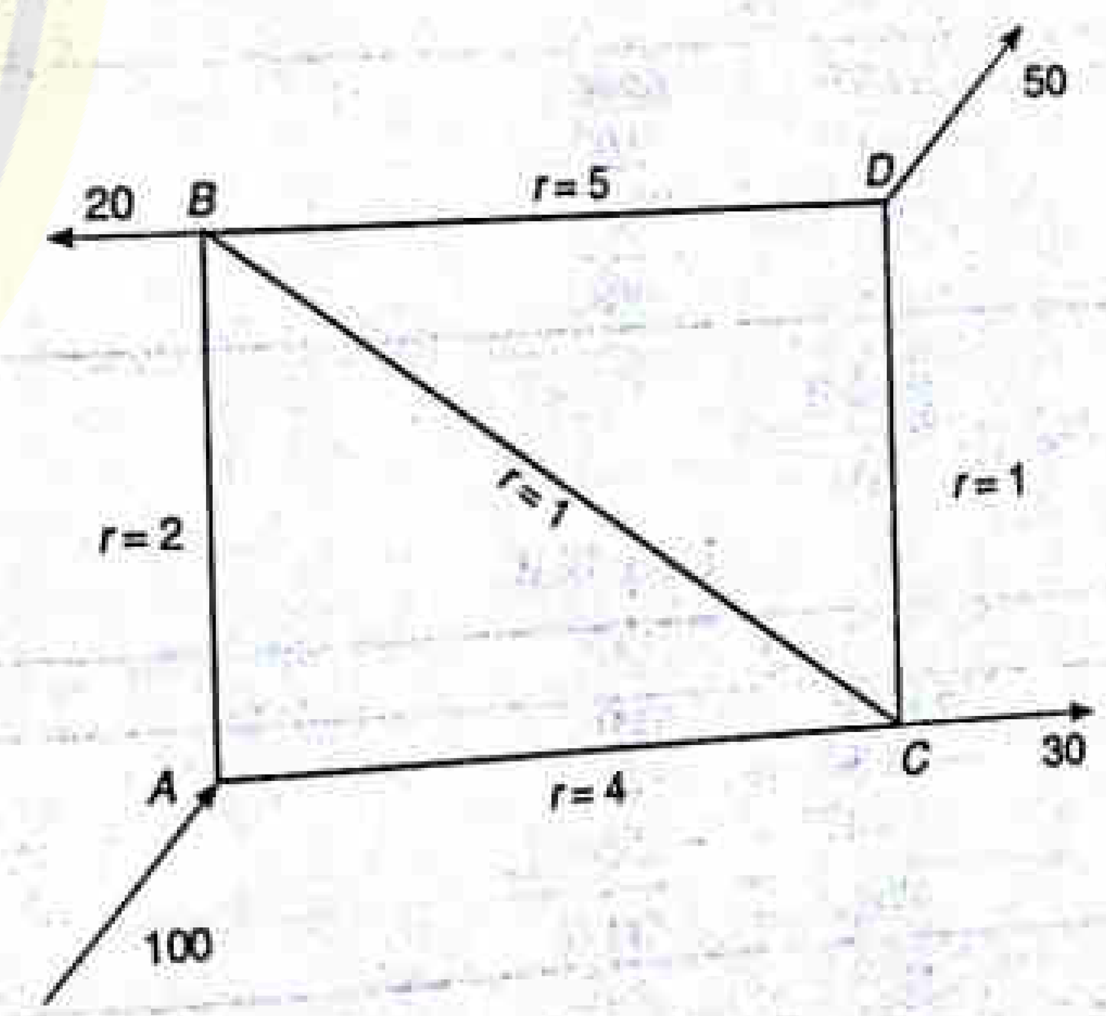


Fig. Ex. 11.20

Solution

For the first trial the distribution is assumed as shown in Fig. 11.20 (a). For this distribution the corrections ΔQ for the loops *ABC* and *BDC* are computed as follows:

Loop ABC

Pipe	rQ^2	$2rQ$
AB	$2 \times 70^2 = 9800$	$2 \times 2 \times 70 = 280$
BC	$1 \times 35^2 = 1225$	$2 \times 1 \times 35 = 70$
AC	$-4 \times 30^2 = -3600$	$2 \times 4 \times 30 = 240$
	$\Sigma rQ^2 = 7425$	$\Sigma 2rQ = 590$

$$\therefore \quad \Delta Q = -\frac{7425}{590} \approx -13$$

Loop BDC

Pipe	rQ^2	$2rQ$
BD	$5 \times 15^2 = 1125$	$2 \times 5 \times 15 = 150$
DC	$-1 \times 35^2 = -1225$	$2 \times 1 \times 35 = 70$
CB	$-1 \times 35^2 = -1225$	$2 \times 1 \times 35 = 70$
	$\Sigma rQ^2 = -1325$	$\Sigma 2rQ = 290$

$$\therefore \quad \Delta Q = -\frac{(-1325)}{290} \approx 5$$

Thus applying the above obtained corrections the modified discharges for various pipes are shown in Fig. Ex. 11.20 (b), which is the distribution for the second trial. For this distribution the corrections ΔQ for the loops *ABC* and *BDC* are computed as follows:

Loop ABC

Pipe	rQ^2	$2rQ$
AB	$2 \times 57^2 = 6498$	$2 \times 2 \times 57 = 228$
BC	$1 \times 17^2 = 289$	$2 \times 1 \times 17 = 34$
AC	$-4 \times 43^2 = -7396$	$2 \times 4 \times 43 = 344$
	$\Sigma rQ^2 = -609$	$\Sigma 2rQ = 606$

$$\therefore \quad \Delta Q = \frac{(-609)}{606} \approx 1$$

Loop BDC

Pipe	rQ^2	$2rQ$
BD	$5 \times 20^2 = 2000$	$2 \times 5 \times 20 = 200$
DC	$-1 \times 30^2 = -900$	$2 \times 1 \times 30 = 60$
CB	$-1 \times 17^2 = -289$	$2 \times 1 \times 17 = 34$
	$\Sigma rQ^2 = 811$	$\Sigma 2rQ = 294$

$$\therefore \quad \Delta Q = \frac{811}{294} \approx -3$$

Thus applying the above obtained corrections the modified discharges for the various pipes are shown in Fig. Ex. 11.20 (c), which is the distribution for the third trial. For this distribution the corrections ΔQ for loops *ABC* and *BDC* are computed as follows:

Loop ABC

Pipe	rQ^2	$2rQ$
AB	$2 \times 58^2 = 6728$	$2 \times 2 \times 58 = 232$
BC	$1 \times 21^2 = 441$	$2 \times 1 \times 21 = 42$
AC	$-4 \times 42^2 = -7056$	$2 \times 4 \times 42 = 336$
	$\Sigma rQ^2 = 113$	$\Sigma 2rQ = 610$

$$\therefore \quad \Delta Q = \frac{113}{610} \approx 0 \text{ (negligible)}$$

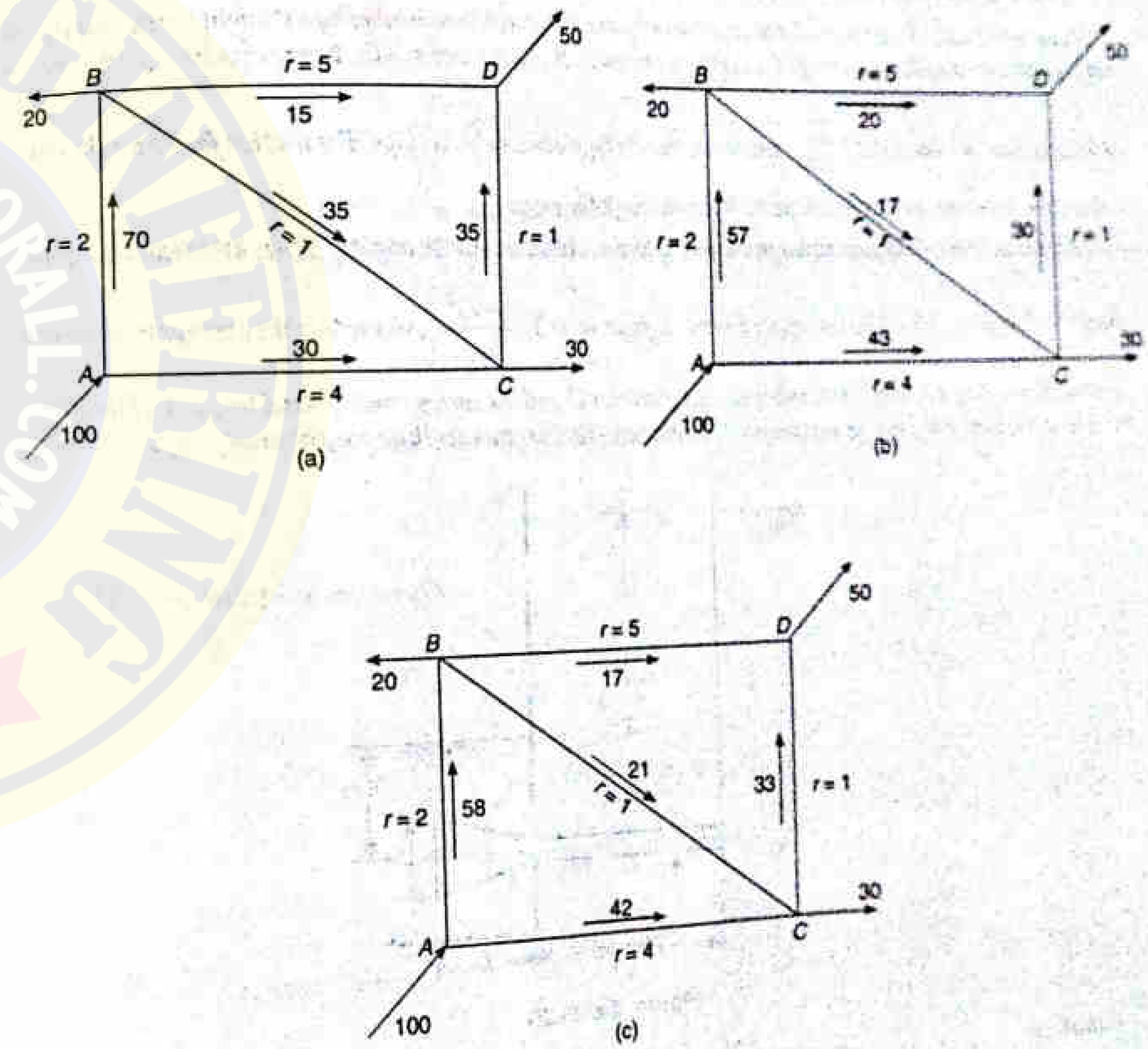


Fig. Ex. 11.20

Loop BDC

Pipe	rQ^2	$2rQ$
BD	$5 \times 17^2 = 1445$	$2 \times 5 \times 17 = 170$
DC	$-1 \times 33^2 = -1089$	$2 \times 1 \times 33 = 66$
CB	$-1 \times 21^2 = -441$	$2 \times 1 \times 21 = 42$
	$\Sigma rQ^2 = -85$	$\Sigma 2rQ = 278$

$$\therefore \quad \Delta Q = \frac{(-85)}{278} = 0 \text{ (negligible)}$$

Thus Fig. Ex. 11.20 (c) shows the correct distribution of flow for the given network of pipes.

Example 11.21. *Two reservoirs of large capacity are connected to each other by a straight horizontal pipe 80 mm in diameter and 800 mm long. The difference between the water levels in the two reservoirs is 10 m. The loss of head at pipe entrance is 0.1 $\frac{v^2}{2g}$, where v is the average velocity in the pipe. The friction factor for the pipe is 0.03. Determine the rate at which water flows through the pipe.*

To increase the rate of flow through the pipe a conical diffuser with an angle of 16° is attached to the pipe as shown in Fig. Ex. 11.18. The loss of head in the diffuser is $\left[0.3\frac{(v-V)^2}{2g}\right]$, where V is the average velocity at the exist section of the diffuser. Determine the exit diameter D and the corresponding axial length L of the diffuser for which the flow rate will be maximum. Compute the discharge under these conditions.

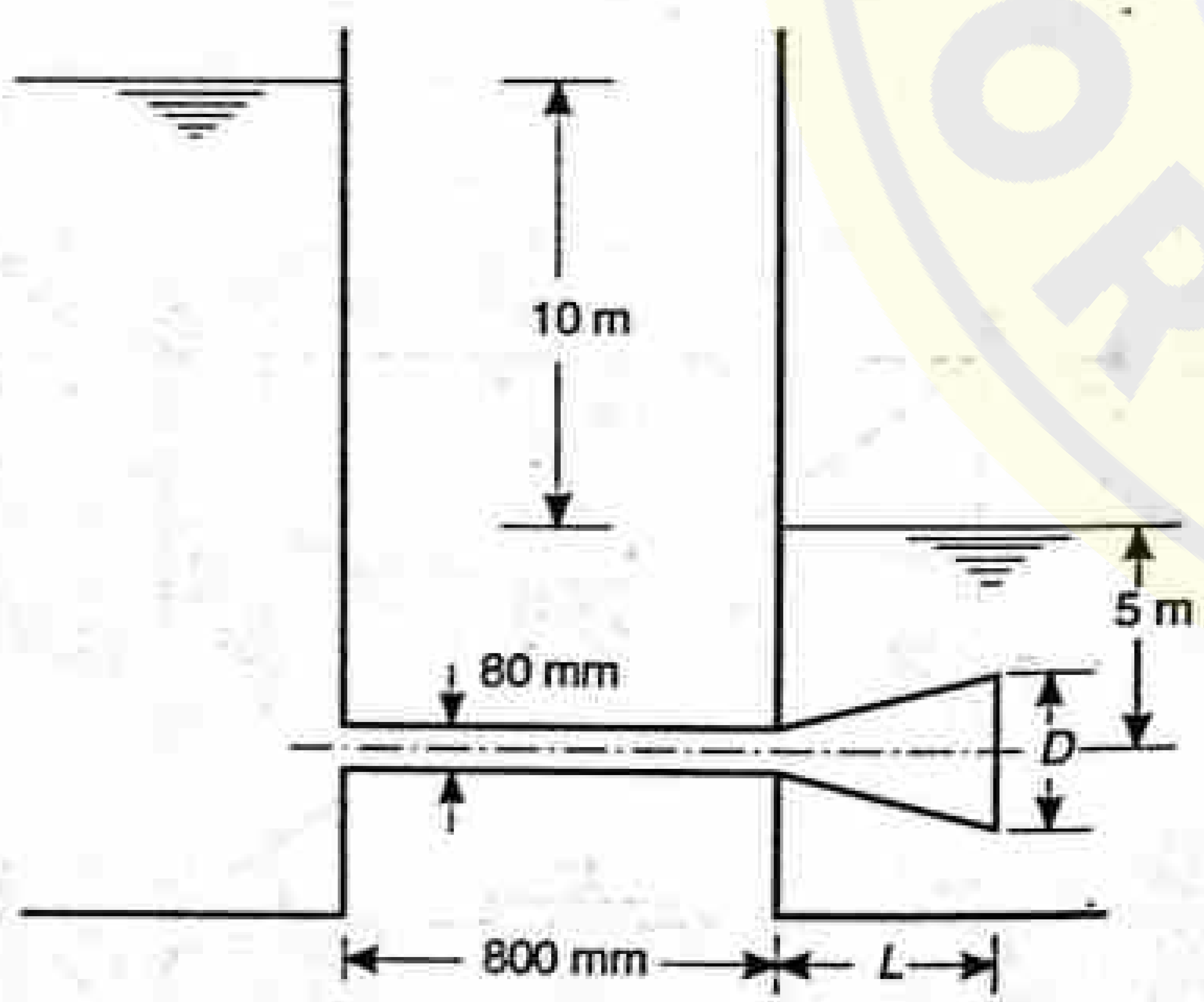


Figure Ex. 11.21

Solution

Applying Bernoulli's equation between the two reservoirs, we get

$$10 = 0.1\frac{v^2}{2g} + \frac{v^2}{2g} + \frac{fLv^2}{2gd}$$

or
$$10 = \frac{v^2}{2g}\left[1.1 + \frac{fL}{d}\right]$$

$$f = 0.03;\ L = 800\text{ mm} = 0.8\text{ m; and } d = 80\text{ mm} = 0.08\text{ m}$$

Thus by substitution, we get

$$10 = \frac{v^2}{2g}\left[1.1 + \frac{0.03 \times 0.8}{0.08}\right]$$

or
$$v = 11.84\text{ m/s}$$

$\therefore$ Discharge
$$Q = \frac{\pi d^2}{4} \times v$$
$$= \frac{\pi \times (0.08)^2}{4} \times 11.84$$
$$= 0.0595\text{ cumec}$$

When diffuser is attached to the pipe, then applying Bernoulli's equation between the two reservoirs, we get

$$10 = 0.1\frac{v^2}{2g} + \frac{V^2}{2g} + \frac{fLv^2}{2gd} + 0.3\frac{(v-V)^2}{2g}$$

If Q is the discharge, then by continuity, we have

$$Q = \frac{\pi d^2}{4} \times v = \frac{\pi D^2}{4} \times V$$

Thus by substitution, we get

$$10 = \frac{Q^2}{(\pi/4)^2 \times 2g}\left[\frac{0.1}{d^4} + \frac{1}{D^4} + \frac{fL}{d} \times \frac{1}{d^4} + 0.3\left(\frac{1}{d^2} - \frac{1}{D^2}\right)^2\right]$$

or
$$Q^2 = \frac{10 \times (\pi/4)^2 \times 2g}{\left[\frac{0.1}{d^4} + \frac{1}{D^4} + \frac{fL}{d} \times \frac{1}{d^4} + 0.3\left(\frac{1}{d^2} - \frac{1}{D^2}\right)^2\right]}$$

Let $\frac{d}{D} = x$, then we have

$$Q^2 = \frac{10 \times (\pi/4)^2 \times 2g}{\left[\frac{0.1}{d^4} + \frac{fL}{d} \times \frac{1}{d^4} + \frac{x^4}{d^4} + \frac{0.3}{d^4}(1 - x^2)^2\right]}$$

or
$$Q = \frac{\sqrt{10\times(\pi/4)^2\times 2g}}{\sqrt{\left[\frac{0.1}{d^4}+\frac{fL}{d}\times\frac{1}{d^4}+\frac{1}{d^4}\left\{x^4+0.3(1-x^2)^2\right\}\right]}}$$

For Q to be maximum, $\frac{dQ}{dx}=0$.

Then since f, L and d are constant, we get

$$4x^3+0.3\times 2(1-x^2)(-2x) = 0$$

or
$$x = 0.4804$$

$\therefore$
$$D = \frac{d}{0.4804} = 2.082\,d$$

$$d = 0.08 \text{ m}$$

$\therefore$
$$D = 0.08\times 2.082 = 0.1666 \text{ m}$$

The maximum discharge is given by

$$Q_{max} = \frac{\sqrt{10\times(\pi/4)^2\times 2\times 9.81}}{\sqrt{\frac{0.1}{(0.08)^4}+\frac{0.03\times 0.8}{(0.08)^5}+\frac{1}{(0.08)^4}\left[(0.4804)^4+0.3\left\{1-(0.4804)^2\right\}^2\right]}}$$

$\therefore$
$$Q_{max} = 0.0887 \text{cumec}$$

The axial length L is given by

$$L = \frac{1}{2}(0.1666-0.08)\cot 8^\circ$$
$$= 0.0433\times 7.1154$$
$$= 0.3081 \text{ m} = 308.1 \text{ mm}$$

SUMMARY OF MAIN POINTS

1. A pipe is a closed conduit used for carrying fluids under pressure. Pipes are commonly circular in section. As pipes carry fluids under pressure, the pipes always run full.
2. The flow of fluid in a pipe be either laminar or turbulent.

   A dimensionless parameter called *Reynolds number* is used to distinguish these two types of flows in a pipe. Reynolds number is given by

   $$\text{Re or } N_R = \frac{\rho VD}{\mu} = \frac{VD}{\nu}$$

   where

   ρ = mass density of the flowing fluid ;
   V = mean or average velocity of flow of fluid;
   D = diameter of the pipe ; a
   μ = absolute or dynamic viscosity or viscosity of the flowing fluid ; and
   ν = kinematic viscosity of the flowing fluid $\left(=\frac{\mu}{\rho}\right)$

   The flow in a pipe will be laminar if the Reynolds number is less than 2000 and turbulent if it is more than 4000. If the Reynolds number is between 2000 and 4000 the flow is considered to be in transition.
3. When a fluid flows through a pipe, certain resistance is offered to the flowing fluid which results in causing a loss of energy or head. The various energy or head losses in pipes may be classified as
   (i) Major losses
   (ii) Minor losses

   The major loss of energy or head is caused by friction and minor losses of energy or head are caused due to sudden expansion or enlargement of pipe, sudden contraction of pipe, bend in pipe and an obstruction in pipe.
4. The loss of energy or head due to friction is given by Darcy–Weisbach equation as

   $$h_f = \frac{fLV^2}{2gD}$$

   where

   f = friction factor ;
   L = length of pipe ;
   V = mean velocity of flow in pipe ;
   D = diameter of pipe ; and
   g = acceleration due to gravity.
5. The loss of energy or head due to friction in pipe may also be determined by using the following formulae.

   (i) Chezy's formula, $V = C\sqrt{mi}$ or $V = C\sqrt{RS}$

   (ii) Manning's formula, $V = \frac{1}{n}R^{2/3}S^{1/2}$.

   (iii) Hazen-Willam's formula, $V = 0.85\,C_1R^{0.63}S^{0.54}$

   where

   V = mean velocity of flow in the pipe ;
   C = Chezy's coefficient ;
   m or R = hydraulic mean depth or hydraulic radius $\left(=\frac{A}{P}\right)$, A being wetted area and P being wetted perimeter of flow section of pipe.
   i or S = slope of energy grade line $\left(=\frac{h_f}{L}\right)$, h_f being head loss due to friction and L being the length of pipe ;
   n = Manning's roughness coefficient ; and
   C_1 = Hazen-William's coefficient since pipe always runs full

   $$m \text{ or } R = \frac{D}{4}$$

   where

   D = diameter of pipe.
6. (i) The loss of energy or head due to sudden enlargement or expansion is given by

   $$h_L = \frac{(V_1-V_2)^2}{2g}$$

   where

   V_1 = velocity of flow of fluid in smaller pipe ; and
   V_2 = velocity of flow of fluid in larger pipe.

   (ii) The loss of energy or head due to sudden contraction is given by

   $$h_L = 0.5\,\frac{V^2}{2g}$$

   where

   V = velocity of flow of fluid in smaller pipe.

   (iii) The loss of energy or head in a bend is given by

   $$h_L = k\frac{V^2}{2g}$$

   where

   k = a coefficient the value of which mainly depends on the angle of the bend ; and
   V = mean velocity of flow of fluid

   (iv) The loss of energy or head due to fluid obstruction is given by

   $$h_L = \left[\frac{A}{C_c(A-a)}-1\right]^2\frac{V^2}{2g}$$

   where

   A = cross-sectional area of pipe ;
   a = maximum cross-sectional area of the obstruction ;
   C_c = coefficient of contraction, assumed to vary from 0.60 to 0.66 ; and
   V = mean velocity of flow of fluid at unobstructed section of pipe.

   (v) The loss of energy or head due to various pipe fittings is given by

$$h_L = k\frac{V^2}{2g}$$

(vi) The loss of energy or head at the entrance to a pipe is given by

$$h_L = 0.5\frac{V^2}{2g}$$

(vii) The loss of energy or head at the exit of a pipe is given by

$$h_L = \frac{V^2}{2g}$$

7. The line representing the sum of pressure head and datum head with respect to some datum is known as *hydraulic grade line* (H.G.L).

The line representing the total energy or head i.e., the sum of pressure head, datum head with respect to some datum and velocity head is known as *energy grade* line or *total energy* line (T.E.L)

8. When several pipes of different lengths and different diameters are connected in series, it is known as compound pipe. The discharge through each pipe connected in series is same. A compouned pipe may be replaced by an equivalent pipe.

An *equivalent pipe* is a single pipe of uniform diameter and having same length, same discharge and same loss of energy or head as a compound pipe consisting of several pipes of different lengths and different diameters connected in series.

The diameter of equivalent pipe is given by Dupuit's equation as

$$\frac{L}{D^5} = \left[\frac{L_1}{D_1^5}+\frac{L_2}{D_2^5}+\frac{L_3}{D_3^5}+.........\right]$$

where

L = length of equivalent pipe (= L_1+ L_2+ L_3+ ....) ;

D = diameter of equivalent pipe ;

L_1, L_2, L_3.... = lengths of different pipes connected in series ; and

D_1, D_2, D_3 ... = diameters of different pipes connected in series .

9. When the pipes are connected in parallel, the loss of head in each pipe is same. The discharge in the main pipe is equal to the sum of the discharges in each pipe connected in parallel.

10. For solving the problems of branched pipes i.e., pipes connecting different reservoirs, the three basic equations viz., continuity equation, Bernoulli's equation and Darcy-Weisbach equation are used.

11. A siphon is a long bent pipe which is used to carry water from a reservoir at a higher elevation to another reservoir at a lower elevation when the two reservoirs are separated by a hill or high level ground. The maximum vacuum pressure at the summit of a siphon may be 7.4 m of water.

12. The loss of energy or head due to friction in a tapering pipe of length l and diameters D_1 and D_2 at the ends (D_1>D_2) is given by

$$h_f = \frac{4fQ^2l^5}{2g\lambda^2D_2^5}\left[\frac{1}{l^4}-\frac{1}{(L+l)^4}\right]$$

in which

$$l = \frac{D_2L}{D_1-D_2}$$

13. The loss of energy or head due to friction in a pipe of length L and diameter D with side tappings through which liquid is removed at a uniform rate of q per unit length of pipe is given by

$$h_f = \frac{f}{D}\frac{V^2}{2g}\frac{Q}{3q}\left[1-\left(1-\frac{qL}{Q}\right)^3\right]$$

where Q is the discharge at the inlet end of the pipe.

If the end of the pipe is closed so that $(Q-qL)=0$; then

$$h_f = \frac{f}{D}\frac{V^2}{2g}\frac{L}{3}$$

14. Power transmitted through a pipe is given by

$$P = wQ=(H-h_f)$$

where

w = specific or unit weight of water ;

Q = discharge ;

H = total head at inlet of pipe ; and

h_f = loss of head due to friction. The efficiency of power transmission through pipe is given by

$$\eta = \frac{H-h_f}{H}$$

The power transmitted through a pipe is maximum when $h_f = \frac{H}{3}$, and corresponding efficiency $\eta = \frac{2}{3}$ or 66.67%

15. The velocity of flow at the outlet of the nozzle fitted at the end of a pipe is given by

$$v = \sqrt{\frac{2gh}{\left(1+\frac{fL}{D}\frac{a^2}{A^2}\right)}}$$

where

H = head at the inlet of the pipe ;

L = length of the pipe ;

D = diameter of the pipe ;

a = cross-sectional area of the nozzle outlet ; and

A = cross-sectional area of the pipe.

The power transmitted by the jet issuing from the nozzle is given by

$$P = wQ\,(H-h_f)$$

in which

$$Q = av = AV \text{ ; and}$$

$$h_f = \frac{fLV^2}{2gD} = \frac{fL}{D}\frac{a^2}{A^2}\frac{v^2}{2g}$$

The efficiency of power transmission through nozzle is given as

$$\eta = \left[\frac{1}{1+\frac{fL}{D}\frac{a^2}{A^2}}\right]$$

The power transmitted by a nozzle is maximum when $h_f = \frac{H}{3}$

The diameter of the nozzle transmitting maximum power is given by

$$d = \left(\frac{D^5}{2fL}\right)^{1/4}$$

where

d = diameter of the nozzle ;

D = diameter of the pipe ;

L = length of the pipe, and

f = friction factor

16. When water flowing in a long pipe is suddenly brought to rest by closing the valve or by any similar cause, there will be a sudden rise in pressure and a pressure wave of high intensity will be transmitted along the pipe. This phenomenon of sudden rise in pressure in the pipe is known as *water hammer* or *hammer blow*.

The intensity of pressure rise due to water hammer is given by

(i) When valve is closed gradually

$$p_i = \frac{wLV}{gt}$$

(ii) When valve is closed suddenly and pipe is elastic

$$p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}}$$

(iii) When valve is closed suddenly and pipe is considered rigid

$$p_i = V\sqrt{\frac{wK}{g}}$$

or

$$\frac{p_i}{w} = \frac{V}{g}\sqrt{\frac{K}{\rho}} = \frac{VV_0}{g} \quad \text{(Allievi formula)}$$

where

L = length of pipe ;

V = velocity of flow ;

t = time taken to close the valve ;

D = diameter of the pipe ;

K = bulk modulus of water ;

E = Young's modulus of pipe material ;

T = wall thickness of pipe ;

m = poisson's ratio ;

w = specific or unit weight of water ;

ρ = mass density of water ; and

g = acceleration due to gravity.

$V_0 = \sqrt{\frac{K}{\rho}}$ = velocity of propogation of sound wave in the liquid.

17. For solving the problems of pipe network the conditions to be satisfied are as follows :

(i) The flow into each junction must be equal to the flow out of the junction

(ii) In each loop the loss of head due to flow in clockwise direction must be equal to the loss of head due to flow in anticlockwise direction.

(iii) The Darcy–Weisbach equation must be satisfied for flow in each pipe, according to which the loss of head h_f through any pipe discharging at the rate of Q can be expressed as

$$h_f = rQ^n$$

where r is a proportionality factor

$$\left(= \frac{fL}{2g(\pi/4)^2 D^5} = \frac{fL}{12.10D^5}\right)$$ and n is an exponent having a numerical value ranging from 1.72 to 2.00

PROBLEMS

11.1 Distinguish between laminar flow and turbulent flow in pipes.

11.2 Explain the terms hydraulic gradient and total energy lines.

11.3 What is a compound pipe? How would you determine the equivalent size of a compound pipe?

11.4 Why are the pipes connected in parallel?

11.5 What is a siphon? Where is it used? Explain its working.

11.6 Obtain the condition for maximum efficiency in transmission of power through a pipeline?

11.7 What is meant by water hammer ? Obtain an expression for the rise in pressure in a thin elastic pipe of circular section in which the flow of water is stopped by sudden closure of valve.

11.8 A pipe line, 16 km long, supplies, 40 million litres of water per day to city. The first 5 km length of the pipe is of 1 m diameter and the remaining part is 0.8 m diameter pipe. If the water to the city is to be supplied at a residual head of 15 m of water, calculate the supply head at the inlet end. Neglect minor losses and assume f = 0.03 for the entire pipe line. Sketch the hydraulic gradient for the pipe line. [*Ans.* 35.53 m]

11.9 A pipe 0.15 m diameter taking off from a reservoir suddenly expands to 0.3 m at the end of 16 m and continues for another 15 m. If the head above the inlet of the pipe is 4.88 m, determine the actual velocity at the exit, taking into consideration all the losses. Assume f = 0.04 for the complete pipe line. [*Ans.* 1.07 m/s]

11.10 A hydraulic power main 4.5 km long and 0.225 m in diameters is tapped at various points throughout its length at an average rate 0.00135 m^3/sec per 100 m length. Assuming f = 0.02, calculate the power lost due to friction in the pipe if (i) the pumping station is situated, at one end of the main, the other end being closed; (ii) pumping station is placed mid-way, the two ends of the main being closed. Start from first principles. [*Ans.* (i) 7.09 kW; (ii) 0.89 kW]

11.11 Two reservoirs are cennected by a pipe 2250 m long and 0.225 m in diameter, the difference in water levels being 7.5 m. Determine the flow through the pipe in litres per minute if f = 0.03. Also find the percentage increase in the discharge if for the last 600 m a second pipe of the same diameter is laid along side the first. [*Ans.* 1666.61 litres/min; 12.1%]

12.12 Two reservoirs are connected by three pipes laid in parallel, their diameters are d, $2d$ and $3d$ respectively, and they are of the same length L. Assuming f to be the same for all pipes, determine the discharge through each of the larger pipes if the smallest pipe is discharging 1 cumec. [*Ans.* 5.66 m^3/sec; 15.6 m^3/sec]

11.13 Water flows out of tank, wherein water stands at El. 140 m; through a pipe 4 km long and 0.3 m diameter upto a point where the elevation is El. 90 m and one-third of the discharge flowing is tapped out to be used by a power machine. The remaining discharge flows down through another pipe, 5 km long and 0.2 m in diameter, into another tank wherein water stands at El. 110 m. Use Darcy's formula $h_f = \left(\frac{LQ^2}{600D^5}\right)$ with usual notation. Determine the power available at the tapping point for the power machine. [*Ans.* 6.63 kW (9.01 h.p.)]

11.14 Two reservoirs whose water surface are at an elevation difference of H meters are connected by a pipeline, of diameter D metres and length L metres, which is uniformly tapped all along its length such that the discharge entering it is halved before it reaches the other end. Derive an expression for the discharge Q in cumec entering the pipeline. Use Darcy's formula $h_f = \left(\frac{LQ^2}{600D^5}\right)$. Start from first principles. Neglect minor losses. $\left[Ans.\ \sqrt{\frac{7200D^5H}{7L}}\right]$

11.15 A pipe 0.2 m diameter and 1800 m long connects two reservoirs one being 30 m below the other. The pipeline crosses a ridge whose summit is 7.5 m above the upper reservoir. What may be the minimum depth of the pipe below the summit of the ridge in order that the pressure at the apex does not fall below 7.5 m vacuum? The length of the pipe from the upper reservoir to the apex is 300 m. Taking f = 0.032, determine the rate of flow to the lower reservoir in litres per minute. [*Ans.* 2.37m; 2688 litres/min]

11.16 A turbine operating under a total fall of 36 m is supplied water through 90 m long 0.2 m diameter pipe. If the flow rate is such that 9.5 m of head is lost in friction in the pipe, what will be the power delivered to the turbine? Take f = 0.022. [*Ans.*35.43 kW(48.16 h.p.)]

11.17 The water levels in the two reservoirs A and B are respectively 66 m and 61.5 m above datum. A pipe joins each to a common point D where the pressure is 103 kN/m^2 gauge and height is 45 m above datum. Another pipe connects D to another tank C. What will be the height of water level in C assuming the same value of friction factor for all pipes?

Pipe	Length	Diameter
AD	2400 m	0.30 m
BD	2700 m	0.45 m
CD	3000 m	0.60 m

[*Ans.* 51.90 m above datum]

11.18 Water is flowing in a long pipe which is 0.15 m diameter and 6.25 mm thick with a velocity of 1.2 m/s and it is suddenly brought to rest by the closing of a valve. Calculate the inertia pressure and the theoretical stress produced in the pipe near the valve, taking K for water as 207 × 10^4 kN/m^2 [21.1 × 10^3 kg(f)/cm^2] and E for the pipe material as 206 × 10^6 kN/m^2 [21 × 10^5 kg(f)/cm^2].

[*Ans.*155 × 10^4 N/m^2 [15.8 kg (f)/cm^2]; 1.86 × 10^4 kN/m^2 [189.6 kg(f)/cm^2]]

11.19 A 0.3 m wrought iron pipeline 560 m long discharges water 100 m below the surface of reservoir. Determine the diameter of the nozzle which will deliver the maximum power. Assume f = 0.022 and the coefficient of velocity for the nozzle is 0.97. [*Ans.* 101.2 mm]

11.20 A water main of concrete pipe 3200 m long and 0.3 m diameter discharges into a reservoir at the rate of 10 × 10^6 litres per day. If this line is gradually closed by operating a valve at the reservoir end in 16 seconds, show that there is a possibility of pipe burst. The safe pressure of concrete pipe is 245.25 kN/m^2 [2.5 kg(f)/cm^2]

[*Ans.* p_i = 327.48 kN/m^2 [3.34 kg(f)/cm^2], which is greater than safe pressure of concrete pipe and hence there is possibility of pipe burst]

11.21 Show that a long pipe with nozzle at its end has greater velocity of efflux but less discharge as compared to a similar pipe without nozzle at its end.

11.22 Water flows through a horizontal pipe 2 m long, which tapers from a diameter of 0.2 m to 0.15 m in the length 2 m. A constant discharge of 40 litres per second flows through the pipe. Starting from first principles, determine the loss of head due to friction. Take f = 0.04. [*Ans.* 0.0714 m]

11.23 A pipe 0.15 m diameter, 15 m long is connected to the bottom of a tank 15 m long and 12 m wide. The original head over the open end of the pipe is 5 m. Find the time of emptying the tank. Assume the entrance to the pipe to be sharp-edged and f = 0.01 in $h_f = \frac{flV^2}{2gd}$. [*Ans.* 4.52 hrs.]

11.24 Two reservoirs A and C having a difference of level of 15.5 m are connected by a pipeline ABC, the elevation of the point B being 4 m below the level of water in the reservoir A. The length AB of the pipeline is 250 m, the pipe being made of mild steel having a friction coefficient f_1, while the length BC is 450 m, the pipe being made of cast iron having a friction coefficient f_2. Both the lengths AB and BC have a diameter 0.20 m. A partially closed valve is located in the length BC at a distance of 150 m from reservoir C. If the flow through the pipeline is 3000 litres/minute, the pressure head at B is 0.5 m and the head loss at the valve is 5.0 m, find the friction coefficients f_1 and f_2. Draw the hydraulic grade line of the pipeline and indicate on the diagram head loss values of significant points. Take into account head loss at entrance and exit points of the pipeline. [*Ans.* $f_1 = 0.0205$, $f_2 = 0.0241$]

11.25 If two pipes of diameter D and d and equal length L are arranged in parallel the loss of head for a flow of Q is h. If the same pipes are arranged in series the loss of head for the same flow Q is H. If $d = 0.5\,D$, find the percentage of total flow through each pipe when placed in parallel and the ratio (H/h). Neglect minor losses and assume f to be constant. [*Ans.* 85%, 15%; 45.7]

11.26 Deduce an expression for loss of head in a pipeline with laterals assuming that velocity changes uniformly from V_1 to V_2 in a length L of the pipeline of constant diameter D and friction factor f.

$$\left[\textit{Ans.}\ \frac{fL}{6gD}\left(V_1^2+V_2^2+V_1V_2\right)\right]$$

[**Note** : The expression obtained in this case is same as given by Eq. 11.22. By considering $\left(Q/\frac{\pi D^2}{4}=V_1\right)$ and $\left[(Q-qL)/\frac{\pi D^2}{4}=V_2\right]$, we obtain $\left[(qL/Q)=((V_1-V_2)/V_1)\right]$ and introducing this in Eq. 11.22 the above noted expression may be obtained.]

11.27 A pipe 0.6 m in diameter takes off from a reservoir 150 m above the datum. The pipe is 5000 m long and is laid completely at the datum level. For the last 1200 m water is drawn by service pipes at a uniform rate of 0.088 cumec per 300 m. Find the pressure at the end of the pipeline (friction factor $f = 0.04$). Take the other end as dead end, i.e., the velocity at the other end as zero.

[*Ans.* 1254.29 kN/m^2 (12.79 kg(f)/cm^2)]

11.28 For supplying water to a colony, it is intended to store water in a large tank whose water surface elevation is presently 85.37 m R.L. This tank is being supplied by a pipeline from a junction J, the friction loss in metres of water in this pipe being $0.6\,Q^{1.95}$, where Q is the discharge in m^3/s. Junction J receives water under pressure and without leakage from (i) a large reservoir I whose water surface elevation is constant at 100.00 m R.L., through a pipe characterised by $h_f = 0.35\,Q^2$; and (ii) a second large reservoir II with its constant water level at 96.00 m R.L., through a pipe characterised by $h_f = 0.46\,Q^{1.85}$. Neglecting losses other than due to pipe friction, determine the incoming flow into the tank in the colony and the piezometric head at the junction J.

[*Ans.* Flow from Tank I = 3.506 m^3/s, from Tank II = 0.797 m^3/s, total flow into the colony Tank = 4.303 m^3/s; Piezometric head atthejunction J = 95.698 m R.L.]

11.29 Derive the expression for the pressure rise due to water hammer caused by a sudden closure of the discharge through the pipe if

(1) the pipe elasticity is considered;

(2) the Poisson's ratio is not considered; and

(3) the pipe elasticity is fully neglected.

$$(1)\ p_i = \frac{V}{\sqrt{\frac{g}{w}\left[\frac{1}{K}+\frac{D}{TE}\left(1-\frac{1}{2m}\right)\right]}};$$

$$(2)\ p_i = \frac{V}{\sqrt{\frac{g}{w}\left(\frac{1}{K}+\frac{D}{TE}\right)}}$$

$$(3)\ p_i = V\sqrt{\frac{wK}{g}}$$

11.30 Two reservoirs with 15 m difference in their water levels are connected by a 300 mm diameter pipe line of 3000 m length. Calculate the discharge. If a parallel pipe line of 300 mm diameter is attached to the last 1500 m length of the existing pipe, determine the modified discharge. Take only wall friction into account. Assume f = 0.04 in Darcy–Weisbach formula.

[*Ans.* 0.0606 m^3/s; 0.0767 m^3/s]

Boundary Layer Theory

Chapter 12

12.1 INTRODUCTION

When a real fluid flows past a solid boundary, a layer of fluid which comes in contact with the boundary surface adheres to it on account of viscosity. Since this layer of fluid cannot slip away from the boundary surface it attains the same velocity as that of the boundary. In other words, at the boundary surface there is no relative motion between the fluid and the boundary. This condition is known as *no slip* condition. If the boundary is moving, the fluid adhering to it will have the same velocity as that of the boundary. However, if the boundary is stationary, the fluid velocity at the boundary surface will be zero. Thus at the boundary surface the layer of fluid undergoes retardation. This retarded layer of fluid further causes retardation for the adjacent layers of the fluid, thereby developing a small region in the immediate vicinity of the boundary surface in which the velocity of flowing fluid increases gradually from zero at the boundary surface to the velocity of the main stream. This region is known as *boundary layer*. In the boundary layer region since there is a larger variation of velocity in a relatively small distance, there exists a fairly large velocity gradient $(\partial v/\partial y)$ normal to the boundary surface. As such in this region of boundary layer even if the fluid has small viscosity, the corresponding shear stress $\tau = \mu\,(\partial v/\partial y)$, is of appreciable magnitude. Farther away from the boundary this retardation due to the presence of viscosity is negligible and the velocity there will be equal to that of the main stream. The flow may thus be considered to have two regions, one close to the boundary in the boundary layer zone in which due to larger velocity gradient appreciable viscous forces are produced and hence in this region the effect of viscosity is mostly confined; and second outside the boundary layer zone in which the viscous forces are negligible and hence the flow may be treated as non-viscous or inviscid. The concept of boundary layer was first introduced by L. Prandtl in 1904 and since then it has been applied to several fluid flow problems. In the following paragraphs the characteristics of flow in the boundary layer are discussed.

12.2 THICKNESS OF BOUNDARY LAYER

The velocity within the boundary layer increases from zero at the boundary surface to the velocity of the main stream asymptotically. Therefore the thickness of the boundary layer represented by δ (Greek 'delta') is arbitrarily defined as that distance from the boundary surface in which the velocity reaches 99% of the velocity of the main stream. In other words, the boundary layer thickness δ may be considered equal to the distance y from the boundary surface at which $v = 0.99\,V$. This definition however gives an proximate value of the boundary layer thickness and hence δ is generally termed an *nominal thickness* the boundary layer. For greater accuracy the boundary layer thickness is defined in terms of certain athematical expressions which are the measures of the effect of boundary layer on the flow. Three ch definitions of the boundary layer thickness which are commonly adopted are the *displacement ckness* δ^*, the *momentum thickness* θ and the *energy thickness* δ_E.

The *displacement thickness* δ^* is defined as the distance by which the boundary surface would have be displaced outwards (as shown in Fig. 12.1) so that the total actual discharge would be same as at of an ideal (or frictionless) fluid past the displaced boundary. Thus, it may be expressed by the uation

$$V\delta^* = \int_0^\infty (V-v)\,v\,dy$$

$$\delta^* = \int_0^\infty \left(1-\frac{v}{V}\right) dy\frac{1}{2} \qquad \text{... (12.1)}$$

Another definition for the displacement thickness may be given which is based on the fact that cause of the retardation of the flow within the boundary layer, in order to maintain the continuity of w the streamlines outside the boundary layer are farther from the boundary than they would be if

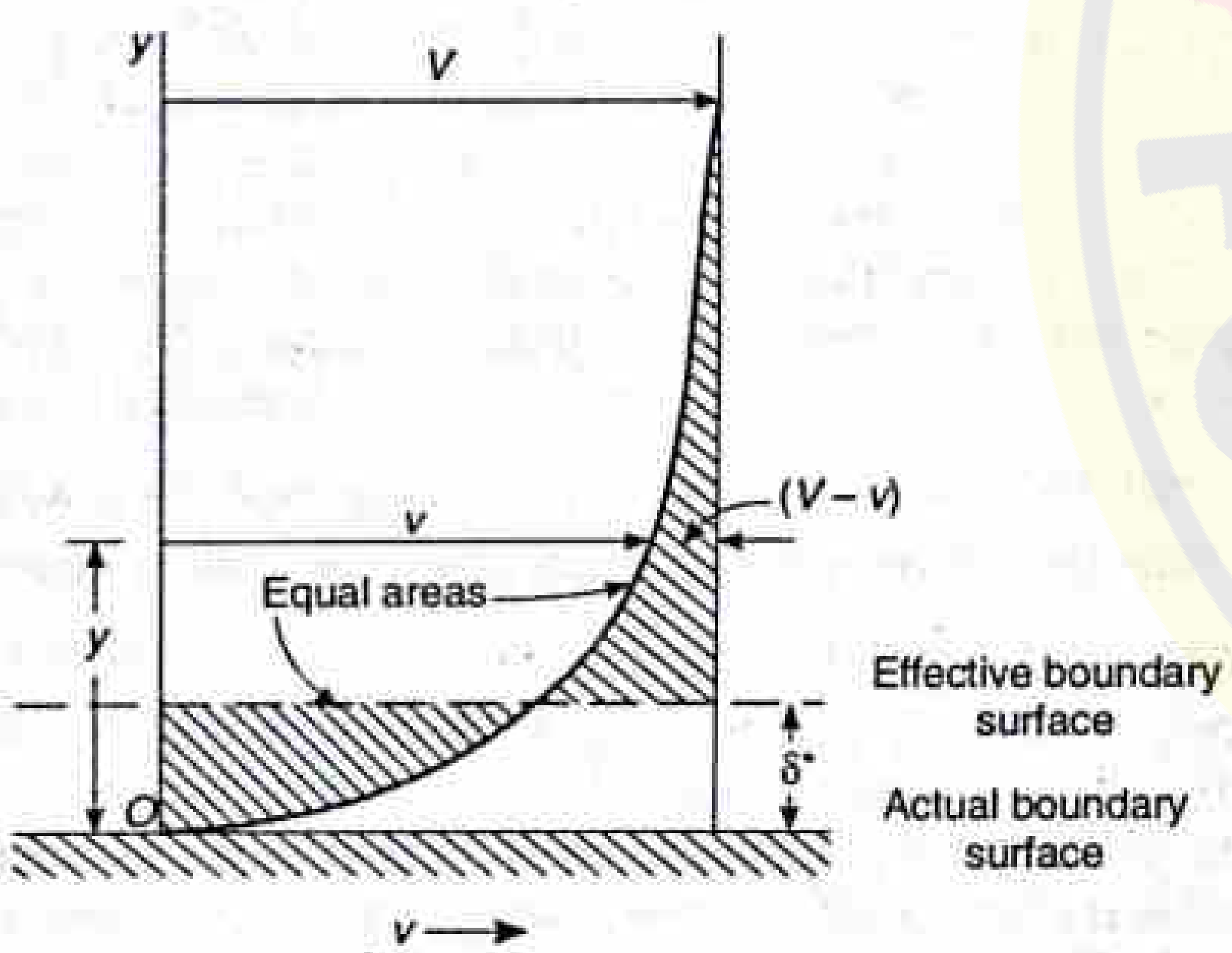


Figure 12.1 *Displacement thickness of boundary layer*

e fluid was ideal or inviscid upto the boundary. The displacement thickness δ^* may then be defined the distance by which the external streamlines are shifted or displaced outwards owing to the rmation of the boundary layer.

The retardation of flow due to viscosity in the boundary layer would cause the reduction in omentum flux. Accordingly the *momentum thickness* θ is defined as the distance from the actual undary surface such that the momentum flux corresponding to the main stream velocity V through s distance θ is equal to the deficiency or loss in momentum due to the boundary layer formation. us

$$\rho V^2\theta = \rho\int_0^\infty (V^2 - v^2)\,v\,dy$$

or

$$\theta = \int_0^\infty \frac{v}{V}\left(1-\frac{v}{V}\right) dy \qquad \text{...(12.2)}$$

Further the retardation of flow due to viscosity in the boundary layer would cause the reduction in the flux of energy. As such the *energy thickness* δ_E is defined as the distance from the actual boundary surface such that the energy flux corresponding to the main stream velocity V through this distance δ_E is equal to the deficiency or loss of energy due to the boundary layer formation. Thus

$$\frac{1}{2}\rho V^3\delta_E = \frac{1}{2}\rho\int_0^\infty (V^2 - v^2)\,v\,dy$$

or

$$\delta_E = \int_0^\infty \frac{v}{V}\left(1-\frac{v^2}{V^2}\right) dy \qquad \text{...(12.3)}$$

12.3 BOUNDARY LAYER ALONG A LONG THIN PLATE AND ITS CHARACTERISTICS

Consider a long thin plate held stationary in the direction parallel to the flow in a uniform stream of velocity V as shown in Fig. 12.2. The plate is said to be held at zero incidence to the velocity of flow and the velocity of flow is known as 'free stream velocity' or 'ambient velocity' or 'potential velocity'. At the leading edge of the plate the thickness of the boundary layer is zero, but on downstream, for the fluid in contact with the boundary the velocity of flow is reduced to zero and at some distance d from the boundary the velocity is nearly V. Hence a velocity gradient is set up which develops shear resistance to the flow and retards the motion of the fluid. Near the leading edge of the plate the fluid is retarded in a thin layer. In other words, the boundary layer near the leading edge is relatively thin. As this retarded layer of fluid moves downstream, due to continued action of shear resistance more and more fluid is retarded. Thus the thickness of the boundary layer δ, goes on increasing in the downstream direction as shown in Fig. 12.2. The various factors which influence the thickness of the boundary layer forming along a flat smooth plate are noted below.

(1) The boundary layer thickness increases as the distance from the leading edge increases.

(2) The boundary layer thickness decreases with the increase in the velocity of flow of the approaching stream of fluid.

(3) Greater is the kinematic viscosity of the fluid greater is the boundary layer thickness.

(4) The boundary layer thickness is considerably affected by the pressure gradient $(\partial p/\partial x)$ in the direction of flow. In the case of a flat plate placed in a stream of uniform velocity V, the pressure may also be assumed to be uniform i.e., $(\partial p/\partial x) = 0$. However, if the pressure gradient is negative as in the

case of a converging flow, the resulting pressure force acts in the direction of flow and it accelerates the retarded fluid in the boundary layer. As such the boundary layer growth is retarded in the presence of negative pressure gradient. On the other hand if the pressure gradient is positive as in the case of

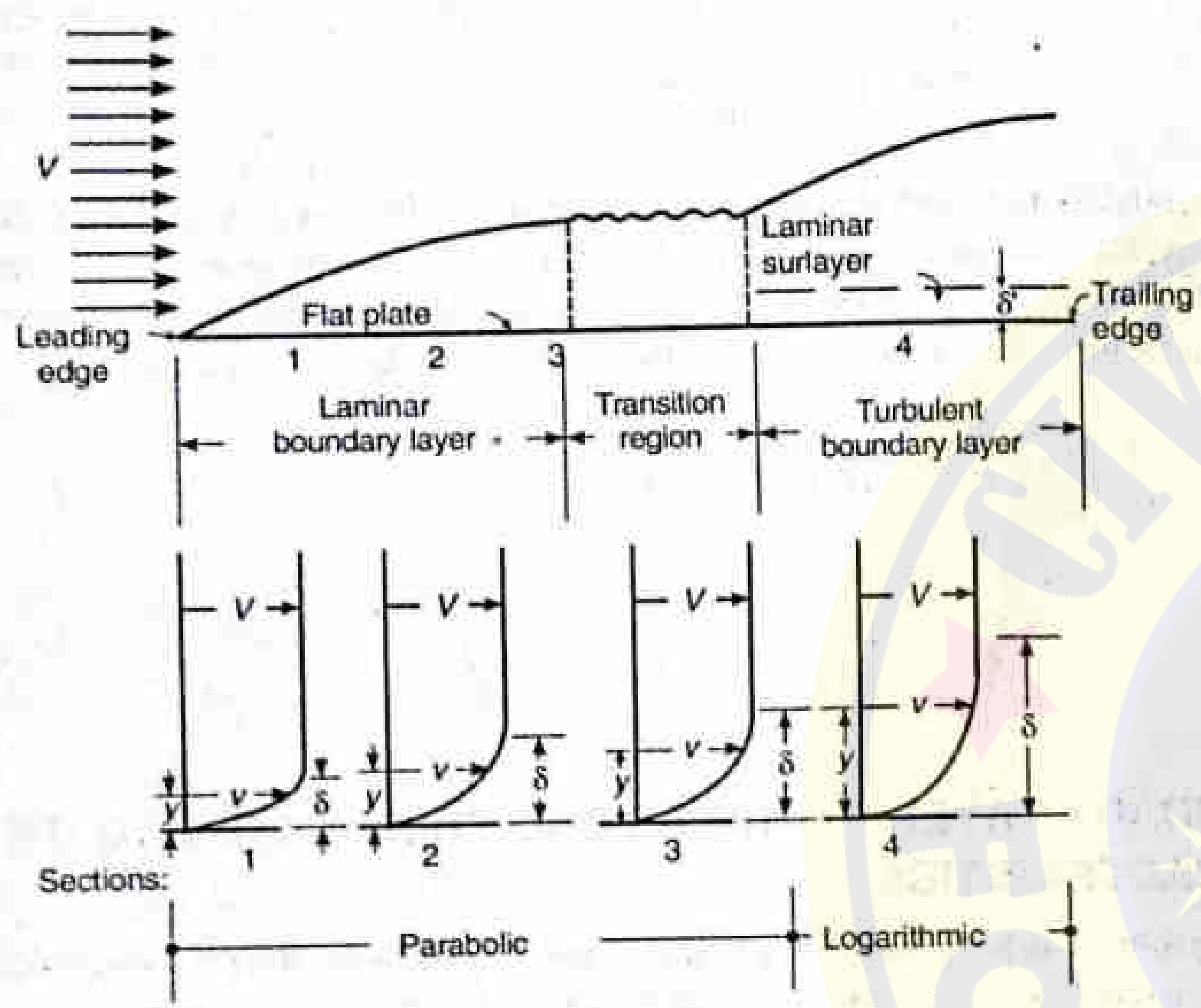


Figure 12.2 *Boundary layer and velocity distribution at successive points along a flat plate*

divergent flow the fluid in the boundary layer is further decelerated and hence assists in thickening of the boundary layer. In the later case back flow and separation may be caused as discussed in Section 12.10.

As the boundary layer develops, upto a certain portion of the plate from the leading edge, the flow in the boundary layer exhibits all the characteristics of laminar flow. This is so irrespective of whether the flow of the incoming stream is laminar or turbulent. This is known as *laminar boundary layer*. If the plate is sufficiently long, then beyond some distance from the leading edge the laminar boundary layer becomes unstable and the flow in the boundary layer exhibits the characteristics between those of laminar and turbulent flow. This region of the boundary layer is usually small and is known as transition region. After the transition region the flow in the boundary layer becomes turbulent. In this portion of the boundary layer there is a rapid increase in its thickness and it is known as *turbulent boundary layer*. If the plate is very smooth, even in the region of turbulent boundary layer, there is a very thin layer just adjacent to the boundary, in which the flow is laminar. This thin layer is commonly known as *laminar sublayer*, and its thickness is represented by δ'.

The velocity distribution in a laminar boundary layer is parabolic $(V-v) \sim (\delta-y)^2$; and for turbulent boundary layer the velocity distribution has been found to follow approximately either the one-

seventh power law: $v \sim y^{1/7}$ or it is logarithmic, $v \sim \log y$. For laminar sublayer the velocity distribution is parabolic, but since its thickness δ' is usually very small, a linear distribution can be assumed.

The change of boundary layer from laminar to turbulent mainly depends on the velocity of flow V, of the approaching stream of fluid, the length x measured along the plate from the leading edge, the mass density ρ of fluid and its dynamic viscosity μ. As such the Reynolds number $Re_x = (\rho V x/\mu)$ (the suffix x indicating that it is calculated with the distance x as the characteristic length) becomes a significant parameter in indicating the change of boundary layer from laminar to turbulent. The value of Re_x at which the boundary layer may change from laminar to turbulent varies from 3×10^5 to 6×10^5. In exceptional cases, where the approaching flow is free from any disturbance the values of critical Reynolds number can reach 10^6 or even higher. However, change of boundary layer from laminar to turbulent is also affected by several other factors such as roughness of the plate, plate curvature, pressure gradient and intensity and scale of turbulence.

12.4 BOUNDARY LAYER EQUATIONS

The equations of continuity and motion for the steady flow of an incompressible, inviscid fluid in two dimensions without body forces are

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

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} \qquad \text{... (ii)}$$

and

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} \qquad \text{... (iii)}$$

where x and y are rectangular cartesian coordinates and u and v are the corresponding velocity components.

Now if a viscous fluid is considered then the equation of continuity will be unchanged, but in the equations of motion additional terms will be introduced due to viscous stresses as indicated below.

Consider the boundary layer on a flat plate with x measured along the plate and y normal to it as shown if Fig. 12.3. The only viscous stress τ that need be considered is that acting in the direction parallel to the plate. Thus if a small element of fluid of area $\delta x\ \delta y$ in the xy plane and unit length normal to the xy plane is considered, then on its lower surface there is a force, due to shear stress, equal to $-\tau\,\delta x$ in the x direction. Likewise on the upper surface there is a force in the x direction due to shear stress as

$$(\tau + \partial\tau)\ \delta x = \left[\tau + \frac{\partial \tau}{\partial y}\delta y\right]\delta x$$

neglecting the terms of higher power of δy.

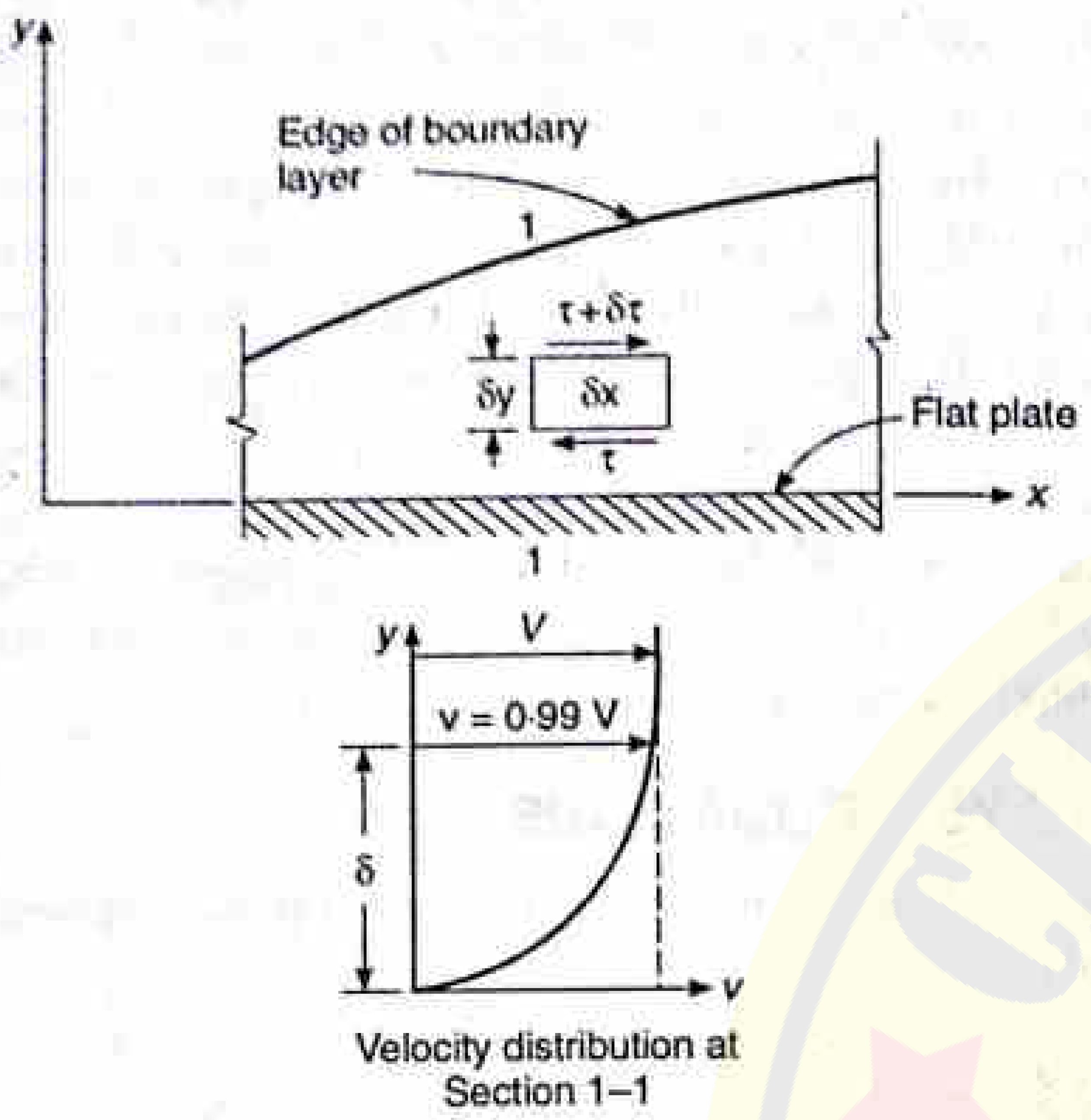


Figure 12.3 *Boundary layer along a flat plate*

The net force in the x direction acting on the element is therefore $(\partial\tau/\partial y)\,\delta x\,\delta y$, and hence as δx and δy tend to be zero, the force per unit mass in the x direction due to viscous stresses becomes

$$\frac{1}{\rho}\frac{\partial t}{\partial y}$$

Since the left-hand side of Eq. (ii) noted above represents the acceleration in x direction of a particle of fluid and the right-hand side represents the components of the force per unit mass in that direction due to static pressure, it follows that the above term due to viscous stresses should be added on the right-hand side of the equation to complete the equation of motion in the x direction in the boundary layer. This equation is then modified as

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{1}{\rho}\frac{\partial \tau}{\partial y}$$

Further for viscous flow in the laminar boundary layer from Newton's law of viscosity

$$\tau = \mu\frac{\partial u}{\partial y}$$

and hence the equation of motion becomes

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon\frac{\partial^2 u}{\partial y^2}$$

where υ is kinematic viscosity (μ/ρ).

The second equation of motion i.e., Eq. (iii) is unchanged by the argument that the shear stress is acting in the x direction only. However, by determining the order of magnitude of each of the terms of Eq. (iii) and neglecting the terms of smaller order of magnitude, Eq. (iii) reduces to

$$-\frac{1}{\rho}\frac{\partial p}{\partial y} = 0.$$

Thus the equations that govern the flow in the steady, two dimensional laminar boundary layer on a flat plate are

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y} = 0 \qquad ...(12.4)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon\frac{\partial^2 u}{\partial y^2} \qquad ...(12.5)$$

and

$$-\frac{1}{\rho}\frac{\partial p}{\partial y} = 0 \qquad ...(12.6)$$

Equations 12.4, 12.5 and 12.6 are generally known as *Prandtl's boundary layer equations* for two-dimensional steady flow of incompressible fluids. In the above analysis, Eq. 12.5 and 12.6 have been derived by adopting a relatively simpler approach. But the same can also be derived directly from the Navier–Stokes equations which are in fact the basic equations of motion for the flow of viscous fluids. The analysis involving the use of Navier–Stokes equations is more accurate and complete, since in this all the viscous stresses are included. However, for deriving Eqs 12.5 and 12.6 from the Navier-Stokes equations the order of magnitude of each of the terms of these equations is determined and the terms of smaller order of magnitude are neglected.

By integrating Eq. 12.6, we get

$$p = f(x) \text{ for steady flow}$$

and

$$p = f(x,t) \text{ for unsteady flow}$$

from which it may be concluded that the pressure p does not vary in the y direction or in other words the pressure remains constant across the boundary layer. Hence the partial derivative $(\partial p/\partial x)$ in Eq. 12.5 can be replaced by the total derivative (dp/dx) and Eq. 12.5 then becomes

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{dp}{dx}+\upsilon\frac{\partial^2 u}{\partial y^2} \qquad ...(12.7)$$

Since outside the boundary layer the fluid may be treated as inviscid (or non-viscous), the Euler's equations of motion may be applied according to which

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

In the region outside the boundary layer since $v = 0$ and $u = V$ the free stream or ambient velocity of the approaching stream, by substituting these values in the above equation, it becomes

$$V\frac{\partial V}{\partial x} = -\frac{1}{\rho}\frac{\partial p}{\partial x}$$

or
$$V\frac{\partial V}{\partial x} = -\frac{1}{\rho}\frac{dp}{dx} \quad ...(12.8)$$

as both V and p are the functions of x only. Integration of Eq. 12.8 leads to the Bernoulli's equation at any section viz.,

$$\frac{p}{\rho}+\frac{V^2}{2} = \text{constant}$$

Further introducing Eq. 12.8 in Eq. 12.7 it becomes

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y} = V\frac{dV}{dx}+\upsilon\frac{\partial^2 u}{\partial y^2} \quad ...(12.9)$$

12.5 MOMENTUM INTEGRAL EQUATION OF THE BOUNDARY LAYER

The boundary layer equations obtained in the preceding sections are nonlinear partial differential equations and hence difficult to solve. As such approximate methods for the solution of these equations have been developed for the case for which either the exact solutions cannot be obtained or the same may involve a lot of computations. One such approximate method developed by Theodore von Karman is based on the application of the momentum equation to the boundary layer and the corresponding equation derived by him is known as *momentum integral equation of the boundary layer or von Karman's integral equation*. The momentum integral equation of the boundary layer expresses the relation that must exist between the overall rate of flux of momentum across a section of the boundary layer, the shear stress at the boundary surface and the pressure gradient in the direction of flow. For steady, two-dimensional flow in the boundary layer along a flat plate this equation may be derived as indicated below.

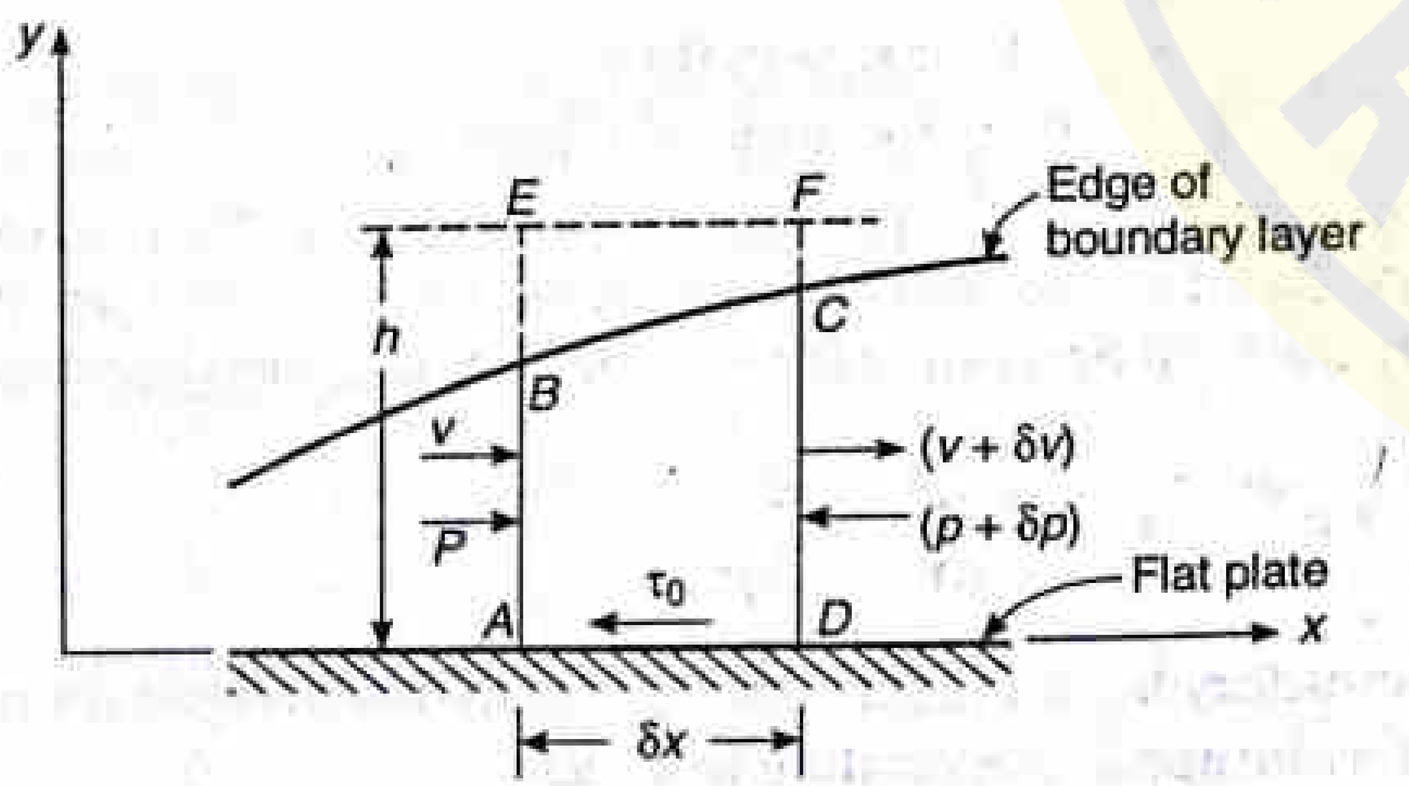


Figure 12.4 *Definition sketch for the derivation of momentum integral equation*

Consider an element of the boundary layer formed along a flat plate, between sections AB and CD, at a small distance δx apart as shown in Fig. 12.4. The boundary layer is of thickness δ and its outer edge is represented by BC. Let AB and CD be extended upto E and F where $AE = DF = h$, such that h is slightly greater than the local boundary layer thickness δ. $AEFD$ may be taken as a suitable control volume for this analysis.

For unit width in the direction perpendicular to the plane of the paper, the rate of mass flow across AE into $AEFD$

$$= \int_0^h \rho v\, dy$$

and the corresponding rate of mass flow across DF out of $AEFD$

$$= \int_0^h \rho v\, dy + \frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x$$

neglecting the terms of higher power of δx.

Hence the net rate of mass flow across DF and AE, out of $AEFD$

$$= \frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x \quad ...(i)$$

If v_h denotes the mean value of velocity in the y direction at height h above the surface over the length EF then the rate of mass flow out of $AEFD$ across EF

$$= \rho v_h \delta x$$

But continuity of mass requires that there is no net rate of change of mass inside $AEFD$ and hence

$$\rho v_h = -\frac{d}{dx}\left[\int_0^h \rho v dy\right] \quad ...(ii)$$

Now for the fluid in $AEFD$, the balance of rate of change of momentum in the x direction and components of applied force in that direction is considered.

The rate of transport of momentum in the x direction across DF minus the rate of transport of momentum in the x direction across AE is

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right]\delta x$$

The rate of transport of momentum in the x direction across EF out of $AEFD$ is

$$\rho v_h V\delta x = -V\frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x$$

from Eq. (ii), where V represents the velocity (in the x direction) of the main stream outside the boundary layer at section AB.

The force in the x direction on $AEFD$ due to the pressure

$$= -h\delta p = -h\frac{dp}{dx}\delta x$$

and that due to the shear stress at the boundary surface

$$= -\tau_0 \delta x$$

where τ_0 is the mean shear stress over the length AD.

Thus equating the net increase in the rate of transport of momentum to the sum of the forces acting in the x direction, we have

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right]\delta x - V\frac{d}{dx}\left[\int_0^h \rho v dy\right]\delta x$$

$$= -h\frac{dp}{dx}\delta x - \tau_0 \delta x$$

Dividing both the sides of the above equation by δx and taking the limit $\delta x \to 0$, we get

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right] - V\frac{d}{dx}\left[\int_0^h \rho v dy\right]$$

$$= -h\frac{dp}{dx} - \tau_0 \quad \text{... (12.10)}$$

By introducing Eq. 12.8 in Eq. 12.10 it becomes

$$\frac{d}{dx}\left[\int_0^h \rho v^2 dy\right] - V\frac{d}{dx}\left[\int_0^h \rho v dy\right]$$

$$= h\rho V\frac{dV}{dx} - \tau_0$$

Further the above equation can be written in the following form

$$\frac{d}{dx}\left[\int_0^h \rho v(v-V)\,dy\right] + \frac{dV}{dx}\left[\int_0^h \rho v dy\right]$$

$$= h\rho V\frac{dV}{dx} - \tau_0$$

or $$\frac{d}{dx}\left[\int_0^h \rho v(V-v)\,dy\right] + \frac{dV}{dx}\left[\int_0^h \rho(V-v)\,dy\right] = \tau_0$$

Since $(V-v)$ becomes zero at the edge of the boundary layer the upper limit to both integrals in the above equation may be changed to ∞. Then from the Eqs. 12.1 and 12.2 defining the displacement thickness δ^* and momentum thickness θ the above equation simplifies to

$$\frac{d}{dx}(\rho V^2\theta) + \frac{dV}{dx}(\rho V\delta^*) = \tau_0$$

or $$\frac{d}{dx}(V^2\theta) + \frac{dV}{dx}(V\delta^*) = \frac{\tau_0}{\rho} \quad \text{... (12.11)}$$

Equation 12.11 is the *momentum integral equation* of the boundary layer which forms the basis for approximate methods of solving boundary layer problems. Since in the derivation of this equation no assumption has been made regarding the nature of flow in the boundary layer, it is applied to both laminar as well as turbulent boundary layers. Further it may be noted that if for a flat plate at zero incidence $(\partial p/\partial x) = 0$ then from Eq. 12.8, $(\partial V/\partial x)$ or $(dV/dx) = 0$ and hence Eq. 12.11 reduces to

$$\frac{\tau_0}{\rho} = \frac{d}{dx}(V^2\theta)$$

or $$\frac{\tau_0}{\rho V^2} = \frac{d\theta}{dx} \quad \text{...(12.12)}$$

The momentum integral Eq. 12.11 may also be derived by integration of boundary layer Eq. 12.9.

12.6 LAMINAR BOUNDARY LAYER

Inside the boundary layer since the viscous forces are predominant, it is reasonable to assume that the inertial and viscous forces are of the same order of magnitude in a laminar boundary layer. The inertial forces per unit volume is given by $(\rho v\partial v/\partial x)$ which is proportional to $(\rho V^2/x)$ for the case of a flat plate. Similarly the viscous force per unit volume is $(\partial v/\partial y)$ which for laminar flow becomes $(\partial/\partial y)(\mu\partial v/\partial y)$ i.e., $(\mu\partial^2 v/\partial y^2)$. In this boundary layer since $(\partial v/\partial y) \sim (V/\delta)$, hence $(\partial\tau/\partial y) \sim (\mu V/\delta)$. Thus if these forces are proportional to each other, then

$$\rho\frac{V^2}{x} = k'\mu\frac{V}{\delta^2}$$

or $$\frac{\delta^2}{x^2} = \frac{k'\mu}{\rho Vx} = \frac{k'}{Re_x}$$

$\therefore$ $$\frac{\delta}{x} = \frac{k}{\sqrt{Re_x}};\ \delta = k\sqrt{\frac{xv}{V}} \quad \text{...(12.13)}$$

in which k' and k are constants. By exact analytical solution of the boundary layer equations Blasius has obtained the value of the constant k as 5. Thus

$$\frac{\delta}{x} = \frac{5}{\sqrt{Re_x}};\ \text{or } \delta = 5\sqrt{\frac{xv}{V}} \quad \text{...(12.14}$$

Further the expression for the shear stress τ_0 can also be obtained as indicated below.

Since in the boundary layer

$$\left(\frac{\partial v}{\partial y}\right)_{y=0} \sim \frac{V}{\delta};\ \tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=0} \sim \mu\frac{V}{\delta}$$

Introducing the value of δ from Eq. 12.13 in the above expression it becomes

$$\tau_0 = \text{constant}\sqrt{\frac{V^3\rho\mu}{x}} \quad \text{...(12.}$$

Equation 12.15 can also be expressed as

$$\frac{\tau_0}{\frac{\rho V^2}{2}} = c_f = \frac{\text{constant}}{\sqrt{\frac{\rho V x}{\mu}}} = \frac{\text{constant}}{\sqrt{\text{Re}_x}} \quad ...(12.16)$$

where the coefficient c_f is known as *local drag coefficient**. In Eq. 12.16 the value of the constant of proportionality has been obtained by Blasius by exact analytical solution of the boundary layer equations as 0.664. Thus Eq. 12.16 becomes

$$\frac{\tau_0}{\frac{\rho V^2}{2}} = \frac{0.664}{\sqrt{\text{Re}_x}} \quad ...(12.17)$$

Further the total horizontal force F_D (or skin friction drag) acting on one side of the plate on which laminar boundary layer is developed can be obtained as

$$F_D = \int_0^L \tau_0 B\,dx = \left(0.664\,\rho V^{3/2} \upsilon^{1/2} L^{1/2} B\right)$$

in which B is the width of the plate and L is the length of the plate.

The average drag coefficient C_f may be obtained as

$$C_f = \frac{(F_D/BL)}{(\rho V^2/2)} = \frac{1.328}{\sqrt{Re_L}} \quad ...(12.18)$$

in which $$Re_L = \frac{\rho V L}{\mu}$$

Further from the exact analytical solution of the boundary layer equations by Blasius the following expressions for the displacement thickness and the momentum thickness have been obtained.

$$\frac{\delta^*}{x} = \frac{1.729}{\sqrt{\text{Re}_x}} \quad ...(12.19)$$

and $$\frac{\theta}{x} = \frac{0.664}{\sqrt{\text{Re}_x}} \quad ...(12.20)$$

The above noted equations for the laminar boundary layer along a flat plate can also be derived by using Eq. 12.12. For this it is essential to assume a suitable velocity distribution for the boundary layer. Thus as an example if it is assumed that the velocity distribution across a section of the boundary layer is linear with y upto the edge of the boundary layer, then we have

$$\frac{v}{V} = \frac{y}{\delta} \text{ for } y \le \delta$$

* For definitions of drag and drag coefficient see Chapter 18.

and the boundary conditions are at $y = 0$, $v = 0$; and $y = \delta$, $v = V$, where V is the velocity of the approaching stream outside the boundary layer.

Thus $$\tau_0 = \mu\left(\frac{\delta v}{\delta y}\right)_{y=0} = \frac{\mu V}{\delta}$$

$$\delta^* = \int_0^\delta\left(1-\frac{v}{V}\right)\delta y = \int_0^\delta\left(1-\frac{y}{\delta}\right)\delta y = \frac{\delta}{2}$$

and $$\theta = \int_0^\delta \frac{v}{V}\left(1-\frac{v}{V}\right)\delta y = \int_0^\delta \frac{y}{\delta}\left(1-\frac{y}{\delta}\right)\delta y = \frac{\delta}{6}$$

Substituting these values in Eq. 12.12 yields

$$\frac{1}{6}\frac{d\delta}{dx} = \frac{\mu}{\rho V \delta} = \frac{\upsilon}{V\delta}$$

or $$\frac{d}{dx}(\delta^2) = \frac{12\upsilon}{V}$$

Integrating the above expression, we get

$$\delta^2 = \frac{12\upsilon x}{V} + C$$

where C is constant of integration.

Since at $x = 0$, $\delta = 0$, hence $C = 0$. Thus

$$\delta^2 = \frac{12\upsilon x}{V}$$

or $$\frac{\delta^2}{x^2} = \frac{12\upsilon}{Vx} = \frac{12}{\text{Re}_x}$$

$$\therefore \quad \frac{\delta}{x} = \sqrt{\frac{12}{\text{Re}_x}} = \frac{3.464}{\sqrt{\text{Re}_x}} \quad ...(12.21)$$

where $$\text{Re}_x = \frac{Vx}{\upsilon}$$

Further local drag coefficient c_f may be obtained as

$$c_f = \frac{\tau_0}{(\rho V^2/2)} = \frac{2\mu}{\rho V \delta} = \frac{0.5774}{\sqrt{\text{Re}_x}} \quad ...(12.22)$$

Skin friction drag F_D on one side of the plate having laminar boundary layer is obtained as

$$F_D = \int_0^L \tau_0 B\,dx = \left(0.5774\,\rho V^{3/2} \upsilon^{1/2} L^{1/2} B\right)$$

where B and L are the width and the length of the plate respectively.

The average drag coefficient C_f may be obtained as

$$C_f = \frac{(F_D/BL)}{(\rho V^2/2)} = \frac{1.1548}{\sqrt{Re_L}} \quad \text{...(12.23)}$$

in which

$$Re_L = \frac{VL}{\upsilon}$$

The momentum thickness is given by

$$\frac{\theta}{x} = \frac{\delta}{6x} = \frac{0.5774}{\sqrt{Re_x}} \quad \text{...(12.24)}$$

and the displacement thickness is given by

$$\frac{\delta^*}{x} = \frac{\delta}{2x} = \frac{1.732}{\sqrt{Re_x}} \quad \text{...(12.25)}$$

It may thus be noted that the results obtained by assuming a linear velocity distribution in the boundary layer, are fairly close to the results obtained by Blasius by exact solution of the boundary layer equations. However, by assuming certain other velocity distributions in the boundary layer better agreement with the exact solution can be obtained. Moreover, in laminar boundary layer the flow exhibits the characteristics of laminar flow, and hence the velocity distribution at any section in the laminar boundary layer will follow parabolic law.

12.7 TURBULENT BOUNDARY LAYER

Turbulent boundary layers are usually thicker than laminar ones. Further as a result of intermingling of fluid particles between different layers of the fluid in a turbulent boundary layer the velocity distribution in a turbulent boundary layer is much more uniform than that in a laminar boundary layer. However, in a turbulent boundary layer, near the boundary large velocity change occurs in a relatively small vertical distance, and hence at the boundary the velocity gradient $(\partial v/\partial y)$ is steeper in a turbulent boundary layer than in a laminar boundary layer. The velocity distribution in a turbulent boundary layer follows a logarithmic law i.e., $v \sim \log y$, which can also be represented by a power law of the type

$$\frac{v}{V} = \left(\frac{y}{\delta}\right)^n \quad \text{...(12.26)}$$

The value of the exponent n is approximately (1/7) for moderate Reynolds number ($Vx/\upsilon < 10^7$ for a flat plate) and decreases somewhat with increasing Reynolds number. Although Eq. 12.26 satisfactorily describes the velocity distribution for most of the region of turbulent boundary layer, it cannot apply at the boundary itself because $(\partial v/\partial y) = (1/7)V\delta^{-1/7}\, y^{-6/7} = \infty$ when $y = 0$. However, immediately adjacent to the boundary there is laminar sublayer, which is so thin that its velocity distribution profile may be taken as linear (instead of parabolic) and tangential to the 'seventh root' profile at the point where the laminar sublayer merges with the turbulent part of the boundary layer.

The expressions for δ, c_f and C_f for the turbulent boundary layer are as noted below:

$$\frac{\delta}{x} = \frac{0.376}{(Re_x)^{1/5}} \quad \text{...(12.27)}$$

$$c_f = \frac{0.059}{(Re_x)^{1/5}} \quad \text{...(12.28)}$$

$$C_f = \frac{0.074}{(Re_L)^{1/5}} \quad \text{...(12.29)}$$

Equation 12.29 is applicable only for values of Reynolds number Re_L ranging from 5×10^5 to 10^7. For values of Re_L ranging between 10^7 to 10^9, H. Schlichting assumed a logarithmic velocity distribution for the flow in the boundary layer and obtained the semi-empirical relation as noted below.

$$C_f = \frac{0.455}{(\log_{10} Re_L)^{2.58}} \quad \text{...(12.30)}$$

A comparison of Eqs. 12.13 and 12.27 indicates that in the case of laminar boundary layer the boundary layer thickness δ increases as $x^{1/2}$, but in the case of turbulent boundary layer δ increases as $x^{4/5}$. Hence the thickness of the turbulent boundary layer increases much faster than that of laminar boundary layer.

Equation 12.18 for the average drag coefficient is applicable only if the boundary layer is laminar for the entire length of the plate, that is when the value of Re_L is less than 5×10^5. Similarly Eq. 12.29 (or 12.30) is applicable if, the boundary layer is turbulent for the entire length of the plate, and the value of Re_L is within the range of 5×10^5 to 10^7 (or 10^7 to 10^9). However, when the plate is of such a length that for some distance from the leading edge of the plate the boundary layer is laminar and it becomes turbulent for the remaining portion of the plate, then combined relations are required. Hence for such cases where the plate is covered with both laminar and turbulent boundary layers, Eq. 12.31 as proposed by Prandtl may be used to compute the average drag coefficient C_f

$$C_f = \frac{0.074}{Re_L^{1/5}} - \frac{A}{Re_L} \quad \text{...(12.31)}$$

The constant A in Eq. 12.31 depends on the value of the Reynolds number Re_x at which the laminar boundary layer becomes turbulent. The values of A for various values of critical Reynolds number are as given below:

Critical $(Re)_x$	3×10^5	5×10^5	10^6	3×10^6
Constant A	1050	1700	3300	8700

It may, however, be mentioned that since for most of the cases the value of critical Re_x may be taken as 5×10^5, Eq. 12.31 then becomes

$$C_f = \frac{0.074}{Re_L^{1/5}} - \frac{1700}{Re_L} \quad \text{...(12.31 a)}$$

Further Eq. 12.31 is applicable for values of Re_L upto 10^7. But for higher values of Re_L ranging from 10^7 to 10^9 the following equation can be used.

$$C_f = \frac{0.455}{(\log_{10} Re_L)^{2.58}} - \frac{A}{Re_L} \quad \text{...(12.32)}$$

in which value of the constant A again depends on the critical Re_x and is the same as given in the above table. Equation 12.32 is known as *Prandtl–Schlichting Equation*. It may, however, be mentioned that Eq. 12.32 is valid in the entire range of Re_L from 5×10^5 upto 10^9, and it agrees with the Eq. 12.31 upto $Re_L = 10^7$.

12.8 LAMINAR SUBLAYER

If the plate is very smooth, even in the zone of turbulent boundary layer, there exists a very thin layer immediately adjacent to the boundary, in which the flow is laminar. This thin layer is commonly known as *laminar sublayer*, and its thickness is represented by δ'. As stated earlier the exponential as well as the logarithmic velocity distribution cannot apply at the boundary itself, although these satisfactorily describe the velocity distribution for most of the region of turbulent boundary layer. This discrepancy may, however, be explained by the existence of the laminar sublayer. Since the flow in the laminar sublayer is laminar, it will follow a parabolic velocity distribution law, which can be approximately taken to be linear as δ' is very small, and hence exponential or logarithmic velocity distribution law cannot be applied within the laminar sublayer. Thus in this case there will be a gradual change from parabolic velocity distribution very close to the boundary to exponential or logarithmic velocity distribution and the intersection of the parabolic and exponential or logarithmic velocity distribution curves may be arbitrarily taken as the limit of the laminar sublayer. Nikuradse's experimental studies have shown that

$$\delta' = \frac{11.6\upsilon}{\sqrt{(\tau_0/\rho)}} = \frac{11.6\upsilon}{V_*} \qquad ...(12.33)$$

in which $V^* = \sqrt{(\tau_0/\rho)}$ is known as shear or friction velocity. As indicated in Chapter 14 the relative magnitudes of the thickness of laminar sublayer δ' and the average height of roughness projections k facilitates the classification of the boundaries as hydrodynamically smooth and rough.

12.9 BOUNDARY LAYER ON ROUGH SURFACES

In the preceding sections the development of boundary layer along smooth plates has been considered. However, in most practical applications connected with the boundary layer development on flat plates such as ships, airplane wings, turbine blades etc., the surfaces cannot be considered smooth. As such the flow past a rough plate is of much practical interest. For a rough plate if k is the average height of roughness projections on the surface of the plate and δ is the thickness of the boundary layer, then the relative roughness (k/δ) is a significant parameter indicating the behaviour of the boundary surface. For k remaining constant, (k/δ) decreases along the plate because δ increases in the downstream direction. As a result the front portion of the plate will behave differently from its rear portion as far as the influence of roughness on drag is concerned. If it is assumed that the boundary layer is turbulent from the leading edge of the plate, the front portion of the plate will act as hydrodynamically rough, followed by a transition region and the downstream portion of the plate will be hydrodynamically smooth if the plate is sufficiently long. The limits between these three regimes are determined by the value of a dimensionless roughness parameter $(V_* k_s/\upsilon)$ as indicated below:

Hydrodynamically smooth : $\dfrac{V_* k_s}{\upsilon} < 5$

Transition : $5 < \dfrac{V_* k_s}{\upsilon} < 70$

Hydrodynamically rough : $\dfrac{V_* k_s}{\upsilon} > 70$

(or completely rough)

in which k_s is equivalent sand grain roughness defined as that value of the roughness which would offer the same resistance to the flow past the plate as that due to the actual roughness on the surface of the plate. Although the above noted limits have been determined for flow through pipes by Nikuradse on the basis of the results of his experiments on the pipes made artificially rough by gluing sand grains of uniform size on the inner surface of the pipe, but the same may also be applied to the flow past a plate. The introduction of the equivalent sand grain roughness k_s instead of the average height of the roughness projections k is due to the fact that as compared to k, k_s is a more representative parameter of the actual roughness. This is so because usually the actual roughness projections on the surface of the plate are of various heights and they do not have any uniform pattern and hence such roughnesses cannot be truely represented in terms of the average height of the roughness projections. Moreover, the consideration of k_s also facilitates the use of Nikuradse's results, which are in terms of the sand grain roughness, in the analysis of flow past rough plates.

In the completely rough regime the local drag coefficient c_f and the average drag coefficient C_f are given by the following expressions

$$c_f = \left[2.87 + 1.58\log_{10}\left(\frac{x}{k_s}\right)\right]^{-2.5} \qquad ...(12.34)$$

$$C_f = \left[1.89 + 1.62\log_{10}\left(\frac{L}{k_s}\right)\right]^{-2.5} \qquad ...(12.35)$$

which are valid for $10^2 < \left(\dfrac{L}{k_s}\right) < 10^6$.

12.10 SEPARATION OF BOUNDARY LAYER

As mentioned in Section 12.3 the boundary layer thickness is considerably affected by the pressure gradient in the direction of flow. If the pressure gradient $(\partial p/\partial x)$ is zero, then the boundary layer continues to grow in thickness along a flat plate. With the decreasing pressure in the direction of flow i.e., with negative pressure gradient, the boundary layer tends to be reduced in thickness. However, with the pressure increasing in the direction of flow i.e., with positive (or adverse) pressure gradient, the boundary layer thickens rapidly. The adverse pressure gradient plus the boundary shear decreases the momentum in the boundary layer and if they both act over a sufficient distance they cause the fluid in the boundary layer to come to rest i.e., the retarded fluid particles, cannot, in general penetrate too far into the region of increased pressure owing to their small kinetic energy. Thus, the boundary layer is deflected sideways from the boundary, separates from it and moves into the main stream. This phenomenon is called *separation*.

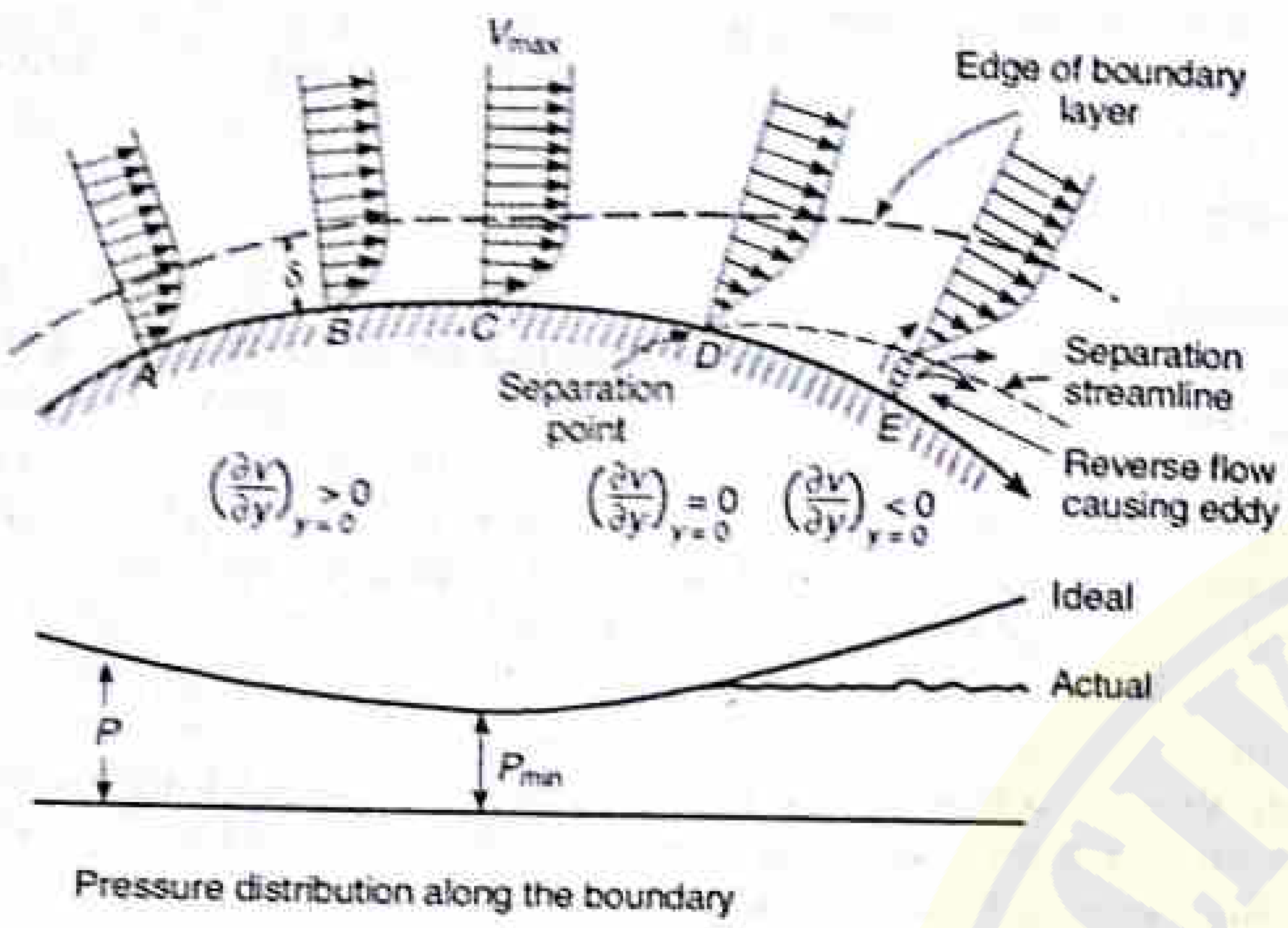


Figure 12.5 *Separation of boundary layer*

Consider flow of fluid over a curved surface as shown in Fig. 12.5. As the fluid flows round the surface it is accelerated over the left hand section until at point C the velocity just outside the boundary layer is a maximum. At this section the pressure is minimum as shown by the graph below the surface. Thus, from A to C the pressure gradient $(\partial p/\partial x)$ is negative and the net pressure force on an element of fluid in the boundary layer is in the direction of flow, which counteracts to some extent the slowing down effect of the boundary on the flowing fluid and thus the rate at which the boundary layer thickens is less than that for a flat plate with zero pressure gradient (at a corresponding value of Re_x).

However, beyond C the pressure increases, and hence the net pressure force on an element of fluid in the boundary layer opposes the forward flow. Thus at a certain distance on the downstream of point C the fluid near the boundary surface is soon brought to a standstill. The value of the velocity gradient $(\partial v/\partial y)$ at the boundary surface is then zero as at point D. The fluid is no longer able to follow the contour of the curved surface and it separates from it. The separation of the flowing fluid from the boundary first occurs at a point where $(\partial v/\partial y)$ at the boundary becomes zero and this point is known as *separation point*. On downstream of the separation point D, a further retardation of the fluid close to the boundary can even have a reverse or back flow as at point E shown in Fig. 12.5. If all the points below which a reverse flow occurs are joined by a smooth curve, a line dividing the forward and reverse flows is obtained which is known as *separation streamline*. In a region in between the boundary surface and the separation streamline, as a result of the reverse flow, large irregular eddies are formed in which much energy is dissipated as heat. This region of disturbed fluid usually extends for some distance on the downstream. Since the energy of the eddies is dissipated as heat, the pressure downstream remains approximately the same as at the separation point.

Separation occurs with both laminar and turbulent boundary layers, but laminar boundary layer is more susceptible to earlier separation than turbulent boundary layer. This is so because in a laminar boundary layer the increase of velocity with distance from the boundary surface is less rapid, and the adverse pressure gradient can more rapidly halt the slow moving fluid close to the boundary surface. On the other hand in a turbulent boundary layer the velocity distribution is much more uniform than in a laminar boundary layer because of intense lateral mixing. As a result relatively higher velocity prevails within a turbulent boundary layer, which reduces tendency of separation.

Separation of the boundary layer greatly affects the flow as a whole. In particular the formation of a wake zone of disturbed fluid on the downstream, in which the pressure is approximately constant and much less than that on the upstream, gives rise to boundary forces. As indicated in Chapter 18, the net force in the direction of flow caused by such pressure difference is known as pressure drag or form drag, the magnitude of which depends on the form or shape of the boundary and the resulting flow pattern obtained due to separation. Further the flow in a divergent passage or diffuser is another example in which the separation of the flow may be caused due to adverse pressure gradient prevailing there unless the angle of divergence is very small.

Since the separation of the boundary layer gives rise to additional resistance to flow, attempts should be made to avoid separation by some means. One of these methods may be by developing such boundary shapes for which the separation is as far as possible eliminated. The flow past a streamlined object such as airfoil is one such example. Apart from this the separation may also be avoided by adopting suitable method of controlling the boundary layer which are described in the next section.

12.11 METHODS OF CONTROLLING THE BOUNDARY LAYER

Several methods of controlling the boundary layer have been developed which are briefly described below.

12.11.1 Motion of Solid Boundary

The formation of the boundary layer is due to the difference between the velocity of the flowing fluid and that of the solid boundary. As such it is possible to eliminate the formation of a boundary layer by causing the solid boundary to move with the flowing fluid. Such a motion of the boundary may be achieved in the simplest ways by rotating a circular cylinder lying in a stream of fluid [see Fig. 12.6 (a)], so that on the upper side of the cylinder, where the fluid as well as the cylinder move in the same direction, the boundary layer does not form and hence the separation is completely eliminated. However, on the lower side of the cylinder, where the fluid motion is opposite to that of the cylinder, separation would occur.

12.11.2 Acceleration of the Fluid in the Boundary Layer

This method consists of supplying additional energy to the particles of fluid which are being retarded in the boundary layer. This may be achieved either by injecting fluid into the region of boundary layer from the interior of the body with the help of some suitable device as shown in Fig. 12.6 (b); or by diverting a portion of the fluid of the main stream from the region of high pressure to the retarded region of boundary layer through a slot provided in the body as in the case of the slotted wing shown in Fig. 12.6 (c). However, a disadvantage of this method is that if the fluid is injected into a laminar boundary layer, it undergoes a transition to turbulent boundary layer which results in an increased skin friction drag.

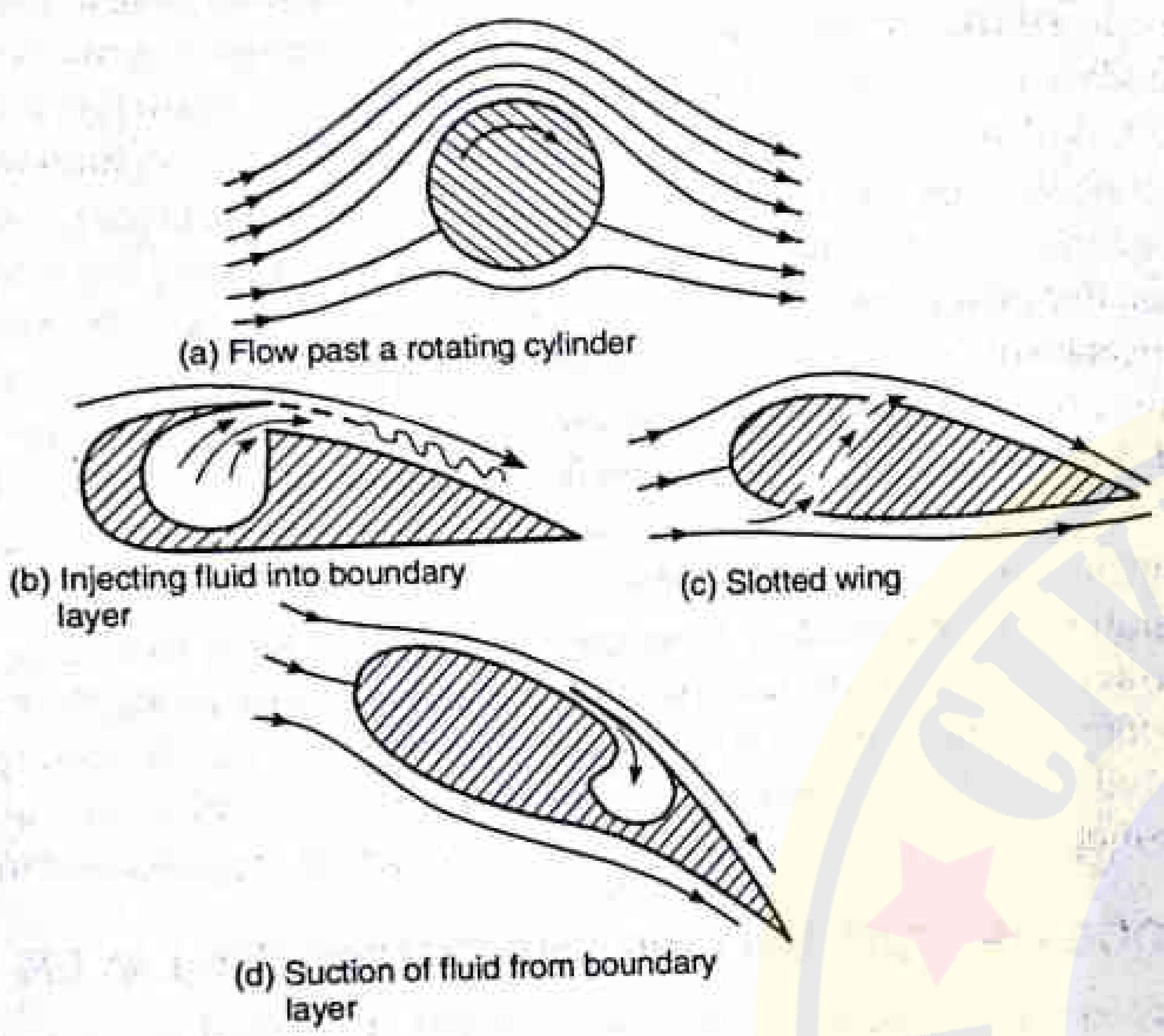


Figure 12.6 *Methods of controlling the boundary layer*

12.11.3 Suction of the Fluid from the Boundary Layer

In this method the slow moving fluid in the boundary layer is removed by suction through slots or through a porous surface as shown in Fig. 12.6 (d), so that on the downstream of the point of suction a new boundary layer starts developing which is able to withstand an adverse pressure gradient and hence separation is prevented. Moreover, the suction of the fluid from the boundary layer also greatly delays its transition from laminar to turbulent due to which skin friction drag is reduced.

12.11.4 Streamlining of Body Shapes

By the use of suitably shaped bodies the point of transition of the boundary layer from laminar to turbulent can be moved downstream which results in the reduction of the skin friction drag. Furthermore by streamlining of body shapes the separation may be eliminated.

ILLUSTRATIVE EXAMPLES

Example 12.1. *The velocity distribution in the boundary layer is given as*

$$\frac{v}{V} = \frac{3}{2}\eta - \frac{1}{2}\eta^2$$

in which $\eta = (y/\delta)$. *Compute* (δ^*/δ) and (θ/δ).

Solution

$$\delta^* = \int_0^\infty \left(1-\frac{v}{V}\right)dy$$

or

$$\delta^* = \int_0^\delta \left(1-\frac{v}{V}\right)dy + \int_\delta^\infty \left(1-\frac{v}{V}\right)dy$$

But outside the boundary layer $(v/V) = 1$, and hence

$$\delta^* = \int_0^\delta \left(1-\frac{v}{V}\right)dy$$

$$\frac{v}{V} = \frac{3}{2}\eta - \frac{1}{2}\eta^2 \text{ and } dy = \delta d\eta$$

Thus

$$\delta^* = \delta\int_0^1 \left(1-\frac{3}{2}\eta+\frac{1}{2}\eta^2\right)d\eta = \frac{5}{12}\delta$$

$$\therefore \quad \frac{\delta^*}{\delta} = \frac{5}{12}$$

Similarly

$$\theta = \int_0^\delta \frac{v}{V}\left(1-\frac{v}{V}\right)dy$$

or

$$\theta = \delta\int_0^1 \left(\frac{3}{2}\eta-\frac{1}{2}\eta^2\right)\left(1-\frac{3}{2}\eta+\frac{1}{2}\eta^2\right)d\eta$$

$$= \frac{19}{120}\delta$$

$$\therefore \quad \frac{\theta}{\delta} = \frac{19}{120}$$

Example 12.2. *Given that a laminar boundary layer at zero pressure gradient over a flat plate is described by the velocity profile*

$$\frac{v}{V_0} = \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)$$

determine the momentum correction coefficient (or factor) and the energy correction coefficient (or factor).

Also show that boundary layer thickness δ, *wall shear stress* τ_0 *and coefficient of drag* C_D *are given by*

$$\delta = \frac{4.795x}{\sqrt{Re_x}};\ \tau_0 = \frac{0.328\rho V_0^2}{\sqrt{Re_x}};\ C_D = \frac{1.312}{\sqrt{Re_L}}$$

where symbols have their usual meaning.

Solution

Momentum correction coefficient (or factor) is given by

$$\beta = \frac{1}{AV^2}\int_A v^2 dA$$

Mean velocity

$$V = \frac{Q}{A} = \frac{\int_A v\,dA}{A}$$

Considering unit width of the plate $dA = 1 \times dy$, and $A = (1 \times \delta)$. Thus

$$V = \frac{\int_0^\delta V_0 \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)(1 \times dy)}{(1 \times \delta)}$$

or

$$V = \frac{V_0}{\delta}\int_0^\delta \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right) dy$$

or

$$V = \frac{2V_0}{\pi}$$

Thus

$$\beta = \frac{\pi^2}{4V_0^2} \times \frac{1}{(1 \times \delta)}\int_0^\delta \left[V_0 \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]^2 (1 \times dy)$$

or

$$\beta = \frac{\pi^2}{4\delta}\int_0^\delta \sin^2\left(\frac{\pi}{2}\frac{y}{\delta}\right) dy$$

or

$$\beta = \frac{\pi^2}{8} = 1.234$$

Energy correction coefficient (or factor) is given as

$$\alpha = \frac{1}{AV^3}\int_A v^3 dA$$

or

$$\alpha = \frac{\pi^3}{8V_0^3} \times \frac{1}{(1 \times \delta)}\int_0^\delta \left[V_0 \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]^3 (1 \times dy)$$

or

$$\alpha = \frac{\pi^3}{8\delta}\int_0^\delta \sin^3\left(\frac{\pi}{2}\frac{y}{\delta}\right) dy$$

or

$$\alpha = \frac{\pi^2}{6} = 1.645$$

From Eq. 12.12 we have

$$\frac{\tau_0}{\rho V_0^2} = \frac{d\theta}{dx} \quad \text{...(i)}$$

At the boundary

$$\tau_0 = \mu\left(\frac{\delta v}{\delta y}\right)_{y=0}$$

$$\left(\frac{\delta v}{\delta y}\right)_{y=0} = \frac{\pi}{2}\frac{V_0}{\delta}\left[\cos\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]_{y=0} = \frac{\pi}{2}\frac{V_0}{\delta}$$

$$\tau_0 = \mu\frac{\pi}{2}\frac{V_0}{\delta} \quad \text{...(ii)}$$

From Eq. 12.2

$$\theta = \int_0^\infty \frac{v}{V_0}\left(1 - \frac{v}{V}\right) dy$$

Outside the boundary layer since $v = V_0$,

$$\theta = \int_0^\delta \frac{v}{V_0}\left(1 - \frac{v}{V_0}\right) dy$$

or

$$\theta = \int_0^\delta \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\left[1 - \sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right] dy$$

or

$$\theta = \frac{2\delta}{\pi}\left(1 - \frac{\pi}{4}\right)$$

By substituing the values of τ_0 and θ in Eq. (i) we get

$$\frac{1}{\rho V_0^2}\left(\mu\frac{\pi}{2}\frac{V_0}{\delta}\right) = \frac{2}{\pi}\left(1 - \frac{\pi}{4}\right)\frac{d\delta}{dx}$$

or

$$\delta\, d\delta = \frac{11.4976\, \mu dx}{\rho V_0} \quad \text{...(iii)}$$

Since δ is a function of x only, integration of Eq. (iii) gives

$$\frac{\delta^2}{2} = \frac{11.4976\, \mu dx}{\rho V_0} + \text{const}$$

or
$$\delta = 4.795\sqrt{\frac{\mu x}{\rho V_0}}$$

or
$$\delta = \frac{4.795x}{\sqrt{Re_x}}$$

where
$$Re_x = \frac{\rho V_0 x}{\mu}$$

Substituting the value of δ in Eq. (ii) yields

$$\tau_0 = \mu \frac{\pi}{2}\frac{V_0}{4.795}\sqrt{\frac{\rho V_0}{\mu x}}$$

or
$$\tau_0 = 0.328\sqrt{\frac{\rho\mu V_0^3}{x}} \quad \text{... (iv)}$$

or
$$\tau_0 = \frac{0.328\rho V_0^2}{\sqrt{Re_x}}$$

where
$$Re_x = \left(\frac{\rho V_0 x}{\mu}\right)$$

The drag F_D on one side of the plate of unit width is

$$F_D = \int_0^L \tau_0 dx$$

Introducing the value of τ_τ from Eq. (iv) we have

$$F_D = \int_0^L 0.328\sqrt{\frac{\rho\mu V_0^3}{x}}\, dx$$

or
$$F_D = 0.656\sqrt{\rho\mu V_0^3 L} \quad \text{... (v)}$$

The drag can be expressed in terms of a drag coefficient C_D times the stagnation pressure $(\rho V_0^2/2)$ and the area of plate L (per unit width). Thus

$$F_D = C_D\frac{\rho V_0^2}{2}L$$

Introducing the value of F_D from Eq. (v) we have

$$0.656\sqrt{\rho\mu V_0^3 L} = C_D\frac{\rho V_0^2}{2}L$$

or
$$C_D = \frac{1.312}{\sqrt{Re_L}}$$

where
$$Re_L = \frac{\rho V_0 L}{\mu}$$

Example 12.3. *Air flows over a flat plate 1 m long at a velocity of 6 m/s. Determine (a) the boundary layer thickness at the end of the plate, (b) shear stress at the middle of the plate, (c) total drag per unit length on the sides of the plate. Take ρ = 1.226 kg/m³ (0.125 msl/m³) and $\upsilon = 0.15 \times 10^{-4}$ m²/s (0.15 stokes) for air*

Solution

$$Re_L = \frac{VL}{\upsilon} = \frac{6\times1}{0.15\times10^{-4}} = 4.0\times10^5$$

Hence the boundary layer is laminar over the entire length of the plate

$$\therefore \quad \delta = 5\sqrt{\frac{x\upsilon}{V}}$$

$$= 5\sqrt{\frac{1\times0.15\times10^{-4}}{6}} = 7.91\times10^{-3}\text{ m} = 7.91\text{ mm}$$

SI units

$$\tau_0 = c_f\frac{\rho V^2}{2}; \text{ and } c_f = \frac{0.664}{\sqrt{Re_x}}$$

For the middle of the plate x = 0.5 m

Thus
$$Re_x = \frac{6\times0.5}{0.15\times10^{-4}} = 2.0\times10^5$$

and
$$c_f = \frac{0.664}{\sqrt{2.0\times10^5}} = 1.485\times10^{-3}$$

$$\therefore \quad \tau_0 = \frac{1.485\times10^{-3}\times1.226\times(6)^2}{2} = 32.77\times10^{-3}\text{ N/m}^2$$

Further
$$C_f = \frac{1.328}{\sqrt{Re_L}} = \frac{1.328}{\sqrt{4.0\times10^5}} = 2.1\times10^{-3}$$

and
$$F_D = 2\times BL\times C_f\frac{\rho V^2}{2}$$

$$\therefore \quad F_D = 2\times(1\times1)\times2.1\times10^{-3}\times\frac{1.226\times(6)^2}{2}$$

$$= 92.69\times10^{-3}\text{ N}$$

Metric units

$$\tau_0 = c_f \frac{\rho V^2}{2} \text{ ; and } c_f = \frac{0.664}{\sqrt{Re_x}}$$

For the middle of the plate $x = 0.5$ m

Thus

$$Re_x = \frac{6 \times 0.5}{0.15 \times 10^{-4}} = 2.0 \times 10^5$$

and

$$c_f = \frac{0.664}{\sqrt{2.0 \times 10^5}} = 1.485 \times 10^{-3}$$

$\therefore$

$$\tau_0 = \frac{1.485 \times 10^{-3} \times 0.125 \times (6)^2}{2}$$

$$= 3.34 \times 10^{-3} \text{ kg(f)/m}^2$$

Further

$$C_f = \frac{1.328}{\sqrt{Re_L}} = \frac{1.328}{\sqrt{4.0 \times 10^5}} = 2.1 \times 10^{-3}$$

and

$$F_D = 2 \times BL \times C_f \frac{\rho V^2}{2}$$

$\therefore$

$$F_D = 2 \times (1 \times 1) \times 2.1 \times 10^{-3} \times \frac{0.125 \times (6)^2}{2}$$

$$= 9.45 \times 10^{-3} \text{ kg (f)}$$

Example 12.4. *A smooth two-dimensional flat plate is exposed to a wind velocity of 100 km per hour. If laminar boundary layer exists upto a value of Re_x equal to 3×10^5, find the maximum distance upto which laminar boundary layer persists, and find its maximum thickness. Assume kinematic viscosity of air as 1.49×10^{-5} m²/s.*

Solution

$$V = \frac{100 \times 10^3}{3600} = 27.78 \text{ m/s}$$

$$Re_x = \frac{Vx}{\upsilon}$$

or

$$3 \times 10^5 = \frac{27.78 \times x}{1.49 \times 10^{-5}}$$

$\therefore$

$$x = 0.161 \text{ m}$$

$$\delta = 5\sqrt{\frac{x\upsilon}{V}}$$

$$= 5\sqrt{\frac{0.161 \times 1.49 \times 10^{-5}}{27.78}}$$

$$= 1.47 \times 10^{-3} \text{ m} = 1.47 \text{ mm}$$

Example 12.5. *Find the power required to tow lengthwise a plate 1.2 m wide and 3 m long at a velocity of 2.4 m/s in water at 23°C. Make allowance for the fact that the boundary layer will change from laminar to turbulent over the plate. υ for water at 23°C is 0.9×10^{-6} m²/s and $\rho = 1000$ kg/m³ (10² msl/m³).*

Solution

$$Re_L = \frac{VL}{\upsilon} = \frac{2.4 \times 3}{0.9 \times 10^{-6}} = 8.0 \times 10^6$$

$$C_f = \frac{0.074}{(Re_L)^{1/5}} - \frac{1700}{Re_L}$$

or

$$C_f = \left[\frac{0.074}{(8.0 \times 10^6)^{1/5}} - \frac{1700}{8.0 \times 10^6}\right] = 2.87 \times 10^{-3}$$

The drag on the plate is given by

$$F_D = 2C_f \times BL \times \frac{\rho V^2}{2}$$

SI units

$$F_D = \left[2 \times 2.87 \times 10^{-3} \times 1.2 \times 3 \times \frac{1000 \times (2.4)^2}{2}\right]$$

$$= 59.51 \text{ N}$$

$\therefore$ Power required $= 59.51 \times 2.4$

$$= 143 \text{ W} = 0.143 \text{ kW}$$

Metric units

$$F_D = \left[2 \times 2.87 \times 10^{-3} \times 1.2 \times 3 \times \frac{102 \times (2.4)^2}{2}\right]$$

$$= 6.07 \text{ kg(f)}$$

$\therefore$ Power required $= \frac{6.07 \times 2.4}{75} = 0.194$ h.p.

Example 12.6. *Find the ratio of skin friction drag on the front two-third and rear one-third of a flat plate kept in a uniform stream at zero incidence. Assume the boundary layer to be turbulent over the entire plate.*

Solution

$$C_f = \frac{0.074}{(Re_L)^{1/5}}$$

For the front two-third portion of the plate

$$Re_L = \frac{V(2/3)L}{\upsilon}$$

Thus $$C_f = \frac{0.074}{\left[\frac{V(2/3)L}{\upsilon}\right]^{1/5}}$$

∴ Drag for the front two-third portion of the plate is

$$F_{D1} = C_f \times BL \times \frac{\rho V^2}{2}$$

$$= \frac{0.074}{\left[\frac{V(2/3)L}{\upsilon}\right]^{1/5}} \times B \times (2/3)\,L \times \frac{\rho V^2}{2}$$

Similarly drag for the entire plate is

$$F_D = \frac{0.074}{\left[\frac{VL}{\upsilon}\right]^{1/5}} \times B \times L \times \frac{\rho V^2}{2}$$

∴ Drag for the rear one-third portion of the plate is

$$F_{D2} = (F_D - F_{D1})$$

$$= \frac{0.074}{\left[\frac{VL}{\upsilon}\right]^{1/5}} \times B \times L \times \frac{\rho V^2}{2} \times \left[1 - \frac{2}{3}\left(\frac{3}{2}\right)^{1/5}\right]$$

$$\therefore \quad = \frac{F_{D_1}}{F_{D_2}} = \frac{\frac{2}{3}\left(\frac{3}{2}\right)^{1/5}}{\left[1-\frac{2}{3}\left(\frac{3}{2}\right)^{1/5}\right]} = 2.61$$

Example 12.7. *Calculate the friction drag on a plate 0.15 m wide and 0.45 m long placed longitudinally in a stream of oil flowing with a free stream velocity of 6 m/s. Also find the thickness of the boundary layer and shear stress at the trailing edge. Sp. gr. of oil is 0.925 and its kinematic viscosity is 0.9 × 10⁻⁴ m²/s (0.9 stokes).*

Solution

$$Re_L = \frac{6 \times 0.45}{0.9 \times 10^{-4}} = 3.0 \times 10^4$$

Hence the boundary layer is laminar over the entire length of the plate.

Thus $$C_f = \frac{1.328}{\sqrt{Re_L}} = \frac{1.328}{\sqrt{3.0 \times 10^4}} = 7.667 \times 10^{-3}$$

SI units

The friction drag for both sides of the plate is

$$F_D = 2 \times BL \times C_f\,\frac{\rho V^2}{2}$$

$$= 2 \times (0.15 \times 0.45) \times 7.667 \times 10^{-3} \times (0.925 \times 1000) \times \frac{(6)^2}{2}$$

$$= 17.233 \text{ N}$$

Thickness of the boundary layer at the trailing edge is given by Eq. 12.14 for $x = L$ as

$$\delta = 5\sqrt{\frac{L\upsilon}{V}}$$

$$= 5\sqrt{\frac{0.45 \times 0.9 \times 10^{-4}}{6}} = 13 \times 10^{-3}\text{ m} = 13 \text{ mm}$$

Shear stress at the trailing edge is given by Eq. 12.17 for $x = L$ as

$$\tau_0 = \frac{0.664 \times \rho V^2}{2\sqrt{Re_L}}$$

$$= \frac{0.664 \times (0.925 \times 1000) \times (6)^2}{2 \times \sqrt{3.0 \times 10^4}} = 63.83 \text{ N/m}^2$$

Metric units

The friction drag for both sides of the plate is

$$F_D = 2 \times BL \times C_f\,\frac{\rho V^2}{2}$$

$$= 2 \times (0.15 \times 0.45) \times 7.667 \times 10^{-3} \times (0.925 \times 102) \times \frac{(6)^2}{2}$$

$$= 1.758 \text{ kg(f)}$$

Thickness of the boundary layer at the trailing edge is given by Eq. 12.14 for $x = L$ as

$$\delta = 5\sqrt{\frac{L\upsilon}{V}}$$

$$= 5\sqrt{\frac{0.45 \times 0.9 \times 10^{-4}}{6}} = 1.3 \times 10^{-2}\text{ m} = 1.3 \text{ cm}$$

Shear stress at the trailing edge is given by Eq. 12.17 for $x = L$, as

$$\tau_0 = \frac{0.664 \times \rho V^2}{2\sqrt{Re_L}}$$

$$= \frac{0.664 \times (0.925 \times 102) \times (6)^2}{2 \times \sqrt{3.0 \times 10^4}} = 6.51 \text{ kg(f)/m}^2$$

Example 12.8. *Assuming that the velocity distribution in the boundary layer is given by*

$$\frac{v}{V} = \left(\frac{y}{\delta}\right)^{1/7}$$

calculate $\frac{\delta^*}{\delta}$, $\frac{\theta}{\delta}$ *and* $\frac{\delta_E}{\delta}$. *If at a certain section, free stream velocity V was observed to be 10 m/s and the thickness of the boundary layer as 25 mm, then calculate the energy loss per unit length due to the formation of the boundary layer. Take ρ = 1.226 kg/m³ (0.125 msl/m³).*

Solution

The displacement thickness is given by Eq. 12.1 as

$$\delta^* = \int_0^\delta \left(1-\frac{v}{V}\right)dy$$

$$= \int_0^\delta \left[1-\left(\frac{y}{\delta}\right)^{1/7}\right]dy = \frac{\delta}{8}$$

$$\therefore \qquad \frac{\delta^*}{\delta} = \frac{1}{8} = 0.125$$

The momentum thickness is given by Eq. 12.2 as

$$\theta = \int_0^\delta \frac{v}{V}\left(1-\frac{v}{V}\right)dy$$

$$= \int_0^\delta \left(\frac{y}{\delta}\right)^{1/7}\left[1-\left(\frac{y}{\delta}\right)^{1/7}\right]dy = \frac{7}{72}\delta$$

$$\therefore \qquad \frac{\theta}{\delta} = \frac{7}{72} = 0.097$$

The energy thickness is given by Eq. 12.3 as

$$\delta_E = \int_0^\delta \frac{v}{V}\left(1-\frac{v^2}{V^2}\right)dy$$

$$= \int_0^\delta \left(\frac{y}{\delta}\right)^{1/7}\left[1-\left(\frac{y}{\delta}\right)^{2/7}\right]dy = \frac{7}{40}\delta$$

$$\therefore \qquad \frac{\delta_E}{\delta} = \frac{7}{40} = 0.175$$

The loss of energy per unit length due to the formation of the boundary layer is given by

$$E_L = \frac{1}{2}\rho V^3 \delta_E$$

SI units

$$E_L = \frac{1}{2} \times 1.226 \times (10)^3 \times \frac{7}{40} \times 25 \times 10^{-3}$$

$$= 2.682 \text{ N.m/s}$$

Metric units

$$E_L = \frac{1}{2} \times 0.125 \times (10)^3 \times \frac{7}{40} \times 25 \times 10^{-3}$$

$$= 0.273 \text{ kg(f)-m/s}$$

Example 12.9. *It is required to determine the frictional drag of a submarine. The length of the hull is 75 m and its surface area is 3000 m². The submarine is travelling at a constant speed of 5 m/s. Critical Reynolds number at which the flow in the boundary layer changes from laminar to turbulent is 5 × 10⁵. Assuming that the boundary layer at the leading edge is laminar, obtain the frictional drag and the power required to propel the submarine at 5 m/s. Take υ = 1 × 10⁻⁶ m²/s (0.01 cm²/s) and ρ = 1000 kg/m³ (102 msl/m³).*

Solution

$$Re_L = \frac{VL}{\upsilon} = \frac{5\times75}{1\times10^{-6}} = 3.75\times10^8$$

Since at leading edge boundary layer is laminar, it changes from laminar to turbulent on the surface of the submarine.

As critical Reynolds number = 5×10^5

$$C_f = \frac{0.074}{(Re_L)^{1/5}} - \frac{1700}{Re_L}$$

$$= \left[\frac{0.074}{(3.75\times10^8)^{1/5}} - \frac{1700}{3.75\times10^8}\right]$$

$$= 1.422 \times 10^{-3}$$

SI units

The frictional drag is

$$F_D = C_f \rho A \frac{V^2}{2}$$

$$= 1.422 \times 10^{-3} \times 1000 \times 3000 \times \frac{(5)^2}{2}$$

$$= 53.325 \times 10^3 \text{ N} = 53.325 \text{ kN}$$

Power required $= (53.325 \times 10^3 \times 5)$

$= 266.625 \times 10^3 \text{ W} = 266.625 \text{ kW}$

Metric units

The frictional drag is

$$F_D = C_f \rho A \frac{V^2}{2}$$

$$= 1.422 \times 10^{-3} \times 102 \times 3000 \times \frac{(5)^2}{2}$$

$$= 5.44 \times 10^3 \text{ kg (f)}$$

Power required $= \dfrac{5.44\times10^3 \times 5}{75} = 362.7$ h.p.

SUMMARY OF MAIN POINTS

1. When a real fluid flows past a soild boundry, a layer of fluid adheres to the boundary surface due to viscosity of the fluid, and a small region is developed in the vicinity of the boundary surface in which the velocity of flowing fluid increases gradually from zero at the boundary surface to the velocity of the main stream or free stream velocity. This region is known as *boundry layer*, in which there exists a large velocity gradient $(\partial v/\partial y)$ normal to the boundary surface.
2. The nominal thickness of the boundary layer δ is considered equal to the distance y from the boundary surface at which the velocity v reaches 99% of the free stream velocity V, i.e., $v = 0.99$ V.
3. The *displacement thickness* δ^* is defined as the distance by which the boundary surface would have to be displaced outwards so that the total actual discharge would be same as that of an ideal fluid past the displaced boundary. It may be expressed by the equation

$$\delta^* = \int_0^{\infty}\left(1-\frac{v}{V}\right)dy$$

4. The *momentum thickness* θ is defined as the distance from the actual boundary surface such that the momentum flux corresponding to the main stream velocity V through this distance θ is equal to the deficiency or loss of momentum due to the boundary layer formation. It may be mathematically express by the equation

$$\theta = \int_0^{\infty}\frac{v}{V}\left(1-\frac{v}{V}\right)dy$$

5. The *energy thickness* δ_E is defined as the distance from the actual boundary surface such that the energy flux corresponding to the main stream velocity V through this distance δ_E is equal to the deficiency or loss of energy due to the boundary layer formation. It may be mathematically expressed by the equation

$$\delta_E = \int_0^{\infty}\frac{v}{V}\left(1-\frac{v^2}{V^2}\right)dy$$

6. Von-Karman momentum integral equation of the boundary layer is given as

$$\frac{\tau_0}{\rho V^2} = \frac{d\theta}{dx}$$

where

τ_0 = shear stress at the boundary surface.

7. In a laminar boundary layer the flow exhibits all the characteristics of laminar flow. The velocity distribution in a laminar boundary layer is parabolic. The laminar boundary layer exists when the Reynolds number $Re_x = \dfrac{Vx}{\nu}$ is less than 5×10^5,

where

V = Free stream velocity of flowing fluid ;

x = distance from the leading edge of the solid boundary ; and

ν = kinematic viscosity of the flowing fluid.

The thickness of laminar boundary layer is given by

$$\delta = \frac{5x}{\sqrt{Re_x}}$$

8. From the exact analytical solution of the boundary layer equations by Blasius the following expressions for the displacement thickness and the momentum thickness have been obtained.

$$\delta^* = \frac{1.729x}{\sqrt{Re_x}}$$

$$\theta = \frac{0.664x}{\sqrt{Re_x}}$$

9. The average drag coefficient C_f obtained by Blasius by exact solution of the boundary layer equations is given by

$$C_f = \frac{1.328}{\sqrt{Re_L}} = \frac{1.328}{\sqrt{VL/\nu}}$$

where

L = length of the plate.

Note : The expressions for δ, δ^*, q and C_f have been obtained by considering a laminar boundary layer formed along a long thin plate held stationary in a uniform stream in the direction parallel to the flow.

10. In turbulent boundary layer the flow exhibits the characteristics of turbulent flow. The velocity distribution in a turbulent boundary layer follows a lagarithmic law i.e., $v \sim \log y$, which can also be represented by a power law of the type

$$\frac{v}{V} = \left(\frac{y}{\delta}\right)^n$$

The value of the exponent n is approximately (1/7) for moderate Reynolds number $\left(Re_x = \dfrac{Vx}{v} < 10^7\right)$ for a flat plate.

Turbulent boundary layer is usually thicker than laminar boundary layer.

11. The thickness of turbulent boundary layer is given by

$$\delta = \frac{0.376x}{(Re_x)^{1/5}}$$

12. The average drag coefficient C_f for turbulent boundary layer developed along a long thin plate held stationary in a uniform stream in the direction parallel to the flow is given by

$$C_f = \frac{0.074}{(Re_L)^{1/5}}$$

13. If the plate is covered with both laminar and turbulent boundary layers the drag coefficient C_f is given by

$$C_f = \frac{0.074}{(Re_L)^{1/5}} - \frac{1700}{Re_L}$$

14. If the plate is smooth, even in the zone of turbulent boundary layer, there exists a very thin layer immediately adjacent to the boundary, in which the folw is laminar. This thin layer is commonly known as *Laminar Sublayer*. The thickness of laminar sublayer is represented by δ' which is given by

$$\delta' = \frac{11.6\,\nu}{V_*}$$

in which $V_* = \sqrt{(\tau_0/\rho)}$ is known as shear or friction velocity.

15. With the pressure increasing in the direction of flow i.e., with positive pressure gradient, the boundary layer separates from the boundary surface, and back flow and formation of eddies takes place due to which a great loss of energy occur. This phenomenon is dalled *separation of flow.*
16. The conditions for attached flow, separation and detached flow are :

(i) For attached flow

$$\left(\frac{\partial v}{\partial y}\right)_{y=0} > 0 \ ; \ \frac{\partial p}{\partial x} < 0$$

(ii) For separation

$$\left(\frac{\partial V}{\partial y}\right)_{y=0} = 0; \ \frac{\partial p}{\partial x} = 0$$

(iii) For detached flow

$$\left(\frac{\partial V}{\partial y}\right)_{y=0} < 0; \ \frac{\partial p}{\partial x} > 0$$

17. The various methods of controlling the formation as well as separation of boundary layer are as follows :

(i) Motion of solid boundary.

(ii) Acceleration of fluid in the boundary layer.

(iii) Suction of fluid from the boundary layer.

(iv) Streamlining of body shapes.

PROBLEMS

12.1 Explain the characteristics of laminar and turbulent boundary layer.

12.2 What are the factors affecting the boundary layer thickness?

12.3 What do you understand by displacement thickness and momentum thickness?

12.4 For a linear velocity distribution in the boundary layer on a flat plate show that $(\delta^*/\theta) = 3$ and $(\delta E/\delta) = 0.25$.

12.5 Discuss the phenomenon of separation in a diverging flow.

12.6 Discuss (i) the concept of the boundary layer with reference to fluid motion over a flat plate; (ii) phenomenon of separation for flow over curved surfaces; (iii) the prevention of separation.

12.7 Define boundary layer and explain the fundamental causes of its existence. Also discuss the various methods of controlling the boundary layer.

12.8 For the following types of velocity distribution obtain the values of (δ^*/δ) and (θ/δ).

(i) $\frac{v}{V} = 2\eta - \eta^2$; (ii) $\frac{v}{V} = 2\eta - 2\eta^3 + \eta^4$ where $\eta = (y/\delta)$.

$$\left[Ans.\ \text{(i)}\ \frac{1}{3}, \frac{2}{15};\quad \text{(ii)} \frac{3}{10}, \frac{37}{31}\right]$$

12.9 If a laminar boundary layer at zero pressure gradient over a flat plate is described by the velocity profile

$$\frac{v}{V_0} = \frac{3}{2}\eta - \frac{\eta^3}{2}$$

in which $\eta = (y/\delta)$, show that boundary layer thickness δ, wall shear stress τ_o and coefficient of drag C_D are given by

$$d = \frac{4.65x}{\sqrt{Re_x}}\ ; t_o = \frac{0.322\rho V_0^2}{\sqrt{Re_x}}\ ;\ C_D = \frac{1.328}{\sqrt{Re_L}}$$

12.10 A smooth flat plate 1 m wide and 1.5 m long is rowed lengthwise through still air with a velocity of 10 m/s. Assuming the boundary layer to be fully laminar, estimate its thickness at the trailing edge. Mass density and kinematic viscosity of the air are 1.216 kg/m³ (0.124 msl/m³) and 0.15 × 10^{-4} m²/s (0.15 cm²/s) respectively. Also calculate the shear stress at that point.

[*Ans.* 7.5 mm; 40.37 × 10^{-3} N/m²[41.17 × 10^{-4} kg(f)/m²]]

12.11 Find the ratio of skin friction drag on the front half and rear half portions of a flat plate kept in a uniform stream of zero incidence. Assume the boundary layer to be turbulent over the entire plate. [*Ans.* 1.35]

12.12 Air (kinematic viscosity 0.15 × 10^{-4} m²/s) flows at 10 m/s past a smooth rectangular flat plate 0.3 m wide and 3 m long. Assuming that the turbulence level in the oncoming stream is low and that transition occurs at $Re_x = 5 \times 10^5$, calculate the ratio of the total drag when the flow is parallel to the length of the plate to the value when the flow is parallel to the width. [*Ans.* 1.08]

10^{-3}

12.13 A smooth flat plate 2.4 m long and 0.9 m wide moves lengthwise at 6 m/s through still atmospheric air of density 1.226 kg/m³ (0.125 msl/m³) and kinematic viscosity 1.49 × 10^{-5} m²/s. Assuming the boundary layer to be entirely laminar, calculate the boundary layer thickness at the trailing edge of the plate, the shear stress half-way along the plate and the power required to move the plate. What power would be required if the boundary layer was made turbulent at the leading edge?

[*Ans.* 12.2 mm; 2.108× N/m² (2.149 × kg(f)/m²); 0.773 × kW (1.05 × h.p.); 2.688× kW (3.655× h.p.)]

Laminar Flow

Chapter 13

13.1 INTRODUCTION

In laminar flow the fluid particles move along straight parallel paths in layers or laminae, such that the paths of the individual fluid particles do not cross those of the neighbouring particles. It occurs at low velocity so that forces due to viscosity predominate over the inertial forces. The viscosity of fluid induces relative motion within the fluid as the fluid layers slide over each other, which in turn gives rise to shear stresses. The magnitude of the viscous shear stress so produced, varies from point to point, being maximum at the boundary and gradually decreasing with increase in the distance from the boundary. The shear stress so produced result in developing a resistance to flow. In order to overcome the shear resistance to flow the pressure drops from section to section in the direction of flow, so that a pressure gradient exists. An expression relating shear and pressure gradients in laminar flow may be developed, which may be used to analyse the various cases of laminar flow as indicated below.

13.2 RELATION BETWEEN SHEAR AND PRESSURE GRADIENTS IN LAMINAR FLOW

Consider a free-body of fluid having the form of an elementary parallelopiped of length δx, width δz, thickness δy as shown in Fig. 13.1. On account of relative motion between different layers of fluid, the velocity distribution is non-uniform. Thus the fluid layer $abcd$ is moving at a higher velocity than the layer below it and hence the layer $abcd$ exerts a shear stress in the positive direction on the lower layer. The lower layer on the other hand, exerts an equal and opposite shear stress on the layer $abcd$. Similarly it can be seen that a shear stress is exerted on the layer $a'b'c'd'$ by the layer above it in the positive x-direction. The magnitudes of shear stresses on the layers $abcd$ and $a'b'c'd'$ will be different. Thus if τ represents the shear stress on the layer $abcd$ then the shear stress on the layer $a'b'c'd'$ is equal to

$$\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right).$$

For two-dimensional steady flow there will be no shear stresses on the vertical faces $abb'a'$ and $cdd'c'$. Thus the only forces acting on the parallelopiped in the direction of flow x will be the pressure and shear forces as shown in Fig. 13.1. The net shear force acting on the parallelopiped

$$= \left[\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right)\delta x \delta z - \tau \delta x \delta z\right] = \frac{\partial \tau}{\partial y}\delta x \delta y \delta z.$$

ensity on face $add'a'$ is p, and since there exists a pressure gradient in the direction

re intensity on the face $bcc'b'$ will be $\left(p + \frac{\partial p}{\partial x}\delta x\right)$. The net pressure force acting on the

$$= \left[p\delta y \delta z - \left(p + \frac{\partial p}{\partial x}\delta x\right)\delta y \delta z\right] = -\left(\frac{\partial p}{\delta x}\right)\delta x \delta y \delta z.$$

uniform flow, there being no acceleration in the direction of motion, the sum of these rection must be equal to zero. Thus

$$\left(\frac{\partial \tau}{\partial y}\right)\delta x\, \delta y\, \delta z - \left(\frac{\partial p}{\partial x}\right)\delta x, \delta y, \delta z = 0$$

$$\frac{\partial \tau}{\partial y} = \frac{\partial p}{\partial x} \quad ...(13.1)$$

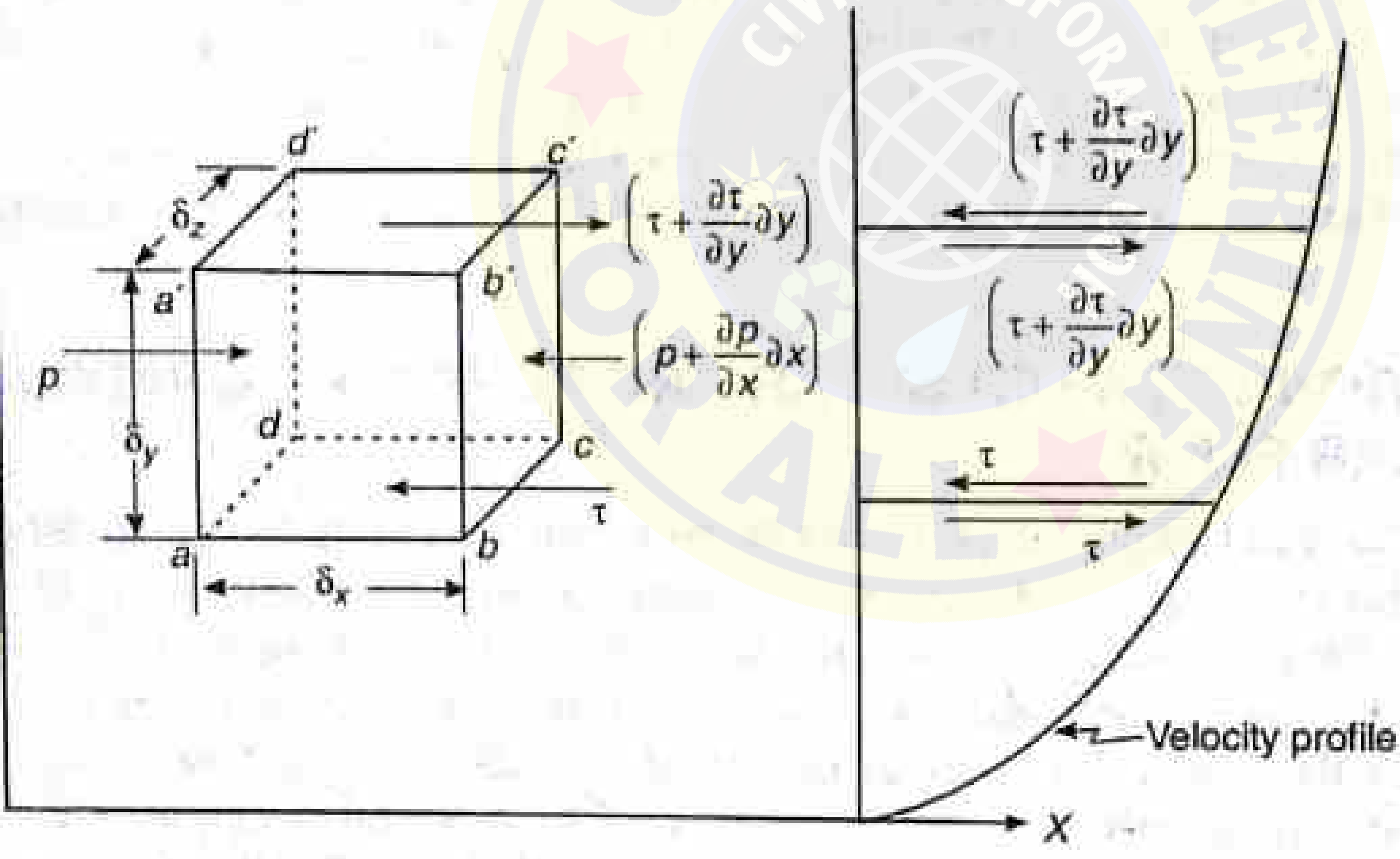


Figure 13.1 *Forces on a fluid element in laminar flow*

1 indicates that in a steady uniform laminar flow the pressure gradient in the direction to the shear stress gradient in the normal direction. Further for steady uniform flow, on is absent, it is apparent that the pressure gradient $(\partial p/\partial x)$ is independent of y and gradient $(\partial \tau/\partial y)$ is independent of x.

r viscous fluids according to Newton's law of viscosity $\tau = \mu(\partial v/\partial y)$. Thus substituting the following differential equation for laminar flow is obtained direction opposite to that of the flow of fluid and on account of symmetry its magnitude will be constant on whole of the periphery. The total pressure forces on the left and the right faces of the cylinder are $(p)\pi r^2$ and $\left(p+\frac{\partial p}{\partial x}dx\right)\pi r^2$ respectively and total shear force acting on the periphery of the cylindrical element is $\tau(2\pi r)dx$. Since the summation of all these forces in the x-direction must be equal to zero,

$$p(\pi r^2)-\left(p+\frac{\partial p}{\partial x}dx\right)\pi r^2-\tau(2\pi r)dx = 0$$

or $$\left(p+\frac{\partial p}{\partial x}dx\right)\pi r^2-\tau(2\pi r)dx = 0$$

Dividing the above equation by the volume of the element $(\pi r^2)dx$ and further simplifying it, we get

$$\tau = -\frac{\partial p}{\partial x}\frac{r}{2} \quad \text{...(13.3)}$$

Equation 13.3 shows that the shear stress τ varies linearly along the radius of the pipe (Fig. 13.3). At the centre of the pipe since $r=0$, the shear stress τ is zero and at the pipe wall, since $r=R$ the maximum value of the stress τ_0 is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} \quad \text{...(13.4)}$$

The negative sign for the pressure gradient $\left(\frac{\partial p}{\partial x}\right)$ indicates a decrease of fluid pressure in the direction of flow. The pressure must decrease in the direction of flow in a horizontal pipe because pressure force is the only means available to compensate for the resistance to the flow, the potential and kinetic energies remain constant. Thus the term $\left(-\frac{\partial p}{\partial x}\right)$ is positive. Equation 13.3 however holds good for both laminar and turbulent flow, since in deriving it no assumptions were made about the nature of the flow.

Since the pipe is uniform the pressure gradient will be constant and hence

$$\left(-\frac{\partial p}{\partial x}\right) = \left(\frac{p_1-p_2}{L}\right) = \frac{wh_f}{L}$$

Introducing the above expression in Eq. 13.3, we get

$$\tau = \frac{wh_f}{2L}r \quad \text{...(13.5)}$$

In laminar flow the shear stress τ is entirely due to viscous action and therefore it may be evaluated by using Newton's law of viscosity, according to which $\tau=\mu\left(\frac{\partial v}{\partial y}\right)$; in which v is the velocity of flow at a distance y from the pipe wall. Since in the present case $y=(R-r)$, it follows that $dy=-dr$ and the expression for τ then becomes

$$\tau = -\mu\frac{\partial v}{\partial r} \quad \text{...(13.6)}$$

in which the negative sign is on account of particular choice of coordinates and predicts mathematically that velocity v decreases as the radius r increases.

Substituting the value of τ from Eq. 13.6 in Eq. 13.3, we have

$$-\mu\frac{\partial v}{\partial r} = -\frac{\partial p}{\partial x}\frac{r}{2}$$

or $$\frac{\partial v}{\partial r} = \frac{1}{\mu}\frac{\partial p}{\partial x}\frac{r}{2} \quad \text{...(13.7)}$$

For steady uniform flow since the drop in pressure depends only on the distance x and is independent of r, the pressure gradient $(\partial p/\partial x)$ in the direction of flow must have a constant value. The integration of Eq. 13.7 with respect to r then yields the expression for the velocity distribution for laminar flow in a circular pipe as

$$v = \frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)r^2+C \quad \text{...(13.8)}$$

The constant of integration C can be evaluated from the boundary condition that at $r=R$, the velocity $v=0$.

$$\therefore \quad C = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)R^2$$

Equation 13.3 then becomes

$$v = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2-r^2) \quad \text{...(13.9)}$$

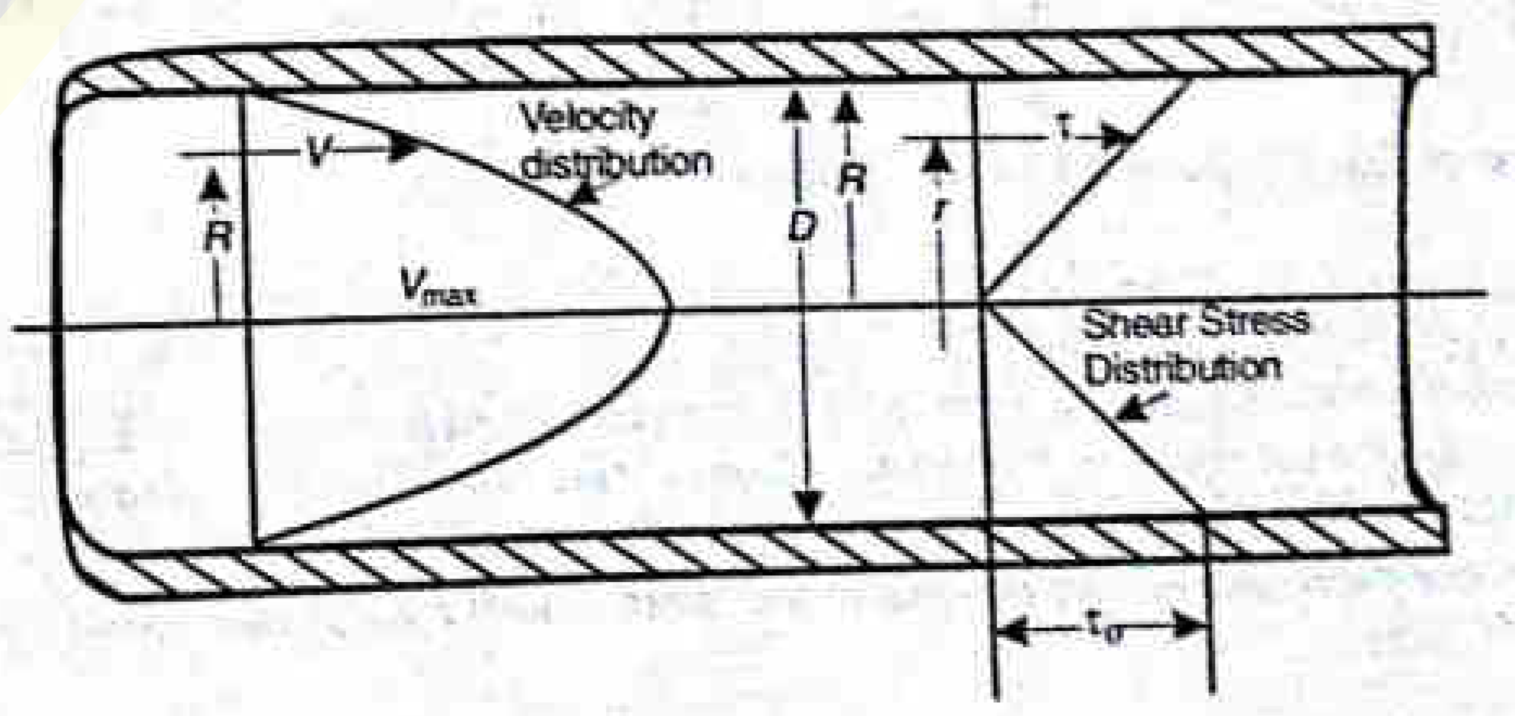


Figure 13.3 Velocity and shear-stress distribution for laminar flow in a circular pipe

Contd.

Equation 13.9 shows that in laminar flow through circular pipes, velocity of flow varies parabolically, and the surface of velocity distribution is a paraboloid of revolution as shown in Fig. 13.3. The maximum velocity v_{max} occurs at the axis of the pipe and has the magnitude of

$$v_{max} = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)R^2 \quad ...(13.10)$$

by combining Eq. 13.9 and 13.10 the expression for the velocity distribution can also be written in terms of the maximum velocity as

$$v = v_{max}\left[1-\left(\frac{r}{R}\right)^2\right] \quad ...(13.11)$$

The discharge Q passing through any cross-sectio of a circular pipe can be obtained by integrating a small discharge passing through an elementary ring of thickness dr, considered at a radial distance r, out of the cross sectional area of the pipe, as shown in Fig. 13.2.

Since $\quad dQ = vdA = v(2pr)\,dr$

$$= \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2-r^2)$$

By integrating both sides, we get

$$Q = \frac{\pi}{2\mu}\left(-\frac{\partial p}{\partial x}\right)\int_0^R (R^2-r^2)\,r\,dr$$

or

$$Q = \frac{\pi}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^4 \quad ...(13.12)$$

If D is the diameter of the pipe, then Eq. 13.12 expressed in terms of D becomes

$$Q = \frac{\pi}{128\mu}\left(-\frac{\partial p}{\partial x}\right)D^4 \quad ...(13.13)$$

The mean velocity of flow V, which is equal to (Q/A) or $(Q/\pi R^2)$, is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^2 = \frac{1}{32\mu}\left(-\frac{\partial p}{\partial x}\right)D^2 \quad ...(13.14)$$

Comparison of Eqs 13.10 and 13.14 reveals that

$$V = \frac{1}{2}v_{max} \quad ...(13.15)$$

The relation between the mean and the maximum velocities of flow, given in Eq. 13.15, also follows from the geometry of a paraboloid of revolution, since it is known that the volume of a paraboloid of revolution is equal to half the volume of the circumscribing cylinder.

The point where the local velocity is equal to the mean velocity V, can be located by combining Eqs 13.11 and 13.15. Thus

$$v = v_{max}\left[1-\left(\frac{r}{R}\right)^2\right] = V = \frac{1}{2} = v_{max}$$

or

$$r = \frac{1}{\sqrt{2}}R = 0.707R \quad ...(13.16)$$

That is the mean velocity of flow V occurs at a radial distance of $(0.707R)$ from the centre of the pipe. Rearranging Eq. 13.14 to solve for pressure drop, we have

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{32\mu V}{D^2}$$

or

$$\frac{p_1-p_2}{L} = \frac{32\mu V}{D^2}$$

or

$$(p_1-p_2) = \frac{32\mu VL}{D^2} = \frac{128\mu QL}{\pi D^4} \quad ...(13.17)$$

Equation 13.17 is known as Hagen–Poiseuille equation for laminar flow in the circular pipes. It was first determined experimentally by a German Engineer G.H.L. Hagen in 1839 and almost simultaneously but independently by J.L.M. Poiseuille, a French physician in 1840. The striking feature of Hagen–Poiseuille equation is that it involves no empirical coefficient or experimental factor of any kind, except for the physical properties such as viscosity and specific weight of the flowing fluid.

Further if h_f represents the drop in pressure head, then

$$\left.\begin{aligned} h_f &= \frac{p_1-p_2}{w} = \frac{32\mu VL}{wD^2} \\ \text{or} \quad h_f &= \frac{64}{Re}\left(\frac{LV^2}{2gD}\right) = \left(\frac{128\mu QL}{w\pi D^4}\right)\end{aligned}\right\} \quad ...(13.18)$$

where Reynolds number $Re = (rDV/\mu)$ and w is the specific weight of the flowing fluid. It may again be noted that the drop in pressure head h_f is nothing but the head loss over length L of the pipe, which varies directly as the first power of the velocity of flow of fluid and inversely as the square of the diameter of the pipe.

Several investigators have verified the validity of the Hagen–Poiseuille equation, and hence it is often employed for experimental determination of the fluid viscosity. From Eq. 13.17, we have

$$\mu = \frac{(p_1-p_2)D^2}{32VL} = \frac{(p_1-p_2)D^4\pi}{128QL} \quad ...(13.19)$$

The pressure drop (p_1-p_2) in a fixed length L of a pipe of diameter D for any given rate of laminar flow can be measured. Thus all the quantities on the right hand side of Eq. 13.19 will be known, from which the value of the coefficient of viscosity μ of the fluid can be obtained.

The loss of head due to frictional resistance in a long straight pipe of length L and diameter D may also be expressed by Darcy–Weisbach equation as

$$h_f = \frac{p_1-p_2}{w} = f\frac{L}{D}\frac{V^2}{2g} \quad ...(13.20)$$

where f is friction factor and V is mean velocity of flow. Equating the two values of h_f given by Eqs. 13.18 and 13.20, we obtain

$$\frac{fLV^2}{2gD} = \frac{32\mu VL}{wD^2}$$

or
$$f = \frac{64\mu}{\rho VD} = \frac{64}{Re} \qquad ...(13.21)$$

Thus the value of friction factor f, for laminar flow of fluid is given by Eq. 13.21.

Further from Eq. 13.4

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} = \left(\frac{p_1 - p_2}{L}\right)\frac{D}{4}$$

By substituting for $(p_1 - p_2)$ from Eq. 13.20 in the above expression, we get

$$\tau_0 = \frac{(wh_f)D}{4L} = \frac{wfV^2}{8g} = \frac{\rho fV^2}{8} \qquad ...(13.22)$$

where ρ is mass density of fluid.

From Eq. 13.22, we obtain

$$\sqrt{\frac{\tau_0}{\rho}} = V_* = V\sqrt{\frac{f}{8}} \qquad ...(13.23)$$

Since the expression $\left[\sqrt{(\tau_0/\rho)}\right]$ has the dimensions of velocity, it is known as shear velocity and is denoted by V_*.

In order to maintain steady uniform laminar flow through pipes, certain power is required to overcome the resistance to flow. Since power is rate of doing work, it is equivalent to the product of force and velocity. The resistance to flow is compensated by the pressure gradient $(-\partial p/\partial x)$ in the direction of flow. The pressure gradient may be considered as the average force per unit volume of fluid. Thus if A and L are the area and length of pipe respectively, then the total force is $(-\partial p/\partial x) \times AL$, and with the mean velocity of flow V the required power is

$$P = \left(-\frac{\partial p}{\partial x}\right)ALV = Q\,(p_1 - p_2) \qquad ...(13.24)$$

since discharge $\quad Q = AV$, and $\left(-\dfrac{\partial p}{\partial x}\right) = \dfrac{(p_1 - p_2)}{L}$.

13.4 LAMINAR FLOW THROUGH INCLINED PIPES

When the pipe having laminar flow of fluid is inclined as shown in Fig. 13.4 in addition to pressure and viscous forces, gravity forces will also become effective. Considering a small concentric cylindrical fluid element, if specific weight of the fluid is w, the weight of the element is $[w(\pi r^2)dx]$ acting vertically downward, and its component in the direction of flow becomes $[w(\pi r^2)dx]\cos\theta$, in which x-direction is taken parallel to the axis of the pipe. Now if z-direction is taken vertically upward, let dz represent a change in elevation corresponding to length dx of the fluid element.

Then from Fig. 13.4, we have

$$\cos\theta = -\frac{dz}{dx} = -\frac{\partial z}{\partial x}$$

in which the minus sign has been considered because the elevation z decreases as the distance x inceases. The component of the weight of the fluid element may thus be written as

$$[w(\pi r^2)dx]\cos\theta = [w(\pi r^2)dx]\left(-\frac{dz}{dx}\right) = -w(\pi r^2)dz.$$

The other forces acting on the fluid element are the shear and pressure forces as shown in Fig. 13.4.

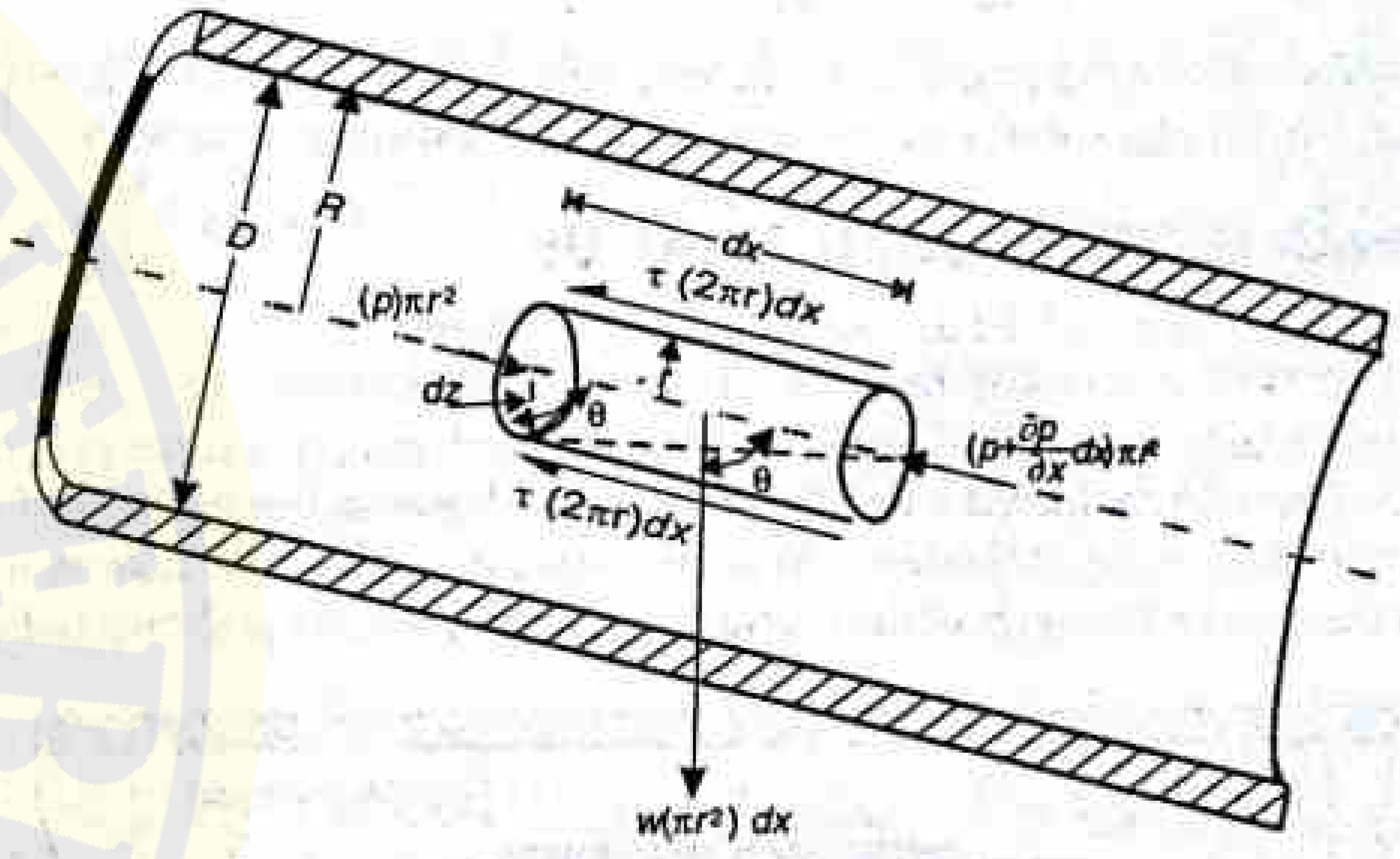


Figure 13.4 *Laminar flow through an inclined pipe*

In the absence of any acceleration, sum of all the forces acting on the element in the direction of flow must be equal to zero. Thus

$$P\pi r^2 - \left(P + \frac{\partial p}{\partial x}dx\right)\pi r^2 - w\pi r^2 dz - \tau(2\pi r)\,dx = 0$$

or
$$\tau = -\frac{\partial}{\partial x}(p + wz)\frac{r}{2} = -\frac{\partial}{\partial x}\left(\frac{p}{w} + z\right)\frac{r}{2} \qquad ...(13.25)$$

or
$$\tau = -w\frac{\partial h}{\partial x}\frac{r}{2} \qquad ...(13.25\ a)$$

in which $h = [(p/w) + z]$ represents piezometric head. Comparison of Eq. 13.25 with Eq. 13.3 would reveal that in the case of inclined pipes the term $\partial(p + wz)/\partial x$ replaces the pressure gradient terms $\partial p/\partial x$. Thus Eq. 13.25 is general expression for the distribution of shear stress for laminar flow in circular pipes, from which Eq. 13.3 can be derived since for horizontal pipe $(\partial z/\partial x) = 0$. For pipes according to Newton's law of viscosity $\tau = -\mu(\partial v/\partial r)$ as indicated in Section 13.3. Thus by substitution Eq. 13.25 becomes

$$-\mu\frac{\partial v}{\partial r} = -w\frac{\partial h}{\partial x}\frac{r}{2}$$

or

$$\frac{\partial v}{\partial r} = \frac{w}{\mu}\frac{\partial h}{\partial x}\frac{r}{2} \qquad ...(13.26)$$

Again for steady uniform flow the piezometric head gradient $(\partial h/\partial r)$ in the direction of flow must have constant value which depends on the distance x and is independent of r. Thus by integrating Eq. 13.26 with respect to r and applying the boundary condition that at $r = R$ the value of $v = 0$, we obtain

$$v = \frac{1}{4\mu}\times w\left(-\frac{\partial h}{\partial x}\right)(R^2 - r^2) \qquad ...(13.27)$$

The other expressions for this case can be similarly obtained by introducing $(w\partial h/\partial x)$ in place of $(\partial p/\partial x)$ and $w(h_1 - h_2)$ in place of $(p_1 - p_2)$ in the various other expressions of Section 13.3.

13.5 LAMINAR FLOW THROUGH ANNULUS

Consider steady laminar flow of fluid through annular space between two concentric circular tubes as shown in Fig. 13.5. Let the radius of the inner surface of the outer tube be R_1 and that of the outer surface of the inner tube be R_2. A fluid element having a shape of small concentric cylindrical sleeve of length dx and thickness dr considered at a radial distance r is chosen as free body. The forces acting on the fluid element in the direction of flow are normal pressure forces over the end areas and shear forces over inner and outer curved surfaces of the cylindrical element, which are shown in Fig. 13.5.

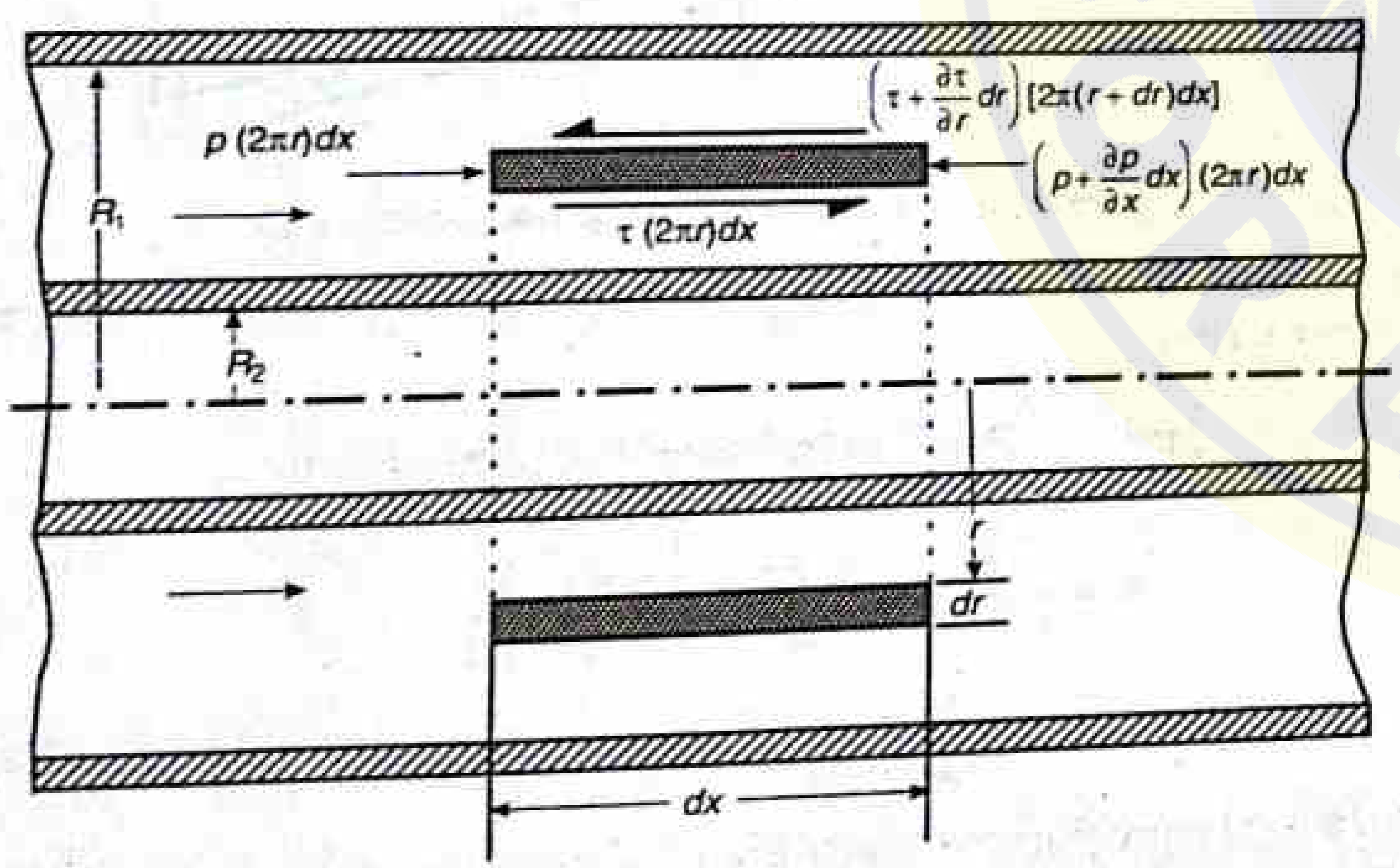


Figure 13.5 *Laminar flow through an annulus*

Again since the flow is steady and uniform, the summation of forces on the free body in the direction of flow must be zero. Thus

$$\left[p(2\pi r)dr - \left(p + \frac{\partial p}{\partial x}dx\right)2\pi r dr\right] + \left[\tau(2\pi r)dx - \left(\tau + \frac{\partial \tau}{\partial r}dr\right)\times 2\pi(r+dr)dx\right] = 0$$

or

$$\left[\left(-\frac{\partial p}{\partial x}dx\right)2\pi r dr - \left(\frac{\partial \tau}{\partial r}dr\right)2\pi r dx\right] - \left[\tau(2\pi dx)dr + \left(\frac{\partial \tau}{\partial r}dr\right)(2\pi dx)dr\right] = 0$$

Dividing through by the volume of the element $(2\pi r)\,dr\,dx$ and neglecting the last term which is a differential of higher order than the first three, we get

$$\left(\frac{\partial p}{\partial x} + \frac{\partial \tau}{\partial r} + \frac{\tau}{r}\right) = 0$$

In the above expression τ is a function of r only, and p is a function of x only. The last two terms may be combined to provide

$$\frac{\partial p}{\partial x} + \frac{1}{r}\frac{\partial}{\partial r}(\tau r) = 0. \qquad ...(13.28)$$

Since $\frac{\partial p}{\partial x}$ is not a function of r, Eq. 13.28 can be integrated with respect to r. Thus, we have

$$\left(\frac{\partial p}{\partial x}\right)\frac{r^2}{2} + \tau r = C_1$$

where C_1 is the constant of integration.

Since $\tau = -\mu(\partial v/\partial r)$, by substitution, we have

$$\left(\frac{\partial p}{\partial x}\right)\frac{r^2}{2} - \mu r\frac{\partial v}{\partial r} = C_1$$

Dividing the above expression throughout by r and again integrating it with respect to r, we get

$$\left(\frac{\partial p}{\partial x}\right)\frac{r^2}{4} - \mu v = C_1 \log_e r + C_2 \qquad ...(13.29)$$

in which C_2 is the second constant of integration.

The two constants of integration can be evaluated by applying the boundary conditions that at $r = R_1$ and $r = R_2$, $v = 0$. Thus the following expression for the velocity distribution in an annulus can be readily obtained:

$$v = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)\times\left[R_1^2 - r^2 - \frac{(R_1^2 - R_2^2)}{\log_e(R_1/R_2)}\log(R_1/r)\right] \qquad ...(13.30)$$

In order to locate the point where maximum velocity occurs, we differentiate Eq. 13.30 with respect to r and equate it to zero. Thus, we have

$$\frac{\partial v}{\partial r} = 0 = -\frac{1}{4\mu}\left(\frac{\partial p}{\partial x}\right)\left[-2r + \frac{1}{r}\frac{(R_1^2 - R_2^2)}{\log_e(R_1/R_2)}\right]$$

$$\therefore \quad r = \left[\frac{R_1^2 - R_2^2}{2\log_e(R_1/R_2)}\right]^{1/2} \quad ...(13.31)$$

By substituting this value of r in Eq. 13.30 the value of maximum velocity may be obtained.

The discharge Q passing through any cross-section of the annulus is

$$Q = \int_{R_2}^{R_1} 2\pi r\, v dr$$

$$= \left(-\frac{\pi}{8\mu}\right)\left(\frac{\partial p}{\partial x}\right)\left[R_1^4 - R_2^4 - \frac{(R_1^2-R_2^2)^2}{\log_e(R_1/R_2)}\right] \quad ...(13.32)$$

The mean velocity of flow through the annulus is given by,

$$V = \frac{Q}{\pi(R_1^2-R_2^2)} = \left(-\frac{1}{8\mu}\right)\left(\frac{\partial p}{\partial x}\right)\left[\left(R_1^2 + R_2^2 - \frac{(R_1^2-R_2^2)}{\log_e(R_1/R_2)}\right)\right] \quad ...(13.33)$$

It will be observed that if $R_2 = 0$ the problem becomes the same as laminar flow through a circular pipe and the above equations reduce to the same from as derived in Section 13.3.

If the tubes are inclined the relevant expressions for the velocity distribution and the discharge for an inclined annulus can be obtained by replacing $(\partial p/\partial x)$ by $[\partial(p+wz)/\partial x)$ as discussed in Section 13.4.

13.6 LAMINAR FLOW BETWEEN PARALLEL PLATES–BOTH PLATES AT REST

Consider laminar flow of fluid between two fixed parallel flat plates located at a distance B apart, as shown in Fig. 13.6. A small rectangular element of the fluid of length δx is considered as free-body as shown in Fig. 13.6. Let the lower face of the element be at a distance y from the lower plate and here let the velocity be v. At the upper face of the element which is at a distance of $(y + \delta y)$ from the lower plate, let the velocity be $(v + \delta v)$.

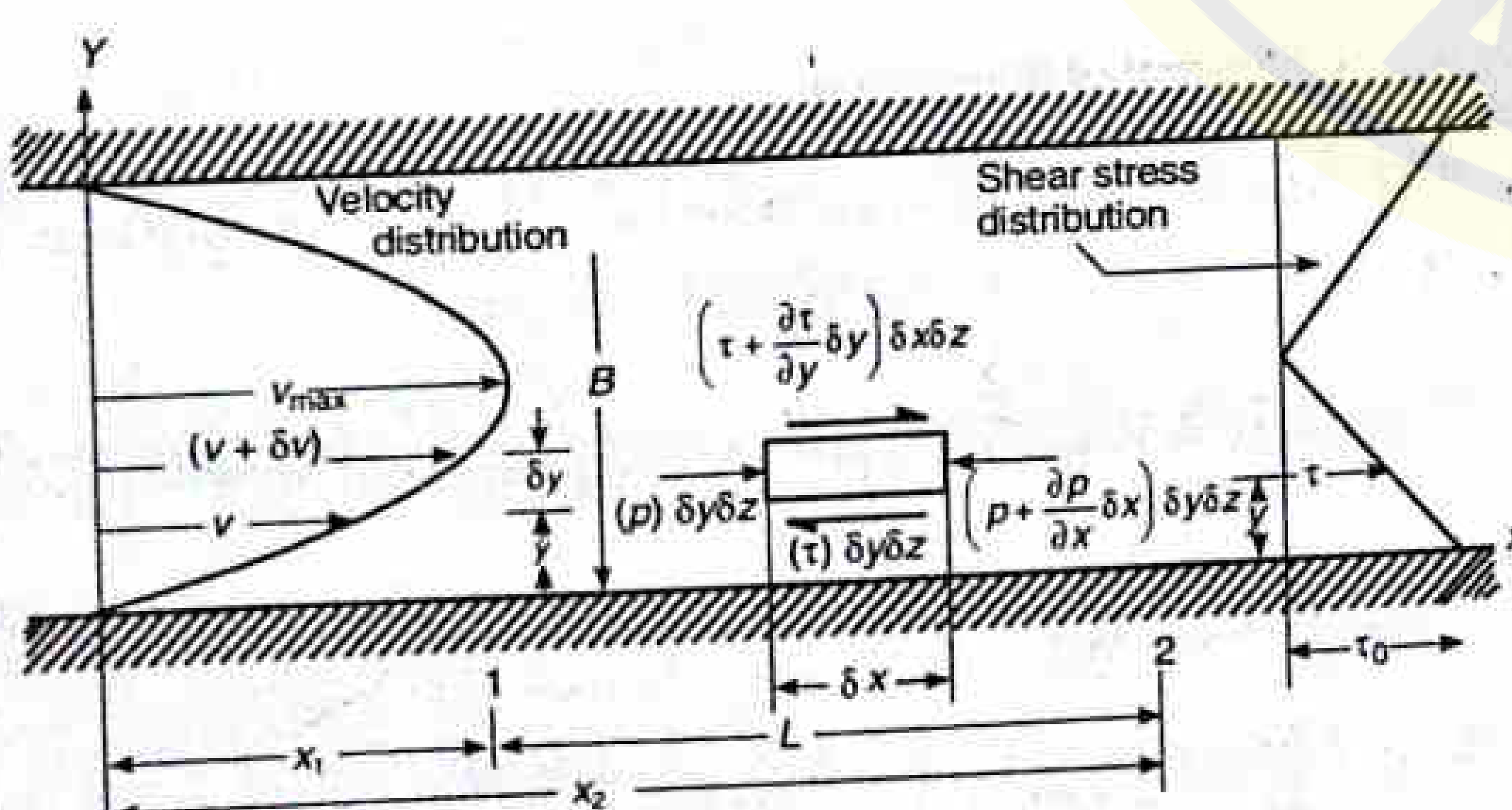


Figure 13.6 *Laminar flow between two fixed parallel flat plates*

If δv is positive, the faster moving fluid just above the upper face of the element exerts a forward force on the upper face. Similarly, the slower moving fluid adjacent to the lower face tends to retard its motion, i.e., it exerts a backward force on the lower face. Thus there are shear stresses of magnitude τ on the lower face and $\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right)$ on the upper face of the element in the directions as shown in Fig. 13.6. In order to balance the shearing forces in the fluid a pressure gradient in the direction of flow must be maintained. Thus if p is the pressure intensity at the left face of the element, then $\left(p + \frac{\partial p}{\partial x}\delta x\right)$ will be the pressure intensity on the right face of the element.

If the width of the element in the direction perpendicular to the paper is δz, the total force acting on the element towards the right is

$$\left[p - \left(p + \frac{\partial p}{\partial x}\delta x\right)\right]\partial y\, \partial z + \left[\left(\tau + \frac{\partial \tau}{\partial y}\delta y\right) - \tau\right]\partial x\, \partial z$$

But for steady and uniform flow, there is no acceleration and hence this total force must be equal to zero.

$$\therefore \quad -\frac{\partial p}{\partial x}\delta x\, \delta y\, \delta z + \frac{\partial \tau}{\partial y}\delta x\, \delta y\, \delta z = 0$$

Dividing by the volume of the element $(\delta x\, \delta y\, \delta z)$, we get

$$\frac{\partial p}{\partial x} = \frac{\partial \tau}{\partial y}$$

which is same as Eq. 13.1 derived above. Again according to Newton's law of viscosity for laminar flow the shear stress $\tau = \mu \frac{\partial v}{\partial y}$. Hence by substitution in the above equation the following differential equation for laminar flow is obtained, which is same as Eq. 13.2.

$$\frac{\partial p}{\partial x} = \mu \frac{\partial^2 v}{\partial y^2}$$

Since $\left(\frac{\partial p}{\partial x}\right)$ is independent of y, integrating the above equation twice with respect to y gives

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2 \quad ...(13.34)$$

The two constants of integration C_1 and C_2 may be evaluated by means of two boundary conditions. There being no slip of fluid at the two solid boundary surfaces of the plates,

$$v = 0 \text{ at } y = 0$$

$$\therefore \quad C_2 = 0$$

and

$$v = 0 \text{ at } y = B$$

$$\therefore \quad C_1 = -\frac{B}{2\mu}\left(\frac{\partial p}{\partial x}\right)$$

Introducing these values in Eq. 13.34 the following equation for the velocity distribution is obtained

$$v = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2) \quad ...(13.35)$$

Equation 13.35 indicates that the velocity distribution curve for the laminar flow between parallel flat plates is a parabola with its vertex being mid-way between the plates. The negative sign for the pressure gradient indicates that there is a drop in pressure in the direction of flow.

The maximum velocity v_{max} which occur at the mid-point between the plates, can be evaluated by substituting $y = B/2$ in Eq. 13.35.

Thus

$$v_{max} = \frac{B^2}{8\mu}\left(-\frac{\partial p}{\partial x}\right) \quad ...(13.36)$$

The discharge q of the laminar flow per unit width of the plate can be obtained by integrating a small discharge dq passing through strip dy.

Since $\quad dq = v\,dy$

$$\therefore \quad q = \int_0^B v\,dy = \int_0^B \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2)\,dy$$

or

$$q = \frac{B^3}{12\mu}\left(-\frac{\partial p}{\partial x}\right) \quad ...(13.37)$$

The mean velocity of flow V is given by

$$V = \frac{q}{B} = \frac{B^2}{12\mu}\left(-\frac{\partial p}{\partial x}\right) \quad ...(13.38)$$

Comparison of Eqs. 13.36 and 13.38 reveals that

$$V = \frac{2}{3}v_{max} \quad ...(13.38a)$$

The relation between the mean and the maximum velocities of flow given in Eq. 13.38 (a) also follows from the geometry of a parabola, since it is known that the area of a parabola is equal to two-thirds the area of the enclosing rectangle.

Rearranging Eq. 13.38 to solve for the pressure gradient, we get

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{12\mu V}{B^2} \quad ...(13.39)$$

Again since the flow is steady and uniform, the pressure drop $(p_1 - p_2)$ between any two sections 1 and 2 in the direction of flow at a distance $L = (x_2 - x_1)$ apart can be determined by integrating Eq. 13.39 with respect to x.

Thus

$$(p_1 - p_2) = \int_{p_1}^{p_2}(-\partial p) = \int_{x_1}^{x_2}\frac{12\mu V}{B^2}(\partial x)$$

or

$$(p_1 - p_2) = \frac{12\mu V}{B^2}(x_2 - x_1)$$

or

$$(p_1 - p_2) = \frac{12\mu VL}{B^2} \quad ...(13.40)$$

If h_f represents the drop in pressure head, then

$$h_f = \frac{p_1 - p_2}{w} = \frac{12\mu VL}{wB^2} \quad ...(13.41)$$

In which w is the specific weight of the flowing fluid. It may be noted from Eq.13.41 that the drop in the pressure head (or the loss of head) for laminar flow varies linearly with the first power of velocity.

The distribution of shear stress in the flowing fluid across any section of the parallel plates may be determined by substituting Eq. 13.35 into Newton's law of viscosity. Thus

$$\tau = \mu\frac{\partial v}{\partial y}$$

or

$$\tau = \mu\frac{\partial}{\partial y}\left[\frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2)\right]$$

or

$$\tau = \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right) \quad ...(13.42)$$

From Eq. 13.42 it is obvious that the shear stress varies linearly with the distance from the boundary. It has the same maximum value at either boundary (i.e., at $y = 0$ or $y = B$) and decreases linearly with the distance from the boundary, with the result that it is equal to zero at the centre line between the two plates $\left(\text{i.e., at } y = \frac{B}{2}\right)$. The shear stress distribution is shown in Fig. 13.6. If τ_0 represents the magnitude of the boundary shear stress, then from Eq. 13.42, for $y = 0$, we have

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2} \quad ...(13.43)$$

It may be noted from Eq. 13.43 that the boundary shear stress is directly proportional to the pressure gradient and the boundary spacing.

If the plates are inclined, gravity forces will also become effective. This effect can be accounted for by considering the piezometric head $h\left(=\frac{p}{w}+z\right)$ instead of pressure head. The various expressions relevant to this case may thus be obtained by inserting $\left(w\frac{\partial h}{\partial x}\right)$ in place of $\left(\frac{\partial p}{\partial x}\right)$ in the various expressions derived for the flow between parallel flat plates placed horizontally.

13.7 LAMINAR FLOW BETWEEN PARALLEL FLAT PLATES—ONE PLATE MOVING AND OTHER AT REST—COUETTE FLOW

Consider laminar flow between two parallel flat plates located at a distance B apart, such that the lower plate is at rest and the upper plate moves uniformly in its own plane with a velocity V as shown in Fig. 13.7.

The flow of this type is known as Couette flow after M.F.A. Couette. The general expression for the velocity distribution for laminar flow between two parallel flat plates as derived earlier [Eq. 13.34] is also applicable for Couette flow. Thus

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2 \quad \text{... (13.44)}$$

with the boundary conditions

$$v = 0 \text{ at } y = 0$$

$$\therefore \quad C_2 = 0$$

and $$v = V \text{ at } y = B$$

$$\therefore \quad C_1 = \frac{V}{B} - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)B$$

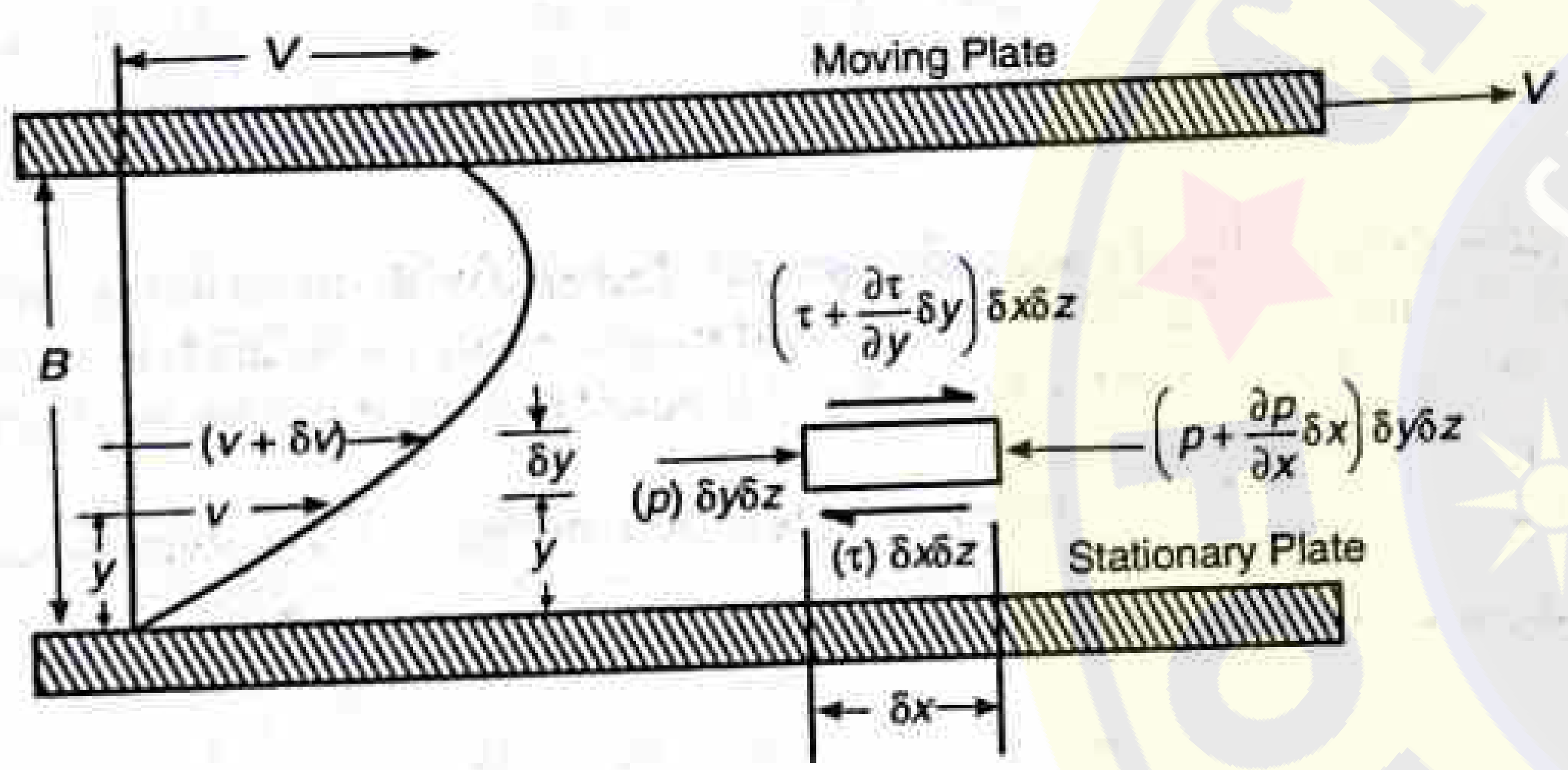


Figure 13.7 *Couette flow*

Hence, substituting the values of C_1 and C_2 in Eq. 13.34, it yields the following equation for the velocity distribution for Couette flow

$$v = \frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2) \quad \text{...(13.44)}$$

It is thus observed that the velocity distribution in Couette flow depends on both V and $\left(\frac{\partial p}{\partial x}\right)$.

However, the pressure gradient $\left(\frac{\partial p}{\partial x}\right)$ in this case may be either positive or negative.

In particular case when $\left(\frac{\partial p}{\partial x}\right)$ equals zero, there is no pressure gradient in the direction of flow, then, we have

$$v = V\frac{y}{B} \quad \text{...(13.45)}$$

which shows that the velocity distribution is linear. This particular case is known as *simple (or plain) Couette flow* or *simple shear flow*. Now, if V equals zero, Eq. 13.44 reduces to Eq. 13.35. Thus it may be stated that the general case of Couette flow is a superposition of the simple Couette flow over a laminar flow between two fixed parallel flat plates.

The distribution of shear stress across any section for the case of Couette flow may also be determined by substituting Eq. 13.44 into Newton's law of viscosity. Thus

$$\tau = \mu\frac{\partial v}{\partial y}$$

or $$\tau = \mu\frac{\partial}{\partial y}\left[\frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)\right]$$

or $$\tau = \mu\frac{V}{B} + \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right) \quad \text{...(13.46)}$$

Evidently in this case also the shear stress varies linearly with the distance from the boundary. However, as shown in Fig. 13.8, in this case the shear stress distribution at any section is asymmetrical with the values as given below.

At $y = 0$

$$\tau = \tau_{0_1} = \left[\frac{\mu V}{B} + \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2}\right]$$

At $y = \frac{B}{2}$

$$\tau = \frac{\mu V}{B}$$

At $$y = \left[\frac{B}{2} + \frac{\mu V}{B}\left(-\frac{1}{\partial p/\partial x}\right)\right]$$

$$\tau = 0$$

At $y = B$

$$\tau = \tau_{0_2} = \left[\frac{\mu V}{B} - \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2}\right]$$

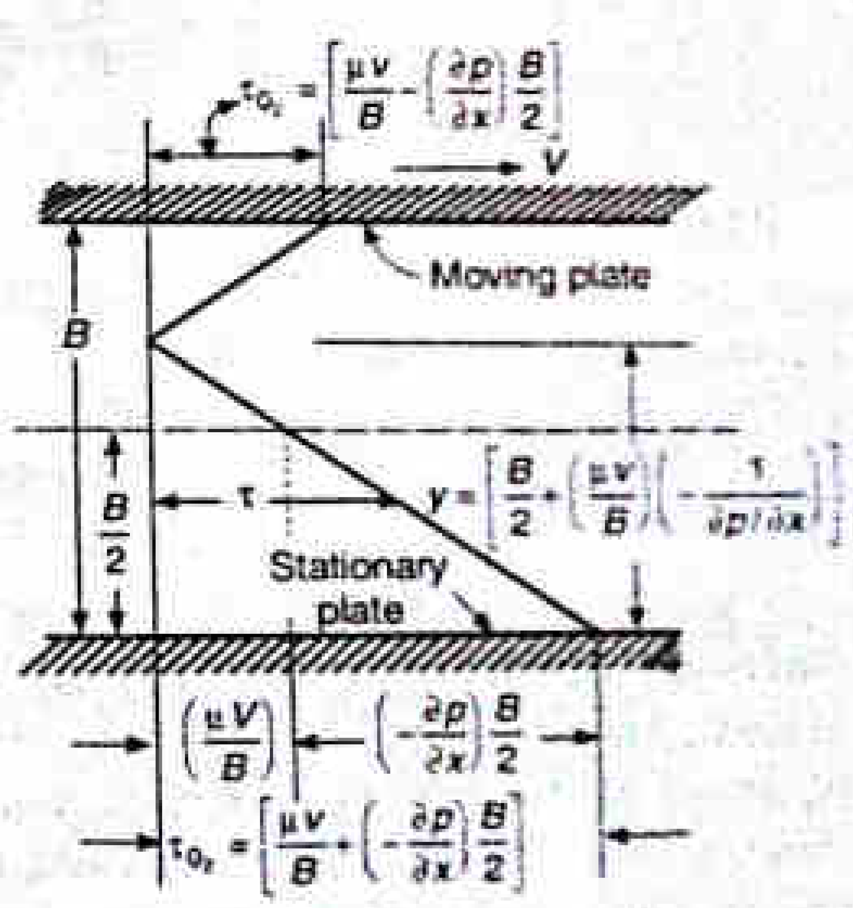


Figure 13.8 *Shear stress distribution for Couette flow*

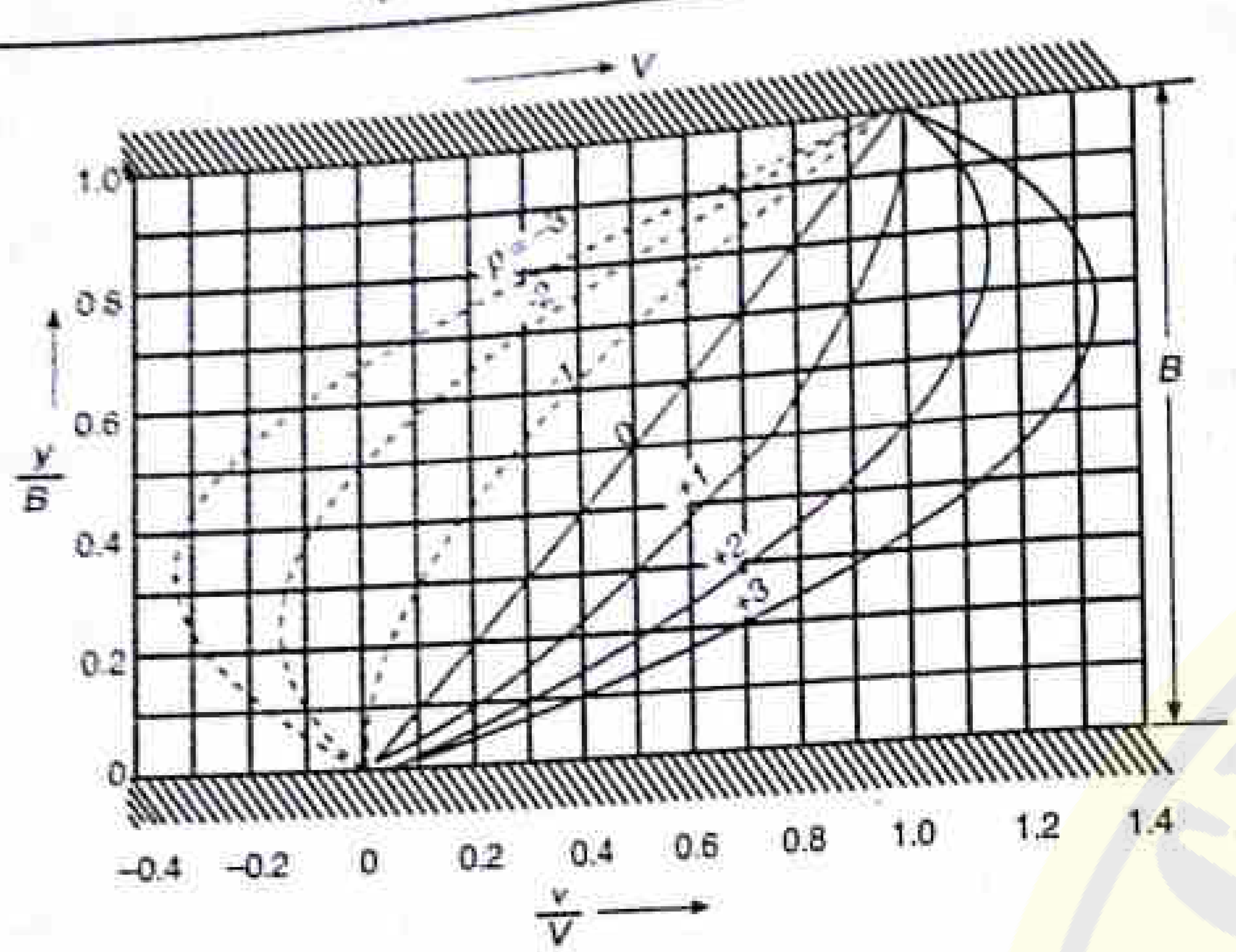


Figure 13.9 *Non-dimensional velocity distribution curves for Couette flow*

The velocity distribution for a general Couette flow can be transformed to the following non-dimensional form.

$$\frac{v}{V} = \frac{y}{B} + \frac{B^2}{2\mu V}\left(-\frac{\partial p}{\partial x}\right)\left(1-\frac{y}{B}\right)\frac{y}{B} \qquad \text{... (13.47)}$$

If the constant term $\left[\frac{B^2}{2\mu V}\left(-\frac{\partial p}{\partial x}\right)\right]$ is replaced by P, called dimensionless pressure gradient, then the above expression becomes

$$\frac{v}{V} = \frac{y}{B} + P\left(1-\frac{y}{B}\right)\frac{y}{B} \qquad \text{... (13.48)}$$

A family of velocity distribution curves can be plotted in terms of (v/V) and (y/B) for different values of P. These are shown in Fig. 13.9. Positive values of P indicate a pressure drop in the direction of flow and negative values of P indicate an increase in fluid pressure in the direction of flow. It will be observed that for $P< (-1)$, for some layers of fluid there exists a backward flow, that is, opposite to the direction of V. This is on account of an adverse pressure gradient in the fluid. Further it may be noted that for a given value of P the maximum value of (v/V) occurs at a point where the shear stress is equal to zero, i.e., at a distance $y = \left[\frac{B}{2} + \frac{\mu V}{B}\left(-\frac{1}{\partial p/\partial x}\right)\right]$ above the lower plate.

The Couette flow with a pressure gradient has its application to the hydrodynamic theory of lubrication. The flow of lubricating oil in the narrow space in between journal and bearing is somewhat identical with Couette flow with a pressure gradient, which has been discussed in Section 13.11.

13.8 LAMINAR FLOW OF FLUID IN AN OPEN CHANNEL

Consider steady, uniform, laminar flow in a very wide open channel. Let D be the depth of flow and $S_0 = (-\partial z/\partial x) = (\sin\theta)$, be the slope of the bed of the channel. In this case there is flow with a free surface, the pressure at every section will be same and therefore no pressure gradient is available to maintain the flow. However, in such a case the flow is maintained by the drop in the potential energy, i.e., the resistance to flow will be compensated by the gravity force. The required drop in potential energy will be provided by the bed slope of the channel. Equation 13.2 can be employed to analyse this case also, but the term $(\partial p/\partial x)$ should be replaced by $w(\partial z/\partial x)$, where w is the specific weight of the fluid. Thus for this case

$$w\frac{\partial z}{\partial x} = \mu\frac{\partial^2 v}{\partial y^2}$$

Again since the flow is uniform $(\partial z/\partial x)$ is independent of y. Thus integrating the above equation with respect to y, we get

$$\frac{\partial v}{\partial y} = \frac{w}{\mu}\left(\frac{\partial z}{\partial x}\right)y + C_1 \qquad \text{...(13.49)}$$

The constant of integration C_1 may be evaluated by applying the boundary condition that at the free surface the velocity is maximum. Thus for $y = D$, $(\partial v/\partial y) = 0$

$$C_1 = -\frac{w}{\mu}\left(\frac{\partial z}{\partial x}\right)D$$

Introducing the value of C_1 in the above expression it becomes

$$\frac{\partial v}{\partial y} = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)(D-y) \qquad \text{... (13.50)}$$

Again integrating it with respect to y the expression for the velocity distribution for laminar flow in an open channel is obtained as

$$v = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy-\frac{y^2}{2}\right) + C_2 \qquad \text{... (13.51)}$$

The second constant of integration C_2 may be evaluated by applying the boundary condition that the velocity of flow is zero at the bed of the channel. Thus at $y = 0$, $v = 0$, therefore, $C_2 = 0$.

The velocity distribution then becomes

$$v = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy-\frac{y^2}{2}\right) \qquad \text{... (13.52)}$$

This is a quadratic equation indicating that the velocity of uniform laminar flow in a wide open channel has a parabolic distribution. Integrating this equation from $y = 0$ to $y = D$ and dividing the result by D, the average velocity is obtained as

$$V = \frac{1}{D}\int_0^D v\,dy = \frac{wS_0D^2}{3\mu} = \frac{gS_0D^2}{3\upsilon}$$

and the discharge per unit width is obtained as

$$q = \frac{wS_0D^3}{3\mu} = C_1D^3 \qquad \text{...(13.53)}$$

where $C_1 = (wS_0/3\mu)$, a coefficient involving slope and viscosity.

Rearranging the above expression for the potential head gradient, we get

$$S_0 = \left(-\frac{\partial z}{\partial x}\right) = \frac{3\mu V}{wD^2} \qquad \text{...(13.54)}$$

Again since the flow is steady and uniform, the above expression can be integrated with respect to x to provide the potential head drop $(z_1 - z_2)$ between any two sections 1 and 2 in the direction of flow at a distance L apart. Thus

$$(z_1 - z_2) = \frac{3\mu VL}{wD^2} \qquad \text{...(13.55)}$$

If h_f represents the loss of head in between the sections 1 and 2 then since $h_f = (z_1 - z_2)$ and hence

$$h_f = \frac{3\mu VL}{wD^2} \qquad \text{...(13.55 a)}$$

The distribution of shear stress in the flowing fluid across any section of the channel may be determined by substituting the value of $(\partial v/\partial y)$ in Newton's law of viscosity. Thus we obtain

$$\tau = w\left(-\frac{\partial z}{\partial x}\right)(D-y) = wS_0(D-y) \qquad \text{...(13.56)}$$

From this expression it is evident that the shear stress varies linearly with distance from the boundary, having a maximum value at the bed and a minimum value of zero at the free surface. The value of the bed shear stress τ_0 is obtained by substituting $y = 0$ in the above expression. Thus we obtain

$$\tau_0 = wS_0D \qquad \text{...(13.57)}$$

13.9 LAMINAR FLOW THROUGH POROUS MEDIA

Another phenomenon that exhibits laminar flow is that of flow of water or any other liquid through porous materials which they saturate. Consider a circular pipe of length L and cross-sectional area A completely filled with porous material of grain diameter d, as shown in Fig. 13.10. Let Q be the discharge of liquid of specific weight w, flowing through the pipe. The flow will take place through the interstices or pores of the porous material which form numerous little conduits. Since it is not possible to analyse the flow through such individual conduit the porous material as a whole is considered in which the passage for liquid is bounded by the impervious solid sand grains. Thus the apparent (or superficial) mean velocity V is equal to (Q/A), but this will not be the actual mean velocity of flow of liquid. This is because part of the cross-sectional area of pipe is blocked by the solid grains of the porous material and the liquid flows only through the pores. Thus if η is the effective ratio of the area of pores A_p to the total area A [i.e., $\eta = (A_p/A)$], then the actual mean velocity of flow of liquid through the pores is given by (Q/A_p) which is same as $(Q/\eta A)$ or (V/η). Since the area of the pores A_p is smaller than the total area A, the actual mean velocity of flow through the pores will be greater than the apparent mean velocity V. However, the actual mean velocity as well as the pore size being quite small

the flow will be laminar, and therefore, the loss of head when liquid flows through a porous media can be determined by using the general expression for head loss in laminar flows as indicated below

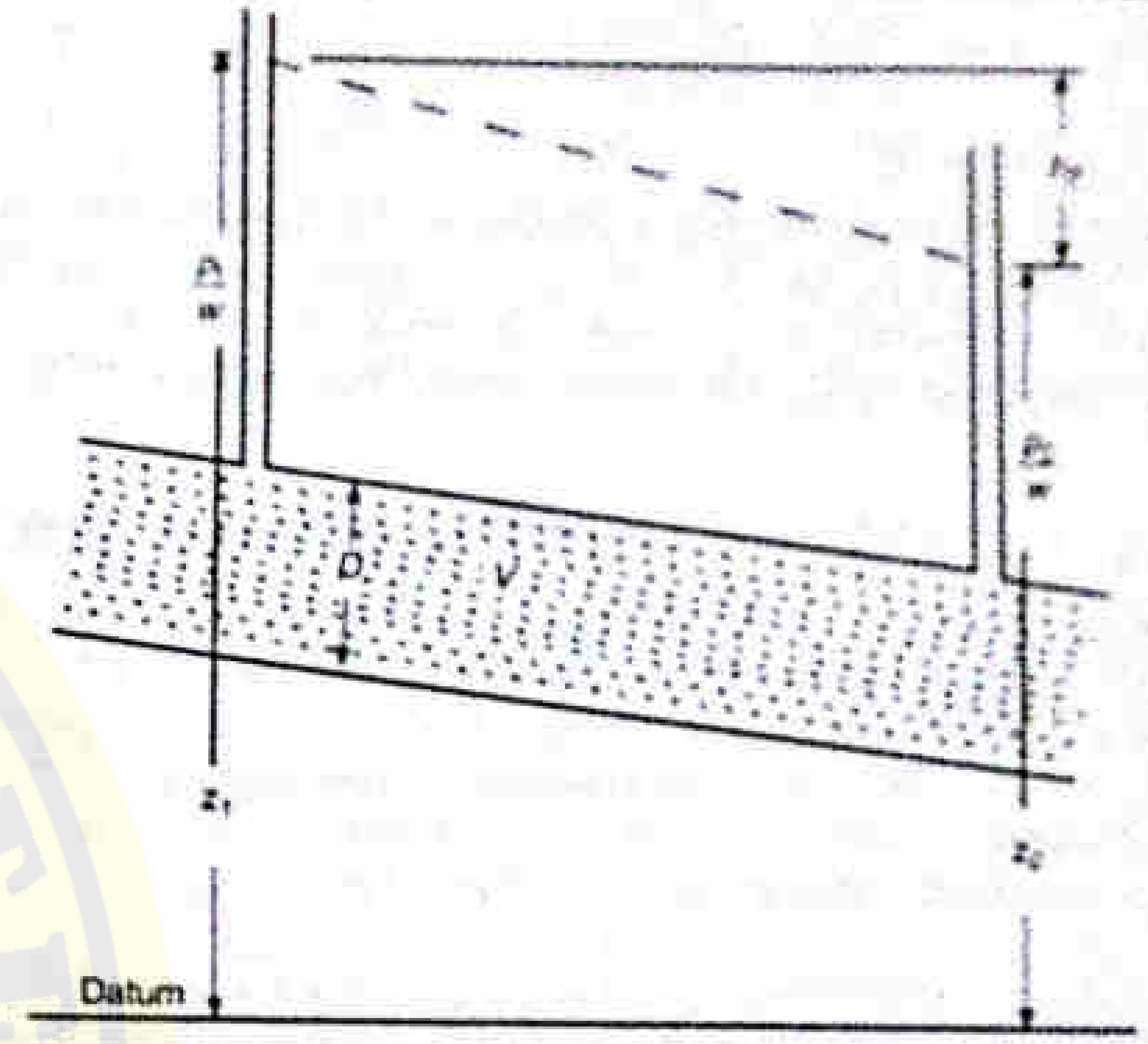


Figure 13.10 *Flow through porous media*

As derived in Section 13.3 the loss of head for laminar flow through a pipe is given by Eq. 13.18 as

$$h_f = \frac{32\mu VL}{wD^2}$$

Similarly the loss of head for laminar flow through parallel plates is given by Eq. 13.41 as

$$h_f = \frac{12\mu VL}{wB^2}$$

Hence the general expression for the loss of head for laminar flow through a passage of any shape may be expressed as

$$h_f = \frac{k\mu VL}{wD^2} \qquad \text{... (13.58)}$$

where h_f is the loss of head in length L, V is the mean velocity of flow, D is a characteristic length representing the geometry of the passage, and k is a constant the value of which depends on the shape of the passage.

In the case of flow through porous media the average size of the pores may be taken equal to the grain diameter d_s. Thus by substituting (V/η) for V and d_s for D in Eq. 13.58, the loss of head for laminar flow through a porous media is given by

$$h_f = \frac{k\mu VL}{w\eta d_s^2} \qquad \text{... (13.59)}$$

which can be rearranged for velocity V as

$$V = \left(\frac{w\eta d_s^2}{k\mu}\right)\left(\frac{h_f}{L}\right) \qquad \ldots (13.60)$$

or

$$V = Ki$$

where i is the hydraulic gradient and K is known as coefficient of permeability which has the dimensions of velocity. Equation 13.60 expresses Darcy's Law in its simplest form, which was first discovered experimentally by Henri Darcy, a French hydraulic engineer in 1856. Darcy's Law has been found to be quite useful in the analysis of various problems which invlove flow of liquid through porous media.

13.10 LAMINAR FLOW AROUND A SPHERE–STOKES' LAW

Laminar flow may also exist around a solid body which moves through a fluid of infinite extent, in such a manner that viscosity is the only fluid property affecting the pattern of motion and the accelerative effects are so small that they may be neglected. The practical examples in this context may be that of a dust particle falling in the air or a fine sand grain falling in water. When a body moves through any fluid, it experiences a resistance which acts in the direction opposite to that of the motion of the body. The resistance called *drag*, depends on the size of the object, the velocity with which it moves and the viscosity of the fluid.

In 1815 G.G. Stokes developed analytically an expression for the resistance F_D experienced by a sphere of diameter D, moving with a constant velocity V (i.e., there is no acceleration) in a fluid of viscosity μ, which is given by

$$F_D = 3\pi\mu VD \qquad \ldots (13.61)$$

It is known as Stokes' Law which has been verified experimentally and is found to hold good for values of Reynolds number (VD/ν) less than 0.2.

If the sphere is dropping down through the fluid under its own weight at a constant velocity V, then the buoyant force plus the resistance to its motion must be equal to its weight. Thus we obtain

$$\left(\frac{\pi}{6}D^3\right)w + F_D = \left(\frac{\pi}{6}D^3\right)w_s$$

or

$$F_D = \left(\frac{\pi}{6}D^3\right)(w_s - w)$$

in which w is specific weight of the fluid, w_s is specific weight of the sphere and $\left(\frac{\pi}{6}D^3\right)(w_s - w)$ represents the submerged weight of the sphere. Introducing the value of F_D from Eq. 13.61 in the above expression and solving for V, we obtain

$$V = \frac{D^2}{18\mu}(w_s - w) \qquad \ldots (13.62)$$

The velocity V given by Eq. 13.62 is called terminal fall velocity, which is defined as the velocity attained by a body in falling through a fluid at rest, when the drag on the body is equal to the submerged weight of the body. As indicated later, the viscosity μ of a fluid can be determined

experimentally, if in Eq. 13.62 the fall velocity V, of a sphere of known diameter D, moving down through the fluid is measured.

The drag F_D for any body moving with velocity V in a fluid of mass density ρ may be expressed as

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

in which C_D is known as coefficient of drag, and A is the projected area of the body perpendicular to the direction of flow. For the sphere $F_D = 3\pi\mu VD$ and $A = (\pi D^2/4)$; thus by substituting for F_D and A in the above equation, we get

$$C_D = 24\left(\frac{\mu}{\rho VD}\right) = \frac{24}{\text{Re}} \qquad \ldots (13.63)$$

Equation 13.63 represents the drag coefficient* for sphere on the basis of Stokes' Law. Experiments have shown that Eq. 13.63 holds good for Reynolds number less than 0.2 provided the sphere is moving in an infinite fluid. However, if the fluid through which the sphere moves is not infinite in extent but is confined within a container of finite dimensions, then the resistance to motion is increased and in such a case the drag coefficient is given by the following modified expression

$$C_D = \frac{24}{\text{Re}}\left[1 + 2.1\frac{D}{D_1}\right]$$

where D_1 is the smallest lateral dimension of the container and D is the diameter of the sphere.

13.11 LUBRICATION MECHANICS

One of the most important applications of laminar flow of viscous fluids arises in the lubrication of various types of bearings. The theory of lubrication was initiated by Reynolds who showed that *two parallel or nearly parallel surfaces can slide one over the other with but slight frictional resistance even under great normal pressure, provided a film of viscous fluid is maintained between them.* This is the working principle of various bearings with lubricating oil as viscous fluid. Although high velocities may be involved in the lubricating systems, the thickness of the film of the lubricant is so small that the Reynolds number is far below its critical value for that system. Therefore, laminar motion may always be assumed in the analysis of lubrication problems.

The primary function of the lubricant is to separate the bearing surfaces so that there is no direct contact between properly finished surfaces of two machine elements which are in relative motion. Further, if the film of lubricant is to keep the bearing surfaces apart when they are in relative motion it must be capable of sustaining a load, which is made possible by the pressure developed in the oil film. However, as indicated later the bearings can support a load only if the surfaces of the relatively moving parts are inclined to one another.

Two common types of bearings, viz, *slipper bearing* and *journal bearing* are described below.

13.11.1 Slipper bearing

In the simplest form a slipper bearing consists of a slipper or slide block moving over a horizontal bearing plate as illustrated in Fig. 13.11. The plane surface of the slipper is kept slightly inclined over

* For definition of drag coefficient see Chapter 18.

the plane surface of the bearing plate which is horizontal. For the purpose of simple analysis it is assumed that the bearing plate is infinite in extent and the slipper has infinite width in the direction

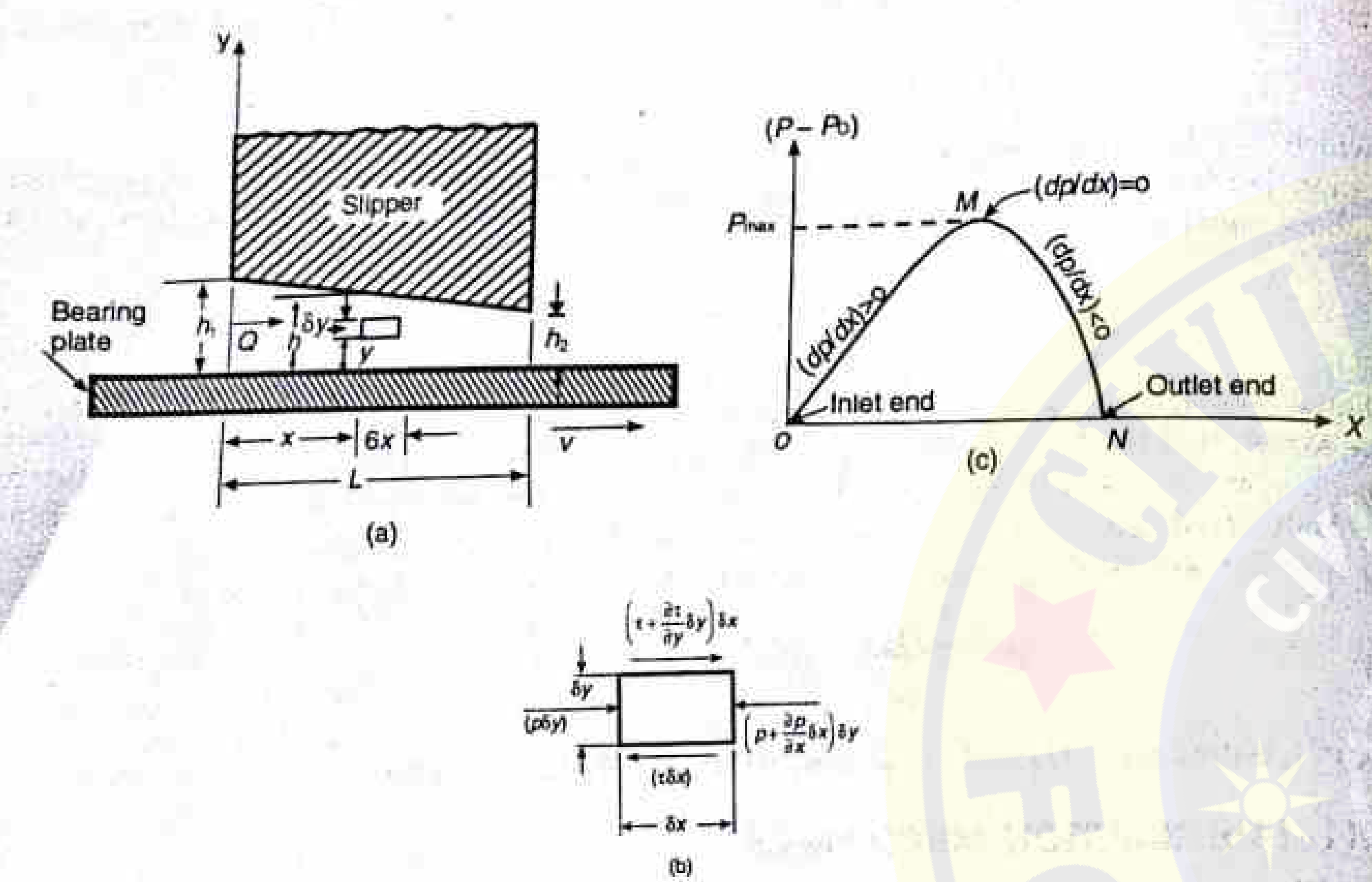


Figure 13.11 *Slipper bearing–a simplified sketch*

perpendicular to the plane of the paper, so that the flow of the lubricant may be considered two dimensional.

Actually the slipper or slide block is moving at a constant velocity V, but in order to simplify the analysis the principle of relative motion is applied to the entire system to bring the slipper at rest while the bearing plate moves past it with a constant velocity V in the x-direction. The behaviour of the lubricant in this case, therfore, corresponds to laminar flow between parallel plates with one plate moving, i.e., Couette flow as discussed in Section 13.7. However, to recapitulate the detailed analysis is given here again.

Consider a rectangular element of fluid situated at (x, y) as shown in Fig. 13.11 (a). This element is shown enlarged in Fig. 13.11 (b) together with the pressure and shear forces acting on its faces. For steady and uniform flow, since there is no acceleration, the sum of all the forces acting on the fluid element must be equal to zero. Hence for unit width of the element in the direction perpendicular to the paper,

$$p\delta y - \left(p+\frac{\partial p}{\partial x}\delta x\right)\delta y + \left(\tau+\frac{\partial \tau}{\partial y}\delta y\right)\delta x - \tau\delta x = 0.$$

or $$\frac{\partial p}{\partial x}\delta x\,\delta y = \frac{\partial \tau}{\partial y}\delta x\,\delta y$$

or $$\frac{\partial p}{\partial x} = \frac{\partial \tau}{\partial y}$$

Since according to Newton's law of viscosity $\tau = \mu(\partial v/\partial y)$, the above expression becomes

$$\frac{\partial p}{\partial x} = \mu\frac{\partial^2 v}{\partial y^2} \quad ...(13.63\ a)$$

Again since $(\partial p/\partial x)$ is independent of y, integrating Eq. 13.63 (a) twice with respect to y gives

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2}+C_1 y+C_2 \quad ...(13.64)$$

which is same as Eq. 13.34 derived earlier. Equation 13.64 is a general equation which may be applied to this case as well. The constants of integration C_1 and C_2 can be evaluated by applying the following boundary conditions:

at $y = 0, v = V$ $\quad \therefore\ C_2 = V$

at $y = h,\ v = 0$ $\quad \therefore\ C_1 = -\frac{V}{h} - \frac{h}{2\mu}\frac{\partial p}{\partial x}$

Thus introducing the values of constants in Eq. 13.64 it becomes

$$v = V\left(1-\frac{y}{h}\right) + \frac{h^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left(\frac{y}{h}\right)\left(\frac{y}{h}-1\right) \quad ...(13.65)$$

By comparison it may be seen that Eq. 13.65 is exactly same as Eq. 13.44 for the general case of Couette flow.

Considering the flow through an elementary strip of unit breadth and thickness dy

$$dQ = v\,dy$$

$$\therefore \quad Q = \int_0^h v\,dy = \text{constant}$$

$$= \int_0^h \left[V\left(1-\frac{y}{h}\right)+\frac{h^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left(\frac{y}{h}\right)\left(\frac{y}{h}-1\right)\right]dy$$

$$= \frac{Vh}{2} - \frac{1}{12\mu}\frac{\partial p}{\partial x}h^3$$

Solving for $(\partial p/\partial x)$, we obtain

$$\frac{\partial p}{\partial x} = \frac{12\mu}{h^3}\left(\frac{Vh}{2}-Q\right) \quad ...(13.66)$$

From Fig. 13.10 it is seen that

$$h = h_1 - \frac{x}{L}(h_1 - h_2) = (h_1 - mx)$$

where $$m = \left(\frac{h_1 - h_2}{L}\right)$$

By substituting the value of h in Eq. 13.66, we obtain

$$\frac{\partial p}{\partial x} = 12\mu\left[\frac{V}{2(h_1 - mx)^2} - \frac{Q}{(h_1 - mx)^3}\right] \quad ...(13.67)$$

Integarating the above expression the pressure distribution along the slipper is given by

$$p = 12\mu\left[\frac{V}{2m(h_1 - mx)} - \frac{Q}{2m(h_1 - mx)^2}\right] + C_3$$

The pressure intensities at the two ends of the slipper are known since atmospheric pressure prevails at the ends, i.e., at $x = 0$, $p = 0$ and $x = L$, $p = 0$. Thus by substitution

$$C_3 = -\frac{6\mu V}{mh_1} + \frac{6\mu Q}{mh_1^2}$$

and $$Q = \frac{Vh_1h_2}{h_1 + h_2} \quad ...(13.68)$$

By substituting the values of C_3 and Q the pressure distribution along the slipper is given by

$$p = \frac{6\mu Vx(L-x)(h_1 - h_2)}{Lh^2(h_1 + h_2)} \quad ...(13.69)$$

The variation of pressure along the slipper as represented by Eq. 13.69 is shown in Fig. 13.11. Further by equating $(\partial p/\partial x)$ to zero it can be readily shown that the maximum pressure intensity p_{max} occurs at

$$x = \frac{h_1(h_1 - h_2)}{m(h_1 + h_2)} = \frac{Lh_1}{(h_1 + h_2)} \quad ...(13.70)$$

Introducing Eq. 13.70 in Eq. 13.69 the maximum pressure intensity is obtained as

$$p_{max} = \frac{3\mu V(h_1 - h_2)L}{2h_1h_2(h_1 + h_2)} \quad ...(13.71)$$

It is on account of the pressure developed in the oil film that the bearing supports the load. As shown in Fig. 13.11 the pressure increases from the zero (or atmospheric) pressure at the inlet end O to a maximum value at some point M, and then decreases again to zero (or atmospheric) at the outlet end N. The slope of the pressure distribution $(\partial p/\partial x)$ must therefore change from positive between O and M to negative between M and N.

From Eq. 13.66 it will be clear that if $(\partial p/\partial x)$ has to change from positive value to negative, $\left(\frac{Vh}{2} - Q\right)$ must change from positive between O and M to negative between M and N. Since Q is constant at all sections along the oil film it follows that the essential conditions for the development of pressure in the oil film are

(i) between O and M, $h > (2Q/V)$

(ii) at M, $h = (2Q/V)$

and (iii) between M and N, $h < (2Q/V)$

It, therefore, follows that for a bearing to sustain a load the thickness of the oil film h must decrease in the direction of flow of the lubricant.

The total force on the bearing surface per unit length can be obtained by integrating the pressure intensity over the width L.

Thus $$P = \int_0^L p\,dx = \int_0^L \frac{6\mu Vx(L-x)(h_1 - h_2)}{L(h_1 + h_2)(h_1 - mx)^2}\,dx.$$

$$\therefore \quad P = \frac{6\mu VL^2}{(h_1 - h_2)^2}\left[\log_e(h_1/h_2) - \frac{2(h_1 - h_2)}{(h_1 + h_2)}\right] \quad ...(13.72)$$

It is evident from this expression that the total load P will be zero when $h_1 = h_2$, that is when the two surfaces are parallel. *In other words the moving parts of the bearing will sustain load only if the surfaces of the relatively moving parts are inclined to one another.*

The maximum load P_{max} is computed with Eq. 13.72 when $h_1 = 2.2\,h_2$ for which

$$P_{max} = \frac{0.16\,\mu VL^2}{h_2^2} \quad ...(13.73)$$

In the above analysis it has been assumed that the bearing plate is of infinite extent and the slipper has infinite width, but the real bearings used in actual practice have a finite width. As such for real bearings the flow is three-dimensional. Michell has investigated the three-dimensional flow problem for bearing pads of finite width. In such cases the oil flows not only in the direction of motion of the bearing but also towards the sides, thus causing a decrease in the load or bearing capacity. According to Michell's analysis the load on the bearing of finite width B is reduced according to the following table:

B/L	∞	1	1/3
Load	P	$0.422P$	$0.031\,P$

In the above table the total load P supported by the slipper bearing is as given by Eq. 13.72.

13.11.2 Journal Bearing

As shown in Fig. 13.12 a journal bearing consists of a sleeve which is wrapped partially or completely around a rotating shaft or journal. It is designed to support a radial load. It is possible to design a journal bearing to operate with its rubbing surfaces separated by lubricant film and still carry a substantial load. The journal bearing adjusts itself automatically to these conditions as discussed below.

When the loaded journal is at rest, there is a metal to metal contact at the lowest point as shown in Fig. 13.12 (a). As the journal begins to rotate say in clockwise direction it moves up the bearing in an anticlockwise direction as shown in Fig. 13.12 (b). The lubricant then adheres to the surface of the journal which now rolls on a lubricated surface. When the speed of journal is sufficiently high, the

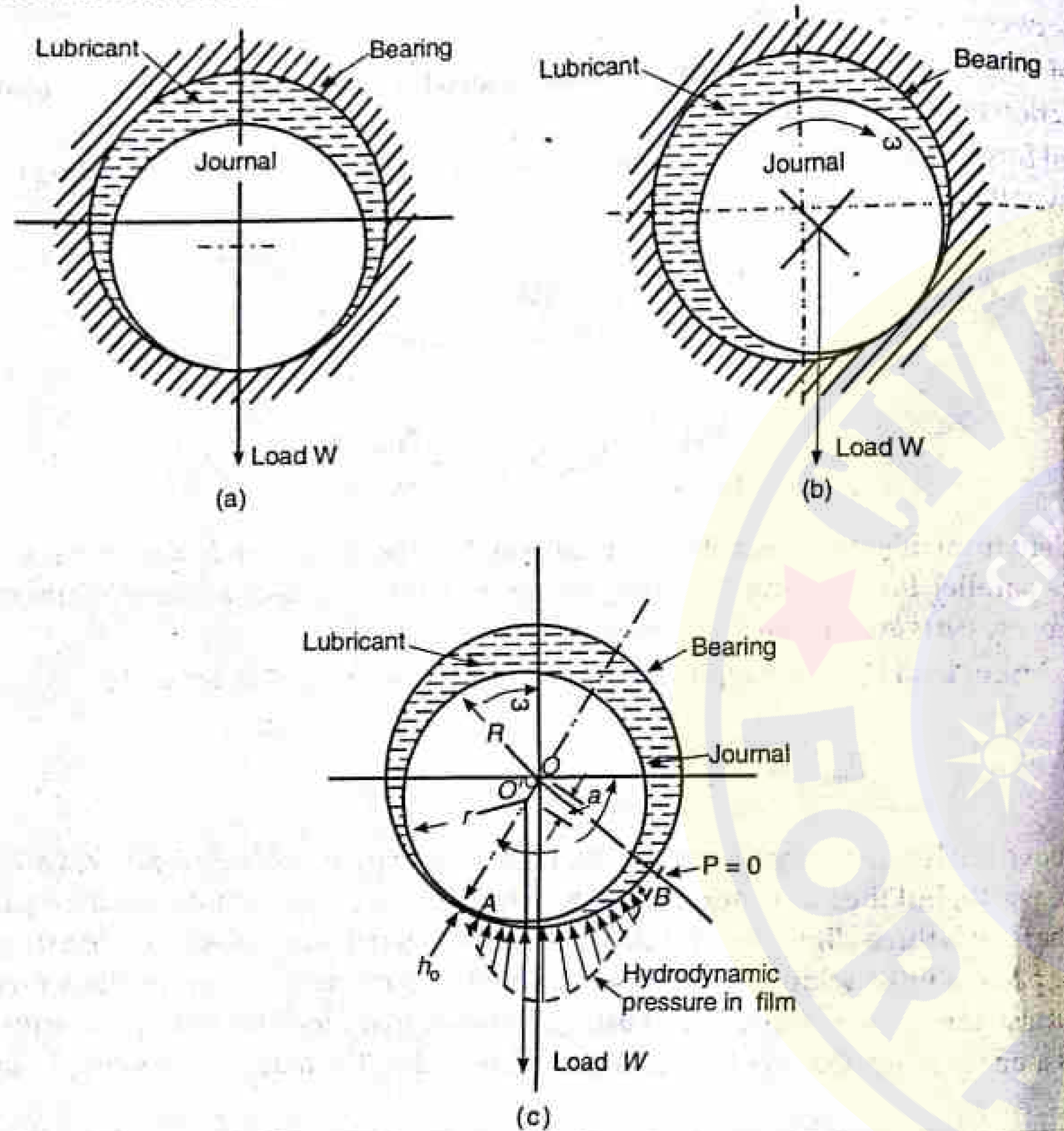


Figure 13.12 *Journal bearing*

lubricant will automatically be drawn between the surfaces, thus floating the journal in the bearing and the centre of the journal moves to the left of the centre of the bearing as shown in Fig. 13.12 (c). Once the condition shown in Fig. 13.12 (c) is reached, the convergent film of lubricant gets established and a positive pressure is developed which can support the load.

A polar diagram of film pressures is shown representing the extent of the arc of actual load carrying lubricant film. The leading edge of the film may be established by an oil distribution groove such as at *B* in Fig. 13.12 (c). The trailing edge of the load carrying film is considered as terminating at the point of closest approach between the shaft (or journal) and bearing. Beyond this point the clearance space is no longer converging, but begins to diverge. The film thickness at the point of closest approach is designated as h_o. Point *O* represents the centre of the bearing and *O'* the centre of the shaft. The radius of the shaft is

r and the radius of the bearing is *R*. Distance *OO'* is indicated by *a* and it is the eccentric distance. The ratio of *a* for the actual operating conditions to the radial clearance in the bearing is called the eccentricity ratio ε. For light loads and high operating speeds the centre of the shaft and the bearing tend to coincide and *a* approaches zero. Thus ε also approaches zero. For heavy loads or extremes of operation such as low speed or very low viscosity the journal becomes more eccentric in the bearing and the distance *a* increases. The limit is where the journal just begins to make the metallic contact with the bearing or the distance *a* equals the radial clearance. The value of the eccentricity ratio ε is then 1.

Although the lubricant film in this case is curved, the same equations as derived for the case of slipper bearing can be applied for the journal bearing also, but with proper modifications as noted below.

The radial clearance in the bearing is *c*, which equals *mr*, where *m* is the clearance modulus of the bearing in *mm/mm*. The value of *m* usually range between 0.0005 and 0.003, with typical values for industrial type bearing running from 0.001 to 0.0025.

The value of ∂x in Eq. 13.66 is replaced by $r\partial\theta$ (Fig. 13.12c) also

$$h_0 + a = c = mr$$

but $$a = \varepsilon mr$$

so $$h_0 = mr(1-\varepsilon)$$

Also $$h + a\cos\theta = c = mr$$

so $$h = mr(1-\varepsilon\cos\theta).$$

Thus substituting these values in Eq. 13.66 it becomes

$$\frac{\partial p}{r\partial\theta} = 6\mu V\left[\frac{1}{m^2r^2(1-\varepsilon\cos\theta)^2} - \frac{2Q}{Vm^3r^3(1-\varepsilon\cos\theta)^3}\right]$$

or $$\frac{\partial p}{\partial\theta} = \frac{6\mu Vr}{c^2}\left[\frac{1}{(1-\varepsilon\cos\theta)^2} - \frac{2Q}{Vc(1-\varepsilon\cos\theta)^3}\right] \qquad ...(13.74)$$

Integrating the above equation, we obtain

$$p = \frac{6\mu Vr}{c^2}\left[\frac{\varepsilon(2-\varepsilon\cos\theta)\sin\theta}{(2+\varepsilon^2)(1-\varepsilon\cos\theta)^2}\right] \qquad ...(13.75)$$

The above expression has been obtained by using the boundary conditions that $p = 0$ at $\theta = 0$ and the term $(2Q/V)$ has been determined by the condition that the pressure returns to its original value when θ has increased by 2π. Further in the above expressions $V = r\omega = (2\pi Nr/60)$, where *N* is the speed of the journal in rev./min.

To obtain the total load that can be carried by the bearing when it has an eccentricity ratio ε we must integrate the pressure over the convergent portion of the oil, that is between points *A* and *B*. This is because in the crown of the bearing a negative pressure will be established which will contribute very little to the load carrying function of the bearing, since the positive oil-film pressures are generally much larger than these negative pressures.

13.11.3 Properties of Lubricant

Oils and greases are the most widely used lubricants. Viscosity is the most important single property involved in lubrication in engineering applications. A detailed discussion of viscosity of fluids is iven in Chapter 1. It is known that viscosity μ of common lubricating oils, like that of most liquids, alls rapidly with rising temperature. The degree to which the viscosity depends on temperature is called *viscosity index* (V.I.) of the oil. Oils which show the smallest change with temperature are assigned a higher value of V.I. say 100 and those which show the greatest change have a V.I. of zero. In between these two extreme categories of oils, there are several other oils with varying values of V.I. A lubricant with high V.I. is always desirable, but it is especially necessary for the systems which are exposed to extreme ranges of temperature.

In addition to viscosity there are some other properties of lubricants which are also required to be considered to judge the suitability of any particular lubricant. However, a detailed discussion of these may be seen in any of the books on lubricants.

13.12 DASH-POT MECHANISM

A dash-pot is a device used for damping vibrations of machines, which may be achieved by making use of a fluid of fairly high viscosity. The simplest kind of a dash-pot is shown in Fig. 13.13. It consists of a piston which may move in a concentric cylinder, the diameter of which is only slightly greater than that of the piston. The cylinder contains a viscous oil, the quantity of which should be sufficient to cover the top of the piston. The piston is connected to the machine component whose movement is to be restrained. If the piston is caused to move downwards, oil is displaced from underneath it. This displaced oil must move to the space above the piston and its only route is through the small annular clearance between the piston and the cylinder. If the oil is highly viscous and the clearance between the piston and the cylinder is small the flow of oil upwards through the clearance space is laminar, and it occurs simply as a result of the pressure developed underneath the piston. Obviously, more viscous is the oil and smaller is the clearance between the piston and the cylinder, the greater is the pressure required to produce a particular movement of oil, thus greater is the force required to move the piston or greater is the resistance to its motion.

Similarly for an upward movement of the piston, oil must flow downwards through the clearance space, since an upward movement of the piston reduces the pressure underneath it and the resulting difference of pressure betwen the top and bottom of the piston causes the downward flow of oil. Consider a force F applied to the piston which causes it to move downwards and, therefore, the oil flow is upward through the clearance space. If the clearance space is small, then the piston and cylinder may be considered as two flat plates of breadth equal to the circumferential length of the piston πD, length L and distance apart equal to the clearance c, through which the oil flows. Thus Eq. 13.34 may be applied to this case also according to which

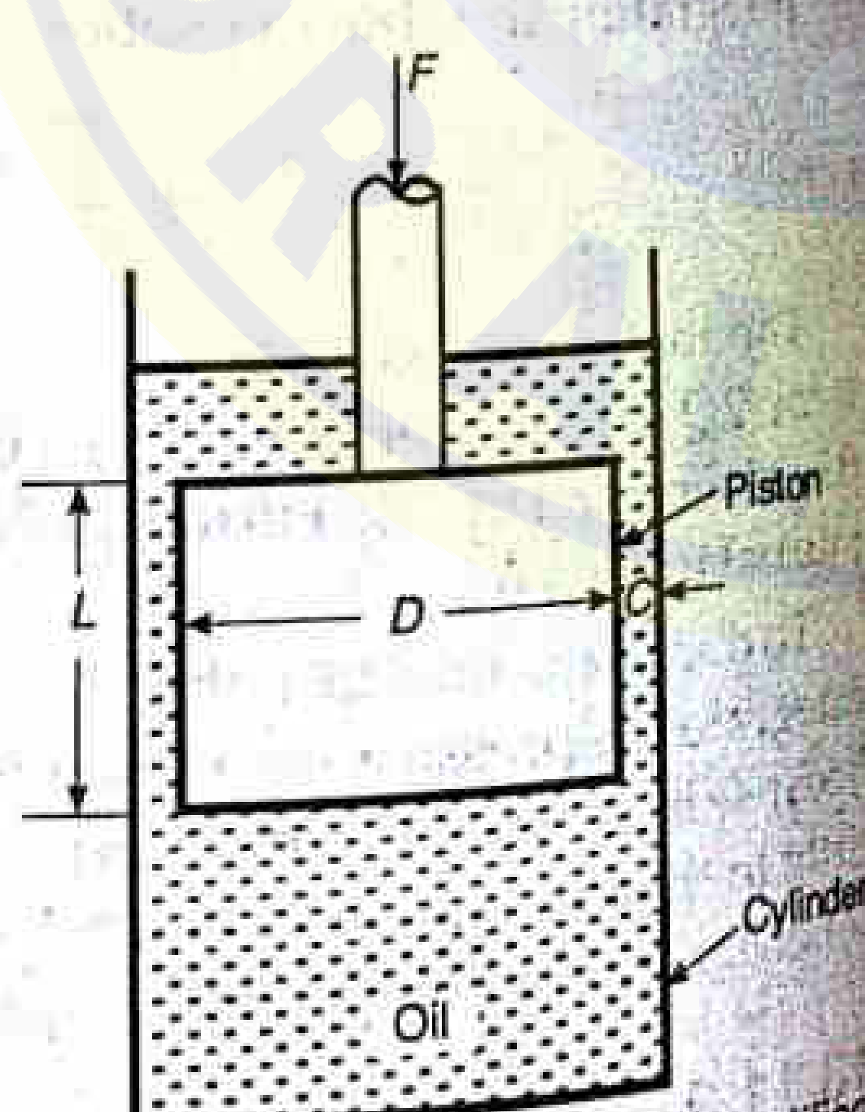


Figure 13.13 *Dash pot–a simplified sketch*

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2$$

Applying the boundary conditions:

when $y = 0,\ v = 0 \quad \therefore C_2 = 0$

when $y = c,\ v = -V_p$

(where V_p is the uniform downward velocity of the piston and hence considered negative).

$$\therefore \quad C_1 = -\frac{V_p}{c} - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{c}{2}$$

$$\therefore \quad v = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy) - \frac{V_p y}{c} \qquad ...(13.76)$$

Considering the flow through an elemental strip of breadth πD and thickness dy, then

$$dQ = v(\pi D)\, dy$$

$$= \pi D\left[\frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy) - V_p\frac{y}{c}\right]dy$$

or

$$Q = \pi D\int_0^c\left[\frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy) - V_p\frac{y}{c}\right]dy$$

$$Q = \pi D\left[-\left(\frac{\partial p}{\partial x}\right)\frac{c^3}{12\mu} - \frac{V_p c}{2}\right] \qquad ...(13.77)$$

Equation 13.77 represents the steady rate of flow of oil through the clearance space, which must be exactly equal to the rate at which oil is being displaced by the piston, i.e., $(\pi D^2/4)\, V_p$.

Thus

$$\pi D\left[-\left(\frac{\partial p}{\partial x}\right)\frac{c^3}{12\mu} - \frac{V_p c}{2}\right] = \frac{\pi D^2}{4}V_p$$

or

$$V_p\left(\frac{D}{2} + c\right) = -\frac{\partial p}{\partial x}\frac{c^3}{6\mu} \qquad ...(13.78$$

But $\frac{\partial p}{\partial x} = -\frac{p}{L}$, where p is the pressure drop over the length of the piston.

$$\therefore \quad p = \frac{6\mu V_p L}{c^3}\left(\frac{D}{2} + c\right) \qquad ...(13.7$$

Due to this pressure difference an upward force will be exerted on the piston which will be opposi its downward motion, and it is given by

$$F_p = p\frac{\pi D^2}{4} = \pi\mu V_p L\left[\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)^2\right] \qquad ...(13.8$$

In addition to this force there will be shear force (i.e., skin friction drag) exerted by the oil on the pist which will also be resisting its downward motion. The shear stress at the piston wall is given by

$$\tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=c}$$

From Eq. 13.34

$$\frac{\partial v}{\partial y} = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)y + C_1$$

As obtained above

$$C_1 = -\frac{V_p}{c} - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{c}{2}$$

$$\therefore \quad \frac{\partial v}{\partial y} = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)y - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{c}{2} - \frac{V_p}{c}$$

when $y = c$

$$\left(\mu\frac{\partial v}{\partial y}\right)_{y=c} = \frac{\partial p}{\partial x}\frac{c}{2} - \frac{\mu V_p}{c}$$

$$\therefore \quad \tau_0 = \frac{\partial p}{\partial x}\frac{c}{2} - \frac{\mu V_p}{c}$$

But from Eq. 13.78

$$\frac{\partial p}{\partial x} = -\frac{6\mu V_p}{c^3}\left(\frac{D}{2}+c\right)$$

$$\therefore \quad \tau_0 = -\left[\frac{3\mu V_p}{c^2}\left(\frac{D}{2}+c\right)+\frac{\mu V_p}{c}\right]$$

Thus shear force resisting the motion of the piston is given by

$$F_s = (\pi D L \tau_0) = \pi DL\left[\frac{3\mu V_p}{c^2}\left(\frac{D}{2}+c\right)+\frac{\mu V_p}{c}\right]$$

Note: The negative sign of shear stress has been omitted since it merely denotes that the shear force opposes the motion of the piston

$$\therefore \quad F_s = \pi\mu V_p L\left[\frac{3}{2}\left(\frac{D}{c}\right)^2 + 4\left(\frac{D}{c}\right)\right] \qquad \text{...(13.81)}$$

Therefore, the total force required to move the piston at a uniform velocity down through the oil is

$$F = F_p + F_s$$

$$F = \pi\mu V_p L\left[\left\{\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)^2\right\} + \left\{\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{4D}{c}\right\}\right] \qquad \text{...(13.82)}$$

If V_p is negligible as compared to the average velocity of oil through the clearance space, then from Eq. 13.76

$$v = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(y^2 - cy)$$

and using the same analysis as indicated above

$$p = \frac{3\mu V_p LD}{c^3}$$

and

$$\tau_0 = \frac{3}{2}\frac{\mu V_p D}{c^2}$$

Thus

$$F = F_p + F_s$$

$$= \pi\mu V_p L\left[\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)^2\right] \qquad \text{... (13.83)}$$

When the shear force (or skin friction drag) on the piston wall is negligible as compared to the force due to pressure difference then

$$F = F_p = \frac{3}{4}\pi\mu V_p L\left(\frac{D}{c}\right)^3 \qquad \text{... (13.84)}$$

Equation 13.82 gives the most general expression for the force resisting the downward motion of the piston and Eqs. 13.83 and 13.84 are only the simplified forms of Eq. 13.82. The criterion which controls the use of either of the simplifications given by the Eqs. 13.83 and 13.84 is the value of the (D/c). Thus larger is the value of (D/c) the smaller will be the effect of the piston velocity and the piston shear.

In the above experssions the downward force F on the piston includes the self-weight of the piston. However, since the piston is completely submereged in the oil, the net weight of the piston should be considered, which will be equal to the weight of the piston minus the weight of the oil displaced by it (or the buoyant force acting upwards on the piston). Further if the piston is moving downwards under its own weight only, then the value of F in the above equations will be equal to the net weight of the piston as indicated above.

13.13 MEASUREMENT OF VISCOSITY—VISCOMETERS

The various methods adopted for the measurement of viscosity of a fluid are based on the use of the following three principles:

(a) Newton's law of viscosity; (b) Hagen-Poiseuille equation; (c) Stokes' law.

(a) *Coaxial cylinder viscometer:* In this viscometer Newton's law of viscosity is employed to measure the viscosity of a fluid. In Newton's equation of viscosity, by measurement of the velocity gradient (dv/

y) and the shear stress τ, the dynamic viscosity μ can be computed.

As shown in Fig. 13.14, this viscometer consists of two coaxial cylinders, one of radius R_1, placed nside the other of radius R_2, such that a very small space $(R_2 - R_1)$ is left in between the two. The space filled with the liquid whose viscosity is to be determined. The inner cylinder is suspended by a orsion wire and it is held stationary while the outer cylinder can rotate at a known constant speed. Vhen the outer cylinder rotates, the torque generated by uch rotation is transmitted by the thin liquid film to the nner stationary cylinder, which causes rotation of the orsion wire. The rotation of the wire can be measured by neans of a circular dial attached to the wire and a fixed pointer. From the previously obtained calibration curve between the torque and the rotation of the torsion wire, the orque exerted on the wire and hence on the inner cylinder, corresponding to the measured rotation of the wire can be known. By knowing the torque on the inner cylinder, shear stress τ can be computed.

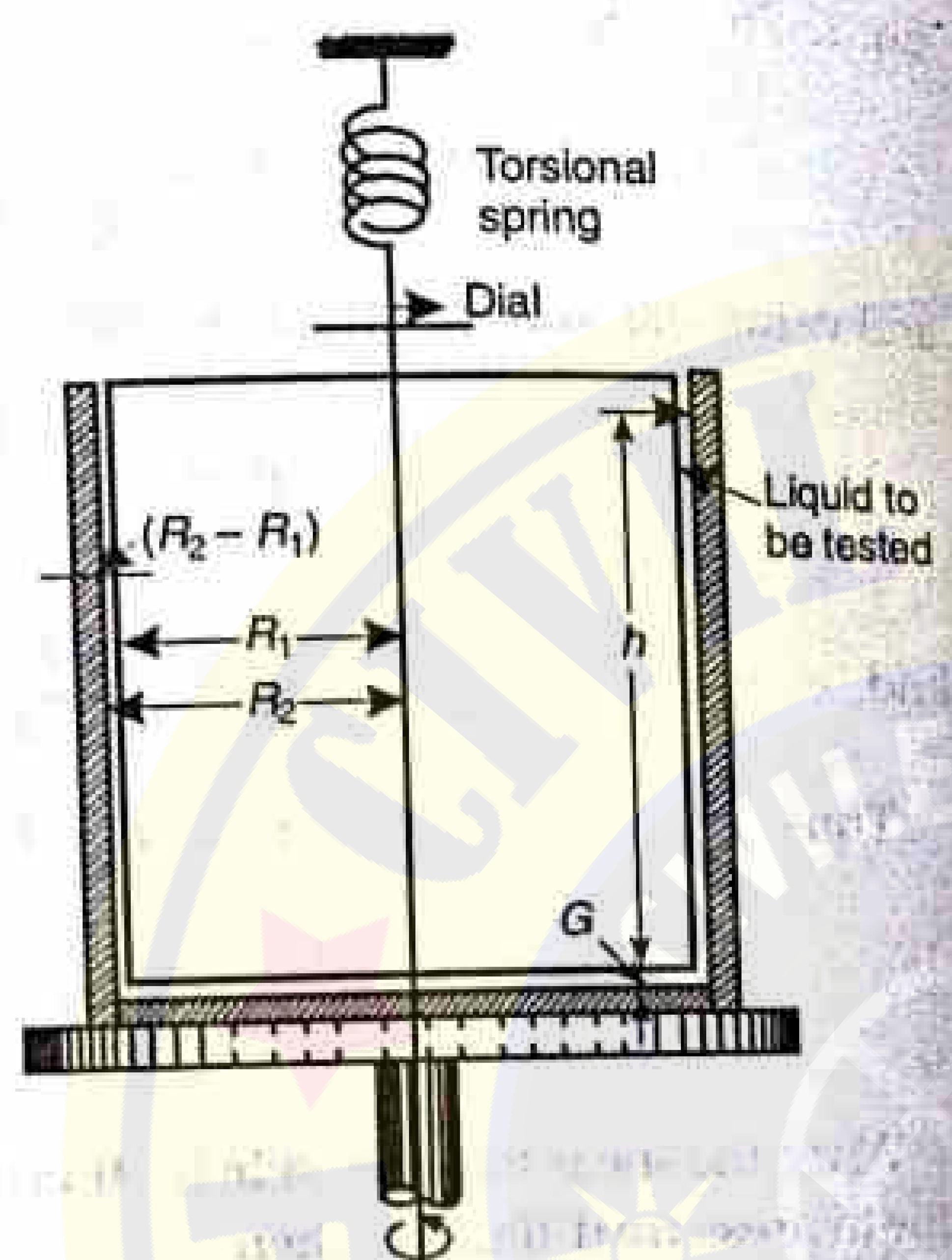


Figure 13.14 *Coaxial cylinder viscometer*

Let T be the torque exerted on the inner cylinder, obtained as indicated above and h be the depth of the liquid in the cylinder. Now if the clerance a between the two cylinders at their bottom is large, then the torque contribution from the bottom may be neglected. The torque T will then be contributed only from the sides, being equal to T_1, and it may be expressed in terms of the shear stress τ as follows:

$$T = T_1 = (2\pi R_1 h)\tau R_1$$

$$= (2\pi R_1^2 h)\tau$$

or

$$\tau = \frac{T_1}{2\pi R_1^2 h}$$

The velocity gradient may be determined as mentioned below.

When the outer cylinder rotates at a speed N rev./min, the velocity of the liquid at the surface of the outer cylinder will be $(2\pi R_2 N/60)$. The inner cylinder being stationary the velocity of the liquid at the surface of the inner cylinder will be zero. Since the space between the cylinders in very small the velocity of the liquid in the space may be assumed to vary linearly from zero at the surface of the inner cylinder to a maximum at the surface of the outer cylinder. Hence the velocity gradient may be expressed as

$$\frac{\partial v}{\partial y} = \frac{2\pi R_2 N}{60(R_2 - R_1)}$$

By substituting the value of τ and $(\partial v/\partial y)$ in Newton's equation of viscosity and solving for μ, we obtain

$$\mu = \frac{15T_1(R_2 - R_1)}{\pi^2 R_1^2 R_2 hN} \qquad \ldots (13.85)$$

However, if the clearance a is so small that the torque exerted from the bottom is appreciable then the total torque T exerted on the inner cylinder will be contributed by both bottom as well as sides. Thus if T_2 is the torque contribution from the bottom, then by performing integration over whole area of the bottom of the inner cylinder, we obtain

$$T_2 = \left(\frac{\mu\pi^2}{a}\right)\frac{N}{60}R_1^4$$

The total torque T may be expressed as

$$T = T_1 + T_2 = \frac{\mu\pi^2 R_1^2 R_2 hN}{15(R_2 - R_1)} + \frac{\mu\pi^2 N R_1^4}{60a}$$

or

$$T = \frac{\mu\pi^2 N R_1^2}{15}\left(\frac{R_2 h}{R_2 - R_1} + \frac{R_1^2}{4a}\right) \qquad \ldots (13.86)$$

Since R_1, R_2, h and a are constants for a viscometer, the torque may be expressed as

$$T = C\mu N$$

in which C is defined as a constant for the viscometer. Solving for μ

$$\mu = \frac{T}{CN} \qquad \ldots (13.87)$$

Thus Eq. 13.86 or 13.87 may be used to determine μ since execpt μ all other quantities are known.

(b) *Capillary tube viscometer*: This viscometer involves use of the Hagen–Poiseuille equation. As shown in Fig. 13.15 it consists of a capillary tube of cross-sectional area a and length L attached horizontally very close to the bottom of a resevoir of area A, filled with liquid whose viscosity μ is to be determined. The tube is allowed to discharge freely in the atmosphere. As the liquid flows through the tube, in the absence of any inflow into the tank, the level of the liquid surface in the tank will be lowered.

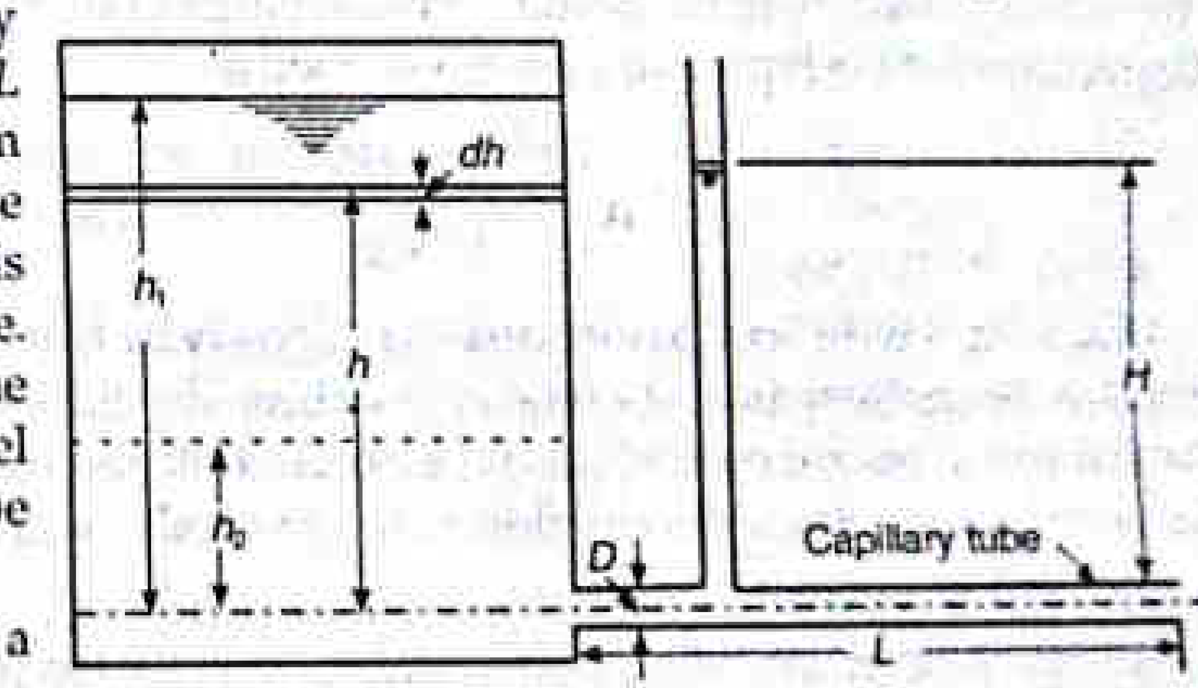


Figure 13.15 *Capillary tube viscometer*

Let at any instant the liquid surface be at a height h above the centre of the tube causing a disharge q and let the surface fall by a height dh in time dt. Then, we have

$$-A\,dh = q\,dt = aV\,dt$$

where V is the velocity of flow in the tube under the head h. Since the flow in the tube is laminar, it may be assumed that the head h is utilised completely to overcome the viscous resistance in the tube, (i.e., the velocity head possessed by the liquid leaving the tube is neglected). Then by Hagen–Poiseuille equation, we have

$$h = \frac{32\mu VL}{wD^2};\ \text{or}\ V = \frac{whD^2}{32\mu L}$$

where D is the diameter of the tube and w is the specific weight of the liquid. By substituting for V, we get

$$-A\,dh = \frac{awD^2 h}{32\mu L}dt$$

or
$$dt = \left(-\frac{32\mu LA}{awD^2}\right)\frac{dh}{h}$$

The time t required to lower the liquid surface from a height h_1 to h_2 above the centre of the tube may be obtained by integrating both sides of the above expression. Thus, we have

$$\int dt = -\frac{32\mu LA}{awD^2}\int_{h_1}^{h_2}\frac{dh}{h}$$

or
$$t = \frac{32\mu LA}{awD^2}\log_e\left(\frac{h_1}{h_2}\right)$$

$$\mu = \frac{tawD^2}{32AL[\log_e(h_1/h_2)]} \quad \ldots (13.88)$$

Alternatively by providing suitable arrangement for inflow into the tank, a constant head may be maintained over the capillary tube and the liquid is allowed to flow through the capillary tube under the constant head. Both the head H and discharge Q are measured and using Hagen–Poiseulle eqation the viscosity of the liquid may be determined as

$$\mu = \frac{\pi wHD^4}{128QL} \quad \ldots (13.89)$$

However, it is difficult to measure the viscosity by this type of viscometer. As such certain modified forms of the capillary tube viscometer have been developed for the measurement of viscosity of liquids. Two common viscometers of this type are Saybolt and Redwood viscometers, both of which involve the principle of vertical gravity flow through capillary tubes.

1. Saybolt Viscometer

It consists of a vertical cylindrical chamber surrounded by a constant temperature bath as shown in Fig. 13.16. A short capillary tube is attached vertically at the bottom of the cylindrical chamber. The diameter of the capillary tube varies from 1.75 mm. to 1.78 mm. and its length varies from 12.15 mm to 12.35 mm. The cylindrical chamber is completely filled with the liquid whose viscosity is to be determined and the outlet is plugged. The surrounding bath will maintain the liquid under test at a required constant temperature. The plug is then removed and the time in seconds is measured for the flow of 0.06 litre or 60 cm³ of the liquid through the capillary tube under a falling head. The time so obtained is designated as Saybolt reading. This viscometer measures kinematic viscosity of the liquid. Assuming the flow through the tube to be laminar, Hagen–Poiseuille equation can be used. Rearranging the Hagen –Poiseuille equation for viscosity μ, we obtain

$$\mu = \frac{wh\pi D^4}{128QL} = \frac{\rho gh\pi D^4}{128QL}$$

in which h represents the head under which the liquid flows through the tube and Q is the volume of liquid flowing per unit time. If t is the time in seconds recorded for the flows of constant volume (viz., 60 cm³) of liquid, then $Q = (\text{vol.}/t)$.Thus the above expression may be written as

$$\frac{\mu}{\rho} = \upsilon = \frac{gh\pi D^4 t}{128(vol.)L}$$

In this expression although the head h varies during the test, its variation is over the same range for all liquids, since a constant volume of liquid is allowed to flow for each test. As such all the terms on the right hand side except t may be considerd as constant and may be grouped together to provide a constant of a particular viscometer. Let C_1 be the constant of the viscometer, then we get

$$\upsilon = C_1 t \quad \ldots (13.90)$$

Equation 13.90 shows that the kinematić viscosity varies directly as the time t. In this case since the capillary tube is quite short, steady laminar flow conditions will usually not exist in the tube. Thus a correction factor will have to be incorporated in Eq. 13.90 in order to compensate for the use of Hagen-Poisuille equation in the analysis and to obtain a correct value of kinematic viscosity of the liquid. The correction is of the form (C_2/t) hence the kinematic viscosity becomes

$$\upsilon = C_1 t + \frac{C_2}{t} \quad \ldots (13.91)$$

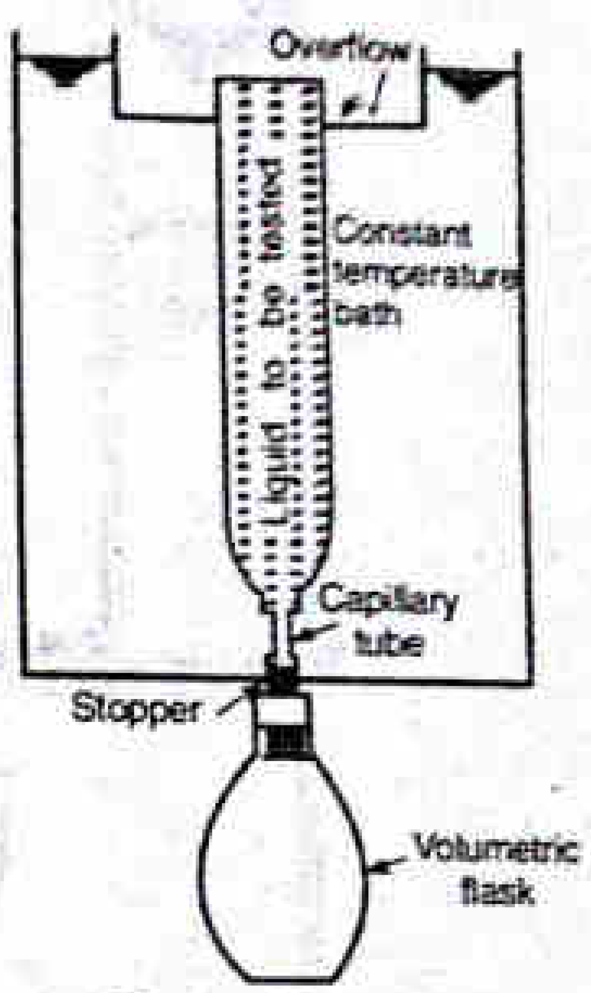


Figure 13.16 *Sybolt Viscometer*

The approximate relationship between kinematic viscosity and time t for a Saybolt Universal viscometer is expressed by

$$\upsilon = 0.0022\,t - \frac{1.80}{t} \quad \ldots (13.92)$$

where υ is in stokes and t is in seconds.

2. Redwood Viscometer

It consists of vertical cylinder provided with an orifice in the centre of its base. The cylinder is surrounded by a water bath which can maintain the liquid to be tested at any required temperature by means of an immersed electric heater as shown in Fig. 13.17. The cylinder is 47.625 mm in diameter and 88.90 mm deep. The orifice is 1.70 mm diameter and 12 mm in length. The cylinder is filled upto a fixed height with the liquid whose viscosity is to be determined, and is heated by the water bath to the required temperature. The orifice is then opened and the time taken for 0.05 litre or 50 cm³ of the flow is noted. The viscosity of the liquid is proportional to this time. Thus an expression similar to Eq. 13.90 can be obtained for this viscometer as well. It may, however, be pointed out that this viscometer does not provide a direct value of the viscosity of any liquid. It only facilitates a comparison of viscosities of different liquids. The exact value of viscosity of any liquid can, however, be obtained by comparison with the value of time t for liquids of known viscosity.

(c) *Falling sphere viscometer.* This method of measuring viscosity of a liquid is based on Stokes' Law. As illustrated in Fig. 13.18, it consists of a long vertical container either cylindrical or prismatic in shape. The container is filled with the liquid whose viscosity is to be determined and is surrounded by a constant temperature bath. A small spherical ball of steel is allowed to fall vertically through the liquid. The falling ball as it is dropped will at first accelerate, but the resistance to its motion increases with its velocity until the resistance to its motion is counter-balanced by the pull of gravity on the ball. When this condition is reached the ball falls with a constant velocity. The time t taken by the ball to fall

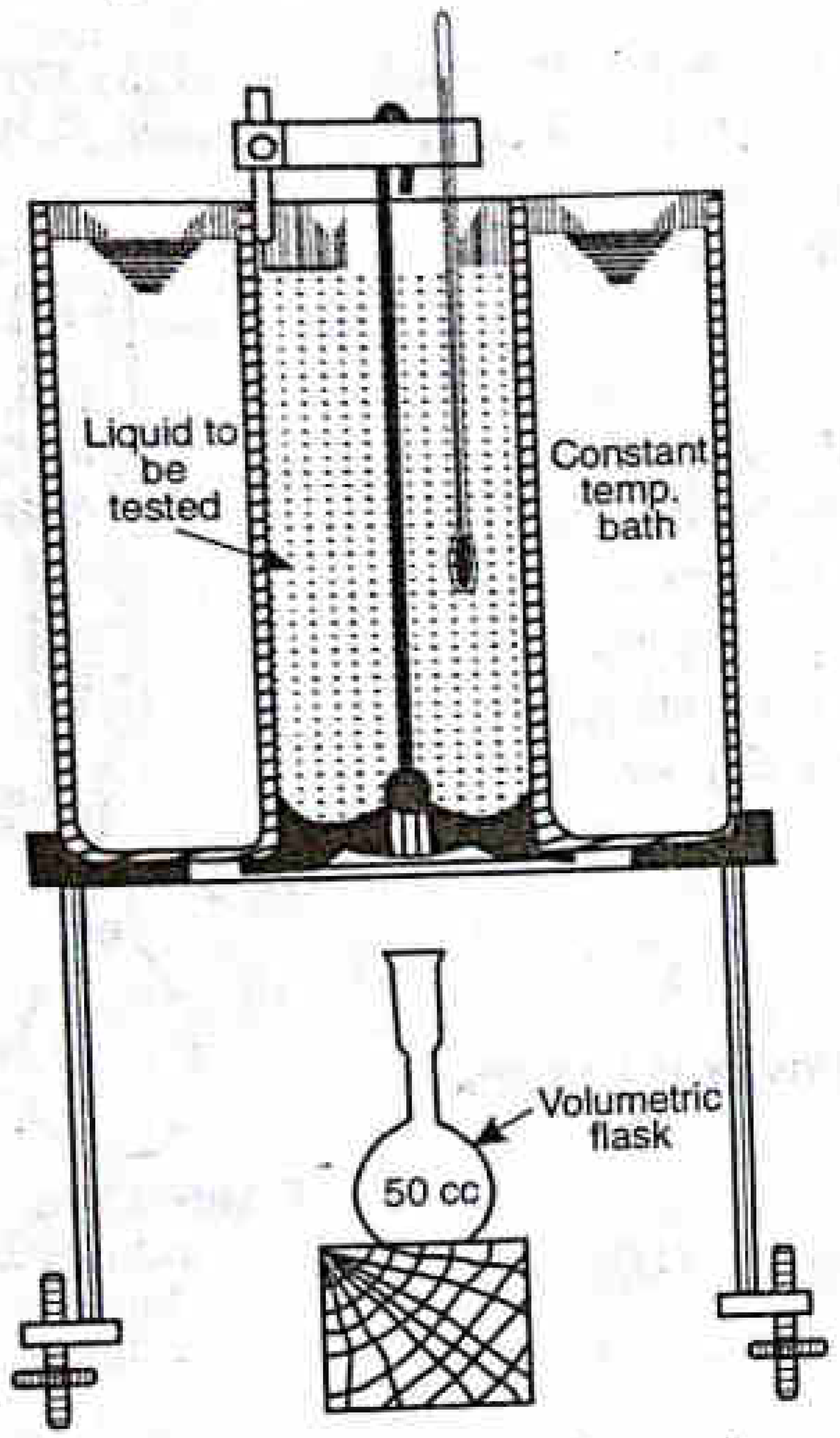


Figure 13.17 *Redwood viscometer*

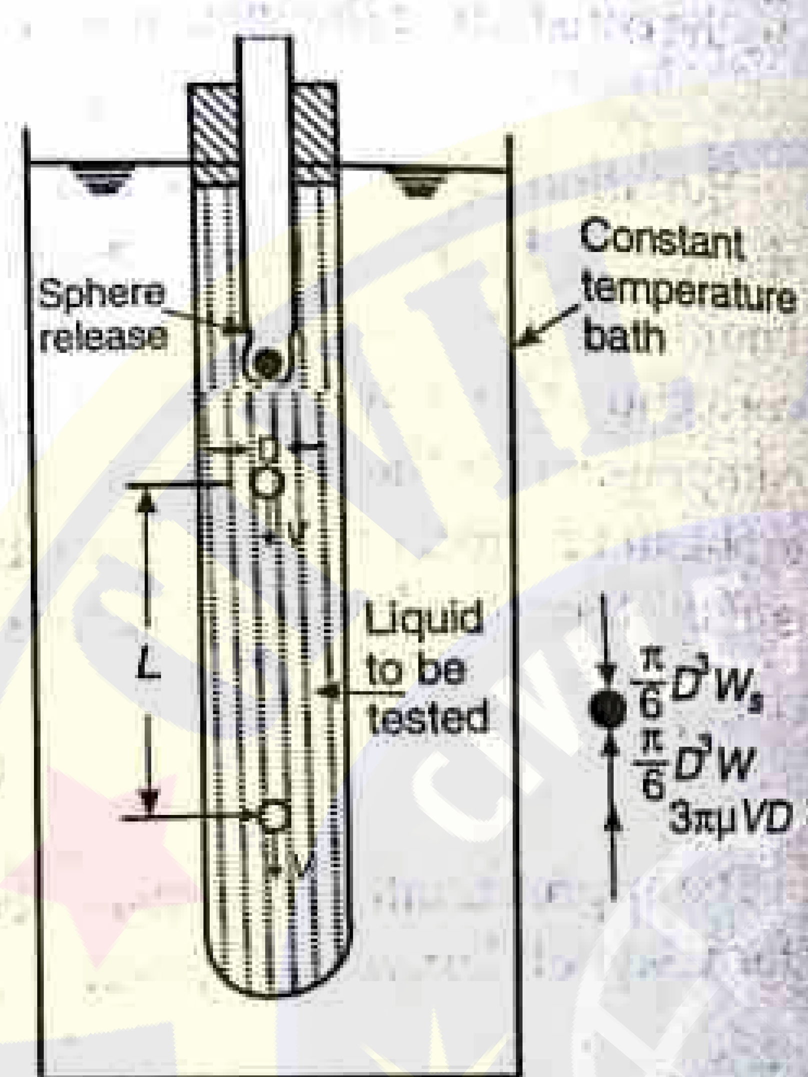


Figure 13.18 *Falling sphere viscometer*

through a known distance L, after a constant velocity has been attained by it, is noted. Then the fall velocity V is known, since $V = (L/t)$.

As indicated in Section 13.10 the viscosity μ may be determined using Eq. 13.62 as

$$\mu = \frac{D^2}{18V}(w_s - w) \quad \text{... (13.93)}$$

where w_s and w are specific weights of the ball and the liquid respectively. This method will, however give accurate value of viscosity only if Stokes' Law is applicable.

ILLUSTRATIVE EXAMPLES

Example 13.1. *Lubricating oil of specific gravity 0.82 and dynamic viscosity 12.066×10^{-2} N.s/m² [1.23×10^{-2} kg (f)–s/m²] is pumped at a rate of 0.02 m³/s through a 0.15 m diameter 300 m long pipe. Calculate the pressure drop, average shear stress at the wall of the pipe and the power reuired to maintain flow (a) if the pipe is horizontal; (b) if the pipe is inclined at 15 degrees with the horizontal and the flow is (i) in the upward direction, (ii) in the downward direction. Also determine the slope of the pipe and the directon of flow so that the pressure gradient along the pipe is zero.*

Solution

$$V = \frac{0.02}{\frac{\pi}{4}(0.15)^2} = 1.132 \text{ m/s}$$

(a) For horizontal pipe from Hagen-Poiseuille equation, we have

$$(p_1 - p_2) = \frac{32\mu VL}{D^2}$$

SI units

$$(P_1 - P_2) = \frac{32 \times 12.066 \times 10^{-2} \times 1.132 \times 300}{(0.15)^2}$$

$$= 58\,277.17 \text{ N/m}^2$$

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= \frac{58277.17}{300} \times \frac{0.15}{2 \times 2} = 7.285 \text{ N/m}^2$$

Power required to maintain the flow is

$$P = Q(p_1 - p_2)$$

$$= 0.02 \times 58\,277.17 = 1166 \text{ W} = 1.166 \text{ kW}$$

Metric units

$$(p_1 - p_2) = \frac{32 \times 1.23 \times 10^{-2} \times 1.132 \times 300}{(0.15)^2}$$

$$= 5\,941 \text{ kg (f)/m}^2$$

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= \frac{5941}{300} \times \frac{0.15}{2 \times 2} = 0.743 \text{ kg(f)/m}^2$$

Power required to maintain the flow is

$$p = \frac{Q(p_1 - p_2)}{75}$$

$$= \frac{0.02 \times 5941}{75} = 1.584 \text{ h.p.}$$

(b) (i) For inclined pipe with flow in upward direction, we have

$$w(h_1 - h_2) = \frac{32\mu VL}{D^2}$$

SI units

$$(h_1 - h_2) = \frac{32 \times 12.066 \times 10^{-2} \times 1.132 \times 300}{(9810 \times 0.082) \times (0.15)^2} = 7.245 \text{ m}$$

or $\left(\frac{p_1}{w}+0\right)-\left(\frac{p_2}{w}+Z_2\right) = 7.245$

Since $\frac{Z_2}{300} = \sin 15° = 0.2588 \; ; Z_2 = 77.64$ m

$\therefore \quad (p_1 - p_2) = (9\,810 \times 0.82)\,(7.245 + 77.64)$
$= 682\,832 \text{ N/m}^2 = 682.832 \text{ kN/m}^2$

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(9810 \times 0.82)\times 7.245}{300} \times \frac{0.15}{2\times 2} = 7.285 \text{ N/m}^2$$

Power required to maintain the flow is

$$P = Q\,(p_1 - p_2)$$
$$= 0.02 \times 682\,832 = 13\,657 \text{ W} = 13.657 \text{ kW}$$

Metric untis

$$(h_1 - h_2) = \frac{32 \times 1.23 \times 10^{-2} \times 1.132 \times 300}{(1000 \times 0.82)\times(0.15)^2} = 7.245 \text{ m}$$

or $\left(\frac{p_1}{w}+0\right)-\left(\frac{p_2}{w}+Z_2\right) = 7.245$

Since $\frac{Z_2}{300} = \sin 15° = 0.2588 \; ; Z_2 = 77.64$ m

$\therefore \quad (p_1 - p_2) = (1000 \times 0.82)\,(7.245 + 77.64)$
$= 6.96 \times 104 \text{ kg (f)/m}^2$

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(1000 \times 0.82)\times 7.245}{300} \times \frac{0.15}{2\times 2}$$
$$= 0.743 \text{ kg (f)/m}^2$$

Power required to maintains the flow is

$$P = \frac{Q(p_1 - p_2)}{75}$$

$$= \frac{0.02 \times (6.96 \times 10^4)}{75} = 18.56 \text{ h.p.}$$

(ii) For inclined pipe with flow in downward direction, we have

$$w\,(h_1 - h_2) = \frac{32\mu VL}{D^2}$$

SI units

$$(h_1 - h_2) = \frac{32 \times 12.066.10^{-2} \times 1.132 \times 300}{(9810 \times 0.82)\times(0.15)^2} = 7.245 \text{ m}$$

or $\left(\frac{p_1}{w}+Z_1\right)\left(\frac{p_2}{w}+0\right) = 7.245$

Since $\frac{Z_1}{300} = \sin 15° = 0.2588; Z_1 = 77.64$ m

$\therefore \quad (p_1 - p_2) = (9810 \times 0.82)\,(7.245 - 77.64)$
$= -566\,271 \text{N/m}^2 = -566.271 \text{ kN/m}^2$

i.e., in this case the pressure increases in the direction of flow, or there is positive pressure gradient.

$$\tau_0 = W\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(9810 \times 0.82)\times 7.245}{300} \times \frac{0.15}{2\times 2} = 7.285 \text{ N/m}^2$$

In this case the resistance to flow is compensated by the excessive downward slope of the pipe and hence no external power is required to maintain the flow. Moreover, in this case the flow will have to be regulated by means of a regulating value to maintain the given flow rate.

For pressure gradient along the pipe to be zero, $p_1 = p_2$.

Then from the two above noted cases of inclination of the pipe we have either, $-Z_2 = 7.245$ m, or $Z_1 = 7.245$ m, from which it may be concluded that point 1 is higher than point 2, so that the flow is in the downward direction. The slope required to be provided for the pipe in this case is given by

$$\sin\theta = \frac{Z_1}{300} = \frac{7.245}{300} = 0.0242$$

$\therefore \quad \theta = 1°23'$

Example 13.2. *Water flows at a steady mean velocity of 1.5 m/s through a 50 mm diameter pipe slopping upwards at 45° to the horizontal. At a section some distance downstream of the inlet the pressure is 700 k Pa and at a section 30 m further along the pipe the pressure is 462 k Pa. Determine the average shear stress at the wall of the pipe and at a radius of 10 mm.*

Solution

Assuming datum to be passing through the lower point,we have

$$(h_1 - h_2) = \left(\frac{p_1}{w}+0\right)-\left(\frac{p_2}{w}+Z_2\right)$$

$$= \left(\frac{700 \times 10^3}{9810}\right)-\left(\frac{462 \times 10^3}{9810}+\frac{30}{\sqrt{2}}\right)$$

$$= (71.356 - 68.308) = 3.048 \text{ m}$$

Average shear stress at the wall of the pipe is

$$\tau_0 = w\left(-\frac{\partial h}{\partial x}\right)\frac{R}{2}$$

$$= \frac{9810 \times 3.048}{30} \times \frac{0.05}{2 \times 2}$$

$$= 12.459 \text{ N/m}^2 = 12.459 \text{ Pa}$$

Average shear stress at a radius of 10 mm is

$$\tau = \left(-\frac{\partial h}{\partial x}\right)\frac{r}{2}$$

$$= \frac{9810 \times 3.048}{30} \times \frac{0.01}{2}$$

$$= 4.983 \text{ N/m}^2 = 4.983 \text{ Pa}$$

Example 13.3. *For Laminar flow of an oil having dynamic viscosity μ = 1.766 Pa.s in a 0.3 m diameter pipe, the velocity distribution is parabolic with a maximum point velocity of 3 m/s at the centre of the pipe. Calculate the shearing stresses at the pipe wall and within the fluid 50 mm from the pipe wall.*

Solution

From Eq. 13.15, we have

$$V = \frac{1}{2}v_{max} = \frac{3}{2} = 1.5, \text{m/s}$$

Also from Eq. 13.17, we have

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{p_1 - p_2}{L} = \frac{32\mu V}{D^2}$$

Thus

$$\left(-\frac{\partial p}{\partial x}\right) = \frac{32 \times 1.766 \times 1.5}{(0.3)^2} = 941.87 \text{ Pa/m}$$

The shear stress at the pipe wall is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= \frac{(941.87 \times 0.3)}{2 \times 2} = 70.64 \text{ Pa}$$

The shear stress at 50 mm from the pipe wall is

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2}$$

$$= 941.87 \times \frac{(0.15 - 0.05)}{2} = 47.09 \text{ Pa}$$

Example 13.4. *Two parallel plates kept 0.1 m apart have laminar flow of oil between them with a maximum velocity of 1.5 m/s. Calculate the discharge per metre width, the shear stress at the plates, the difference in pressure in pascals between two points 20 m apart, the velocity gradient at the plates and velocity at 0.02 m from the plate. Take viscosity of oil to be 2.453 N.s/m².*

Solution

In this case the mean velocity of flow V is equal to two-thirds of the maximum velocity.

$$\therefore \quad V = \frac{2}{3}(1.5) = 1.0 \text{ m/s}$$

The discharge q per metre width the plate is given by

$$q = VB = (1.0 \times 0.1) = 0.1 \text{m}^3\text{/s per m}$$

From Eq. 13.38, we have

$$V = \frac{B^2}{12\mu}\left(-\frac{\partial p}{\partial x}\right)$$

$$\therefore \quad \left(-\frac{\partial p}{\partial x}\right) = \frac{12\mu V}{B^2}$$

$$= \frac{12 \times 2.453 \times 1.0}{(0.1)^2} = 2943.6 \text{ N/m}^2 \text{ per m}$$

The shear stress at the plates is given by Eq. 13.43 as

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2} = \frac{2943.6 \times 0.1}{2} = 147.18 \text{ N/m}^2$$

The pressure difference between the two points is given by Eq. 13.40 as

$$(p_1 - p_2) = \frac{12\mu V}{B^2}$$

$$= \frac{12 \times 2.453 \times 1.0 \times 20}{(0.1)^2}$$

$$= 58872 \text{ N/m}^2 = 58.872 \text{kN/m}^2$$

The shear stress at plates is also given by

$$\tau_0 = \mu\left(\frac{\partial v}{\partial y}\right)_{y=0}$$

As such the velocity gradient at the plates is given by

$$\left(\frac{\partial v}{\partial y}\right)_{y=0} = \frac{\tau_0}{\mu} = \frac{147.18}{2.453} = 60 \text{ s}^{-1}$$

The velocity v at a distance of 0.02 m from the plate is given by Eq. 13.35 as

$$v = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2)$$

$$= \frac{1}{2 \times 2.453} \times (2943.6) \times \left[(0.1 \times 0.02) - (0.02)^2\right]$$

$$= 0.96 \text{ m/s}$$

Example 13.5. *Oil of viscosity 0.1 Pa.s and specific gravity 0.90, flows through a horizontal pipe of 25 mm diameter. If the pressure drop per metre length of the pipe is 12 k Pa, determine (a) the rate of flow in N/min; (b) the shear stress at the pipe wall; (c) the Reynolds number of the flow; and (d) the power required per 50 m length of pipe to maintain the flow.*

Solution

From Eq. 13.17, we have

$$V = \frac{(p_1 - p_2)D^2}{32\mu L} = \frac{12\times10^3\times(0.025)^2}{32\times0.1\times1} = 2.34 \text{ m/s}$$

(a) Rate of flow in N/min is given by

$$Q = \frac{\pi}{4}(0.025)^2\times2.34\times(9810\times0.90)\times60$$
$$= 608.5 \text{ N/min}$$

(b) From Eq. 13.4, we have

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{R}{2} = (12\times10^3)\times\frac{0.025}{2\times2} = 75 \text{ N/m}^2$$

(c) Reynolds number

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(1000\times0.90)\times2.34\times0.025}{0.1} = 526.5$$

(d) From Eq. 13.24, we have

Power $$P = Q(p_1 - p_2)$$
$$= \frac{\pi}{4}(0.025)^2\times2.34\times(12\times10^3\times50) = 689.2 \text{ W}$$

Example 13.6. *A total of 12 litres per second of oil is pumped through two pipes in parallel, one 10 cm in diameter and the other 12 cm in diameter, both pipes being 1000 metres long. The specific gravity of the oil is 0.97 and the kinematic viscosity 9 cm² per second. Calculate the flow rate through each pipe and the horse-power of the pump.*

Solution

Assuming the flow in each pipe to be laminar, the head loss is given by Eq.13.18 as

$$h_f = \frac{32\mu VL}{wD^2} = \frac{128\mu QL}{w\pi D^4}$$

Let Q_1 and Q_2 be the flow rates through each of the two pipes, which being in parallel, we have

$$h_{f_1} = h_{f_2}$$

or
$$\frac{128\mu_1 Q_1 L_1}{w_1\pi D^2} = \frac{128\mu_2 Q_2 L_2}{w_2\pi D^2}$$

$$\mu_1 = \mu_2 = (\rho\upsilon) = (0.97\times1\times9) \text{ g/cm.s} = 8.73 \text{ g/cm.s};$$
$$L_1 = L_2 = 1000 \text{ m}; D_1 = 10 \text{ cm} = 0.10 \text{ m}; D_2 = 12 \text{ cm} = 0.12 \text{ m};$$
and
$$w_1 = w_2 = 0.97\times9810 \text{ N/m}^3 = 9515.7 \text{ N/m}^3$$

By substituting, we get

$$\frac{128\times8.73\times Q_1\times1000}{9515.7\times\pi\times(0.10)^4} = \frac{128\times8.73\times Q_2\times1000}{9515.7\times\pi\times(0.12)^4}$$

or
$$\frac{Q_1}{(0.10)^4} = \frac{Q_2}{(0.12)^4}$$

or
$$\frac{Q_1}{Q_2} = \left(\frac{0.10}{0.12}\right)^4 = 0.4823$$

or
$$Q_1 = 0.4823\, Q_2$$

But
$$Q_1 + Q_2 = 12 \text{ l/s}$$

$\therefore$
$$Q_1 = 3.904 \text{ l/s; and } Q_2 = 8.096 \text{ l/s}$$

The Reynolds number is given by

$$\text{Re} = \frac{Vd}{\upsilon}$$

For pipe of diameter 10 cm,

$$V = \frac{3.904\times10^3}{(\pi/4)(10)^2} = 49.71 \text{ cm/s}$$

$\therefore$
$$\text{Re} = \frac{49.71\times10}{9} = 55$$

Similarly for pipe of diameter 12 cm

$$V = \frac{8.096\times10^3}{(\pi/4)(12)^2} = 71.58 \text{ cm/s}$$

$\therefore$
$$\text{Re} = \frac{71.58\times12}{9} = 95$$

Thus the assumption of laminar flow in both the pipes is correct.

The head loss
$$h_f = \frac{128\mu QL}{w\pi D^4}$$
$$= \frac{128\mu QL}{\rho g\pi D^4}$$
$$= \frac{128\,\upsilon QL}{g\pi D^4}$$

$$\upsilon = 9 \text{ cm}^2/\text{s}; Q = 3.904\times10^3 \text{ cm}^3/\text{s}; L = 1000\times100 = 10^5 \text{ cm};$$
$$g = 981 \text{ cm/s}^2; \text{ and } D = 10 \text{ cm}$$

Thus by substitution, we get

$$h_f = \frac{128\times9\times3.904\times10^3\times10^5}{981\times\pi\times(10)^4}$$
$$= 1.46\times10^4 \text{ cm} = 146 \text{ m}$$

Power of the pump is

$$P = \frac{wQh_f}{75} \text{ metric horse-power}$$
$$w = 0.97\times1000 = 970 \text{ kg(f)/m}^3$$
$$Q = 12 \text{ l/s} = 12\times10^{-3} \text{ m}^3/\text{s}$$

$$\therefore \quad P = \frac{970\times12\times10^{-3}\times146}{75}$$

$$= 22.66 \text{ metric horse-power}$$

$$= 22.66 \times 735.5 \text{ W}$$

$$= 16.7 \times 10^3 \text{ W}$$

$$= 16.7 \text{ kW}$$

Example 13.7. *Determine the depth d below the free surface at which the point velocity is equal to the average velocity of flow for a uniform laminar flow with a free surface.*

Solution

From Eqs. 13.52 and 13.54, equating the point velocity and the average velocity, we have

$$\frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy-\frac{y^2}{2}\right) = \frac{w}{3\mu}\left(-\frac{\partial z}{\partial x}\right)D^2$$

or $$6Dy - 3y^2 = 2D^2$$

or $$3y^2 - 6Dy + 2D^2 = 0$$

or $$y = 0.423D$$

Since the distance y in the above expression is measured from the bed of the channel, the required depth

$$d = D - 0.423D = 0.577D$$

Example 13.8. *A horizontal column of sand 0.6 m long and 75 mm in diameter has water flowing through it. If the pressure drop across the column of sand is 20.601 k Pa and the discharge is 4.55 litres/min, what is the value of transmission constant?*

Solution

From Eq. 13.60, we have

$$V = Ki$$

$$V = \frac{4.55\times10^6}{60\times\frac{\pi}{4}\times(75)^2} = 17.165 \text{ mm/s}$$

$$i = \frac{h_f}{L} = \frac{20.601\times10^3}{9810\times0.6} = 3.5$$

$$K = \frac{V}{i} = \frac{17.165}{3.5} = 4.904 \text{ mm/s}$$

i.e., the transmission constant $K = 4.904$ mm/s

Example 13.9. *In a certain capillary tube viscometer the reservoir is of 100 mm diameter and a slender tube fixed horizontally at the bottom of the reservoir is 1 mm in diameter and 300 mm long. Determine the viscosity of a liquid of specific gravity 0.82 if the level in the reservoir is 0.1265 m at the beginning of the test and 0.12 m at the end of 2 minutes.*

Solution

From Eq. 13.88, we have

$$\mu = \frac{tawD^2}{32AL[\log_e(h_1/h_1)]}$$

Thus by substitution, we get

$$\mu = \frac{2\times60\times\frac{\pi}{4}(0.001)^2\times9810\times0.82\times(0.001)^2}{32\times\frac{\pi}{4}(0.1)^2\times0.3\times\left(\log_e\frac{0.1265}{0.12}\right)}$$

$$= 1.906 \times 10^{-4} \text{ N.s/m}^2$$

$$= 1.906 \times 10^{-3} \text{ poise}$$

Example 13.10. *0.02 litres of a liquid (specific gravity 0.9) flow in 40 seconds through a pipe 0.25 m lo[ng] and 1.5 mm diameter emerging from the bottom of a tank holding the liquid maintained at a constant level 0.35 m above the outlet end of the pipe discharging freely. Compute the dynamic viscosity of the liquid.*

Solution

From Eq. 13.89, we have

$$\mu = \frac{\pi wHD^4}{128QL}$$

$$w = (9\,810\times0.9)\ \text{N/m}^3;\quad H = 0.35\text{ m};\quad D = (1.5\times10^{-3})\text{ m}$$

$$Q = \frac{(0.02\times10^{-3})}{40}\text{ m}^3/\text{s};\quad L = 0.25\text{ m}$$

Thus by substitution, we get

$$\mu = \frac{\pi\times(9810\times0.9)\times0.35\times(1.5\times10^{-3})^4\times40}{128\times(0.02\times10^{-3})\times0.25}$$

or $$\mu = 3.072\times10^{-3}\text{ N.s/m}^2$$

or $$\mu = 3.072\times10^{-2}\text{ poise}$$

Example 13.11. *A small glass sphere of specific gravity 2.65 has a diameter of 3 mm. If the sphere [falls] 0.6 m in 30 seconds through a liquid of specific gravity 0.85, what is the absolute viscosity of the liqui[d]*

Solution

From Eq. 13.93, we have

$$\mu = \frac{D^2}{18V}(w_s - w)$$

$$V = \frac{0.6}{30} = 0.02\text{ m/s}$$

Thus by substitution, we have

$$\mu = \frac{(0.003)^2}{18\times(0.02)}\times9810\,(2.65-0.85)$$

$= 0.441\ N.s/m^2$

$= 0.0441$ poise

Example 13.12. *The space between two parallel plates distance h apart is filled with incompressible fluid of viscosity μ. The bottom plate moves at velocity $(-V_1)$ and the top plate moves at velocity $+V_2$. Obtain the expression for velocity distribution (v/V_2) for this case.*

Solution

From Eq. 13.34 we have

$$v = \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2} + C_1 y + C_2$$

In the present case the boundary conditions are as follows:

at $\quad y = 0, v = -V_1$ and at $y = h, v = V_2$

Thus $\quad C_2 = -V_1$

and $\quad C_1 = \frac{V_2}{h} + \frac{V_1}{h} - \frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{h}{2}$

∴ By substitution, we get

$$v = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(hy - y^2) + \left(\frac{V_2 y}{h} + \frac{V_1 y}{h} - V_1\right)$$

or

$$\frac{v}{V_2} = \frac{y}{h} = -\frac{V_1}{V_2}\left(1 - \frac{y}{h}\right) + \left[\frac{h^2}{2\mu V_2}\left(-\frac{\partial p}{\partial x}\right)\frac{y}{h}\left(1 - \frac{y}{h}\right)\right]$$

Example 13.13. *A dash-pot has a cylinder 70 mm diameter in which slides a piston 80 mm long having a radial clearance of 1 mm. The cylinder is filled with oil of viscosity 1 poise. Determine the uniform velocity of the piston when the force acting on it is 180 N. What percentage error is involved if the shear force on the piston wall is neglected?*

Solution

From Eq. 13.82, we have

$$F = p\mu V_p L\left[\left\{\left(\frac{3}{4}\left(\frac{D}{c}\right)^3 + \frac{3}{2}\left(\frac{D}{c}\right)\right)^2\right\} + \left\{\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{4D}{c}\right\}\right]$$

$$\frac{D}{c} = 68; \left(\frac{D}{c}\right)^2 = (68)^2 = 46.24 \times 10^2$$

$$\left(\frac{D}{c}\right)^3 = (68)^3 = 314.4 \times 10^3$$

$$\frac{3}{4}\left(\frac{D}{c}\right)^3 = \frac{3}{4} \times 314.4 \times 10^3 = 235.8 \times 10^3$$

$$\frac{3}{2}\left(\frac{D}{c}\right)^2 = \frac{3}{2} \times 46.24 \times 10^2 = 69.36 \times 10^2$$

$$4\frac{D}{c} = 4 \times 68 = 272$$

$$\left[\frac{3}{4}\left(\frac{d}{C}\right)^3 + \frac{3}{2}\left(\frac{d}{C}\right)^2\right] = 10^3(235.8 + 6.936) = 242.736 \times 10^3$$

$$\left[\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{4D}{c}\right] = 10^3(6.936 + 0.272) = 7.208 \times 10^3$$

$$1 \text{ poise} = 0.1\ N.s/m^2$$

Thus by substitution we get

$$180 = \pi \times 0.1 \times V_p \times 0.08 \times 10^3(242.736 + 7.208)$$

$$\therefore \quad V_p = \frac{180}{\pi \times 0.1 \times 0.08 \times 10^3 \times 249.944} = 2.865 \times 10^{-2}\ m/s$$

If shear force at the piston wall is neglected, then $\left[\frac{3}{2}\left(\frac{D}{c}\right)^2 + \frac{D}{c}\right]$ is negligible.

Hence $\quad V_p = \frac{180}{\pi \times 0.1 \times 0.08 \times 10^3 \times 242.736} = 2.951 \times 10^{-2}\ m/s$

∴ Percentage error $= \left(\frac{2.951 \times 10^{-2} - 2.865 \times 10^{-2}}{2.865 \times 10^{-2}}\right) \times 100 = 3\%$

Example 13.14. *The thrust of a shaft is taken by a collar bearing filled with a forced lubrication system which maintains a film of oil of constant thickness 0.3 mm between the surface of the collar and the bearing. The outer and inner diameters of the collar are 156 mm and 120 mm respectively. The coefficient of viscosity of the oil is 1.2 poise. Calculate the power lost in friction of the bearing when the shaft rotates at 500 revolutions per minute.*

Solution

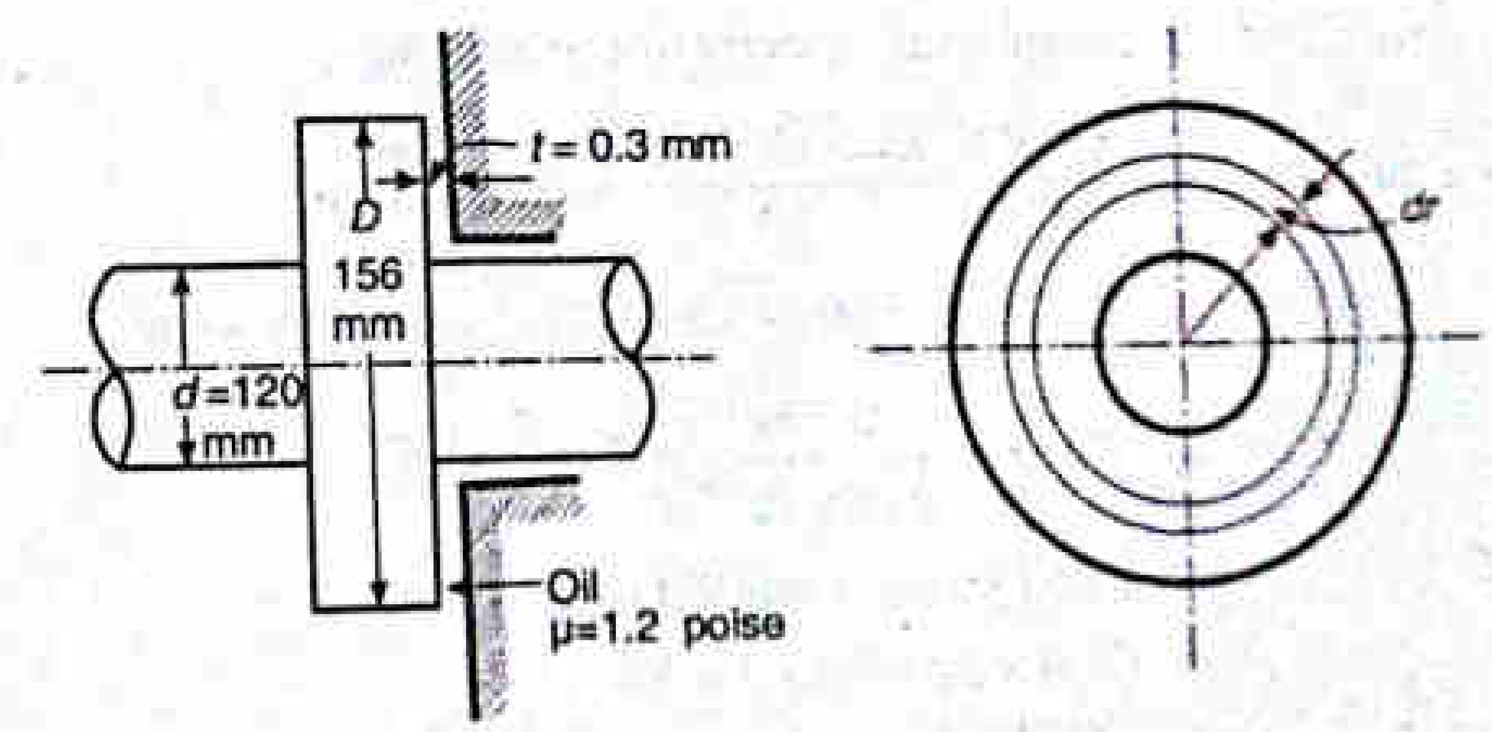


Figure Ex. 13.14

The arrangement of the collar thrust bearing is shown in the Fig. Ex. 13.14. In this case the peripheral speed varies with the radius, hence considering an elemental ring of radius r and thickness dr, the shear stress at the ring.

$$\tau = \mu \frac{v}{t}$$

$$v = r\omega = \frac{2\pi N r}{60} = \frac{2\pi \times 500 r}{60} = \frac{50\pi r}{3} \text{ m/s}$$

$$\mu = 1.2 \text{ poise} = 0.12 \text{ N.s/m}^2$$

$$t = 0.3 \text{ mm} = 0.3 \times 10^{-3} \text{ m}$$

$$\therefore \quad \tau = 0.12 \times \frac{50\pi r}{3} \times \frac{1}{0.3 \times 10^{-3}} = 6667\,\pi\, r \text{ N/m}^2$$

Shear force on the ring $= \tau \times 2\pi r\, dr$

$= 13\,334\, \pi^2 r^2 dr$

Torque on the ring $=$ Shear force $\times$ radius

$= 13\,334\, \pi^2 r^3 dr$

Total torque required to overcome the viscous resistance of oil

$$T = 13\,334\, \pi^2 \int_{0.06}^{0.078} r^3 dr = 0.791 \text{ Nm}$$

$$\therefore \quad \text{Power} = \frac{2\pi NT}{60} = \frac{2\pi \times 500 \times 0.791}{60} = 41.417 \text{ W}$$

Example 13.15. *A 0.15 m diameter journal runs in a bearing 0.3 m long. The lubricant used has a specific gravity of 0.855 and a kinematic viscosity 1.81 stokes. If the radial clearance may be assumed to be uniform and equal to 0.05 mm, determine the power required to overcome the viscous resistance of lubricant when the journal rotates at 300 r.p.m.*

Solution

Shear stress $\tau = \mu \dfrac{v}{C}$

where C is the clearance and v is peripheral velocity of the journal

$$C = 0.05 \times 10^{-3} \text{ m}; \; v = \frac{\pi DN}{60} = \frac{\pi \times 0.15 \times 300}{60} = 0.75\,\pi \text{ m/s}$$

$$\mu = \rho\upsilon = (0.855 \times 1000) \times 1.81 \times 10^{-4} = 0.155 \text{ N.s/m}^2$$

$$\therefore \quad \tau = 0.155 \times \frac{0.75\pi}{0.05 \times 10^{-3}} = 7304 \text{ N/m}^2$$

Shear force $=$ Shear stress $\times$ wetted area

$= 7304 \times (\pi \times 0.15 \times 0.3)$

$= 1032.58$ N

Torque to overcome viscous resistance of lubricant

$T =$ Shear force $\times$ radius

or $T = 1032.58 \times \dfrac{0.15}{2} = 77.44$ Nm

$$\text{Power} = \frac{2\pi NT}{60}$$

$$= \frac{2\pi \times 300 \times 77.44}{60} = 2433 \text{ W} = 2.433 \text{ kW}$$

Example 13.16. *A flow of 420 litres/min of oil (specific gravity = 0.91, and viscosity = 1.24 poise) is pumped through a pipeline 75 mm diameter having a length of 62 m and whose outlet is 3 m higher than its inlet. Estimate the power required for the pump if its efficiency is 60%.*

Solution

Discharge $Q = \dfrac{420 \times 10^{-3}}{60} = 7 \times 10^{-3}$ m^3/s

Velocity $V = \dfrac{7 \times 10^{-3}}{(\pi/4) \times (0.075)^2} = 1.584$ m/s

Reynolds number $\text{Re} = \dfrac{\rho V D}{\mu}$

$\mu = 1.24$ poise

$= (1.24 \times 0.1)$ N.s/m^2

$$\therefore \quad \text{Re} = \frac{(0.91 \times 1000) \times 1.584 \times 0.075}{(1.24 \times 0.1)}$$

$= 871.84$

Since Re < 2000, the flow is laminar.

∴ Head loss due to friction

$$h_f = \frac{32 \mu V L}{w D^2}$$

$$= \frac{32 \times (1.24 \times 0.1) \times 1.584 \times 62}{(9810 \times 0.91) \times (0.075)^2}$$

$= 7.76$ m

Total head $H = 3 + \dfrac{V^2}{2g} + h_f$

$$= 3 + \frac{(1.584)^2}{2\times 9.81} + 7.76$$

$$= 10.888 \text{ m}$$

Power required for the pump

$$P = \frac{wQH}{\eta}$$

$$= \frac{(9810\times 0.91)\times 7\times 10^{-3}\times 10.888}{0.60}$$

$$= 1134 \text{ W} = 1.134 \text{ kW}$$

Example 13.17. *Derive an expression for the power lost in a footstep bearing shown in Fig. Ex. 13.17.*

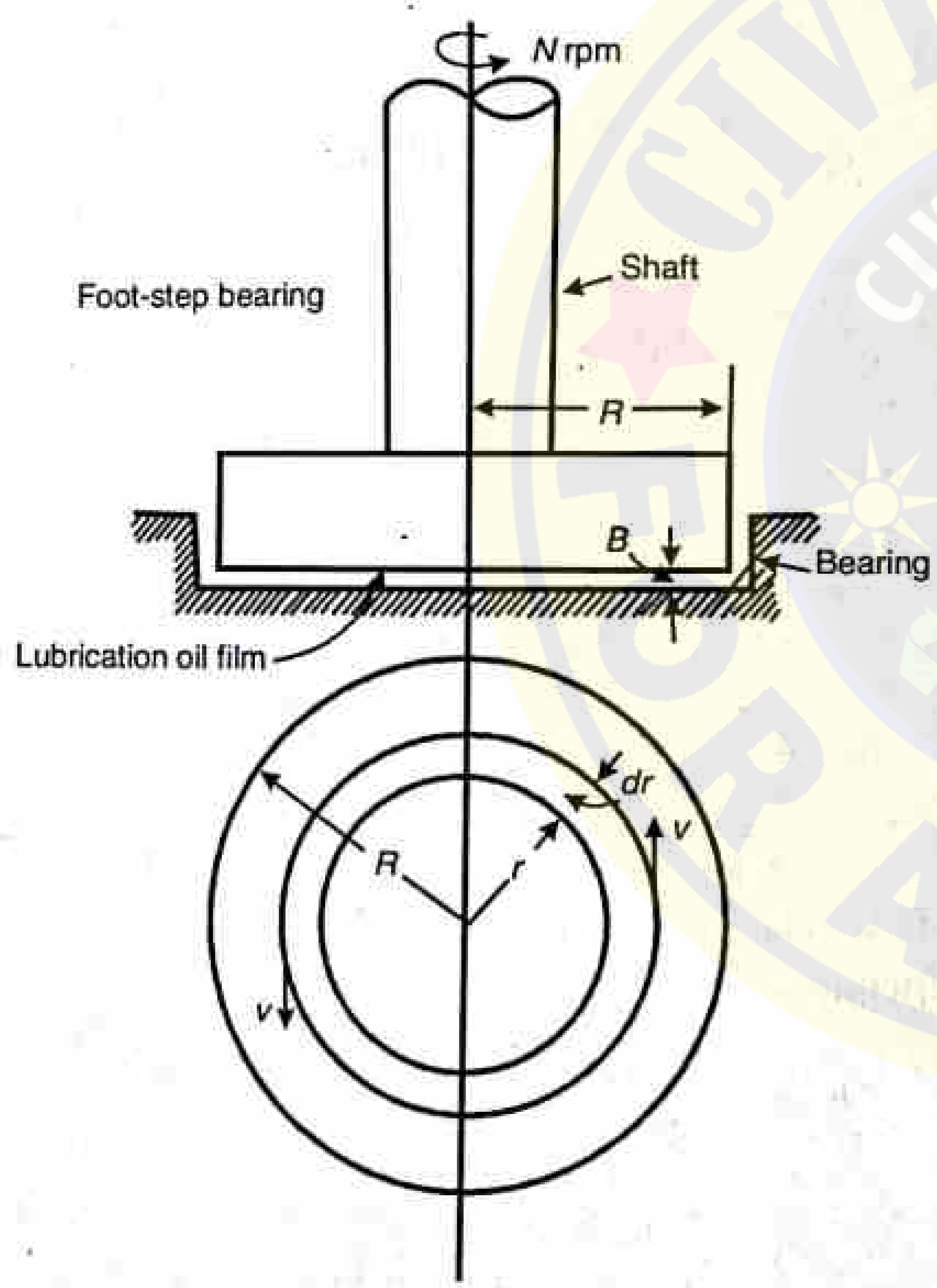


Figure Ex. 13.17

Solution

Since the lubrication film thickness B is generally very small, in this portion a linear velocity distribution may be assumed, or in other words plain (or simple) Couette flow may be considered for which from Eq. 13.45, we have

$$v = V\frac{y}{B}; \text{ or } \frac{\partial v}{\partial y} = \frac{V}{B}$$

where V is the peripheral velocity of the shaft base at a radial distance r.

If the shaft is rotating at a speed of N r.p.m. then

$$V = \frac{2\pi Nr}{60}$$

Considering an elemental ring of radius r and thickness dr the shear stress at the ring.

$$\tau = \mu\frac{\partial v}{\partial y} = \mu\frac{V}{B} = \frac{2\pi Nr\mu}{60\times B}$$

Shear force on the ring $= \tau \times 2\pi r\, dr$

$$= \frac{4\pi^2 Nr^2\mu dr}{60\times B}$$

Torque on ring = Shear force × radius

$$= \frac{4\pi^2 Nr^3\mu dr}{60\times B}$$

Total torque required to overcome the viscous resistance of oil

$$T = \int_0^R \frac{4\pi^2 Nr^3\mu dr}{60\times B} = \frac{\pi^2 N\mu R^4}{60\times B}$$

$\therefore$ Power lost in the bearing $= \dfrac{2\pi NT}{60\times 1000}$ kW

$$= \frac{2\pi^3 N^2\mu R^4}{B\times 3600\times 1000} \text{ kW}$$

SUMMARY OF MAIN POINTS

1. In laminar flow the fluid particles move along straight parallel paths in layers or laminae, such that the paths of the individual fluid particles do not cross those of the neighbouring particles. It occurs at low velocity so that forces due to viscosity predominate over the inertial forces and hence it is also known as *viscous flow*.
2. A flow in a pipe will be Laminar if the Reynolds number Re $\left(= \frac{\rho VD}{\mu}\right)$ is less than 2000.
3. For laminar flow of fluid through pipes shear stress distribution is given by

$$\tau = -\frac{\partial p}{\partial x}\frac{r}{2}$$

where

τ = shear stress at a radial distance r from the pipe axis; and

$\frac{\partial p}{\partial x}$ = pressure gradient.

The shear stress τ varies linearly along the radius of the pipe. Thus at the pipe axis since $r = 0$, $\tau = 0$, and at the pipe surface since $r = R$, the shear stress τ_0 is given by

$$\tau_0 = -\frac{\partial p}{\partial x}\frac{R}{2}$$

4. For laminar flow of fluid through pipes the velocity distribution is given by

$$V = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)(R^2 - r^2)$$

The velocity distribution in laminar flow is parabolic with velocity $V = 0$ at the pipe surface since $r = R$, and at the axis of the pipe since $r = 0$, $V = V_{max}$ given by

$$V_{max} = \frac{1}{4\mu}\left(-\frac{\partial p}{\partial x}\right)R^2$$

The mean velocity of flow V, which is equal to (Q/A) or $(Q/\lambda R^2)$ is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)R^2$$

Thus

$$V = \frac{1}{2}v_{max}$$

The point where the local velocity is equal to the mean velocity is given by

$$r = 0.707R$$

where

R = radius of the pipe ; and

μ = dynamic viscosity of the folwing fluid.

5. For Laminar flow of fluid in a pipe the drop in pressure head h_f due to friction is given by Hagen–Poiseuille equation as

$$h_f = \frac{p_1 - p_2}{w} = \frac{32\mu VL}{wD^2}$$

or

$$h_f = \frac{64}{Re}\left(\frac{LV^2}{2gD}\right)$$

where

Re = Reynolds number = $\frac{\rho VD}{\mu}$; and

w = specific wieght of the flowing fluid.

The friction factor f for laminar flow in a pipe is given by

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

The shear velocity $V_* \left(= \sqrt{\tau_0/\rho}\right)$

$$V_* = V\sqrt{\frac{f}{8}}$$

6. For laminar flow of fluid through inclined pipes the various expressions may be obtained by considering the piezometric head gradient ($\partial h/\partial x$) in place of pressure gradient ($\partial p/\partial x$).

7. Mean velocity of laminar flow through annulus of radius of the inner surface of the outer tube R_1 and radius of the outer surface of the inner tube R_2 is given by

$$V = \frac{1}{8\mu}\left(-\frac{\partial p}{\partial x}\right)\left[R_1^2 + R_2^2 - \frac{(R_1^2 - R_2^2)}{\log_e(R_1/R_2)}\right]$$

8. For laminar flow of fluid between two parallel flat plates located at a distance B apart is given by

$$V = \frac{1}{2\mu}\left(-\frac{\partial p}{\partial x}\right)(By - y^2)$$

The maximum velocity v_{max} which occurs at the mid-point between the plates is obtained by substituting $y = \frac{B}{2}$, as

$$v_{max} = \frac{B^2}{8\mu}\left(-\frac{\partial p}{\partial x}\right)$$

The mean velocity of flow V is given by

$$V = \frac{B^2}{12\mu}\left(-\frac{\partial p}{\partial x}\right)$$

Thus

$$V = \frac{2}{3}v_{max}$$

9. For laminar flow of fluid between two parallel flat plates the drop in pressure head h_f due to friction is given by

$$h_f = \frac{p_1 - p_2}{w} = \frac{12\mu VL}{wB^2}$$

10. The distribution of shear stress at any section for laminar flow of fluid between two parallel flat plates is given by

$$\tau = \left(-\frac{\partial p}{\partial x}\right)\left(\frac{B}{2} - y\right)$$

The shear stress varies linearly with the distance from the plate. It has the same maximum value at either plate i.e., at $y = 0$ or $y = B$ which is given by

$$\tau_0 = \left(-\frac{\partial p}{\partial x}\right)\frac{B}{2}$$

The shear stress decreases linearly with the distance from the plate and it is equal to zero at the centre line between the plates.

11. The laminar flow of fluic between two parallel flat plates with one plate moving and the other being at rest is known as Couette flow for which the velocity distribution is given by

$$v = \frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)$$

where

V = velocity of the moving plate.

When there is no pressure gradient in the direction of flow i.e., $\left(\frac{\partial p}{\partial x} = 0\right)$ it is known as *simple (or plain) Couette flow* for which the velocity distribution is linear given by

$$v = V\frac{y}{B}$$

12. For laminar flow of fluid in a very wide open channel with depth of flow D and bed slope $S_0(=-\partial z/\partial x)$ the velocity distribution is given by

$$v = \frac{w}{\mu}\left(-\frac{\partial z}{\partial x}\right)\left(Dy - \frac{y^2}{2}\right)$$

The loss of head h_f between two sections of channel L distance apart is given by

$$h_f = \frac{3\mu VL}{wD^2}$$

13. For laminar flow of fluid through a porous media the velocity of flow of fluid V is given by Darcy's law as

$$V = Ki$$

where

i = hydraulic gradient ; and

K = coefficient of permeability

The loss of head for laminar flow of fluid through a porous media is given by

$$h_f = \frac{k\mu VL}{w\eta d_s^2}$$

where

η = effective ratio the area of pores A_p to the total area A i.e., $\eta = A_p/A$; and

d_s = grain diameter.

14. The resistance F_D experienced by a sphere of diameter D moving with a constant velocity V in a fluid of velocity μ is given by Stokes' law as

$$F_D = 3\pi\mu VD$$

15. For Journal Bearing velocity, shear force, torque and power absorbed are given as

Velocity $$V = \frac{\pi DN}{60} = \frac{\partial v}{\partial y} = \frac{V}{t} = \frac{\pi DN}{60t}$$

Shear stress $$\tau = \mu\frac{\partial v}{\partial y} = \frac{\mu\pi DN}{60t}$$

Shear force $$F = \frac{\mu\pi^2 D^3 NL}{60t}$$

Torque $$T = \frac{\mu\pi^2 D^3 NL}{2 \times 60t}$$

Power $$P = \frac{\mu\pi^3 D^3 NL}{60 \times 60t}$$

where

L = Length of bearing;

D = diameter of shaft;

N = speed of shaft; and

t = clearance between shaft and bearing.

16. For Foot Step Bearing shear force, torque and power absorbed are given as

Shear force $$F = \frac{\mu}{15}\frac{\pi^2 N}{t}\frac{R^3}{3}$$

$$\text{Torque } T = \frac{\mu}{60t}\pi^2 NR^4$$

$$\text{Power } P = \frac{\mu\pi^3 N^2 R^4}{60\times 30\times t}$$

where

R = radius of the shaft; and

N = speed of the shaft.

17. For collar bearing torque and power absorbed are given as

$$\text{Torque } T = \frac{\mu}{60t}\pi^2 N\left(R_2^4 - R_1^4\right)$$

$$\text{Power } P = \frac{\mu\pi^3 N^2}{60\times 30t}\left(R_2^4 - R_1^4\right)$$

where

R_1 = internal radius of the collar;

R_2 = external radius of the collar;

t = thickness of oil film; and

P = power in watts.

18. A dash-pot is a device used for damping vibrations of machines, which may be achieved by making use of a fluid of fairly high viscosity.

19. The coefficient of viscosity of a liquid may be determined experimentally by the following methods:

(i) Coaxial cylinder viscometer

$$\mu = \frac{60(R_2 - R_1)\alpha T}{\pi^2 R_1^2 N\left[4\alpha h R_2 + R_1^2(R_2 - R_1)\right]}$$

where

R_1 = radius of inner stationary cylinder;

R_2 = radius of outer rotating cylinder;

h = depth of liquid in the cylinder;

α = clearance between the two cylinders at the bottom;

N = speed of rotation; and

T = torque

(ii) Capillary tube viscometer

$$\mu = \frac{\pi w H D^4}{128 QL}$$

where

w = specific weight of liquid;

H = head over capillary tube;

D = diameter of capillary tube;

L = length of capillary tube; and

Q = discharge through capillary tube.

(iii) Falling sphere viscometer

$$\mu = \frac{D^2}{18V}(w_s - w)$$

where

D = diameter of the spherical ball;

V = velocity of falling ball;

w_s = specific weight of ball; and

w = specific weight of the liquid.

PROBLEMS

13.1 Derive an expression for mean velocity for laminar flow (i) through a pipe; (ii) between parallel plates.

13.2 A horizontal pipe 50 mm diameter carrying glycerine has shear stress at the pipe boundary as 196.2 N/m² [20 kg (f)/m²]. Determine the pressure gradient, mean velocity and Reynolds number. Take for glycerine r = 1275.3 kg/m³ (130 msl/m³) and $\mu = 80.442 \times 10^{-2}$ N.s/m² [8.20 × 10^{-2} kg(f)–s/m²].

[*Ans.* 15.696 kN/m² per m (1600 kg(f)/m² per m); 1.52 m/s; 120.5]

13.3 A steady flow of oil (sp. gr. 0.85) occurs in a 0.1 m diameter pipe which has a downward slope of 1 in 100. If the discharge measured is 2 litres per second, determine the pressure drop between two points on the pipe 200 m apart. Take μ for oil as 1.2 poise.

[*Ans.* 2877 N/m³ (293.25 kg(f) /m³)]

13.4 Using Hagen-Poiseuille equation derive an expression for the head loss in a pipe of diameter D and length L in terms of Reynolds number and velocity head.

13.5 An oil having viscosity of 1.43 poise and specific gravity 0.9 flows through a pipe 25 mm diameter and 300 m long at 1/10 of the critical velocity for which Reynolds number is 2500. Find (a) the velocity of flow through the pipe; (b) the head in metres of oil across the pipe length required to maintain the flow; and (c) the power of the flow.

[*Ans.* (a) 1.59 m/s; (b) 396 m of oil; (c) 2.729 kW (3.71 h.p.)]

13.6 Two parallel plates kept 75 mm apart have laminar flow of glycerine between them with a maximum velocity of 1 m/s. Calculate the discharge per metre width, the shear stress at the plates, the difference in pressure in pascals (or N/m²) between two points 25 m apart, the velocity gradients at the plates and velocity at 15 mm from the plate. Take viscosity of glycerine as 8.35 poise.

[*Ans.* 0.05 m³/s per m; 44.53 N/m² (4.54 kg(f)/m²);29.69 kN/m² (0.303 kg(f)/cm²); 53.3 s^{-1}; 0.64 m/s]

13.7 Two parallel plates are placed horizontally 10 mm apart. The bottom plate is fixed and the top plate is moved at a uniform speed of 0.25 m/s. The fluid between them has a dynamic viscosity μ equal to 1.472 N.s/m² [0.15 kg(f)-s/m²]. Determine the pressure gradient which corresponds to the condition of zero discharge between the plates and the shearing stress at each plate. [**Hint.** For zero discharge

$$q = \int_0^B \left[\frac{V}{B}y - \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right)(By - y^2)\right]dy = 0$$

[*Ans.* 22.08 kN/m² per m (2.25 × 10³ kg(f)/m² per m);– 73.6 N/m² (–7.5 kg(f)/m²); 147.2 N/m² (15 kg(f)/m²]

13.8 Lubricating oil of specific gravity 0.90 and viscosity 6.0 poise is pumped at the rate of 0.02 m³/s through a 0.2 m diameter of the pipe 400 m long. Determine the slope of the pipe and the direction of flow so that the pressure gradient along the pipe is zero.

[*Ans.* 1 in 28.89; downward direction]

13.9 A horizontal column of sand 1 m long and 0.1 m in diameter has water flowing through it. If the discharge passing through it is 5 litres/min and the transmission constant of the material K is 4.5 mm/s, determine the pressure drop across the column.

[*Ans.* 23.131 kN/m² (2358 kg(f)/m²)]

13.10 In a rotary viscometer the radii of the cylinders are respectively 50 mm and 50.5 mm, and the outer cylinder is rotated steadily at 30 rad/s. For a certain liquid the torque is 0.4503 N.m [0.0459 kg(f)-m] when the depth of the liquid is 0.05 m and the torque is 0.8103 N.m [0.0826 kg(f)-m] when the depth is 0.1 m. Estimate the viscosity of the liquid.

[*Ans.* 0.1513 N.s/m² (0.0154 kg(f)-s/m²)]

13.11 A small lead sphere of specific gravity 11.5 has a terminal fall velocity of 7 mm/s in a fluid of specific gravity 1.25 and dynamic viscosity 7.848 poise. Determine the diameter of the sphere.

[*Ans.* 1 mm]

13.12 Show that when laminar flow occurs with mean velocity V between extensive stationary flat plates, the mean kinetic energy per unit weight of the fluid is (1.543 $V^2/2g$).

13.13 In a Saybolt universal viscometer, the time for the flow of 60 cm³ of a liquid of specific weight 8.633 kN/m³ [880 kg(f)/m³] is 170 seconds. Calculate the absolute viscosity of the liquid.

[*Ans.* 3.198 × 10^{-2} N.s/m² (3.26 × 10^{-3} kg(f) – s/m²)]

13.14 A force of 2.453 N [0.25 kg(f)] is required to drag a thin plate at a constant velocity of 0.3 m/s between two fixed plates when the space between these two fixed plates is filled with a liquid. If the area of the plate is 0.1 m², it travels equidistant from and parallel to the fixed plates, and the distance from the moving plate to the fixed plates is 3 mm, calculate the viscosity of the liquid.

[*Ans.* 1.227 × 10^{-2} N.s/m² (1.25 × 10^{-3} kg(f) – s/m²)]

13.15 A uniform circular tube of bore radius a has a fixed coaxial cylindrical core of radius b. An incompressible viscous fluid flows through the annular passage under a pressure gradient $(-\partial p/\partial x)$. Determine the radius at which the shear stress in the stream is zero, given that the flow is laminar under steady state condition.

$$\left[\textit{Ans.}\left\{\frac{a^2 - b^2}{2\log_e(a/b)}\right\}^{1/2}\right]$$

13.16 A horizontal circular tube of radius a has a fixed coaxial cylindrical rigid core of radius b. τ_a and τ_b are the shear stresses along the tube and core surfaces when a viscous liquid is flowing through the annulus. The flow is laminar and the rate of variation of pressure along the length of the passage is $(-\partial p/\partial x)$. Show that $a\tau_a - b\tau_b = \frac{1}{2}(a^2 - b^2)\left(\frac{\partial p}{\partial x}\right)$.

Turbulent Flow in Pipes

Chapter **14**

14.1 INTRODUCTION

Mostly the flow in pipes is turbulent. As stated earlier if Reynolds number is greater than 4000 the flow is turbulent. The velocity distribution in turbulent flow is relatively uniform and the velocity profile of turbulent flow is much flatter than the corresponding laminar flow parabola for the same mean velocity. It becomes even flatter with increasing Reynolds number as shown in Fig. 14.1 in which a comparison between velocity profiles for three different values of Reynolds number Re = 10^3, 10^4 and 10^6 is shown. In turbulent flow the fluid particles are in an extreme state of disorder, their movement is haphazard and large scale eddies are developed, which results in a complete mixing of the fluid. Such irregular motion of fluid particles in turbulent flow is on account of the fact that at any fixed point in space, the velocity and pressure do not remain constant with time, but fluctuate in irregular manner. In other words, in turbulent flow there are irregular velocity and pressure fluctuations of high frequency superimposed on the main flow. On account of these fluctuations the analytical treatment of turbulent flow is rendered extremely complicated. No complete theory has yet been developed for the analysis of turbulent flow and one has to depend at some point on experimentally derived information. However, in turbulent flow also the mean components of velocity and pressure may be considered, which may be assumed to have a constant value at any section if the average are taken over a sufficiently long interval of time.

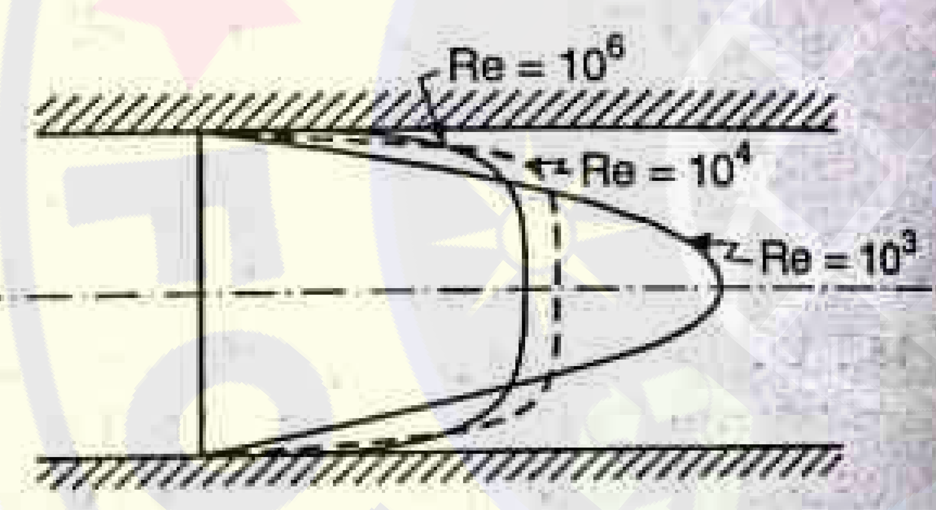


Figure 14.1 *Velocity distribution in laminar and turbulent flow*

In the case of turbulent flow the velocity fluctuations influence the mean motion in such a way that an additional shear (or frictional) resistance to flow is caused. This shear stress produced in turbulent flow is in addition to the viscous shear stress and it is termed as turbulent shear stress which may be evaluated as explained in the next section.

14.2 SHEAR STRESSES IN TURBULENT FLOW

As stated earlier, in turbulent flow velocity fluctuations cause a continuous interchange of fluid masses between the neighbouring layers, which is accompanied by a transfer of momentum. A large

number of lumps of fluid particles move from one fluid layer into the adjacent layers above and below and get merged in them. Since each fluid layer possesses a different velocity, such movement of fluid particles brings about a transfer of momentum from one layer to another. But the change of momentum is equivalent to the force in a particular direction. Hence such momentum transport due to fluctuations results in developing additional shear stresses of high magnitude between adjacent layers. In order to determine the magnitude of the turbulent shear stress a number of semiempirical theories have been developed, some of which are discussed below.

In analogy with the expression for the viscous shear. J. Boussinesq, a French mathematician in 1877 developed an expression for the turbulent shear stress which may be expressed as

$$\tau = \eta \frac{d\bar{v}}{dy} = \rho\varepsilon \frac{d\bar{v}}{dy} \quad \text{...(14.1)}$$

in which $\bar{v}$ denotes the average velocity at a distance y from boundary. The turbulent mixing coefficient η (Greek 'eta') introduced by Boussinesq which is analogous to the absolute viscosity μ, is often called 'apparent' or 'virtual' or 'eddy' viscosity. Similar to kinematic viscosity $\nu = \left(\frac{\mu}{\rho}\right)$, the 'apparent' or 'virtual' or 'eddy' kinematic viscosity ε (Greek 'epsilon') is also obtained by dividing eddy viscosity η, by the mass density of the fluid ρ, thus

$$\varepsilon = \frac{\eta}{\rho} \quad \text{...(14.2)}$$

When viscous action is also included the total shear stress may be expressed as

$$\tau = \mu\frac{d\bar{v}}{dy} + \eta\frac{d\bar{v}}{dy} = (\mu+\eta)\frac{d\bar{v}}{dy} \quad \text{...(14.3)}$$

The magnitude of η may vary from zero (if the flow is laminar) to several thousand times of μ. The value of η depends on the momentum carried by the migrating particles and thus on the density of the flowing fluid and the characteristics of flow. Further the kinematic eddy viscosity $\varepsilon = \left(\frac{\eta}{\rho}\right)$ may be considered to be independent of the properties of the fluid but depends on the characteristics of flow. Thus whereas μ is a fluid property and is a function of temperature of fluid alone, η and ε are mainly the functions of the characteristics of flow, and may be expected to vary from point to point in the flow. Since the values of η and ε cannot be predicted, the Boussinesq's hypothesis is, however, of limited use.

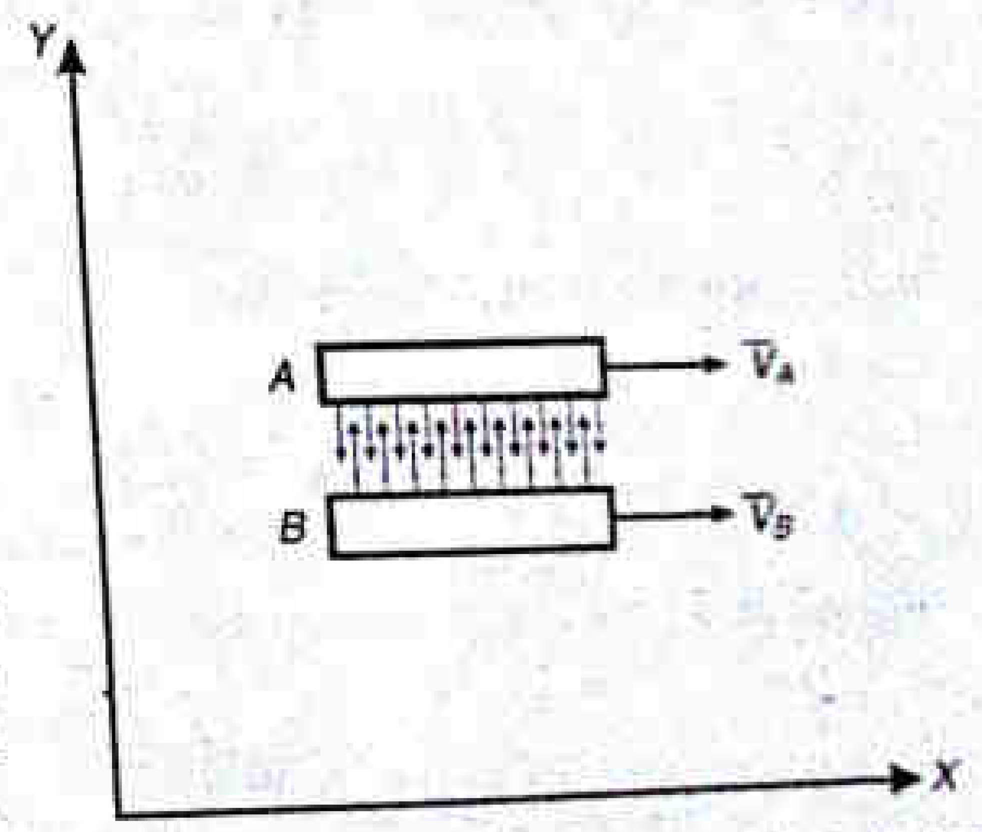


Figure 14.2 *Transfer of momentum in turbulent flow*

In 1886 Reynolds developed an expression for the turbulent shear stress which may be derived as follows. Consider two fluid layers A and B at a small distance apart as shown in Fig. 14.2, having the average velocities $\bar{V}_A$ and $\bar{V}_B$ respectively. It is assumed that $\bar{V}_A > \bar{V}_B$ and the relative velocity of layer A with respect of layer B is $(\bar{V}_A - \bar{V}_B) = v_x$; where v_x is the fluctuating component of velocity in the x-direction due to turbulence. It then means that if layer B is assumed to be stationary then layer A may be assumed to be moving at a uniform velocity v_x. Further in the y-direction let the fluctuating component be v_y. Now if over a surface of area A perpendicular to the y-direction and separating two adjacent fluid layers the component v_y are uniformly distributed, then the mass of fluid transferred across that surface from one layer to the other per second is $(\rho A v_y)$. Since this mass of fluid has started moving with a relative velocity v_x, a transfer of momentum equal to $(\rho A v_y v_x)$ takes place, resulting in developing tangential forces on each of the layers. The corresponding turbulent shear stress exerted on the fluid layers is equal to

$$\tau = \frac{\text{Force}}{\text{Area}} = \frac{\rho A v_y v_x}{A} = (\rho v_y v_x) \quad \text{...(14.4)}$$

Since v_x and v_y are varying, the magnitude of τ will also vary. Hence usually the time average value of shear stress is considered. Thus by taking the time average on both sides of Eq.14.4 it becomes

$$\bar{\tau} = \overline{\rho v_y v_x} \quad \text{...(14.5)}$$

The stress represented by Eq. 14.5 is usually termed as Reynolds stress.

In 1925 L. Prandtl, a German Engineer made an important advance in this direction by presenting mixing length hypothesis, by means of which the turbulent shear stress can be expressed in terms of measurable quantities related to the average flow characteristics.

According to Prandtl, mixing length is that distance in the transverse direction which must be covered by a lump of fluid particles travelling with its original mean velocity in order to make the difference between its velocity and the velocity of the new layer equal to the mean transverse fluctuation in turbulent flow. Prandtl has further indicated that the velocity fluctuation in the x-direction, v_x may be related to the mixing length l by the following expression

$$v_x = l\frac{d\bar{v}}{dy} \quad \text{...(14.6)}$$

Further the fluctuation component v_y is of the same order of magnitude as v_x and can be written as

$$v_y = l\frac{d\bar{v}}{dy} \quad \text{...(14.7)}$$

and hence $\overline{v_x v_y}$ can be written as

$$\overline{v_x v_y} = l^2\left(\frac{d\bar{v}}{dy}\right)^2 \quad \text{...(14.8)}$$

From Eqs. 14.5 and 14.8 the expression for turbulent shear stress becomes

$$\bar{\tau} = \rho l^2\left(\frac{d\bar{v}}{dy}\right)^2 \quad \text{...(14.9)}$$

Again the total shear stress at any point is the sum of the viscous shear stress and turbulent shear stress and it may be expressed as

$$\bar{\tau} = \mu\frac{d\bar{v}}{dy} + \rho l^2\left(\frac{d\bar{v}}{dy}\right)^2 \quad \text{...(14.10)}$$

However, the viscous shear stress is negligible as compared with the turbulent shear stress.

The superiority of Eq. 14.9 given by Prandtl lies in the fact that it is possible to make suitable assumptions regarding the variation of the mixing length. In several problems of fluid flow it is possible to establish a simple relation between l and a characteristic length of the flow and therefore Prandtl's Eq. 14.9 has been successfully applied to the various turbulent flow problems.

14.3 FORMATION OF BOUNDARY LAYER IN PIPES—ESTABLISHMENT OF FLOW IN PIPES

The concept of boundary layer formation along a flat plate, indicated in Chapter 12, may also be used in the case of flow of a fluid in a pipe. When a fluid flows through a pipe, close to the pipe wall the fluid is retarded, thus resulting in the formation of the boundary layer in the same manner as in the case of a plane boundary. However, the formation of boundary layer in a pipe is somewhat different from that on a flat plate. Whereas the boundary layer over a flat plate may expand indefinitely (provided the depth of flow and the length of the plate are infinite), the boundary layer in a pipe may attain a maximum thickness equal to the pipe radius. This is so because from the entrance section of the pipe the boundary layer gradually increases and at a certain section in the downstream when it attains a thickness equal to the radius of the pipe it cannot expand any more. Thus from this section onwards in the downstream direction the thickness of the boundary layer remains unchanged. With the increase in the thickness of the boundary layer from the entrance section, more and more fluid will be retarded in the downstream direction, and hence in order to maintain a constant discharge, the velocity in the central core will increase. As such, as shown in Fig. 14.3, in between the entrance section and the section AA, where the boundary layer attains thickness equal to the radius of the pipe, the velocity

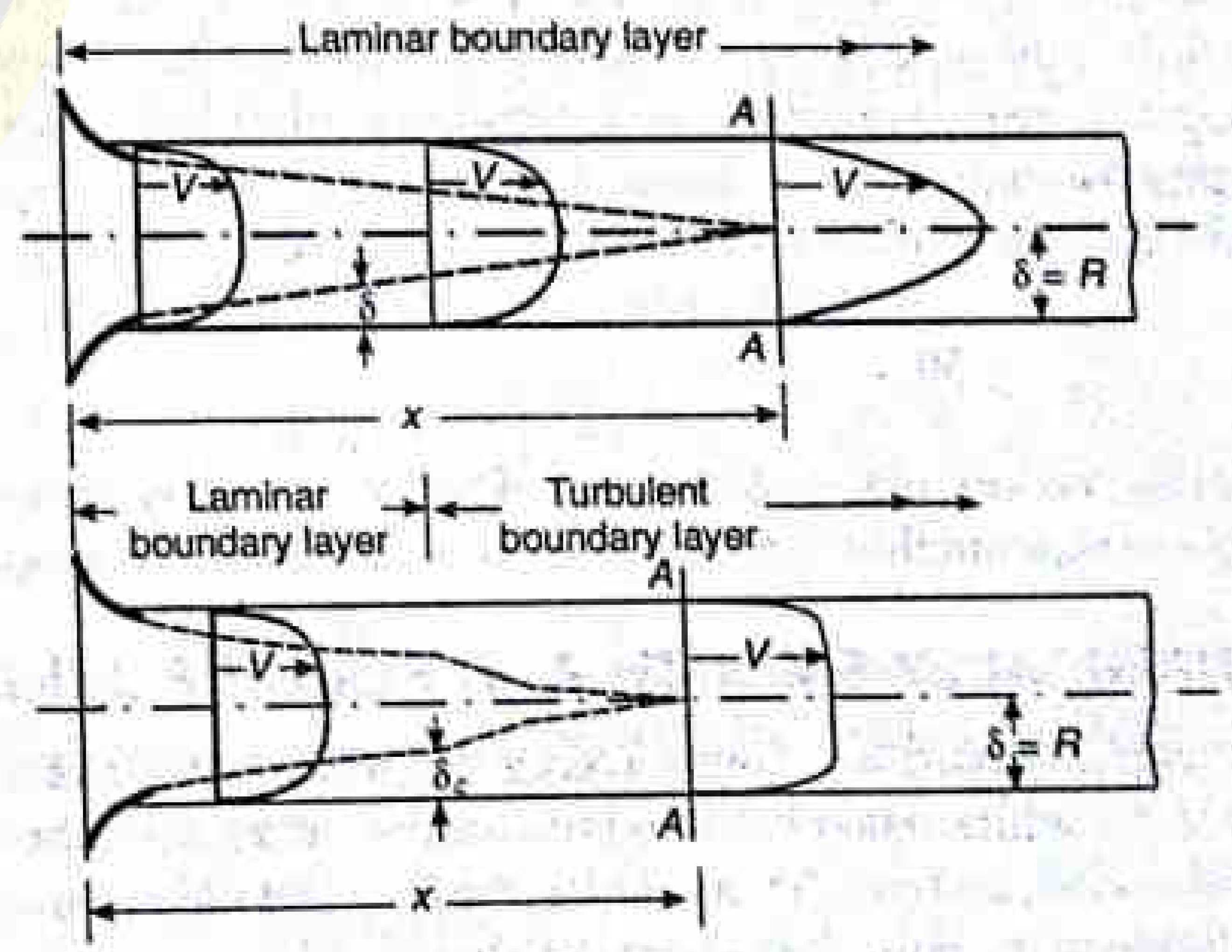


Figure 14.3 *Establishment of flow in a pipe*

distributions will vary from section to section due to the variation in the thickness of the boundary layer, but the velocity distributions at the section AA and at any other section on the downstream will be exactly the same. Accordingly it is only beyond section AA, the flow in the pipe will be truly uniform and the flow is then said to be established. The length of the pipe from the entrance section upto the section AA is termed as the length required for the establishment of flow in the pipe. As described below different lengths are required for the establishment of laminar and turbulent flows in pipes.

For laminar flow in a pipe, laminar boundary layer will be developed for the entire length of the pipe and as shown in Fig. 14.3(a), at section AA thickness of the boundary layer will become equal to the radius of pipe. Since laminar flow has parabolic velocity distribution, at section AA and downstream of it the velocity distributions will be parabolic as shown in Fig. 14.3(a). The length of pipe x, from the entrance of the pipe upto section AA, is therefore the length required for the establishment of fully developed laminar flow in the pipe. Experiments have shown that for laminar flow (x/D) is a function of Reynolds number $Re\left(=\frac{\rho VD}{\mu}\right)$ where D is diameter of the pipe, ρ and μ are mass density and viscosity of the flowing fluid and V is the mean velocity of flow in the pipe. Rouse has given the following relationship between (x/D) and Re for the establishment of laminar flow in pipes.

$$\frac{x}{D} = 0.07\,Re \qquad \text{...(14.11)}$$

If the flow in a pipe is turbulent, even then as shown in Fig. 14.3 (b), for a small distance from the entrance section, laminar boundary layer will be formed, which will change to turbulent boundary layer before the thickness of the boundary layer becomes equal to the radius of the pipe. However, in some cases, if the pipe is rough and the intensity of turbulence of the incoming flow is high, from the entrance section itself turbulent boundary layer may be formed. The thickness of the boundary layer will increase in the downstream direction upto a certain section AA [see Fig. 14.3 (b)] where it will become equal to the radius of the pipe. As such from section AA onwards there will exist a fully developed turbulent flow in the pipe. Since a turbulent flow has logarithmic velocity distribution, it is much more uniform and hence the length of pipe x, required for the establishment of fully developed turbulent flow in a pipe is relatively less. Rouse has indicated that the distance required for the establishment of a fully developed turbulent flow in a pipe is given by the following expression

$$\frac{x}{D} = 50 \qquad \text{...(14.12)}$$

It may be noted that the various expressions derived for laminar flow in pipes in Chapter 13 and those for turbulent flow in pipes in this chapter are applicable only for the established flows.

14.4 HYDRODYNAMICALLY SMOOTH AND ROUGH BOUNDARIES

In general a boundary with irregularities of large average height k, on its surface is considered to be a rough boundary and the one with smaller k values is considered as a smooth boundary. However for a proper classification of smooth and rough boundaries, the flow and fluid characteristics are required to be considered in addition to the boundary characteristics.

Consider a boundary such that the average height k, of the irregularities projecting from its surface is much less than the thickness of the laminar sublayer δ', Fig. 14.4. As the flow outside the laminar sublayer is turbulent, eddies of various sizes are present which try to penetrate through the laminar sublayer. But due to greater thickness of the laminar sublayer the eddies cannot reach the surface irregularities, and thus the boundary acts as a smooth boundary. Such a boundary is known as *hydrodynamically smooth boundary*.

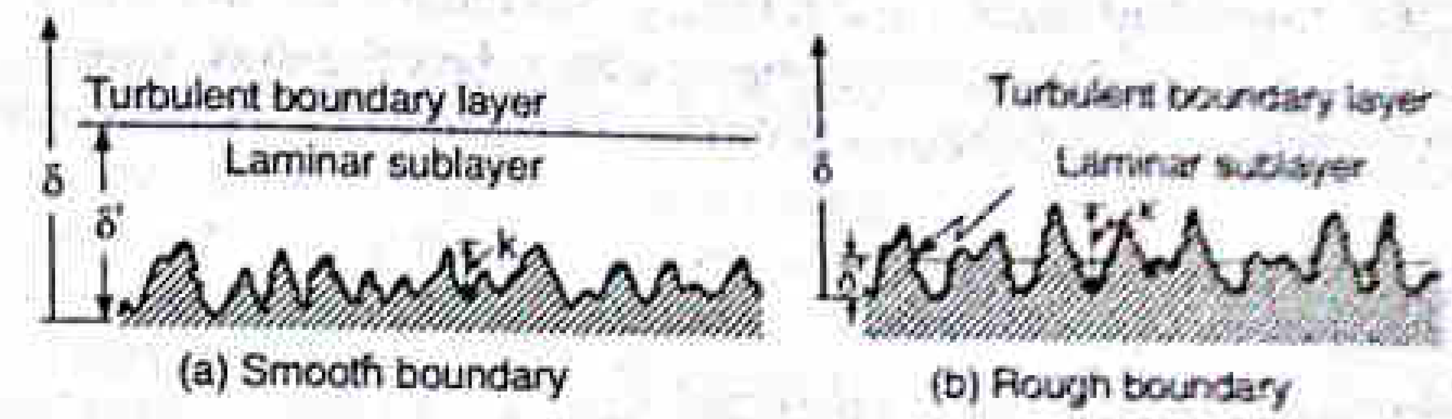


Figure 14.4 *Definition of smooth and rough boundaries*

With the increase in Reynolds number the thickness of the laminar sublayer decreases, and it can even become much smaller than the average height k, of surface irregularities. The irregularities will then project through the laminar sublayer and the laminar sublayer is completely destroyed. The eddies will thus come in contact with the surface irregularities and large amount of energy loss will take place such a boundary is known as *hydrodynamically rough boundary*.

As indicated later it has been found by Nikuradse's experiment that the boundary behaves as hydrodynamically smooth boundary when $(k//\delta')$ is less than 0.25 and as hydrodynamically rough when (k/δ') is greater than 6.0. The boundary is classified as boundary in transition for $0.25 < (k//\delta') < 6.0$.

14.5 VELOCITY DISTRIBUTION FOR TURBULENT FLOW IN PIPES

For turbulent flow in circular pipes Prandtl assumed the mixing length l to be linear function of the distance y from the pipe wall i.e.,

$$l \sim y$$

or

$$l = \kappa y \qquad \text{...(14.13)}$$

in which κ (Greek 'kappa') is the constant of proportionality and is the so called 'Karman universal constant'. The Prandtl's hypothesis $l = \kappa y$, has been confirmed for the regions very close to the pipe boundary, by the experimental studies of Prandtl's student. J. Nikuradse, who used smooth pipes as well as pipes artificially roughened by cemented coatings of sand grains of uniform size. The analysis of Nikuradse's experimental results has shown that in the immediate neighbourhood of the pipe boundary the expression for the mixing length is approximately

$$l = 0.4y - 0.44\left(\frac{y^2}{R}\right) \qquad \text{...(14.14)}$$

For small values of y, close to the pipe boundary, Eq. 14.14 reduces to

$$l = 0.4y \qquad \text{...(14.15)}$$

By comparing Eqs 14.13 and 14.15 the value of Karman Universal constant κ is found to be 0.4.

Now by substituting for the mixing length from Eq. 14.13 in Eq. 14.9 the turbulent shear stress may be expressed as

$$\tau = \rho\kappa^2 y^2\left(\frac{dv}{dy}\right)^2 \qquad \text{...(14.16)}$$

For the sake of convenience, the bars denoting the time average of the various quantities are omitted.

Since turbulent shear stress also varies linearly with radius (Chapter 13) it can be expressed as

$$\tau = \tau_0\left(\frac{r}{R}\right) = \tau_0\left(\frac{R-y}{R}\right) = \tau_0\left(1-\frac{y}{R}\right) \qquad ...(14.17)$$

where τ_0 represents the turbulent shear stress at the pipe boundary, i.e., at $y = 0$ or $r = R$.

However, for small values of y that is close to the pipe boundary the shear stress τ may be assumed to be constant, being approximately equal to τ_0 from Eq. 14.17. Thus Eq. 14.16 becomes

$$\tau_0 = \rho\kappa^2 y^2\left(\frac{dv}{dy}\right)^2 \qquad ...(14.18)$$

Equation 14.18 can be rearranged to solve for (dv/dy) as

$$\left(\frac{dv}{dy}\right) = \frac{1}{\kappa y}\left(\frac{\tau_0}{\rho}\right)^{1/2} = \frac{V_*}{\kappa y} \qquad ...(14.19)$$

since $(\tau_0/\rho)^{1/2} = V_*$, shear velocity or friction velocity (as explained in Chapter 13). For a given case of turbulent flow, V_* is constant and hence Eq. 14.19 may be integrated to yield the following relationship

$$v = \frac{V_*}{\kappa}\log_e y + C \qquad ...(14.20)$$

Equation 14.20 indicates that in turbulent flow the velocity varies directly with the logarithm of the distance from the boundary or the velocity distribution in turbulent flow is logarithmic in nature. The constant of integration C may be determined from the boundary condition at the centre of the pipe:

$$v = v_{max} \text{ (maximum velocity) at } y = R$$

$$C = v_{max} - \frac{V_*}{\kappa}\log_e R$$

The use of boundary condition at the centre of the pipe is, however, not strictly agreeable to Prandtl's original assumption that Eq. 14.19 is applicable only to the flow close to the pipe boundary. But Nikuradse's experimental studies have indicated that Eq. 14.19 holds good for the entire cross-section of the pipe. Thus by substituting the expression for C in Eq. 14.20 and using the value of $\kappa = 0.4$, the expression for velocity distribution becomes

$$v = v_{max} + 2.5V_*\log_e\left(\frac{y}{R}\right) \qquad ...(14.21)$$

Equation 14.21 is known as 'Prandtl's universal velocity distribution equation, for turbulent flow in pipes. In obtaining this equation no assumption has been made as to whether the pipe boundary is hydrodynamically smooth or rough. Hence Eq. 14.21 is applicable for both smooth and rough pipe boundaries. By dividing both sides of Eq. 14.21 by V_* and rearranging the terms, the equation may be expressed in a non-dimensional form:

$$\frac{v_{max} - v}{V_*} = 2.5\log_e\left(\frac{R}{y}\right) \qquad ...(14.22)$$

Changing from natural (to the base e) to common (to the base ten) logarithms Eq. 14.22 may be written as

$$\frac{v_{max} - v}{V_*} = 5.75\log_{10}\left(\frac{R}{y}\right) \qquad ...(14.23)$$

Fluid Mechanics.

The difference between the maximum velocity v_{max} and the local velocity at any point v, that is $(v_{max} - v)$ is commonly known as 'velocity defect'.

It is obvious from Eq. 14.22 that $\left[\frac{(v_{max} - v)}{V_*}\right]$ is a function of (y/R) alone. Hence the velocity distribution curve given by Eq. 14.22 appears to be independent of the nature of the boundary. However, the experiments conducted by Fritsch have indicated that the velocity distribution curves for smooth and rough pipes are identical over a substantial portion of the pipe except for a narrow region in the vicinity of the boundary, where the nature of the pipe boundary affects the velocity distribution curve. It may therefore be stated that Prandtl's equation [Eq. 14.22] applies only to the turbulent flow in the central region of the pipe. Moreover different velocity distribution equations will have to be derived for the hydrodynamically smooth and rough boundaries as discribed below.

14.6 VELOCITY DISTRIBUTION FOR TURBULENT FLOW IN HYDRODYNAMICALLY SMOOTH AND ROUGH PIPES—KARMAN PRANDTL VELOCITY DISTRIBUTION EQUATION

As derived in the previous section the velocity distribution for turbulent flow in pipes is given by Eq. 14.20 as

$$v = \frac{V_*}{\kappa}\log_e y + C \qquad ...(14.20)$$

This velocity distribution for turbulent flow has a peculiarity that at the boundary, that is for $y = 0$, it gives a velocity v equal to $-\infty$ (minus infinity). Hence it is only at a certain finite distance above the boundary, say, $y = y'$, that the velocity will be equal to zero as shown in Fig. 14.5. Thus with this condition that at $y = y'$ the velocity $v = 0$, the constant C can be obtained as

$$C = -\frac{V_*}{\kappa}\log_e y'$$

Introducing this value of the constant C in Eq. 14.20, it may be expressed as

$$v = \frac{V_*}{\kappa}\log_e\left(\frac{y}{y'}\right) \qquad ...(14.24)$$

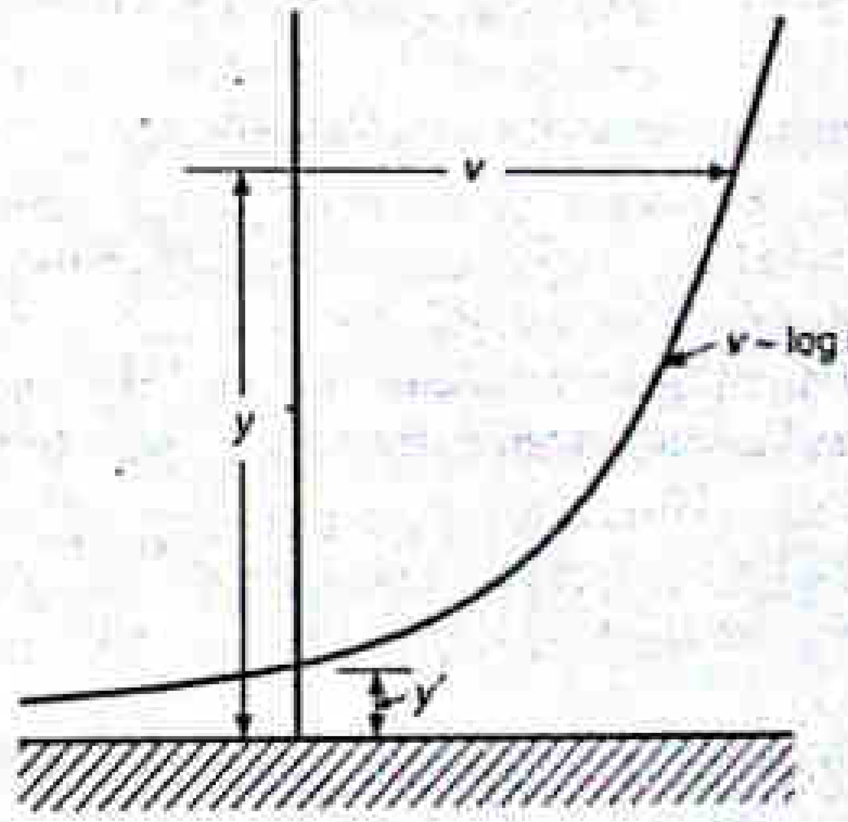


Figure 14.5. *Logarithmic velocity distribution curve for turbulent flow*

The tendency of the logarithmic velocity distribution curve to indicate a zero velocity at a certain distance above boundary is, however, in disagreement with the physical fact that the velocity at the boundary is zero. As such Eq. 14.20 for logarithmic velocity distribution applies only to turbulent flow in the central region of the pipe, and it cannot be applied to the regions close to the pipe boundary.

(a) Velocity Distribution in Smooth Pipes. In the vicinity of a smooth boundary there exists a laminar sublayer. The flow in the laminar sublayer being laminar has a parabolic velocity distribution.

As such the velocity which is zero at the pipe boundary increases parabolically in the zone of laminar motion, which extends upto certain distance from the boundary. Above the zone of laminar motion there exists a transition zone where the flow changes from laminar to turbulent. Beyond the transition zone the flow is turbulent having logarithmic velocity distribution. Since the change from parabolic to logarithmic distribution is gradual, the zone of laminar motion will extend well beyond the distance y', as shown in Fig. 14.6. However, in the absence of any specific line of demarcation between the different zones of flow near the boundary, the intersection of the parabolic and the logarithmic velocity distribution curves, as shown in Fig. 14.6 is arbitrarily chosen as nominal border line between the two types of flow. Moreover the same point of intersection between the two curves is arbitrarily taken as the limit of laminar sublayer. Since the thickness of the laminar sublayer δ' is generally very small, the parabolic velocity distribution in this region may be approximated by a straight line without appreciable error, as shown in Fig. 14.6.

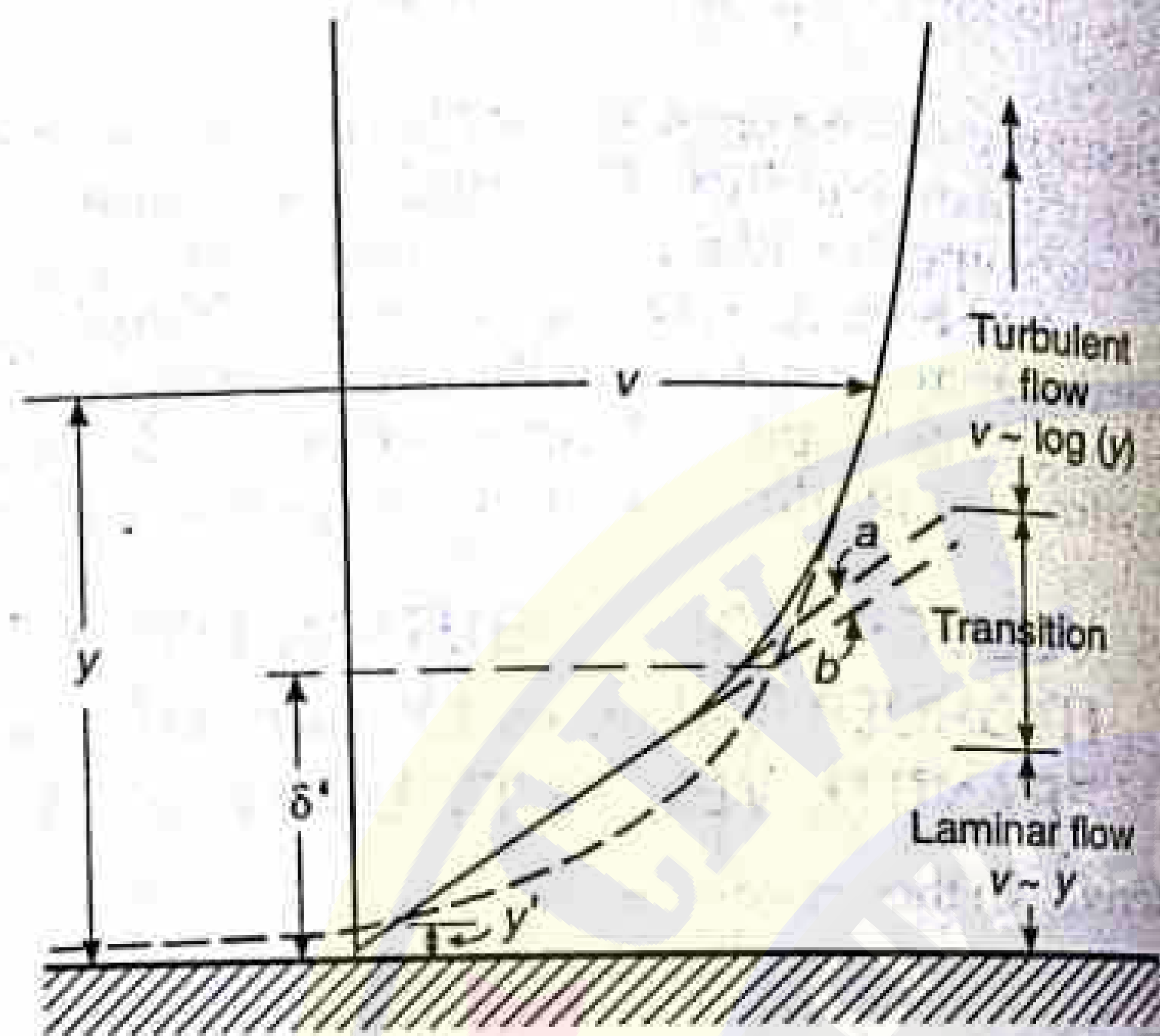


Figure 14.6 *Velocity distribution for turbulent flow near a smooth boundary*

In the zone of laminar sublayer, since the flow is laminar, the viscous stress predominate and the turbulent stresses tend to become zero. Therefore, in the laminar sublayer the shear stress $\tau = \mu(dv/dy)$.

For linear velocity distribution within the laminar sublayer, the velocity gradient (dv/dy) becomes (v/y). Furthermore, if it is assumed that in the laminar sublayer, that is upto $y = \delta'$, the shear stress τ remains constant equal to τ_0, the shear stress at the pipe boundary, then

$$\tau = \tau_0 = \mu \frac{v}{y} \qquad \text{...(14.25)}$$

By dividing both sides of Eq. 14.25 by mass density ρ and replacing the ratio (τ_0/ρ) by V_*^2 Eq. 14.25 may be rearranged in the following non-dimensional form:

$$\frac{v}{V_*} = \frac{\rho V_* y}{\mu}$$

or

$$\frac{v}{V_*} = \frac{V_* y}{\upsilon} \; [\text{since } (\mu/\rho) = \upsilon\,] \qquad \text{...(14.26)}$$

Equation 14.26 thus provides the velocity distribution within laminar sublayer, that is from $y = 0$ to $y = \delta'$. The term $(V_* y/\upsilon)$ on the right hand side of Eq. 14.26 is dimensionally equivalent to a form of Reynolds number.

Nikuradse's experimental studies of turbulent flow in smooth pipes have shown that for turbulent flow in smooth pipes of any size the value of the parameter $(V_* y/\upsilon)$ for $y = \delta'$ is approximately 11.6 and for $y = y'$ it is approximately 0.108. That means

$$\frac{V_* \delta'}{\upsilon} = 11.6$$

or

$$\delta' = \frac{11.6\upsilon}{V_*} \qquad \text{...(14.27)}$$

and

$$\frac{V_* y'}{\upsilon} = 0.108$$

or

$$y' = \frac{0.108\upsilon}{V_*} = \frac{\delta'}{107} \qquad \text{...(14.28)}$$

Now by substituting the value of y' given by Eq. 14.28 in Eq. 14.24 the following expression is obtained

$$v = \frac{V_*}{\kappa} \log_e \left(\frac{V_* y}{0.108\upsilon}\right) \qquad \text{...(14.29)}$$

Changing from natural (to the base e) to common (to the base ten) logarithms and using the value of $\kappa = 0.4$, Eq. 14.29 may be written as

$$v = 5.75 V_* \log_{10} \left(\frac{V_* y}{0.108\upsilon}\right) \qquad \text{...(14.30)}$$

By dividing both sides of Eq. 14.30 by V_* it may be rearranged in the following form

$$\frac{v}{V_*} = 5.75\, V_* \log_{10} \left(\frac{V_* y}{\upsilon}\right) + 5.5 \qquad \text{...(14.31)}$$

Equation 14.31 represents the velocity distribution in the region of turbulent flow near hydrodynamically smooth boundaries and it is applicable for regions outside the laminar sublayer that is for y equal to or greater than δ'. It is known as Karman-Prandtl equation for the velocity distribution near hydrodynamically smooth boundaries. Though Eq. 14.31 has been derived to represent the velocity distribution near smooth boundaries only, but it has been found that the experimental data of Nikuradse on velocity measurements in hydrodynamically smooth pipes follow closely the trend of Eq. 14.31, even for the distances farther away from the pipe boundary. As such Eq. 14.31 is used to represent the velocity distribution in the entire region of turbulent flow in the pipe i.e., $y = \delta'$ to $y = R$. Therefore Eq. 14.31 may be considered as Karman–Prandtl equation for velocity distribution for turbulent flow in hydrodynamically smooth pipes.

The velocity distribution for turbulent flow in smooth pipe may also be represented by an exponential equation instead of the logarithmic equation. It has been found empirically that for $(V_* y/v)$ between 70 and 700, Equation 14.31 for turbulent flow in smooth pipes may be approximated by

$$\frac{v}{V_*} = 8.74 \left(\frac{V_* y}{\upsilon}\right)^{1/7} \qquad \text{...(14.32)}$$

Further at the centreline of the pipe i.e., for $y = R$ the velocity v will be equal to v_{max}. Then from Eq. 14.32

$$\frac{v_{max}}{V_*} = 8.74\left(\frac{V_* R}{\upsilon}\right)^{1/7} \quad ...(14.33)$$

By dividing Eq. 14.32 by Eq. 14.33 the so called 'Blasius one-seventh power velocity distribution equation' is obtained as

$$\frac{v}{v_{max}} = \left(\frac{y}{R}\right)^{1/7} = \left(1-\frac{r}{R}\right)^{1/7} \quad ...(14.34)$$

since $y = (R-r)$

However, in a general form the exponential equation for velocity distribution for turbulent flow in smooth pipes may be expressed as

$$\frac{v}{v_{max}} = \left(\frac{y}{R}\right)^{1/n} = \left(1-\frac{r}{R}\right)^{1/n} \quad ...(14.35)$$

where exponent $(1/n)$ decreases with increasing Reynolds number, but tends to approach a constant numerical value asymptotically for very large Reynolds numbers.

(b) Velocity Distribution in Rough Pipes. The flow conditions near hydrodynamically rough boundaries are different from those near hydrodynamically smooth boundaries. This is so because in the case of rough boundaries the surface irregularities protrude well beyond the laminar sublayer, which is therefore completely destroyed, as shown in Fig. 14.7. As such velocity distribution in turbulent flow near hydrodynamically rough boundaries is considerably affected by the surface protrusions. Equation 14.24 applies equally to the turbulent flow in hydrodynamically rough pipes. But in order to obtain an equation which would represent the velocity distribution for turbulent flow near hydrodynamically rough boundaries, the distance y' must be evaluated in terms of the average height of the surface protrusions k. Again from the experiments by Nikuradse and others, using pipes artificially roughened by cemented coatings of sand grains of diameter k, it has been found that y' is directly proportional to k and $y' = (k/30)$. Introducing this value of y' in Eq. 14.24 and using $\kappa = 0.4$, the following expression is obtained

$$v = \frac{V_*}{0.4}\log_e\left(\frac{30y}{k}\right) \quad ...(14.36)$$

By converting to log base ten and dividing both sides of Eq. 14.36 by V^* it may be rearranged in the following non-dimensional form

$$\frac{v}{V_*} = 5.75\log_{10}(y/k) + 8.5 \quad ...(14.37)$$

Equation 14.37 represents the velocity distribution for turbulent flow near hydrodynamically rough boundaries. It is known as Karman-Prandtl equation for the velocity distribution near hydrodynamically rough boundaries. Though Eq. 14.37 has been derived to represent the velocity distribution for turbulent flow near rough boundaries only, but on the basis of the Nikuradse's experimental data it has been found that it may be used to represent the velocity distribution in the complete region of turbulent flow in hydrodynamically rough pipes. Accordingly Eq. 14.37 may be

considered as Karman-Prandtl equation for the velocity distribution for turbulent flow in hydrodynamically rough pipes.

(c) Criterion for Smooth and Rough Pipes. As stated in section 14.4, a pipe boundary will behave as hydrodynamically smooth or rough depending upon the relative magnitude of the average height of the surface protrusions k, and the thickness of the laminar sublayer δ' that is (k/δ') must be the parameter which will determine whether a pipe boundary is hydrodynamically smooth or rough. By substituting the value of δ' from Eq. 14.27 the value of (k/δ') becomes

$$\frac{k}{\delta'} = \frac{V_* k}{\upsilon}\frac{1}{11.6} \quad ...(14.38)$$

Equation 14.38 thus indicates that the parameter (k/δ') is a function of $(V_* k/\upsilon)$. Thus it may be stated that a pipe boundary may be characterized as hydrodynamically smooth or rough depending upon the value of the parameter $(V_* k/\upsilon)$. On the basis of Nikuradse's experiments the following criterion for hydrodynamically smooth and rough pipes has been given.

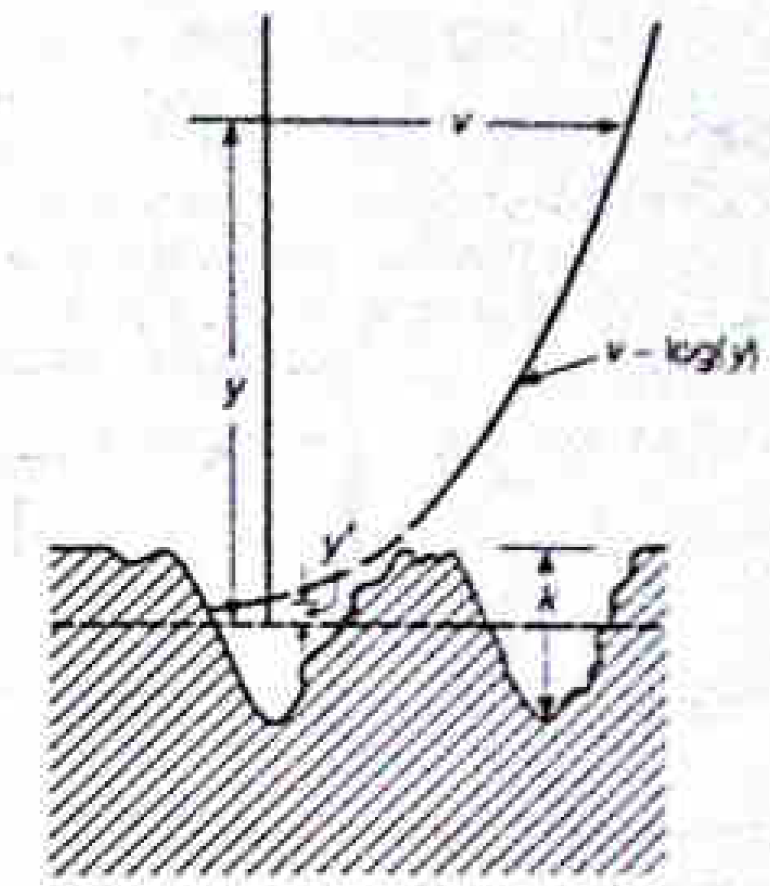


Figure 14.7 *Velocity distribution for turbulent flow near a rough boundary*

For hydrodynamically smooth: $\left(\frac{V_* k}{\upsilon}\right) \le 3$; or $\left(\frac{k}{\delta'}\right) \le 0.25$

For transition: $3 < \left(\frac{V_* k}{\upsilon}\right) < 70$; or $0.25 < \left(\frac{k}{\delta'}\right) < 6.0$

For hydrodynamically rough: $\left(\frac{V_* k}{\upsilon}\right) \ge 70$; or $\left(\frac{k}{\delta'}\right) \ge 6.0$.

14.7 VELOCITY DISTRIBUTION EQUATION FOR TURBULENT FLOW IN TERMS OF MEAN VELOCITY, FOR SMOOTH AND ROUGH PIPES

Equations 14.31 and 14.37 represent the velocity distribution equations for turbulent flow in smooth and rough pipes respectively. From these equations the velocity distribution in terms of the mean velocity of flow V, may be obtained for both smooth and rough pipes as described below.

Consider an elementary ring of thickness dr at a radial distance r from the pipe centre or at a distance y from the pipe boundary as shown in Fig. 14.8. Thus $y = (R-r)$; where R is the radius of the pipe.

Using Eq. 14.31, the mean velocity V, for turbulent flow in smooth pipes is given as follows :

$$V = \frac{Q}{\pi R^2} = \frac{1}{\pi R^2}\int_A (v)dA$$

$$= \frac{1}{\pi R^2}\int_0^R v(2\pi r)dr$$

or

$$V = \frac{1}{\pi R^2}\int_0^R 2pr\, V^*\left[5.75\log_{10}\frac{V_*(R-r)}{\upsilon}+5.5\right]dr$$

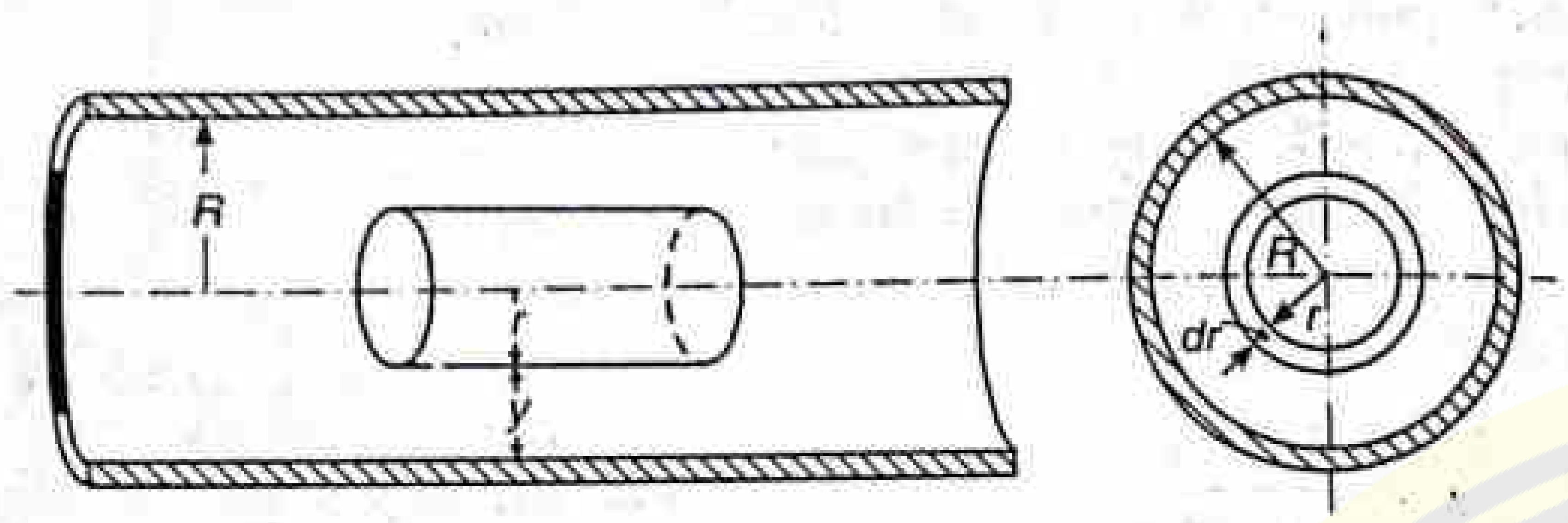


Figure 14.8 *Elementary ring of thickness dr at radial distance r from pipe centre*

On integrating and simplifying the above expression the following relation is obtained for mean velocity V, for turbulent flow in smooth pipe:

$$\frac{V}{V_*} = 5.75\log_{10}\left(\frac{V_* R}{\upsilon}\right)+1.75 \qquad ...(14.39)$$

Similarly using Eq. 14.37, the mean velocity V, for turbulent flow in rough pipes is given by

$$V = \frac{1}{\pi R^2}\int_0^R \pi r\, V_*\left[5.75\log_{10}\frac{(R-r)}{k}+8.5\right]dr$$

Integration of the above expression and subsequent simplification will give the following relation for mean velocity V, for turbulent flow in a rough pipe:

$$\frac{V}{V_*} = 5.75\log_{10}\left(\frac{R}{k}\right)+4.75 \qquad ...(14.40)$$

By subtracting Eq. 14.39 from 14.31 for smooth pipes or Eq. 14.40 from 14.37 for rough pipes the following identical relationship is obtained in both the cases:

$$\frac{v-V}{V_*} = 5.75\log_{10}\left(\frac{y}{R}\right)+3.75 \qquad ...(14.41)$$

Thus when referred to mean velocity of flow, the Karman-Prandtl expressions for velocity distribution in smooth and rough pipes become identical. Therefore, when Eq. 14.41 is used to plot a velocity distribution curve, the pipe wall roughness k need not be known. Further at $y = R$ the velocity v is equal to v_{max}, Eq. 14.41 becomes

$$\frac{v_{max}-V}{V_*} = 3.75 \qquad ...(14.41\text{ a})$$

14.8 RESISTANCE TO FLOW OF FLUID IN SMOOTH AND ROUGH PIPES

When a fluid flows through a pipe frictional resistance is offered to the motion of the fluid which results in the dissipation of certain amount of energy, thereby causing a reduction of pressure intensity in the direction of flow. As derived earlier the loss of head, due to friction in pipes, may be expressed by Darcy-Weisbach equation. But the loss of head due to friction in the case of flow through pipes may be correctly predicted if the friction factor f can be evaluated accurately. By dimensional analysis* it can be shown that the friction factor f depends upon the Reynolds number (VD/υ) and the ratio (k/D).

Thus

$$f = \phi\left[\left(\frac{VD}{\upsilon}\right),\left(\frac{k}{D}\right)\right] \qquad ...(14.42)$$

where k is the average height of the pipe wall roughness protrusions, D is the diameter of the pipe and υ is the kinematic viscosity of the fluid flowing in the pipe. The term (k/D) is commonly known as relative roughness. Sometimes (k/D) is replaced by (R/k) where R is the radius of the pipe. The term (R/k) is known as relative smoothness. Equation 14.42 is a general equation which is applicable for laminar as well as turbulent flow in pipes. For laminar flow as well as for turbulent flow with smooth boundary the relative roughness (k/D) is unimportant. As such for both these cases the friction factor f will be a function of only the Reynolds number. On the other hand for a rough boundary, since viscosity of fluid no longer plays a significant role, Reynolds number is unimportant. Hence in the case of turbulent flow with rough boundary the friction factor f will be a function of only the relative roughness (k/D).

(a) Variation of Friction Factor f for Laminar Flow. As derived in Chapter 13, the friction factor f for laminar flow in pipe is given as

$$f = \frac{64}{\text{Re}} \qquad ...(14.43)$$

The above equation shows that for laminar flow the friction factor f varies inversely with Reynolds number and it is independent of the relative roughness (k/D). It has been verified experimentally that upto Reynolds number equal to 2000, which is the limit for the flow to be laminar in circular pipes, the experimental points closely follows the trend of Eq. 14.43 as shown in Fig. 14.9. Hence for the laminar flow in circular pipes the friction factor f may be computed by using Eq. 14.43.

(b) Variation of Friction Factor f for Turbulent Flow. As stated earlier the flow of fluid in circular pipes is turbulent if the Reynolds number is greater than 4000. As such for the Reynolds number more than 2000 but less than 4000 there exists a zone of transition from laminar to turbulent flow. In the transition zone, however, there exists no specific relationship between the friction factor f and the Reynolds number Re.

In the fully developed turbulent flow, as stated earlier, the friction factor f depends on either Re or (k/D) or both, depending on whether the boundary is hydrodynamically smooth or rough or it is in transition. As such in the following paragraphs the relationship for the friction factor f for the different boundaries have been developed.

(c) Variation of Friction Factor for Turbulent Flow in Smooth Pipes. For turbulent flow in smooth pipes the friction factor depends only on the Reynolds number and it is independent of the relative

*See chapter 17.

roughness (k/D). This is so because in case of hydrodynamically smooth pipes the roughness protrusions are submerged in the laminar sublayer and hence these will have no effect on the friction factor.

In 1911 Blasius studied the experimental data of Saph and Schoder and developed the following empirical equation for turbulent flow with smooth boundaries

$$f = \frac{0.316}{(Re)^{1/4}} \qquad \ldots (14.44)$$

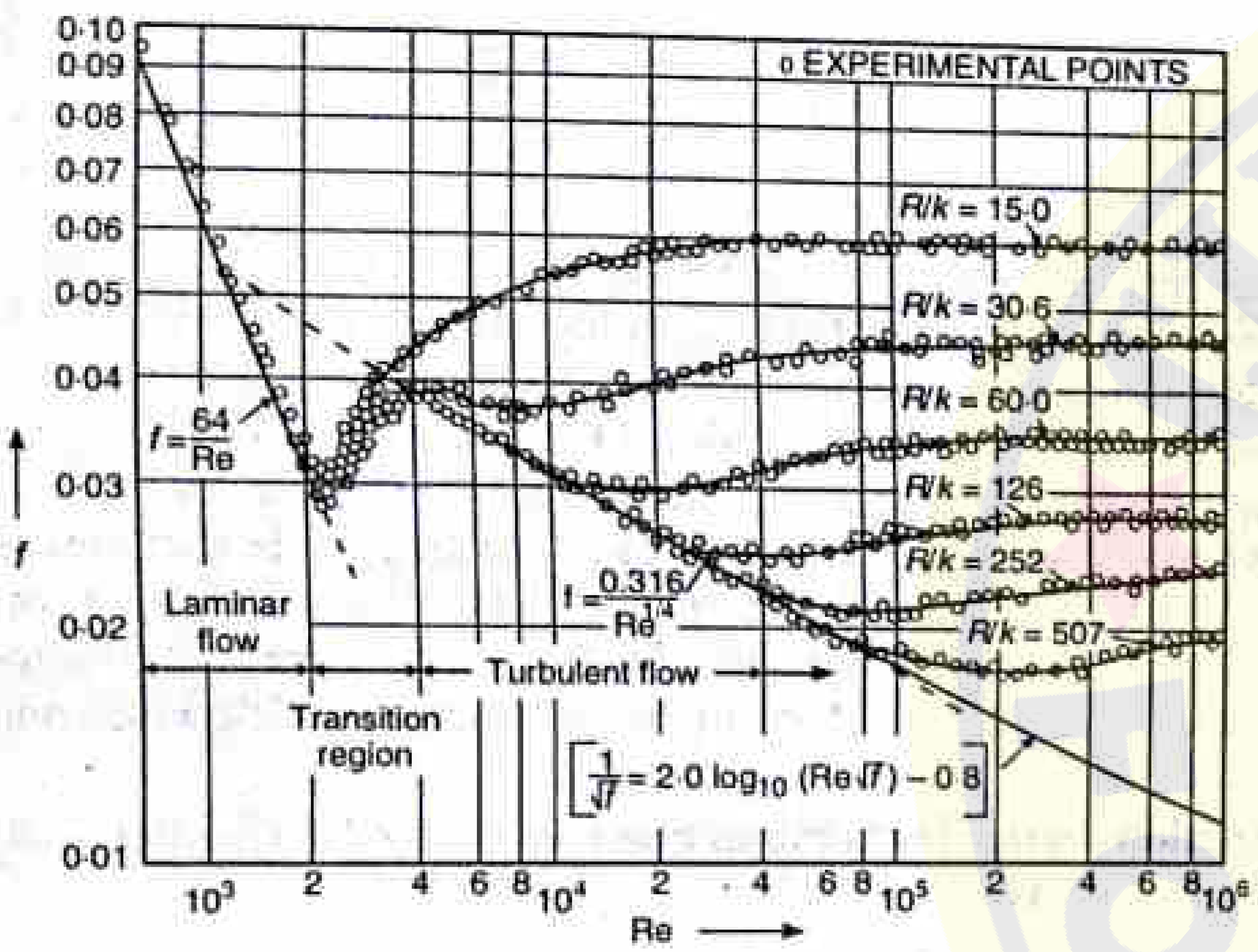


Figure 14.9 *Variation of friction factor with Reynolds number for artificially roughened pipes*

Equation 14.44 shows that the coefficient f varies inversely with the one-fouth power of the Reynolds number. Equation 14.44 has been plotted as shown in Fig. 14.9, using log f and log Re as coordinates. On the same plot Nikuradse's experimental data for smooth pipes is also plotted. It is thus observed that the experimental points follow closely the trend of Eq. 14.44 for Reynolds number varying from 4×10^3 to 10^5. For Reynolds number greater than 10^5, there is an apparent deviation of the experimental points from the straight line represented by Eq. 14.44. It was shown by Nikuradse on the basis of his experimental data that for Reynolds number greater than 10^5 the values of friction factor f given by Blasius equation were lower than the values obtained experimentally. It may thus be concluded that the Blasius equation is applicable for hydrodynamically smooth pipes for Reynolds number from 4×10^3 to 10^5.

The friction factor f for Reynolds numbers greater than 10^5 may be obtained from the expression derived from the logarithmic law of velocity distribution for smooth pipes. For turbulent flow in smooth pipes the mean velocity V of flow has been expressed by Eq. 14.39 as

$$\frac{V}{V_*} = 5.75 \log_{10}\left(\frac{V_* R}{\upsilon}\right) + 1.75 \qquad \ldots (14.39)$$

Since $V_* = V\sqrt{\frac{f}{8}}$, so by substituting the value of V_* in Eq. 14.39, it becomes

$$\frac{V}{V(f/8)^{1/2}} = 5.75 \log_{10}\left(\frac{V(f/8)^{1/2} R}{\upsilon}\right) + 1.75$$

which may be simplified and rearranged in the following form

$$\frac{1}{\sqrt{f}} = 2.03 \log_{10}\left(\frac{VD}{\upsilon}\sqrt{f}\right) - 0.91$$

or

$$\frac{1}{\sqrt{f}} = 2.03 \log_{10}\left(\mathrm{Re}\sqrt{f}\right) - 0.91 \qquad \ldots (14.45)$$

However, on the basis of Nikuradse's experimental data for turbulent flow in smooth pipes, it has been indicated that the experimental results, instead of following the trend of Eq. 14.45, follow closely the trend of the following equation:

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\mathrm{Re}\sqrt{f}\right) - 0.8 \qquad \ldots (14.46)$$

A comparison of Eq. 14.45 and 14.46 reveals that there is only a slight difference in the values of the numerical constants of the two equations. Equation 14.46 is known as 'Karman-Prandtl resistance equation' for turbulent flow in smooth pipes. From Nikuradse's experimental measurements equation 14.46 has been found to be valid from Reynolds number 5×10^4 to a Reynolds number as high as 4×10^7. Therefore Eq. 14.46 may be used to predict the value of f, for any arbitrary large Reynolds number of flow beyond the range of available experimental data. However Eq. 14.46 can be solved for f only by trial and error method, because f appears on both sides of this equation. As such the following empirical relationship given by Nikuradse, which fits the experimental data very well, can be used alternatively to obtain the value of f directly.

$$f = 0.0032 + \frac{0.221}{(\mathrm{Re})^{0.237}} \qquad \ldots (14.47)$$

(d) Variation of Friction Factor for Turbulent Flow in Rough Pipes. For turbulent flow in rough pipes the friction factor depends only on the relative roughness and it becomes independent of the Reynolds number. An expression for the friction factors for rough pipes may also be obtained on the same line as for smooth pipes. For turbulent flow in rough pipes the mean velocity V of flow has been expressed by Eq. 14.40 as

$$\frac{V}{V_*} = 5.75 \log_{10}\left(\frac{R}{k}\right) + 4.75 \qquad \ldots (14.40)$$

Again by substituting the value of V_* in Eq. 14.40 it becomes

$$\frac{V}{V(f/8)^{1/2}} = 5.75 \log_{10}\left(\frac{R}{k}\right) + 4.75$$

which may be simplified as

$$\frac{1}{\sqrt{f}} = 2.03 \log_{10}\left(\frac{R}{k}\right) + 1.68 \qquad ...(14.48)$$

Again Nikuradse's experimental data on turbulent flow in rough pipes has shown that the experimental results instead of following the trend of Eq. 14.48, follow closely the trend of the following equation

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74 \qquad ...(14.49)$$

A comparison of Eqs.14.48 and 14.49 indicated that there is only a slight difference in the value of the numerical constants of the two equations. Equations 14.49 is known as 'Karman-Prandtl resistance equation for turbulent flow in rough pipes'. It is indicated from Eq. 14.49 that the friction factor for a rough pipe depends only on the parameter (R/k), and is therefore independent of the Reynolds number. In other words for any given value of the relative pipe wall smoothness (R/k) the friction factor f has a constant value, which is indicated by the horizontal portion of various (R/k) curves in Fig. 14.9.

In Fig. 14.9 are shown the curves representing the variation of friction factor with Reynolds number for artificially roughened pipes based on the results obtained by Nikuradse. The plot of this type with f versus Re for different values of (R/k) on a log-log graph is called 'Stanton diagram' because Thomas E. Stanton was the first to present the experimental data on friction factor in this form.

A study of Fig. 14.9 reveals that the various curves for different values of (R/k) appear to bifurcate, at different points, from the curve for the smooth boundary condition, which may be considered to represent a curve corresponding to (k/R) equal to zero. Therefore it may be stated that in the turbulent flow there is a range of Reynolds number over which a pipe of given relative roughness behave in the same way as a smooth pipe. The rough pipe can then be said to be hydrodynamically smooth in this range and f depends on Re alone as indicated by Eq. 14.44 and 14.46. With further increase in Reynolds number since δ' becomes smaller than k, the boundary can no longer act as smooth boundary. Hence at a certain value of Reynolds number [whose magnitude increases as (R/k) increases], the resistance curve for a rough pipe deviates from that for a smooth pipe and after passing through a transitory phase, it finally becomes horizontal. The horizontal lines, which closely follow the trend of Eq. 14.49, correspond to the pipe boundary being hydrodynamically rough, and show that in this region of turbulent flow, f is independent of Re but depends only on the parameter (R/k). Figure 14.9 indicates that greater is the value of the parameter (R/k), larger is the Reynolds number at which the pipe which originally behaves as a smooth pipe begins to follow the pattern of a rough pipe.

(e) Criteria for Hydrodynamically Smooth and Rough Boundaries. The nature of the boundary *viz.*, smooth, rough or in transition depends on the value of (k/δ'). From Eq. 14.27

$$\delta' = \frac{11.6\nu}{V_*}$$

in which

$$V_* = V\sqrt{(f/8)}$$

$$\therefore \quad \delta' = \frac{11.6\nu}{V(f/8)^{1/2}} = \frac{11.6\nu(\sqrt{8})}{V\sqrt{f}}$$

or

$$\frac{\delta'}{R} = \frac{11.6\nu(\sqrt{8})}{VR\sqrt{f}} = \frac{11.6\sqrt{8}(2\nu)}{VD\sqrt{f}} = \frac{65.6}{\text{Re}\sqrt{f}} \qquad ... (14.50)$$

Equation 14.50 indicates that the ratio (δ'/R) is inversely proportional to the quantity $(\text{Re}\sqrt{f})$. Further from Eq. 14.50 the value of (k/δ') may be obtained as

$$\frac{k}{\delta'} = \frac{\text{Re}\sqrt{f}}{65.6(R/k)} \qquad ... (14.51)$$

From Eq. 14.51 it follows that in addition to (k/δ'), $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is another parameter the value of which may also define the nature of the boundary. The criteria for a pipe to behave as hydrodynamically smooth or rough pipe may be established on the basis of Nikuradse's experimental data as indicated below.

Equations 14.46 and 14.49 may be rearranged in the following form:

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}(R/k) = 2.0 \log_{10}\left(\frac{\text{Re}\sqrt{f}}{R/k}\right) - 0.8$$

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}(R/k) = 1.74$$

from which it follows that when the pipe is smooth $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ is a function of $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ and when it is rough $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ attains a constant value of 1.74. In other words by plotting $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ v/s $\log_{10}\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ a unique curve would be obtained which would define the nature of the boundary. Thus on the basis of Nikuradse's experimental data a plot of $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ v/s $\log_{10}\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is made as shown in Fig. 14.10 from which it is observed that as long as the parameter $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is less than 17 the pipe will behave as hydrodynamically smooth pipe and when $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ becomes greater than 400, then the pipe will behave as hydrodynamically rough pipe. For $17 < \left(\frac{Re\sqrt{f}}{Rk}\right) < 400$ the pipe boundary will behave as in transition. From Eq. 14.51 the corresponding values of (k/δ') representing the boundary in transition are $0.25 < (k/\delta') < 6$. Therefore a pipe will behave as hydrodynamically smooth pipe if (k/δ') is less than 0.25 and it will behave as hydrodynamically rough pipe when (k/δ') is greater than 6.0.

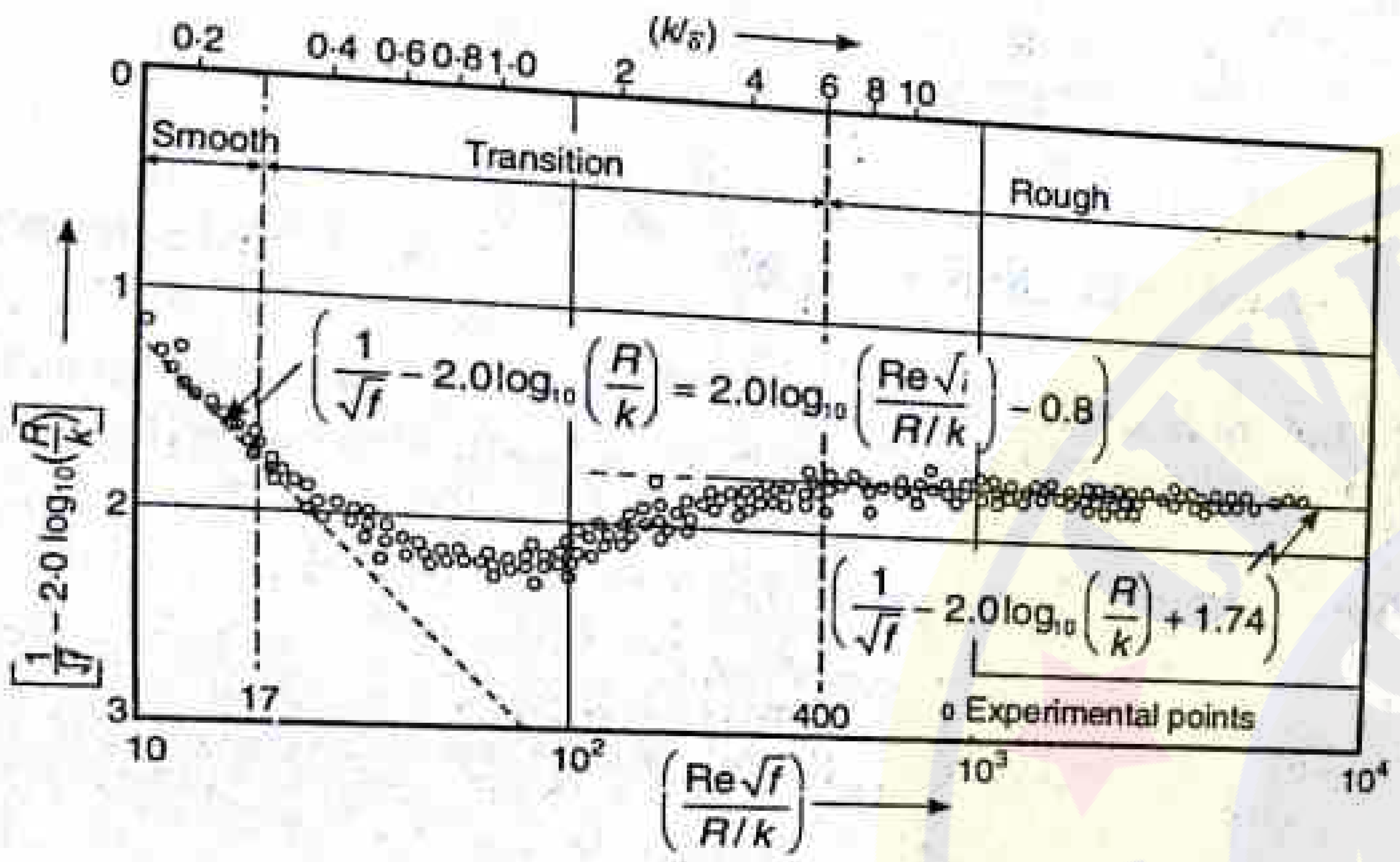
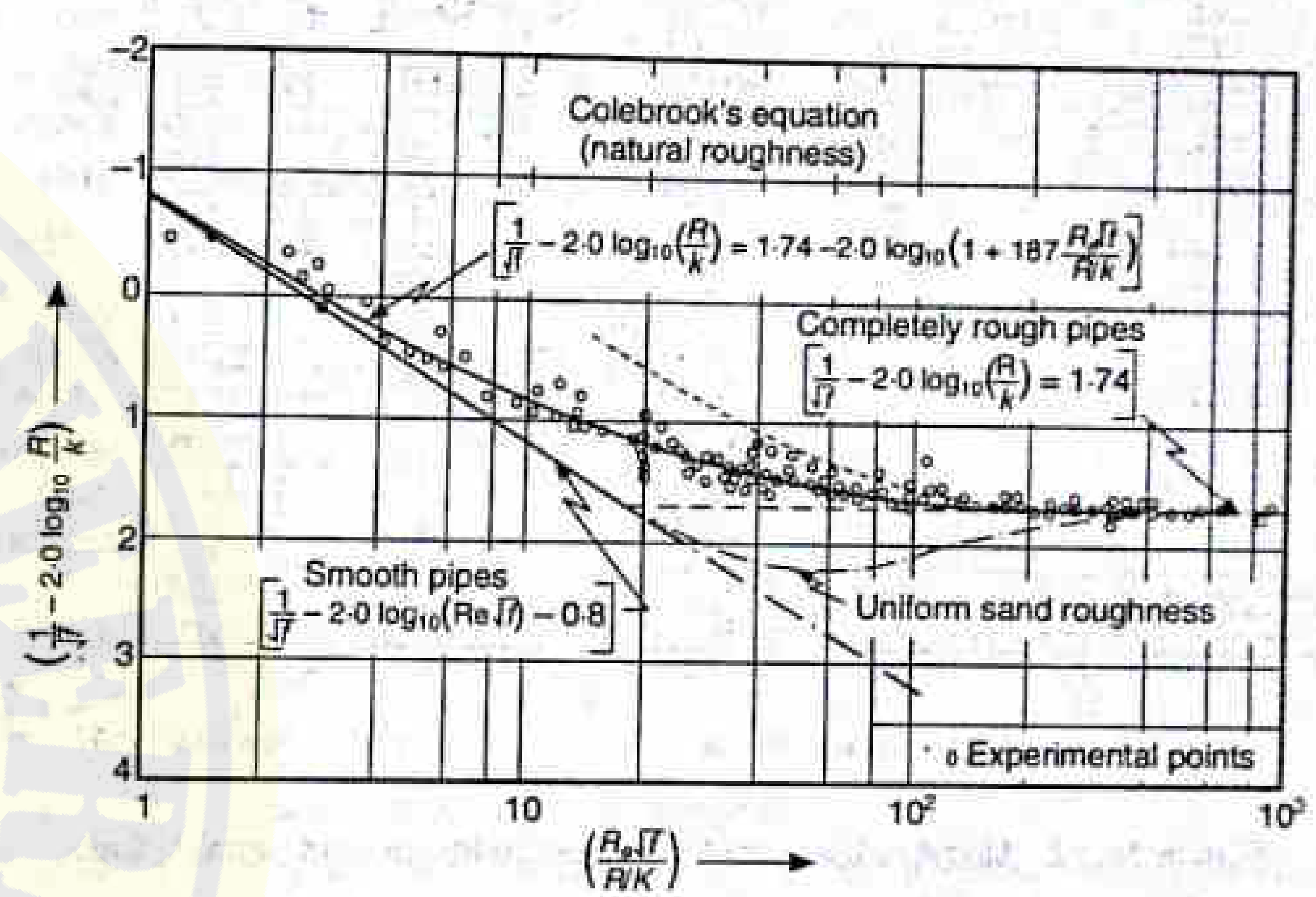


Figure 14.10 *Plot of* $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ *v/s* $\log_{10}\left(\frac{Re\sqrt{f}}{R/k}\right)$ *for artificially roughened pipes*

(f) Variation of Frictional Factor for Commercial Pipes. Although Nikuradse's experimental data on turbulent flow in smooth and artifically roughened pipes have been used to verify the various theoretically developed relationships of turbulent pipe flow, yet Nikuradse's experimental curves of Fig. 14.9 cannot be used directly to evaluate the friction factor f for commercial pipes. This is so because the actual wall roughness pattern of commercial pipes is very much different from the uniform sand grain roughness used by Nikuradse. However, for commercial pipes it is possible to evaluate the surface roughness of such pipes in terms of a uniform sand grain diameter k, which is then called *'equivalent sand grain roughness'*. In other words for any given pipe it is possible to find a uniform sand grain diameter such that, when the pipe is coated with these sand grains, it will give the same limiting value (i.e., for rough condition) of friction factor f as that for the commercial pipe in its natural form. In order to find the equivalent sand grain roughness k, for any commercial pipe, a series of experiments are conducted on the pipe at sufficiently large Reynolds numbers and the limiting value of the friction factor f is determined from Darcy-Weisbach equation. This value of f is then substituted in Eq. 14.49 and it is solved for (R/k) from which the value of equivalent sand grain roughness k for the pipe may be computed.

In the transitional region of the boundary where the value of f depends on both Re and k/D, the artificially roughened pipes would produce results totally different from those for commercial pipes. This difference can be seen from Fig. 14.11 in which test results from both Nikuradse's artificially roughened pipes and commercial pipes are plotted. The plotting of the experimental results for the commercial pipes as shown in Fig. 14.11 was carried out by C.F. Colebrook, and it is seen that the experimental data for commercial pipes in the transition region deviate from Nikuradse's data on turbulent flow in artificially roughened pipes. The coordinates of Fig. 14.11 are chosen to be

$$\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}\left(\frac{R}{k}\right)\right] \text{ and } \log_{10}\left(\frac{Re\sqrt{f}}{R/k}\right)$$



Fiigure14.11 *Plot of* $\left[\frac{1}{\sqrt{f}} - 2.0\log_{10}(R/k)\right]$ *versus* $\log_{10}\left(\frac{Re\sqrt{f}}{R/k}\right)$ *for commercial pipes in transition region*

so that for smooth pipes in accordance with Eq. 14.46 a straight line with a slope of 2 is obtained and for rough pipes in accordance with Eq. 14.49 a horizontal line is obtained. The experimental points for commercial pipes in the transition region are seen to follow closely the trend of a single curve which is asymptotic to the plots corresponding to the equations for both the smooth and the rough pipes. Colebrook and White developed an empirical equation for this curve which represents the variation of the friction factor f for commercial pipes. It is of the form

$$\frac{1}{\sqrt{f}} - 2.0\log_{10}\left(\frac{R}{k}\right) = 1.74 - 2.0\log_{10}\left(1 + 18.7\frac{R/k}{Re\sqrt{f}}\right) \quad \text{... (14.52)}$$

Equation 14.52 may therefore be used to predict the friction factor f for commercial pipes.

L.F. Moody has plotted Eq. 14.52 as shown in Fig. 14.12 which is in the form of f versus Re curves for various values of (R/k). Figure 14.12 is commonly known as 'Moody's diagram' which is essentially the same as Nikuradse's plot (Fig. 14.9) except for the transition regions. For any turbulent pipe flow problem the value of friction factor f can therefore be determined from Moody's diagram (Fig. 14.12) if the numerical values of R/k for the pipe and Re of flow are known.

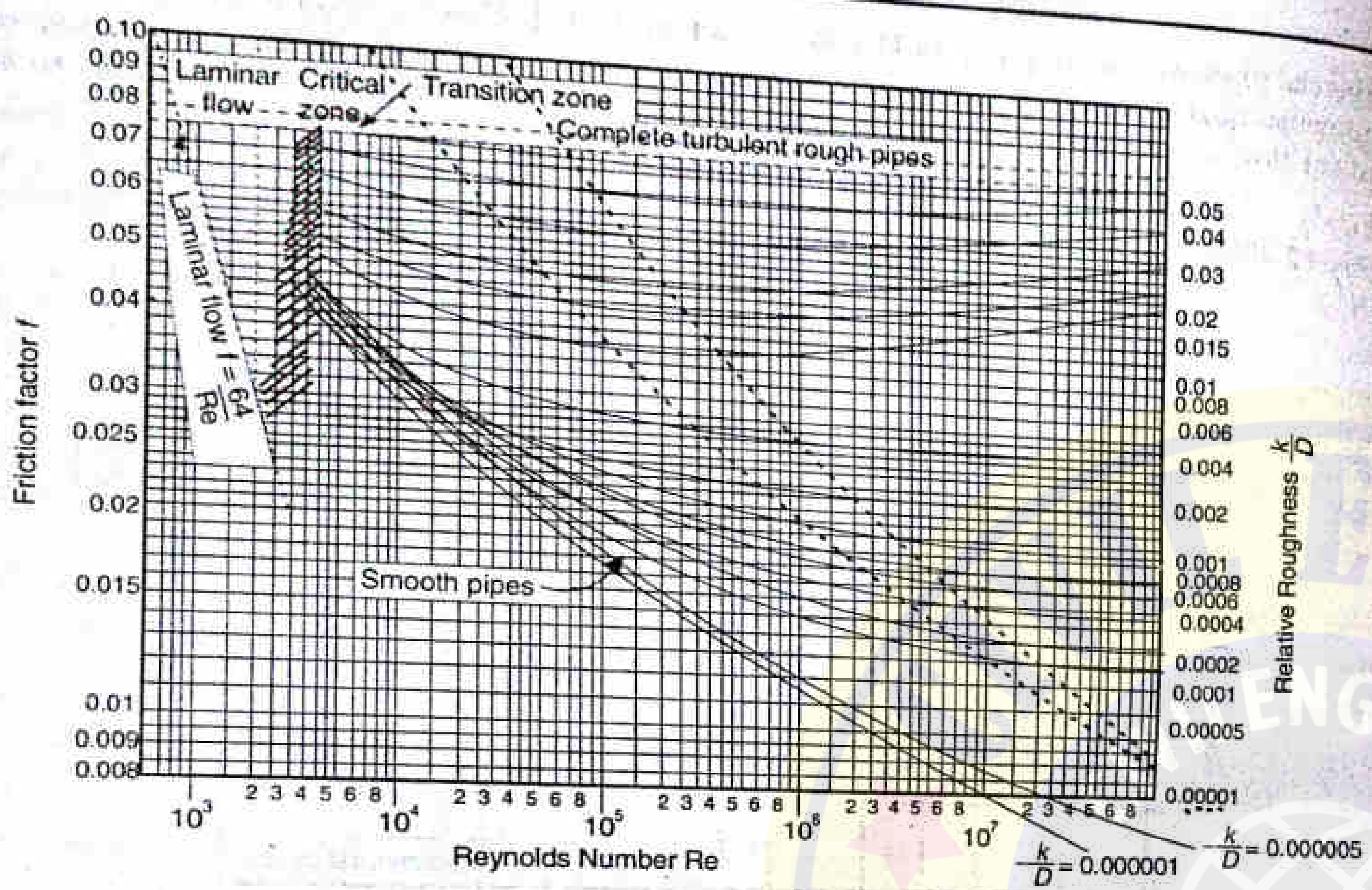


Figure 14.12 *Moody's diagram for friction factor for commercial pipes*

The values of k which may be adopted for the pipes of some of the common materials are given below in Table 14.1.

TABLE 14.1

Pipe Materials	*k in mm*
1. Glass, Brass, Copper, Lead	Smooth
2. Steel, Wrought iron	0.045
3. Asphalted cast iron	0.120
4. Galvanized iron	0.150
5. Cast iron	0.260
6. Concrete	0.30 to 3.0
7. Rivetted steel	0.90 to 9.0

The value of equivalent sand grain roughness k given in Table 14.1 correspond to material in new and clean condition. As the pipes become older the roughness increases due to corrosion. Colebrook and White have shown that the boundary roughness may increase with time approximately in accordance with the following expression

$$k = k_0 + \alpha t \quad \text{...(14.53)}$$

in which k_0 is the equivalent sand grain roughness of the new pipe material, k is the equivalent sand grain roughness at any time t, and α is the time rate of increase of roughness. Resistance measurements taken on the same pipe at two different times will enable the evaluation of the constant α which may be used to estimate the value of k at any future time.

14.9 TYPES OF PROBLEMS IN PIPELINE DESIGNS

In practice there are three types of problems which occur most frequently in the design of pipelines as mentioned below:

(1) Determination of the head lost due to friction h_f from the known values of discharge Q, length of pipe L, pipe diameter D, roughness k, and kinematic viscosity υ of the flowing fluid.

(2) Determination of discharge Q from the known values of head loss due to friction h_f, pipe diameter D, length of pipe L, roughness k, and kinematic viscosity υ of the flowing fluid.

(3) Determination of the diameter D of a pipe from the known values of discharge Q, head loss due to friction h_f, length of pipe L, roughness k and kinematic viscosity υ of the flowing fluid.

In the first case the Reynolds number Re and the relative roughness k/D can be calculated from the given data. For these values of Re and (k/D), the value of friction factor f can be found from Moody's diagram (Fig. 14.12) or from the relevant resistance equations.

In the second case the relative roughness (k/D) is known and the parameter $(\text{Re}\sqrt{f})$ can be computed from the following expression

$$\text{Re}\sqrt{f} = \frac{VD}{\upsilon}\left(\frac{2gh_f D}{V^2 L}\right)^{1/2} = \frac{D^{3/2}}{\upsilon}\left(\frac{2gh_f}{L}\right)^{1/2} \quad \text{...(14.54)}$$

From these values of (k/D) and $(\text{Re}\sqrt{f})$, the value of friction factor f can be found from Moody's diagram (Fig. 14.12) or from the relevant resistance equation.

In the third case, since the diameter of the pipe is unknown, the relative roughness (k/D), Reynolds number and friction factor f are also unknown. As such only trial and error solution is possible. The procedure will be to first assume a suitable value of f and compute the value of diameter D from Darcy-Weisbach equation. This will give a trial value of D. With this trial value of D, values of Re and (k/D) can be computed to find a new value of f from Moody's diagram (Fig. 14.12). The new values of f is used for the next trial and the procedure is repeated until the value of f does not change.

The illustrative examples below indicate the above discussed methods for solving the different pipeline problems.

14.10 FRICTION IN NON-CIRCULAR CONDUITS

Mostly the closed conduits (or pipes) used in engineering practice are of circular cross-section. However, sometimes non-circular conduits, such as rectangular air ducts are also employed in engineering installations, for which the friction loss is required to be estimated. Experiments have shown that the expressions developed for turbulent flow in circular conduits may be applied to non-circular conduits of many shapes, if in place of the diameter D of the circular conduit an 'equivalent diameter' is used, such that the hydraulic mean depth m is the same. For a circular conduit flowing full $m = (D/4)$ and so the equivalent diameter must equal $4m$, (i.e., D must be replaced by four times the hydraulic mean depth). The concept of equivalent diameter is, however, not applicable to laminar flow in non-circular conduits and also to non-circular conduits whose cross-sections deviate considerably from the circular section even if the flow is turbulent.

ILLUSTRATIVE EXAMPLES

Example 14.1. *If mixing length l is given by*

$$l = \frac{\kappa(dv/dy)}{(d^2v/dy^2)}$$

and $\tau = \tau_0\left(1-\frac{y}{R}\right)$, *derive the expression for velocity distribution for a fully developed turbulent flow in a circular pipe of radius R.*

Solution

From Eq. 14.9, we have

$$\tau = \rho l^2\left(\frac{dv}{dy}\right)^2$$

For small values of y, we have

$$\tau = \tau_0\left(1-\frac{y}{R}\right) = \tau_0$$

Thus

$$\tau_0 = \rho l^2\left(\frac{dv}{dy}\right)^2$$

Introducing the given value of l, we get

$$\tau_0 = \frac{\rho\kappa(dv/dy)^4}{(d^2v/dy^2)^2}$$

Dividing both sides by ρ and taking square root of both sides, we get

$$\sqrt{\frac{\tau_0}{\rho}} = \pm\frac{\kappa(dv/dy)^2}{(d^2v/dy^2)}$$

or

$$V_* = \pm\frac{\kappa(dv/dy)^2}{(d^2v/dy^2)}$$

or

$$\frac{d^2v}{dy^2} = \pm\frac{\kappa}{V_*}\left(\frac{dv}{dy}\right)^2$$

Since (d^2v/dy^2) will be negative, only negative sign is taken on the R.H.S. Thus

$$\frac{d^2v}{dy^2} = -\frac{\kappa}{V_*}\left(\frac{dv}{dy}\right)^2$$

Let

$$\frac{dv}{dy} = z; \quad \therefore \frac{d^2v}{dy^2} = \frac{dz}{dy}$$

Thus

$$\frac{dz}{dy} = -\frac{\kappa}{V_*}z^2$$

or

$$\frac{dz}{z^2} = -\frac{\kappa}{V_*}dy$$

Integrating both sides of the above equation, we get

$$-\frac{1}{z} = -\frac{\kappa}{V_*}y$$

or

$$z = \frac{V_*}{\kappa}\times\frac{1}{y}$$

or

$$\frac{dv}{dy} = \frac{1}{y}$$

or

$$dv = \frac{V_*}{\kappa}\times\frac{1}{y}$$

Integrating both sides of the above equation, we get

$$v = \frac{V_*}{\kappa}\log_e y + C$$

The constant of integration C may be determined from the boundary condition at the centre of the pipe:

at

$$y = R$$
$$v = v_{max} \text{ (maximum velocity)}$$

Thus

$$C = v_{max} - \frac{V_*}{\kappa}\log_e R$$

Introducing this value of C, the expression for velocity distribution for a fully developed turbulent flow in a circular pipe becomes

$$v = v_{max} + \frac{V_*}{\kappa}\log_e (y/R)$$

or

$$\frac{v - v_{max}}{V_*} = \frac{1}{\kappa}\log_e (y/R)$$

Taking $\kappa = 0.4$ and changing from natural to common logarithms the above equation becomes

$$\frac{v - v_{max}}{V_*} = 5.75\log_{10}(y/R)$$

Example 14.2. *At a point in a turbulent flow field the instantaneous values of u and v velocity components measured at an interval of 0.05 seconds are listed below.*

u (mm/s)	+105	+110	+84	+89	+102	+94	+111	+101	+87	+95	+89
v (mm/s)	−3	−16	+11	+25	−6	−20	−20	+4	+21	−3	+6

Determine $\bar{u}$ $\bar{v}$ $\overline{u'v'}$ *and local value of Reynolds' shear stress. Take* $\bar{\rho} = 1.23 kg/m^3$.

Solution

The computations are performed in tabular form as shown below.

n	u(mm/s)	$u' = (u - \bar{u})$	v (mm/s)	$v' = (v - \bar{v})$	$u'v'$
1	+105	+8	−3	−3	−24
2	+110	+13	−16	−16	−208
3	+84	−13	+11	+11	−143
4	+89	−8	+25	+25	−200
5	+102	+5	−6	−6	−30
6	+94	−3	−20	−20	+60
7	+111	+14	−20	−20	−280
8	+101	+4	+4	+4	+16
9	+87	−10	+21	+21	−210
10	+95	−2	−2	−2	+4
11	+89	−8	+6	+6	−48
	$\Sigma u = 1067$		$\Sigma v = 0$		$\Sigma(u'v') = -1063$

$\therefore \bar{u} = \frac{1067}{11}$ $= 97$ mm/s $\bar{v} = 0$ $\overline{u'v'} = -\frac{1063}{11} = -96.64$

Reynold's shear stress

$$\tau = \overline{\rho u'v'}$$

$$= (1.23 \quad 96.64) = 118.87 \text{ N/m}^2$$

Example 14.3. *For a turbulent flow in pipes determine the value of y at which the point velocity is equal to the mean velocity of flow.*

Solution

From Eq. 14.41

$$\frac{v - V}{V_*} = 5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75$$

for

$$v = V$$

$$5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75 = 0$$

or

$$\left(\frac{y}{R}\right) = 0.223 \quad \text{i.e., } y = 0.223\ R.$$

Example 14.4. *For turbulent flow in pipes (smooth as well as rough) show that*

$$\frac{v_{max}}{V} = 1.33\sqrt{f} + 1$$

Solution

For turbulent flow in pipes from Eq. 14.41

$$\frac{v - V}{V_*} = 5.75 \log_{10}\left(\frac{y}{R}\right) + 3.75$$

which is applicable to both smooth and rough pipes.

For $\quad y = R, v = v_{max}$

Thus

$$\frac{v_{max} - V}{V_*} = 3.75$$

But $\quad V_* = V\sqrt{(f/8)}$

Hence

$$\frac{v_{max} - V}{V\sqrt{(f/8)}} = 3.75$$

or

$$\frac{v_{max}}{V} = 3.75\sqrt{(f/8)} = 1.33\sqrt{f} + 1$$

which is the required expression.

Note. It may however be stated that Nikuradse's experimental data instead of following the trend of above derived equation, closely follow the trend of the following equation

$$\frac{v_{max}}{V} = 1.43\sqrt{f} + 1.$$

Example 14.5. *A smooth brass pipeline 75 mm in diameter and 900 m long carries water at the rate of 7 litres per second. If the kinematic viscosity of water is 0.0195 stokes, calculate the loss of head, wall shearing stress, centre line velocity, shear stress and velocity at 25 mm from the centre line and the thickness of the laminar sublayer. Take* $\rho = 1000 kg/m^3$ *(10^2 msl/m^3).*

Solution

Mean velocity $\quad V = \frac{Q}{A} = \frac{7 \times 10^{-3}}{\frac{\pi}{4}(0.075)^2} = 1.584$ m/s

Reynolds number $\quad \frac{VD}{\nu} = \frac{1.584 \times 0.075}{0.0195 \times 10^{-4}} = 6.09 \times 10^4$

Hence the flow is turbulent. For this value of Reynolds number of flow in smooth pipes, the value of friction factor f is given by Blasius equation as

$$f = \frac{0.316}{(Re)^{1/4}} = \frac{0.316}{(6.09\times10^4)^{1/4}} = 0.0201$$

From Darcy-Weisbach equation the head loss is given as

$$h_f = \frac{fLV^2}{2gD} = \frac{0.0201\times900\times(1.584)^2}{2\times9.81\times0.075} = 31\text{ m}$$

Wall shearing stress is given as

SI units

$$\tau_0 = \frac{\rho V^2 f}{8} = \frac{1000\times(1.584)^2\times0.0201}{8} = 6.304\text{ N/m}^2$$

Metric units

$$\tau_0 = \frac{\rho V^2 f}{8} = \frac{102\times(1.584)^2\times0.0201}{8} = 0.643\text{ kg(f)/m}^2$$

Shear velocity, $V_* = V\sqrt{\frac{f}{8}} = 1.584\sqrt{\frac{0.0201}{8}}$ 0.079 4 m/s

For turbulent flow in smooth pipes from Eq. 14.31, we have

$$\frac{v}{V_*} = 5.75\log_{10}\left(\frac{V_* y}{\upsilon}\right) + 5.5$$

For $y = \frac{D}{2} = \frac{0.075}{2} = 0.0375\text{ m}, v = V_{max}$ (i.e., centre line velocity)

Thus by substitution, we get

$$\frac{V_{max}}{0.0794} = 5.75\log_{10}\left(\frac{0.0794\times0.0375}{0.0195\times10^{-4}}\right) + 5.5$$

or $V_{max} = 1.89\text{ m/s}$

From Eq. 14.17, we have

$$\tau = \tau_0\frac{r}{R}$$

Thus shear stress at $r = 0.025$ m is obtained as

SI units

$$\tau = 6.304\left(\frac{0.025}{0.0375}\right) = 4.203\text{ N/m}^2$$

Metric units

$$\tau = 0.643\left(\frac{0.025}{0.0375}\right) = 0.429\text{ kg (f)/m}^2$$

Again from Eq. 14.31, for

$$y = (0.0375 - 0.025) = 0.0125\text{ m}$$

$$\frac{v}{0.0794} = 5.75\log_{10}\left(\frac{0.0794\times0.0125}{0.0195\times10^{-4}}\right) + 5.5$$

or $v = 1.672\text{ m/s}$

Thickness of laminar sublayer is given by Eq. 14.27 as

$$\delta' = \frac{11.6\upsilon}{V_*} = \frac{11.6\times0.0195\times10^{-4}}{0.0794} = 2.85\ 10^{-4}\text{ m}$$

Example 14.6. *Assuming the following velocity distribution in a circular pipe,*

$$v = v_{max}\left(1-\frac{r}{R}\right)^{1/7}$$

where v_{max} = maximum velocity, R = radius of pipe, v = velocity at a radius r from the centre of pipe, calculate:

(i) The ratio of the mean and the maximum velocity ;

(ii) The radius at which the actual velocity equal the mean velocity.

Solution

Consider an elementry ring of thickness dr at a radial distance r from the centre of the pipe as shown in Fig. 14.8.

The discharge dQ through the elementry ring is given by

$$dQ = v(2\pi r)dr$$

∴ Total discharge

$$Q = \int_0^R dQ$$

$$= \int_0^R v(2\pi r)\,dr$$

$$= \int_0^R v_{max}\left(1-\frac{r}{R}\right)^{1/7}\times2\pi r\,dr$$

$$= 2\lambda v_{max}\int_0^R r\left(1-\frac{r}{R}\right)^{1/7}dr$$

$$= \frac{49}{60}\times(\pi R^2)\times v_{max}$$

Mean velocity of flow $V = \frac{Q}{\lambda R^2}$

$$V = \frac{49}{60} v_{max}$$

or
$$\frac{V}{v_{max}} = \frac{49}{60} = 0.817$$

For
$$v = V$$

$$V = v_{max}\left(1-\frac{r}{R}\right)^{1/7}$$

or
$$\frac{V}{v_{max}} = v_{max}\left(1-\frac{r}{R}\right)^{1/7} = \frac{49}{60}$$

$\therefore$
$$r = 0.7579\ R$$

i.e., actual velocity is equal to mean velocity at a radius of 0.7579 R from the centre of the pipe.

Alternataively if y is the distance from the pipe boundary, then since $y = (R - r)$ and hence

$$y = (1 - 0.7579)\ R = 0.2421R$$

i.e., actual velocity is equal to mean velocity at a distance of 0.2421 R from the pipe.

Example 14.7. *Mean point velocities measured with the help of a pitot tube at mid-point and quarter point of a 0.2 m diameter pipe were found to be 1.50 m/s and 1.35 m/s respectively. If the flow in the pipe is turbulent, determine the discharge, friction factor and average height of roughness projections.*

Solution

From Eq. 14.23, we have

$$\frac{v_{max} - v}{V_*} = 5.75 \log_{10}\left(\frac{R}{y}\right)$$

$$v_{max} = 1.50\ \text{m/s};\ v = 1.35\ \text{m/s};\ R = 0.1\ \text{m; and}$$
$$y = (R - r) = (0.1 - 0.05) = 0.05\ \text{m}$$

Thus by substitution, we get

$$\frac{1.50 - 1.35}{V_*} = 5.75 \log_{10}\left(\frac{0.1}{0.05}\right)$$

$\therefore$
$$V^* = 0.0867\ \text{m/s}$$

From Eq. 14.41 (a), we have

$$\frac{V_{max} - V}{V_*} = 3.75$$

By substituting the given values, we get

$$\frac{1.50 - V}{0.0867} = 3.75\ ;\ \text{or}\ V = 1.175\ \text{m/s}$$

Thus discharge
$$Q = \frac{\pi D^2}{4} \times V$$

$$= \frac{\pi(0.2)^2}{4}\ 1.175 = 0.037\ \text{m3/s}$$

Also
$$V^* = V\sqrt{f/8}$$

or
$$0.0867 = 1.175\sqrt{f/8}$$

$\therefore$
$$f = 0.0436$$

From Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

or
$$\frac{1}{\sqrt{0.0436}} = 2.0 \log_{10}\left(\frac{0.1}{k}\right) + 1.74$$

$\therefore$
$$k = 2.99\ 10^{-3}\ \text{m} = 2.99\ \text{mm}$$

Example 14.8. *A rough pipeline of diameter 0.1 m carries water at 20°C at the rate of 50 l/s. If the average height of the protrusions on the pipe surface is 0.15 mm, calculate the friction factor, maximum velocity, shear stress at the pipe surface and the shear velocity. For water at 20°C take $\nu = 1.0 \times 10^{-6}$ m²/s and $\rho = 1000$ kg/m³ (102 msl/m³).*

Solution

Mean velocity
$$V = \frac{Q}{A} = \frac{50\times10^{-3}}{\frac{\pi}{4}(0.1)^2} = 6.37\ \text{m/s}$$

Reynolds number
$$\text{Re} = \frac{VD}{\nu} = \frac{6.37\times0.1}{1.0\times10^{-6}} = 6.37\times 10^5$$

Hence the flow is turbulent in the rough pipe.

For rough turbulent flow from Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

$$R = 0.05\ \text{m; and}\ k = 0.15\ 10^{-3}\ \text{m}$$

Thus by substitution, we get

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{0.05}{0.15\times10^{-3}}\right) + 1.74$$

or
$$f = 0.0217$$

Shear stress at the pipe surface is given by $\tau_0 = \frac{\rho V^2 f}{8}$

In SI units

$$\tau_0 = \frac{1000\times(6.37)^2\times0.0217}{8} = 110.065\ \text{N/m}^2$$

In Metric units

$$\tau_0 = \frac{102 \times (6.37)^2 \times 0.0217}{8} = 11.227 \text{ kg(f)/m}^2$$

Shear velocity $$V_* = \sqrt{\frac{\tau_0}{\rho}} = \sqrt{\frac{110.065}{1000}} = 0.332 \text{ m/s}$$

Also $$V_* = \sqrt{\frac{11.227}{102}} = 0.332 \text{ m/s}$$

From Eq. 14.37 the velocity distribution in rough pipe is given as

$$\frac{v}{V_*} = 5.75 \log_{10}\left(\frac{y}{k}\right) + 8.5$$

For $$y = R, v = v_{max}$$

Hence by substitution, we get

$$\frac{v_{max}}{0.332} = 5.75 \log_{10} + 8.5$$

$\therefore$ $$v_{max} = 7.64 \text{ m/s}$$

Example 14.9. *A liquid of specific gravity 0.88 and absolute viscosity 6.533 10^{-4} N.s/m² flows through a pipe of diameter 0.15 m at the rate of 60 litres per second. If the loss of head in 100 m length of pipe is 4.56 m, determine whether the pipe is rough or smooth.*

Solution

Mean velocity $$V = \frac{Q}{A} = \frac{60 \times 10^{-3}}{\frac{\pi}{4}(0.15)^2} = 3.40 \text{ m/s}$$

Reynolds number $$Re = \frac{\rho V D}{\mu}$$

$$= \frac{0.88 \times 1000 \times 3.4 \times 0.15}{6.533 \times 10^{-4}} = 6.87 \times 10^5$$

Hence the flow is turbulent.

According to Darcy-Weisbach equation

$$h_f = \frac{fLV^2}{2gD}$$

By substituting the given values, we get

$$4.56 = \frac{f \times 100 \times (3.40)^2}{2 \times 9.81 \times 0.15}$$

or $$f = 0.0116$$

Let the pipe be assumed as hydrodynamically rough, then from Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

By substitution, we have

$$\frac{1}{\sqrt{0.0116}} = 2.0 \log_{10}\left(\frac{R}{k}\right) + 1.74$$

or $$(R/k) = 5921$$

Thus $$\left[\frac{Re\sqrt{f}}{R/k}\right] = \frac{6.87 \times 10^5 \times (0.0116)^{1/2}}{5921} = 12.5 \approx 13$$

Since the parameter $\left[\frac{Re\sqrt{f}}{R/k}\right]$ is less than 17, the assumption made for the pipe to be rough is not correct. Hence the pipe is hydrodynamically smooth.

Example 14.10. *Find the rate of flow of water at 40°C through a cast iron pipe of 100 mm diameter, if the loss of head is 1.05 m per 100 m length of the pipe. The average height of the roughness projection on the pipe wall is 0.25 mm and υ for water at 40°C is 7.5 × 10^{-3} stokes.*

Solution

$$\frac{R}{k} = \frac{50}{0.25} = 200$$

From Eq. 14.54, we have

$$Re\sqrt{f} = \frac{D^{3/2}}{\upsilon}\sqrt{\frac{2gh_f}{L}}$$

$$Re\sqrt{f} = \frac{(0.1)^{3/2}}{7.5 \times 10^{-3} \times 10^{-4}}\left[\frac{2 \times 9.81 \times 1.05}{100}\right]^{1/2}$$

$$= 1.914 \times 10^4$$

$\therefore$ $$\left[\frac{Re\sqrt{f}}{R/k}\right] = \frac{1.914 \times 10^4}{200} = 95.7$$

Since the parameter is more than 17 but less than 400, the boundary acts in transition. Thus from Eq. 14.52, we have

$$\frac{1}{\sqrt{f}} - 2.0 \log_{10}\left(\frac{R}{k}\right) = 1.74 - 2.0 \log_{10}\left(1 + 18.7 \frac{R/k}{Re\sqrt{f}}\right)$$

or $$\frac{1}{\sqrt{f}} - 2.0 \log_{10}(200) = 1.74 - 2.0 \log_{10}\left(1 + \frac{18.7 \times 200}{1.914 \times 10^4}\right)$$

or
$$f = 0.0261$$
Now by substituting in Darcy-Weisbach equation, we have
$$1.05 = \frac{0.0261 \times 100 \times V^2}{2 \times 9.81 \times 0.1}$$
or
$$V = 0.888 \text{ m/s}$$
∴ Discharge
$$Q = \frac{\pi D^2}{4} \times V$$
$$= \frac{\pi \times (0.1)^2}{4} \times 0.888$$
$$= 0.007 \text{ m}^3/\text{s} = 7 \text{ l/s}$$

Example 14.11. *A pipe of diameter 0.1m carries water at 15°C. If sand grains 0.75 mm in diameter are cemented on the inner surface of the pipe for test purposes, at what velocity of water will (a) the surface roughness just begins to disturb the laminar sublayer, and (b) the pipe wall behaves as rough boundary. Kinematic viscosity of water at 15°C is 1.14 × 10⁻² stokes.*

Solution
$$\frac{R}{k} = \frac{0.05}{0.75 \times 10^{-3}} = 66.67$$
(a) For smooth pipes from Blasius equation, we have
$$f = \frac{0.316}{\text{Re}^{1/4}}$$
and in the limiting condition the parameter $\left[\frac{\text{Re}\sqrt{f}}{R/k}\right]$ is equal to 17,

or
$$\left(\text{Re}\sqrt{f}\right) = (17 \times 66.67)$$
By substituting the value of f from the Blasius equation, we get
$$(\text{Re})^{7/8} = \frac{17 \times 66.67}{\sqrt{0.316}} = 2016$$
∴
$$\text{Re} = 5978$$
Thus the velocity at which the surface roughness will just begin to disturb the laminar sublayer is
$$V = \frac{\text{Re}\,\upsilon}{D} = \frac{5978 \times 1.14 \times 10^{-2} \times 10^{-4}}{0.1} = 0.068 \text{ m/s}$$
(b) For rough pipes from Eq. 14.49, we have
$$\sqrt{f} = \frac{1}{2\log_{10}\left(\frac{R}{k}\right) + 1.74}$$
or
$$\sqrt{f} = \frac{1}{2\log_{10}(66.67) + 1.74} = 0.1856$$

In the limiting condition the parameter $\left[\frac{\text{Re}\sqrt{f}}{R/k}\right] = 400$

or
$$\text{Re} = \frac{400 \times 66.67}{0.1856} = 1.44 \times 10^5$$
Hence the minimum velocity at which the pipe wall will behave as rough boundary is
$$V = \frac{\text{Re}\,\upsilon}{D} = \frac{1.44 \times 10^5 \times 1.14 \times 10^{-2} \times 10^{-4}}{0.1} = 1.642 \text{ m/s}$$

Example 14.12. *Determine the diameter of a cast iron pipe which is required to carry water at 10°C at the rate of 250 litres per second, if the loss of head is not to exceed 2 m per 100 m length of pipe. The average height of the pipe wall projection is 0.36 mm and kinematic viscosity of water at 10°C is 0.013 stokes.*

Solution

Discharge $Q = 250 \; 10^{-3} = 0.25 \text{ m}^3/\text{s}$

Assume $f = 0.018$

From Darcy-Weisbach equation, we have
$$h_f = \frac{fLV^2}{2gD} = \frac{8fLQ^2}{\pi^2 g D^5}$$
or
$$D^5 = \frac{8fLQ^2}{\pi^2 g h_f} = \frac{8 \times 0.018 \times 100 \times (0.25)^2}{\pi^2 \times 9.81 \times 2} = 465 \times 10^{-5}$$
or
$$D = 0.342 \text{ m}$$
Mean velocity
$$V = \frac{Q}{A} = \frac{0.25}{\frac{\pi}{4}(0.342)^2} = 2.721 \text{ m/s}$$
Reynolds number
$$\text{Re} = \frac{VD}{\upsilon} = \frac{2.721 \times 0.342}{0.013 \times 10^{-4}} = 7.16 \times 10^5$$
Relative roughness
$$\frac{k}{D} = \frac{0.36 \times 10^{-3}}{0.342} = 0.001\,05$$
From Fig 14.12 the value of f corresponding to these values of Re and (k/D) is obtained as $f = 0.02$.

Computing a new value of D based on $f = 0.02$
$$D^5 = \frac{8fLQ^2}{\pi^2 g h_f} = \frac{8 \times 0.02 \times 100 \times (0.25)^2}{\pi^2 \times 9.81 \times 2} = 516 \times 10^{-5}$$
or
$$D = 0.349 \text{ m}$$
Thus
$$V = \frac{0.25}{\frac{\pi}{4}(0.349)^2} = 2.613 \text{ m/s}$$
$$\text{Re} = \frac{VD}{\upsilon} = \frac{2.613 \times 0.349}{0.013 \times 10^{-4}} = 7.01 \times 10^5$$

and $$\frac{k}{D} = \frac{0.36 \times 10^{-3}}{0.349} = 0.00103$$

Again from Fig. 14.12 for these values of Re and (k/D), $f = 0.0198$ which agrees with the assumed value of f.

Hence $$D = 0.349 \text{ m}$$

Note. The next larger standard size of pipe should be selected.

Example 14.13. *Design the diameter of a steel pipe to carry oil having kinematic viscosity of 0.20 stokes with a velocity of 1.5 m/s, if Darcy's friction factor over almost the whole turbulent range is given by*

$$f = 0.0055\left[1+\left(20000\frac{\varepsilon}{D}+\frac{10^6}{Re}\right)^{0.33}\right]$$

in which ε = *equivalent sand grain roughness,* D = *diameter of pipe,* Re = *Reynolds number.*

The head loss is to be limited to 0.4 m per 100 metres of pipe. The equivalent sand grain roughness for steel may be assumed as 0.045 mm.

Solution

If D is the diameter of the pipe, then

$$\text{Re} = \frac{VD}{\upsilon} = \frac{1.5 \times D}{0.20 \times 10^{-4}} = 7.5 \times 10^4 D$$

$$\varepsilon = 0.045 \text{ mm} = 0.045 \times 10^{-3} \text{ m}$$

Substituting these values in the given expression for f it becomes

$$f = 0.0055\left[1+\left(\frac{0.9}{D}+\frac{13.33}{D}\right)^{0.33}\right]$$

or $$f = 0.0055 + \frac{0.0132}{D^{0.33}} \quad \text{... (i)}$$

Further according to Darcy-Weisbach equation

$$h_f = \frac{fLV^2}{2gD}$$

or $$\frac{0.4}{100} = \frac{f \times (1.5)^2}{2 \times 9.81 \times D}$$

or $$f = 0.0349\, D \quad \text{... (ii)}$$

Solving Eqs (i) and (ii) by trial the pipe diameter D may be obtained. However, the pipe diameter which may be adopted should be such that corresponding to this value of the diameter, the value of f obtained by Eq. (i) is slightly less than that obtained by Eq. (ii), so that the head loss will be less than the given limiting value. Accordingly the value of the pipe diameter which may be adopted is

$$D = 0.625 \text{ m}$$

However, if the pipe of diameter 0.625 m is not available, the next larger standard size of the pipe may be adopted, which would have head loss less than the given limiting value.

Example 14.14. *A pipeline of diameter 0.3 m carries oil at the rate of 540 l/s. If the specific gravity of the oil is 0.80 and its kinematic viscosity is equal to 0.023 stokes, determine (i) the maximum permissible height of the protrusions upto which the pipe acts as smooth pipe, (ii) the height of the protrusions beyond which the pipe would become rough.*

Solution

Mean velocity, $$V = \frac{Q}{A} = \frac{540 \times 10^{-3}}{(\pi/4)(0.3)^2} = 7.64 \text{ m/s}$$

Reynolds number $$\text{Re} = \frac{VD}{\upsilon} = \frac{7.64 \times 0.3}{0.023 \times 10^{-4}} = 9.97 \times 10^5$$

(i) For turbulent flow in smooth pipes, from Eq. 14.46, we have

$$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(\text{Re}\sqrt{f}\right) - 0.8$$

or $$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\left(9.97 \times 10^5 \sqrt{f}\right) - 0.8$$

or $$\frac{1}{\sqrt{f}} = 2.0 \log_{10}\sqrt{f} + 11.2$$

Solving by trail, we obtain

$$f = 0.0116$$

Alternatively for tublent flow in smooth pipes, from Eq. 14.47, we have

$$f = 0.0032 + \frac{0.221}{(\text{Re})^{0.237}}$$

or $$f = 0.0032 + \frac{0.221}{(9.97 \times 10^5)^{0.237}}$$

or $$f = 0.01157 \approx 0.0116$$

Thus $$\left(\text{Re}\sqrt{f}\right) = (9.97 \times 10^5) \times \sqrt{0.0116}$$
$$= 1.074 \times 10^5$$

The pipe will act as smooth pipe as long as

$$\left(\frac{\text{Re}\sqrt{f}}{R/k}\right) < 17$$

or $$\frac{R}{k} > \frac{1.074 \times 10^5}{17}$$

or $$k < \frac{17 \times 0.15}{1.074 \times 10^5}$$

or $$k < 2.37 \times 10^{-5} \text{ m}$$

or $$k < 0.0237 \text{ mm}$$

Hence the maximum permissible height of the surface protrusions = 0.0237 mm.

(ii) The pipe will act as rough when

$$\left(\frac{\mathrm{Re}\sqrt{f}}{R/k}\right) \geq 400$$

or

$$\sqrt{f} \geq \frac{400(R/k)}{9.97\times10^5}$$

For rough turbulent flow from Eq. 14.49, we have

$$\frac{1}{\sqrt{f}} = 2.0\log_{10}(R/k) + 1.74$$

or

$$\frac{9.97\times10^5}{400(R/k)} = 2.0\log_{10}(R/k) + 1.74$$

Solving by trail

$$\frac{R}{k} = 363.3$$

$$\therefore \quad k = \frac{0.15}{363.3}$$

$$= 4.13\times10^{-4}\ \mathrm{m} = 0.413\ \mathrm{mm}$$

Hence the pipe will behave as rough when the height of the surface protrusions is equal to or greater than 0.413 mm.

Example 14.15. *Water flows in a 80 mm pipe at Reynold's number 80 000. The pipe is estimated to have an equivalent sand grain roughness of size 0.16 mm. Determine the head loss expected in 500 m length of the pipe. How much head loss would be expected if this pipe were smooth? Take kinematic viscosity of water as 10^{-6} m²/s. The following explicit equation for friction factor may be used.*

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{k}{D} + \frac{21.25}{Re^{0.9}}\right)$$

in which k is the equivalent sand grain roughness, D is the pipe diameter and Re is the pipe Reynold's number.

Solution

If V is the velocity of flow, D is the pipe diameter and υ is kinematic viscosity of water, then Reynolds' number

$$\mathrm{Re} = \frac{VD}{\upsilon}$$

$$\mathrm{Re} = 80\,000;\ D = 80\ \mathrm{mm} = 0.08\ \mathrm{m};\ \text{and}$$

$$\upsilon = 10^{-6}\ \mathrm{m^2/s}$$

Thus by substitution, we get

$$8000 = \frac{V\times0.08}{10^{-6}}$$

$$\therefore \quad V = 1\ \mathrm{m/s}$$

From Darcy-Weisbach equation, we have

$$h_f = \frac{fLV^2}{2gD}$$

L = 500 m; and f is given by

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{k}{D} + \frac{21.25}{\mathrm{Re}^{0.9}}\right)$$

k = 0.16 mm; D = 80 mm; and Re = 80 000

Thus by substitution, we get

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left[\frac{0.16}{80} + \frac{21.25}{(80000)^{0.9}}\right]$$

or

$$\frac{1}{\sqrt{f}} = 6.2396$$

$$\therefore \quad f = 0.0257$$

Thus

$$h_f = \frac{0.0257\times500\times(1)^2}{2\times9.81\times0.08} = 8.19\ \mathrm{m}$$

If the pipe were smooth, $k = 0$ and hence the equation for friction factor becomes

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left(\frac{21.25}{\mathrm{Re}^{0.9}}\right)$$

Thus by substitution, we get

$$\frac{1}{\sqrt{f}} = 1.14 - 2\log_{10}\left[\frac{21.25}{(80000)^{0.9}}\right]$$

or

$$\frac{1}{\sqrt{f}} = 7.311$$

$$\therefore \quad f = 0.0187$$

Thus in this case, we have

$$h_f = \frac{0.0187\times500\times(1)^2}{2\times9.81\times0.08} = 5.96\ \mathrm{m}$$

Thus when the pipe is smooth the head loss is reduced by about 27%.

Example 14.16. *Starting from the velocity distribution in pipes given below*

$$\frac{v_{max} - v}{V_*} = 5.75\log_{10}\left(\frac{R}{y}\right)$$

obtain the relationship for the ratio of maximum velocity to average velocity in terms of Darcy-Weisbach friction factor and show that this ratio increases with the friction factor.

Solution

From the given velocity distribution, we have

$$v = v_{max} - 5.75 \quad V_* \log_{10}\left(\frac{R}{y}\right)$$

Changing from common (to the base 10) to normal (to the base e) logarithms, above equation becomes

$$v = v_{max} - \frac{5.75}{2.30} V_* \log_e\left(\frac{R}{y}\right)$$

or
$$v = v_{max} - 2.5 \quad V_* \log_e\left(\frac{R}{y}\right)$$

or
$$v = v_{max} + 2.5 \quad V_* \log_e\left(\frac{y}{R}\right)$$

Consider an elementary ring of thickness dr at a radial distance r from the pipe centre or at a distance y from the pipe boundary (see Fig. 14.8). Thus $y = (R-r)$ and the above equation becomes

$$v = v_{max} + 2.5 V_* \log_e\left(\frac{R-r}{R}\right)$$

The average or mean velocity V is given as follows

$$V = \frac{Q}{\pi R^2} = \frac{1}{\pi R^2}\int_A (v) dA$$

or
$$V = \frac{1}{\pi R^2}\int_0^R v(2\pi r) dr$$

or
$$V = \frac{1}{\pi R^2}\int_0^R 2\pi r\left[v_{max} + 2.5 V_* \log_e\left(\frac{R-r}{R}\right)\right] dr$$

or
$$V = \frac{2}{R^2}\int_0^R r\left[v_{max} + 2.5 V_* \log_e\left(\frac{R-r}{R}\right)\right] dr$$

or
$$V = \frac{2}{R^2}\left[\frac{v_{max} R^2}{2} + 2.5 V_* \int_0^R r \log_e\left(\frac{R-r}{R}\right) dr\right]$$

or
$$V = v_{max} + \frac{5 V_*}{R^2}\int_0^R r \log_e\left(\frac{R-r}{R}\right) dr$$

or
$$V = v_{max} + \frac{5 V_*}{R^2}\left[\left\{\frac{r^2}{2}\log_e\left(\frac{R-r}{R}\right)\right\}_0^R - \int_0^R \frac{-(r^2)}{2(R-r)} dr\right]$$

or
$$V = v_{max} + \frac{5 V_*}{R^2}\left[\frac{R^2}{2}\log_e(0) + \frac{1}{2}\int_0^R \frac{r^2}{(R-r)} dr\right]$$

or
$$V = v_{max} + \frac{5 V_*}{R^2}\left[\frac{R^2}{2}\log_e(0) + \frac{1}{2}\int_0^R \left\{-r - R + \frac{R^2}{(R-r)}\right\} dr\right]$$

or
$$V = v_{max} + \frac{5 V_*}{R^2}\left[\frac{R^2}{2}\log_e(0) + \frac{1}{2}\left(-\frac{R^2}{2} - R^2\right) - \frac{R^2}{2}\log_e(0)\right]$$

or
$$V = v_{max} - 3.75 \, V_*.$$

Since $V_* = V\sqrt{\frac{f}{8}}$, we get

$$V = v_{max} - 3.75 \, V \sqrt{\frac{f}{8}}$$

or
$$v_{max} = V\left(1 + 3.75\sqrt{\frac{f}{8}}\right)$$

or
$$\frac{v_{max}}{V} = 1 + 3.75\sqrt{\frac{f}{8}} \quad \text{(It is same as Eq. 14.41(a))}$$

which is the required expression for the ratio (v_{max}/V). It shows that the ratio (v_{max}/V) increases as the friction factor f increases.

Example 14.17. *After 10 years of service, an asphalted cast iron water main 0.45 m in diameter is found to require 30% more power to deliver 240 litres per second for which it was originally designed. Determine the corresponding magnitude of the rate of roughness increase α. Take ν for water as 0.014 stokes.*

Solution

The rate of flow remaining constant, a 30% increase in power corresponds to a 30% increase in the value of friction factor f. Thus if f_{10} represents the value of f after 10 years then

$$f_{10} = 1.3 f_0$$

where f_0 is the value of f for the new pipe at the time of its installation.

Discharge $\quad Q = 240 \times 10^{-3} = 0.24 \text{ m}^3/\text{s}$

Mean velocity $\quad V = \frac{Q}{A} = \frac{0.24}{(\pi/4)(0.45)^2} = 1.51 \text{ m/s}$

Reynolds number $\quad Re = \frac{VD}{\nu} = \frac{1.51 \times 0.45}{0.014 \times 10^{-4}} = 4.85 \times 10^5$

From Table 14.1 the value of k for a new asphalted cast iron pipe is $k = k_0 = 0.12$ mm

Thus relative roughness k/D, at the time of installation of the pipe is

$$\left(\frac{k}{D}\right)_0 = \frac{0.12}{0.45 \times 10^3} = 0.000\,27$$

From Fig. 14.12 for Re = 4.85×10^5 and $(k/D) = 0.00027$, the value of the friction factor $f_0 = 0.0165$

Thus

$$f_{10} = (1.3 \quad 0.0165) = 0.0215$$

Again from Fig. 14.12 for Re = 4.85×105 and $f_{10} = 0.0215$, the value of relative roughness $(k/D)_{10}$ = 0.00125.

Now if k_{10} represents the value of k after 10 years then

$$k_{10} = (0.00125 \times 0.45)$$
$$= 0.5625 \times 10^{-3}\,\text{m} = 0.5625\text{ mm}$$

and from Eq. 14.51 we have

$$\alpha = \frac{k_{10} - k_0}{t}$$
$$= \frac{0.5625 - 0.12}{10} = 0.04425 \text{ mm per year.}$$

SUMMARY OF MAIN POINTS

1. The flow in pipes is turbulent if Reynolds number is more than 4000.
2. The velocity distribution in turbulent flow is relatively uniform and the velocity profile of turbulent flow is much flatter than the corresponding laminar flow parabola for the same mean velocity.
3. Shear stress in turbulent flow is sum of shear stress due to viscosity and shear stress due to turbulence, i.e.,

$$\tau = \mu\frac{dv}{dy} + \eta\frac{dv}{dy}$$

where

μ = absolute viscosity; and

η = eddy viscocity;

4. Turbulent shear stress by Reynolds is given as

$$\tau = \overline{\rho v_x' v_y'}$$

where

v_x' and v_y' = the fluctuating components of velocity in the x and y directions respectively.

5. According to Prandtl the shear stress in turbulent flow is given by

$$\tau = \rho l^2 \left(\frac{dv}{dy}\right)^2$$

where

l = mixing length.

6. According to Prandtl, mixing length is that distance in the transverse direction which must be covered by a lump of fluid particles travelling with its original mean velocity in order to make the difference between its velocity and the velocity of the new layer equal to the mean transverse fluctuation in turbulent flow.
7. When a fluid flows through a pipe, close to the pipe wall a boundary layer is formed which may attain a maximum thickness equal to the radius of the pipe at a certain section of the pipe at which the flow is considered to have been established. For laminar flow in a pipe laminar boundary layer will be developed for the entire length of the pipe. According to Rouse the length of the pipe x from the entrance of the pipe upto the section where the flow is established is given as

$$\frac{x}{D} = 0.07\text{Re}$$

where

D = diameter of the pipe ; and

Re = Reynolds number $\left(= \frac{\rho V D}{\mu}\right)$

According to Rouse the distance required for the establishment of a fully developed turbulent flow is given as

$$\frac{x}{D} = 50$$

8. The boundary may be classified as
(i) Hydrodynamically smooth boundary
(ii) Hydrodynamically rough boundary
A boundary is known as hydrodynamically smooth boundary when (k/δ') is less than 0.25 and it is known as hydrodynamically rough boundary when (k/δ') is greater than 6.0, where k is the average height of the irregularities projecting from the boundary surface, and δ' is thickness of the laminar sublayer.
The boundary is classified as boundary in transition when $0.25 < (k/\delta') < 6.0$
9. The velocity distribution for turbulent flow in hydrodynamically smooth pipes is given by

$$\frac{v}{V_*} = 5.75\log_{10}\left(\frac{V_* y}{\nu}\right) + 5.5$$

where

V_* = shear or friction velocity $= \left(= \sqrt{(\tau_0/\rho)}\right)$;

and

ν = kinematic viscosity of the fluid.

10. The velocity distribution fort urbulent flow in hydrodynamically rough pipes is given by

$$\frac{v}{V_*} = 5.75\log_{10}(y/k) + 8.5$$

where

k = average height of irregularities projecting from the boundary surface.

11. The mean velocity V for turbulent flow in hydrodynamically smooth pipes of radius R is given by -

$$\frac{V}{V_*} = 5.75\log_{10}\left(\frac{V_* R}{\nu}\right) + 1.75$$

12. The mean velocity V for turbulent flow in hydrodynamically rough pipes of radius R is given by

$$\frac{V}{V_*} = 5.75\log_{10}\left(\frac{R}{k}\right) + 4.75$$

13. The velocity distribution identical for both hydrodynamically smooth and hydrodynamically rough pipes is given as

$$\frac{v - V}{V_*} = 5.75\log_{10}\left(\frac{y}{R}\right) + 3.75$$

For $y = R$, i.e., at pipe axis $v = v_{max}$, thus

$$\frac{v_{max} - V}{V_*} = 3.75$$

14. The loss of head due to friction for turbulent flow in pipes is given by Darcy-Weisbach equation as

$$h_f = \frac{fLV^2}{2gD}$$

15. For hydrodynamically smooth pipes the friction factor f is given by
(i) Blasius formula

$$f = \frac{0.316}{(\text{Re})^{1/4}} \text{ ...for } 4 \times 10^3 < \text{Re} < 10^5$$

(ii) Karman-Prandlt resistance equation

$$\frac{1}{\sqrt{f}} = 2.0\log_{10}\left(\text{Re}\sqrt{f}\right) - 0.8 \text{ ...for Re} > 10^5$$

(iii) Nikuradse's empirical equation

$$f = 0.0032 + \frac{0.021}{(\text{Re})^{0.237}} \text{ ...for Re} > 10^5$$

16. For hydrodynamically rough pipes the friction factor f is given by
(i) Karman-Prandtl resistance equation

$$\frac{1}{\sqrt{f}} = 2.0\log_{10}\left(\frac{R}{k}\right) + 1.74$$

17. For commercial pipes the friction factor f is given by

$$\frac{1}{\sqrt{f}} - 2.0\log_{10}\left(\frac{R}{k}\right)$$
$$= 1.74 - 2.0\log_{10}\left\{1 + 18.7\frac{R/k}{\text{Re}\sqrt{f}}\right\}$$

18. The pipe behave as hydrodynamically smooth pipe when the parameter $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is less than 17 and the pipe behaves as hydrodynamically rough when the parameter $\left(\frac{\text{Re}\sqrt{f}}{R/k}\right)$ is more than 400,

For $17 < \left(\frac{\text{Re}\sqrt{f}}{R/k}\right) < 400$ the pipe boundary behaves as in transition. This is another criterion for the classification of the boundary surface of pipes.

PROBLEMS

14.1 Define 'shear velocity' or 'friction velocity' for turbulent flow in circular pipes.

14.2 Show that for laminar flow in circular pipes the friction factor is inversely proportional to the Reynolds number.

14.3 What do you understand by hydrodynamically smooth and rough pipes?

14.4 How is head loss in commercial pipes determined?

14.5 Describe Nikuradse's experiments on the resistance of artificially roughened pipes. Discuss the characteristic features of the results obtained.

14.6 With the help of Prandtl's pitot tube installed at mid point and quarter point of a 300 mm diameter pipe, mean point velocities at these points were observed as 1.5 m/s and 1.35 m/s respectively. Assuming the flow to be rough turbulent, determine the discharge, friction factor and relative roughness.

[*Ans.* 83 *l*/s; 0.0436; (*k*/*D*) = (1/66.93)]

14.7 A smooth pipeline 0.1 m in diameter and 1000 m long carries water at the rate of 7.5 litres per second. If the kinematic viscosity of water is 0.02 stokes, calculate the head lost, wall shearing stress, centreline velocity, shear stress and velocity at 40 mm from the centreline, and the thickness of the laminar sublayer.

[*Ans.* 9.95 m; 2.44 N/m² [0.25 kg (f)/m²]; 1.15 m/s; 1.95 N/m² [0.2 kg (f)/m²]; 0.95 m/s; 0.47 mm]

14.8 Rough turbulent flow occurs in a pipe 0.2 m diameter conveying water. If at y = 20 mm, v = 2.0 m/s and $(\partial v/\partial y)$ = 10.5 s⁻¹, determine k, v_0, t_0, f and V. Assume ρ for water as 1000 kg/m³ (102 msl/m³).

[*Ans.* 0.12 mm; 8.82 N/m² [0.90 kg (f)/m²]; 0.017; 2.025 m/s]

14.9 A pipe of diameter 0.3 m is to convey water at 40°C at the rate of 200 litres per second. If the power required to maintain the flow in 100 m length is 61.8 kW (84 h.p.), calculate the value of k, V_{max}, t_0 *and* V^*. Take v for water at 40°C as 0.0075 stokes.

[*Ans.* 0.57 mm; 3.4 m/s; 23.18 N/m² [2.363 kg (f)/m²] ; 0.152 m/s]

14.10 If a pipe 3 m in diameter is to carry a flow of 7.5 m³/s of water with a minimum expenditure of energy, determine the permissible height of the roughness projection on the pipe wall. v for water is 0.015 m² s.

[*Ans.* 0.12 mm]

14.11 A smooth pipe of diameter 0.25 m and length 3000 m carries oil at the rate of 50 litres per second. The oil has specific gravity of 0.85 and viscosity of 0.044 N.s/m². If an increase in temperature causes the viscosity to decrease to 0.025 N.s/m² and if the power input remains the same, determine the change in the rate of flow.

[*Ans.* Increase at the rate of 2.67 *l*/s]

14.12 The oil is pumped through a pipe of diameter 0.4 m at the rate of 500 litres per second to a distance of 10 km. Determine the power input required for the pump if the pump efficiency is 80 percent. Take ρ = 804.42 kg/m³ (82 msl/m³) and μ= 3.924× 10⁻²N.s/m² [4 × 10⁻³ kg (f)⁻ˢ/m²] for the oil and *k* for the pipe as 0.12 mm.

[*Ans.* 2.34 ×10³ kW (3.18 × 10³ h.p.)]

14.13 A 300 mm diameter pipeline carries water at 20°C with a mean velocity of 7.5 m/s. The pipeline is new with no surface irregularities at the beginning, but it was found that the surface irregularities grow at the rate of 0.075 mm per year. Find the number of years after which the surface irregularities will affect the flow. Take υ = 0.01 stokes.

[*Ans.* 2.67 years]

14.14 After 15 years of service a steel water main 0.6 m in diameter is found to require 40% more power to deliver the 300 litres per second, for which it was originally designed. Determine the corresponding magnitude of the rate of roughness increase α. Take υ = 0.015 stokes.

[*Ans.* 0.033 mm per year]

14.15 A smooth wrought iron pipe 0.2 m in diameter carries crude oil at a velocity of 2.5 m/s. What is the loss of head per 100 metres? What is the power required to maintain the flow? Assume kinematic viscosity for oil 0.40 stokes and sp. gr. 0.90.

[*Ans.* 4.78 m; 3.315 kW (4.505 h.p.)]

14.16 Water with kinematic viscosity 0.01 stokes flows in a 200 mm dia. pipe with absolute roughness 0.04 mm with a mean velocity of 1 m/s and the observed head loss is 0.6 m in 100 m length of pipe. If a glycerine-water mixture of kinematic viscosity 0.02 stokes flows in a 100 mm dia pipe of roughness 0.02 mm with a velocity of 4 m/s, how much head loss per 100 m of pipe should be expected?

[*Ans.* 19.2 m]

Flow in Open Channels

Chapter 15

15.1 INTRODUCTION

A channel may be defined as a passage through which water flows under atmospheric pressure. As such in channels the flow of water takes place with a free surface which is subjected to atmospheric pressure. Figure 15.1 shows a comparison between the flow through a pipe and a channel. In the case of flow through a pipe there is no free surface as in the case of flow through a channel. This is so because the water flowing in a pipe is under pressure, which at any section of a pipe is indicated by the level of water column in a piezometric tube installed on the pipe at the section. The water levels in

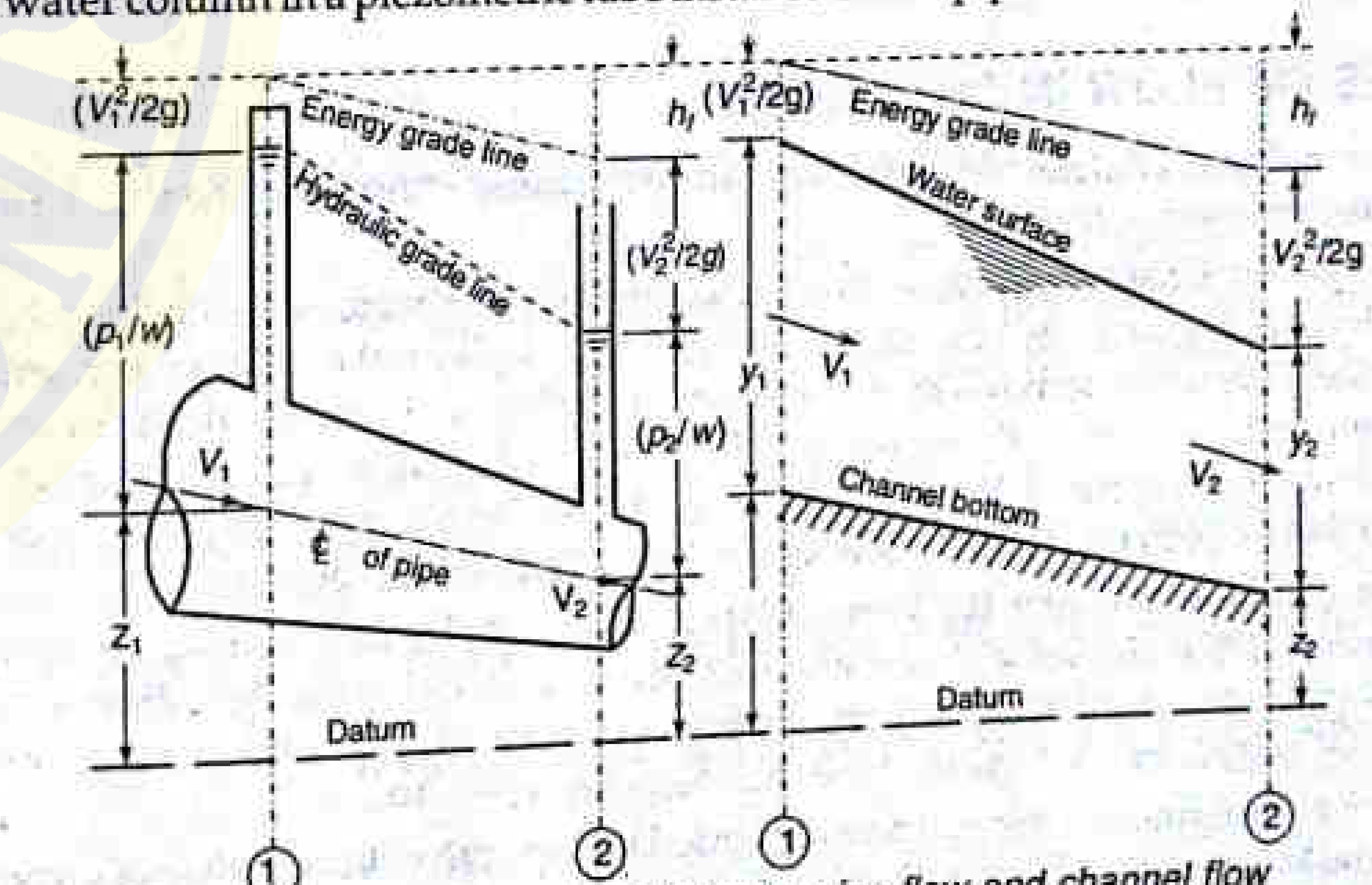


Figure 15.1 *Comparison between pipe flow and channel flow*

the piezometric tubes installed at different sections of a pipe indicate the hydraulic grade line. On the other hand in the case of channel flow the water surface itself is the hydraulic grade line. However, the total energy lines in both the cases lie at a distance of $(V^2/2g)$ above the hydraulic grad line at every section, where V is the mean velocity of flow at the section.

Since the pressure on the top surface of water in a channel is constant, no pressure difference can be built up between any two sections along the channel as in the case of pipes. Moreover, as the water flows in a channel, it will always be subjected to some resistance. As such in order to overcome the resistance and to cause the flow of water in a channel, it is constructed with its bottom slopping towards the direction of flow, so that a component of the weight of the flowing water in the direction of flow is developed which causes the flow of water in the channel.

The channels may be classified according to different considerations as described below. On the basis of the cross-sectional form of the channel it may be classified as *natural channel* or *artificial channel*. A natural channel is the one which has irregular sections of varying shapes, which is developed in a natural way. The examples of natural channels are rivers, streams etc. On the other hand an artificial channel is the one which is built artificially for carrying water for various purposes. Obviously artificial channels have their cross-sections with regular geometrical shapes, which usually remain same throughout the length of the channel. The artificial channels may be further classified according to the shape of the cross-section as, *rectangular channel, trapezoidal channel, triangular channel, parabolic channel*, and *circular channel*.

The channels may also be classified as *open channels* and *closed channels*. The channels without any cover at the top are known as open channels. On the other hand the channels having cover at the top are known as closed channels. However, the closed channels will always be running partly full of water, in order that the flow may be a channel flow with atmospheric pressure prevailing over its entire top surface. Some of the common examples of closed channels are closed conduits or pipes flowing partly full of water, underground drains, tunnels etc., not running full of water.

A channel having the same shape of various sections along its length and laid on a constant bottom slope is known as *prismatic channel*, otherwise the channel is *non-prismatic*. In this chapter only prismatic channels have been dealt with.

15.2 TYPES OF FLOW IN CHANNLES

The flow in channels can be classified into following types depending upon the change in the depth of flow with respect to space and time.

(a) Steady Flow and Unsteady Flow. Flow in a channel is said to be steady if the flow characteristics at any point do not change with time that is $(\partial V/\partial t) = 0$, $(\partial y/\partial t) = 0$ etc. However, in the case of prismatic channels the conditions of steady flow may be obtained if only the depth of flow does not change with time, that is $(\partial y/\partial t) = 0$. On the other hand if any of the flow characteristics changes with time the flow is unsteady. Most of the open channel problems involve the study of flow under steady conditions and therefore in this chapter only steady flow has been considered.

(b) Uniform and Non-uniform (or Varied) Flow. Flow in a channel is said to be uniform if the depth, slope, cross-section and velocity remain constant over a given length of the channel. Obviously, a uniform flow can occur only in a prismatic channel in which the flow will be uniform if only the depth of flow y is same at every section of the channel, that is $(\partial y/\partial s) = 0$.

Flow in channels is termed as non-uniform or varied if the depth of flow y, changes from section to section, along the length of the channel, that is $(\partial y/\partial s)$ is not equal to zero. Varied flow may be further classified as rapidly varied flow (R.V.F.) and gradually varied flow (G.V.F.). If the depth of flow changes abruptly over a comparatively short distance, the flow is characterized as a rapidly varied flow. Typical examples of rapidly varied flow are hydraulic jump and hydraulic drop. In a gradually varied flow the change in the depth of flow takes place gradually in a long reach of the channel.

(c) Laminar Flow and Turbulent Flow. Just as in pipes, the flow in channels may also be characterized as laminar, turbulent or in a transitional state, depending on the relative effect of viscous and inertia forces and alike pipes, Reynolds number Re is a measure of this effect in channel flow also.

However, the Reynolds number for flow in channel is commonly defined as $Re = \left(\frac{\rho V R}{\mu}\right)$ where V is the mean velocity of flow, R is the hydraulic radius (or hydraulic mean depth) of the channel section and ρ and μ are respectively the mass density and absolute viscosity of water. On the basis of the experimental data it has been found that upto Re equal to 500 to 600 the flow in channels may be considered to be laminar and for Re greater than 2000 the flow in channels is turbulent. Thus for Re in between 500 to 2000 the flow in channels may be considered to be in transitional state. Since most of the channel flows are turbulent, in this chapter only the turbulent flow has been dealt with.

(d) Subcritical Flow, Critical Flow and Supercritical Flow. As stated earlier gravity is a predominant force in the case of channel flow. As such depending on the relative effect of gravity and inertia forces the channel flow may be designated as subcritical, critical or supercritical. The ratio of the inertia and the gravity forces is another dimensionless parameter called Froude number Fr which is defined as $Fr = \frac{V}{\sqrt{gD}}$; where V is the mean velocity of flow, g is acceleration due to gravity and D is hydraulic depth of channel section (as defined in Section 15.3). As indicated later, when $Fr = 1$, that is $V = \sqrt{gD}$, the channel flow is said to be in a critical state. If $Fr < 1$, or $V < \sqrt{gD}$, the flow is described as subcritical or tranquil or streaming. If $Fr > 1$, or $V > \sqrt{gD}$, the flow is said to be supercritical or rapid or shooting or torrential.

15.3 GEOMETRICAL PROPERTIES OF CHANNEL SECTION

The geometrical properties of a channel section can be defined entirely by the geometry of the section and the depth of flow. Some of the geometrical properties of basic importance are defined below.

The depth of flow y, is the vertical distance of the lowest point of a channel section from the free surface.

The top width T is the width of the channel section at the free surface.

The wetted area A (or water area or area of flow section) is the cross-sectional area of the flow normal to the direction of flow.

The wetted perimeter P is the length of the channel boundary in contact with the flowing water at any section.

The hydraulic radius R (or hydraulic mean depth) is the ratio of the wetted area to its wetted perimeter,

or $$R = \left(\frac{A}{P}\right)$$

The hydraulic depth D is the ratio of the wetted area to the top width T,

or $$D = \left(\frac{A}{T}\right)$$

The section factor for critical flow computation Z is the product of the wetted area and the square root of the hydraulic depth,

TABLE 15.1 *Geometric Elements of Channel Section*

	Section	Area A	Wetted perimeter P	Hydraulic radius R	Top width T	Hydraulic depth D	Section factor Z
Rectangle		By	$B + 2y$	$\frac{By}{B+2y}$	B	y	$B\,(y)^{1.5}$
Trapezoid		$(B + zy)\,y$	$B+2y\sqrt{z^2+1}$	$\frac{(B+zy)y}{B+2\sqrt{z^2+1}}$	$B + 2zy$	$\frac{(B+zy)y}{B+2zy}$	$\frac{[(B+zy)y]^{1.5}}{\sqrt{B+2zy}}$
Triangle		zy^2	$2y\sqrt{z^2+1}$	$\frac{zy}{2\sqrt{z^2+1}}$	$2zy$	$\frac{1}{2}y$	$\frac{\sqrt{2}}{2}z(y)^{2.5}$
Circle		$\frac{1}{8}(\theta-\sin\theta)D^2$	$\frac{1}{2}\theta D$	$\frac{1}{4}\left(1-\frac{\sin\theta}{\theta}\right)D$	$D\left(\sin\frac{\theta}{2}\right)$, or $2\sqrt{y(D-y)}$	$\frac{1}{8}\left(\frac{\theta-\sin\theta}{\sin\frac{\theta}{2}}\right)D$	$\frac{\sqrt{2}}{32}\frac{(\theta-\sin)^{1.5}}{\left(\sin\frac{\theta}{2}\right)^{0.5}}D^{2.5}$
Parabola		$\frac{2}{3}Ty$	$T+\frac{\theta}{3}\frac{y^2}{T}$	$\frac{2T^2y}{3T^2+By^2}$	$\frac{3A}{2Y}$	$\frac{2}{3}y$	$\frac{2}{9}\sqrt{6}Ty^{1.5}$

or

$$Z = A\sqrt{D} = A\sqrt{\left(\frac{A}{T}\right)} = \left(\frac{A^3}{T}\right)^{1/2}$$

The section factor for uniform flow computation $AR^{2/3}$ is the product of the wetted area and the hydraulic radius to two-thirds power.

Shown in Table 15.1 provides a list of the expressions for the geometrical properties of five commonly used channel sections.

15.4 VELOCITY DISTRIBUTION IN A CHANNEL SECTION

The velocity of flow at any channel section is not uniformly distributed. The non-uniform distribution of velocity in an open channel is due to the presence of a free surface and the frictional resistance along the channel boundary. The velocity distribution in a channel is measured either with the help of a pitot tube or a current meter, described later. The general patterns for velocity distribution as represented by lines of equal velocity, in some of the common channel sections are illustrated in Fig. 15.2. A velocity distribution curve along a vertical line of the channel section is also shown in Fig. 15.2. In a straight reach of a channel maximum velocity usually occurs below the free surface at a distance of 0.05 to 0.15 of the depth of flow. The velocity distribution in a channel section depends on the various factors such as the shape of the section, the roughness of the channel and the presence of bends in the channel alignment.

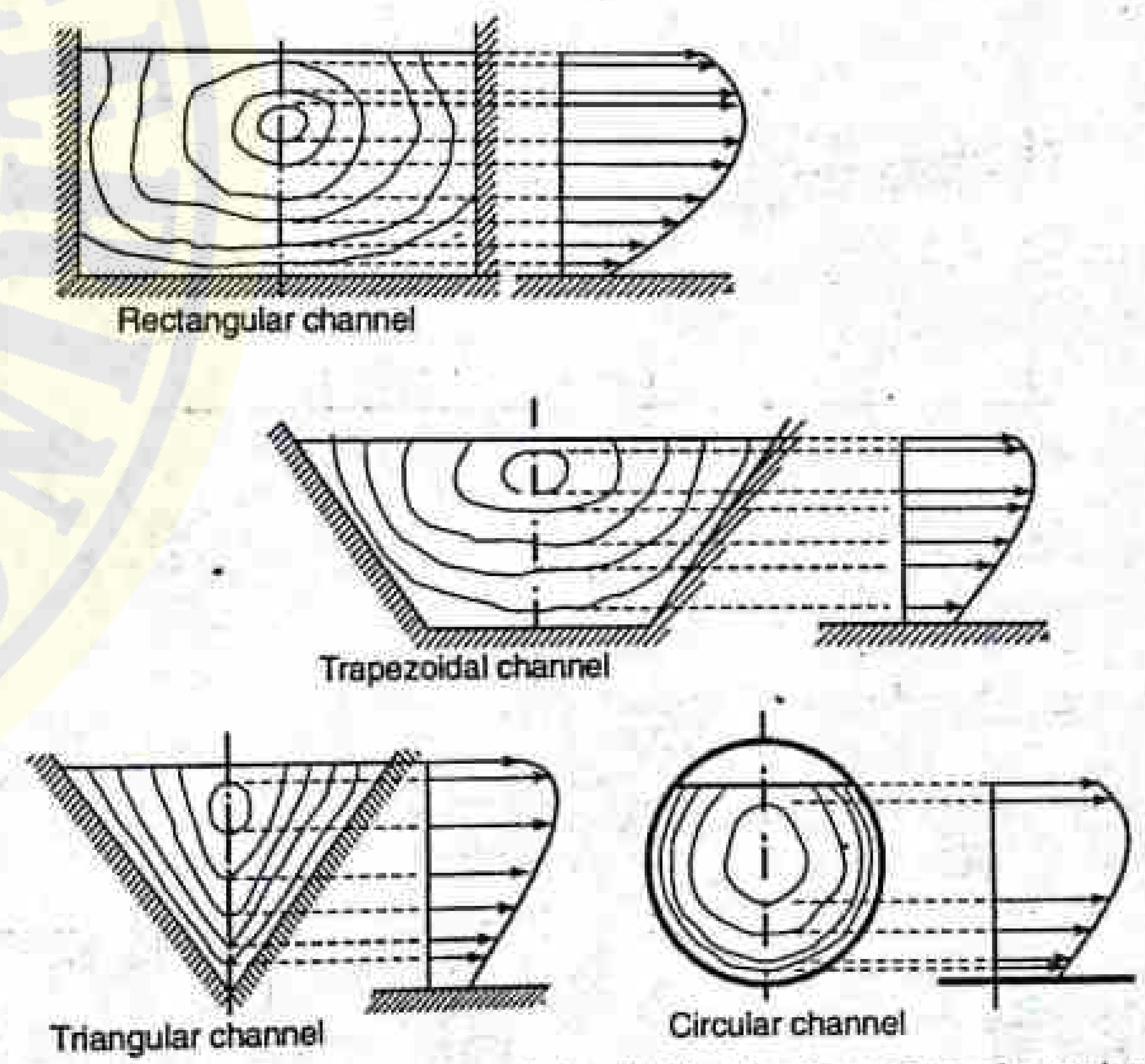


Figure 15.2 *Typical patterns of velocity distribution in various channel sections*

The mean velocity of flow in a channel section can be computed from the vertical velocity distribution curve obtained by actual measurements. It is observed that the velocity at 0.6 depth from the free surface is very close to the mean velocity of flow in the vertical section. A still better approximation for the mean velocity of flow is obtained by taking the average of the velocities measured at 0.2 depth and 0.8 depth from the free surface.

On account of non-uniform distribution of velocities over a channel section, correction factors will have to be applied while computing the kinetic energy and the momentum. The kinetic energy correction factor α (Greek 'alpha') is also known as Coriolis coefficient in honour of G. Coriolis who first proposed it. The momentum correction factor β (Greek 'beeta') is also called Boussinesq coefficient after J. Boussinesq who first proposed it. As mentioned in previous chapters the values of these correction factors can be obtained if the actual velocity profile at a section is known. Experimental data indicate that for turbulent flow in fairly straight prismatic channels, the value of α varies from 1.03 to 1.36 and that of β from about 1.01 to 1.12. However, for the sake of simplicity, both α and β are assumed to be unity in the present analysis.

15.5 UNIFORM FLOW IN CHANNLES

When water flows in an open channel resistance is offered to it, which results in causing a loss of energy. The resistance encountered by the flowing water is generally counteracted by the components of gravity forces acting on the body of the water in the direction of motion [Fig. 15.3 (b)]. A uniform flow

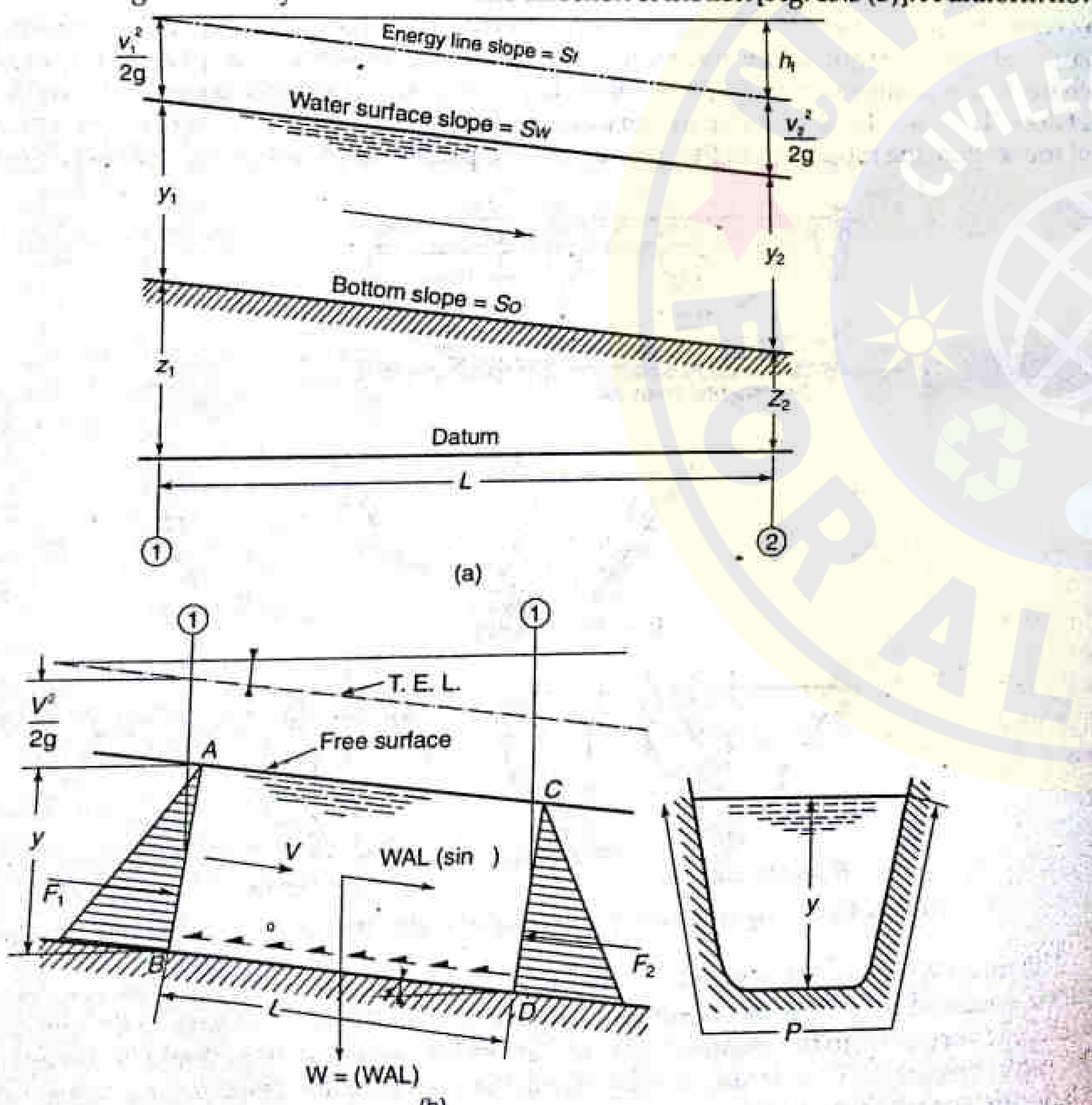


Figure 15.3 *(a) Uniform flow in open channel (b) Forces on a segment of channel having uniform flow*

will be developed if the resistance is balanced by the gravity forces. The magnitude of the resistance, when other physical factors of the channel are kept unchanged, depends on the velocity of flow. When water enters the channel, the velocity and hence the resistance are smaller than the gravity forces, which results in an accelerating flow in the upstream reach of the channel. The velocity and the resistance increase gradually until a balance between the resistance and gravity forces is reached. From this point onwards the flow becomes uniform. Several *uniform-flow formulae* have been developed which correlate the mean velocity of uniform flow in open channels with the hydraulic radius, energy line slope and a factor of flow resistance. The most widely used uniform-flow formulae are the Chezy and Manning formulae which are described in the following paragraphs.

The main features of uniform flow in a channel can be summarized as follows:

(1) The depth of flow, wetted area, velocity of flow and discharge are constant at every section along the channel reach.

(2) The total energy line, water surface and the channel bottom are all parallel, that is, their slopes are all equal, or $S_f = S_w = S_0 = S$, where S_f is energy line slope, S_w is water surface slope and S_0 is channel bottom slope.

Figure 15.3 (a) shows a short reach of prismatic channel having uniform flow. Applying Bernoulli's equation between sections 1 and 2 at a distance L apart, we obtain

$$y_1 + \frac{V_1^2}{2g} + Z_1 = y_2 + \frac{V_2^2}{2g} + Z_2 + h_f$$

where h_f is the loss of energy between the two sections. Since the flow is uniform, $y_1 = y_2$; $V_1 = V_2$. Thus from the above expression

$$Z_1 - Z_2 = h_f$$

or

$$\frac{Z_1 - Z_2}{L} = S_0 = \frac{h_f}{L} = S_f = S_w.$$

The depth of a uniform flow is called the *normal depth* and it is generally represented by y_n.

The fundamental equation for uniform flow in channels may be derived by applying Newton's second law of motion. In uniform flow since the velocity of flow does not change along the length of the channel, there is no acceleration. Hence the sum of the components of all the external forces in the direction of flow must be equal to zero. Consider a short reach of channel of length L in which uniform flow occurs, as shown in Fig. 15.3 (b). The forces acting on the free-body of water $ABCD$ in the direction of flow are as follows:

(1) The forces of hydrostatic water pressure F_1 and F_2 acting on the two ends of the freebody. As the depths of water at these two sections are the same, the forces F_1 and F_2 are equal and hence they cancel each other.

(2) The component of weight of the water in the direction of flow, which is ($wAL \sin \theta$), where w is specific weight of water, A is the wetted cross-sectional area of channel and θ is the angle of inclination of the channel bottom with the horizontal.

(3) The resistance to the flow is exerted by the wetted surface of the channel. If P is the wetted perimeter of the channel and τ_0 is the average shear stress at the channel boundary, the total resistance to flow will be ($PL\tau_0$).

According to Newton's second law of motion, in the direction of flow

or
$$wAL\sin\theta - PL\tau_0 = 0$$
$$\tau_0 = w\sin\theta = wRS_0 \qquad ...(15.1)$$

where R is the hydraulic radius and S_0 (= sin θ = S) is the slope of the channel bottom which for small values of θ may be considered equal to tan θ, since sin θ ≃ tan θ.

In Chapter 13 the shear stress τ_0 has been related to mass density ρ and average velocity V by the equation $\tau_0 = \frac{f}{8}\rho V^2$. Thus equating the two expressions for τ_0, we get

$$wRS = \frac{f}{8}\rho V^2$$

or
$$V = \sqrt{\frac{8w}{\rho f}}\sqrt{RS} = \sqrt{\frac{8g}{f}}\sqrt{RS}$$

Since
$$(w/\rho) = g$$

The above expression is generally written as

$$V = C\sqrt{RS} \qquad ...(15.2)$$

in which
$$C = \sqrt{\left(\frac{8g}{f}\right)}$$

Equation 15.2 is known as Chezy's formula named after the French Engineer Antoine Chezy who developed this formula in 1775. The term C is known as Chezy's coefficient. It may, however, be observed that the Chezy's coefficient C varies inversely as the square root of the Darcy-Weisbach resistance coefficient f. Further it may be noted that Chezy's coefficient is not a dimensionless coefficient but it has the dimensions $[L^{1/2}\,T^{-1}]$. Although Chezy's equation is quite simple the selection of a correct value of C is rather difficult. As such many empirical formulae have been developed to determine the value of Chezy's C. Some of the important formulae developed for this purpose are as noted below.

(a) The Ganguillet-Kutter Formula. On the basis of a series of flow measurements taken in channels of various types, two Swiss engineers Ganguillet and Kutter proposed an empirical formula in 1869 expressing the value of Chezy's C in terms of the slope S, hydraulic radius R and a roughness coefficient n. In SI or metric system the formula is

$$C = \frac{23 + \dfrac{0.00155}{S} + \dfrac{1}{n}}{1 + \left(23 + \dfrac{0.00155}{S}\right)\dfrac{n}{\sqrt{R}}} \qquad ...(15.3)$$

The roughness coefficient n is known as Kutter's n. The value of n varies widely depending upon the channel surface and its condition. Some typical values of n for different types of surfaces commonly encountered in practice are given in Table 15.3.

(b) The Bazin Formula. In 1897, the French hydraulician H. Bazin proposed another empirical formula for Chezy's C. According to this formula Chezy's C is expressed in terms of hydraulic radius R and another roughness factor m, known as Bazin's roughness coefficient. In metric system Bazin's formula may be expressed as

$$C = \frac{157.6}{1.81 + \dfrac{m}{\sqrt{R}}} \qquad ...(15.4)$$

Some of the values of m proposed by Bazin are given in Table 15.2.

TABLE 15.2 *Value of Bazin's m for Various Surface Materials*

Type of Channel Boundary Surface	*m*
Very smooth cement, or planed wood	0.11
Concrete or brick or unplaned wood	0.21
Ashlar, rubble masonry or poor brick work	0.83
Earth channels in very good condition	1.54
Earth channels in ordinary condition	2.36
Earth channels in rough condition	3.17

(c) The Manning Formula. In 1889 an Irish engineer Robert Manning presented a formula according to which the mean velocity V of uniform flow in a channel is expressed in terms of a coefficient of roughness n, called Manning's n, hydraulic radius R and channel bottom slope S. The Manning's formula expressed in SI or metric units is written as

$$V = \frac{1}{n}R^{2/3}S^{1/2} \qquad ...(15.5)$$

Owing to its simplicity of form and to the satisfactory result it yields in practice, the Manning formula has now becomes the most widely used of all the empirical formula for the computation of uniform flow in open channels.

If the Manning formula is compared with the Chezy formula it can be seen that

$$C = \frac{1}{n}R^{1/6} \qquad ...(15.6)$$

The Eq. 15.6 provides an important relationship between Chezy's C and Manning's n.

The values of Manning's n are found to be approximately equal to those of Kutter's n for the normal ranges of slope and hydraulic radius. Some of the typical values which hold good for both Kutter's n and Manning's n are given in Table 15.3.

TABLE 15.3 *Average Values of Manning's Roughness Coefficient n*

Type of channel boundary surface	*Value of n*
Very smooth surface such as glass, plastic or brass	0.010
Very smooth concrete and planed timber0.011	
Smooth concrete	0.012
Ordinary concrete lining	0.013
Glazed brick work	0.013
Good wood	0.014
Vitrified clay	0.014
Brick surface lined with cement mortar	0.015
Cement concrete finish	0.015
Unfinished cement surface	0.017

Contd.

Table 15.3 Contd.

Type of channel boundary surface	Value of n
Earth channel in best condition	0.017
Neatly excavated rock	0.020
Straight unlined earth canals in good condition	0.020
Rubble masonry	0.020
Corrugated metal surface	0.020
Rivers and earth canals in fair condition 0.025	
Earth channel with gravel bottom	0.025
Earth channel with dense weed	0.035
Mountain stream with rock beds and rivers with variable sections and some vegetation along banks	0.045

In addition to the above mentioned uniform-flow formulae, a few other formulae for uniform flow in channels have been derived on the basis of theoretical velocity distribution across the channel section. For turbulent flow in channels the velocity distribution can be shown to be approximately logarithmic. Thus using the Prandtl-von Karman universal velocity-distribution law viz.,

$$[(v/V_*) = 2.5 \log_e y + C],$$

Keulegan has derived the following equations for the mean velocity of uniform turbulent flow in channels.

For smooth channels

$$(V/V_*) = 3.25 + 5.75 \log_{10}\left(\frac{RV_*}{\upsilon}\right) \quad \text{...(15.7)}$$

For rough channels

$$(V/V_*) = 6.25 + 5.75 \log_{10}(R/k) \quad \text{...(15.8)}$$

where V is mean velocity of flow, V_* is shear velocity, R is hydraulic radius, υ is kinematic viscosity and k is average height of roughness protrusions.

From Eq. 15.1, $\quad V_* = \sqrt{(\tau_0/\rho)} = \sqrt{gRS}\,;$

and from Chezy's formula $\quad V = C\sqrt{RS}$

$$\text{Thus} \quad (V/V_*) = (C/\sqrt{g}) \quad \text{...(15.9)}$$

Substituting Eq. 15.9 and the Reynolds number Re $= (VR/\upsilon)$ in Eqs 15.7 and 15.8, the following expressions for Chezy's C are obtained.

For smooth channels

$$C = 18 \log_{10}\left(\frac{11.5Re}{C}\right) \quad \text{...(15.10)}$$

For rough channels

$$C = 18 \log_{10}\left(\frac{12.2R}{k}\right) \quad \text{...(15.11)}$$

However, some of the later investigators have indicated that the values of the additive constants in Keulegan's Eqs 15.7 and 15.8 vary with Froude number.

For rough channels, by eliminating Chezy's C from Eqs 15.6 and 15.11, Manning's n may be expressed as follows.

$$\text{Thus} \quad n = \phi(R/k)k^{1/6} \quad \text{... (15.12)}$$

$$\text{where} \quad \phi(R/k) = \frac{(R/k)^{1/6}}{18\log(12.2R/k)} \quad \text{...(15.13)}$$

For a wide range of (R/k), the variation in the values of $\phi(R/k)$ is small and hence as an approximation, $\phi(R/k)$ may be assumed to be constant. On the basis of actual obervations, Strickler has given a formula for n, which when compared with Eq. 15.12, gives an average value of $\phi(R/k) = 0.0476$ or $(1/21)$. Thus according to Strickler formula

$$n = \frac{k^{1/6}}{21} \quad \text{...(15.14)}$$

in which k is in metres. It may however be stated that the roughness height used by Strickler is the median sieve size of the material (or d_{50} which represents such a size that fifty per cent of the material is finer than d_{50}). One of the later investigators has however shown that the value of the constant in the Strickler formula should be 24 instead of 21, in which case Eq. 15.14 becomes

$$n = \frac{k^{1/6}}{24} \quad \text{...(15.14 a)}$$

Equation 15.14 leads to the conclusion that Manning's n varies with one-sixth power of the roughness height and hence a slight error in the value of roughness height does not have much effect on the value of Manning's n computed from Eq. 15.14. However, it is extremely difficult to predict the correct value of n or C, since each of these parameters is highly variable and depends on several factors.

15.6 MOST ECONOMICAL OR MOST EFFICIENT SECTION OF CHANNEL

A channels section is considered to be the most economical or most efficient when it can pass a maximum discharge for given cross-sectional area, resistance coefficient and bottom slope. From the equation of continuity it is evident that for area of cross-section being constant, discharge is maximum when velocity of flow is maximum. From Chezy formula and Manning formula it can be seen that for a given value of slope and surface roughness the velocity of flow will be maximum if hydraulic radius R is maximum. Further the area being constant hydraulic radius is maximum if the wetted perimeter is minimum. As such this condition is utilized to determine the dimensions of economical sections of different forms of channels as discussed below.

(a) Most Economical or Most Efficient Rectangular Channel Section. For a rectangular channel section of bottom width B and depth of flow y (Figure15.4), following expressions for wetted area A and wetted perimeter P can be written

$$A = By \quad \text{...(i)}$$

$$P = B + 2y \quad \text{...(ii)}$$

From Eq. (i)

$$B = \frac{A}{y}$$

Substituting the value of B in Eq. (ii) it becomes

$$P = \frac{A}{y} + 2y \qquad \text{...(iii)}$$

ssuming the area A to be constant, Eq. (iii) can be differentiated with respect to y and equated to zero or obtaining the condition for minimum P.

Thus
$$\frac{dP}{dy} = -\frac{A}{y^2} + 2 = 0$$

$$A = 2y^2 = By$$

∴
$$B = 2y$$

$$y = \frac{B}{2} \qquad \text{...(15.15)}$$

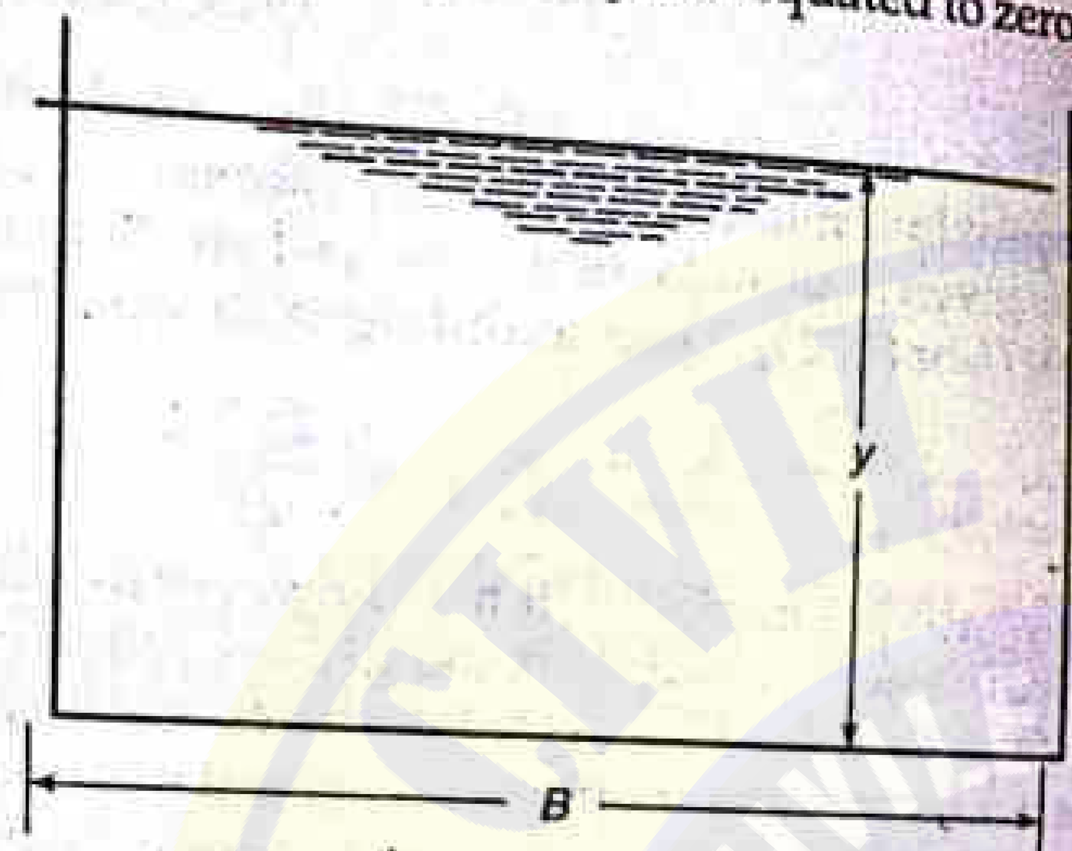


Figure 15.4 *Rectangular channel section*

lso hydraulic radius
$$R = \frac{A}{P} = \frac{By}{B+2y}$$

Substituting the value of

$$B = 2y$$

$$R = \frac{2y^2}{2y+2y} = \frac{y}{2} \qquad \text{... (15.16)}$$

Thus it can be seen that a rectangular channel section will be most economical or most efficient hen either the depth of flow is equal to half the bottom width, or hydraulic radius is equal to half the epth of flow.

(b) Most Economical or Most Efficient Trapezoidal Channel Section. For a trapezoidal channel ction of bottom width B, depth of flow y and side slope z horizontal to 1 vertical (Fig. 15.5), following pression for wetted area A and wetted perimeter P can be written.

$$A = (B + zy)\,y \qquad \text{... (i)}$$

$$P = B + 2y\sqrt{1+z^2} \qquad \text{... (ii)}$$

From Eq. (i)

$$B = \frac{A}{y} - zy$$

Substituting the value of B in Eq. (ii) it becomes

$$P = \frac{A}{y} - zy + 2y\sqrt{1+z^2} \qquad \text{... (iii)}$$

uming area A and side slope z to be constant, Eq. (iii) can be differentiated with respect to y and ated to zero for obtaining the condition for minimum P.

Thus
$$\frac{dP}{dy} = -\frac{A}{y^2} - z + 2\sqrt{1+z^2} = 0$$

or
$$\frac{A}{y^2} + z = 2\sqrt{1+z^2}$$

Substituting the value of A from Eq. (i) the above equation becomes

$$\frac{(B+zy)y}{y^2} + z = 2\sqrt{1+z^2}$$

or
$$\frac{B+2zy}{2} = y\sqrt{1+z^2} \qquad \text{... (15.17)}$$

Equation 15.17 is the required condition for a trapezoidal channel section to be most economical or most efficient. It shows that for a trapezoidal channel section to be most economical or most efficient half the top width must be equal to one of the sloping sides of the channel.

The hydraulic radius R can be expressed as

$$R = \frac{A}{P}$$

$$= \frac{(B+zy)y}{B+2y\sqrt{1+z^2}}$$

Substituting the values of B from Eq. 15.17 in the above expression, we get

$$R = \frac{(2y\sqrt{1+z^2} - 2zy + zy)y}{2y\sqrt{1+z^2} - 2zy + 2y\sqrt{1+z^2}}$$

or
$$R = \frac{y}{2} \qquad \text{... (15.18)}$$

Thus it can be seen that for a trapezoidal channel section to be most economical or most efficient the hydraulic radius must be equal to half the depth of flow.

Further let O be the centre of top width of the trapezoidal channel section. Drop perpendicular OA from O to the slopping side of the channel, (Fig. 15.5.) If θ is the angle made by the slopping side with the horizontal, then from the right-angled triangle OAC, $OA = OC \sin\theta$.

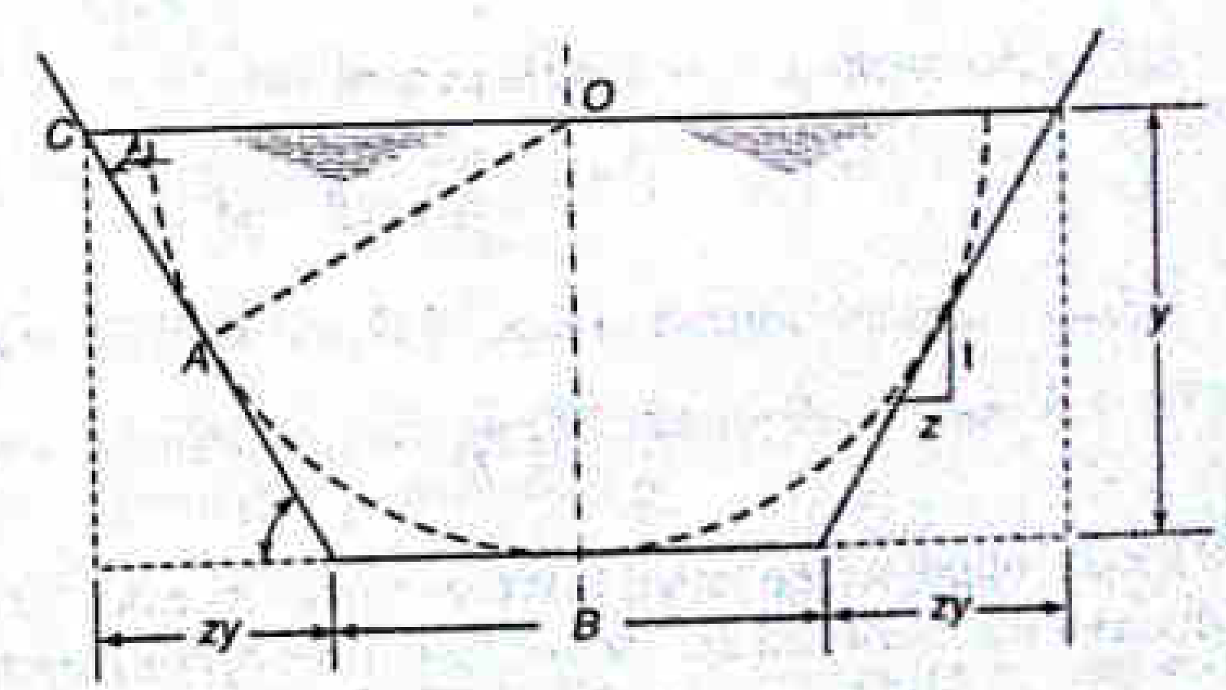


Figure 15.5 *Trapezoidal channel section*

But
$$OC = \left(\frac{B+2zy}{2}\right)$$

and

$$\sin\theta = \frac{y}{y\sqrt{1+z^2}}$$

$$\therefore \quad OA = \frac{B+2zy}{2} \times \frac{y}{y\sqrt{1+z^2}}$$

$$= \frac{B+2zy}{2\sqrt{1+z^2}}$$

Now substituting for $(B + 2zy)$ from Eq. 15.17, we obtain for most economical or most efficient trapezoidal channel section

$$OA = y \qquad \text{... (15.19)}$$

That is, if a semi-circle is drawn with O as centre and radius equal to the depth of flow y, the three sides of a most economical or most efficient trapezoidal channel section, viz., the bottom and the two slopping sides will be tangential to the semi-circle.

Thus Eqs. 15.17, 15.18 and 15.19 represent the three conditions, either of which must be satisfied for a trapezodial channel section to be most economical or most efficient.

The above derived conditions are based on the assumption that side slope z is constant. As such these conditions will not provide a most economical or most efficient trapezoidal channel section if either the bottom width or the depth of flow has to be limited due to physical considerations. However, it is possible to derive the conditions which should be satisfied for a trapezoidal channel section to be most economical or most efficient when either the bottom width or the depth of flow has to be kept constant. By adopting the same procedure as indicated above, the following two conditions can be obtained which may be used for the design of a most economical or most efficient trapezoidal channel section.

(1) For the bottom width to be kept constant

$$\frac{B}{y} = \frac{1-z^2}{z} \qquad \text{... (15.20)}$$

or

$$A = \frac{y^2}{z} \qquad \text{... (15.21)}$$

(2) For the depth of flow to be kept constant

$$z = \frac{1}{\sqrt{3}}; \text{or } \theta = 60° \qquad \text{... (15.22)}$$

If Eq. 15.22 is introduced in Eqs. 15.17 and 15.20, then in both the cases, we get

$$\frac{B}{y} = \frac{2}{\sqrt{3}}$$

It may, therefore, be stated that the most economical or most efficient trapezoidal channel section, which would satisfy all the above noted three conditions, may be obtained by adopting side slope $z = (1/\sqrt{3})$ in the design of the channel section. This fact is indicated in Example 15.2.

(c) Most Economical or Most Efficient Triangular Channel Section. For a triangular channel section, if θ is the angle of inclination of each of the slopping sides with the vertical and y is the depth of flow (Fig.15.6), the following expression for the wetted area A and the wetted perimeter P can be written

$$A = y^2 \tan\theta$$

or

$$y = \sqrt{A/\tan\theta} \qquad \text{... (i)}$$

$$P = (2y)\sec\theta \qquad \text{... (ii)}$$

Substituting the value of y from Eq. (i) in Eq. (ii) it becomes

$$P = \frac{2\sqrt{A}}{\sqrt{\tan\theta}}(\sec\theta) \qquad \text{... (iii)}$$

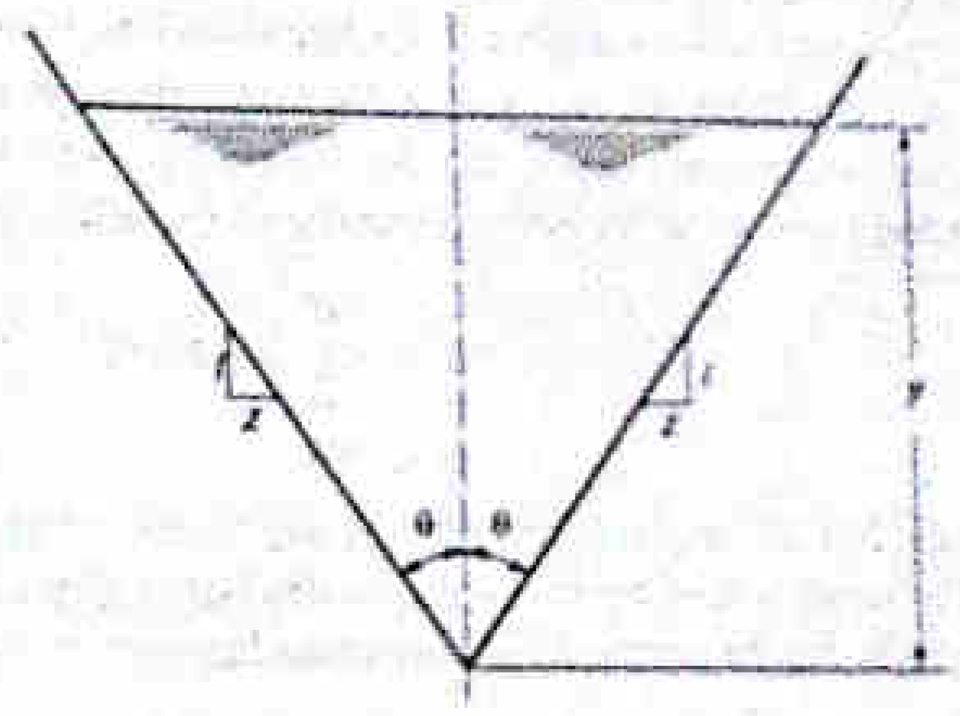


Figure 15.6 *Triangular channel section*

Assuming the area A to be constant, Eq. (iii) can be differentiated with respect to θ and equated to zero for obtaining the condition for minimum P.

Thus
$$\frac{dP}{d\theta} = 2\sqrt{A}\left[\frac{\sec\theta\tan\theta}{\sqrt{\tan\theta}} - \frac{\sec^3\theta}{2(\tan\theta)^{3/2}}\right] = 0$$

or
$$\sec\theta\,(2\tan^2\theta - \sec^2\theta) = 0$$

Since
$$\sec\theta \neq 0$$

$$2\tan^2\theta - \sec^2\theta = 0$$

or
$$\sqrt{2}\tan\theta = \sec\theta$$

$$\therefore \quad \theta = 45°; \text{or } z = 1 \qquad \text{... (15.23)}$$

Hence a triangular channel section will be most economical or most efficient when each of its slopping sides makes an angle of 45° with the vertical.

The hydraulic radius R of a triangular channel section can be expressed as

$$R = \frac{A}{P} = \frac{y^2\tan\theta}{2y\sec\theta}$$

Substituting the value of θ from Eq. 15.23 in the above expression

$$R = \frac{y}{2\sqrt{2}} \qquad \text{... (15.24)}$$

Thus it can be seen that the most economical or most efficient triangular channel section will be half square described on a diagonal and having equal slopping sides.

It may, however, be noted that in all these cases the conditions for the discharge to be maximum would be the same if instead of the area of cross-section the perimeter is given.

(d) Most Economical or Most Efficient Circular Channel Section. For a circular channel section of any radius, as the depth of flow varies, the shape of the flow area also varies due to the convergence of the boundaries towards the top. As such both the wetted area as well as the wetted perimeter varies with the depth of flow, and hence in the case of circular channels the condition of area of flow section being constant cannot be applied. Thus in the case of circular channels two separate conditions may be derived for the maximum discharge and the maximum mean velocity of flow, as explained below.

(i) ***Condition for maximum discharge through a circular channel section.*** For a circular channel section of radius r if y is the depth of flow, as shown in Fig. 15.7, the following expression for the wetted area A and the wetted perimeter P can be written

$$A = \frac{r^2}{2}(\theta - \sin\theta) \qquad \text{...(i)}$$

$$P = r\theta \qquad \text{...(ii)}$$

where θ is the angle subtended at the centre by the portion of circular arc in contact with water. In accordance with Chezy' formula, discharge Q passing through the channel may be expressed as

$$Q = AC\sqrt{RS} = CA\left(\sqrt{\frac{A}{P}S}\right)$$

$$= C\sqrt{\frac{A^3}{P}S}$$

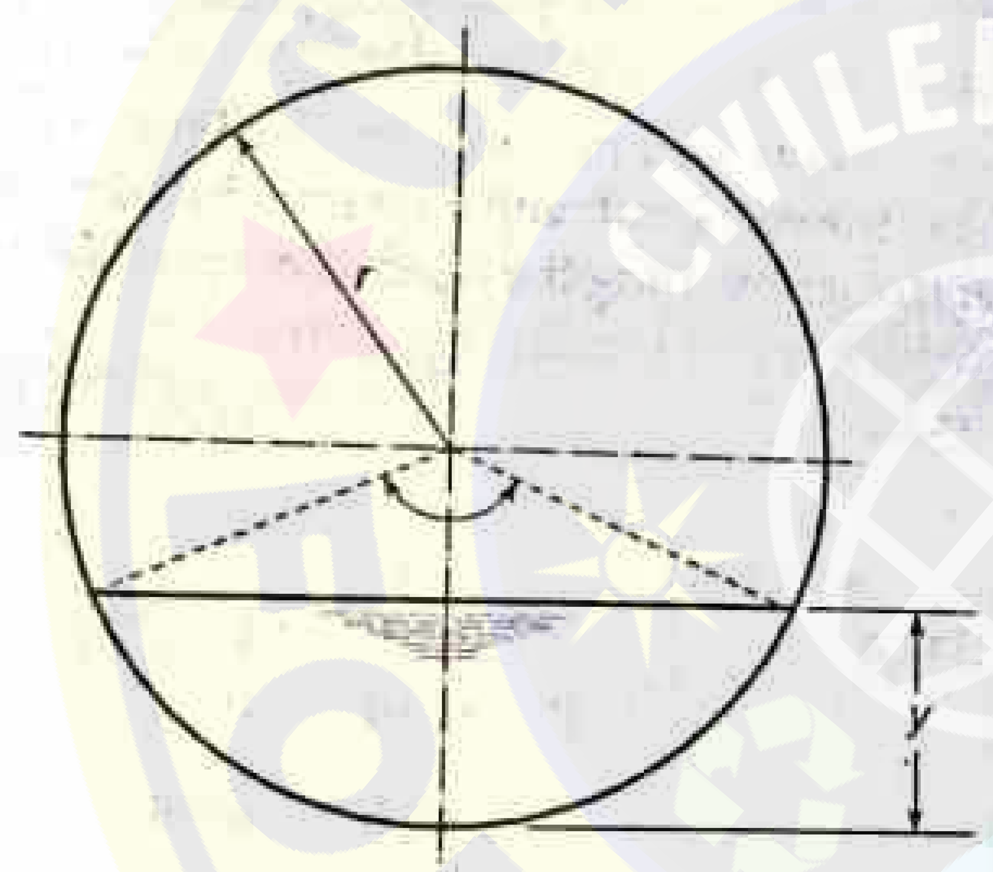


Figure 15.7 *Circular channel section*

where S is the bottom slope and C is Chezy coefficient. Assuming C and S to be constant, the discharge is maximum when $\left(\frac{A^3}{P}\right)$ is maximum. Thus differentiating $\left(\frac{A^3}{P}\right)$ with respect to θ and equating to zero for a maxima, we get

$$\frac{d\left(\frac{A^3}{P}\right)}{d\theta} = \frac{P\times 3A^2\left(\frac{dA}{d\theta}\right) - A^3\left(\frac{dP}{d\theta}\right)}{P^2} = 0$$

or $$3P\frac{dA}{d\theta} - A\frac{dP}{d\theta} = 0$$

But from Eqs. (i) and (ii)

$$\frac{dA}{d\theta} = \frac{r^2}{2}(1-\cos\theta)$$

and $$\frac{dP}{d\theta} = r$$

Therefore substituting for A, P, $\left(\frac{dA}{d\theta}\right)$ and $\left(\frac{dP}{d\theta}\right)$ the above expression becomes

$$\frac{3}{2}r^3\theta(1-\cos\theta) - \frac{r^3}{2}(\theta - \sin\theta) = 0$$

or $$3\theta(1-\cos\theta) - (\theta - \sin\theta) = 0$$

or $$2\theta - 3\theta\cos\theta + \sin\theta = 0$$

The solution of this equation is $\theta = 308°$ (approximately).

Then for maximum discharge the depth of flow in the channel is

$$y = r + r\cos\left(180° - \frac{\theta}{2}\right)$$

$$= r(1+\cos 26°)$$

$$= 1.8988r = 0.95\,D \qquad \text{...(15.25)}$$

where D is the diameter of the channel.

It is thus shown that for maximum discharge in a circular channel the depth of flow is equal to 0.95 times the diameter of the channel.

Further hydraulic radius for a circular channel is

$$R = \frac{A}{P} = \frac{r}{2\theta}(\theta - \sin\theta)$$

Substituting the value of θ as 308° for maximum discharge

$$R = \frac{r}{2\pi\left(\frac{308}{180}\right)}\left(\frac{\pi\times 308}{180} - \sin 308°\right)$$

$$= 0.5733\,r = 0.29\,D \qquad \text{...(15.25 a)}$$

That is, for maximum discharge in a circular channel the hydraulic radius is equal to 0.29 times the diameter of the channel.

The above derived conditions for the maximum discharge in a circular channel are based on the Chezy's formula which has been used in the above analysis. However, if instead of Chezy's formula Manning's formula is considered then a slightly different condition is obtained for maximum discharge in a circular channel as indicated below.

According to Manning's formula the discharge through the channel may be expressed as

$$Q = \frac{1}{n}AR^{2/3}S^{1/2} = \frac{1}{n}\left(\frac{A^5}{P^2}\right)^{1/3}S^{1/2}$$

Assuming n and S to be constant the discharge is maximum when $\left(\frac{A^5}{P^2}\right)$ is maximum. Thus differentiating $\left(\frac{A^5}{P^2}\right)$ with respect to θ and equating to zero for a maxima, we get

$$\frac{d\left(\frac{A^5}{P^2}\right)}{d\theta} = \frac{P^2\times 5A^4\left(\frac{dA}{d\theta}\right) - A^5\times 2P\left(\frac{dP}{d\theta}\right)}{P^4} = 0$$

or $$\left(5P\frac{dA}{d\theta} - 2A\frac{dP}{d\theta}\right) = 0$$

Substituting the values of $A, P, \left(\frac{dA}{d\theta}\right)$ and $\left(\frac{dP}{d\theta}\right)$ the above expression becomes

$$\frac{5}{2}r^3(1-\cos\theta) - r^3(\theta - \sin\theta) = 0$$

or $$3\theta - 5\theta\cos\theta + 2\sin\theta = 0$$

The solution of this equation is $\theta = 302°\ 20'$ (approximately).

The corresponding depth of flow is then

$$y = r + r\cos\left(180° - \frac{\theta}{2}\right)$$
$$= r(1 + \cos 28°50')$$
$$= 1.876\,r = 0.938\,D.$$

That is, when Manning's formula is used, for maximum discharge in a circular channel the depth of flow is equal to 0.938 times the diameter of the channel.

(ii) Condition for maximum mean velocity of flow through a circular channel section. According to Chezy's or Manning's formula the mean velocity of flow through a circular channel is proportional to the hydraulic radius. As such the mean velocity of flow will have a maximum value when the hydraulic radius $\left(\frac{A}{P}\right)$ is maximum. Thus differentiating $\left(\frac{A}{P}\right)$ with respect to θ and equating to zero for a maxima, we get

$$\frac{d\left(\frac{A}{P}\right)}{d\theta} = \frac{P\left(\frac{dA}{d\theta}\right) - A\left(\frac{dP}{d\theta}\right)}{P^2} = 0$$

or $$P\frac{dA}{d\theta} - A\frac{dP}{d\theta} = 0$$

Substituting the values of $A, P, \left(\frac{dA}{d\theta}\right)$ and $\left(\frac{dP}{d\theta}\right)$ the above expression becomes

$$\frac{r^3\theta}{2}(1-\cos\theta) - \frac{r^3}{2}(\theta - \sin\theta) = 0$$

or $$\theta = \tan\theta$$

The solution of the above equation is $\theta = 257.5°$ (approximately).

The depth of flow in the channel, for maximum velocity is

$$y = r + r\cos\left(180° + \frac{\theta}{2}\right)$$

$$= r\left[1 + \cos\left(180° - \frac{257.5°}{2}\right)\right]$$
$$= r(1 + \cos 51.25°)$$
$$= 1.626r \approx 0.81\,D \qquad \text{...(15.26)}$$

That is, the depth of flow for maximum mean velocity of flow in circular channel is equal to 0.81 times the diameter of the channel.

Further the hydraulic radius for a circular channel is

$$R = \frac{A}{P} = \frac{r}{2\theta}(\theta - \cos\theta)$$

Substituting the value of θ as 257.5° for maximum mean velocity

$$R = \frac{r}{2\pi\left(\frac{257.5}{180}\right)}\left(\frac{\pi \times 257.5}{180} - \sin 257.5°\right)$$
$$= 0.6086r \approx 0.30\,D$$

That is, the hydraulic radius is equal to 0.30 times the diameter, for maximum mean velocity of flow in a circular channel.

15.7 OPEN CHANNEL SECTION FOR CONSTANT VELOCITY AT ALL DEPTHS OF FLOW

According to Chezy's or Manning's formula it is seen that the hydraulic radius is the sole shape parameter on which the velocity of flow in a channel laid on a constant bottom slope depends. As such the velocity of flow will be constant at all depths if the hydraulic radius is constant for any depth of flow.

Consider a profile of channel section as shown in Fig. 15.8, assumed to have constant hydraulic radius R for any depth of flow. For a small portion of the channel section considered at a depth of y and dy in thickness, as shown in Fig. 15.8, the area dA and wetted perimeter dP may be expressed as

$$dA = 2x(dy)$$
$$dP = 2\sqrt{(dx)^2 + (dy)^2}$$

where x and $(x + dx)$ are respectively half the bottom and top widths of the small portion of the area under consideration.

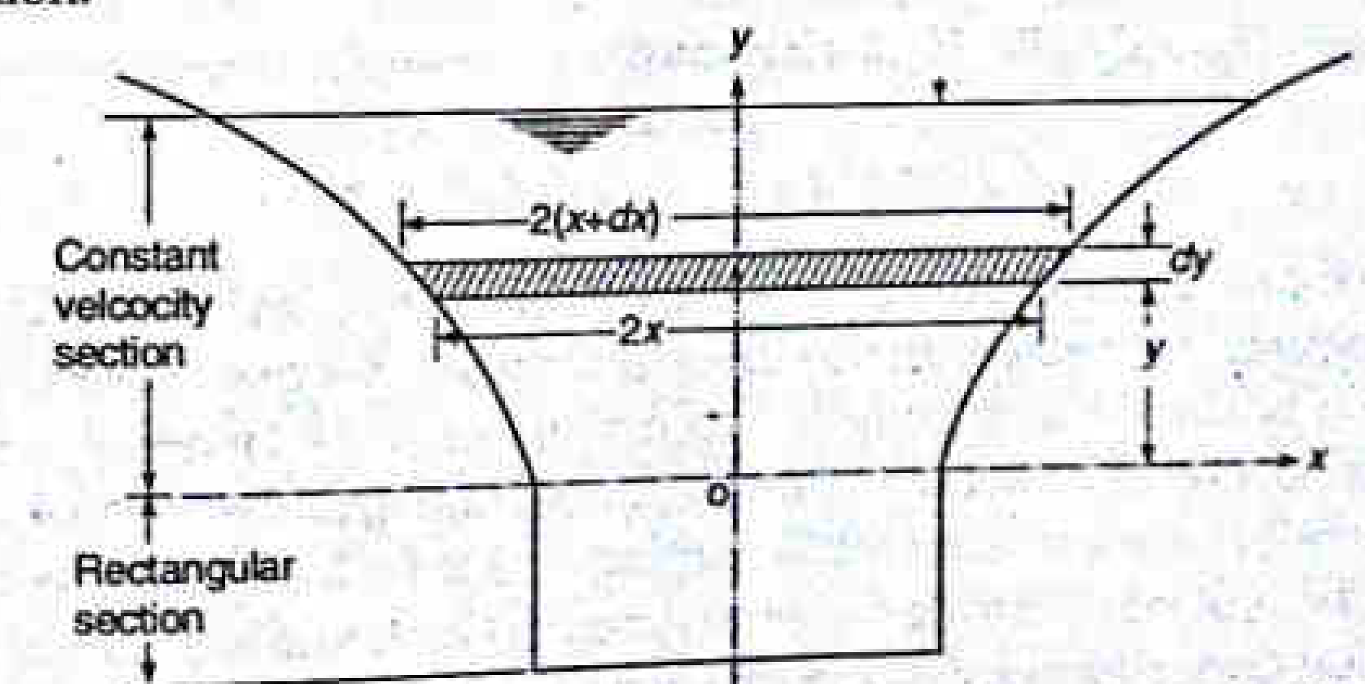


Figure 15.8 *Channel section for constant velocity at all depths of flow*

From the given condition

$$R = \frac{A}{P} = \frac{dA}{dP} = \frac{2xdy}{2\sqrt{(dx)^2+(dy)^2}}$$

or
$$R^2[(dx)^2+(dy)^2] = x^2(dy)^2$$

or
$$R^2\left[1+\left(\frac{dy}{dx}\right)^2\right] = x^2\left(\frac{dy}{dx}\right)^2$$

Solving for $\left(\frac{dy}{dx}\right)$, we get

$$\frac{dy}{dx} = \frac{R}{\sqrt{x^2-R^2}}$$

or
$$dy = \frac{Rdx}{\sqrt{x^2-R^2}}$$

Integrating the above equation, we get

$$y = R\log_e\left(x+\sqrt{x^2-R^2}\right)+C \quad ...(15.27)$$

where C is constant of integration.

Equation 15.27 defines the cross-section of a channel with constant velocity at all depths of flow. The two sides of this channel section are formed by two curves, mathematically known as catenaries. Since the theoretical section of the channel so developed is bottomless, an artificial bottom should be provided. As such below the point $y = 0$, the channel section may be either of the various types, viz., rectangular, trapezoidal, triangular, semi-circular, parabolic etc. Depending on the shape of the channel section to be provided for the bottom portion or on account of limitations imposed by certain conditions, the constant of integration C may be evaluated. For instance if the condition is such that at $y = 0, x = R$, the constant C becomes

$$C = -R\log_e(R)$$

Substituting this value of C in Eq. 15.27 it becomes

$$y = R\left[\log_e(x+\sqrt{x^2-R^2}-\log_e R\right]$$

In practice it is observed that for an ordinary channel section with an increase in the depth of flow, the velocity is considerably increased, thereby causing damage to the channel section. On the contrary if the depth of flow is small, the velocity is considerably reduced. Hence in order to avoid such fluctuations of the velocity of flow and to prevent the damaging of the channel section it becomes necessary, in practice, to have a channel section having constant velocity at all depths. Particularly in the case of large sewers, in which the discharge ranges from a certain minimum value that flows daily to a very large value during the rainy season, channel sections of constant velocity are designed.

15.8 COMPUTATION OF UNIFORM FLOW

The discharge of uniform flow in a channel may be expressed by using Chezy's formula as

$$Q = AV = CA\sqrt{RS} = K\sqrt{S} \quad ...(15.28)$$

where
$$K = CA\sqrt{R} \quad ...(15.29)$$

The term K is known as 'conveyance' of the channel section which is a measure of the carrying capacity of the channel section, since it is directly proportional to the discharge Q. Similarly when the Manning's formula is used, the discharge becomes

$$Q = AV = \frac{1}{n}AR^{2/3}S^{1/2} = K\sqrt{S} \quad ...(15.30)$$

where
$$K = \frac{1}{n}AR^{2/3} \quad ...(15.31)$$

From Eq. 15.28 or 15.30 the conveyance is

$$K = \frac{Q}{\sqrt{S}} \quad ...(15.32)$$

Equation 15.32 can be used to compute the conveyance when the discharge and bottom slope of the channel are given. On the other hand Eqs. 15.29 and 15.31 are used to compute the conveyance when the geometry of the wetted area of the channel section and the Chezy's C or Manning's n are given.

Since Manning's formula is used extensively, the expression $(AR^{2/3})$ is called the section factor for uniform flow computation, which is an important element in the computation of uniform flow as indicated below. From Eq. 15.31 this factor may be expressed as

$$(AR^{2/3}) = nK \quad ...(15.33)$$

and from Eq. 15.32

$$(AR^{2/3}) = \frac{nQ}{\sqrt{S}} \quad ...(15.34)$$

Equation 15.34 applies to a channel section when the flow is uniform. The right hand side of this equation contains the terms n, Q and S, but the left hand side contains the terms which depend only on the geometry of the wetted area. Therefore it indicates that for given values of n, Q and S, there is only one possible depth of flow at which the uniform flow will be maintained in any channel section, provided the value is $(AR^{2/3})$ always increases with increase in depth, which is true in most of the cases. This depth of flow is known as *normal depth* which is represented by y_n. Similarly if n, S and the depth of flow are given for a channel section, it can be seen from Eq. 15.34 that there can be only one discharge for maintaining a uniform flow through the channel section. This discharge is known as *normal discharge*, which is represented by Q_n. Furthermore when Q, n and depth of flow are given for a channel section, Eq. 15.34 can be used to determine the bottom slope which will maintain the uniform flow at a given depth of flow y in the channel. The bottom slope thus determined is known as *normal slope* which is represented by S_n. Equation 15.34 is quite useful for computation and analysis of uniform flow.

15.9 SPECIFIC ENERGY AND CRITICAL DEPTH

Since the free surface in the case of a channel flow represents the hydraulic gradient, the Bernoulli's equation applied between the sections (1) and (2) L distance apart may be written as

$$\frac{V_1^2}{2g} + y_1 + Z_1 = \frac{V_2^2}{2g} + y_2 + Z_2 + h_f$$

where h_f is the energy loss between the two sections. The term $\left(\frac{V^2}{2g} + y + Z\right)$ represents the total energy per unit weight of water at any section. However, in the solution of many problems of channel flow the concept of *specific energy* is quite useful, which was first introduced by Bakhmeteff in 1912. The specific energy of flow at any channel section is defined as the energy per unit weight of water measured with respect to the channel bottom as the datum. Thus specific energy E at any section is the sum of the depth of flow at that section and the velocity head, that is

$$E = y + \frac{V^2}{2g} = y + \frac{Q^2}{2gA^2} \quad \ldots(15.35)$$

Since $$V = \left(\frac{Q}{A}\right)$$

It can be seen from Eq. 15.35 that, for a given channel section and discharge Q, the specific energy is a function of the depth of flow only. Thus for a given channel section and discharge, Eq. 15.35 may be represented graphically in which specific energy is plotted against the depth of flow. The curve so obtained is known as specific energy curve and is shown in Fig. 15.9. This curve has two limbs AC and BC. The lower limb AC approaches the horizontal axis (or the specific energy axis) asymptotically towards the right. The upper limb BC approaches asymptotically to the line OB which passes through the origin and has an angle of inclination equal to 45°. At any point P on this curve the ordinate represents the depth of flow and the abscissa represents the specific energy, which is equal to the sum of the depth of flow and the velocity head.

It can be seen from the specific energy curve that, there is one point C on the curve which has a minimum specific energy, thereby indicating that below this value of the specific energy the given discharge cannot occur. The depth of flow at which the specific energy is minimum is called *critical depth* y_c. Similarly the velocity of flow at the critical depth is known as *critical velocity* V_c. For any other value of the specific energy there are two possible depths, one greater than the critical depth and the other smaller than the critical depth, at which a given discharge can occur with the same specific energy. These two depths for given specific energy are called the *alternate depths*–y_1 smaller than y_c and y_2 greater than y_c as shown in Fig. 15.9. Apparently at point C the two alternate depths become one, equal to critical depth y_c. For any depth of flow greater than the critical depth, the specific energy increases with the increase in the depth. Moreover when the depth of flow is greater than the critical depth, the velocity of flow is less than the critical velocity for the given discharge. Hence the flow at the depths greater than the critical depth is known as *subcritical flow* or *tranquil flow*. On the other hand when the depth of flow is less than the critical depth, the specific energy increases as the depth of flow decreases. Furthermore for given discharge when the depth of flow is less than critical depth, the velocity of flow is greater than the critical velocity. Hence the flow at the depths less than the critical depth is known as *supercritical flow* or *rapid flow*. As shown in Fig. 15.9 y_1 is the depth of a supercritical flow, and y_2 is the depth of a subcritical flow.

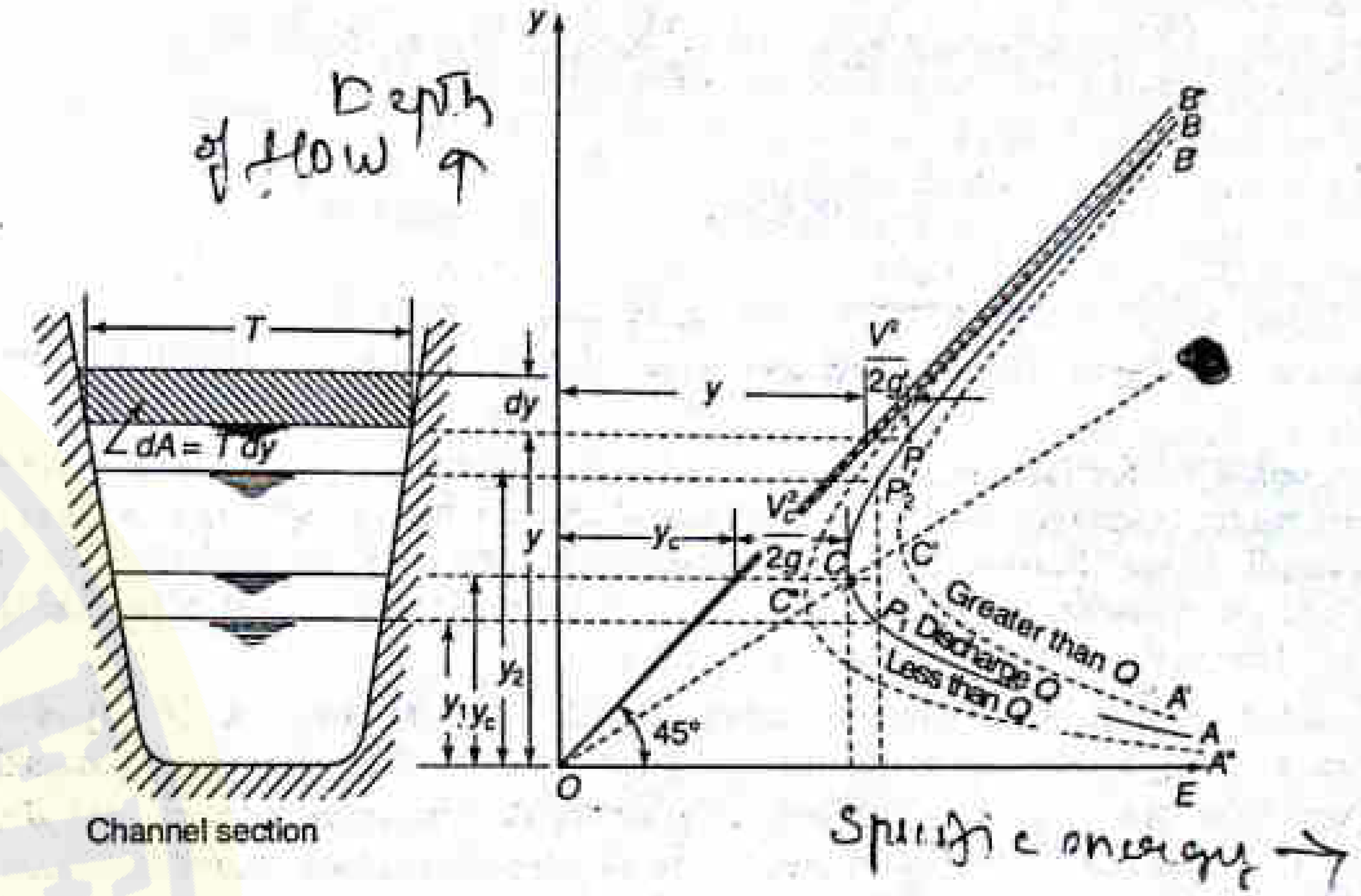


Figure 15.9 *Specific energy curve*

It is evident from Eq. 15.35 that for any channel section if the discharge changes, the specific energy will be changed accordingly. The two curves $A''C''B''$ and $A'C'B'$ shown in Fig. 15.9 represent the positions of the specific energy curve when the discharge is respectively less and greater than the discharge used for the construction of the curve ACB.

For a given discharge the condition for minimum specific energy can be obtained by differentiating Eq. 15.35 with respect to y and then considering $\left(\frac{dE}{dy}\right) = 0$. Therefore

$$\frac{dE}{dy} = \frac{d}{dy}\left(y + \frac{Q^2}{2gA^2}\right) = \left[1 - \frac{2Q^2}{2gA^3}\left(\frac{dA}{dy}\right)\right] = 0$$

Since Q is a constant and A is a function of y. As shown in Fig. 15.9, in a channel section if T is the top width of flow then differential water area dA near the free surface is equal to $T dy$ i.e., $dA = T dy$ or $\left(\frac{dA}{dy}\right) = T$. By substituting this value of $\left(\frac{dA}{dy}\right)$ the above equation becomes

$$1 - \frac{Q^2 T}{gA^3} = 0$$

or

$$\frac{Q^2}{g} = \frac{A^3}{T} \qquad \text{...(15.36)}$$

Since $V = \left(\frac{Q}{A}\right)$ and hydraulic depth $D = \left(\frac{A}{T}\right)$, Eq. 15.36 may be written as

$$\frac{V^2}{g} = D; \text{or } V^2 = gD \qquad \text{...(15.37)}$$

Equation 15.37 may also be written as $\left(\frac{V}{\sqrt{gD}}\right) = 1$. Since $\left(\frac{V}{\sqrt{gD}}\right)$ represents for Froude number it may be stated that for the critical flow the Froude number, $Fr = 1$. Thus the critical state of flow may be defined in a more common way as 'it is that state of flow at which the specific energy is a minimum for the given discharge'. Further from the foregoing discussion it is obvious that in any channel section carrying a given discharge Q, if the flow is in the critical state, the cross-sectional area A and the top width T at the water surface must satisfy Eq. 15.36.

Moreover, as stated earlier when the flow is in subcritical state, since the velocity of flow is less than the critical velocity and the depth of flow is more than the critical depth, it then follows that the Froude number is less than one (i.e., $Fr < 1$). Similarly when the flow is in supercritical state, since the velocity of flow is more than the critical velocity and the depth of flow is less than the critical depth, it then follows that the Froude number is more than one (i.e., $Fr > 1$).

Solving Eq. 15.35 for Q, we get

$$Q = \sqrt{2A^2 g(E-y)} \qquad \text{...(15.38)}$$

In a given channel section for a given value of specific energy E, the condition for maximum discharge is obtained by putting $\left(\frac{dQ}{dy}\right) = 0$. Thus differentiating Eq. 15.38 with respect to y and equating it to zero, we get

$$\frac{dQ}{dy} = \sqrt{2g}\left[\frac{dA}{dy}\sqrt{Ey}\,\frac{A}{2\sqrt{Ey}}\right] = 0$$

or

$$E = \frac{1}{2}\frac{A}{T} + y\,; \text{since } \frac{dA}{dy} = T$$

Substituting the value of E obtained above in Eq. 15.38, it may be simplified and rearranged as

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

which is the criterion for the critical state of flow as represented by Eq. 15.36. It is thus observed that for a given specific energy the discharge in a given channel section is maximum when the flow is in the critical state.

15.10 MOMENTUM IN OPEN-CHANNEL FLOW-SPECIFIC FORCE

If V is the mean velocity of flow of discharge Q in a channel, the momentum of the flow passing a channel section per unit time may be expressed by (wQV/g), where w is the specific weight of water. According to Newton's second law of motion the rate of change of momentum in the body of water flowing in a channel is equal to the resultant of all the forces that are acting on the body. Thus applying this principle to a channel of large bed slope, the following expression for the rate of change of momentum in the body of water enclosed between sections 1 and 2 may be written

$$\frac{wQ}{g}(V_2 - V_1) = P_1 - P_2 + W\sin\theta - F_f \qquad \text{...(15.39)}$$

where w, Q and V are previously defined with subscripts referring to sections 1 and 2; P_1 and P_2 are the resultant pressures acting on the two sections; W is the weight of water enclosed between the sections; θ is the angle of inclination of the channel bottom with the horizontal; and F_f is the total external force of frictional resistance acting in the direction opposite to the flow along the surface of contact between the water and the channel.

When the momentum equation is applied to a short horizontal reach of a prismatic channel, the external force of friction and the component of the weight of water can be ignored. Thus with $\theta = 0$ and $F_f = 0$ Eq. 15.39 becomes

$$\frac{wQ}{g}(V_2 - V_1) = P_1 - P_2 \qquad \text{...(15.40)}$$

Assuming hydrostatic pressure distribution at sections 1 and 2

$$P_1 = wA_1\bar{z}_1; \text{and } P_2 = wA_2\bar{z}_2$$

where A_1, $\bar{z}_1$ and A_2, $\bar{z}_2$ are cross-sectional area and the vertical depth of the centroid of the area below the surface of flow at sections 1 and 2 respectively. Also $V_1 = (Q/A_1)$ and $V_2 = (Q/A_2)$, thus Eq. 15.40 may be written as

$$\frac{Q^2}{gA_1} + A_1\bar{z}_1 = \frac{Q^2}{gA_2} + A_2\bar{z}_2 \qquad \text{...(15.41)}$$

The terms on both sides of Eq. 15.41 are analogous, hence Eq. 15.41 may be expressed for any channel section by a general function

$$F = \frac{Q^2}{gA} + A\bar{z} \qquad \text{...(15.42)}$$

This function consists of two terms. The first term is the momentum of the flow passing through the channel per unit time per unit weight of water, and the second term is the force per unit weight of water. Since both these terms are force per unit weight of water, their sum is known as *specific force*. As such Eq. 15.41 may be expressed as $F_1 = F_2$, which means that the specific forces of sections 1 and 2 are equal, provided that external force of friction and the component of the weight of water in the channel reach between the two sections can be ignored.

It is evident from Eq. 15.42 that for a given channel section and discharge Q the specific force depends only on the depth of flow. Thus by plotting the depth of flow against the specific force for a given channel section and discharge, a specific force curve is obtained as shown in Fig. 15.10. Alike specific energy curve this curve also has two limbs AC and BC. The limb AC approaches the horizontal axis asymptotically towards right. The limb BC rises upward and extends indefinitely to the right. It can be seen from the curve that for a given value of specific force there are two possible depths y_1 and y_2. It has been shown in the next chapter that these two depths y_1 and y_2 constitute the initial and sequent depths of a hydraulic jump. At point C on the curve the two depths become one and corresponding to this depth the specific force is minimum.

For a given discharge the condition for minimum specific force can be obtained by differentiating Eq. 15.42 with respect to y and then considering $(dF/dy) = 0$. Thus

$$\frac{dF}{dy} = -\left(\frac{Q^2}{gA^2}\right)\frac{dA}{dy} + \frac{d(A\bar{z})}{dy} = 0$$

since Q is constant and both A and $\bar{z}$ are the functions of y. As shown in Fig.15.10 for a change dy in the depth, the corresponding change $d(A\bar{z})$ in the moment of the cross-sectional area about the free surface may be expressed as

$$\left[A(\bar{z}+dy)+\frac{T(dy)^2}{2}\right]-(A\bar{z}) = d(A\bar{z})$$

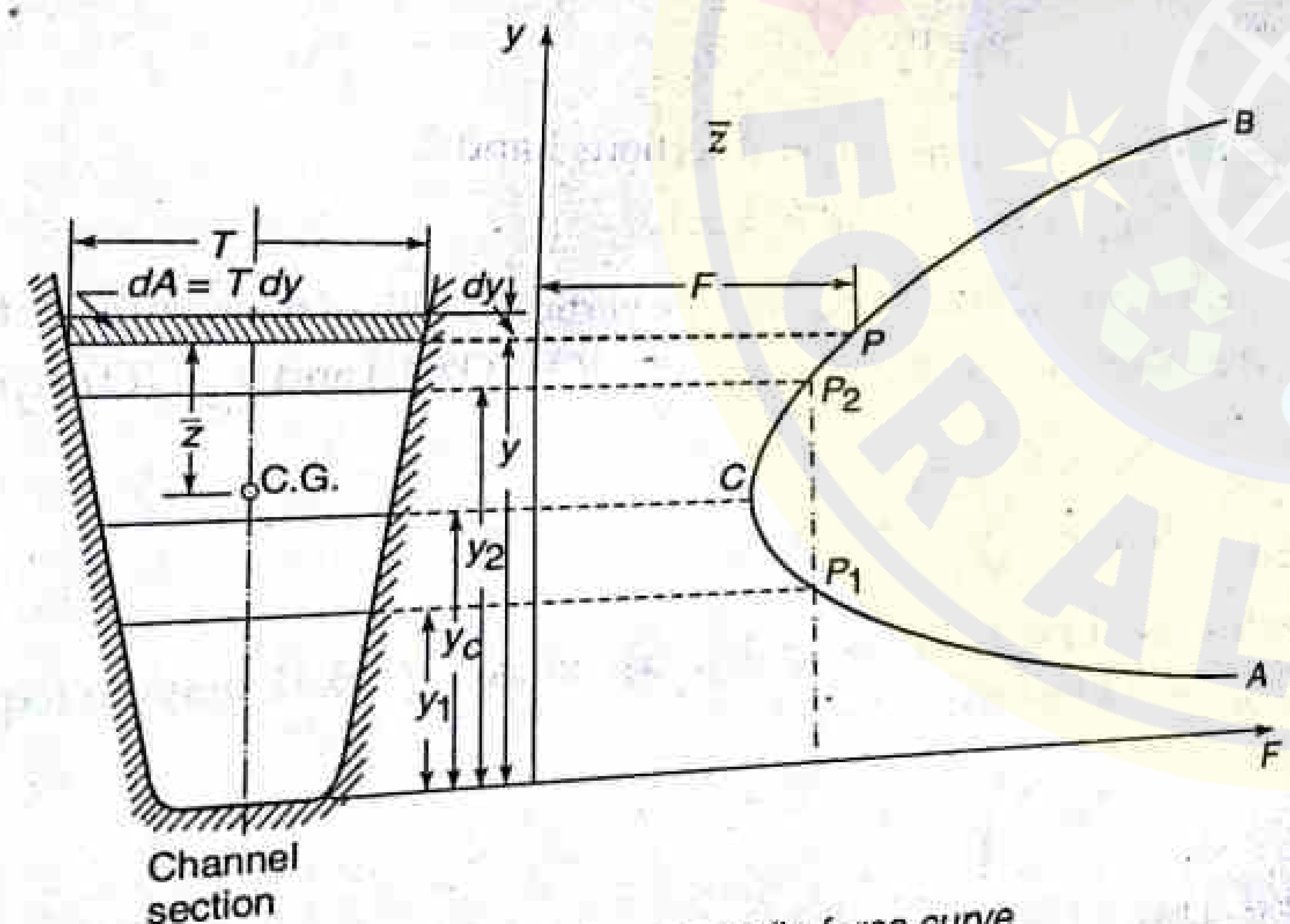


Figure 15.10 *Specific-force curve*

Assuming $(dy)^2 \approx 0$, the above expression becomes

$$d(A\ \ ) = A(dy)$$

Substituting this value of $d(A\bar{z})$ the preceding equation becomes

$$\frac{dF}{dy} = -\left(\frac{Q^2}{gA^2}\right)\frac{dA}{dy} + \frac{A(dy)}{dy} = 0$$

Again since $(dA/dy) = T$, the above equation reduces to

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

which is the condition for the critical state of flow as indicated earlier. It is thus seen that the depth of flow at the minimum value of the specific force is equal to the critical depth.

Further solving Eq. 15.42 for Q, it may be expressed as

$$Q = \sqrt{(F - A\bar{z})gA} \qquad \text{...(15.43)}$$

In a given channel section for a given value of specific force F, the condition for maximum discharge can be obtained by putting $(dQ/dy) = 0$. Thus differentiating Eq. 15.43 with respect to y

$$\frac{dQ}{dy} = \frac{g\left[F\left(\frac{dA}{dy}\right) - A\frac{d(A\bar{z})}{dy} - (A\bar{z})\frac{dA}{dy}\right]}{2\sqrt{(F-A\bar{z})(gA)}} = 0$$

or

$$F = \frac{A^2}{T} + A\bar{z}$$

Substituting the value of F obtained above in Eq. 15.43 it may be simplified and rearranged as

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

which is again the criterion for the critical state of flow as indicated earlier. Therefore it may be stated that for a given specific force the discharge in a given channel section is maximum when the flow is in the critical state.

15.11 CRITICAL FLOW AND ITS COMPUTATION

When the depth of flow of water over a certain reach of a given channel is equal to the critical depth y_c, the flow is described as critical flow or in critical state. As indicated by the critical flow criterion, Eq. 15.36, the critical depth for a given discharge Q is the depth y_c corresponding to which the cross-sectional area A and top width T of the channel section are such that the value of (A^3/T) is given by the following expression,

$$\left(\frac{A^3}{T}\right)_c = \frac{Q^2}{g}$$

or

$$\left(A\sqrt{\frac{A}{T}}\right)_c = Z_c = \frac{Q}{\sqrt{g}} \qquad \text{...(15.44)}$$

Equation 15.44 indicates that the section factor for critical flow computation, $Z[= A\sqrt{A/T}]$ for a channel section at the critical state of flow is equal to (Q/g). For a prismatic channel the section factor Z is a function of the depth of flow. Hence it is evident from Eq. 15.44 that in a prismatic channel there is only one possible depth y_c which makes the given discharge Q, in a channel to flow in critical state.

Since the depth y_c will be same at all sections, critical flow in a prismatic channel is essentially a uniform flow. Similarly it may also be stated that, when the depth of flow in a channel section is fixed, there can be only one discharge, which maintains the critical flow at the chosen depth. This discharge is known as *critical discharge* and is represented by Q_c.

Furthermore, for a given channel section the value of the bottom slope may be so adjusted that for a given discharge it is possible to develop uniform flow at the depth of flow equal to the corresponding critical depth (or in other words it is possible to make the given discharge to flow in a critical state). This value of the bottom slope is known as *critical slope* and it is represented by S_c. On the other hand by adjusting the bottom slope and the discharge, a critical flow may be obtained at the given normal depth. The slope thus obtained is known as the *critical slope at the given normal depth or normal critical slope* which is represented by S_{cn}.

As indicated in Sections 15.9 and 15.10 the various characteristics of the critical state of flow through a channel section are as follows:

(i) The specific energy is a minimum for a given discharge.
(ii) The discharge is a maximum for a given specific energy.
(iii) The specific force is a minimum for a given discharge.
(iv) The discharge is a maximum for a given specific force.
(v) The velocity head is equal to half the hydraulic depth in a channel of small slope.
(vi) The Froude number is equal to unity.

For the computation and analysis of critical flow in an open channel, Eq. 15.44 is very useful. When the discharge is given critical section factor Z_c can be computed from Eq. 15.44. For a given channel section, since section factor Z is a function of the depth of flow y, that is $Z = f(y)$, a curve between Z and y can be prepared. With the help of this curve, corresponding to the known value of $Z = Z_c$ (determined from Eq. 15.44) the value of the depth of flow can be obtained which will be critical depth y_c. On the other hand when the depth of flow is given the corresponding value of the section factor Z can be obtained from the same curve and from this value of Z the critical discharge Q_c can be computed by Eq. 15.45, which has been obtained from Eq. 15.44 in the following form

$$Q_c = \left(\sqrt{\frac{A^3}{T}}\right) \times \left(\sqrt{g}\right) = Z\sqrt{g} \qquad ...(15.45)$$

For the channel sections of different shapes the computation of critical flow may be carried out by adopting the procedure as indicated below.

(a) Critical Flow in Rectangular Channels. Since for a channel of rectangular section the top width T is equal to the bottom width B, Eq. 15.36 can be further simplified. If it is assumed that q represents the discharge per unit width of the channel section then the total discharge Q passing through a channel of rectangular section of bottom width B may be expressed as $Q = (B \times q)$ or $q = \frac{Q}{B}$. Corresponding to a critical depth of flow y_c the area of rectangular channel section $A = (B \times y_c)$. By substituting these values in Eq. 15.44 it becomes

$$\frac{qB}{\sqrt{g}} = By_c\sqrt{\frac{By_c}{B}}$$

$$\therefore \qquad \frac{q^2}{g} = (y_c)^3$$

or

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} \qquad ...(15.46)$$

Equation 15.46 indicates that for a rectangular channel section the critical depth y_c can be determined if the discharge per unit width is known.

Further for a rectangular channel the specific energy Eq. 15.35 may be written as

$$E = y + \frac{(qB)^2}{2g(By)^2} = y + \frac{q^2}{2gy^2} \qquad ...(15.47)$$

Solving Eq. 15.47 for q it becomes

$$q = \sqrt{2g(E-y)y^2} \qquad ...(15.48)$$

For a given specific energy E, the condition for the discharge per unit width q to be maximum is obtained by putting $\left(\frac{dq}{dy}\right) = 0$. Thus differentiating Eq. 15.48 with respect to y and equating to zero, we get

$$\frac{dq}{dy} = \frac{\sqrt{2g}(2yE-3y^2)}{2\sqrt{(E-y)y^2}} = 0$$

or

$$E = \frac{3}{2}y \qquad ...(15.49)$$

Substituting the value of E from Eq. 15.49 in Eq. 15.48 it becomes

$$q = \sqrt{gy^3}$$

$$y^3 = \frac{q^2}{g} = y_c^3$$

It is thus seen that for a given specific energy E, the discharge per unit width q is maximum when the depth of flow y is equal to critical depth y_c. Equation 15.49 may now be expressed using y_c instead of y in the following two forms:

$$E = \frac{3}{2}y_c$$

or

$$y_c = \frac{2}{3}E \qquad ...(15.50)$$

Using Eq. 15.48 a plot between q and y can be prepared for a given value of specific energy E. This is shown in Fig. 15.11 and is known as discharge diagram. As in the case the specific energy curve

Fig. 15.9, for a given value of discharge, it can be seen that in the discharge diagram (Fig. 15.11) also there are two possible depths for a certain value of q, as long as q is less than a certain maximum value. Beyond this maximum value of discharge the flow is physically impossible at the given specific energy. Further corresponding to the maximum discharge the two depths become one, equal to the critical depth y_c.

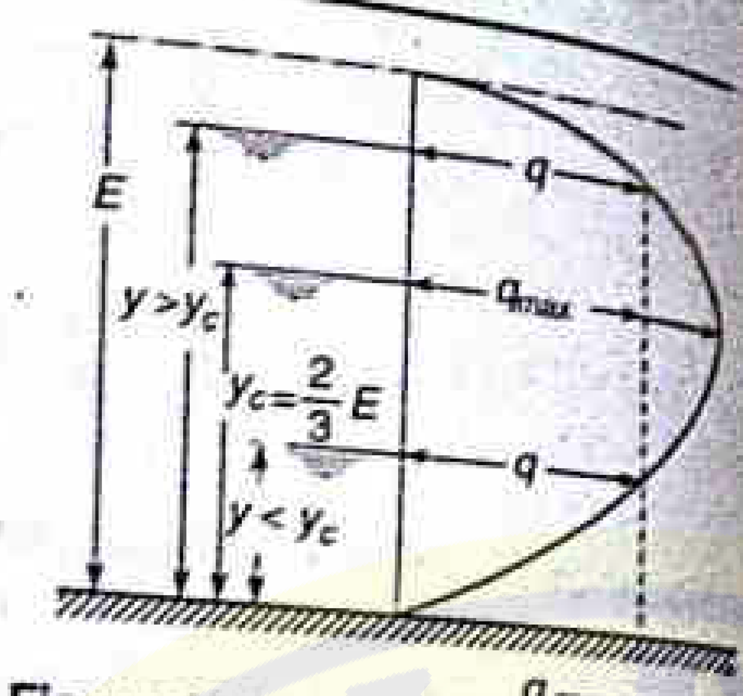


Figure 15.11 *Discharge diagram*

For a discharge q the depth of flow less than y_c represents the supercritical state of flow, while that greater than y_c corresponds to the subcritical state of flow.

(b) Critical Flow in Channal Sections of other Shapes. Equation 15.36 represents a general condition for the critical flow in channel sections of any shape. As such using Eq. 15.36 the following expressions for the critical depth y_c may be obtained for the channel sections of different shapes.

(i) Triangular channel section. For a channel of triangular section with side slope z horizontal to 1 vertical the critical depth y_c is given by the following expressions

$$y_c = \left(\frac{2Q^2}{gz^2}\right)^{1/5} \quad \text{...(15.51)}$$

and

$$y_c = \frac{4}{5}E \quad \text{...(15.52)}$$

(ii) Parabolic channel section. For a channel of parabolic section if the top width at the water surface is expressed as $T = ky^{1/2}$, the critical depth y_c is given by the following expressions

$$y_c = \left(\frac{27Q^2}{8gk^2}\right)^{1/4} \quad \text{...(15.53)}$$

and

$$y_c = \frac{3}{4}E \quad \text{...(15.54)}$$

(iii) Trapezoidal channel section. For a channel of trapezoidal section no explicit expressions for y_c can be obtained, but the following expressions in terms of dimensionless parameters may be developed which can be used for the computation of the critical depth y_c. Thus for a channel section of bottom width B and side slope z horizontal to 1 vertical, we have

$$\left(\frac{Q^2z^3}{gB^5}\right) = \frac{\left[\frac{B}{zy_c}+1\right]^3}{\left(\frac{B}{zy_c}\right)^3\left(\frac{B}{zy_c}+2\right)} \quad \text{...(15.55)}$$

and

$$\frac{Ez}{B} = \frac{\left[3\frac{B}{zy_c}+5\right]}{2\frac{B}{zy_c}\left(\frac{B}{zy_c}+2\right)} \quad \text{...(15.56)}$$

15.12 APPLICATION OF SPECIFIC ENERGY AND DISCHARGE DIAGRAMS TO CHANNEL TRANSITIONS

In the case of long channels often it becomes necessary to provide transitions. A transition is the portion of a channel with varying cross-section, which connects one uniform channel to another, which may or may not have the same cross-sectional form. The variation of a channel section may be caused either by reducing or increasing the width or by raising or lowering the bottom of the channel. Various channel transitions may be broadly classified as sudden transitions and gradual transitions. Sudden transitions are those in which the change of cross-sectional dimensions occur in a relatively short length. On the other hand in the case of a gradual transition the change of cross-sectional area takes place gradually in a relatively long length of the channel. Some of the functions which channel transitions are made to serve are metering of flow, dissipation of energy, reduction or increase of velocities, change in channel section or alignment with a minimum of energy dissipation etc.

Some of the devices commonly used for measuring discharge through channels are venturi flume, standing wave flume and Parshall flume which are provided with transition as described later. The dissipation of energy may be caused by providing a sudden drop in the channel bottom. In the case of irrigation channels, in order to prevent scouring, velocities are required to be reduced, while for navigation channels in order to prevent shoaling higher velocities are required to be developed. Such reduction or increase of velocities may be made possible by expansion or contraction of channel section. The change in channel section with a minimum of energy dissipation is obtained by providing a gradual expansion or a gradual contraction with well rounded corners. Apart from the above mentioned transitions, often channel cross-section may be reduced if a bridge is constructed across it and bridge piers are constructed. Some of the channel transitions described below are shown in Fig. 15.12.

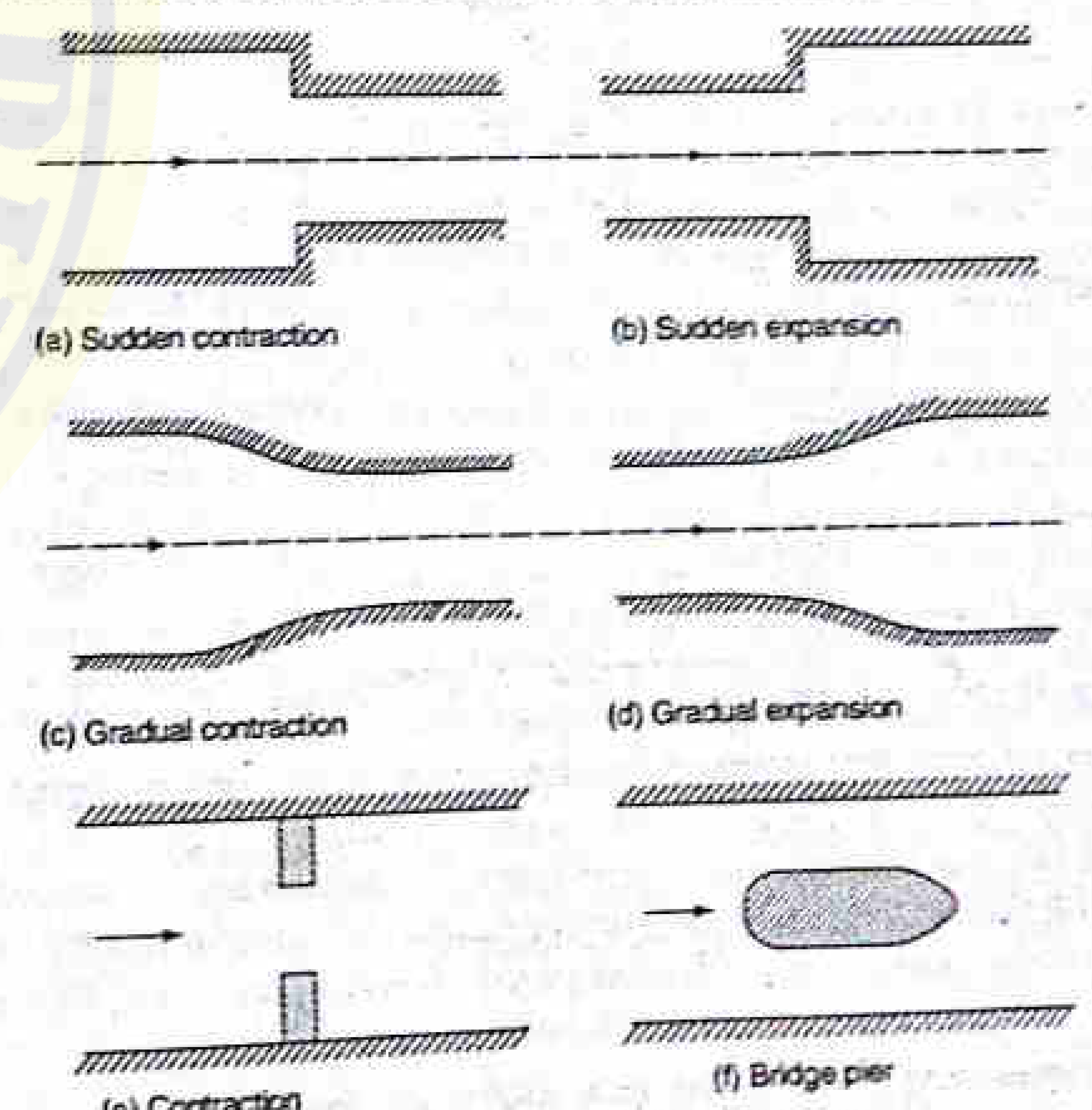


Figure 15.12 *Open channel transitions*

For a rectangular channel, transitions of decreasing channel cross-section may be analysed with the help of specific energy and discharge diagrams. The decrease in cross-section or a channel contraction may be obtained either by a decrease in width or by a decrease in depth or by a combination of both. In the following paragraphs transitions for rectangular channels with reduced width and raised bottom have been discussed separately. The combination can, however, be analysed by using the same principles in combination.

(a) Transition with Reduction in Width of a Rectangular Channel. Consider a channel of rectangular cross-section which has a reduced width as shown in Fig. 15.3. Since the flow is converging between sections 1–1 and 2–2, it may appropriately be assumed that no loss of energy takes place between the two sections. Further if it is assumed that the bottom of the channel is horizontal, the specific energy will be equal at both the sections. Let Q be the total discharge flowing through the channel. Since the total discharge Q must remain the same from section 1–1 to section 2–2, the discharge per unit width q will change from a smaller value at section 1–1 to a greater value at section 2–2.

Thus if B_1 and B_2 are the widths of the channel at sections 1–1 and 2–2 respectively then the discharge per unit width at section 1–1 is $q_1 = \left(\frac{Q}{B_1}\right)$ and at section 2–2 is $q_2 = \left(\frac{Q}{B_2}\right)$. Since B_1 is greater than B_2, q_1 is less than q_2. The contracted section 2–2 is often termed as 'throat section' or 'throat'. If the width of the throat is further reduced to B_3, as shown in Fig. 15.13, for the same discharge Q flowing

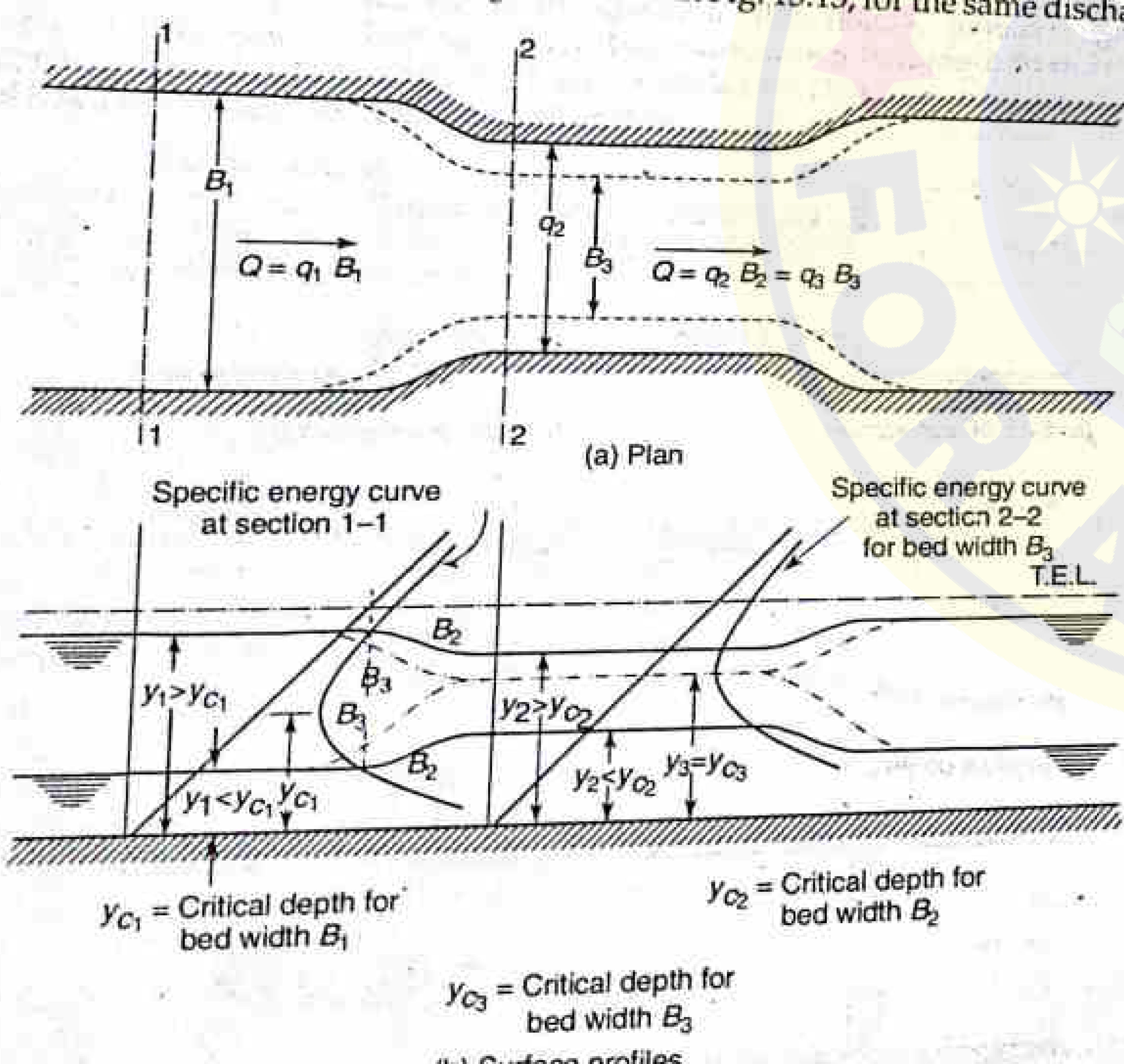


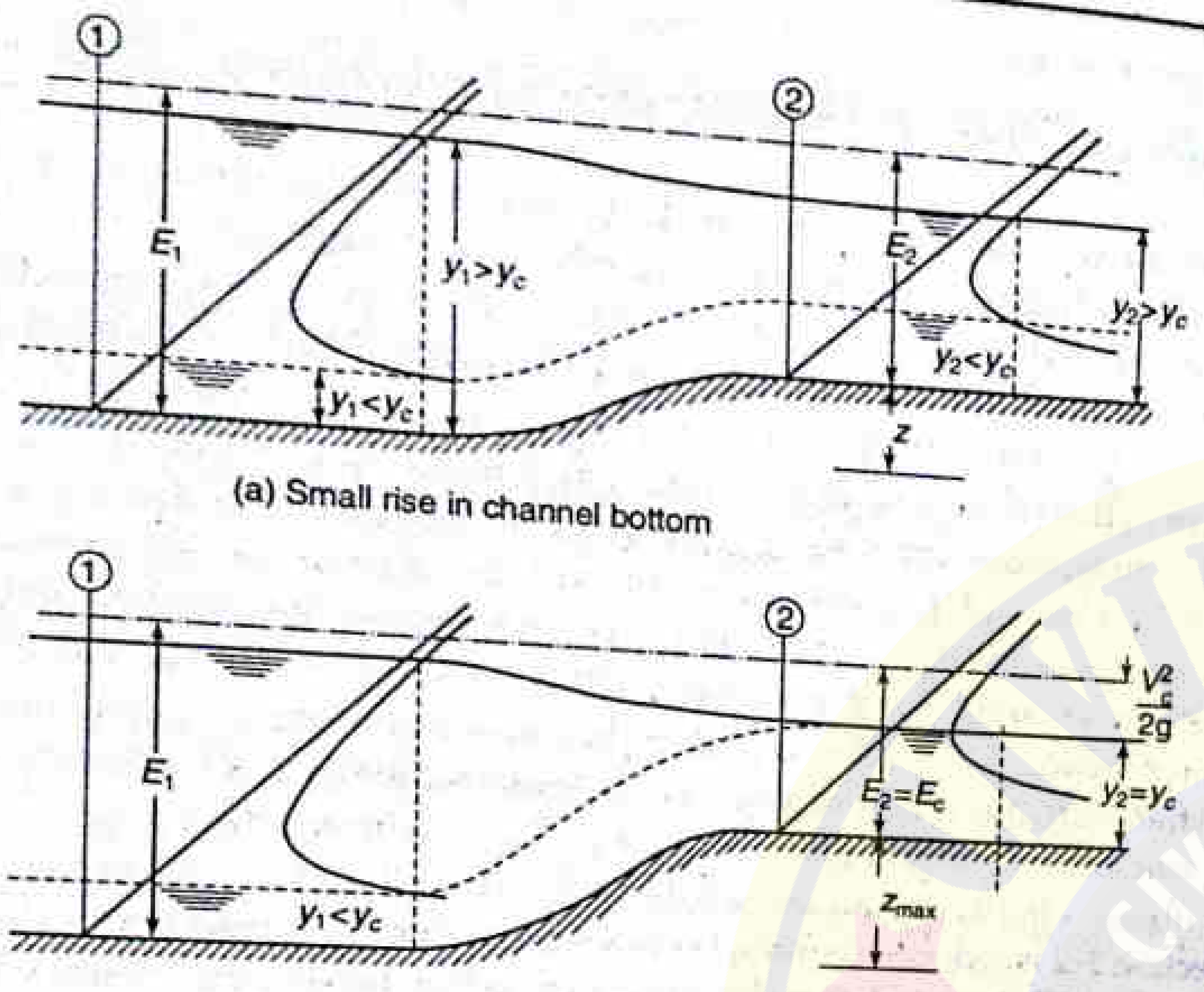
Figure 15.13 *Flow in rectangular channel with reduction in width*

through the channel, the discharge per unit width will be increased to q_3 which is equal to $\left(\frac{Q}{B_3}\right)$. It can be seen from the discharge diagram, Fig. 15.11, that in the subcritical region of flow for a given specific energy as the discharge per unit width q is increased, the depth of flow y will decrease. This decrease in the depth of flow continues upto a certain minimum value of y equal to y_c corresponding to which the discharge per unit width is maximum equal to q_{max}. On the other hand, in the supercritical region of flow for a given specific energy, as q increases the depth of flow y also increases. Again this increase in depth of flow continues upto a certain maximum value of y equal to y_c corresponding to which q becomes maximum equal to q_{max}. In either of the two above discussed flow conditions a further increase in q beyond q_{max} is not possible at that value of specific energy. However, if q is increased further by reducing the width of the channel section the specific energy will not remain same, but it will be increased as explained below. When the flow in the channel is in subcritical state and if the width of section 2–2 is reduced so that q becomes more than q_{max} then there will be rise in the free surface on the upstream side of section 2–2, thereby resulting in an increase in the specific energy E by such an amount that at this new value of E, the maximum discharge can pass through the contracted section. On the other hand when the flow in the channel is in supercritical state and if the width of the section 2–2 is reduced so that q becomes more than q_{max}, then free surface on the upstream side of section 2–2 will be lowered, thereby again increasing the specific energy E by such an amount that at this new value of E, the maximum discharge can pass through the contracted section.

From the foregoing discussion it is thus observed that for a given discharge flowing in a rectangular channel, there is a certain limiting value of the contracted width (say B_3) which when provided for the throat, then there will be critical depth of flow y_c (corresponding to the given discharge and the contracted section of the channel) developed at this section. If for the same total discharge flowing through the channel, the contraction at the throat is less than the above indicated value, then for subcritical flow entering the throat the depth of flow in the throat is greater than critical depth, and for a supercritical flow entering the throat, the depth of flow in the throat is less than critical depth. On the contrary for the same total discharge if the contraction at the throat is more than the above noted limiting value, then for a subcritical flow entering the throat there is heading up of water on the upstream side, and for a supercritical flow entering the throat the water surface on the upstream side is lowered; and in both these cases the depth of flow at the contracted section will be equal to the critical depth corresponding to the given discharge in the contracted section of the channel.

(b) Transition with Raised Bottom in a Rectangular Channel. Figure 15.14 shows channel of rectangular cross-section having uniform width and provided with a rise in the bottom of the channel, which is frequently known as hump. In this case since the width of the channel is not changing, the discharge per unit width will be same at different sections for a total discharge Q flowing through the channel. Since specific energy is measured with respect to channel bottom as the datum, a rise in the bottom of the channel causes a decrease in specific energy. As such if it is assumed that there is no loss of energy between the sections 1 and 2, (Fig. 15.14) the specific energy E_2 at the hump will be less than the specific energy E_1 at section 1 by an amount equal to the height of the hump ΔZ, that is

$$E_2 = E_1 - \Delta Z$$



(a) Small rise in channel bottom

(b) Critical rise in channel bottom

Figure 15.14 *Flow in a rectangular channel with a hump in the bottom*

Now if the flow approaching the hump is in subcritical state, the free surface drops down at the hump. This drop in the free surface at the hump can be explained with the help of specific energy diagram shown in Fig. 15.14. It can be seen from this diagram that as long as the flow is in subcritical state a decrease in specific energy is associated with a decrease in the depth of flow and increase in the velocity. Thus as shown in Fig. 15.14 (a) over the hump the depth of flow decreases and the velocity head increase. However, if the flow approaching the hump is in supercritical state, the depth of flow at the hump is increased. This increase in the depth of flow at the hump can also be explained with the help of specific energy diagram, from which it is indicated that as long as the flow is in supercritical state a decrease in specific energy is associated with an increase in the depth of flow and decrease in the velocity. It may be observed from the specific energy diagram that there is a limit upto which the specific energy for a given discharge can be reduced by increasing the height of the hump to a certain value ΔZ. Thus for given condition of flow at section 1 there is limiting or maximum value of ΔZ, as shown in Fig. 15.14 (b), at which the specific energy at section 2 is equal to the minimum specific energy for the given discharge. In other words, the height of hump can be increased to a certain maximum value without altering the condition of the approaching flow upstream of the hump. This limiting value of the height of the hump ΔZ_{max} may be computed from the following expression,

$$E_1 = E_c + \Delta Z_{max} \quad \text{...(15.57)}$$

in which E_1 is the specific energy at section 1 and E_c is the minimum possible specific energy for the given discharge. It is of course known that for a given discharge the specific energy at a channel section is minimum when the flow is in the critical state. As such when the height of the hump provided in the channel bottom is equal to that given by Eq. 15.57, the depth of flow over the hump will be equal to the critical depth y_c for the given discharge and the velocity of flow will be equal to the corresponding critical velocity V_c.

However, if the height of the hump is increased further, then in order to pass the same discharge the specific energy will have to be increased. In this case for subcritical flow approaching the hump, the required increase in the specific energy will be provided by the increase in the depth of flow at section 1; and for supercritical flow approaching the hump the depth of flow at section 1 will be reduced, thereby providing the required increase in the specific energy. However, if the specific energy is held constant with the hump of height more than the above indicated limiting value, the discharge is decreased until the given specific energy is equal to the minimum specific energy corresponding to the new discharge.

15.13 METERING FLUMES

It was indicated in Section 15.12 that critical depth of flow may be obtained at certain sections in an open channel where the channel bottom is raised by the construction of a low hump or the channel is constricted by reducing its width. Since at critical state of flow the relationship between the depth of flow and discharge is definite and is independent of the channel roughness and other uncontrollable factors, it provides a theoretical basis for the measurement of discharge in open channels. As such various devices which have been developed for flow measurement are based on the principle of critical flow. Some of the devices commonly used for the measurement of flow in open channels are described below.

(a) Venturi Flume. As shown in Fig. 15.15, a venturi flume is a structure in a channel which has a contracted section called throat, downstream of which follows a flared transition section designed to restore the stream to its original width. It is an open channel counterpart of a venturi meter, which is used for measuring discharge in open channels. At the throat section there will be a drop in the water surface and this drop in water surface may be related to the discharge. The velocity of flow at the throat is always less than critical velocity and hence the discharge passing through it will be a function of the difference between the depths of flow upstream of the entrance section and at the throat. Further since the velocity of flow at the throat is less than critical velocity, standing wave or hydraulic jump will not be formed at any section in the venturi flume.

The discharge Q flowing through the channel can be calculated by measuring the depths of flow at the entrance and the throat of the flume and applying the following formula which may be readily derived

$$Q = k\frac{Aa\sqrt{2g}}{\sqrt{A^2 - a^2}}\left(\sqrt{H-h}\right) \quad \text{...(15.58)}$$

Here A, a and H in Fig. 15.58 are the areas and depths of flow section at entrance and throat of the flume respectively and k is the discharge coefficient of the flume. The discharge coefficient must be determined by calibration through the entire range of head. For a rectangular channel Eq. 15.58 may be expressed in the following form

$$Q = k\frac{BH \times bh \times \sqrt{2g}}{\sqrt{(BH)^2 - (bh)^2}}\left(\sqrt{H-h}\right) \quad \text{...(15.59)}$$

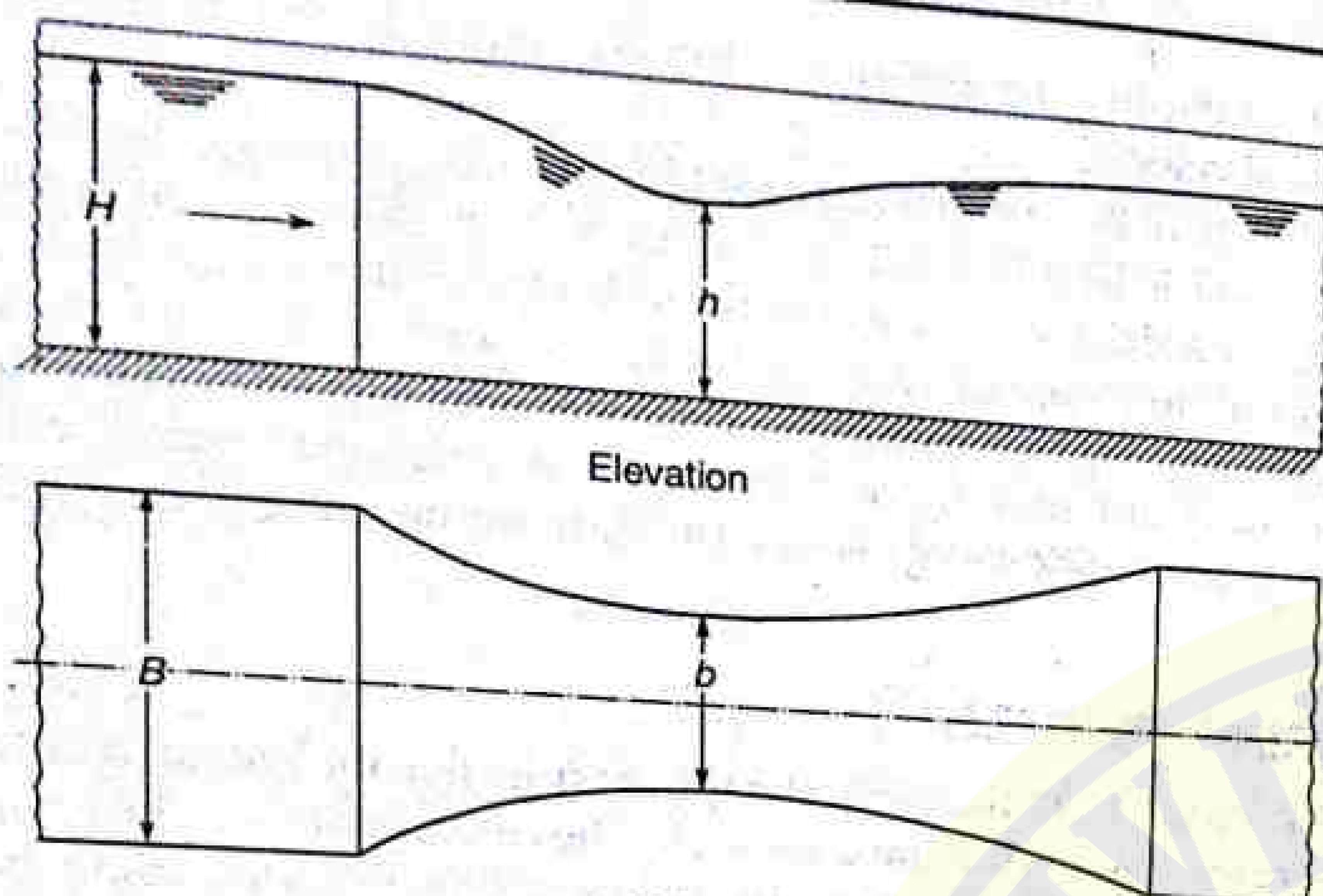


Figure 15.15 *Venturi flume*

in which B and b are the bottom widths at the entrance and the throat of the flume.

(b) Standing Wave Flume or Critical Depth Flume. A standing wave flume is a structure in a channel which has a narrowed throat having a raised floor (or hump) at the bottom as shown in Fig. 15.16, which acts as a broad-crested weir. The downstream of the throat section is followed by a flared

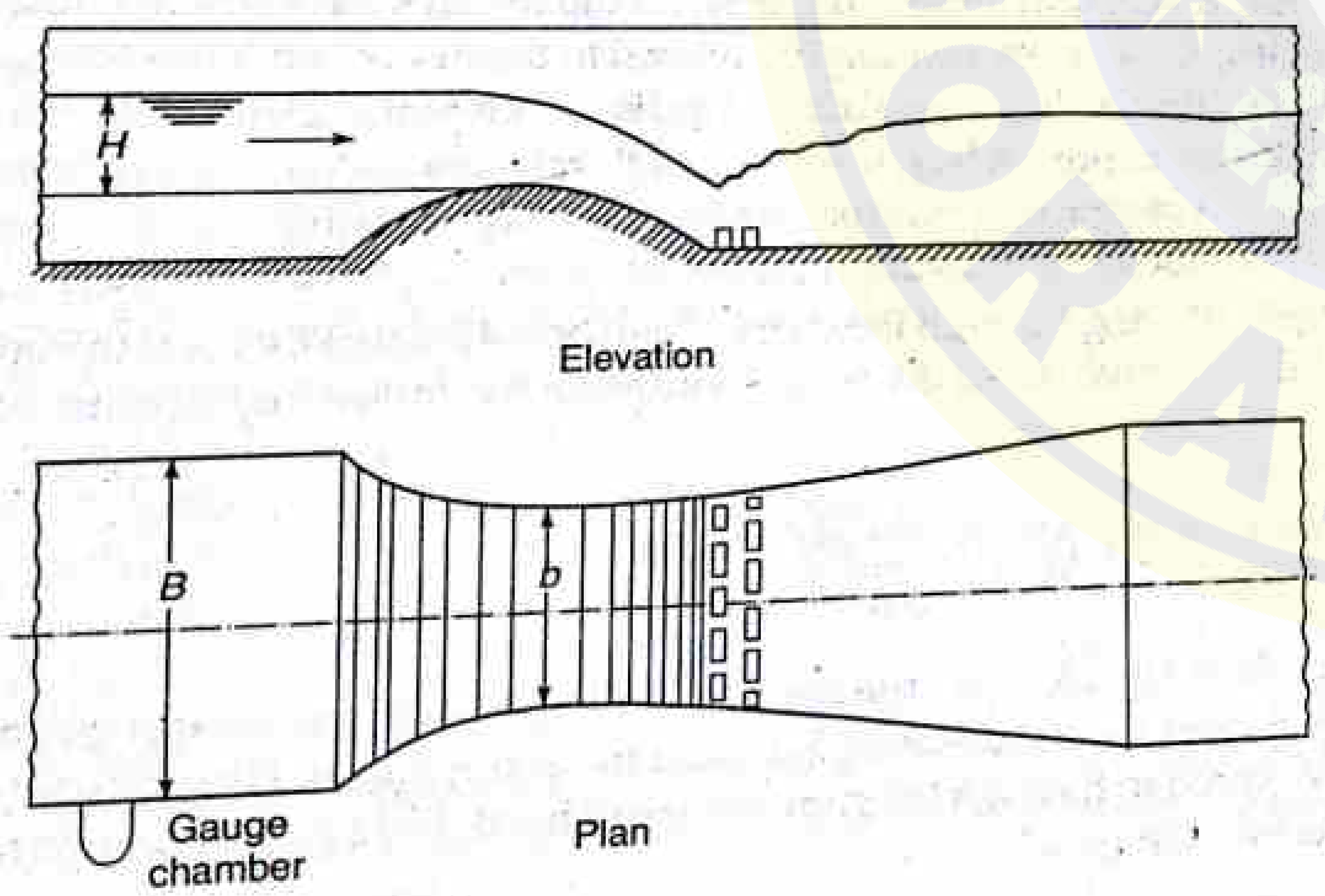


Figure 15.16 *Standing wave flume*

transition section designed to restore the stream to its original width. For any discharge flowing in the channel the velocity of flow at the throat of the flume is greater than critical velocity. As such a standing wave or hydraulic jump is invariably formed at or near the downstream end of the raised

floor. Moreover, since the velocity of flow at the throat is more than critical velocity, the depth of flow at a section upstream of the entrance of the flume remains unaffected by variations in the downstream depth until the downstream depth of submergence becomes greater than about 0.7 of the upstream depth. It may thus be concluded that as long as the downstream depth of flow is kept below the limiting value just mentioned above, the discharge passing through a standing wave flume will be a function of only the depth of flow H (above the raised floor) at a section well upstream of the entrance section, as shown in Fig. 15.16.

Thus in order to determine the discharge flowing in the channel, only the depth of flow H is required to be measured and the discharge Q may be expressed by the following relationship:

$$Q = CbH^{3/2} \quad \text{...(15.60)}$$

in which b is the width of the throat and C is the discharge coefficient of the flume. The value of C may be approximately taken as $[(2/3)^{3/2} \times \sqrt{g}\,]$ which in metric system of units becomes 1.706. However an exact value of C for any flume may be determined by calibration.

(c) Parshall Flume. The Parshall flume is one of the most widely used standing wave flume in U.S.A., which was developed by R.L. Parshall in 1920. As shown in Fig. 15.17 the converging section of the flume has a level floor, the throat section has a downward slopping floor, while the floor in the diverging section slopes upwards. The flume has no raised floor as in the case of an ordinary standing wave flume, but the upward slopping floor of the diverging section facilitates the formation of a standing wave in this portion of the flume as shown in Fig. 15.17. As such for the flumes of intermediate size having downstream depth of flow or submergence less than 0.7 times the upstream depth of flow, free flow will occur and the discharge flowing through the flume is a function of only the upstream depth. The depth–discharge relationship for such flumes may be expressed as

$$Q = CbH^{n} \quad \text{...(15.61)}$$

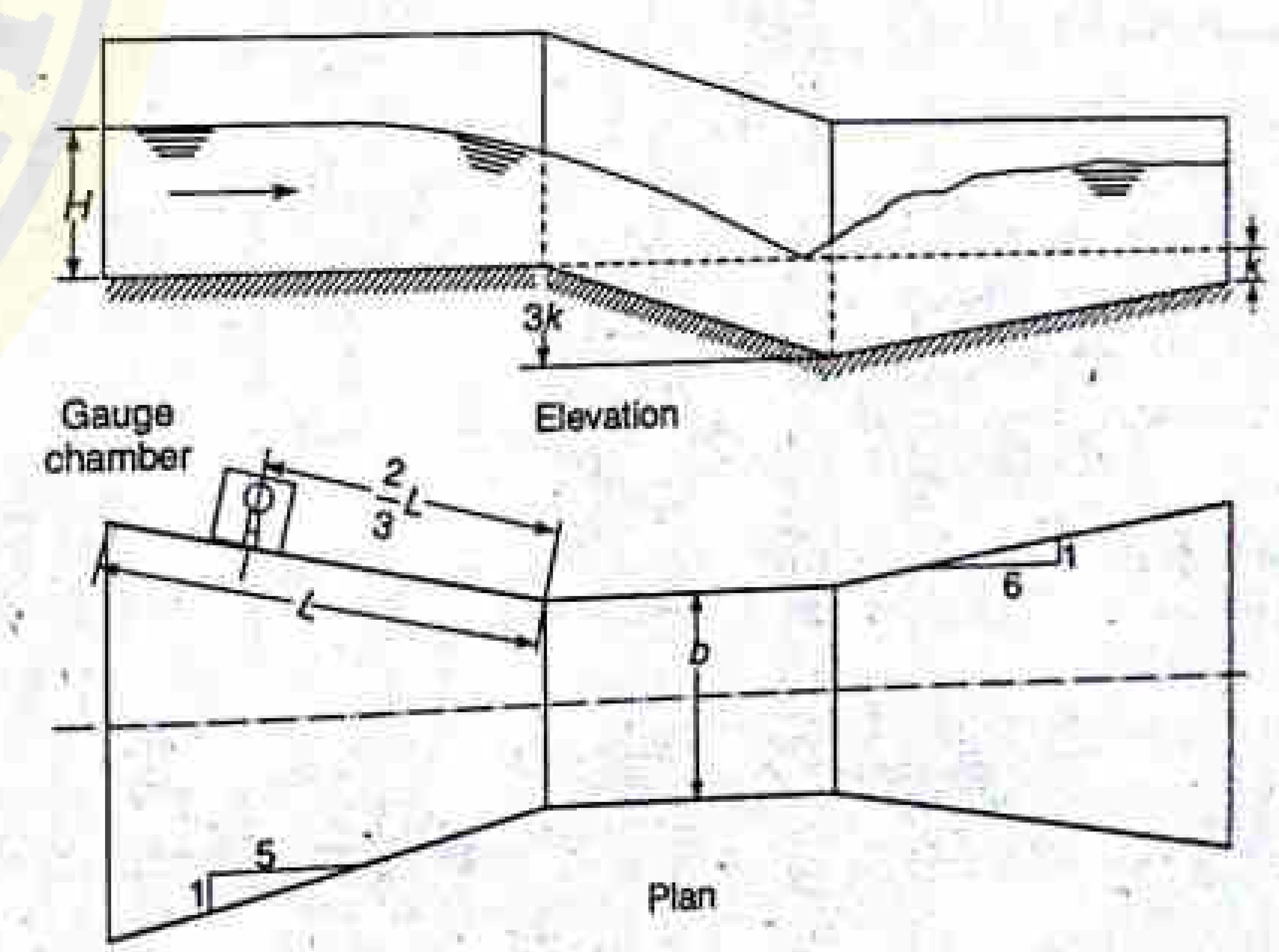


Figure 15.17 *Parshall flume*

In Eq. 15.61, Q is the free discharge, b is the width of the throat, H is the upstream depth of flow measured at gauge chamber shown in Fig. 15.17 and C and n are the constants of the flume which are obtained by calibrations. When the submergence exceeds the above mentioned limit the discharge gets reduced and now if the equation for the free flow condition is used to compute the discharge, the obtained value of Q will have to be corrected by applying a suitable correction factor. Standard design for wood and concrete flumes of various capacities have been worked out for field use and the same are readily available.

15.14 DETERMINATION OF MEAN VELOCITY OF FLOW IN CHANNELS

In order to determine the discharge flowing through a channel the mean velocity of flow is required to be determined. The velocity of flow in channels may be measured by the following methods.

(a) Pitot Tube. As described in Chapter 7 pitot tube is used to measure the velocity of flow at any point. Thus with the help of a pitot tube, velocity distribution along vertical at any section of the channel can be obtained from which the mean velocity of flow can be computed. However, the mean velocity of flow in channels may also be directly obtained by measuring the velocity of flow at 0.6 times the depth below the free surface at any section; or at 0.2 and 0.8 times the depth below the free surface and then taking the mean of the two.

(b) Current Meter. A current meter as shown in Fig. 15.18 is another instrument which is commonly used for measuring the velocity of flow in channel. In general it consists of a wheel or revolving element containing blades or cups, and a tail on which flat vanes or fins are fixed. According to the shape of the revolving element the current meters may be classified as *cup type* and *screw type* or *propeller type*. In a cup type current meter the wheel or the revolving element has the form of a series of

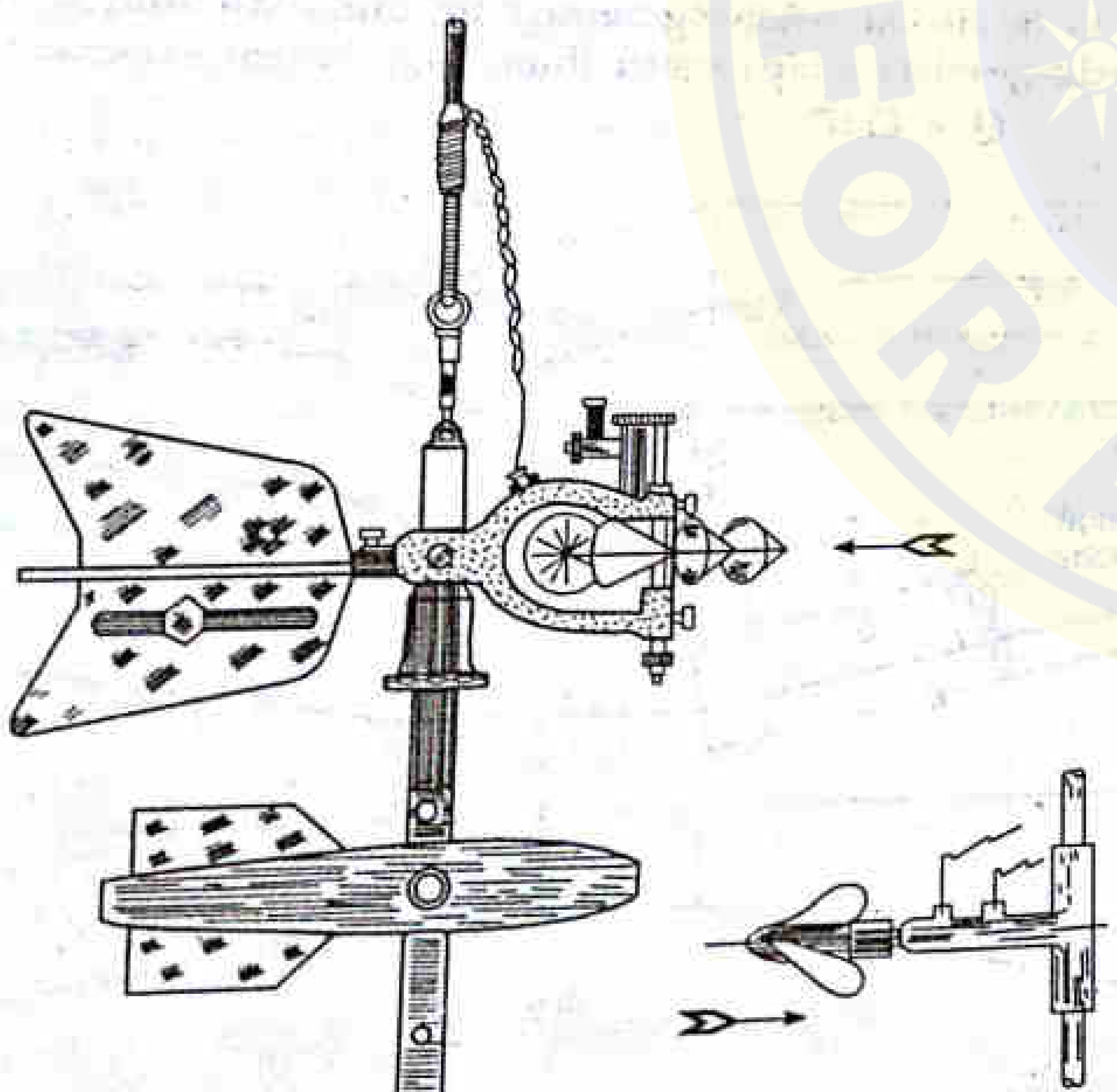


Figure 15.18 *Current meters : (a) Cup type; (b) Screw type*

conical cups, mounted on a spindle which is held vertical at right angle to the direction of flow, Fig. 15.18 (a). In a screw or propeller type current meter the revolving element consists of a shaft, with its axis parallel to the direction of flow, which carries a number of curved vanes or propeller blades mounted around the periphery of the shaft, as shown in Fig. 15.18 (b).

The current meter is suspended by means of a cable and it is held vertically immersed in the flowing stream of water to the required depth, such that the wheel or the revolving element is facing towards the upstream direction. The wheel is thus rotated by the dynamic thrust exerted on it by the flowing water. The number of revolutions of the wheel per unit time are proportional to the velocity of the flowing water. Thus after setting the current meter to the required point in the flowing stream of water the number of revolutions of the wheel per unit time are noted and from the calibration curve the velocity of flow of water may be determined. In order to facilitate the counting of the number of revolutions of the wheel, usually an electrical transmission system is provided. An electric current is passed to the wheel from a battery above the water by means of wires, and a commutator is fixed to the spindle of the revolving blades or cups which makes and breaks the electric circuit during each revolution. An electric bell or a head phone is connected in series in the electric circuit to give an audible signal thereby facilitating the counting of the number of revolutions. Sometimes instead of these a revolution counter is provided. The number of signals in a given time is noted by the observer with a stop watch, from which the speed of the wheel in terms of revolutions per minute may be computed. The current meter is required to be calibrated experimentally. This is done by towing the current meter at various speeds through still water contained in a long tank, and noting the speed of the towing carriage and the rotational speed of the wheel of the current meter. The calibration curve for the current meter may then be prepared by plotting the rotational speed in revolution per minute versus the speed of the towing carriage in metres per second, which will correspond to the velocity of flow of water in the channel when the current meter is held stationary in the water flowing in the channel.

(c) Floats. A float is a small object made of wood or other suitable material which is lighter than water and thus capable of floating on the water surface. It provides a simple way of measuring the velocity of flow of water in rivers and channels. The surface velocity at any section may be obtained by a *single float*. For this the time taken by the float to traverse a known distance is measured and the velocity may be computed. Since the mean velocity of flow is equal to about 0.8 to 0.95 times the surface velocity, the approximate value of the mean velocity of flow may be determined from the known value of the surface velocity.

A better method of determining the velocity of flow of water is to use *double float*, as shown in Fig. 15.19 (a). It consists of a spherical surface float to which is attached a hollow metal sphere, heavier than water and suspended from the float by a cord of known length. The depth of the lower float which is completely submerged may be regulated by the length of the cord. The velocity is then obtained by noting the time taken by the float to traverse a known distance as in the previous case of a single float. However this float gives a velocity equal to the mean between the velocity of the surface and the velocity of the layer traversed by the lower float.

A still better type of float is the *rod float* as shown in Fig. 15.19 (b). It consists of a vertical wooden rod which is weighted at the bottom to keep it vertical with its top end emerging out of free water surface when floating. The length of the rod float should be such that it reaches the bottom of the stream

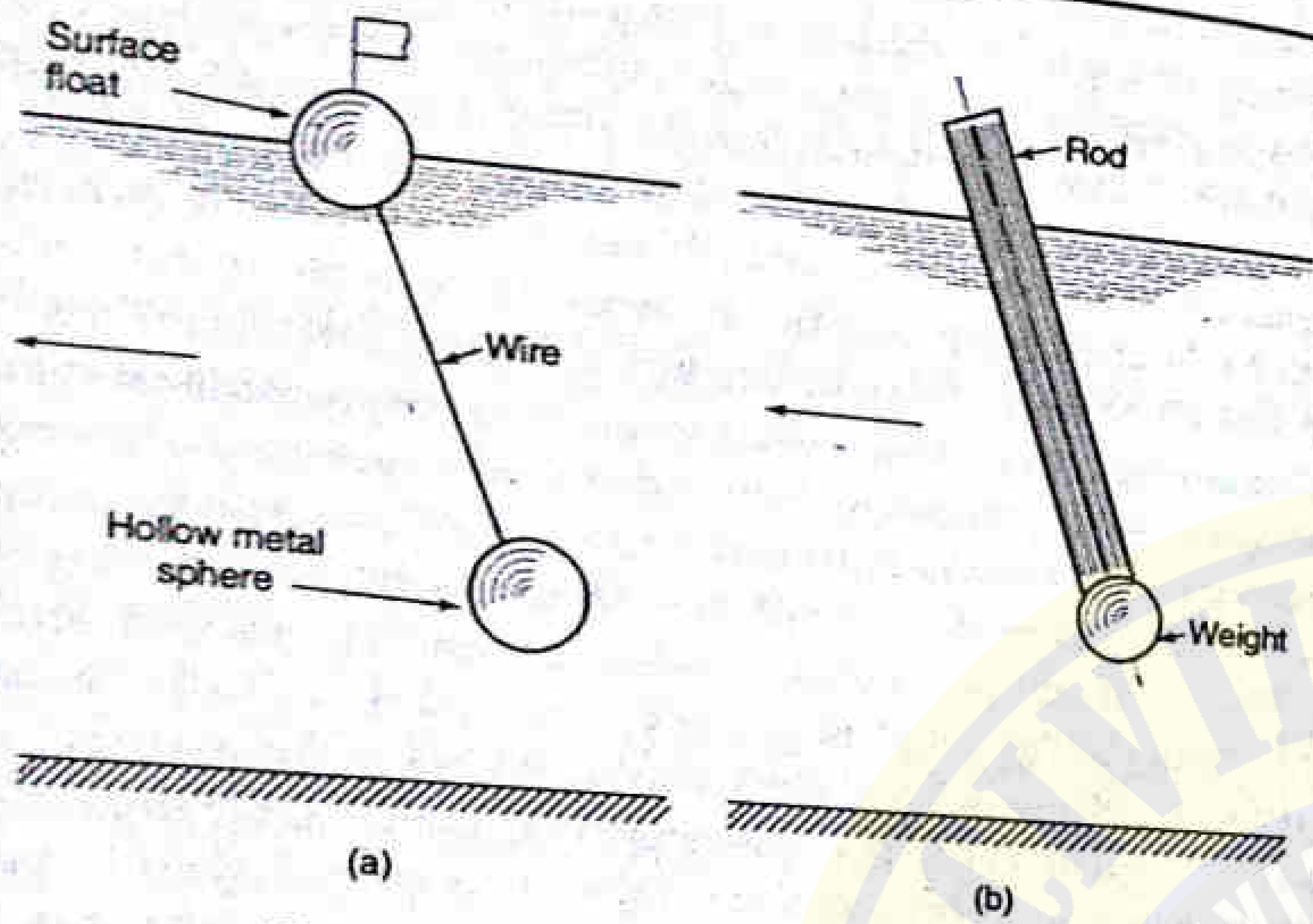


Figure 15.19 *(a) Double float; (b) Rod float*

without touching the sand, weed or mud at the bottom of the stream. Sometimes the rod floats are made telescopic so that the length may be adjusted to suit any depth of flow. The rod floats will travel with a velocity equal to the mean velocity of flow and hence a rod float gives the mean velocity of flow at the section.

15.15 PRACTICAL CHANNEL SECTIONS

In practice the most economical sections for channels obtained in Section 15.6 can not be adopted. This is so because of the two main reasons. Firstly the side slopes to be provided for a channel depends on the type of the soil through which the channel is carried and therefore it may not be possible to adopt 1 : 1 side slope for a triangular channel as indicated by Eq. 15.23. Secondly the sharp corners in he channel cross-section are virtually the zones of stagnation and may lead to the deposition of silt. As such in actual practice the side slopes of the channel sections are required to be modified and also t is desirable to have rounded corners. However for unlined channels the corners can not be made ound during construction but due to silting the channel section gets modified. Moreover, for the esign of an unlined channel altogether different considerations are required to be made as indicated n Chapter 28. For a lined channel the Indian practice is to adopt a triangular section with permissible de slopes and rounded bottom as shown in Fig. 15.20 (a) for discharges less than 55 cumec and a apezoidal section with rounded corners as shown in Fig. 15.20 (b) for higher discharges. In a lined hannel relatively higher mean velocity is permissible which is restricted to 2.5 m/s. As such the esign of a lined channel is based on a limiting velocity rather than any other consideration.

The expressions for the area A, perimeter P and hydraulic mean radius R for the channel sections own in Fig. 15.20 are as follows.

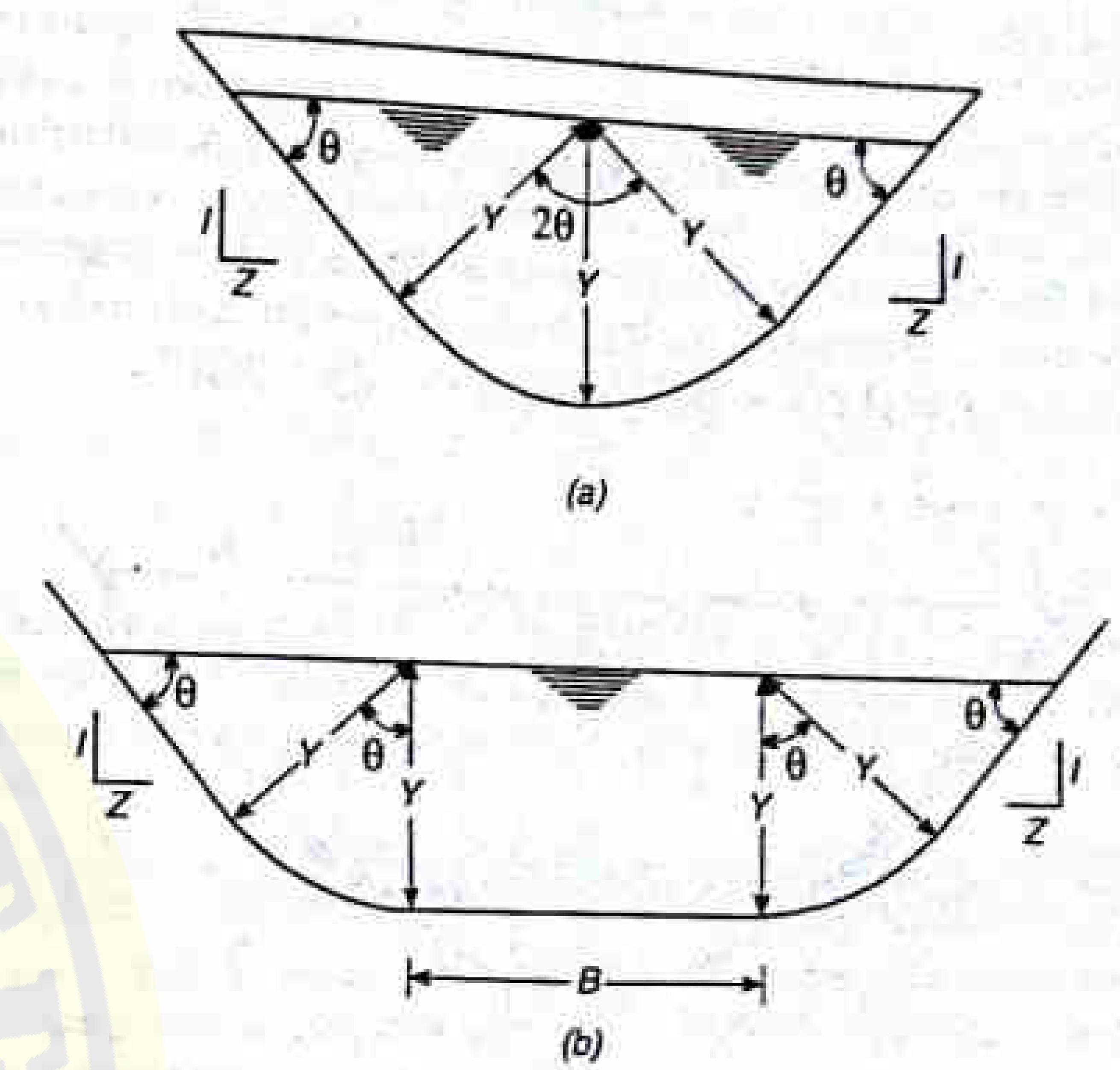


Figure 15.20 *Lined channel section (a) for discharge less than 55 cumec, (b) for discharge more than 55 cumec.*

(a) Triangular Channel Section

$$A = y^2(\theta + \cot\theta)$$
$$P = 2y(\theta + \cot\theta)$$
$$R = \frac{y}{2}$$

(b) Trapezoidal Channel Section

$$A = y(B + y\theta + y\cot\theta)$$
$$P = (B + 2y\theta + 2y\cot\theta)$$
$$R = \frac{y(B + y\theta + y\cot\theta)}{(B + 2y\theta + 2y\cot\theta)}$$

15.16 MEASUREMENT OF DISCHARGE IN RIVERS

A river is an open channel of irregular cross-section. The discharge through a small stream or river may be obtained by fitting a notch or weir across the stream and then measuring the head over the notch as indicated in Chapter 10. However, this method cannot be used for measuring the discharge through a large river. In order to obtain the discharge through a large river, the cross-section of the river is divided into a number of segments, AB, BC, CD, etc., of length l_1, l_2, l_3, etc. as shown in

Fig. 15.22. The vertical lines 1, 2, 3, etc., represent the centre lines of each of the segments. The mean depth of flow d of each of the segments may be measured along the centre lines by soundings. Also the mean velocity of flow through each of the segments may also be measured with the help of a current meter by inserting it along the centre line of the segment to a depth equal to 0.6 times the mean depth below the free water surface. Thus the mean area of each segment may be computed and knowing the mean velocity of flow V through each segment the discharge through any segment is $(ld)V$ and the total discharge through the river is obtained by adding the discharges through each of the segments as

$$Q = [l_1 d_1 V_1 + l_2 d_2 V_2 + l_3 d_3 V_3 + ...] = \Sigma\,(ld)V.$$

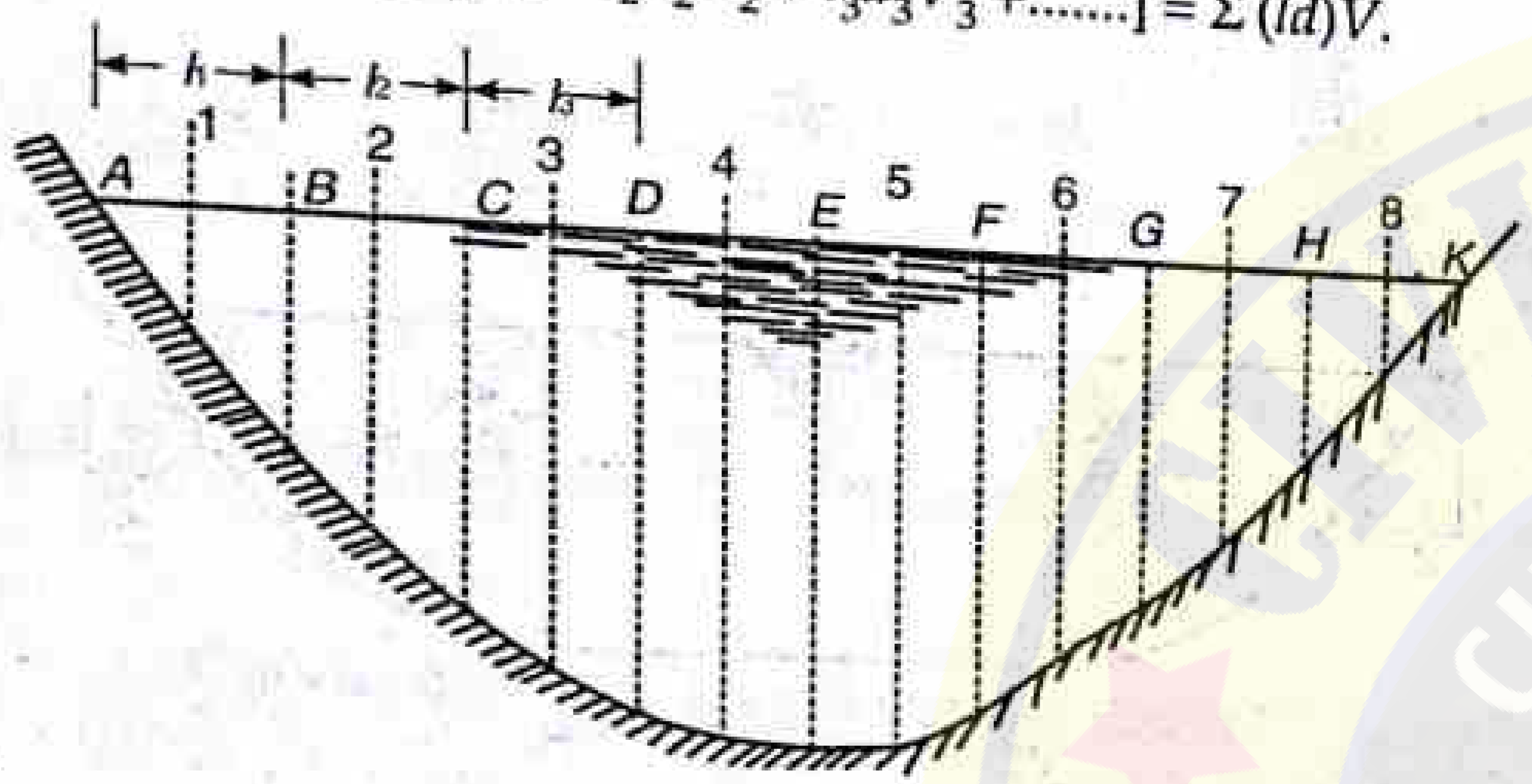


Figure 15.21 *Cross-section of river with unequal segments*

A greater accuracy in the computation of the discharge may be obtained by dividing the river width into the segments of equal width l as shown in Fig. 15.22 and then using either Simpson's rule or Weddle's rule. Simpson's rule requires that the number of segments should be a multiple of 2 and Weddle's rule requires that the number of segments should be a multiple of 6.

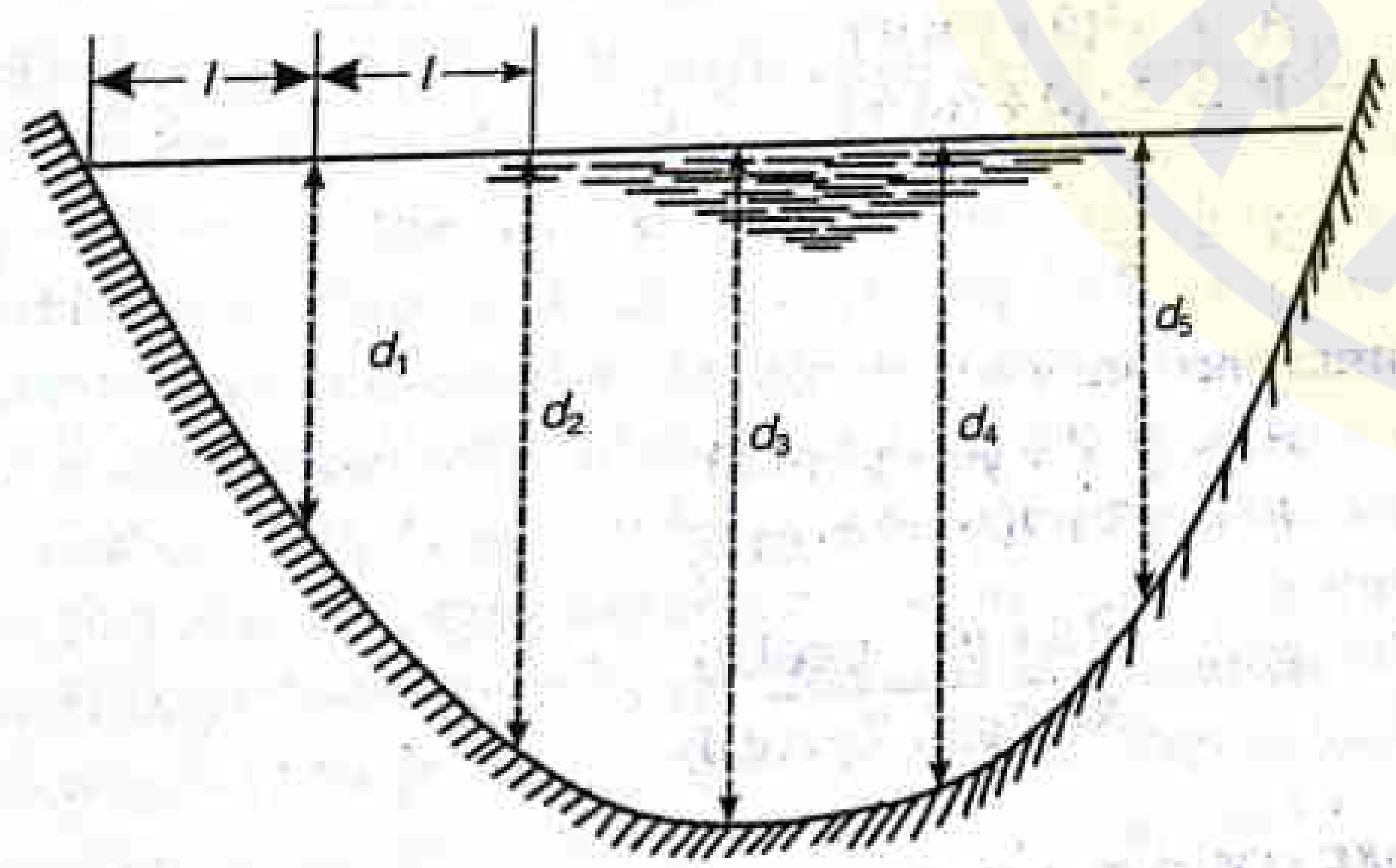


Figure 15.22 *Cross-section of river with equal segments*

Now if d_1, d_2, d_3, etc. and V_1, V_2, V_3, etc., are the respective depths and velocities at the ends of the segments marked 1, 2, 3, etc., and the velocity at the bank being taken zero, then according to Simpson's rule total discharge Q through the river is given as

$$Q = \frac{l}{3}[0 + 4d_1V_1 + 2d_2V_2 + 4d_3V_3 + 2d_4V_4 + 4d_5V_5 + 0]$$

$$= \frac{2l}{3}[d_2V_2 + d_4V_4 + 2(d_1V_1 + d_3V_3 + d_5V_5)]$$

Similarly according to Weddle's rule the total discharge Q through the river is given as

$$Q = \frac{3l}{10}[0 + 5d_1V_1 + d_2V_2 + 5d_3V_3 + d_4V_4 + 5d_5V_5 + 0]$$

$$= \frac{3l}{10}[d_2V_2 + d_4V_4 + 5(d_1V_1 + d_3V_3 + d_5V_5)]$$

If the cross-section of the river in the central portion is more or less uniform then it is preferred to use a combination of Simpson's and Weddle's rule. For this the channel is divided into ten equal segements. The discharge through the central six segments is calculated by Weddle's rule and that of the side portions by Simpson's rule.

ILLUSTRATIVE EXAMPLES

Example 15.1. *An irrigation channel of trapezoidal section, having side slopes 3 horizontal to 2 vertical, is to carry a flow of 10 cumec on a longitudinal slope of 1 in 5000. The channel is to be lined for which the value of friction coefficient in Manning's formula is $n = 0.012$. Find the dimensions of the most economic section of the channel.*

Solution

For most economic trapozoidal channel section from Eq. 15.17, we have

$$\frac{B + 2zy}{2} = y\sqrt{1+z^2}$$

Since $z = \frac{3}{2} = 1.5$, we have

$$B + 3y = 2y\sqrt{1+(1.5)^2} = 3.6y$$

or $B = 0.6y$

Also from Eq. 15.18

$$R = \frac{y}{2}$$

Using Manning's equation

$$V = \frac{1}{n}R^{2/3}S^{1/2}$$

or

$$Q = \frac{1}{n}AR^{2/3}S^{1/2}$$

$$Q = 10 \text{ cumec}; S = \frac{1}{5000}; n = 0.012$$

Also

$$A = (B+zy)y = (0.6y+1.5y)y = 2.1y^2$$

Thus by substitution, we get

$$10 = \frac{1}{0.012} \times 2.1y^2 \times \left(\frac{y}{2}\right)^{2/3} \left(\frac{1}{5000}\right)^{1/2}$$

or

$$y^{8/3} = 6.414$$

$\therefore$

$$y = 2 \text{ m and } B = 1.2 \text{ m.}$$

Example 15.2. *A most efficient trapezoidal section is required to give a maximum discharge of 21.5 m³/s of water. The slope of the channel bottom is 1 in 2500. Taking C = 70 m^{1/2}/s in Chezy's equation, determine the dimensions of the channel. Also determine the value of Manning's n, taking the value of velocity of flow as obtained for the channel by Chezy's equation.*

Solution

For most efficient trapezoidal section of channel taking side slope $z = (1/\sqrt{3})$, from Eq. 15.17 or 15.20, we get

$$\frac{B}{y} = \frac{2}{\sqrt{3}} = 1.1547$$

or

$$B = 1.1547y$$

Using Chezy's equation

$$V = C\sqrt{RS}$$

or

$$Q = CA\sqrt{RS}$$

$$Q = 21.5 \text{ m}^3/\text{s};\ C = 70;\ S = \frac{1}{2500};$$

$$A = (B+zy)\,y$$

$$= \left(1.1547y + \frac{1}{\sqrt{3}}y\right)y = 1.7321y^2$$

and

$$R = \frac{y}{2} = 0.5y \quad \text{(Eq. 15.18)}$$

Thus by substitution, we get

$$21.5 = 70 \times 1.7321y^2 \times \sqrt{0.5y \times \frac{1}{2500}}$$

or

$$y^{5/2} = 12.539$$

$\therefore$

$$y = 2.75 \text{ m}$$

and

$$B = 1.1547 \times 2.75 = 3.175 \text{ m}$$

The wetted perimeter P is given by

$$P = B + 2y\sqrt{1+z^2}$$

$$= 3.175 + 2 \times 2.75 \times \sqrt{1 + \left(\frac{1}{\sqrt{3}}\right)^2}$$

$$= 9.526 \text{ m}$$

As a check it may be shown that if side slope z is taken either more or less than $(1/\sqrt{3})$ the wetted perimeter P is more than the one obtained above for side slope $z = (1/\sqrt{3}) = 0.5774$.

Thus taking side slope $z = 1$, we have from Eq. 15.17

$$\frac{B+2zy}{2} = y\sqrt{1+z^2}$$

or

$$B = 0.828\,y$$

Using Chezy's equation

$$V = C\sqrt{RS}$$

or

$$Q = CA\sqrt{R}$$

$$Q = 21.5 \text{ m}^3/\text{s};\ C = 70;\ S = \frac{1}{2500};$$

$$A = (B+zy)\,y$$

$$= (0.828y + 1 \times y)y = 1.828y^2$$

and

$$R = \frac{y}{2} = 0.5y \quad \text{(Eq. 15.18)}$$

Thus by substitution, we get

$$21.5 = 70 \times 1.828y^2 \times \sqrt{0.5y \times \frac{1}{2500}}$$

or

$$y^{5/2} = 11.881$$

$\therefore$

$$y = 2.692 \text{ m}$$

and

$$B = 2.229 \text{ m}$$

The wetted perimeter is given by

$$P = B + 2y\sqrt{1+z^2}$$

$$= 2.229 + 2 \times 2.692 \times \sqrt{1+(1)^2}$$

$$= 9.843 \text{ m}$$

which is more than the one obtained above for side slope $z = (1/\sqrt{3}) = 0.5774$.

Similarly taking side slope $z = 0.4$, we have from Eq. 15.18

$$\frac{B+2zy}{2} = y\sqrt{1+z^2}$$

or
$$B = 1.354y$$

Using Chezy's equation
$$V = C\sqrt{RS}$$
or
$$Q = CA\sqrt{RS}$$
$$Q = 21.5\ \text{m}^3/\text{s};\ C = 70;\ S = \frac{1}{2500}$$
$$A = (B + zy)y$$
$$= (1.354y + 0.4y)y = 1.754y^2$$
and
$$R = \frac{y}{2} = 0.5y$$

Thus by substitution, we get
$$21.5 = 70 \times 1.754y^2 \times \sqrt{0.5y \times \frac{1}{2500}}$$
or
$$y^{5/2} = 12.382$$
$\therefore$
$$y = 2.736\ \text{m}$$
and
$$B = 3.705\ \text{m}$$

The wetted perimeter is given by
$$P = B + 2y\sqrt{1+z^2}$$
$$= 3.705 + 2 \times 2.736 \times \sqrt{1+(0.4)^2}$$
$$= 9.599\ \text{m}$$

which is more than the one obtained above for side slope $z = (1/\sqrt{3}) = 0.5774$.

Thus the dimensions of the most efficient trapezoidal section of the channel are
$$B = 3.175\ \text{m};\ y = 2.75\ \text{m};\ \text{and}\ z = (1/\sqrt{3}) = 0.5774.$$

From Eq. 15.6, we have
$$C = \frac{1}{n}R^{1/6}$$
$$R = \frac{y}{2} = \frac{2.75}{2} = 1.375\ \text{m}$$
$$C = 70$$

Thus by substitution, we get
$$70 = \frac{1}{n} \times (1.375)^{1/6}$$
or
$$n = \frac{(1.375)^{1/6}}{70} = 0.015$$
$$\text{Manning's } n = 0.015$$

Example 15.3. *An irrigation channel is to carry a discharge of 14 cumec with a velocity of 0.9 m/s and bed slope of 1 in 2500. The side slopes are 1 to 1. Find the depth and bottom width. The values of Chezy's C for this channel for different values of hydraulic radius R are as tabulated below.*

Hydraulic Radius R	0.7	0.8	0.9	1.0	1.1	1.2	1.3	1.4
Chezy's C	34	35	37	38	39	40	41	41

Solution

Discharge $Q = 14\ \text{m}^3/\text{s}$

Velocity $V = 0.9\ \text{m/s}$

Area of flow section
$$A = \frac{Q}{V}$$
$\therefore$
$$A = \frac{14}{0.9} = 15.56\ \text{m}^2$$

From Chezy's formula
$$V = C\sqrt{RS}$$

Bed slope
$$S = \frac{1}{2500}$$

Thus by substitution, we get
$$0.9 = C\sqrt{R \times \frac{1}{2500}}$$
or
$$C\sqrt{R} = 45$$

Assuming $R = 1.0$, then $C = 38$

$\therefore$ $C\sqrt{R} = 38$

Assuming $R = 1.3$, then $C = 41$

$\therefore$ $C\sqrt{R} = 46.75$

Assuming $R = 1.241$, then $C = 40.41$

$\therefore$ $C\sqrt{R} = 45$

which is same as the value for this case as obtained above

$\therefore$ $R = 1.241$

We have
$$A = (B + y)y = 15.56 \quad \ldots\text{(i)}$$
$$R = \frac{15.56}{B + 2y\sqrt{2}} = 1.241 \quad \ldots\text{(ii)}$$

From Eq. (ii), we have

$$B = (12.54 - 2.83y)$$

Substituting the value of B in Eq. (i), we get

$$(12.54 - 2.83\,y + y)\,y = 15.56$$

or
$$12.54\,y - 1.83\,y^2 = 15.56$$

or
$$y^2 - 6.85\,y + 8.50 = 0$$

∴
$$y = 1.63 \text{ m}$$

Substituting the value of y in Eq. (i), we get

$$B = 7.92 \text{ m}$$

Hence the required dimensions of the channel are

$$B = 7.92 \text{ m; and } y = 1.63 \text{ m}$$

Example 15.4. *A channel carries a discharge of 5 cumec with a flow depth 1 m. The side slopes are 1 to 1 and bed slope 1 in 3520. Find the bed width and the velocity. The values of Chezy's C for this channel for different values of hydraulic radius R are as tabulated below.*

Hydraulic Radius R	0.7	0.8	0.9	1.0	1.1	1.2	1.3	1.4
Chezy's C	34	35	37	38	39	40	41	41

Solution

Discharge $Q = 5 \text{ m}^3/\text{s}$

Flow depth $y = 1 \text{ m}$

Bed slope $S = \dfrac{1}{3520}$

Assuming $R = 1.0$, then $C = 38$

From Chezy's formula

$$V = C\sqrt{RS}$$

Thus by substitution, we get

$$V = 38\sqrt{1 \times \frac{1}{3520}}$$

or
$$V = 0.64 \text{ m/s}$$

∴ Area of flow section

$$A = \frac{5}{0.64} = 7.81 \text{ m}^2$$

We have

$$A = (B + y)\,y$$

Thus by substitution, we get

$$(B+1)\,1 = 7.81$$

∴
$$B = 6.81 \text{ m}$$

Wetted perimeter $P = B + 2y\sqrt{2}$

or
$$P = (6.81 + 2.83) = 9.64 \text{ m}$$

Hence the correct value of R is obtained as

$$R = \frac{7.81}{9.64} = 0.81$$

For this value of R, $C = 35.2$

Thus
$$V = 35.2\sqrt{0.81 \times \frac{1}{3520}}$$

or
$$V = 0.53 \text{ m/s}$$

∴
$$A = \frac{5}{0.53} = 9.43 \text{ m}^2$$

We have
$$A = (B+1) \times 1 = 9.43$$

∴
$$B = 8.43 \text{ m}$$

Hence
$$V = 0.53 \text{ m/s; and } B = 8.43 \text{ m}$$

Example 15.5. *A channel is 2m wide at bottom; the length of each sloping side is 1.95 m; the width at water surface is 5.5 m the flow depth 1.2 m; and bed slope 1 in 5280. What is the discharge per minute? The values of Chezy's C for this channel for different values of hydraulic radius R are as tabulated below.*

Hydraulic Radius R	0.7	0.8	0.9	1.0	1.1	1.2	1.3	1.4
Chezy's C	34	35	37	38	39	40	41	41

Solution

Bottom width $B = 2 \text{ m}$

Flow Depth $y = 1.2 \text{ m}$

Bed slope $S = \dfrac{1}{5280}$

Length of slopping side

$$y\sqrt{1+z^2} = 1.95 \text{ m}$$

Width at water surface

$$B + 2zy = 5.5 \text{ m}$$

Substituting the value of B, we get

$$2 + 2zy = 5.5$$

or
$$zy = 1.75$$

∴ Area of flow section

$$A = (B + zy)y = (2 + 1.75)\,1.2 = 4.5 \text{ m}^2$$

Wetted perimeter

$$P = B + 2y\sqrt{1+z^2} = 2 + 2(1.95)$$

$$= 5.9 \text{ m}$$

∴ Hydraulic radius

$$R = \frac{4.5}{5.9} = 0.76$$

For this value of R, $C = 34.6$

∴ Velocity of flow

$$V = C\sqrt{RS}$$

Thus by substitution, we get

$$v = 34.6\sqrt{0.76 \times \frac{1}{5280}}$$

$$= 0.415 \text{ m/s}$$

∴ Discharge $Q = 4.5 \times 0.415$

$$= 1.8675 \text{ m}^3/\text{s}$$

$$= 112.05 \text{ m}^3 \text{ per minute}$$

Example 15.6. *Water flows at a uniform depth of 2 m in a trapezoidal channel having a bottom width 6 m, side slopes 2 horizontal to 1 vertical. If it has to carry a discharge of 65 m³/s, compute the bottom slope required to be provided. Take Manning's n = 0.025.*

Solution

According to Manning's formula $\sqrt{1+z^2}$

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

From the given data

$$A = (B + zy) = (6 + 2 \times 2)2 = 20 \text{ m}^2$$

$$P = B + 2y \qquad = 6 + 2 \times 2\left(\sqrt{5}\right) = 14.94 \text{ m}$$

$$R = \frac{A}{P} = \frac{20}{14.94} = 1.34$$

$$V = \frac{Q}{A} = \frac{65}{20} = 3.25 \text{ m/s}$$

By substituting, we have

$$3.25 = \frac{1}{0.025}(1.34)^{2/3} S^{1/2}$$

or

$$S^{1/2} = 0.066\ 85$$

∴

$$S = 0.0045$$

Example 15.7. *A flow of 100 litres per second flows down in rectangular laboratory flume of width 0.6 m and having adjustable bottom slope. If Chezy's C is 56 determine the bottom slope necessary for uniform flow with a depth of flow 0.3 m. Also find the conveyance and the state of flow (i.e., tranquil or rapid).*

Solution

The discharge flowing in the flume is

$$Q = 100 \text{ l/s} = 100 \times 10^{-3} = 0.1 \text{ m}^3/\text{s}$$

The area of flow section is

$$A = 0.6 \times 0.3 = 0.18 \text{ m}^2$$

Thus mean velocity of flow is

$$V = \frac{Q}{A} = \frac{0.10}{0.18} = 0.56 \text{ m/s}$$

The wetted perimeter of the flow section is

$$P = [B + 2y] = [0.6 + 2(0.3)] = 1.2 \text{ m}$$

The hydraulic radius is

$$R = \frac{A}{P} = \frac{0.18}{1.2} = 0.15 \text{ m}$$

From Chezy's formula, $V = C\sqrt{RS}$

Thus by substituting the computed quantities

$$0.56 = 56\sqrt{0.15 \times S}$$

$$\therefore \quad S = \frac{1}{0.15 \times 10^4} = \frac{1}{1500}$$

i.e., the required bottom slope is 1 in 1 500.

From Eq. 15.29, conveyance is given by

$$K = CA\sqrt{R} = 56 \times 0.18\,(0.15)^{1/2} = 3.904$$

The Froude number of the flow is

$$Fr = \frac{V}{\sqrt{gD}}$$

$$D = 0.30 \text{ m}$$

Thus

$$Fr = \frac{0.56}{\sqrt{9.81 \times 0.30}} = 0.326$$

Since $Fr < 1$, the flow in the flume is tranquil.

Example 15.8. *Derive the correlation between roughness factor f and Manning's coefficient n, in open channel flow.*

Solution

According to Chezy's formula from Eq. 15.2, we have

$$V = \sqrt{\frac{8g}{f}}\sqrt{RS} \qquad \text{...(i)}$$

From Manning's formula, we have

$$V = \frac{1}{n}R^{2/3}S^{1/2} \quad \text{(ii)}$$

From Eq. (i) and (ii), we obtain

$$\sqrt{\frac{8g}{f}}R^{1/2}S^{1/2} = \frac{1}{n}R^{2/3}S^{1/2}$$

or

$$\sqrt{\frac{8g}{f}} = \frac{1}{n}R^{1/6}$$

or

$$\frac{8g}{f} = \frac{1}{n^2}R^{1/3}$$

which is the required correlation between roughness factor f, and Manning's coefficient n.

Example 15.9. *An earthen channel with a base width 2 m and side slope 1 horizontal to 2 vertical carries water with a depth of 1 m. The bed slope is 1 in 625. Calculate the discharge if n = 0.03. Also calculate the average shear stress at the channel boundary.*

Solution

$$B = 2\,\text{m}, y = 1\,\text{m}, z = \frac{1}{2}$$

Thus area of flow section is

$$A = (B + zy)y = \left(2 + 1 \times \frac{1}{2}\right) 1 = 2.5\,\text{m}^2$$

Also wetted perimeter is

$$P = (B + 2y\sqrt{1+z^2}) = [2 + 2 \times 1 \times \sqrt{(5/4)}] = 4.24\,\text{m}$$

Thus

$$R = \frac{A}{P} = \frac{2.5}{4.24} = 0.59$$

Using Manning's formula

$$V = \frac{1}{n}R^{2/3}S^{1/2}$$

$$= \frac{1}{0.03} \times (0.59)^{2/3} \times \left(\frac{1}{625}\right)^{1/2} = 0.94\,\text{m/s}$$

$\therefore$ Discharge $Q = A \times V = (2.5 \times 0.94) = 2.35\,\text{m}^3/\text{s}$

Average shear stress at the channel boundary,

$$\tau_0 = wRS = 9810 \;\; 0.59 \times \left(\frac{1}{625}\right) = 9.26\,\text{N/m}^2$$

Example 15.10. *Compute the average shear stress at the sides and the bottom of a rectangular flume 3.2 m wide, flowing 1.1 m deep and laid on a slope of 1.3 m per 1000 m.*

Solution

Bed slope $S = \frac{1.3}{1000}$

Aera of flow section $A = (3.2 \times 1.1)$
$= 3.52\,\text{m}^2$

Wetted perimeter $P = [3.2 + (2 \times 1.1)]$
$= 5.4\,\text{m}$

Hydraculic radius $R = \frac{3.52}{5.4}$
$= 0.652\,\text{m}$

Average shear stress on sides

$= 0.75\,wRS$

$= 0.75 \times 9810 \times 0.652 \times \frac{1.3}{1000}$

$= 6.236\,\text{N/m}^2$

Average shear stress on bottom

$= wRS$

$= 9810 \times 0.652 \times \frac{1.3}{1000}$

$= 8.315\,\text{N/m}^2$

Example 15.11. *A trapezoidal channel with bottom width 3.5 m and side slopes 1 H : 1 V on the left and 1.5H : 1V on the right, with n = 0.016, and a bed slope of 2.6 in 10000 carries a discharge of 8 m³/s. Determine the normal depth and the average shear stress on the channel bed.*

Solution

Let y be the normal depth of flow.

Then

$$A = \left(By + \frac{1}{2}z_1y^2 + \frac{1}{2}z_2y^2\right)$$

$$= (3.5y + 0.5y^2 + 0.75y^2)$$

$$= (3.5y + 0.25y^2)$$

and

$$P = \left(B + y\sqrt{1+z_1^2} + y\sqrt{1+z_2^2}\right)$$

$$= \left(3.5 + \sqrt{2y} + \frac{1}{2}\sqrt{13y}\right)$$

$$= (3.5 + 3.217y)$$

$$R = \frac{A}{p} = \frac{(3.5y + 1.25y^2)}{(3.5 + 3.217y)}$$

From Manning's formula, we have

$$Q = \frac{1}{n} A R^{2/3} S^{1/2}$$

or

$$8 = \frac{1}{0.016}\left[(3.5y+1.25y^2)\times\left\{\frac{(3.5y+1.25y^2)}{(3.5+3.217y)}\right\}^{2/3}\times\left(\frac{2.6}{10000}\right)^{1/2}\right]$$

or

$$8 = \frac{1}{0.016}\left[\frac{(3.5y+1.25y^2)^{5/3}}{(3.5+3.217y)^{2/3}}\times\left(\frac{2.6}{10000}\right)^{1/2}\right]$$

Solving this equation for y by trial, we get

$$y = 1.505 \text{ m}$$

Average shear stress on the channel bed is given by Eq. 15.1 as

$$\tau_0 = wRS$$

$$R = \frac{[3.5\times1.505+1.25(1.505)^2]}{[3.5+(3.217\times1.505)]}$$

$$= \frac{8.09878}{8.34159} = 0.9709$$

$\therefore$ Average shear stress on the channel bed

$$\tau_0 = 9810\times0.9709\times\frac{2.6}{10000}$$

$$= 2.476 \text{ N/m}^2$$

Example 15.12. *Water flows in a channel of the shape of isosceles triangle of bed width 'a' and sides making an angle of 45° with the bed. Determine the relations between depth of flow 'd' and bed width 'a' for maximum velocity condition and for maximum discharge condition. Use Manning's formula and note that 'd' is less th 0.5 'a'.*

Solution

From Manning's equation, we have

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

Thus for maximum velocity, we have

$$\frac{dR}{dd} = 0 \text{ ; or } \frac{d(A/P)}{dd} = 0$$

Since

$$\frac{d(A/P)}{dd} = \frac{P(dA/dd)-A(dP/dd)}{P^2}$$

$$\therefore \quad P(dA/dd) - A(dP/dd) = 0$$

In this case

$$A = (a-d)d$$

$$\therefore \quad (dA/dd) = (a-2d)$$

and

$$P = (a+2\sqrt{2}d)$$

$$\therefore \quad (dP/dd) = 2\sqrt{2}$$

Thus by substitution, we get

$$\frac{(a+2\sqrt{2}d)(a-2d)-(a-d)d\times2\sqrt{2}}{(a+2\sqrt{2}d)^2}$$

or

$$2\sqrt{2}d^2 + 2ad - a^2 = 0$$

or

$$d = \frac{-2a\pm\sqrt{4a^2+8\sqrt{2}a^2}}{4\sqrt{2}}$$

$$\therefore \quad d = 0.338\,a$$

Discharge Q is given by

$$Q = \frac{1}{n} A R^{2/3} S^{1/2}$$

or

$$Q = \frac{1}{n} A\left(\frac{A}{P}\right)^{2/3} S^{1/2}$$

or

$$Q = \frac{1}{n}\left(\frac{A^5}{P^2}\right)^{1/3} S^{1/2}$$

Thus for maximum discharge, we have

$$\frac{d(A^5/P^2)}{dd} = 0$$

Since

$$\frac{d(A^5/P^2)}{dd} = \frac{P^2\times5A^4(dA/dd)-A^5\times2P(dP/dd)}{P^4}$$

$$\therefore \quad 5P(dA/dd) - 2A(dP/dd) = 0$$

Thus by substitution, we get

$$5\left(a+2\sqrt{2}d\right)\ (a-2d)-2(a-d)\ d\times\sqrt{2}=0$$

or $$16\sqrt{2}\,d^2+(10-6\sqrt{2})\,ad-5a^2=0$$

or $$22.627\ d^2+1.515\ ad-5a^2=0$$

or $$d=\frac{-1.515a\pm\sqrt{2.295a^2+452.54a^2}}{45.254}$$

$\therefore$ $$d=0.438\ a$$

Example 15.13. *A trapezoidal channel having a cross–sectional area A_1, wetted perimeter P_1, Manning's coefficient n and laid to a slope S, carries a certain discharge Q_1 at a depth of flow equal to d. To increase the discharge the base width of the channel is widened by x, keeping all other parameters viz., S, d, side slope and n same. Prove that*

$$\left[\frac{Q_2}{Q_1}\right]^3\left[1+\frac{x}{P_1}\right]^2=\left[1+\frac{xd}{A_1}\right]^5$$

where Q_2 is the new discharge in the channel.

Solution

Let b be the initial base width of the channel and the side slopes be z horizontal to 1 vertical. Then

$$A_1=(b+zd)\,d$$

and $$P_1=(b+2d\sqrt{1+z^2})$$

When the base width is increased by x, the new cross-sectional area $A2$ and wetted perimeter $P2$ are given by

$$A_2=(b+x+zd)\,d=(A1+xd)\qquad\text{...(i)}$$

and $$P_2=(b+x+2d\sqrt{1+z^2})=(P1+x)\qquad\text{...(ii)}$$

Discharge Q_1 is given by

$$Q_1=\frac{1}{n}A_1\left(\frac{A_1}{P_1}\right)^{2/3}S^{1/2}$$

or $$Q_1=\frac{1}{n}\frac{A_1^{5/3}}{P_1^{2/3}}\qquad\text{...(iii)}$$

Similarly discharge $Q2$ is given by

$$Q_2=\frac{1}{n}A_2\left(\frac{A_2}{P_2}\right)^{2/3}S^{1/2}$$

or $$Q_2=\frac{1}{n}\frac{A_2^{5/3}}{P_2^{2/3}}S^{1/2}\qquad\text{...(iv)}$$

From Eqs (iii) and (iv), we get

$$\left[\frac{Q_2}{Q_1}\right]^3=\left[\frac{A_2}{A_1}\right]^5\left[\frac{P_1}{P_2}\right]^2$$

By substituting the values of A_2 and P_2 from Eqs (i) and (ii), we get

$$\left[\frac{Q_2}{Q_1}\right]^3=\left[\frac{A_1+xd}{A_1}\right]^5\left[\frac{P_1}{P_1+x}\right]^2$$

or $$\left[\frac{Q_2}{Q_1}\right]^3\left[1+\frac{x}{P_1}\right]^2=\left[1+\frac{xd}{A_1}\right]^5$$

Example 15.14. *An earth canal in good condition is 17 m wide at the bottom and has side slope of 2 horizontal to 1 vertical. One side slope extends to a height of 7.8 m above the bottom level and the other side extends to an elevation of 1.8 m, then extends flat to a distance of 150 m and rises vertically. If the slope of the canal is 0.7 m per 1 610 m, estimate the discharge when the depth of water is 2.5 m. Assume Chezy's C = 35.*

Solution

As shown in the above figure the area of flow section of the channel is

$$A=ABK+KBCH+HCDG+GDEF$$

$$=\left(\frac{1}{2}\times5\times2.5\right)+(17\times2.5)+\left[\left(\frac{2.5+0.7}{2}\right)\times3.6\right]+(150\ 0.7)$$

$$=6.25+42.5+5.76+105$$

$$=159.51\ \text{m}^2$$

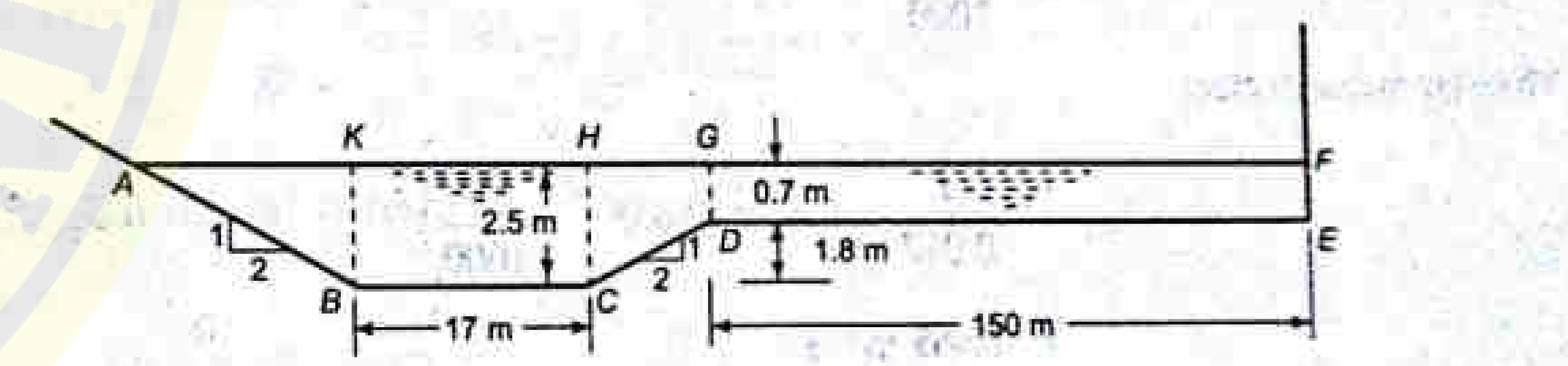


Fig. Ex. 15.14

Wetted perimeter is

$$P=AB+BC+CD+DE+EF$$

$$=(2.5\sqrt{5})+17+(1.8\sqrt{5})+150+0.70$$

$$=5.59+17+4.02+150+0.70$$

$$=177.31\ \text{m}$$

Thus $$R=\frac{A}{P}=\frac{159.51}{177.31}=0.90$$

Using Chezy's formula

$$Q=AC\sqrt{RS}$$

$$S = \frac{0.7}{1610} = \frac{1}{2300} \;; C = 35$$

Thus by substitution, we get

$$Q = 159.51 \times 35 \left(0.90 \times \frac{1}{2300}\right)^{1/2}$$

$$= 110.44 \text{ m}^3/\text{s}$$

Example 15.15. *A rectangular channel 5.4 m wide and 1.2 m deep has a slope of 1 in 1000 and is lined with good rubble masonry; for which Manning's n = 0.017. It is desired to increase the discharge to a maximum by changing the channel slope or the form of section. The dimensions of the section may be changed but the channel must contain the same amount of lining. Compute the new dimensions and probable increase in discharge.*

Solution

According to Manning's formula the discharge through the channel with original dimensions is

$$Q_1 = \frac{1}{n} AR^{2/3} S^{1/2}$$

$$A = (5.4 \times 1.2) = 6.48 \text{ m}^2$$

$$P = (5.4 + 2 \times 1.2) = 7.8 \text{ m}$$

$$R = \frac{A}{P} = \frac{6.48}{7.8} = 0.83$$

$$S = \frac{1}{1000}$$

Thus by substitution

$$Q_1 = \frac{1}{0.017} \times 6.48 \times (0.83)^{2/3} \left(\frac{1}{1000}\right)^{1/2}$$

$$= 10.65 \text{ m}^3/\text{s}$$

Let B and y be the bottom width and the depth of flow respectively for the new section of the channel. In order to have the same amount of lining as in the previous case,

$$P = (B + 2y) = 7.8 \qquad \text{...(i)}$$

Further the condition for the discharge to be maximum is

$$B = 2y \qquad \text{...(ii)}$$

Solving Eq. (i) and (ii), we have

$$B = 3.9 \text{ m}; y = 1.95 \text{ m}$$

The area of cross-section of the new channel section becomes

$$A = (3.90 \times 1.95) = 7.605 \text{ m}^2$$

$$R = \frac{A}{P} = \frac{7.605}{7.8} = 0.975 \text{ m}$$

Again using Manning's formula the discharge through the new section is

$$Q_2 = \frac{1}{0.017} \times 7.605 \times (0.975)^{2/3} \left(\frac{1}{1000}\right)^{1/2}$$

$$= 13.91 \text{ m}^3/\text{s}$$

The probable increase in the discharge is

$$Q_2 - Q_1 = (13.91 - 10.65) = 3.26 \text{ m}^3/\text{s}$$

and percentage increase in the discharge

$$= \frac{Q_2 - Q_1}{Q_1} \times 100 = \frac{3.26}{10.65} \times 100 = 30.61\%$$

If the original section of the channel is retained then the same increase in the discharge would be obtained if the channel slope is changed to 1 in 586.

Example 15.16. *A trapezoidal channel having bottom width 6 m and side slopes of 2 horizontal to 1 vertical is laid on a bottom slope of 0.0016. If it carries a uniform flow of water at the rate of 10 m³/s, compute the normal depth and the mean velocity of flow. Take Manning's n as 0.025.*

Solution

Let y be the normal depth of flow.

Then

$$A = (B + zy)\, y = (6 + 2y)y$$

and

$$P = \left(B + 2y\sqrt{1+z^2}\right) = 6 + 2y\sqrt{5}$$

$$R = \frac{A}{P} = \frac{(6+2y)y}{6+2y\sqrt{5}} = \frac{(3+y)y}{3+y\sqrt{5}}$$

Using Manning's formula, we have

$$\frac{10}{(6+2y)y} = \frac{1}{0.025} \times \left[\frac{(3+y)y}{3+y\sqrt{5}}\right]^{2/3} \times (0.0016)^{1/2}$$

or

$$16.57 + 12.35\, y = [(3+y)y]^{5/2}$$

Solving this equation for y by trial, normal depth $y = 0.963$ m

Alternative Method

From Eq. 15.32, conveyance

$$K = \frac{Q}{\sqrt{S}} = \frac{10}{\sqrt{0.0016}} = 250$$

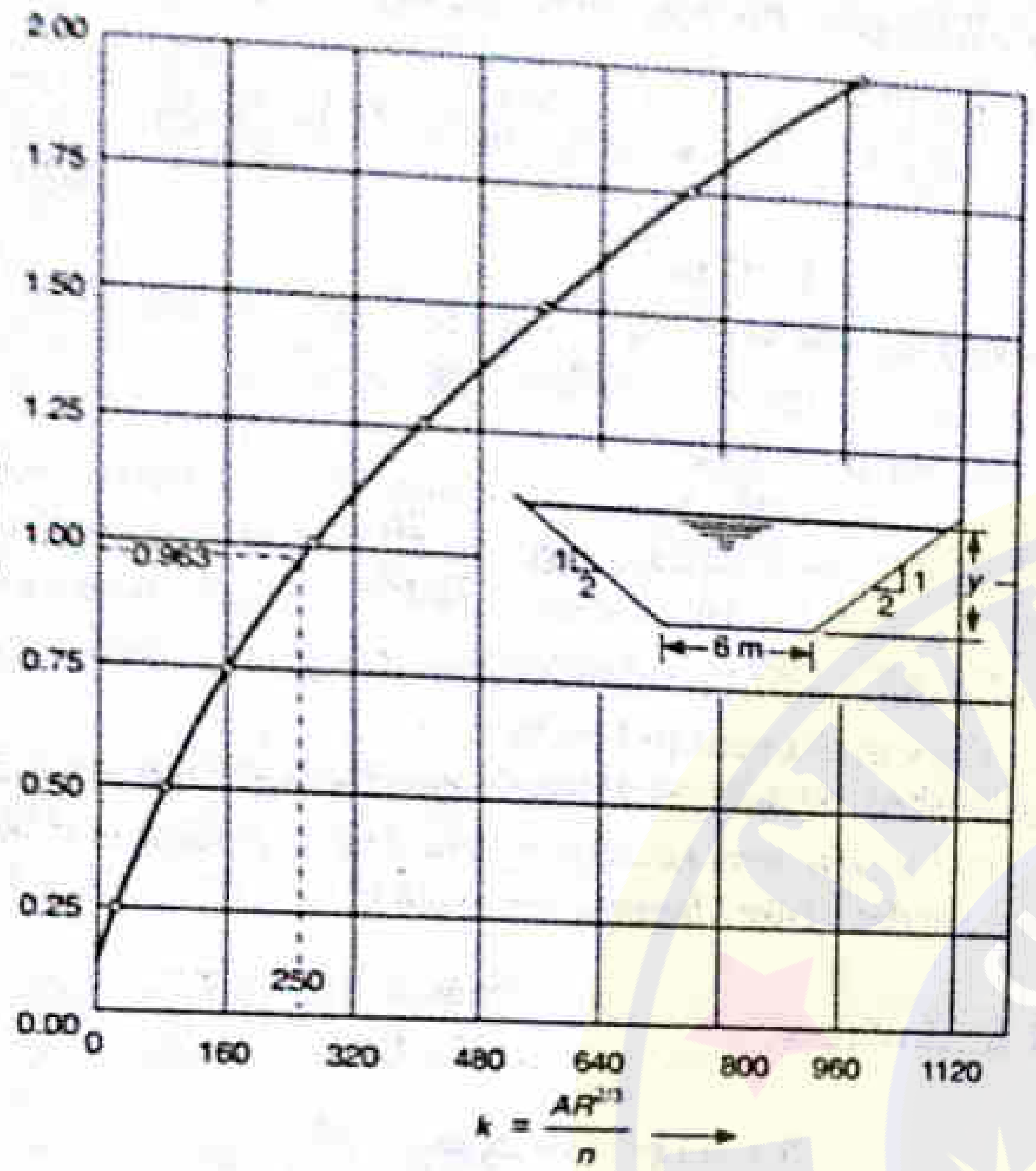


Figure Ex. 15.16

From Eq. 15.31, we have

$$K = \frac{1}{n}AR^{2/3} = \frac{[(6+2y)y]^{5/3}}{0.025(6+2y\sqrt{5})^{2/3}} \quad \text{...(i)}$$

For different values of y the values of conveyance K are computed from Eq. (i) and the same are tabulated below.

y	0	0.25	0.50	0.75	1.00	1.25	1.50	1.75	2.00
K	0	24.28	79.13	160.30	267.42	401.07	562.20	751.96	971.54

A plot of K v/s y is as shown in Fig. Ex. 15.13 from which for $K = 250$, $y = 0.963$ m which is the required normal depth.

The corresponding area $A = 7.62$ m^2, and the mean velocity

$$V = \frac{10}{7.62} = 1.31 \text{ m/s}$$

Example 15.17. *A circular sewer 0.6 m inside diameter has a slope of 1 in 400. Find the depth when the discharge is 0.283 m³/s, taking C in Chezy's formula as 50.*

Solution

Refer Fig. 15.7; $r = 0.3$ m

$$A = \frac{r^2}{2}(\theta - \sin\theta) = \frac{0.09}{2}(\theta - \sin\theta)$$

$$P = r\theta = 0.3\,\theta$$

$$R = \frac{A}{P} = \frac{r(\theta - \sin\theta)}{2\theta} = \frac{0.3(\theta - \sin\theta)}{2\theta}$$

Using Chezy's formula

$$Q = AC\sqrt{RS}$$

$$Q = 0.283 \text{ m}^3/\text{s}; C = 50$$

Thus by substitution, we get

$$0.283 = \frac{0.09}{2}(\theta - \sin\theta)\, 50 \left[\frac{0.3(\theta - \sin\theta)}{2\theta} \times \frac{1}{400}\right]^{1/2}$$

or $$6.495 = \frac{(\theta - \sin\theta)^{3/2}}{\theta^{1/2}}$$

or $$42.19\,\theta = (\theta - \sin\theta)^3$$

Solving the above equation by trial, we get

$$\theta = 286^\circ \text{ (approximately)}$$

Thus depth of flow is

$$y = r + r\cos\left(180^\circ - \frac{\theta}{2}\right)$$

$$= 0.3\left[1 + \cos\left(180^\circ - \frac{286^\circ}{2}\right)\right]$$

$$= 0.3\,[1 + \cos 37^\circ]$$

$$= 0.54 \text{ m}$$

Example 15.18. *A channel has vertical walls 1.2 m apart and a semicircular invert. If the centreline depth is 0.9 m and the bed slope is 1 in 2500, find the discharge using Chezy's formula with C = 54.*

Solution

As shown in the Fig. Ex. 15.15 area of flow section is

$$A = \left[\frac{\pi \times (0.6)^2}{2} + (1.2 \times 0.3)\right] = 0.925 \text{ m}^2$$

Wetted perimeter $$P = [\pi \times 0.6 + 2 \times 0.3]$$

Thus,

$$= 2.485 \text{ m}$$

$$R = \frac{A}{P} = \frac{0.925}{2.485} = 0.372 \text{ m}$$

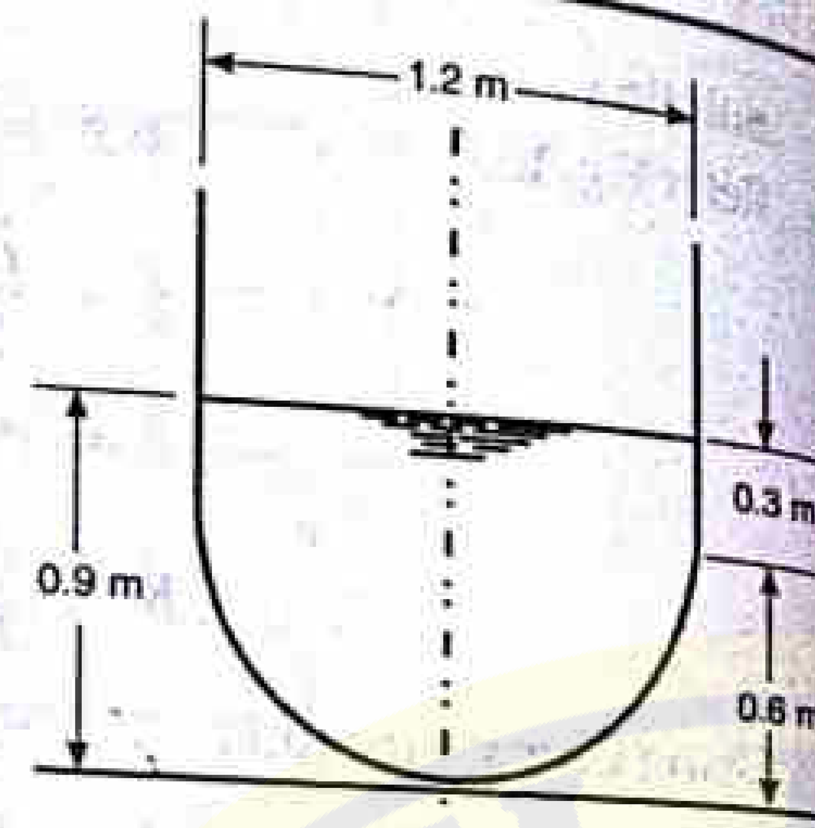


Figure Ex. 15.18

Using Chezy's formula

$$Q = AC\sqrt{RS}$$

$$S = \frac{1}{2500}; C = 54$$

Thus by substitution, we get

$$Q = 0.925 \times 54 \times \sqrt{0.372 \times \frac{1}{2500}}$$

$$= 0.609 \text{ m}^3/\text{s}$$

Example 15.19. *A channel is to be designed to give a constant mean velocity of flow of 1.8 m/s at all depths of flow. The lower portion of the channel to carry the minimum discharge is rectangular and has the best proportion, the bottom width being 1.5 m. Determine the depth of flow when the width of the water surface is 9 m.*

If Manning's n = 0.015, find the channel bed slope.

Solution

The bottom portion is rectangular and has the best proportion, thus

$$B = 2y$$

or

$$1.5 = 2y\,; y = 0.75 \text{ m}$$

Also for the best channel section

$$R = \frac{y}{2} = \frac{0.75}{2} = 0.375 \text{ m}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

$$V = 1.8 \text{ m/s}\,; n = 0.015$$

Thus by substitution, we have

$$1.8 = \frac{1}{0.015} \times (0.375)^{2/3} S^{1/2}$$

$\therefore$

$$S = 0.0027\,; \text{or } 1 \text{ in } 370$$

From Eq. 15.27, we have

$$y = R \log_e \left[x + \sqrt{x^2 - R^2}\right] + C$$

For

$$x = \frac{1.5}{2} = 0.75 \text{ m}; y = 0$$

$\therefore$

$$C = -R \log_e \left[0.75 + \sqrt{(0.75)^2 - R^2}\right]$$

Thus

$$y = R \log_e \left[\frac{x + \sqrt{x^2 - R^2}}{0.75 + \sqrt{(0.75)^2 - R^2}}\right]$$

For

$$x = \frac{9}{2} = 4.5 \text{ m}; R = 0.375 \text{ m}$$

Thus by substitution, we get

$$y = (0.375) \log_e \left[\frac{4.5 + \sqrt{(4.5)^2 - (0.375)^2}}{0.75 + \sqrt{(0.75)^2 - (0.375)^2}}\right]$$

or

$$y = 0.375 \log_e (6.42) = 0.697 \text{ m}$$

$\therefore$ Total depth of flow

$$= (0.75 + 0.697) = 1.447 \text{ m}$$

Example 15.20. *A rectangular channel which is laid on a bottom slope of 0.0064 is to carry 20 m³/s of water. Determine the width of the channel when the flow is in critical condition. Take Manning's n = 0.015.*

Solution

Let the width of the channel be B.

Then

$$q = \frac{Q}{B} = \frac{20}{B}$$

The critical depth for a rectangular channel section is expressed by Eq. 15.46 as

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left(\frac{400}{B^2 \times 9.81}\right)^{1/3} = \frac{3.442}{B^{2/3}}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

or

$$\frac{20}{By_c} = \frac{1}{0.015} \left(\frac{By_c}{B + 2y_c}\right)^{2/3} (0.0064)^{1/2}$$

or

$$\frac{20}{B^{1/3}(3.442)} = \frac{0.08}{0.015} \left[\frac{3.442 B^{1/3}}{B + (6.884/B^{2/3})}\right]^{2/3}$$

or $$\left(\frac{3.75}{3.442}\right)^{2/3} \times \frac{1}{B^{1/2}} = \frac{3.442B}{B^{5/3}+6.884}$$

or $$0.33\,(B^{5/3} + 6.884) = B^{3/2}$$

Solving this equation for B by trial, we obtain

$$B = 2.41 \text{ m}$$

Example 15.21. *For a constant specific energy of 1.8 N.m /N, calculate the maximum discharge that may occur in a rectangular channel 5.0 m wide.*

Solution

For a given specific energy the discharge is maximum when the flow is in critical state.

Thus the depth of flow $y = y_c$

But from Eq. 15.49

$$y = y_c = \frac{2}{3}E = \frac{2}{3} \times 1.8 = 1.2 \text{ m}$$

Further for a rectangular channel from Eq. 15.46

$$y_c = \left(\frac{q^2}{g}\right)^{1/3}$$

or $$1.2 = \left(\frac{q^2}{g}\right)^{1/3}$$

or $$q = [(1.2)^3 \times 9.81]^{1/2}$$

or $$q = 4.12 \text{ m}^3/\text{s per m}$$

∴ Total discharge

$$Q = (4.12 \times 5) = 20.6 \text{ m}^3/\text{s}$$

Example 15.22. *A trapezoidal channel has a bottom width of 6 m and side slopes of 2 horizontal to 1 vertical. If the depth of flow is 1.2 m at a discharge of 10 m³/s, compute the specific energy and the critical depth.*

Solution

$$A = (B + zy)\,y = [(6 + 2 \times 1.2)1.2]$$

$$= 10.08 \text{ m}^2$$

Specific energy is given by

$$E = y + \frac{1}{2g}\left(\frac{Q}{A}\right)^2$$

$$= 1.2 + \frac{1}{2 \times 9.81}\left(\frac{10}{10.08}\right)^2$$

$$= 1.25 \text{ m}$$

For critical flow

$$\frac{Q^2}{g} = \frac{A^3}{T}$$

or $$\frac{(10)^2}{9.81} = \frac{[(6+2y_c)y_c]^3}{6+4y_c}$$

Solving the above Equation for y_c by trial, we get

$$y_c = 0.611 \text{ m}$$

Alternative Method

Section factor for critical flow computation is given by

$$Z = \left(\frac{A^3}{T}\right)^{1/2} = \frac{[(6+2y)y]^{3/2}}{(6+4y)^{1/2}} \quad \text{...(i)}$$

For different values of y the values of Z are computed from Eq. (i) and the same are tabulated below.

y	0	0.25	0.50	0.75	1.00	1.25	1.50
Z	0	0.78	2.32	4.45	7.16	10.44	14.32

A plot of Z v/s y is made as shown in the Fig. Ex. 15.22

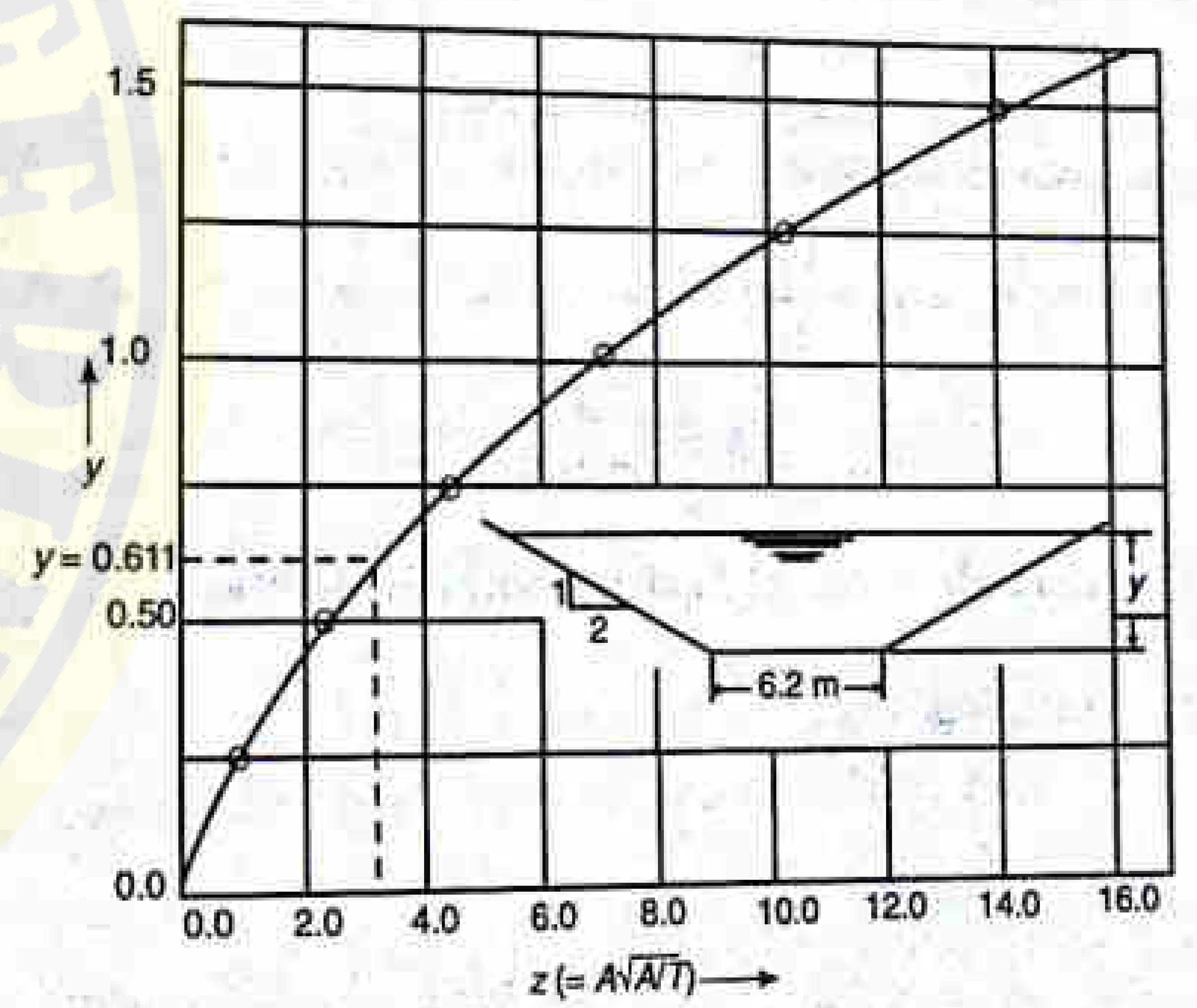


Figure Ex. 15.22

For critical flow from Eq. 15.44, we have

$$\left(\frac{A^3}{T}\right)^{1/2} = Z_c = \frac{Q}{\sqrt{g}} = \frac{10}{\sqrt{9.81}} = 3.19$$

Thus from the plot of Z v/s y, for Z = 3.19, y = 0.611 m which is the required critical depth.

Example 15.23. *In a rectangular channel 3.5 m wide laid at a slope of 0.0036, uniform flow occurs at a depth of 2 m. Find how high can the hump be raised without causing afflux? If the upstream depth of flow is to be raised to 2.5 m, what should be the height of the hump? Take Manning's n equal to 0.015.*

Solution

The area of flow section at the upstream section is

$$A = (3.5 \times 2) = 7.0 \text{ m}^2$$

The wetted perimeter is

$$P = 3.5 + (2 \times 2) = 7.5 \text{ m}$$

Thus $$R = \frac{A}{P} = \frac{7.0}{7.5} = 0.93 \text{ m}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

$$= \frac{1}{0.015} \times (0.93)^{2/3} (0.0036)^{1/2}$$

$$= 3.81 \text{ m/s}$$

Discharge $$Q = (7.0 \times 3.81) = 26.67 \text{ m}^3/\text{s}$$

Specific energy at upstream section is

$$E_1 = y_1 + \frac{V_1^2}{2g}$$

$$= 2.0 + \frac{(3.81)^2}{2 \times 9.81} = 2.74 \text{ m}$$

For the rectangular channels the critical depth is given by Eq. 14.39 as

$$y_c = \left(\frac{q^2}{g}\right)^{1/3}$$

$$q = \frac{Q}{B}$$

$$= \frac{26.67}{3.5} = 7.62 \text{ m}^3/\text{s per metre}$$

Thus $$y_c = \left[\frac{(7.62)^2}{9.81}\right]^{1/3} = 1.81 \text{ m}$$

The corresponding minimum specific energy is

$$E_c = \frac{3}{2} y_c = \frac{3}{2} \times 1.81 = 2.715 \text{ m}$$

From Eq. 15.57 the maximum height of the hump without causing afflux is obtained as

$$\Delta Z_{max} = (E_1 - E_c) = (2.74 - 2.715) = 0.025 \text{ m}$$

If the upstream depth of flow is raised to 2.5 m, the area of flow section becomes

$$A = (3.5 \times 2.5) = 8.75 \text{ m}^2$$

Then for the same discharge the velocity of flow will be

$$V = \frac{26.67}{8.75} = 3.05 \text{ m/s}$$

Specific energy at the upstream section will be

$$E_1 = 2.5 + \frac{(3.05)^2}{2 \times 9.81} = 2.97 \text{ m}$$

$\therefore$ The required height of the hump is

$$\Delta Z = (E_1 - E_c) = (2.97 - 2.715) = 0.255 \text{ m}$$

Example 15.24. *For purpose of discharge measurement the width of a rectangular channel is reduced gradually from 3 m to 2 m and the floor is raised by 0.3 m at a given section. When the approaching depth of flow is 2 m, what rate of flow will be indicated by a drop of 0.15 m in the water surface elevation at the contracted section?*

Solution

From the given data the depth of flow at the contracted section is

$$y = (2.0 - 0.30 - 0.15) = 1.55 \text{ m}$$

If Q is the discharge flowing through the channel then specific energy at the upstream section is

$$E_1 = 2.0 + \frac{Q^2}{2g(3 \times 2)^2} = 2.0 + \frac{Q^2}{72 \times 9.81}$$

Also the specific energy at the contracted section is

$$E_2 = 1.55 + \frac{Q^2}{2g(2 \times 1.55)^2} = 1.55 + \frac{Q^2}{19.22 \times 9.81}$$

Assuming no loss of energy between two sections, we have

$$E_1 = E_2 + \Delta Z$$

Thus $$2.0 + \frac{Q^2}{72 \times 9.81} = 1.55 + \frac{Q^2}{19.22 \times 9.81} + 0.30$$

or $$Q = 6.21 \text{ m}^3/\text{s}$$

Example 15.25. *A trapezoidal channel of base width 6 m and side slopes of 2 horizontal to 1 vertical carries a flow of 60 cumec at a depth of 2.5 m. There is a smooth transition to a rectangular section 6 m wide. (i) If the channel bed is horizontal determine whether the upstream flow is possible as specified; if not determine the upstream depth of flow. (ii) Determine the minimum amount by which the bed must be lowered for the upstream flow to be possible as specified. (iii) Determine the depth of flow in the rectangular section and the change in the water surface level if the transition is accompanied by a gradual lowering of the channel bed by 0.6 m. (iv) Determine the amount by which the bed must be lowered if the drop in water surface is to be restricted to 0.3 m. Assume no losses.*

Solution

At the trapezoidal section

$$A_1 = (6 + 2 \times 2.5)\,2.5 = 27.5\ \text{m}^2$$

$$V_1 = \frac{60}{27.5} = 2.182\ \text{m/s}$$

and

$$E_1 = 2.5 + \frac{(2.182)^2}{2 \times 9.81} = 2.743\ \text{m}$$

(i) The upstream flow as specified would be possible if

$$E_1 \le E_c$$

where E_c is specific energy at critical flow condition at the rectangular section.

$$E_c = \frac{3}{2} y_c$$

and

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(60/6)^2}{9.81}\right]^{1/3} = 2.168\ \text{m}$$

$\therefore$

$$E_c = \frac{3}{2}(2.168) = 3.252\ \text{m}$$

Since $E_1 < E_c$, the upstream flow as specified would not be possible.

Assuming the upstream flow to be subcritical there will be increase in the depth of flow at which the flow will take place.

Thus if y is the new depth of flow in the trapezoidal section then, we have

$$A_1' = (6 + 2y)y = 2(3 + y)y$$

$$V_1' = \frac{60}{2(3+y)y} = \frac{30}{(3+y)y}$$

and

$$E_1' = y + \frac{1}{2g}\left[\frac{30}{(3+y)y}\right]^2$$

For no loss of energy between the two sections, we have

$$E_1' = E_c$$

$$y + \frac{1}{2g}\left[\frac{30}{(3+y)y}\right]^2 = 3.252$$

$$y + \frac{900}{2 \times 9.81} \times \frac{1}{(3+y)^2 y^2} = 3.252$$

$$y + \frac{450}{9.81} \times \frac{1}{(3+y)^2 y^2} = 3.252$$

Solving for y by trial and error, we get

$$y = 3.127\ \text{m}$$

Thus in this case there will be heading up of water on the upstream side in the trapezoidal section with new depth of flow being 3.127 m.

(ii) Let x be the minimum amount by which the bed must be lowered for the upstream flow to be possible as specified, i.e., the depth of flow in the trapezoidal section to be 2.5 m as given.

Thus in this case, we have

$$E_1 + x = E_c$$

or

$$2.743 + x = 3.252$$

$\therefore$

$$x = 0.509\ \text{m}$$

(iii) Let y be the depth of flow at the rectangular section, then

$$A_2 = 6 \times y$$

$$V_2 = \frac{60}{6 \times y} = \frac{10}{y}$$

and

$$E_2 = y + \frac{(10/y)^2}{2 \times 9.81} = \left(y + \frac{5.097}{y^2}\right)$$

For no loss of energy between the two sections, we have

$$E_1 = 0.6 = E_2$$

or

$$2.743 + 0.6 = y + \frac{5.097}{y^2}$$

or

$$y^3 - 3.343\,y^2 + 5.097 = 0$$

Solving for y by trial and error, we get

$$y = 2.573\ \text{m};\ \text{or}\ y = 1.844\ \text{m}$$

These are the alternate depths out of which one being greater than the critical depth will be the depth of subcritical flow, and the other one being less than the critical depth will be the depth of supercritical flow.

Thus if subcritical flow occurs, then

Change in water surface level

$$= (2.5 + 0.6) - 2.573 = 0.527\ \text{m}$$

However, if supercritical flow occurs, then

Change in water surface level

$$= (2.5 + 0.6) - 1.844 = 1.256\ \text{m}$$

(iv) Let the channel bed be required to be lowered by an amount x when the drop in water surface level is to be restricted to 0.3 m, and the depth of flow at the rectangular section be y, then

$$x + 2.5 = y + 0.3$$

or

$$y - x = 2.2$$

Again for no loss of energy between the two sections, we have

$$E_1 + x = E_2$$

or $$2.743 + x = y + \frac{5.097}{y^2} \quad ...(ii)$$

Adding Eq. (i) and (ii), we get

$$2.743 = \frac{5.097}{y^2} + 2.2$$

or $$y = 3.064 \text{ m}$$

$\therefore$ $$x = 0.864 \text{ m}$$

i.e., the channel bed is required to be lowered by 0.864 m.

Again with a lowering of channel bed by 0.864 m, there will be two possible depths of flow at the rectangular section as indicated below.

If y is the depth of flow at the rectangular section, then

$$E_2 = y + \frac{5.097}{y^2}$$

For no loss of energy between the two sections, we have

$$E_1 + 0.864 = E_2$$

or $$2.743 + 0.864 = y + \frac{5.097}{y^2}$$

or $$y^3 - 3.607\, y^2 + 5.097 = 0$$

Solving for y by trial and error, we get

$$y = 3.064 \text{ m; or } y = 1.589 \text{ m}$$

These are the alternate depths out of which one being greater than the critical depth will be the depth of subcritical flow, and the other one being less than the critical depth will be the depth of supercritical flow.

Thus if subcritical flow occurs, then

Change in water surface level

$$= (2.5 + 0.864) - 3.064 = 0.3 \text{ m}$$

However, if supercritical flow occurs, then

Change in water surface level

$$(2.5 + 0.864) - 1.589 = 1.775 \text{ m}$$

Example 15.26. *A trapezoidal channel has a bed width 10 m and side slopes of 2 horizontal to 1 vertical. At a section 200 m downstream the bed rises by 0.08 m gradually, bed width increases to 0.15 m and side slopes become 3 horizontal to 1 vertical. Rise in bed level, increase in bed width and flattening of side slopes are gradual. Flow rate in the channel is 200 m³/s and the flow depth at the upstream section is 7 m. If Manning's n = 0.035 estimate the depth at the downstream section.*

Solution

At the upstream section

$$A_1 = (10 + 2 \times 7)\, 7 = 168 \text{ m}^2$$

$$P_1 = 10 + 2\sqrt{5} \times 7 = 41.305 \text{ m}$$

$$R_1 = \frac{168}{41.305} = 4.067 \text{ m}$$

$$V_1 = \frac{200}{168}\, 1.19 \text{ m/s}$$

If E_1 and E_2 are the specific energies at the upstream and the downstream sections and h_L is the head loss between these sections, then

$$E_1 = E_2 + 0.08 + h_L \quad ...(i)$$

$$E_1 = 7 + \frac{(1.19)^2}{2 \times 9.81} = 7.072 \text{ m}$$

Let y be the depth of flow at the downstream section.

Thus $$A_2 = (15 + 3y)\, y$$

$$P_2 = 15 + 2\sqrt{10}\, y$$

$$V_2 = \frac{200}{(15 + 3y)y}$$

$\therefore$ $$E_2 = y + \frac{(200)^2}{(15 + 3y)^2 y^2 \times 2 \times 9.81}$$

Further if S_{f1} is the slope of the energy line at the upstream section, then

$$S_{f1} = \left(\frac{V_1 n}{R_1^{2/3}}\right)^2$$

$$= \left[\frac{1.19 \times 0.035}{(4.067)^{2/3}}\right]^2 = 2.67 \times 10^{-4}$$

Assuming the energy line slope to be the same at both the sections, we have

$$\frac{h_L}{200} = 2.67 \times 10^{-4}$$

$\therefore$ $$h_L = 0.0534 \text{ m}$$

By substituting in Eq. (i), we get

$$7.072 = y + \frac{(200)^2}{(15 + 3y)^2 y^2 \times 2 \times 9.81} + 0.08 + 0.0534$$

or $$6.9386 = y + \frac{2038.736}{(15 + 3y)^2 y^2}$$

Solving for y by trial and error, we get

$$y = 6.905 \text{ m}$$

Thus $$A_2 = (15 + 3 \times 6.905)\, 6.905 = 246.61 \text{ m}^2$$

$$P_2 = \left(15 + 2\sqrt{10} \times 6.905\right) = 58.67 \text{ m}$$

$$R_2 = \frac{246.61}{58.67} = 4.203 \text{ m}$$

$$V_2 = \frac{200}{246.61} = 0.811 \text{ m/s}$$

If S_{f2} is the slope of the energy line at the downstream section, then

$$S_{f2} = \left(\frac{V_2 n}{R_2^{2/3}}\right)^2$$

$$= \left[\frac{0.811 \times 0.035}{(4.203)^{2/3}}\right]^2 = 1.19 \times 10^{-4}$$

Thus mean value of slope S_f of the energy line is

$$S_f = \frac{S_{f_1} + S_{f_2}}{2}$$

$$= \frac{(2.67 + 1.19)10^{-4}}{2}$$

$$= 1.93 \times 10^{-4}$$

Thus $$\frac{h_L}{200} = 1.93 \times 10^{-4}$$

$\therefore$ $$h_L = 0.0386 \text{ m}$$

Again by substituting in Eq. (i), we get

$$7.072 = y + \frac{(200)^2}{(15+3y)^2 y^2 \times 2 \times 9.81} + 0.08 + 0.0386$$

or $$6.953\,4 = y + \frac{2038.736}{(15+3y)^2 y^2}$$

Solving for y by trial and error, we get

$$y = 6.92 \text{ m}$$

Example 15.27 *Calculate the possible depths of flow at which a discharge of 26.67 cumec may be carried in a rectangular channel 3.5 m wide with a specific energy equal to 2.74 m.*

Solution

Let y be the depth of flow at which the given discharge may be carried with the given special energy in the given channel.

From Eq. 15.35, specific energy is given as

$$E = y + \frac{Q^2}{2gA^2}$$

or $$E = y + \frac{Q^2}{2g(By)^2}$$

By substituting the given values, we get

$$2.74 = y + \frac{(26.67)^2}{2 \times 9.81 \times (3.5 \times y)^2}$$

or $$y^3 - 2.74y^2 + 2.96 = 0$$

Solving for y by trial and error, we obtain the following three values of y, viz.,

$$y = 2.00 \text{ m}, y = 1.64 \text{ m and } y = -0.90 \text{ m}$$

The third value of y being negative has no practical significance and hence it may be ignored.

Therefore the two possible depths of flow for the given discharge and given specific energy in the given rectangular channel are 1.64 m and 2.00 m, which are known as alternate depths.

Note: It may be noted that for a given discharge carried in a given channel with a given specific energy, Eq. 15.35 reduces to a cubic equation for the depth of flow y. The solution of this cubic equation gives three values of y, out of which the two values of y are always positive and the third value is negative. It therefore indicates that for given discharge a plot of specific energy, E v/s depth of flow, y has three limbs, the two of which are as shown in the ordinary specific energy curve (Fig. 15.9), corresponding to the positive values of y. The third limb corresponds to the negative values of y which is normally not considered because it has no practical significance.

Example 15.28. *Design a concrete lined channel to carry a discharge of 500 m³/s at a slope of 1 in 4000. The side slopes of the channel may be taken as 1:1. The Manning's roughness coefficient for the lining, is 0.014. Assume permissible velocity in the section as 2.5 m/s.*

Solution

For a lined channel section as shown in Fig. 15.20, we have

$$A = By + y^2(\theta + \cot\theta)$$

and $$P = B + 2y(\theta + \cot\theta)$$

$$\theta = 45° = \frac{\pi}{4}$$

$$Q = 500 \text{ m}^3/\text{s};\ V = 2.5 \text{ m/s}$$

$\therefore$ $$A = \frac{Q}{V} = \frac{500}{2.5} = 200 \text{ m}^2$$

Thus $$200 = By + y^2\left(\frac{\pi}{4} + 1\right) \quad \text{...(i)}$$

From Manning's equation, we have

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

or $$2.5 = \frac{1}{0.014} \times R^{2/3} \times \left(\frac{1}{4000}\right)^{1/2}$$

$$\therefore \quad R = 3.293 \text{ m}$$

Also $$R = \frac{A}{P} = \frac{200}{P}$$

$$\therefore \quad P = \frac{200}{3.293} = 60.735 \text{ m}$$

Thus $$60.735 = B + 2y\left(\frac{\pi}{4}+1\right) \qquad \ldots(ii)$$

From Eq. (i) and (ii), we have

$$\left(\frac{\pi}{4}+1\right)y^2 - 60.735\,y + 200 = 0$$

or $$y^2 - 34.018\,y + 112.02 = 0$$

$$\therefore \quad y = 3.694 \text{ m}$$

From Eq. (ii), we have

$$B = 60.735 - 2y\left(\frac{\pi}{4}+1\right)$$

or $$B = 60.735 - 2 \times 3.694\left(\frac{\pi}{4}+1\right)$$

$$\therefore \quad B = 47.544 \text{ m}$$

Note. In this case if a trapezoidal channel section with sharp corners is designed then a very wide and shallow section is obtained which is not a practical section.

Example 15.29. *Water flows at a velocity of 1 m/s and a depth of 2 m in an open channel of rectangular cross section 3 m wide. At a certain section the width is reduced to 1.8 m and the bed is raised by 0.65 m. Will the upstream depth be affected? If so, to what extent?*

Solution

The area of flow section at the upstream section is

$$A = (3 \times 2) = 6 \text{ m}^2$$

and velocity of flow is

$$V = 1 \text{ m/s}$$

$\therefore$ Discharge $$Q = (6 \times 1) = 6 \text{ m}^3/\text{s}$$

Specific energy at the upstream section is

$$E_1 = y_1 + \frac{V_1^2}{2g}$$

$$= 2.0 + \frac{(1)^2}{2 \times 9.81} = 2.051 \text{ m}$$

The upstream depth will not be affected if

$$E_1 \geq E_c + 0.65$$

where E_c is specific energy at critical flow condition at the contracted section.

$$E_c = \frac{3}{2}y_c$$

and $$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(6/1.8)^2}{9.81}\right]^{1/3} = 1.042 \text{ m}$$

$$\therefore \quad E_c = \frac{3}{2} \times 1.042 = 1.563 \text{ m}$$

and $$E_c + 0.65 = (1.563 + 0.65) = 2.213 \text{ m}$$

Since $E_1 < E_c + 0.65$, the upstream depth will be affected and there will be heading up of water at the upstream section to such an extent that the flow at the contracted section will be in a critical state.

Let y be the new depth of flow at the upstream section, then

$$V = \frac{6}{3 \times y} = \frac{2}{y}$$

and $$E_1 = y + \frac{(2/y)^2}{2 \times 9.81} = y + \frac{0.204}{y^2}$$

Specific energy at the contracted section is

$$E_2 = E_c + 0.65$$

$$= (1.563 + 0.65) = 2.213 \text{ m}$$

For no loss of energy between the two sections, we have

$$E_1 = E_2$$

or $$y + \frac{0.204}{y^2} = 2.213$$

Solving for y by trial and error, we get

$$y = 2.17 \text{ m}$$

i.e., the depth of flow at the upstream section will increase by 0.17 m.

Example 15.30. *An unlined irrigation canal has its bed and side composed of cohesionless material having mean diameter 6 mm. Angle of repose of the material is 40°. The bed width of the canal is 5 m and the side slope 1.5(H) : 1(V). Determine the minimum discharge that can be admitted into the canal without any sediment movement. Longitudinal slope of the canal is 1 in 5000, Manning's n = 0.025 and specific gravity of the material s = 2.65.*

Solution

Angle of repose $$\phi = 40° ; \tan 40° = 0.8391$$

Side slope $= 1.5(H):1(V)$

$$\therefore \quad \tan\theta = \frac{1}{1.5} = 0.6667\ ;\ \cos\theta = \frac{3}{\sqrt{13}} = 0.8321$$

The ratio of unit tractive force τ_s on the side of the channel and unit tractive force τ_L on the level surface is given by

$$K = \frac{\tau_s}{\tau_L} = \cos\theta\sqrt{1-\frac{\tan^2\theta}{\tan^2\phi}}$$

Thus by substitution we get

$$K = 0.8321\sqrt{1-\frac{(0.6667)^2}{(0.8391)^2}} = 0.5053$$

$$\therefore \quad \tau_s = 0.5053\tau_L$$

According to Shield's formula critical unit tractive force τ_{cr} for just no movement of the sediment is given as

$$\tau_{cr} = 0.06\,w(s-1)d$$

$$= 0.06 \times 9.81(2.65-1) \times \frac{6}{1000}$$

$$= 0.0058\ \text{kN/m}^2$$

$$\therefore \quad \tau_L = \tau_{cr} = 0.0058\ \text{kN/m}^2$$

$$\therefore \quad \tau_s = 0.5053 \times 0.0058\ \text{kN/m}^2$$

But

$$\tau_s = 0.75\,wRS_0 = 0.75 \times 9.81 \times R \times \frac{1}{5000}$$

$$\therefore \quad 0.75 \times 9.81 \times R \times \frac{1}{5000} = 0.5053 \times 0.0058$$

or

$$R = 2\ \text{m}$$

$$A = (5+1.5y)\,y$$

$$P = 5+\sqrt{13}y$$

$$\therefore \quad 2 = \frac{(5+1.5y)y}{5+\sqrt{13}y}$$

$$1.5y^2 - 2.21y + 10 = 0$$

$$y = 3.422\ \text{m}$$

$$Q = \frac{1}{n}AR^{2/3}S_0^{1/2}$$

$$= \frac{1}{0.025}(5+1.5\times 3.422)\,3.422 \times (2)^{2/3} \times \frac{1}{5000}$$

$$= 0.44\ \text{m}^3/\text{s}$$

SUMMARY OF MAIN POINTS

1. A channel is a flow passage through which water flows under atmospheric pressure, and hence in channels the flow of water takes place with a free surface which is subjected to atmospheric pressure.
2. In order to overcome the resistance and to cause the flow of water in a channel, it is constructed with its bottom slopping towards the direction of flow.
3. A channel may be classified as natural channel and artifical channel. A *natural channel* is the one which has irregular sections of varying shapes, which is developed in a natural way. The examples of natural channels are rivers, streams, etc. An *artificial channel* is the one which is built artificially for carrying water for various purposes. The artificial channels may be further classified according to the shape of the cross-section as *rectangular channel, trapezoidal channel, triangular channel, parabolic channel* and *circular channel*.

   A channel having the same shape of various sections along its length and laid on a constant bottom slope is known as *prismatic channel*, otherwise the channel is *non-prismatic*.
4. The flow is channels may be classified as *uniform flow* and *non-uniform (or varied)* flow.

   The flow in a channel is said to be uniform if the depth, slope, cross-section and velocity remain constant over a given length of the channel. Thus in a prismatic channel the flow will be uniform if only the depth of flow y is same at every section of the channel, that is $(\partial y/\partial w) = 0$.

   Flow in channels is termed as non-uniform or varied if the depth of flow y, changes from section to section along the length of the channel, that is $(\partial y/\partial s)$ is not equal to zero.

   Varied flow may be further classified as rapidly varied flow (R.V.F.) and gradually varied flow (G.V.F.). If the depth of flow changes abruptly over a comparatively short distance, the flow is characterized as a rapidly varied flow. In a gradually varied flow the change in the depth of flow takes place gradually in a long reach of channel.
5. The flow in channels may also be characterized as the laminar, turbulent or in a transitional state on the basis of Reynolds number Re which in the case of channels is defined as

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

where
ρ = mass density of water;
V = mean velocity of flow;
R = hydraulic radius or hydraulic mean depth of the channel section $\left(=\frac{A}{P}\right)$; and
μ = absolute viscosity of water.

The flow in channels may be considered to laminar if Re is less than 500 and for Re greater than 2000 the flow in channels is turbulent. For Re in between 500 to 2000 the flow in channels may be considered to be in a transitional state.

6. The mean velocity of flow in channels is given by Chezy's formula as

$$V = C\sqrt{RS} \quad \text{or } V = C\sqrt{mi}$$

where
V = mean velocity of flow;
R or m = hydraulic radius or hydraulic mean depth of the channels section $\left(=\frac{A}{P}\right)$;
S or i = slope of the channel bottom; and
C = Chezy's coefficient.

7. The value of Chezy's coefficient is given by empirical formula as

(i) Ganguillet–Kutter's formula

$$C = \frac{23+\frac{0.00155}{S}+\frac{1}{n}}{1+\left(23+\frac{0.00155}{S}\right)\frac{n}{\sqrt{R}}}$$

where
n = Kutter's roughness coefficient.

(ii) Bazin's formula

$$C = \frac{157.6}{1.81+\frac{m}{\sqrt{R}}}$$

where
m = Bazin's roughness coefficient

8. The mean velocity of flow in channels is given by Manning's formula as

$$V = \frac{1}{n}R^{2/3}S^{1/2}$$

where n = Manning's roughness coefficient.

The values of Manning's n and Kutter's n are approximately the same.

A comparison of Manning's formula with Chezy's formula indicates that

$$C = \frac{1}{n}R^{1/6}$$

9. A channel section is considered to be the most economical or most efficient when it can pass maximum discharge for given cross-sectional area, resistance coefficient and bottom slope.

The conditions for the most economical or most efficient channel section are

(i) For rectangular channel section

$$B = 2y$$

and

$$R = \frac{y}{2}$$

where B = Bed width of the channel; and
y = depth of flow.

(ii) For trapezoidal channel section

(a) When side slope z is constant

$$\frac{B+2zy}{2} = y\sqrt{1+z^2}$$

and

$$R = \frac{y}{2}$$

Also a semi-circle drawn from the mid-point of the top width as centre and radius equal to the depth of flow will touch the three sides of a most economical or most efficient trapezoidal channel section, viz., the bottom and the two sloping sides tangentially

(b) When the bottom width B is constant

$$\frac{B}{y} = \frac{1-z^2}{z}$$

(c) When the depth of flow is constant

$$z = \frac{1}{\sqrt{3}}; \text{ or } \theta = 60°$$

where

θ = angle made by the sloping size with the horizontal

(iii) For triangular channel section

$$\theta = 45°; \quad \text{or } z = 1$$

and

$$R = \frac{y}{2\sqrt{2}}$$

where

θ = angle of inclination of each of the sloping sides with the vertical.

(iv) For circular channel section, area cannot be maintained constant and hence there are two different conditions, one for maximum velocity and the other for maximum discharge.

(a) For maximum velocity through a circular channel section

$$y = 0.81\,D$$

and

$$R = 0.30\,D$$

(b) For maximum discharge through a circular channel section

$$y = 0.95D$$

and

$$R = 0.29D$$

where

D = diameter of the circular channel section.

For a circular channel section

Area of flow section $A = \frac{r^2}{2}(\theta - \sin\theta)$; and

Wetted perimetes $P = r\theta$

where

r = radius of channel section; and

θ = angle subtended at the centre by the portion of the circular arc in contact with water.

10. The discharge Q of uniform flow in a channel may be expressed as

$$Q = K\sqrt{S}$$

where

K = conveyance of the channel section.

The value of K is given as

(i) When Chezy's formula is used

$$K = CA\sqrt{R}$$

(ii) When Manning's formula is used

$$K = \frac{1}{n}AR^{2/3}$$

11. In channels total energy or head per unit weight of flowing water at any channel section is given by

$$\frac{V^2}{2g} + y + Z$$

12. The specific energy of flow at any channel section is defined as the energy per unit weight of flowing water measured with respect to the channel bottom as the datum. Thus specific energy E at any channel section is given by

$$E = y + \frac{V^2}{2g} = y + \frac{Q^2}{2gA^2}$$

For a given channel section and discharge Q, the specific energy may be plotted against the depth of flow to obtain specific energy curve. It may be seen from this curve that there are two possible depths for the same specific energy. These two depths for given specific energy are known as *alternate depths*.

13. The depth of flow at which the specific energy is minimum is called *critical depth* y_c.

14. The velocity of flow at critical depth is known as *critical velocity* V_c.

15. The flow in a channel with depth of flow greater than critical depth is known as *subcritical flow* or *tranquil flow*.

17. The flow in a channel with depth of flow less than critical depth is known as *supercritical flow* or *rapid flow*.

18. For critical flow Froude number $Fr = 1$; for subcritical flow $Fr < 1$; and for supercritical flow $Fr > 1$. The Froude number Fr is defined as

$$Fr = \frac{V}{\sqrt{gD}}$$

where

D = hydraulic depth $= \frac{A}{T}$; and

T = top width

19. For a rectangular channel the critical depth y_c is given as

(i) $$y_c = \left(\frac{q^2}{g}\right)^{1/3}$$

where

q = discharge per unit width of the channel section

(ii) $$y_c = \frac{2}{3}E$$

20. The specific force F is given by

$$F = \frac{Q^2}{gA} + A\bar{z}$$

where

$\bar{z}$ = vertical depth of centroid of the area of flow section below the free surface.

For a given channel section and discharge Q, the specific force may also be plotted against the depth of flow to obtain specific force curve. It may be seen from this curve that there are two possible depths for the same specific force. These two depths for the given specific force are known as *conjugate* depths.

21. For a given channel section and discharge, the specific force is also minimum at critical depth of flow.

22. The section factor for critical flow computation Z is given by

$$Z = \left(\frac{A^3}{T}\right)^{1/2} = A\sqrt{D}$$

PROBLEMS

15.1 What do you understand by (a) steady and unsteady flow; (b) uniform and non-uniform flow in the case of channels?

15.2 What is Chezy's formula? How is it derived?

Show that Chezy's coefficient $C = \left(\frac{R^{1/6}}{n}\right)$, where R is the hydraulic radius and n is Manning's roughness coefficient.

15.3 Explain the significance of channels of most efficient section.

15.4 Show that for a trapezoidal channel of given area of flow, the condition of maximum flow requires that hydraulic mean depth is equal to one half the depth of flow.

15.5 State the conditions under which the rectangular section of an open channel will be most economical. Derive these conditions.

15.6 Show that the discharge formula for a trapezoidal channel, having Manning's coefficient $n = 0.0126$ and carrying maximum flow is given by

$$Q = 100\, y^{8/3} \left(\sqrt{1+z^2} - \frac{z}{2}\right) S_0^{1/2}$$

where Q is discharge in m³/s; y is depth of flow in m in the channel of side slopes 1 vertical to z horizontal and S_0 is the bed slope of the channel.

15.7 An open channel is V-shaped, each side being inclined at 45° to the vertical. If it carries a discharge of 0.04 m³/s, when the depth of flow at the centre is 225 mm, calculate the slope of the channel assuming that Chezy's C = 50. [*Ans.* 0.00314]

15.8 An open channel is to be constructed of trapezoidal section and with side slopes 1 vertical to $1\frac{1}{2}$ horizontal. Find the proportions (i.e., relation between bottom width and depth of flow) for minimum excavation (i.e., best hydraulic section).
If the flow is to be 2.7 cumec, calculate the bottom width and the depth of flow assuming C in Chezy's formula as 44.5 and the bed slope is 1 in 4000. [*Ans.* 0.606; 0.885 m; 1.46 m]

15.9 Comment on the statement: "The maximum discharge from a circular cross-section not under pressure, occurs at a little less than full depth".

15.10 (a) Explain the terms "cross-section of greatest efficiency" and "critical velocity" for flow in an open channel. (b) Discuss the factors that influence the adoption of safe permissible velocities in open earthen channels. (c) It is required to excavate a canal of rectangular section out of rock to bring 15 m³ of water per second from a distance of 6.4 km with a velocity of 2.25 m/s. Determine the most suitable section for the canal and its gradient. Take $n = 0.02$. [*Ans.* 3.65 m; 1.825 m; 0.002 29]

15.11 Determine the most efficient section of a trapezoidal channel with side slopes 1 vertical to 2 horizontal, carrying a discharge of 11.25 m³/s with a velocity of 0.75 m/s. What should be the bed slope of the channel ? Take Manning's $n = 0.025$. [*Ans.* 2.46 m; 1.16 m; 0.000 27]

15.12 A canal is to have a trapezoidal section with one side vertical and the other sloping at 45 degrees. It has to carry 30 m³ of water per second with a mean velocity of 1 m/s. Compute the dimensions of the section which will require the minimum lining. [*Ans.* $y = 3.96$ m; $B = 5.60$ m]

15.13 An open channel laid at a constant slope is required to carry a maximum discharge of 5 m³/s and a minimum discharge of 1 m³/s at a constant velocity of 1 m/s at all depths of flow. Compute graphically or otherwise the top width at the free surface and the depths of flow corresponding to minimum and maximum discharges. For minimum discharge a rectangular channel section of the most economical type may be designed. [*Ans.* 12.56 m; 0.707 m; 1.505 m]

15.14 A trapezoidal channel with side slopes of 2 horizontal to 1 vertical has to carry a discharge of 20 m³/s. If the bottom width is 4 m, calculate the bottom slope required to maintain a uniform flow at a depth of 1.5 m. Take Manning's $n = 0.015$. What would be the normal depth of flow for the above channel to carry a discharge of 27 m³/s? [*Ans.* 1 in 1193; 1.745 m]

15.15 Show that relation between the alternate depths y_1 and y_2 in a rectangular channel can be expressed by

$$\frac{2y_1^2 y_2^2}{(y_1 + y_2)} = y_c^3$$

15.16 (a) Explain the energy variation in a short hydraulic jump using the specific energy equation.
(b) A uniform flow of 9 m³/s occurs at a depth of 1.5 m in a long rectangular channel 3 m wide. Compute the minimum height of a flat top hump that can be built on the channel floor in order to produce a critical depth. [*Ans.* 246 mm]

15.17 A trapezoidal channel having bottom width 5 m and side slopes 1:1 carries a discharge of 12 m³/s. Compute the critical depth and the critical velocity. If Manning's $n = 0.02$ determine the bottom slope required to maintain the critical depth. [*Ans.* 0.793 m; 2.61 m/s; 1 in 200]

15.18 For a constant energy of 2.4 joules per newton (or 2.4 newton metre per newton) calculate the maximum discharge that may occur in a rectangular channel 4 m wide. [*Ans.* 25.36 m³/s]

15.19 A rectangular channel 7.5 m wide carries 12 m³ of water per second with a velocity of 1.5 m/s. Compute the specific energy. Also find the depth of flow in the channel when the specific energy would be minimum. What will be the value of critical velocity as well as minimum specific energy ? [*Ans.* 1.18 J/N ; 0.639 m; 2.503 m/s; 0.959 J/N]

15.20 Uniform flow occurs at a depth of 1.5 m in a long rectangular channel 3 m wide and laid at a slope of 0.0009. If Manning's n is given as 0.015, calculate the width of contraction which will produce critical depth without increasing the upstream depth of flow. [*Ans.* 2.08 m]

15.21 A rectangular channel is reduced gradually from 2 m to 1.5 m and the floor is raised by 0.25 m at a given section. When the approaching depth of flow is 1.5 m, what rate of flow will be indicated if the flow at the contracted section is at critical depth. [*Ans.* 3.95 m³/s]

15.22 Define 'critical tractive stress' and state the main factors on which it depends. A wide unlined channel carrying silt free water has a depth of 2.0 m. The maximum tractive stress permissible on the bed to prevent scour is 1.962 N/m². What is the maximum slope that can be given to the channel?
[Hint. For 'critical tractive stress' refer Chapter 28. However, for this problem tractive stress may be taken same as boundary shear stress $\tau_0 = wRS_0 = wyS_0$]. [*Ans.* 1 in 10000]

15.23 A rectangular channel 2.4 m wide carries uniform flow of 7 cumec at a depth of 1.5 m. If there is a local rise of 150 mm in the bed level, calculate the change of water elevation. What can be the maximum rise in the bed elevation such that the upstream depth is not affected? [*Ans.* 73.3 mm; 262 mm]

15.24 A trapezoidal channel with a base width of 6 m and side slopes of 2 hor. to 1 ver. conveys water at 17 m³/s with a depth of 1.5 m. Is the flow situation sub or super critical? [*Ans.* $Fr = 0.38$, hence flow is subcritical]

15.25 In a wide rectangular channel if the normal depth is increased by 20%, determine the corresponding percentage increase in the discharge. Use Manning's equation. [*Ans.* 35.5%]

15.26 A 3.6 m wide rectangular channel carries water to a depth of 1.8 m. In order to measure the discharge, the channel width is reduced to 2.4 m and a hump of 0.3 m height is provided in the bottom. Calculate the discharge if water surface in the contracted section drops by 0.15 m. Assume no losses. [*Ans.* 6.42 m³/s]

15.27 A trapezoidal channel has a bottom width of 6 m and side slopes of 1:1. The depth of flow is 1.5 m at a discharge of 15 m³/s. Determine the specific energy. If the critical depth is 0.9 m, discuss the type of flow corresponding to the critical depth. [*Ans.* 1.591 m; Flow is in critical state with min. specific energy = 1.197 m]

Non-uniform Flow in Channels

Chapter 16

16.1 INTRODUCTION

As explained in Chapter 15 the non-uniform or varied flow in a channel is the one in which depth of flow changes from section to section along the length of the channel. It may be further classified as *gradually varied flow* (G.V.F.) and *rapidly varied flow* (R.V.F.). The gradually varied flow is a steady non-uniform flow in which the depth of flow varies gradually. Many cases of gradually varied flow are of practical interest to engineers such as flow upstream of a weir or a dam, flow downstream of a sluice gate, flow in channels with break in bottom slopes etc., wherein study of back water and the location of hydraulic jump is of major importance. In a rapidly varied flow the depth of flow changes abruptly over a comparatively short distance. Typical examples of rapidly varied flow are hydraulic jump and hydraulic drop. In this chapter both the types of non-uniform flows have been discussed.

16.2 GRADUALLY VARIED FLOW

The problem of gradually varied flow is that of predicting overall flow pattern, or in other words prediction of the water surface profile to be expected in a given channel with given steady discharge. Such problems can be solved by writing the differential equation for the water surface profile and then integrating it.

1. Dynamic Equation of Gradually Varied Flow. The dynamic equation for gradually varied flow can be derived from the basic energy equation with the following assumptions:

(a) The uniform flow formulae (such as Manning's or Chezy's) may be used to evaluate the energy slope of a gradually varied flow and the corresponding coefficients of roughness developed primarily for uniform flow are applicable to the gradually varied flow also.

Thus,
$$(S_f)_{\text{G.V.F.}} = \left(\frac{Vn}{R^{2/3}}\right)^2 \qquad \text{...(Manning's)}$$

$$(S_f)_{\text{G.V.F.}} = \left(\frac{V}{C\sqrt{R}}\right)^2 \qquad \text{...(Chezy's)}$$

(b) The bottom slope of the channel is very small.

(c) The channel is prismatic.

(d) The energy correction factor α is unity.

(e) The pressure distribution in any vertical is hydrostatic.

(f) The roughness coefficient is independent of the depth of flow and it is constant throughout the channel reach considered.

Considering a short reach of channel having gradually varied flow as shown in Fig. 16.1, the energy equation at any section may be written as

$$H = \frac{V^2}{2g} + y + z$$

or
$$H = \frac{Q^2}{2gA^2} + y + z \qquad \left(\text{since } V = \frac{Q}{A}\right)$$

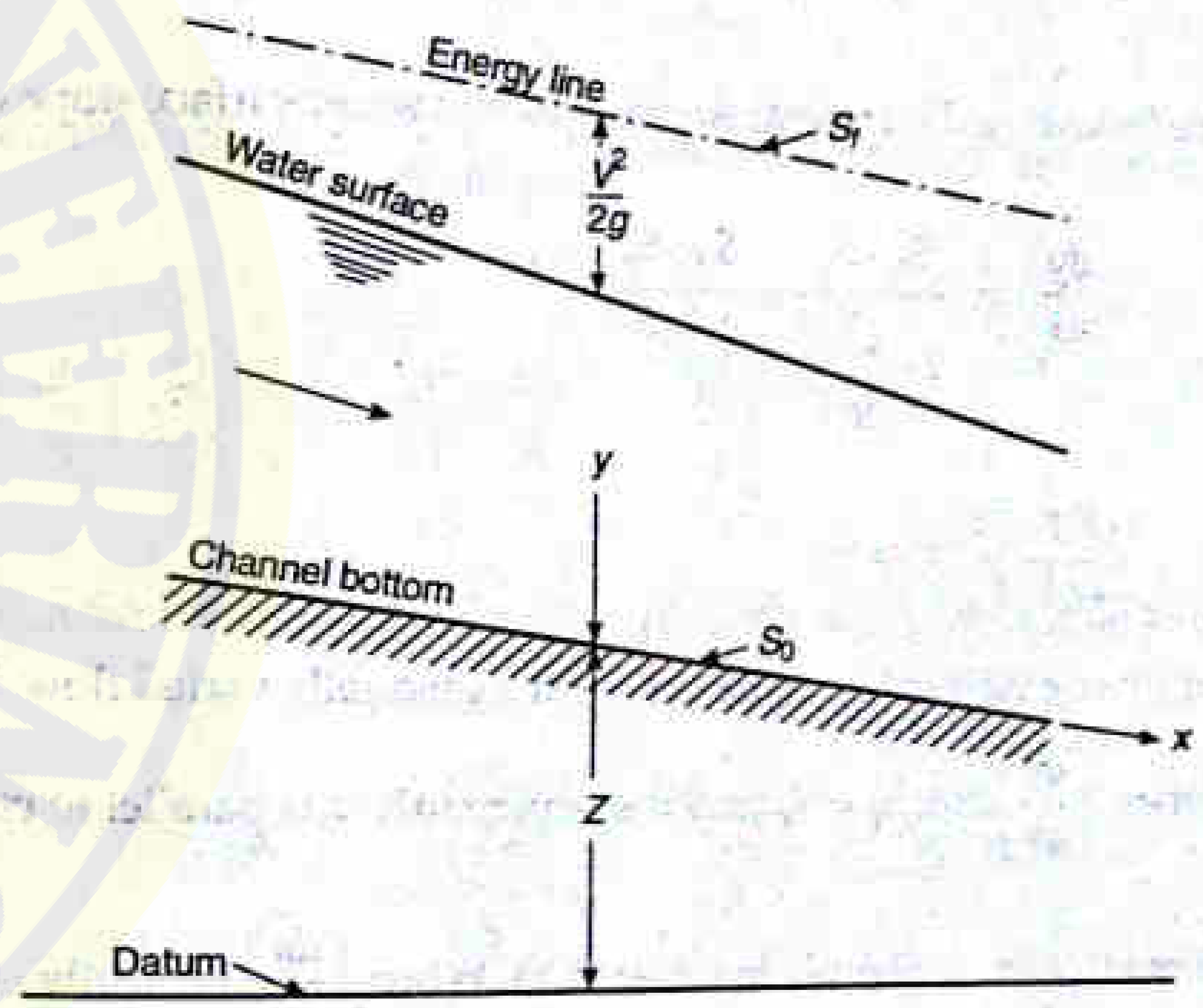


Figure 16.1 *Gradually varied flow*

Differentiating each term of the above equation with respect to x, where x is measured along the channel bottom, the following differential equation can be obtained

$$\frac{dH}{dx} = \frac{d}{dx}\left(\frac{Q^2}{2gA^2}\right) + \frac{dy}{dx} + \frac{dz}{dx}$$

or
$$\frac{dH}{dx} = -\frac{Q^2}{gA^3}\frac{dA}{dx} + \frac{dy}{dx} + \frac{dz}{dx}$$

In the above differential equation, $\frac{dH}{dx}$ is the slope of the energy gradient and hence $\frac{dH}{dx} = -S_f$;

$\frac{dz}{dx}$ is the slope of the channel bed and hence $\frac{dz}{dx} = -S_0$ (the negative signs for S_f and S_0 indicate that as x increases H and z decreases; and $\frac{dy}{dx}$ is the slope of the water surface with respect to the channel bottom. Further the term $\frac{dA}{dx}$ can be written as $\frac{dA}{dx} = \frac{dA}{dy} \times \frac{dy}{dx} = T\frac{dy}{dx}$, since $\frac{dA}{dy}$ is equal to the surface width T.

The substitution of these terms in the above differential equation yields

$$-S_f = -\frac{Q^2T}{gA^3}\frac{dy}{dx} + \frac{dy}{dx} - S_0$$

Solving for $\frac{dy}{dx}$, the following differential equation for the water surface slope can be obtained,

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2T}{gA^3}} = \frac{S_0 - S_f}{1 - Fr^2} \quad \text{...(16.1)}$$

Since $$\frac{Q^2T}{gA^3} = Fr^2$$

Equation 16.1 is the basic differential equation for the gradually varied flow. It may be observed from Eq. 16.1 that when $\left(\frac{dy}{dx}\right) = 0$; $S_0 = S_f$ and the water surface is parallel to channel bottom thus representing a uniform flow as explained in Chapter 15. When $\left(\frac{dy}{dx}\right)$ is positive the water surface is rising and when $\left(\frac{dy}{dx}\right)$ is negative the water surface is falling.

2. Alternative Derivation for Gradually Varied Flow Equation. Equation 16.1 can also be derived by considering the total energy (or total head) at sections 1 and 2, dx apart along the bottom of the channel. Thus

$$z_1 + y_1 + \frac{V_1^2}{2g} = z_2 + y_2 + \frac{V_2^2}{2g} + S_f(dx)$$

or $$(z_1 - z_2) - S_f\,(dx) = (y_2 - y_1) + \frac{V_2^2}{2g} - \frac{V_1^2}{2g}$$

or $$S_0(dx) - S_f\,(dx) = dy + d\left(\frac{V^2}{2g}\right)$$

or $$S_0 - S_f = \frac{dy}{dx} + \frac{d}{dx}\left(\frac{V^2}{2g}\right)$$

or $$S_0 - S_f = \frac{dy}{dx} + \frac{d}{dy}\left(\frac{V^2}{2g}\right)\frac{dy}{dx}$$

or $$\frac{dy}{dx} = \frac{S_0 - S_f}{1 + \frac{d}{dy}\left(\frac{V^2}{2g}\right)}$$

In the above equation the term $\frac{d}{dy}\left(\frac{V^2}{2g}\right)$ represents the change in velocity head which may also be expressed as

$$\frac{d}{dy}\left(\frac{V^2}{2g}\right) = \frac{d}{dy}\left(\frac{Q^2}{2gA^2}\right) = \frac{Q^2}{gA^3}\left(\frac{dA}{dy}\right) = -\frac{Q^2T}{gA^3}$$

Thus by substitution the value of $\frac{d}{dy}\left(\frac{v^2}{2g}\right)$ in the above equation, it becomes

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1 - \frac{Q^2T}{gA^3}}$$

which is same as Eq. 16.1 derived earlier.

Equation 16.1 may also be derived by adopting another approach which is based on the concept of the specific energy as described below. For a given discharge Q flowing in a channel the specific energy at a channel section may be expressed as

$$E = y + \frac{Q^2}{2gA^2}$$

Obviously E is function of the depth of flow. However, in uniform flow since the depth of flow remains constant from section to section, $(dE/dx) = 0$. On the other hand in varied flow since the depth of flow varies from section to section, the specific energy will also vary. Thus differentiating both the sides of the above noted expression for E with respect to the direction of flow x, we have

$$\frac{dE}{dx} = \frac{dy}{dx} + \frac{Q^2}{2g}\left(\frac{2}{A^3}\right)\frac{dA}{dx}$$

or
$$\frac{dE}{dx} = \frac{dy}{dx} - \frac{Q^2}{gA^3}\left(\frac{dA}{dy}\times\frac{dy}{dx}\right)$$

or
$$\frac{dE}{dx} = \frac{dy}{dx}\left(1-\frac{Q^2T}{gA^3}\right)$$

The above expression represents the change of specific energy in a small reach dx of a channel having the nonuniform flow. Further if S_0 is the bottom slope of the channel then $(S_0\,dx)$ is the work done by gravity in a small reach dx. Similarly if S_f is the slope of the energy line than $(S_f dx)$ is the loss of energy which is spent in overcoming the resistance in a small reach dx. Thus in a small reach dx of the channel the net change of specific energy dE becomes

$$dE = (S_0\,dx - S_f\,dx)$$

or
$$\frac{dE}{dx} = S_0 - S_f$$

By substituting this value of (dE/dx) and solving for (dy/dx)

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1-\dfrac{Q^2T}{gA^3}}$$

which is again same as Eq. 16.1 derived earlier.

3. Dynamic Equation for G.V.F. in Wide Rectangular Channel. Equation 16.1 can be expressed in simplified form for a wide rectangular channel as indicated below.

For a wide rectangular channel section of width B the hydraulic radius can be replaced by the depth of flow y since

$$R = \frac{By}{B+2y} \approx \frac{By}{B} = y$$

Further according to Manning's formula

$$Q = \frac{1}{n}(By)\,y^{2/3}S_f^{1/2}$$

$$= \frac{1}{n}(By_n)(y_n)^{2/3}S_0^{1/2}$$

It may be noted that the hydraulic radius has been replaced by the depth of flow and n is assumed to be same for uniform and non-uniform flows. Therefore

$$\frac{S_f}{S_0} = \left(\frac{y_n}{y}\right)^{10/3}$$

However, if Chezy's formula is used instead of the Manning's, the value of

$$\frac{S_f}{S_0} = \left(\frac{y_n}{y}\right)^3$$

In the same manner for a channel of rectangular section

$$\frac{Q^2T}{gA^3} = \frac{Q^2B}{g(B^3y^3)} = \frac{q^2}{gy^3} = \left(\frac{y_c}{y}\right)^3$$

By substituting in Eq. 16.1 it becomes

$$\frac{dy}{dx} = S_0\,\frac{1-\left(\dfrac{y_n}{y}\right)^{10/3}}{1-\left(\dfrac{y_c}{y}\right)^3} \quad ...(16.2)$$

and
$$\frac{dy}{dx} = S_0\,\frac{1-\left(\dfrac{y_n}{y}\right)^3}{1-\left(\dfrac{y_c}{y}\right)^3} \quad ...(16.3)$$

4. Relation between Water Surface Slopes and Channel Bottom Slope. The term (dy/dx) represents the slope of the water surface with respect to the channel bottom. But often the water surface slope S_w with respect to horizontal may be required to be determined. As such a relation between the water surface slope S_w, the channel bottom slope S_0 and the slope (dy/dx) may be developed which facilitates the determination of S_w when S_0 and (dy/dx) are known. For rising water surface as shown in Fig. 16.2 (a), from triangle abd,

$$S_w = \sin\alpha = \frac{bd}{ba} = \frac{cd-cb}{ba}$$

But
$$\frac{cd}{ba} = \sin\theta = S_0, \text{ and } \frac{cb}{ba} = \frac{dy}{dx}$$

$\therefore$
$$S_w = S_0 - \left(\frac{dy}{dx}\right) \quad ...(16.4)$$

However, in this case if the water surface is such that point b lies above point d, then $(dy/dx) > S_0$ and hence Eq. 16.4 becomes

$$S_w = \left(\frac{dy}{dx}\right) - S_0 \quad \text{...(16.4 a)}$$

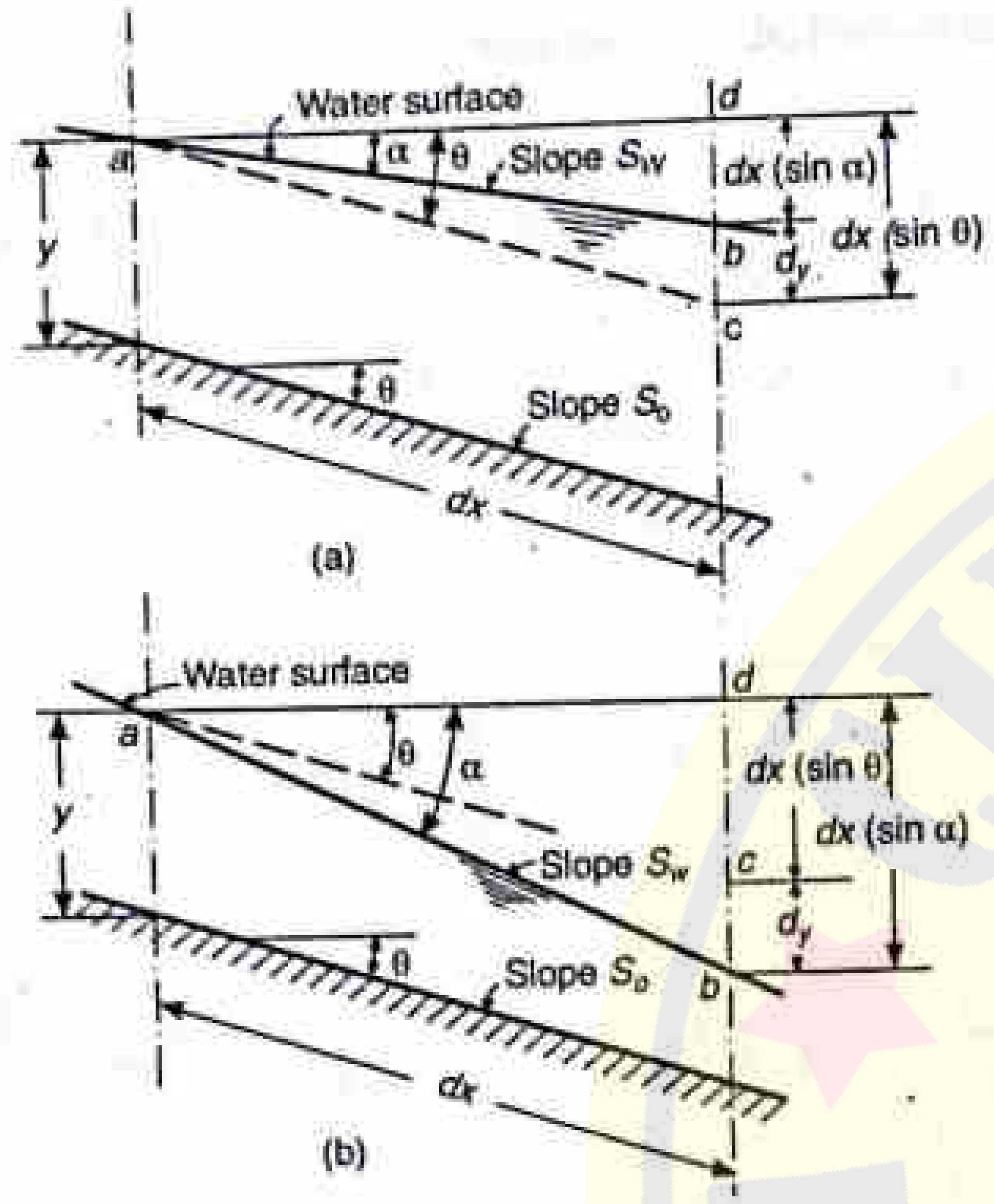


Figure16.2 *Relation between water surface and channel bottom slopes*

For falling water surface as shown in Fig. 16.2 (b), from triangle *abd*,

$$S_w = \sin\alpha = \frac{bd}{ba} = \frac{cd + cb}{ba}$$

Again $$\frac{cd}{ba} = \sin\theta = S_0, \text{ and } \frac{cb}{ba} = \frac{dy}{dx}$$

$$\therefore \quad S_w = S_0 + \left(\frac{dy}{dx}\right) \quad \text{...(16.5)}$$

16.3 CLASSIFICATION OF CHANNEL BOTTOM SLOPES

The channel bottom slopes are classified in the various categories as mentioned below:

(i) Critical Slope. The channel bottom slope is designated as critical when the bottom slope S_0 is equal to the critical slope S_c, i.e., $S_0 = S_c$. Thus in this case the normal depth of flow will be equal to the critical depth, i.e., $y_n = y_c$.

(ii) Mild Slope. The channel bottom slope is designated as mild when the bottom slope S_0 is less than the critical slope S_c, i.e., $S_0 < S_c$. The application of Manning's or Chezy's formula will then indicate that when the bottom slope is mild, the normal depth of flow is greater than the critical depth i.e., $y_n > y_c$.

(iii) Steep Slope. The channel bottom slope is designated as steep when the bottom slope S_0 is greater than the critical slope, i.e., $S_0 > S_c$. Again the application of Manning's or Chezy's formula will indicate that when the bottom slope is steep, the normal depth of flow is less than the critical depth i.e., $y_n < y_c$.

(iv) Horizontal Slope. When the channel bottom slope is equal to zero i.e., $S_0 = 0$, the bottom slope is designated as horizontal. Obviously for a channel with horizontal bottom the normal depth of flow $y_n = \infty$ (infinity).

(v) Adverse Slope. When the channel bottom slope instead of falling rises in the direction of flow it is designated as an adverse slope. Thus in a channel with adverse bottom slope, S_0 is less than zero [i.e., $(S_0 < 0)$] or it is negative. Obviously for an adverse-slopped channel the normal depth of flow y_n is imaginary or it is non-existent.

16.4 CLASSIFICATION OF SURFACE PROFILES

The various water surface profiles occurring in the channels are designated with reference to the bottom slopes of the channels. Thus surface profiles which occur in mild-sloped channels are known

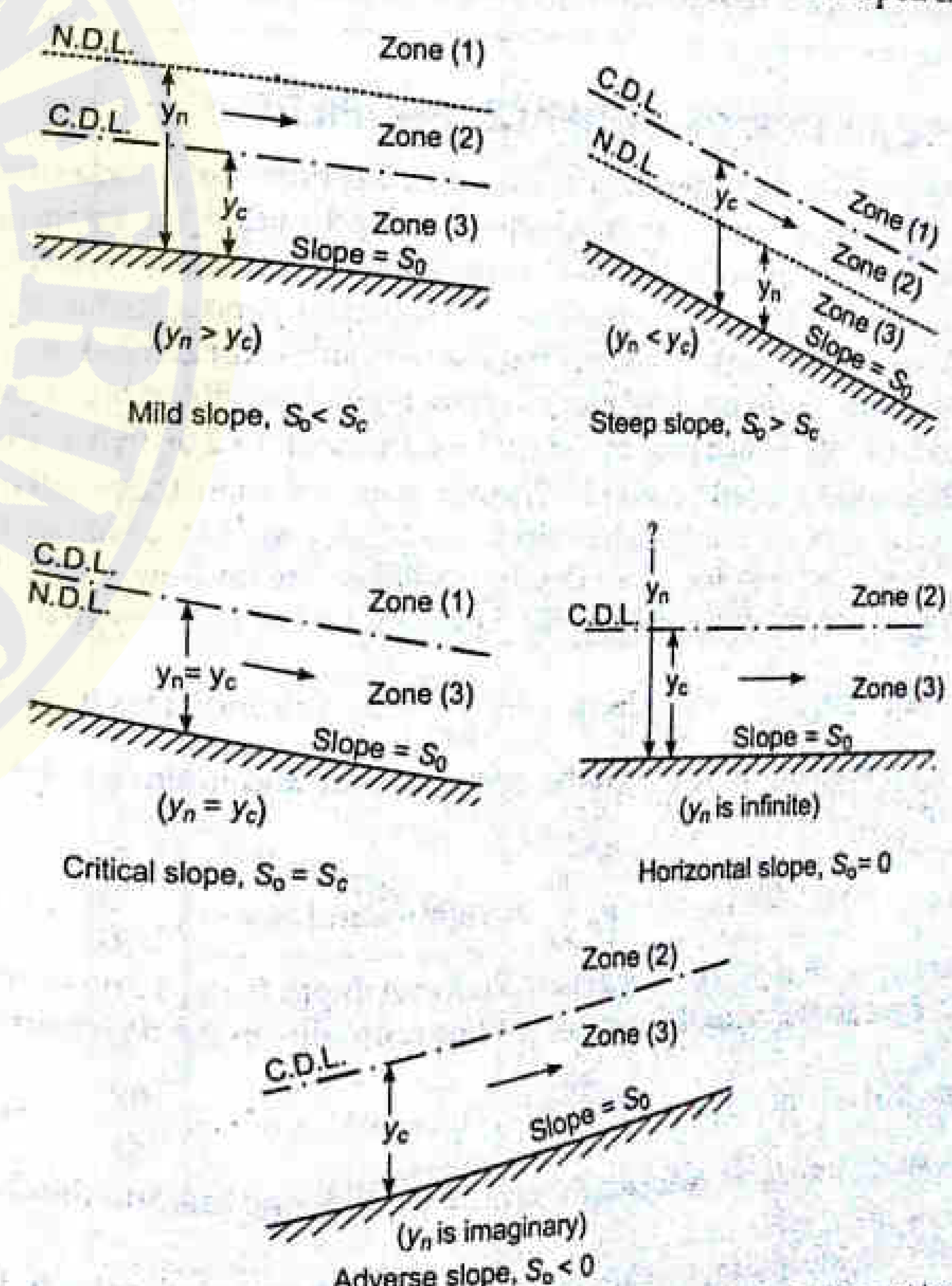


Figure 16.3 *Different zones for water surface profiles in channels of different bottom slopes*

as M-curves; those which occur in steep-slopped channels are known as S-curves; those which occur in critical-slopped channels are known as C-curves; those which occur in horizontal channels are known as H-curves; and those which occur in adverse-slopped channels are known as A-curves. These water surface profiles may be further classified depending upon the position of water surface relative to critical depth y_c and normal depth y_n. For the given discharge and channel section the normal depth line (N.D.L.) and the critical depth line (C.D.L.) divide the entire space above the channel bottom into three zones (Fig. 16.3). In zone 1 the given depth y lies above y_n and y_c; in zone 2 the given depth y lies between y_n and y_c and in zone 3, y lies below y_n and y_c. For mild and steep slopes three such zones are possible and the corresponding three surface profiles are designated as M_1, M_2, M_3 and S_1, S_2, S_3 on the mild and steep slopes respectively. On the critical slope the normal depth y_n and the critical depth y_c being the same, zone 2 vanishes and only two zones 1 and 3 exist, so that the given depth, y lies either above the critical depth or below the critical depth resulting in either C_1 or C_3 curves. For horizontal-slopped channels normal depth is infinite and for adverse-slopped channels it is imaginary, as such only two zones 2 and 3 are possible, resulting in only two types of profiles H_2, H_3 and A_2, A_3 on these two slopes. Thus it can be seen that in all twelve surface profiles are possible as shown in Fig. 16.4.

16.5 CHARACTERISTICS OF SURFACE PROFILES

Figure 16.4 shows the various surface profiles. The surface profiles can be classified as back water curves and drawdown curves depending on whether the depth of flow increases or decreases in the direction of flow (or in other words, dy/dx is positive or negative). The study of surface profiles shown in Fig. 16.4 will indicate that all the surface profiles with subscript 1 and 3, that is $M_1, M_3, S_1, S_3, C_1, C_3, H_3$ and A_3, are back water (or rising) curves while those with subscript 2, that is M_2, S_2, H_2 and A_2 are drawdown (or falling) curves. In order to plot a certain type of profile, it is necessary to know the characteristics of each profile, which may be determined from Eq. 16.2 or 16.3 as discussed below.

1. Surface Profile in Mild-Sloped Channels. In a mild sloped channel there will be three zones viz., $y > y_n > y_c$, $y_n > y > y_c$ and $y_n > y_c > y$, in which respectively M_1, M_2 and M_3 curves will be formed. In the first zone, that is, when $y > y_n > y_c$, the given depth y can have the limiting values as $y \to y_n$ on the upstream side and $y \to \infty$ on the downstream side. Equation 16.2 or 16.3 shows that when $y > y_n > y_c$, $\frac{dy}{dx}$ is positive and as $y \to y_n$, $\frac{dy}{dx} \to 0$ and as $y \to \infty$, $\frac{dy}{dx} \to S_0$. This indicates that M_1 curve meets the normal depth (or y_n) line asymptotically on the upstream side and it tends to be horizontal on the downstream side.

In the second zone, that is, when $y_n > y > y_c$, $\frac{dy}{dx}$ is negative and as $y \to y_n$, $\frac{dy}{dx} \to 0$ and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$, thereby indicating that M_2 curve meets the normal depth (or y_n) line asymptotically on the upstream side and it meets the critical depth (or y_c) line normally on the downstream side.

In the third zone, that is, when $y_n > y_c > y$, $\frac{dy}{dx}$ is positive, and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$ and as $y \to 0$, $\frac{dy}{dx} \to \infty$, thereby indicating that M_3 curve meets the critical depth (or y_c) line and the channel bottom line normally.

2. Surface Profiles in Steep-Sloped Channels. In a steep-sloped channel also there will be three zones viz., $y > y_c > y_n$, $y_c > y > y_n$, and $y_c > y_n > y$, in which respectively S_1, S_2 and S_3 curves will be formed. In the first zone, that is when $y > y_c > y_n$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to \infty$ and when $y \to \infty$, $\frac{dy}{dx} \to S_0$. That is on the upstream side S_1 curve meets the critical depth (or y_c) line normally and on the downstream side it tends to be horizontal.

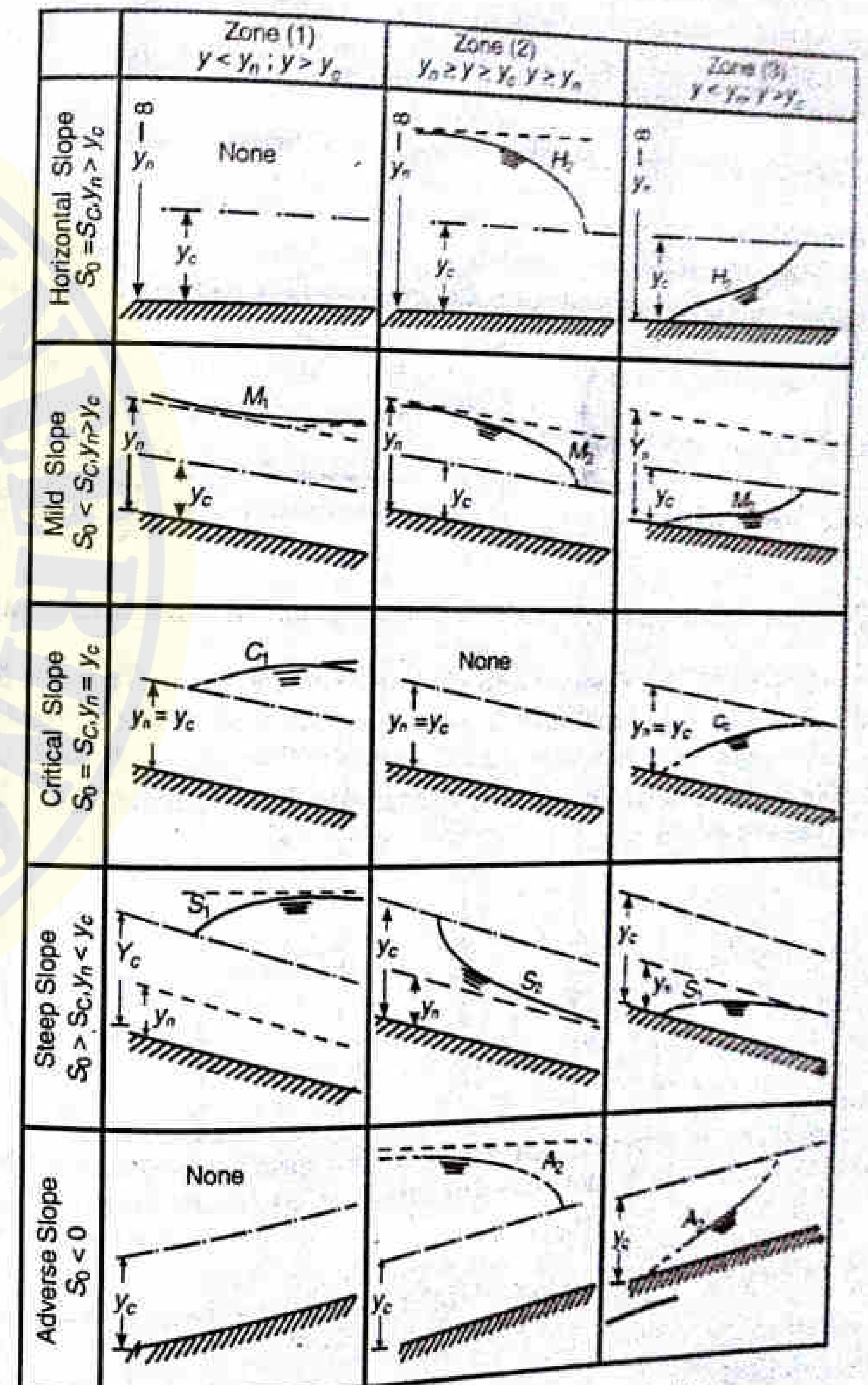


Figure 16.4. *Flow profiles of gradually varied flow*

In the second zone, that is, when $y_c > y > y_n$, $\frac{dy}{dx}$ is negative and as $y \to y_c$, $\frac{dy}{dx} \to \infty$ and as $y \to y_n$, $\frac{dy}{dx} \to 0$, thereby indicating that the S_2 curve meets the critical depth (or y_c) line normally on the upstream side and on the downstream side it meets the normal depth (or y_n) line asymptotically.

In the third zone, that is, when $y_c > y_n > y$, $\frac{dy}{dx}$ is positive and as $y \to y_n$, $\frac{dy}{dx} \to -\infty$ 0, and as $y \to 0$, $\frac{dy}{dx} \to \infty$, thereby indicating that the S_3 curve meets the channel bed normally and it is asymptotic to the normal depth (or y_n) line.

3. Surface Profiles in Critical-Slopped Channels. In a critical-slopped channel as discussed earlier only two zones 1 and 3 exist and hence only C_1 and C_3 curves will be formed. In the first zone, that is, when $y > y_c = y_n$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to S_0 = S_c$, and as $y \to \infty$, $S_0 = S_c$, thereby indicating that C_1 curve will be more or less a horizontal line.

In the third zone, that is, when $y < y_c = y_n$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to S_0 = S_c$, and as $y \to 0$, $\frac{dy}{dx} \to S_0 = S_c$, thereby indicating that C_3 curve will also be more or less a horizontal line.

It may however be stated that if Chezy's formula is used and accordingly Eq. 16.3 is considered then the C_1 and C_3 curves will be horizontal straight lines, but if Manning's formula is used and accordingly Eq. 16.2 is considered then the C_1 and C_3 curves will be slightly curved.

4. Surface Profiles in Horizontal Channels. In a channel with horizontal bottom $S_0 = 0$ and hence Eq. 16.1 may be expressed as

$$\frac{dy}{dx} = \frac{-S_f}{1 - \frac{Q^2 T}{g A^3}} = \frac{-\frac{n^2 V^2}{y^{4/3}}}{1 - \left(\frac{y_c}{y}\right)^3} \qquad \text{...(16.6)}$$

if Manning's formula is used, and

$$\frac{dy}{dx} = \frac{-\frac{V^2}{C^2 y}}{1 - \left(\frac{y_c}{y}\right)^3} \qquad \text{...(16.7)}$$

if Chezy's formula is used.

Further in a channel with horizontal bottom since $y_n = \infty$ only two zones 2 and 3 exist and accordingly only H_2 and H_3 curves will be formed.

In the second zone, that is, when $y > y_c$, $\frac{dy}{dx}$ is negative and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$, thereby indicating that H_2 curve meets the critical depth (or y_c) line normally at the downstream end and at the upstream end it tends to approach horizontal line tangentially.

In the third zone, that is, when $y < y_c$, $\frac{dy}{dx}$ is positive and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$ and as $y \to 0$, $\frac{dy}{dx} \to \infty$, thereby indicating that H_3 curve meets the critical depth (or y_c) line and the channel bottom line normally (i.e., vertically).

TABLE 16.1 *Types of Flow Profiles in Prismatic Channels*

Channel Slope	Symbol	Depth Relations	$\frac{dy}{dx}$	Type of profile	Type of flow
Horizontal $[S_0 = 0]$	None	$y > y_n > y_c$		None	None
	H_2	$y_n > y > y_c$	–	Drawdown	Subcritical
	H_3	$y_n > y_c > y$	+	Backwater	Supercritical
Mild $[0 < S_0 < S_c]$	M_1	$y > y_n > y_c$	+	Backwater	Subcritical
	M_2	$y_n > y > y_c$	–	Drawdown	Subcritical
	M_3	$y_n > y_c > y$	+	Backwater	Supercritical
Critical $[S_0 = S_c > 0]$	C_1	$y > y_c = y_n$	+	Backwater	Subcritical
	None	$y_c = y = y_n$		None	None
	C_3	$y_c = y_n > y$	+	Backwater	Supercritical
Steep $[S_0 > S_c > 0]$	S_1	$y > y_c > y_n$	+	Backwater	Subcritical
	S_2	$y_c > y > y_n$	–	Drawdown	Supercritical
	S_3	$y_c > y_n > y$	+	Backwater	Supercritical
Adverse $[S_0 < 0]$	None			None	None
	A_2	$y > y_c$	–	Drawdown	Subcritical
	A_3	$y_c > y$	+	Backwater	Supercritical

5. Surface Profiles in Adverse-Slopped Channels. In a channel with adverse bottom slope since $S_0 < 0$ (i.e., S_0 is negative) the normal depth is imaginary and hence only two zones 2 and 3 exist in which A_2 and A_3 curves will be formed.

In the second zone, that is, when $y > y_c$, Eq. 16.1 indicates that $\frac{dy}{dx}$ is negative. Again when $y \to y_c$, $\frac{dy}{dx} \to -\infty$ and when $y \to \infty$, $\frac{dy}{dx} \to S_0$ thereby indicating that A_2 curve tends to be horizontal at the upstream end and at the downstream end it meets the critical depth (or y_c) line normally.

In the third zone that is when $y < y_c$, $\frac{dy}{dx}$ is positive, and as $y \to y_c$, $\frac{dy}{dx} \to -\infty$, and as $y \to 0$, $\frac{dy}{dx} \to \infty$ thereby indicating that A_3 curve meets the critical depth (or y_c) line and the channel bottom line normally.

For the sake of convenience the above discussed characteristics of the various surface profiles have been tabulated in Table 16.1

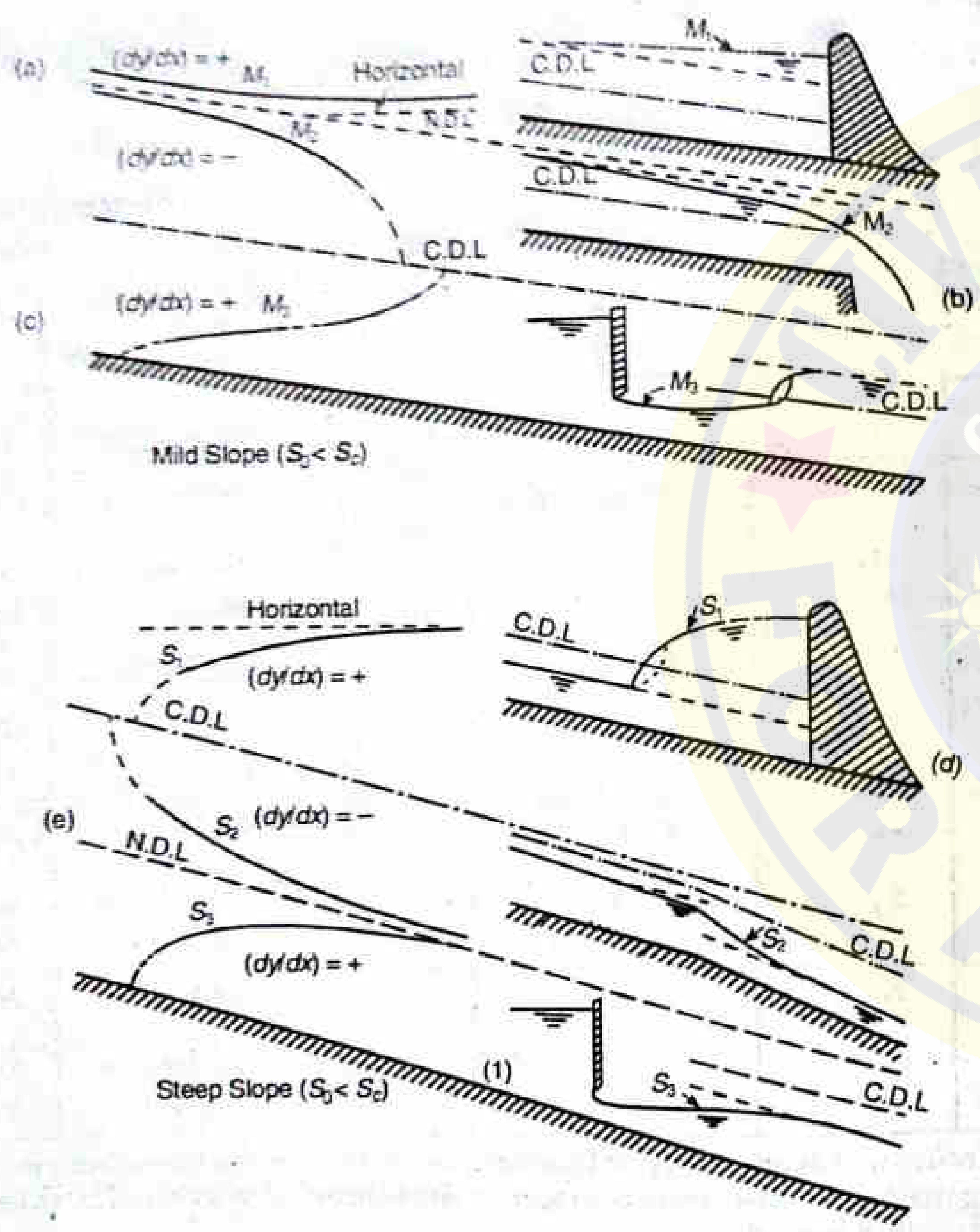


Figure 16.5 *Practical examples of M and S surface profiles*

Some of the practical examples of the various surface profiles are shown in Figs. 16.5 and 16.6. Since the flow profiles near the critical depth line and the channel bottom cannot be accurately defined by the theory of gradually varied flow, they are shown with dotted lines.

In order to determine the class of surface profile in a given channel the following procedure may be adopted :

(i) Compute y_n and y_c for the given discharge and plot the lines representing the normal and the lines of normal and critical depths.

(ii) By comparing the normal depth and the critical depth determine whether the channel slope is mild, critical, steep, adverse or horizontal.

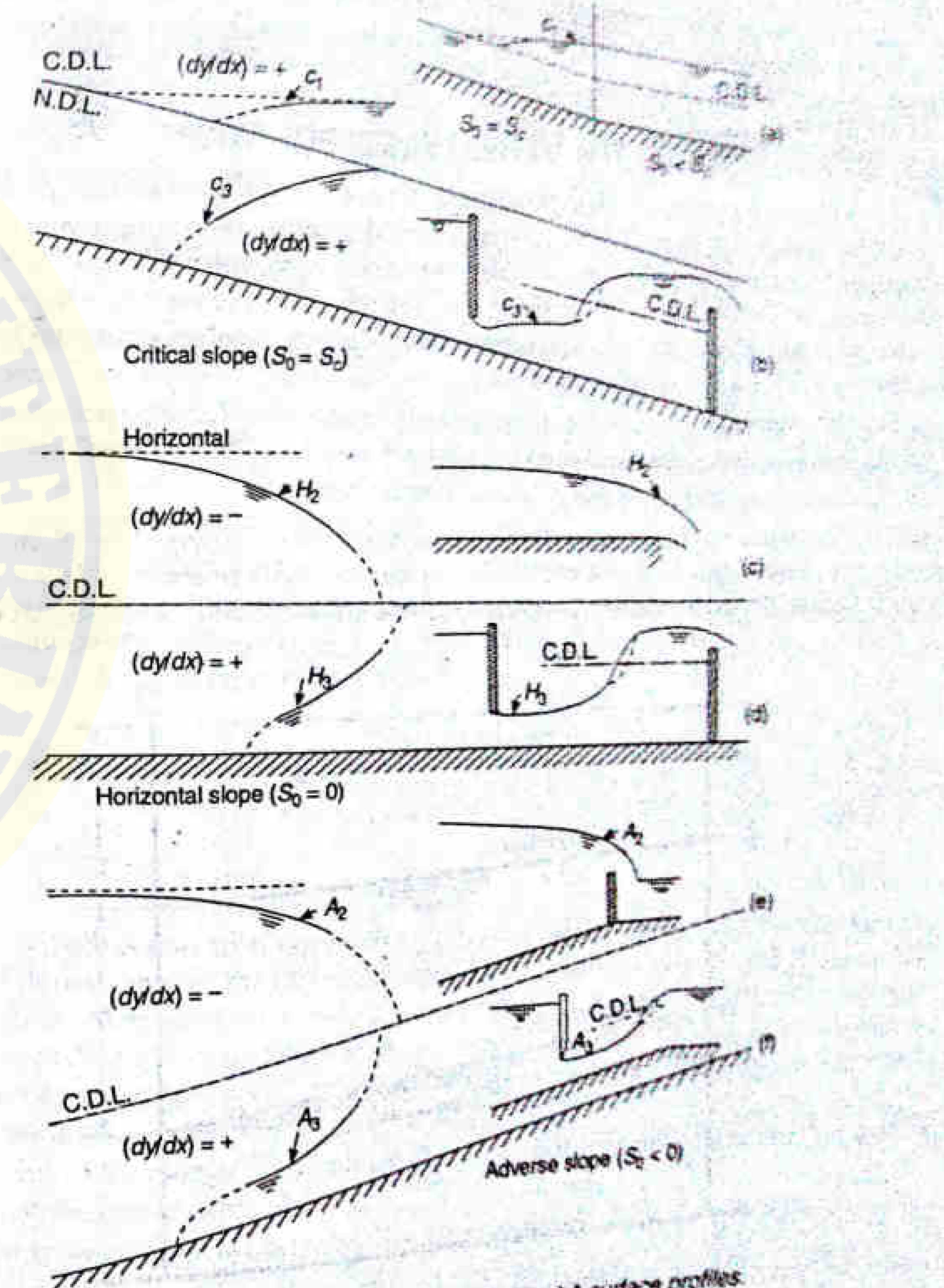


Figure 16.6 *Practical examples of C, H and A surface profiles*

(iii) Knowing the normal depth and the depth at the control section, determine the type of the surface profile.

A *control section* is that at which for a given discharge the depth of flow is known or it can be controlled to a required value. If the depth of non-uniform flow y is above the critical depth it is governed by a downstream control and if it is less than the critical depth it is governed by an upstream control.

Thus knowing the values of the given depth y, normal depth y_n and the critical depth y_c the appropriate surface profile can be sketched.

(iv) If in any reach the supercritical depth has to meet the subcritical depth, i.e., when the stream of water has to cross the critical depth line then there will be a hydraulic jump developed in between, as shown in Figs. 16.5 and 16.6.

16.6 INTEGRATION OF THE VARIED FLOW EQUATION

In practice it is often required to determine the distance upto which the surface profile of gradually varied flow extends. For instance, if a weir is constructed across a river having a mild slope then it may be required to estimate the distance on the upstream side upto which the effect of the resulting M_1 profile exists. In order to solve the problems of this type it is necessary to integrate the dynamic equation of gradually varied flow. The various methods developed for integrating the varied flow equation may be broadly classified as follows:

(a) The step method.

(b) The graphical integration method.

(c) The direct integration method.

(a) The Step Method. In the step method of integration the entire length of the channel is divided into short reaches and the computation is carried out step by step from one end of the reach to the other. Figure 16.7 illustrates a channel reach of length dx, which is sufficiently small so that in this reach the

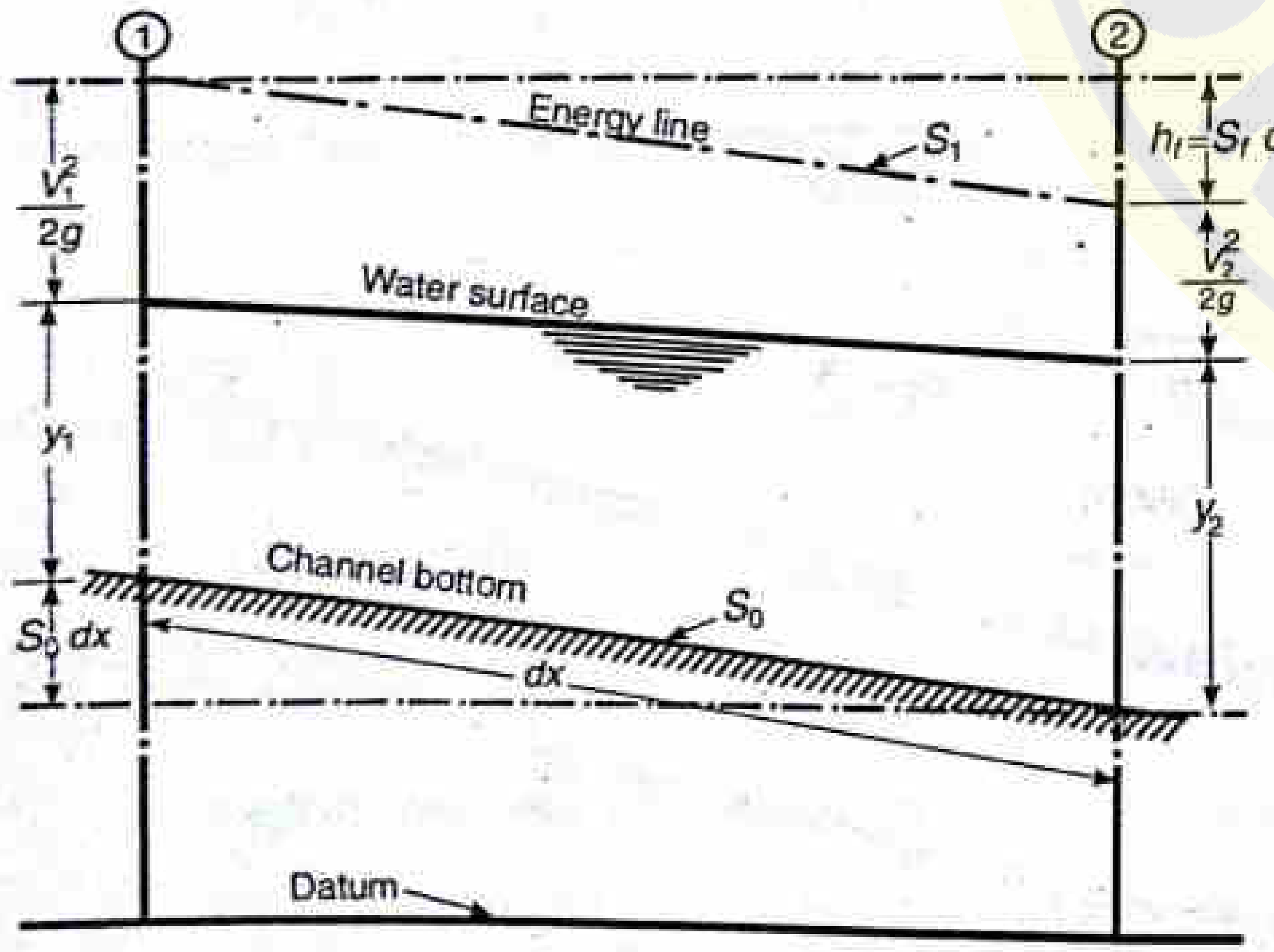


Figure 16.7 *Short reach of channel*

surface profile may be approximated by a straight line. Equating the total energies at the sections 1 and 2 at the either end of the channel reach under consideration the following expression is obtained

$$(S_0)\,dx + y_1 + \frac{V_1^2}{2g} = y_2 + \frac{V_2^2}{2g} + (S_f)\,dx$$

in which all the notations are as shown in Fig. 16.7.

Since $$E_1 = y_1 + \frac{V_1^2}{2g} \quad \text{and} \quad E_2 = y_2 + \frac{V_2^2}{2g}$$

where E_1 and E_2 are the values of the specific energy at sections 1 and 2, the above expression may be written as

$$(S_0)\,dx + E_1 = E_2 + (S_f)\,dx$$

Solving the above expression for dx

$$dx = \frac{E_2 - E_1}{S_0 - S_f} = \frac{\Delta E}{S_0 - S_f} \quad \ldots(16.8)$$

where ΔE represents the change in specific energy between the sections 1 and 2.

The step method of integration is based on Eq. 16.8. For the computation of the length x of the surface profile the various quantities that should be known are discharge Q, the channel shape, the bottom slope S_0, Manning's n or Chezy's C and the depth of flow at one of the sections (usually at one of the control sections). The procedure for computation is as follows:

1. Compute the area of flow section A, wetted perimeter P and the hydraulic radius R corresponding to the known value of the depth of flow y.

2. Compute the mean velocity V, the velocity head $(V^2/2g)$, the specific energy E and the energy line slope S_f at the channel section where the depth of flow is known. The energy line slope S_f may be computed by using either Manning's or Chezy's formula as

$$S_f = \frac{n^2V^2}{R^{4/3}}$$

or $$S_f = \frac{V^2}{C^2R}$$

3. Assume a new depth of flow for another section lying very close to the previous one and from the assumed depth of flow y compute the various quantities as indicated in steps (1) and (2) above. The assumed depth will be either more or less than the known depth at the control section, depending on whether the surface profile is rising or falling from the control section. In order to keep the distance dx between the two adjacent sections small, as far as possible small incremental increase or decrease in depth should be assumed.

4. Obtain the value of the change in specific energy ΔE, from the computed values of the specific energies E_1 and E_2 at the two adjacent sections.

5. In Eq. 16.8 the average value of the energy line slopes at the two adjacent sections is to be used. The average value of S_f is denoted by $\bar{S}_f$ and is equal to $\frac{1}{2}(S_{f1} + S_{f2})$, where S_{f1} and S_{f2} are the values

of the slope of the energy line at the two adjacent sections. Thus compute $\bar{S}_f$ from the computed values of S_{f1} and S_{f2} at the two adjacent sections.

6. Substitute the values of ΔE, S_0 and $\bar{S}_f$ in Eq. 16.8 and compute dx, which represents the length of the portion of the surface profile lying between the two adjacent sections.

7. Repeat the process by assuming different values of the depths of flow y at the adjacent sections until the required section is reached. Add the resulting dx values to obtain the required length x of the flow profile.

The step method of integration is very simple. However, the accuracy of this method depends on the depth increments chosen.

(b) The Graphical Integration Method. The graphical method of integration of varied flow equation can be used for both prismatic as well as non-prismatic channels. Consider Eq. 16.1 viz.,

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1-\frac{Q^2T}{gA^3}}$$

Since for a given discharge, A, T, and S_f are functions of the depth of flow y, we may write

$$dx = f(y)\,dy \qquad \text{...16.9)}$$

in which $$f(y) = \left(1-\frac{Q^2T}{gA^3}\right)\Big/\left(S_0 - S_f\right)$$

Now if y_1 and y_2 are the depths of flow at points distant x_1 and x_2 from the control section, then by integrating Eq. 16.9, we get

$$(x_2 - x_1) = \int_{x_1}^{x_2} dx = \int_{y_1}^{y_2} f(y)dy \qquad \text{...(16.10)}$$

By assuming several values of y, the corresponding values of $f(y)$ may be computed. A curve of y versus $f(y)$ is then prepared and the area under the curve between $y = y_1$ and $y = y_2$ is determined which will give the longitudinal distance $(x_2 - x_1)$ required for this change in the depth of flow, in accordance with Eq. 16.10. If y_1 and y_2 do not differ much then approximately

$$(x_2 - x_1) = \frac{f(y_1)+f(y_2)}{2}(y_2 - y_1)$$

in which $f(y_1)$ and $f(y_2)$ are the values of $f(y)$ at $y = y_1$ and $y = y_2$.

(c) The Direct Integration Method. There are several methods of direct integration of varied flow equation, however, only Bresse's method is discussed here.

Bresse's Method. Bresse's method of integrating the varied flow equation is applicable in the case of very wide rectangular channels. In this method Chezy's formula is used for the evaluation of the effect of frictional resistance to the flow. The general varied flow equation may be expressed for this case as

$$\frac{dy}{dx} = S_0 \frac{1-\left(\frac{y_n}{y}\right)^3}{1-\left(\frac{y_c}{y}\right)^3} \qquad \text{...(16.10 a)}$$

Now if $(y/y_n) = u$, then $dy = y_n du$, and the above equation can be rearranged as

$$dx = \frac{y_n}{S_0}\left[1-\left(1-\frac{y_c^3}{y_n^3}\right)\frac{1}{1-u^3}\right]du$$

Integrating the above equation, we get

$$x = \frac{y_n}{S_0}\left[u-\left\{1-\left(\frac{y_c}{y_n}\right)^3\right\}\int\frac{du}{1-u^3}\right] + \text{constant} \qquad \text{...(16.11)}$$

The distance x can be calculated from Eq. 16.11 if the value of the integral $\left[\int\frac{du}{1-u^3}\right]$ is known. The function $\int\frac{du}{1-u^3}$ is known as Bresse's varied flow function and the tables are available giving the numerical values of this function for different values of u. However, it can also be calculated from the following expression.

$$\int\frac{du}{1-u^3} = \left[\frac{1}{6}\log_e\frac{u^2+u+1}{(u-1)^2} - \frac{1}{\sqrt{3}}\cot^{-1}\left(\frac{2u+1}{\sqrt{3}}\right)\right] \qquad \text{...(16.12)}$$

Further the ratio $\left(\frac{y_c}{y_n}\right)$ in a wide rectangular channel can be expressed as

$$\left(\frac{y_c}{y_n}\right)^3 = \frac{C^2S_0}{g}$$

where C is Chezy's coefficient.

Thus substituting the above expression in Eq. 16.11 and writing $\int\frac{du}{1-u^3}$ as $\phi(u)$, the Bresse's solution for the varied flow equation becomes

$$(x_2 - x_1) = \frac{y_n}{S_0}\left[(u_2 - u_1) - \left(1-\frac{C^2S_0}{g}\right)\times\{\phi(u_2)-\phi(u_1)\}\right] \qquad \text{...(16.13)}$$

in which u_2 and u_1 are the values of u at x_2 and x_1 respectively and $\phi(u_2)$ and $\phi(u_1)$ are the values of $\phi(u)$ for u equal to u_2 and u_1 respectively.

16.7 HYDRAULIC JUMP

The hydraulic jump is defined as the sudden and turbulent passage of water from a supercritical state to subcritical state. It has been classified as rapidly varied flow, since the change in depth of flow from rapid to tranquil state is in an abrupt manner over a relatively short distance. The flow in a hydraulic jump is accompanied by the formation of extremely turbulent rollers and there is a considerable dissipation of energy.

A hydraulic jump will form when water moving at a supercritical velocity in a relatively shallow stream strikes water having a relatively large depth and subcritical velocity. It occurs frequently in a canal below a regulating sluice, at the foot of a spillway, or at the place where a steep channel bottom slope suddenly changes to a flat slope.

In order to study the conditions of flow before and after the hydraulic jump the application of the energy equation does not provide an adequate means of analysis, because hydraulic jump is associated with an appreciable loss of energy which is initially unknown. As such in the analysis of hydraulic jump the momentum equation is used by considering the portion of the hydraulic jump as the control volume. The following assumptions are, however, made in this analysis:

(1) It is assumed that before and after jump formation the flow is uniform and the pressure distribution is hydrostatic.

(2) The length of the jump is small so that the losses due to friction on the channel floor are small and hence neglected.

(3) The channel floor is horizontal or the slope is so gentle that the weight component of the water mass comprising the jump is negligibly small.

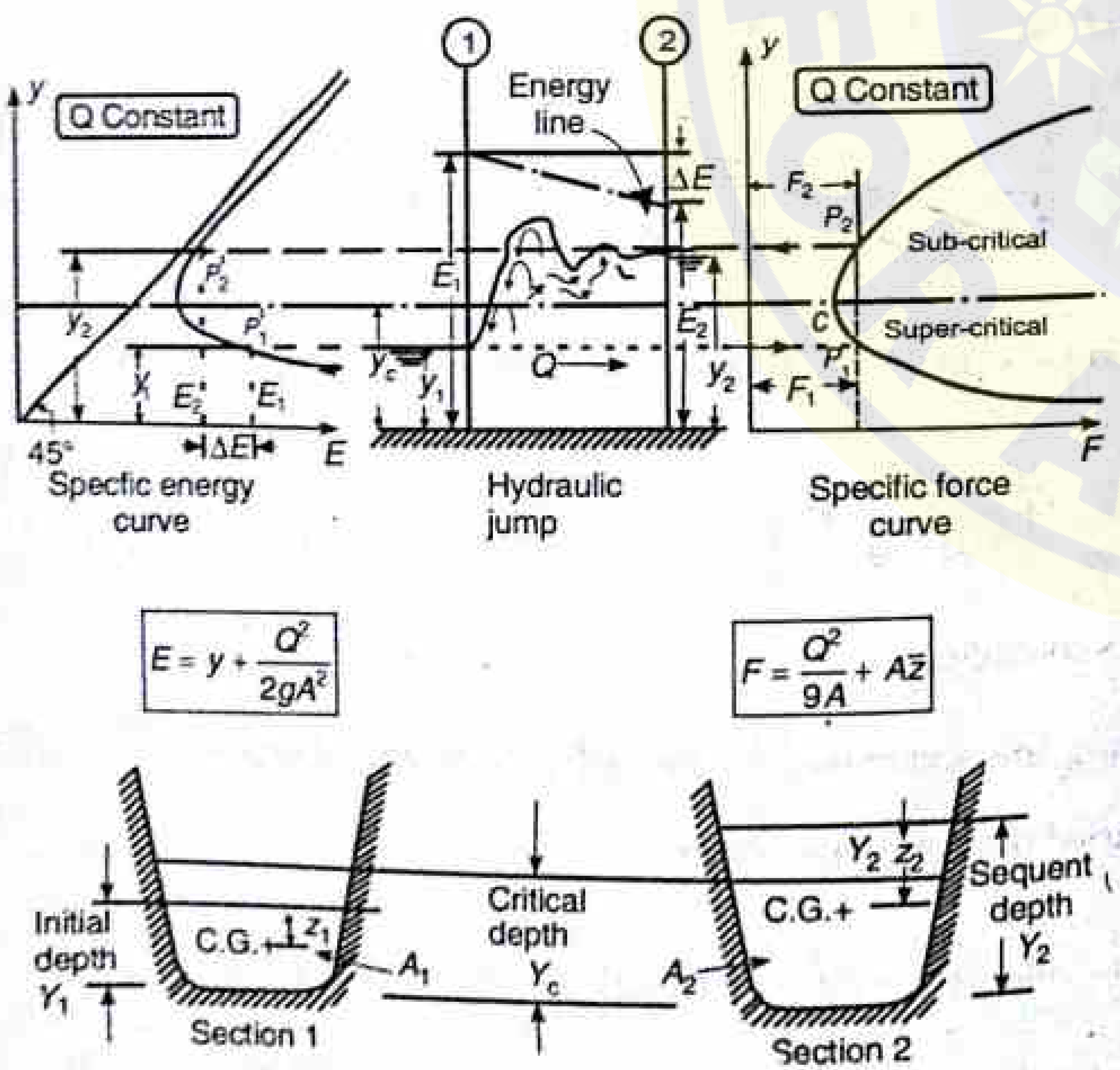


Figure 16.8 *Hydraulic jump in a prismatic channel*

Consider a hydraulic jump formed in a prismatic channel with horizontal floor carrying a discharge Q as shown in Fig. 16.8. Let the depth of flow before the jump at section 1 be y_1 and the depth of flow after the jump at section 2 be y_2. The depth y_1 is known as initial depth and y_2 is known as sequent depth. The symbols A_1, V_1 and $\bar{z}_1$ represent the area of cross-section, mean velocity of flow and the depth of the centroid of area A_1 below the free surface respectively at section 1 before the jump and A_2, V_2 and $\bar{z}_2$ are the corresponding quantities at section 2 after the jump.

The only external forces acting on the mass of water between the sections 1 and 2 are the hydrostatic pressures P_1 and P_2 at sections 1 and 2 respectively, as the frictional loss has been assumed to be negligible. Hence in accordance with the momentum equation

$$(P_2 - P_1) = \rho Q (V_1 - V_2)$$

or
$$wA_2\bar{z}_2 - wA_1\bar{z}_1 = \frac{w}{g} Q\left(\frac{Q}{A_1} - \frac{Q}{A_2}\right)$$

or
$$\frac{Q^2}{gA_1} + A_1\bar{z}_1 = \frac{Q^2}{gA_2} + A_2\bar{z}_2 \quad ...(16.14)$$

As already explained in Chapter 15, the sum $\left(\frac{Q^2}{gA} + A\bar{z}\right)$ is called the *specific force*, designated by F.

Thus if F_1 and F_2 represent the specific force at sections 1 and 2 respectively then Eq. 16.14 may be written as

$$F_1 = F_2 \quad ...(16.15)$$

The specific force F is a function of the depth of flow y and hence it can be plotted against the depth of flow y to obtain specific force curve as shown in Fig. 16.8. It may be noted from the specific force curve that alike specific energy the specific force also attains a minimum value at critical depth y_c.

It is obvious from Eq. 16.15 that the specific force F_1 corresponding to y_1 and the specific force F_2 corresponding to y_2 are same which may also be seen from the specific force curve shown in Fig. 16.8. The initial depth y_1 and the sequent depth y_2 are commonly known as the conjugate depths, which indicate the same specific force (in order to distinguish them from the alternate depths which indicate the same specific energy).

Equation 16.14 enables the determination of y_2, if y_1 is given or vice versa for a known discharge Q flowing in a given channel section. Alternatively from the specific force curve also, knowing y_1 and hence F_1, y_2 can be determined or vice versa, since both y_1 and y_2 correspond to same specific force.

Knowing the conjugate depths y_1 and y_2 for a known discharge Q in a given channel section the specific energies E_1 and E_2 at the sections 1 and 2 respectively may be computed from which the loss of energy in the jump can be determined. Alternatively the loss of energy in the jump may also be determined by using the specific energy curve in combination with specific force curve as indicated below.

Knowing y_1 the corresponding specific force F_1 can be found from specific force curve as indicated by the point P_1. A vertical through point P_1 will cut the curve at point P_2. Since $F_1 = F_2$, the ordinate of point P_2 will indicate the depth y_2. Horizontal lines drawn through points P_1 and P_2 will cut the specific energy curve at points P'_1 and P'_2 respectively, indicating the values of the specific energies before and after the jump, i.e. E_1 and E_2 respectively. The horizontal distance between points P'_1 and P'_2 is the energy loss ΔE, due to hydraulic jump.

(a) Hydraulic Jump in Rectangular Channels. For rectangular channels Eq. 16.14 can be further simplified and a relation between the conjugate depths y_1 and y_2 can be obtained. Thus if B is the width of the rectangular channel, then $A_1 = By_1$; $A_2 = By_2$; $\bar{z}_1 = (y_1/2)$, $\bar{z}_2 = (y_2/2)$ and $q = (Q/B)$. By substituting these values in Eq. 16.14, it becomes

$$\frac{Q^2}{g(By_1)} + By_1\left(\frac{y_1}{2}\right) = \frac{Q^2}{g(By_2)} + By_2\left(\frac{y_2}{2}\right)$$

or

$$\frac{Q^2}{gB^2}\left(\frac{1}{y_1} - \frac{1}{y_2}\right) = \frac{1}{2}(y_2^2 - y_1^2)$$

or

$$\frac{q^2}{g}\left(\frac{y_2 - y_1}{y_1 y_2}\right) = \frac{1}{2}(y_2 - y_1)(y_2 + y_1)$$

or

$$\frac{2q^2}{g} = y_1 y_2 (y_1 + y_2) \qquad \text{...(16.16)}$$

Equation 16.16 is the momentum equation for hydraulic jump in rectangular channels.

Equation 16.16 can be considered as quadratic equation in terms of y_1 or y_2 and its solution gives the relations between the conjugate depths y_1 and y_2 as

$$y_2 = -\frac{y_1}{2} + \sqrt{\left(\frac{y_1}{2}\right)^2 + \frac{2q^2}{gy_1}} \qquad \text{...(16.17)}$$

$$y_1 = -\frac{y_2}{2} + \sqrt{\left(\frac{y_2}{2}\right)^2 + \frac{2q^2}{gy_2}} \qquad \text{...(16.18)}$$

In Eqs 16.17 and 16.18 the negative sign before the square root has not been used since it gives negative values.

Equations 16.17 and 16.18 may also be written as

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + \frac{8q^2}{gy_1^3}}\right] \qquad \text{...(16.19)}$$

$$\frac{y_1}{y_2} = \frac{1}{2}\left[-1 + \sqrt{1 + \frac{8q^2}{gy_2^3}}\right] \qquad \text{...(16.20)}$$

Since for a rectangular channel $\left(\frac{q^2}{g}\right) = y_c^3$, where y_c is the critical depth for discharge q. The Eqs 16.19 and 16.20 may also be written as

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + 8\left(\frac{y_c}{y_1}\right)^3}\right] \qquad \text{...(16.21)}$$

$$\frac{y_1}{y_2} = \frac{1}{2}\left[-1 + \sqrt{1 + 8\left(\frac{y_c}{y_2}\right)^3}\right] \qquad \text{...(16.22)}$$

Since the Froude numbers Fr_1 and Fr_2 before and after the hydraulic jump respectively are given as

$$Fr_1 = \frac{q}{\sqrt{gy_1^3}} \quad \text{and } Fr_2 = \frac{q}{\sqrt{gy_2^3}}$$

Equations 16.21 and 16.22 may be written as

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + 8Fr_1^2}\right] \qquad \text{...(16.23)}$$

and

$$\frac{y_1}{y_2} = \frac{1}{2}\left[-1 + \sqrt{1 + 8Fr_2^2}\right] \qquad \text{...(16.24)}$$

When the conjugate depths are known the energy loss ΔE in a hydraulic jump may be computed as

$$\Delta E = \left(y_1 + \frac{V_1^2}{2g}\right) - \left(y_2 + \frac{V_2^2}{2g}\right)$$

or

$$\Delta E = \left[y_1 + \frac{Q^2}{2g(By_1)^2}\right] - \left[y_2 + \frac{Q^2}{2g(By_2)^2}\right]$$

or

$$\Delta E = \frac{q^2}{2g}\left(\frac{1}{y_1^2} - \frac{1}{y_2^2}\right) - (y_2 - y_1)$$

or

$$\Delta E = \frac{q^2}{2g}\frac{y_2^2 - y_1^2}{(y_1 y_2)^2} - (y_2 - y_1)$$

By substituting for $\left(\frac{q^2}{g}\right)$ from Eq. 16.16 the above expression becomes

$$\Delta E = \frac{1}{4}\frac{y_1 y_2 (y_1 + y_2)(y_2^2 - y_1^2)}{(y_1 y_2)^2} - (y_2 - y_1)$$

which on simplification becomes

$$\Delta E = \frac{(y_2 - y_1)^3}{4y_1y_2} \quad ...(16.25)$$

However, as indicated in Illustrative Example 16.11, the energy loss ΔE in a hydraulic jump in a rectangular channel may also be expressed as

$$\Delta E = \frac{(V_1 - V_2)^3}{2g(V_1 + V_2)} \quad ...(16.25\text{ a})$$

where V_1 and V_2 are the mean velocities of flow before and after the jump respectively.

The *height of the jump* h_j may be defined as the difference between the depths after and before the jump, i.e., $h_j = (y_2 - y_1)$.

The *length of the jump* L_j may be defined as the distance measured from the front face of the jump to a point on the surface immediately downstream from the roller. However, the length of the jump cannot be determined analytically. In addition, practical complications arise from the general instability of the phenomenon and the difficulty of defining the beginning and the end sections of the jump. The length of the jump has been investigated experimentally by many hydraulicians and as a general statement it may be said that for a rectangular channel the length of the jump L_j varies between 5 and 7 times the height of the jump, that is,

$$L_j = (5 \text{ to } 7)\, h_j = (5 \text{ to } 7)\,(y_2 - y_1) \quad ...(16.26)$$

(b) Types of Hydraulic Jump. Equation 16.23 emphasizes the importance of the Froude number Fr_1 of the incoming supercritical flow, as a parameter describing the phenomenon of hydraulic jump. As such according to the studies of U.S. Bureau of Reclamation, depending upon the value of Froude number Fr_1 of the incoming flow, there are five distinct types of the hydraulic jump which may occur on a horizontal floor. These different types of hydraulic jump are shown in Fig. 16.9 and are described below.

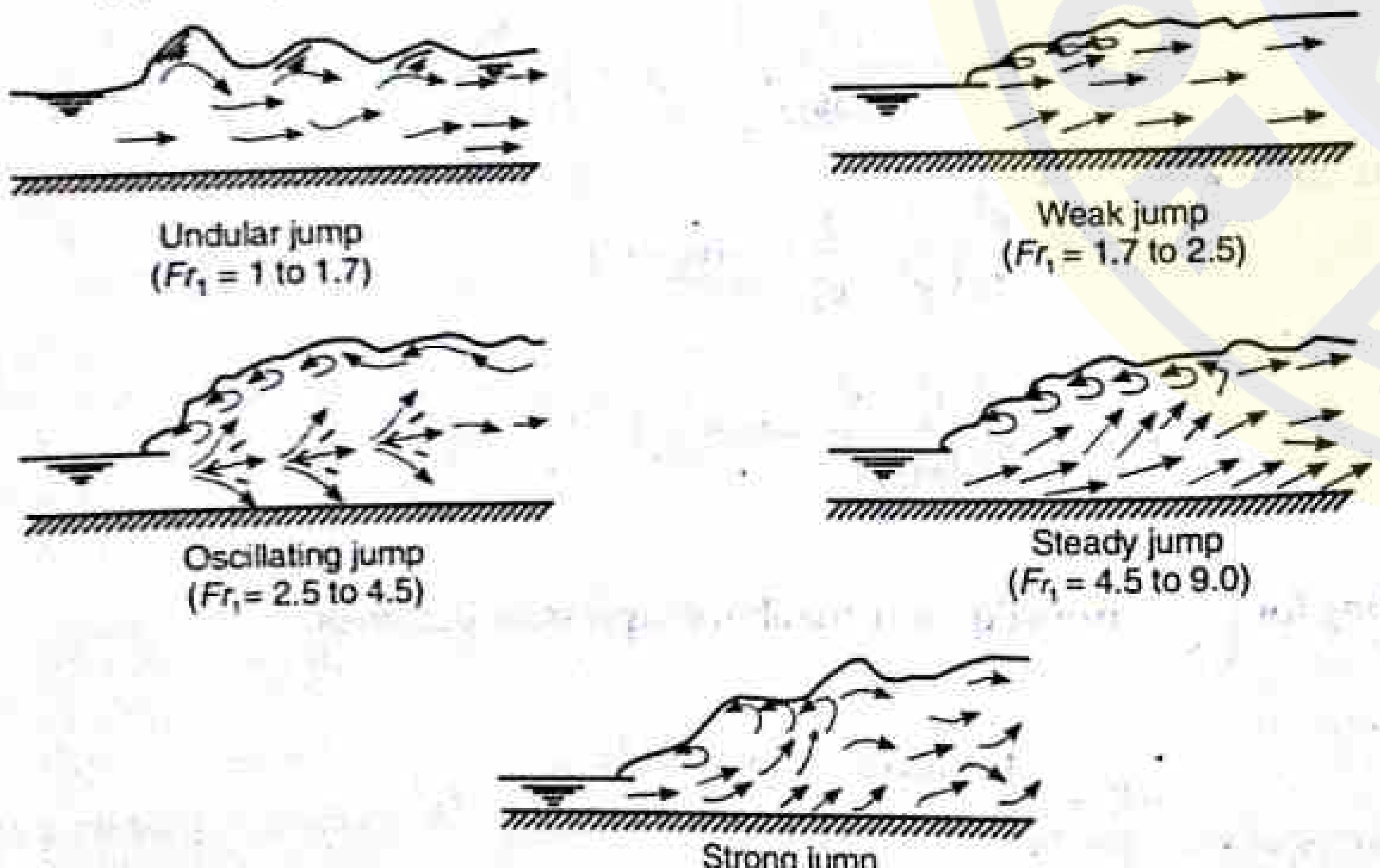


Figure 16.9 *Types of hydraulic jump*

(1) For Fr_1 = 1.0 to 1.7, the water surface shows undulations and the jump is called an *undular jump*.

(2) For Fr_1 = 1.7 to 2.5, the jump formed is called *weak jump*, as the velocity throughout is fairly uniform and only a small amount of energy is dissipated. In this case a series of small rollers form on the jump surface, but the downstream water surface remains quite smooth.

(3) For Fr_1 = 2.5 to 4.5, jump formed is known as an *oscillating jump*. In this case the entering jet of water oscillates back and forth from the bottom to the surface and back again.

(4) For Fr_1 = 4.5 to 9.0 the jump formed is well stabilized and is called a *steady jump*. For this jump the energy dissipation ranges from 45 to 70 per cent.

(5) For Fr_1 = 9.0 and larger the jump formed is called a *strong jump*. In this case a rough surface prevails which continues downstream for a long distance. The jump action is quite rough but is effective since the energy dissipation may reach 85 per cent.

The above described different types of hydraulic jump refer only to channels of rectangular section. In other channel sections the shape of the jump is often complicated additionally by cross currents.

In the above paragraphs only the hydraulic jump in rectangular channel has been discussed, but it may however be mentioned that by using Eq. 16.14 the hydraulic jump in prismatic channel of any shape can be analysed.

(c) Applications of Hydraulic Jump. The phenomenon of hydraulic jump has many practical applications as listed below.

(1) It is a useful means of dissipating excess energy of water flowing over spillways and other hydraulic structures or through sluices and thus preventing possible erosion on the downstream side of these structures.

(2) It raises the water level in the channels for irrigation etc.

(3) It increases the weight on an apron of a hydraulic structure due to increased depth of flow and hence the uplift pressure acting on the apron is considerably counterbalanced.

(4) It increases the discharge through a sluice by holding back the tail water.

(5) It may be used for mixing chemicals in water and other liquids, since it facilitates thorough mixing due to turbulence created in it.

16.8 LOCATION OF HYDRAULIC JUMP

Often it is required to locate the exact position of the hydraulic jump in a channel under different conditions of flow. As such the following three typical cases for the location of the exact position of the hydraulic jump are described below.

Case (1)

In this case a jump forms below a regulating sluice in a mild sloped channel, see Fig. 16.10 (a). The jet of water issuing from the sluice will contract upto vena contracta section at a distance L_c from the sluice which is taken approximately equal to the sluice opening h. Thereafter it will follow M_3 profile as indicated by DE in Fig. 16.10 (a). The location of the jump in this case will be considerably affected by the length of the channel reach on the downstream side of the sluice. Thus if there exists a long reach of channel with same slope on the downstream side of the sluice, then after the formation of the jump, uniform flow with depth of flow equal to the normal depth of flow will be developed. Hence the depth of flow after the jump or the sequent depth will be equal to the normal depth of flow. As such from the vena contracta section the depth of flow will gradually increase, following M_3 profile, upto a certain section on the downstream side, where the depth of flow will be equal to the initial depth required for the formation of the jump corresponding to the sequent depth equal to the normal depth of flow. At this section the jump will be developed and it will extend from this section on the downstream side upto a distance equal to the length of the jump. Therefore, in order to locate the jump section, compute the initial depth corresponding to the sequent depth equal to the normal depth of flow in the channel and obtain the length of the M_3 profile between the vena contracta section and the section having the depth of flow equal to this initial depth. This method has been illustrated in Illustrative Example 16.6.

On the other hand if there exists a short reach of channel on the downstream side of the sluice, with a free overfall or an abrupt drop at its end, then there will be a M_2 profile developed as shown by FG in Fig. 16.10 (a). As such in this case after the formation of the jump the depth of flow will be gradually varying and the sequent depth will lie along the M_2 profile. However, from the vena contracta section the depth of flow will gradually increase following M_3 profile as shown by DE in Fig. 16.10 (a). The jump in this case will be developed at a section where the depth of flow along the M_3 profile will be

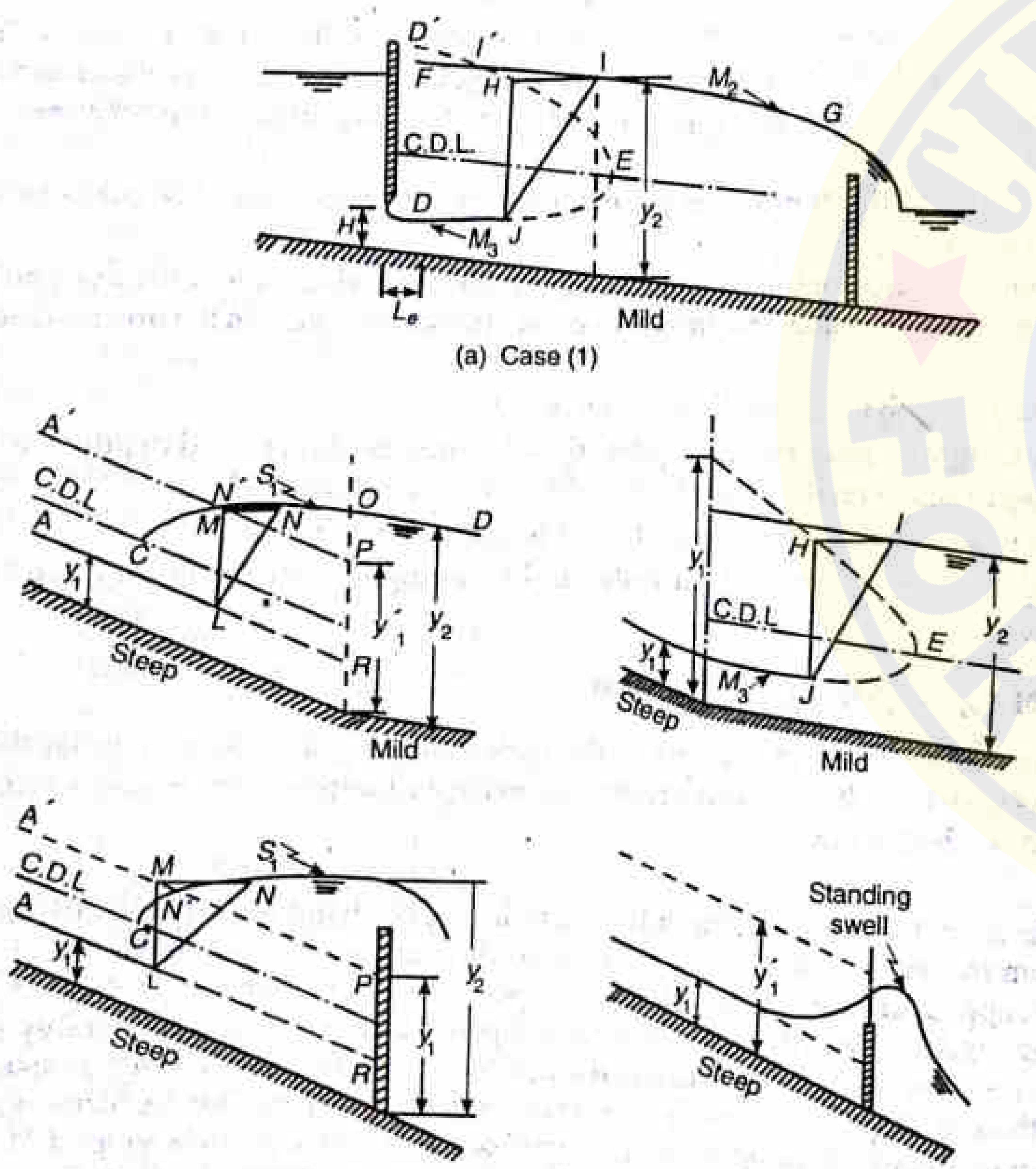


Figure 16.10 *Location of hydraulic jump*

equal to the initial depth required for the formation of the jump corresponding to a sequent depth along the M_2 profile. As such for locating the jump in this case, different values of initial depths are

assumed along the M_3 profile. Corresponding to these initial depths the sequent depths are computed and the same are plotted. The curve $D'E$ shown in Fig. 16.10 (a) is a plot of the depth sequent to curve DE (i.e., M_3 profile). The intersection of M_2 profile i.e., curve FG and the curve $D'E$ at point I' gives the approximate position of the jump if the length of the jump is not taken into account.

However, for taking the length of the jump into account find the initial and sequent depths corresponding to point I' and compute the length of the jump from Eq. 16.26. Now by trial locate a horizontal intercept HI equal to the length of the jump, between the curves $D'E$ and FG. Locate a point J vertically below the point H, on the curve DE. Thus it is obvious that the jump will form between the points J and I, since the depth at I is equal to the sequent depth corresponding to the depth at J and the distance HI is equal to the length of the jump. Thus if the length of the jump is not taken into account, the error involved in the location of the jump is represented by $I'I$.

It can be seen from the above analysis that in the later case the jump can be moved upstream by increasing the downstream depth or raising the curve FG. On the other hand, decreasing the downstream depth or lowering the curve FG will move the jump downstream.

Furthermore, in both the cases discussed above if the depth of flow at the vena contracta section is equal to the initial depth required for the formation of the jump corresponding to the available sequent depth, then the M_3 profile will not be developed and the jump will be located just at the vena contracta section. On the other hand if the depth of flow at the vena contracta section is more than the required initial depth, then no clear jump will be formed but a submerged jump will be formed close to the sluice.

The above discussed method also applies in the case of the location of the jump formed at the foot of an overflow spillway or a weir.

Case (2)

In this case the jump forms in a channel having a break in the bottom slope that changes from steep to mild, see Fig. 16.10 (b). It is assumed that the flow is uniform in the channel section except in the reach between the jump and the change of slope.

Now if the normal depth in the mild channel is greater than the depth y_1' sequent to the normal depth in the steep channel, then the jump will form in the steep channel. Thus compute the profile S_1 in the steep channel as shown by OC in Fig. 16.10 (b). The S_1 profile starts from the point O at the section where the change in slope occurs. Compute the depth y_1' sequent to the normal depth in the steep channel, and the line $A'P$ is drawn parallel to the bottom slope of the steep channel, at distance y_1'. Then the intersection of S_1 profile (i.e., curve OC) and the line $A'P$ at point N' gives the approximate position of the jump if the length of the jump is not taken into account. On the other hand if the length of the jump is to be taken into account, then compute the length of the jump using Eq. 16.26, taking initial depth as equal to the normal depth in the steep channel and sequent depth as obtained above. Now by trial and error locate horizontal intercept MN equal to the length of the jump, between the line $A'P$ and the curve OC. Locate the point L vertically below the point M, on the normal depth line in the steep channel. Thus it is obvious that the jump will be formed between the points L and N since the depth at N is equal to the sequent depth corresponding to the depth at L and the distance MN measures the length of the jump.

On the other hand if the normal depth in the mild channel is less than the depth y_1' sequent to the normal depth in the steep channel, then the jump will form in the mild channel. This case thus becomes similar to case (1) and hence the jump can be located by the same procedure as indicated in case (1).

Case (3)

In this case the jump forms behind an overflow barrier provided in a steep channel. Theoretically, in this case the jump will form if the depth at the barrier is greater than the depth y_1' sequent to the approaching normal depth in the steep channel. The location of the jump in this case is similar to the case (2) when the jump forms in the steep portion. When the depth at barrier is less than the sequent depth y_1' there will be no jump and the barrier will be crossed by a standing swell.

16.9 SURGES IN OPEN CHANNELS

A surge or surge wave is a moving wave front which brings about an abrupt change in depth of flow. A surge is also often referred to as moving hydraulic jump and is caused by sudden increase or decrease in flow, such as that caused by sudden opening or closing of a gate fixed in the channel. Surges are usually classified as positive surges and negative surges. A positive surge is one which results in an increase in the depth of flow and a negative surge causes a decrease in the depth of flow. Figures 16.11 (a) and (b) shows two types of positive surges and Figs. 16.11 (c) and (d) shows two types of negative surges. Type A (Fig. 16.11 a) is a positive surge having an advancing wave front moving downstream. Type B (Fig. 16.11 b) is also a positive surge having an advancing wave front moving upstream. Type C (Fig. 16.11 c) is a negative surge having a retreating wave front moving downstream. Type D (Fig. 16.11 d) is also a negative surge having a retreating wave front moving upstream. The

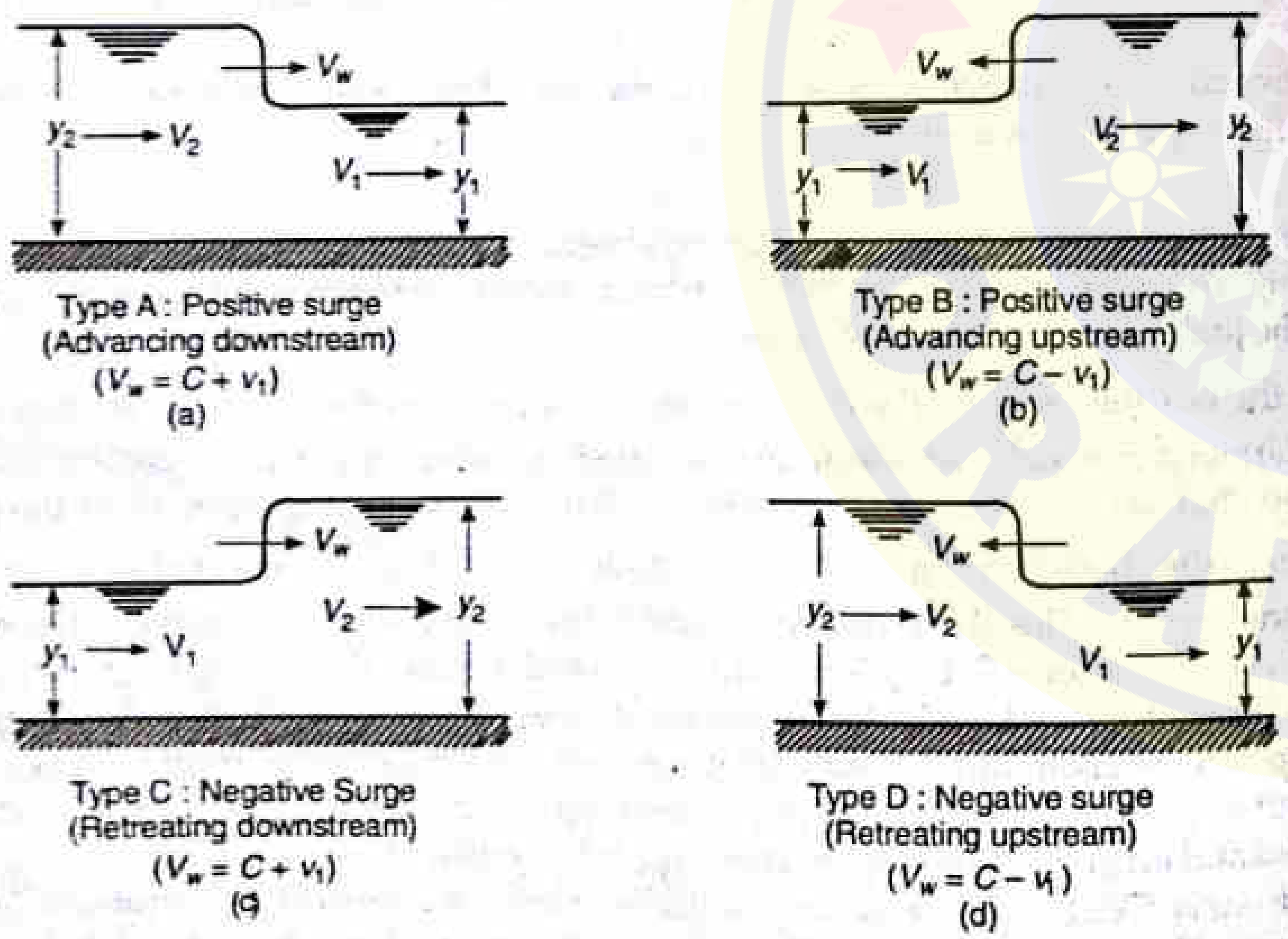


Figure 16.11 *Different types of surges.*

positive surge of type A may occur when a gate provided at the head of a channel is suddenly opened. The positive surge of the type B may occur when a gate provided at the tail end of a channel is suddenly closed. The negative surge of type C may occur when a gate provided at the head of a channel is suddenly closed. The negative surge of type D may occur when a gate provided at the tail end of a channel is suddenly opened.

Although the occurrence of a surge is an unsteady flow phenomenon, but when the surge is moving at a constant velocity, it can be converted into a case of a steady flow by applying a velocity of the same magnitude but in opposite direction to the flowing stream as well as to the surge.

Positive Surges

Case (a) Surge due to sudden increase of flow

Consider a positive surge of type A created in a rectangular channel by suddenly opening a gate as shown in Fig. 16.12. Let V_w be the absolute velocity of the surge moving towards right and let V_1 and V_2 be the velocities and y_1 and y_2 be the corresponding depths at sections 1 and 2 respectively. In order to make it a case of steady flow, apply velocity V_w in opposite directions to the velocities V_1 and V_2 and the surge. Thus the velocities at sections 1 and 2 becomes $(V_1 - V_w)$ and $(V_2 - V_w)$ respectively. Then from continuity, we have

$$y_2(V_2 - V_w) = y_1(V_1 - V_w)$$

from which solving for V_w, we have

$$V_w = \frac{V_1 y_1 - V_2 y_2}{y_1 - y_2} \quad ...(16.27)$$

and solving for V_2, we have

$$V_2 = \frac{y_1(V_1 - V_w) + y_2 V_w}{y_2} \quad ...(16.28)$$

Further due to the surge developed there is a change in the momentum caused between the sections 1 and 2 and hence the momentum equation may be expressed as

$$\frac{1}{2}wy_2^2 - \frac{1}{2}wy_1^2 = \frac{w}{g}\, y_2(V_2 - V_w)[(V_1 - V_w) - (V_2 - V_w)]$$

or

$$\frac{1}{2}(y_2^2 - y_1^2) = \frac{1}{g}\, y_2(V_2 - V_w)(V_1 - V_2) \quad ...(16.29)$$

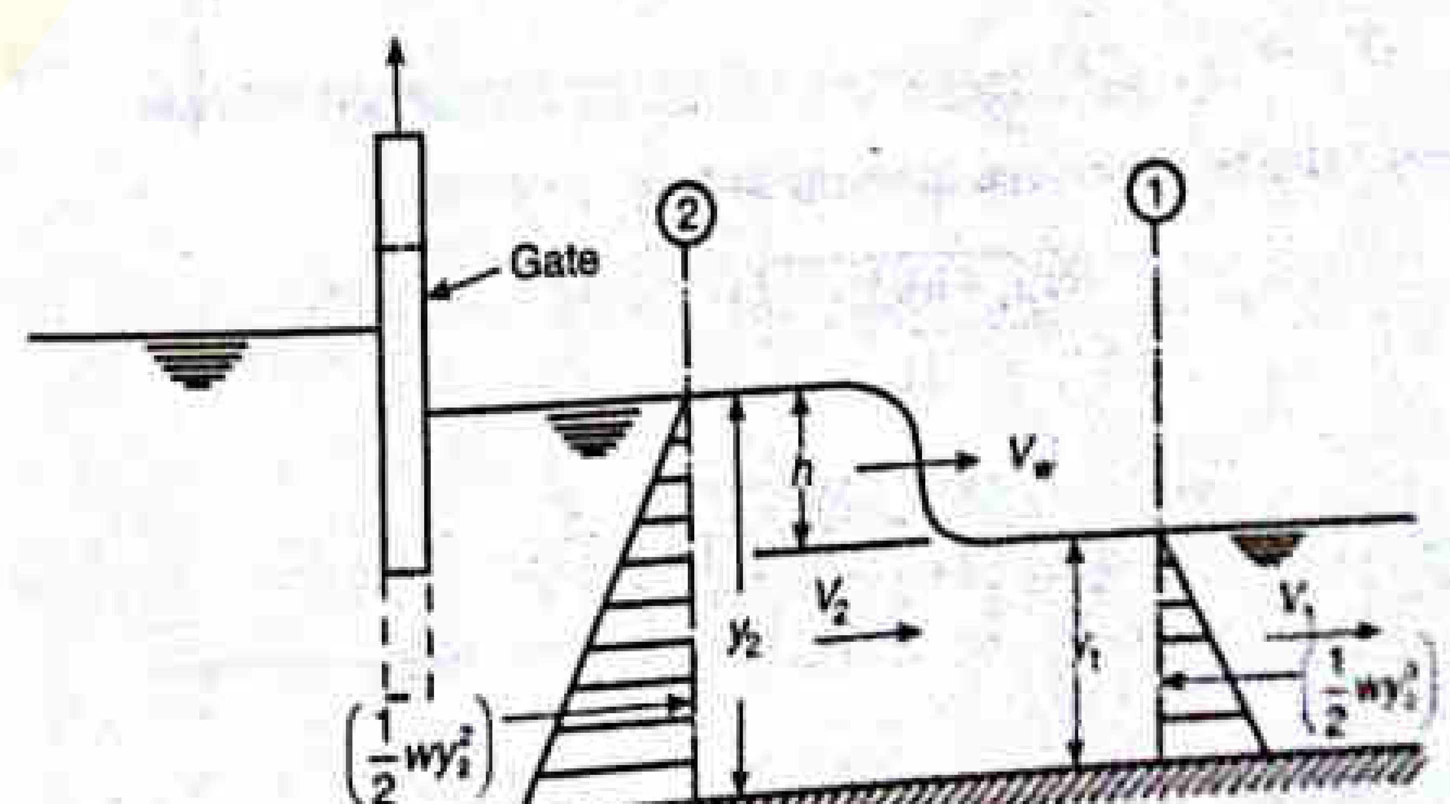


Figure 16.12 *Positive surge of type A in a rectangular channel*

Substituting the value of V_2 from Eq. 16.28 in Eq. 16.29 and simplifying, we get

$$(V_w - V_1)^2 = \frac{g y_2}{2y_1}(y_2 + y_1) \qquad ...(16.30)$$

or

$$V_w = \left[\sqrt{\frac{g y_2}{2y_1}(y_2 + y_1)}\right] + V_1 \qquad ...(16.31)$$

Eliminating V_w from Eqs 16.27 and 16.31 and simplifying, we have

$$(V_1 - V_2)^2 = (y_1 - y_2)^2 \frac{(y_1 + y_2)g}{2y_1 y_2} \qquad ...(16.32)$$

Equation 16.32 represents the relationship among the initial and final velocities and depth of the surge.

Further Eq. 16.31 represents the absolute velocity of the surge. The celerity of the wave C is defined as the velocity of the wave relative to the velocity of flow and hence $C = (V_w - V_1)$.

$$\therefore \quad C = \sqrt{\frac{g y_2}{2y_1}(y_2 + y_1)} \qquad ...(16.33)$$

Multiplying Eq. 16.32 by the square of Eq. 16.33 and simplifying, we have

$$(V_1 - V_2) = \frac{(y_2 - y_1)}{C}\left(\frac{y_2 + y_1}{2y_1}\right)g$$

or

$$(V_1 - V_2) = \frac{h}{C}\left(\frac{y_2 + y_1}{2y_1}\right)g \qquad ...(16.34)$$

where $h = (y_2 - y_1)$ is the height of the surge.

When the height of the surge is small compared with the depth of flow, then $y_1 \approx y_2$ and Eq. 16.34 may be written as

$$(V_1 - V_2) = \frac{hg}{C} \qquad ...(16.35)$$

Equation 16.33 may also be expressed in terms of h as

$$C = \sqrt{\frac{g(y_1 + h)(2y_1 + h)}{2y_1}}$$

or

$$C = \sqrt{g y_1}\sqrt{1 + \frac{3}{2}\left(\frac{h}{y_1}\right) + \frac{1}{2}\left(\frac{h}{y_1}\right)^2} \qquad ...(16.36)$$

If h is less than y_1, the term $\frac{1}{2}\left(\frac{h}{y_1}\right)^2$ becomes much small and hence it can be neglected. Equation 16.36 then simplifies to

$$C = \sqrt{g y_1}\sqrt{1 + \frac{3}{2}\left(\frac{h}{y_1}\right)}$$

or

$$C = \sqrt{g y_1}\left(1 + \frac{3}{4}\frac{h}{y_1}\right) \qquad ...(16.37)$$

Again when the ratio $\left(\frac{h}{y_1}\right)$ is very small, the term $\left(\frac{3}{4}\frac{h}{y_1}\right)$ can also be neglected and then Eq. 16.33 reduces to

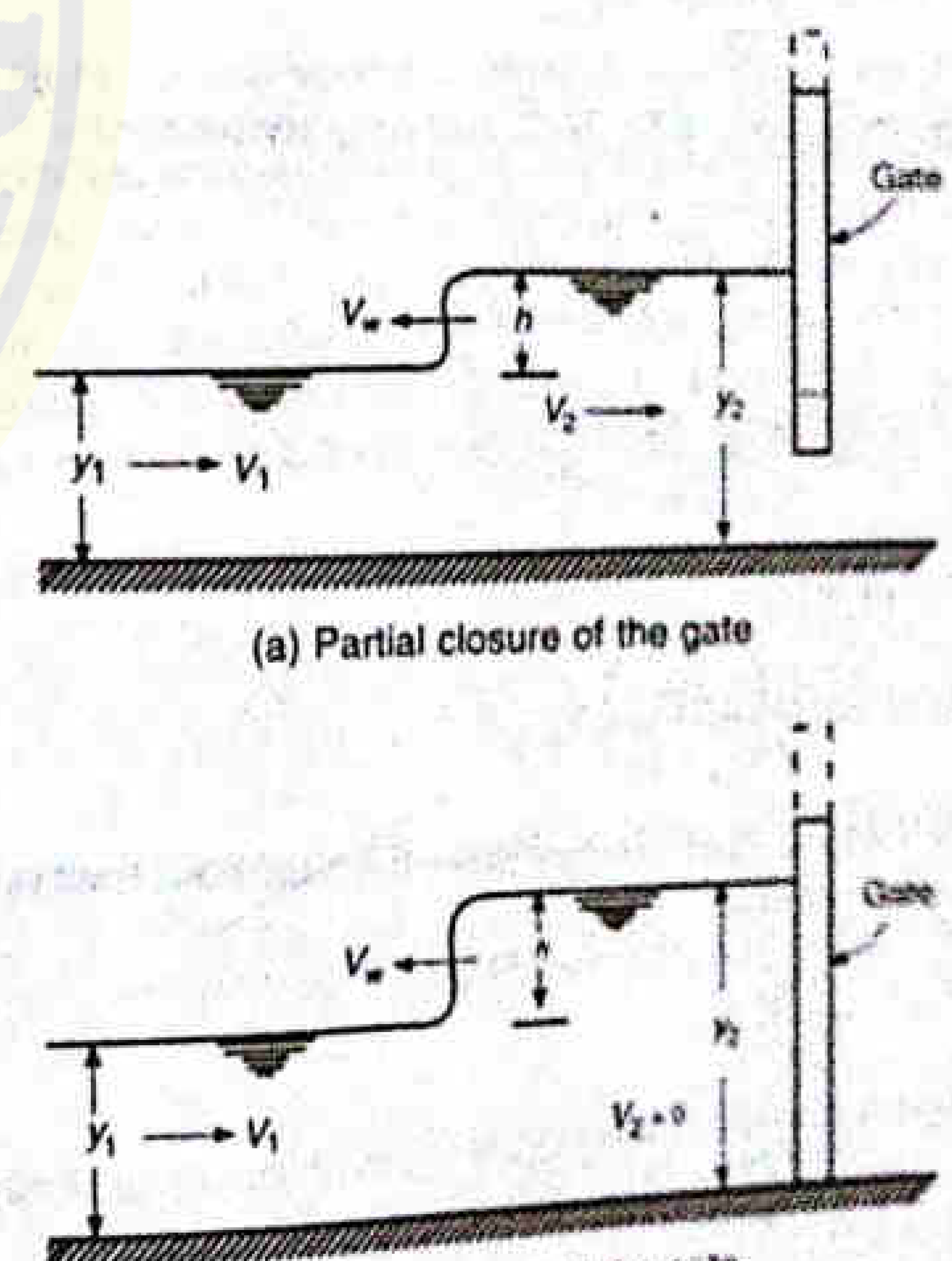
$$C = \sqrt{g y_1} \qquad ...(16.38)$$

The above simplification was suggested by Lagrange and hence Eq. 16.38 is known as 'Lagrange's celerity equation'.

As indicated earlier a surge is often termed as moving hydraulic jump. However, in the case of a stationary hydraulic jump, the absolute velocity V_w is zero. Hence by substituting $V_w = 0$ in Eq. 16.30 and simplifying it, Eq. 16.16 may be obtained. As such a stationary hydraulic jump may be considered as a special case of a surge.

Case (b) Surge due to sudden reduction of flow

Consider a positive surge of type *B* created in a rectangular channel by sudden reduction of flow due to partial or complete closing of a gate. If V_w is the absolute velocity of the surge moving towards left



(a) Partial closure of the gate

(b) Total closure of the gate

Figure 16.13 Positive surge of type *B* in a rectangular channel

(Fig. 16.13 a) then in order to make it a case of steady flow apply a velocity V_w in the direction opposite to that of surge. The velocities at sections 1 and 2 will then be $(V_1 + V_w)$ and $(V_2 + V_w)$ respectively. The continuity equation in this case is

$$y_1(V_1 + V_w) = y_2(V_2 + V_w) \quad \text{...(16.39)}$$

and the momentum equation is

$$\frac{1}{2}w\left(y_2^2 - y_1^2\right) = \frac{w}{g}\, y_2(V_2+V_w)\,[(V_1+V_w)-(V_2+V_w)] \quad \text{...(16.40)}$$

If the flow is completely stopped (Fig. 16.13 b) the velocity $V_2 = 0$. The above equations will then be reduced to

$$y_1(V_1 + V_w) = y_2 V_w \quad \text{...(16.39 a)}$$

and

$$\frac{1}{2}w\left(y_2^2 - y_1^2\right) = \frac{w}{g}\, y_2 \times V_w V_1 \quad \text{...(16.40 a)}$$

Eliminating V_w from Eqs 16.39 (a) and 16.40 (a) and simplifying, we have

$$V_1^2 = (y_2 - y_1)^2 \frac{(y_1 + y_2)g}{2y_1 y_2} \quad \text{...(16.41)}$$

In this case the celerity of wave $C = (V_1 + V_w)$, and from Eq. 16.39 (a) and 16.41, we have

$$C = \sqrt{\frac{g y_2}{2y_1}(y_1 + y_2)} \quad \text{...(16.42)}$$

Multiplying Eq. 16.41 by the square of Eq. 16.42 and simplifying, we have

$$V_1 = \frac{(y_2 - y_1)}{C}\left(\frac{y_2 + y_1}{2y_1}\right) g$$

or

$$V_1 = \frac{h}{C}\left(\frac{y_2 + y_1}{2y_1}\right) g \quad \text{...(16.43)}$$

Thus the height of the surge is

$$h = \frac{C}{g}\left(\frac{2y_1}{y_1 + y_2}\right) V_1 \quad \text{...(16.44)}$$

Again if the height of the surge is small retative to the depth of flow, then $y_1 \approx y_2$, and Eqs 16.43 and 16.44 may be approximately written as

$$V_1 = \frac{hg}{C} \quad \text{...(16.45)}$$

and

$$h = \frac{V_1 C}{g} \quad \text{...(16.46)}$$

Negative Surges

Negative surges are not stable in form because as indicated below the upper portions of the wave travel faster than the lower portions. As shown in Fig. 16.14 if y_2 is the depth below the top $a_1\, b_1$ of the wave and y_1 is the depth below the bottom or trough $c_1\, d_1$ of the wave, then since celerity $C = \sqrt{gy}$, its value at the top $a_1\, b_1$ of the wave is $C_2 = \sqrt{gy_2}$ and at the bottom $c_1\, d_1$ of the wave is $C_1 = \sqrt{gy_1}$; and C_2 is greater than C_1. Thus if the initial profile of the surge is assumed to have a steep front, it will soon flatten out as the surge moves through the channel. If the height of the surge is moderate or small compared with the depth of flow, the equations derived for a positive surge can be applied to determine approximately the propagation of the negative surge. However, if the height of the surge is relatively large, a more elaborate analysis as indicated below may be adopted.

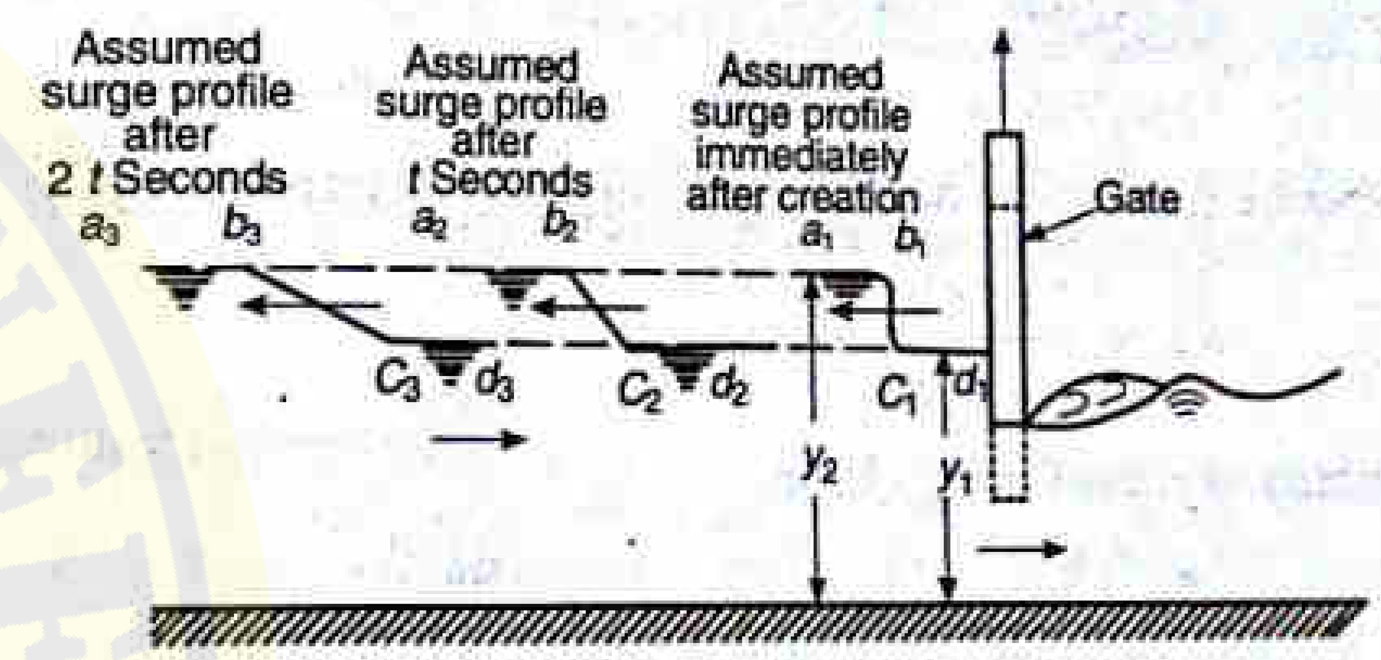


Figure 16.14 *Propagation of a negative surge due to sudden lift of a gate*

Figure 16.15 shows a negative surge of type D of relatively large height retreating in upstream direction in a rectangular channel. The surge is caused by the sudden lifting of a gate. In this case the velocity of the surge actually varies from point to point. Let V_w be the velocity of the surge at a point on its surface where the depth is y and the velocity of flow through the section is V. During a time interval dt, let the change in y be dy. The value of dy is positive for an increase of y and negative for a decrease of y. Assuming that the rate of change in y varies uniformly from O to dy in time dt, the average change is $\frac{dy}{2}$. The mass of water that moves per unit width of the channel is equal to $\frac{w}{g}\left(y + \frac{dy}{2}\right)$ $[V_w - (-\frac{w}{g}\left(y + \frac{dy}{2}\right))] = \frac{w}{g}\left(y + \frac{dy}{2}\right)(V_w + V)$, as V is in a direction opposite to V_w. If the change in velocity is dV, then the rate of change of momentum of the vertical element between y and $(y + dy)$ is equal to $\left[\frac{w}{g}\left(y + \frac{dy}{2}\right)(V_w + V)dV\right]$. By the momentum principle this rate of change of momentum is equal to the corresponding change in hydrostatic pressure which is equal to $\left[\frac{w}{2}y^2 - \frac{w}{2}(y + dy)^2\right]$.

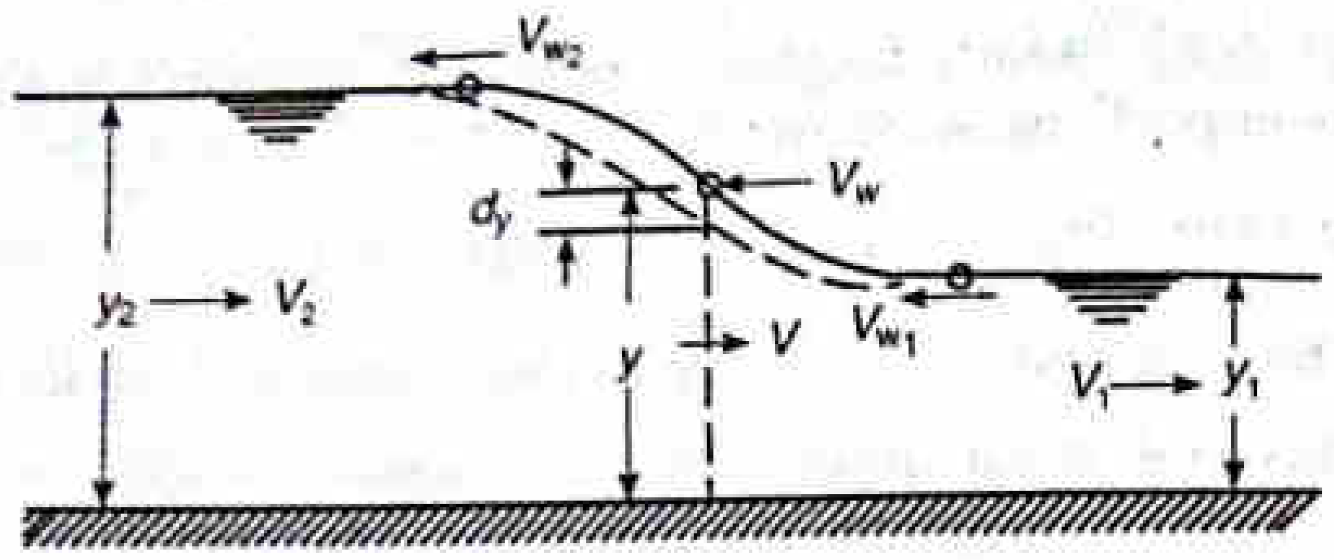


Figure 16.15 *Analysis of a negative surge*

Thus

$$\frac{w}{2}y^2 - \frac{w}{2}(y+dy)^2 = \frac{w}{g}\left(y+\frac{dy}{2}\right)(V_w+V)dV \quad ...(16.47)$$

Simplifying the above equation and neglecting the differential terms of higher order, we get

$$dy = -\frac{(V+V_w)dV}{g} \quad ...(16.48)$$

In this case the celerity of wave $C = V_w + V$, or

$$V_w = C - V$$

or

$$V_w = \sqrt{gy} - V \quad ...(16.49)$$

Thus the velocity at the crest (or top) of the surge wave is

$$V_{w_2} = \sqrt{gy_2} - V_2 \quad ...(16.50)$$

and at the trough (or bottom) of the surge wave is

$$V_{w_1} = \sqrt{gy_1} - V_1 \quad ...(16.51)$$

When the surge is not too high, a straight-line relation between V_{w1} and V_{w2} may be assumed. Thus the mean velocity of the surge wave may be considered to be

$$V_m = \frac{V_{w_1}+V_{w_2}}{2} \quad ...(16.52)$$

Eliminating V_w between Eqs 16.48 and 16.49, we get

$$\frac{dy}{\sqrt{y}} = -\frac{dV}{\sqrt{g}} \quad ...(16.53)$$

Integrating Eq. 16.53 from y_2 to y and V_2 to V, and solving for V, we get

$$V = V_2 + 2\sqrt{gy_2} - 2\sqrt{gy} \quad ...(16.54)$$

From Eq. 16.49, we have

$$V_w = 3\sqrt{gy} - 2\sqrt{gy_2} - V_2 \quad ...(16.55)$$

Thus the velocity V_{w1} at the trough (or bottom) of the surge wave is

$$V_{w_1} = 3\sqrt{gy_1} - 2\sqrt{gy_2} - V_2 \quad ...(16.56)$$

The velocity V_{w2} at the crest (or top) of the surge wave is

$$V_{w_2} = \sqrt{gy_2} - V_2$$

which is same as given by Eq. 16.50.

Further let t be the time elapsed since the surge was created or, in this case since the gate was opened. At $t = 0$, the wavelength $\lambda = 0$. After t seconds the wavelength is equal to

$$\lambda = (V_{w_2} - V_{w_1})t \quad ...(16.57)$$

The above analysis can also be similarly applied to a negative surge of type C.

ILLUSTRATIVE EXAMPLES

Example 16.1. *A rectangular channel 7.5 m wide has a uniform depth of flow of 2.0 m and has a bed slope of 1 in 3000. If due to weir constructed at the downstream end of the channel, water surface at a section is raised by 0.75 m, determine the water surface slope with respect to horizontal at this section. Assume Manning's n = 0.02.*

Solution

According to Manning's formula, we have

$$Q = \frac{1}{n}AR^{2/3}S_0^{1/2}$$

or

$$Q = \frac{1}{0.02}(7.5\times 2.0)\left[\frac{(7.5\times 2.0)}{7.5+(2\times 2.0)}\right]^{2/3}\left(\frac{1}{3000}\right)^{1/2}$$

or

$$Q = 16.35 \text{ m}^3/\text{s}$$

From Eq. 16.1, we have

$$\frac{dy}{dx} = \frac{S_0 - S_f}{1-\frac{Q^2T}{gA^3}}$$

$$S_0 = \frac{1}{3000} = 3.333\times 10^{-4}$$

$$S_f = \frac{n^2V^2}{R^{4/3}}$$

For

$$y = (2.0+0.75) = 2.75 \text{ m}$$
$$A = 20.625 \text{ m}^2$$
$$P = 13 \text{ m}$$
$$R = 1.5865 \text{ m}$$
$$V = 0.793 \text{ m/sec}$$

$\therefore$

$$S_f = 1.359\ 10^{-4}$$

$$\frac{Q^2T}{gA^3} = \frac{(16.35)^2 \times 7.5}{9.81 \times (20.625)^3}$$

$$= 0.0233$$

Thus
$$\frac{dy}{dx} = \frac{(3.333 - 1.359)\times 10^{-4}}{(1 - 0.0233)}$$

$$= 2.021 \times 10^{-4}$$

The water surface slope S_w with respect to horizontal is given by

$$S_w = S_0 - \frac{dy}{dx} = (3.333 - 2.021) \times 10^{-4}$$

$$= 1.312 \times 10^{-4}.$$

Example 16.2. *A rectangular channel 10 m wide carries a discharge of 30 m³/s. It is laid at a slope of 0.0001. If at a section in this channel the depth is 1.6 m, how for (upstream or downstream) from the section will the depth be 2.0 m? Take Manning's n as 0.015.*

Solution

If y_n is the normal depth of flow then

$$A = 10y_n$$

and
$$R = \frac{10y_n}{10 + 2y_n}$$

The velocity of flow is

$$V = \frac{30}{10y_n} = \frac{3}{y_n}$$

Using Manning's formula

$$V = \frac{1}{n} R^{2/3} S^{1/2}$$

or
$$\frac{3}{y_n} = \frac{1}{0.015} \times \left(\frac{10y_n}{10 + 2y_n}\right)^{2/3} \times (0.0001)^{1/2}$$

or
$$1.909(5 + y_n) = y_n^{5/2}$$

Solving the equation for y_n by trial

$$y_n = 2.97 \text{ m}$$

The critical depth y_c is given by

$$y_c = \left[\frac{q^2}{g}\right]^{1/3} = \left[\frac{(30/10)^2}{9.81}\right]^{1/3} = 0.972 \text{ m}$$

TABLE FOR EXAMPLE 16.2

y m	A m²	R m	V m/s	$\frac{V^2}{2g}$ m	E m	ΔE m	S_f	$\bar{S}_f$	$S_0 - \bar{S}_f$	Δx
1.6	16	1.21	1.88	0.179	1.779		0.000617			
						0.080		0.000562	0.000462	173.16
1.7	17	1,27	1.76	0.159	1.859		0.000507			
						0.083		0.000470	0.000370	224.32
1.8	18	1.32	1.67	0.142	1.942		0.000433			
						0.085		0.000400	0.000300	283.33
1.9	19	1.38	1.58	0.127	2.027		0.000366			
						0.088		0.000340	0.000240	366.67
2.0	20	1.43	1.50	0.115	2.115		0.000314			

The required distance between the two sections is

$$\Sigma(\Delta x) = (173.16 + 224.32 + 283.33 + 366.67) = 1\,047.48 \text{ m}$$

Since $y_n > y_c$, it is a mild-sloped channel. Further the given depths of 1.6 m and 2.0 m are between y_n and y_c hence the surface profile lies in zone (2) or the surface profile will be M_2 type curve. As such the depth of 2.0 m will be found upstream from that of 1.6 m.

Example 16.3. *A trapezoidal channel having bottom width 6 m, side slope 2 horizontal to 1 vertical, Manning's roughness coefficient 0.025, and bottom slope 0.0016, carries a discharge of 10 m³/s. Compute the back water profile created by a dam which backs up the water to a depth of 2.0 m immediately behind the dam. Use the direct step method for computation.*

Solution

If y_n is the normal depth of flow, then according to Manning's formula

$$10 = \frac{1}{0.025}(6 + 2y_n)\, y_n \times \left[\frac{(6 + 2y_n)y_n}{6 + (2\sqrt{5})y_n}\right]^{2/3} (0.0016)^{1/2}$$

or
$$16.57 + 12.35y_n = [(3 + yn)y_n]^{5/2}$$

Solving by trial,

$$y_n = 0.963 \text{ m}$$

Further if y_c is the critical depth then

$$\frac{(10)^2}{9.81} = \frac{[(6 + 2y_c)y_c]^3}{6 + 4y_c}$$

or
$$10.19\,(6 + 4y_c) = [(6 + 2y_c)y_c]^3$$

Solving by trial,

$$y_c = 0.611 \text{ m.}$$

Since in this case y_n is greater than y_c it is a mild sloped channel and hence the profile is of the M_1

type. For the computation of the length of the surface profile the various steps are shown in Table below

TABLE FOR EXAMPLE 16.3

y m	A m^2	R m	V m/s	$\frac{V^2}{2g}$ m	E m	ΔE m	S_f	$\bar{S}_f$	$S_0-\bar{S}_f$	Δx
2.0	20.00	1.34	0.55	0.013	2.013		0.106×10^{-3}			
						0.196		0.204×10^{3}	1.467×10^{-3}	133.61
1.80	17.28	1.23	0.58	0.017	1.817		0.160×10^{-3}			
						0.193		0.204×10^{-3}	1.396×10^{-3}	138.25
1.60	14.72	1.12	0.68	0.024	1.624		0.248×10^{-3}			
						0.191		0.329×10^{-3}	1.271×10^{3}	150.28
1.40	12.32	1.00	0.81	0.033	1.433		0.410×10^{-3}			
						0.183		0.563×10^{-3}	1.037×10^{-3}	176.47
1.20	10.08	0.89	0.99	0.050	1.250		0.716×10^{-3}			
						0.170		1.062×10^{-3}	0.538×10^{-3}	315.99
1.00	8.00	0.76	1.25	0.080	1.080		1.408×10^{-3}			
						0.024		1.493×10^{-3}	0.107×10^{-3}	224.30
0.97	7.70	0.74	1.30	0.086	1.056		1.578×10^{-3}			

The length of the water surface profile is

$$\Sigma(\Delta x) = 1138.90 \text{ m.}$$

Note. The upstream end of the water surface profile is assumed at a depth equal to 1% greater than the normal depth and hence the depth at the upstream end of the profile has been considered as $(1.01 \times 0.963) = 0.97$ m.

Example 16.4. *Solve example 16.3 by Bresse's method.*

Solution

According to Bresse's method the length of the surface profile is given by Eq. 16.13 as

$$x = \frac{y_n}{S_0}\left[(u_2-u_1)-\left(1-\frac{C^2S_0}{g}\right)\{\phi(u_2)-\phi(u_1)\}\right]$$

where $\phi(u) = \int \frac{du}{1-u^3}$ and $u = \left(\frac{y}{y_n}\right)$

$y_n = 0.963$ m ; and $y_c = 0.611$ m

The depth at the upstream end of the profile is $(1.01\times 0.963) = 0.97$ m and at the downstream end the depth is 2.0 m.

As the depth of flow varies from 0.97 m to 2.0 m, an average value of $y = 1.485$ m may be assumed for the evaluation of Chezy's C.

Chezy's C may be expressed as

$$C = \frac{1}{n} R^{1/6}$$

For $y = 1.485$ m; $R = 1.054$ m

Thus
$$C = \frac{1}{0.025}(1.054)^{1/6} = 40.35$$

Hence
$$\frac{C^2S_0}{g} = \frac{(40.35)^2(0.0016)}{9.81} = 0.266$$

and
$$\left(1-\frac{C^2S_0}{g}\right) = 0.734$$

The values of u and $\phi(u)$ may be computed for the two extreme sections of the profile and the same are given in the following table.

y	u	$\phi(u)$
2.00	2.08	0.122
0.97	1.01	1.419

The values of the varied flow function $\phi(u)$ as given in the third column of the above table may be computed either by Eq. 16.12 or the same may be directly obtained from the Bakhmeteff's varied flow function tables.

By substituting the values in the above expression

$$x = \frac{0.963}{0.0016}[(2.08-1.01)-0.734\,(0.122-1.419)]$$

or
$$x = 1217 \text{ m.}$$

The length of the water surface profile obtained by the Bresse's method is about 7% more than the value obtained in Example 16.3. This is so because Bresse's method is derived primarily for a wide rectangular channel. As such when this method is applied to channels of other cross-sectional shapes, the solution is quite approximate.

Example 16.5. *A rectangular flume 2 m wide carries discharge at the rate of 2 m³/s. The bed slope of the flume is 0.0004. At a certain section the depth of flow is 1 m. Calculate the distance of the section downstream where the depth of flow is 0.9 m. Solve by single step method. Assume rugosity coefficient as 0.014. Is the slope of the channel mild or steep? How is this type of surface profile classified?*

Solution

If y_n is the normal depth of flow, then according to Manning's formula

$$2 = \frac{1}{0.014}\times(2\times y_n)\times\left(\frac{2y_n}{2+2y_n}\right)^{2/3}(0.0004)^{1/2}$$

or $$0.5857\,(1+y_n) = y_n^{5/2}$$

Solving by trial

$$y_n = 1.082 \text{ m}$$

Further if y_c is the critical depth, then

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left[\frac{(2/2)^2}{9.81}\right]^{1/3} = 0.467 \text{ m.}$$

Since in this case y_n is greater than y_c, the bed slope of the flume is mild. Further the given depths of flow viz., 1.0 m and 0.90 m are between y_n and y_c, the surface profile lies in zone (2) and it is M_2 type curve.

The length of the surface profile between the sections where the depths of flow are 1.0 m and 0.90 m, may be computed by step method as follows.

At section 1 where the depth of flow is 1 m, the specific energy is

$$E_1 = 1.0 + \left[\frac{\{2/(2\times1)\}^2}{2\times9.81}\right] = 1.051 \text{ N.m/N}$$

Similarly at section 2 where the depth of flow is 0.90 m, the specific energy is

$$E_2 = 0.90 + \left[\frac{\{2/(2\times0.9)\}^2}{2\times9.81}\right] = 0.963 \text{ N.m/N}$$

$$\therefore \quad \Delta E = (E_2 - E_1) = (0.963 - 1.051)$$
$$= -0.088 \text{ N.m/N}$$

According to Manning's formula at section 1

$$S_{f1} = \frac{V^2n^2}{R^{4/3}} = \frac{\{2/(2\times1)\}^2\times(0.014)^2}{\left(\dfrac{2\times1}{2+2\times1}\right)^{4/3}} = 4.94\times10^{-4}$$

Similarly at section 2

$$S_{f2} = \frac{\{2/(2\times0.9)\}^2\times(0.014)^2}{\left(\dfrac{2\times0.90}{2+2\times0.90}\right)^{4/3}} = 6.55\times10^{-4}$$

$$\therefore \quad \bar{S}_f = \frac{S_{f1}+S_{f2}}{2} = \frac{(4.94\times10^{-4}+6.55\times10^{-4})}{2}$$
$$= 5.745\times10^{-4}$$

From Eq. 16.8 the distance between sections 1 and 2 is

$$x = \frac{\Delta E}{S_0 - \bar{S}_f} = \frac{-0.088}{(4\times10^{-4} - 5.745\times10^{-4})}$$
$$= 504.3 \text{ m.}$$

Example 16.6. *Water flows from an under sluice into a very wide rectangular channel. The channel has a bed slope of 1 in 1000. The sluice is regulated to discharge 6 m³/s per m width of channel, the depth at venacontracta being 0.5 m. Will a hydraulic jump form? If so determine its location. Use a single step for the computation with Manning's n = 0.015.*

Solution

For a wide rectangular channel $R \approx y$

Thus if y_n is the normal depth of flow then according to Manning's formula

$$\frac{6}{y_n} = \frac{1}{0.015}\times y_n^{2/3}\times\left(\frac{1}{1000}\right)^{1/2}$$

or $$y_n = 1.873 \text{ m}$$

The critical depth y_c is given by

$$y_c = \left(\frac{q^2}{g}\right)^{1/3} = \left(\frac{36}{9.81}\right)^{1/3} = 1.542 \text{ m}$$

Since in this case $y_n > y_c$, the bed slope of the channel is mild. Further the depth of flow at vena contracta is 0.5 m, which is less than the critical depth. As such in this case hydraulic jump will form. The jump will be located at a section where the depth of flow will be conjugate to the normal depth of flow y_n equal to 1.873 m. For this the depth of flow will increase from 0.5 m at vena contracta to this conjugate depth along M_3 type surface profile.

From Eq. 16.20 the depth y_1 conjugate to the normal depth y_n equal to 1.873 m is obtained as

$$\frac{y_1}{1.873} = \frac{1}{2}\left[-1+\sqrt{1+\frac{8\times(6)^2}{9.81\times(1.873)^3}}\right]$$

or $$y_1 = 1.253 \text{ m}$$

The distance between vena contracta and the section where the jump will be located may be obtained as follows.

At vena contracta section the specific energy is

$$E_1 = 0.50 + \frac{(6/0.50)^2}{2\times9.81} = 7.839 \text{ N.m/N}$$

Similarly at the section when the depth of flow is 1.253 m, the specific energy is

$$E_2 = 1.253 + \frac{(6/1.253)^2}{2\times9.81} = 2.422 \text{ N.m/N}$$

$$\Delta E = (E_2 - E_1) = (2.422 - 7.839)$$
$$= -5.417 \text{ N.m/N}$$

According to Manning's formula at vena contracta section

$$S_{f1} = \frac{V^2n^2}{R^{4/3}} = \frac{(6/0.50)^2\times(0.015)^2}{(0.50)^{4/3}} = 8.164\times10^{-2}$$

Similarly at the section where the depth of flow is 1.253 m

$$S_{f2} = \frac{(6/1.253)^2 \times (0.015)^2}{(1.253)^{4/3}} = 0.382 \times 10^{-2}$$

$$\therefore \quad \bar{S}_f = \frac{S_{f_1} + S_{f_2}}{2} = \frac{(8.164 + 0.382) \times 10^{-2}}{2}$$

$$= 4.273 \times 10^{-2}$$

From Eq. 16.8 the distance between the vena contracta section and the section where the jump will be located is

$$x = \frac{\Delta E}{S_0 - \bar{S}_f} = \frac{-5.417}{(0.10 \times 10^{-2} - 4.273 \times 10^{-2})}$$

$$= 130 \text{ m}$$

Note. It has been assumed that there exists a long reach of the channel with same slope on the downstream side of the sluice.

Example 16.7. *A horizontal rectangular channel 4 m wide carries a discharge of 16 m³/s. Determine whether a jump may occur at an initial depth of 0.5 m or not. If a jump occurs, determine the sequent depth to this initial depth. Also determine the energy loss in the jump.*

Solution

The jump may occur at an initial depth of 0.5 m if the initial Froude number Fr_1 is greater than one.

$$Fr_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{16}{(4 \times 0.5)\sqrt{9.81 \times 0.5}} = 3.61$$

Since $Fr_1 > 1$, the jump may occur at an initial depth of 0.5 m. From Eq. 16.23

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + 8Fr_1^2}\right]$$

or

$$\frac{y_2}{0.5} = \frac{1}{2}\left[-1 + \sqrt{1 + 8(3.61)^2}\right]$$

or

$$y_2 = 2.32 \text{ m}$$

From Eq. 16.25 the energy loss in the jump is given as

$$\Delta E = \frac{(y_2 - y_1)^3}{4y_1 y_2} = \frac{(2.32 - 0.5)^3}{4(2.32 \times 0.5)} = 1.30 \text{ m.}$$

Example 16.8. *In a rectangular channel there occurs a jump corresponding to $Fr_1 = 2.5$. Determine the critical depth and head loss in terms of the initial depth y_1.*

Solution

For a rectangular channel

$$Fr_1^2 = \frac{q^2}{gy_1^3} = \left(\frac{y_c}{y_1}\right)^3$$

or

$$(2.5)^2 = \left(\frac{y_c}{y_1}\right)^3$$

or

$$y_c = (2.5)^{2/3} y_1 = 1.84\, y_1$$

From Eq. 16.23

$$\frac{y_2}{y_1} = \frac{1}{2}\left[-1 + \sqrt{1 + 8(2.5)^2}\right] = 3.07$$

From Eq. 16.25

$$\frac{\Delta E}{y_1} = \frac{[(y_2/y_1) - 1]^3}{4(y_2/y_1)} = \frac{(2.07)^3}{4 \times (3.07)} = 0.722$$

or

$$\Delta E = 0.722\, y_1.$$

Example 16.9. *A trapezoidal channel having bottom width 8 m and side slope 1:1, carries a discharge of 80 m³/s. Find the depth conjugate to initial depth of 0.75 m before the jump. Also determine the loss of energy in the jump.*

Solution

The area of flow section corresponding to $y_1 = 0.75$ m is

$$A_1 = (B + zy_1)y_1 = [8 + (0.75 \times 1)]0.75 = 6.56 \text{ m}^2$$

and

$$\bar{z}_1 = \frac{y_1}{6}\frac{(3B + 2zy_1)}{(B + zy_1)} = \frac{0.75(24 + 1.5)}{6(8 + 0.75)} = 0.364 \text{ m}$$

If y_2 is the conjugate depth after the jump then the corresponding values of A_2 and $\bar{z}_2$ are given by

$$A_2 = (8 + y_2)y_2$$

and

$$\bar{z}_2 = \frac{y_2}{6}\frac{(24 + 2y_2)}{(8 + y_2)}$$

By substituting the above obtained values in Eq. 16.14, it becomes

$$\frac{(80)^2}{(9.81 \times 6.56)} + (6.56 \times 0.364) = \frac{(80)^2}{9.81(8 + y_2)y_2} + \frac{y_2^2}{6} \times (24 + 2y_2)$$

or

$$101.84 = \frac{652.4}{(8 + y_2)y_2} + \frac{y_2^2}{3}(12 + y_2)$$

Solving the above equation for y_2 by trial and error

$$y_2 = 4.066 \text{ m}$$

The specific energies before and after the jump are

$$E_1 = y_1 + \frac{Q^2}{2gA_1^2} = 0.75 + \frac{(80)^2}{2 \times 9.81 \times (6.56)^2}$$

$$= 8.33 \text{ m}$$

$$E_2 = y_2 + \frac{Q^2}{2gA_2^2} = 4.066 + \frac{(80)^2}{2\times9.81\times(49.06)^2}$$

$$= 4.20 \text{ m}$$

Thus, the loss of energy in the jump is

$$\Delta E = (E_1 - E_2)$$

$$= (8.33 - 4.20) = 4.13 \text{ m}.$$

Example 16.10. *A hydraulic jump occurs in a 90° triangular channel. Derive an equation relating the two depths and the flow rate. If the depths before and after the jump in the above channel are 0.5 m and 1.0 m, determine the flowrate and obtain the Froude numbers before and after the jump.*

Solution

From Eq. 16.14, we have

$$\frac{Q^2}{gA_1} + A_1\bar{z}_1 = \frac{Q^2}{gA_2} + A_2\bar{z}_2$$

If y_1 and y_2 are the depths before and after the jump, then in this case

$$A_1 = y_1^2;\ \bar{z}_1 = (y_1/3);\quad A_2 = y_2^2;\text{ and } \bar{z}_2 = (y_2/3)$$

Thus by substitution, we get

$$\frac{Q^2}{gy_1^2} + y_1^2\times\frac{y_1}{3} = \frac{Q^2}{gy_2^2} + y_2^2\times\frac{y_2}{3}$$

or
$$\frac{Q^2}{g}\left(\frac{1}{y_1^2} - \frac{1}{y_2^2}\right) = \frac{1}{3}\left(y_2^3 - y_1^3\right)$$

or
$$\frac{Q^2}{g}\left(\frac{y_2^2 - y_1^2}{y_1^2y_2^2}\right) = \frac{1}{3}\left(y_2^3 - y_1^3\right)$$

or
$$\frac{Q^2}{gy_1^5}\left[\frac{(y_2/y_1)^2 - 1}{(y_2/y_1)^2}\right] = \frac{1}{3}\left[(y_2/y_1)^3 - 1\right] \quad \text{...(i)}$$

But $\frac{Q^2}{gy_1^5} = \frac{V_1^2}{gy_1} = Fr_1^2$; and if ; then, we have

$$3Fr_1^2 = \frac{r^2(r^3-1)}{(r^2-1)} \quad \text{...(ii)}$$

The depths before and after the jump are given as

$$y_1 = 0.5 \text{ m; and } y_2 = 1.0 \text{ m}$$

$$\therefore \quad (y_2/y_1) = 2$$

Introducing these values in Eq. (i), we get

$$\frac{Q^2}{9.81\times(0.5)^5}\left[\frac{(2)^2-1}{(2)^2}\right] = \frac{1}{3}\left[(2)^3 - 1\right]$$

$$\therefore \quad Q = 0.977 \text{ m}^3/\text{s}$$

Froude number before the jump is given by

$$Fr_1 = \frac{Q}{\sqrt{gy_1^5}}$$

$$= \frac{0.977}{\sqrt{9.81\times(0.5)^5}} = 1.764$$

Alternatively the value of may be obtained by Eq. (ii) as

$$3Fr_1^2 = \frac{(2)^2(2^3-1)}{(2^2-1)}$$

$$\therefore \quad Fr_1 = 1.764$$

Froude number after the jump is given by

$$Fr_2 = \frac{Q}{\sqrt{gy_2^5}}$$

$$= \frac{0.977}{\sqrt{9.81\times(1)^5}} = 0.312$$

Example 16.11. *A rectangular channel carries a discharge of 2 m³/s per metre width. If the loss of energy in the hydraulic jump is found to be 2.75 m, determine the conjugate depths before and after the jump.*

Solution

From Eq. 16.25 the loss of energy in the jump may be expressed as

$$\Delta E = \frac{(y_2 - y_1)^3}{4y_1y_2}$$

or
$$\frac{\Delta E}{y_1} = \frac{\left[\left(\frac{y_2}{y_1}\right) - 1\right]^3}{4\left(\frac{y_2}{y_1}\right)} = \frac{(r-1)^3}{4r}$$

where
$$\left(\frac{y_2}{y_1}\right) = r$$

By substitution

$$\frac{2.78}{y_1} = \frac{(r-1)^2}{4r} \quad ...(i)$$

Further from Eq. 16.19, we have

$$\frac{y_2}{y_1} = \frac{1}{2}\left[1+\sqrt{1+\frac{8q^2}{gy_1^3}}\right]$$

or

$$r = \frac{1}{2}\left[-1+\left\{1+\frac{8(2)^2}{9.81y_1^3}\right\}^{1/2}\right]$$

or

$$r = \frac{1}{2}\left[-1+\left(1+\frac{3.26}{y_1^3}\right)^{1/2}\right] \quad ...(ii)$$

From Eqs (i) and (ii) the values of y_1 and r can be obtained by trial and error. Thus we have

$$y_1 = 0.214 \text{ m}, r = 8.64, \text{ and } y_2 = 1.849 \text{ m}$$

∴ Conjugate depths are 0.214 m and 1.849 m.

Example 16.12. *Show that the head loss in a hydraulic jump formed in a rectangular channel may be expressed as*

$$\Delta E = \frac{(V_1-V_2)^3}{2g(V_1+V_2)}$$

Solution

The head loss in a hydraulic jump may be expressed as

$$\Delta E = \left(y_1+\frac{V_1^2}{2g}\right)-\left(y_2+\frac{V_2^2}{2g}\right)$$

or

$$\Delta E = \left(\frac{V_1^2-V_2^2}{2g}\right)-(y_2-y_1)$$

If q is the discharge per unit width of the channel, then

$$q = y_1V_1 = y_2V_2$$

or

$$y_1 = \frac{q}{V_1} \text{ and } y_2 = \frac{q}{V_2}$$

By substitution, we get

$$\Delta E = \left(\frac{V_1^2-V_2^2}{2g}\right)-\left(\frac{q}{V_2}-\frac{q}{V_1}\right)$$

or

$$\Delta E = \left(\frac{V_1^2-V_2^2}{2g}\right)-q\left(\frac{V_1-V_2}{V_1V_2}\right) \quad ...(i)$$

From Eq. 16.16, we have

$$\frac{2q^2}{g} = y_1y_2(y_1+y_2)$$

or

$$\frac{2q^2}{g} = \frac{q^2}{V_1V_2}\left(\frac{q}{V_1}+\frac{q}{V_2}\right)$$

or

$$q = \frac{2(V_1V_2)^2}{g(V_1+V_2)} \quad ...(ii)$$

Introducing the value of q from Eq. (ii) in Eq. (i), we get

$$\Delta E = \left(\frac{V_1^2-V_2^2}{2g}\right)-\frac{2V_1V_2(V_1-V_2)}{g(V_1+V_2)}$$

or

$$\Delta E = \frac{(V_1^2-V_2^2)(V_1+V_2)-4V_1V_2(V_1-V_2)}{2g(V_1+V_2)},$$

or

$$\Delta E = \frac{(V_1-V_2)^3}{2g(V_1+V_2)}$$

Example 16.13. *A concrete lined trapezoidal irrigation canal has a bottom width of 10 m, side slopes of 1h : 1V and longitudinal bottom slope of 0.0005. If the channel is several kilometres long, what is the flow depth near the downstream end for a flow of 60 m³/s under free fall condition ?*

Solution

Let y be the normal depth of flow.

Then

$$A = (B+zy)y = (10+y)y$$

and

$$P = B = 2y\sqrt{1+2^2} = \left(10+2y\sqrt{2}\right)$$

$$R = \frac{A}{P} = \frac{(10+y)y}{\left(10+2y\sqrt{2}\right)}$$

For concrete lined channel value of Manning's n may be taken as 0.012. Using Manning's formula, we have

$$\frac{60}{(10+y)y} = \frac{1}{0.012}\times\left[\frac{(10+y)y}{\left(10+2y\sqrt{2}\right)}\right]^{2/3}\times(0.0005)^{1/2}$$

or $$1827.14 + 516.79y = [(10+y)y]^{5/2}$$

Solving this equation for y by trial, we get

$$y = 2.0108 \text{ m}$$

Let y_c be the critical depth.

For critical flow, we have

$$\frac{Q^2}{g} = \left[\frac{A^3}{T}\right]_c$$

$$\frac{(60)^2}{9.81} = \frac{[(10+y_c)y_c]^3}{(10+2y_c)}$$

or $$366.97(10+2y_c) = [(10+y_c)y_c]^3$$

Solving this equation for y_c by trial, we get

$$y_c = 1.4657 \text{ m.}$$

Since normal depth y > critical depth y_c, the flow is subcritical, and hence the depth of flow near the downstream end for free fall condition is equal to

$$y_c = 1.4657 \text{ m}$$

Example 16.14. *Water discharges at the rate of 298 m³/s over a spillway 14 m wide into a stilling basin of the same width. The water level behind the spillway is 61 m over datum. The river water level in the downstream is 29.2 m over datum. The river bed is paved flush with the bed level to just adequate length to form the jump basin. Assuming that no energy is dissipated in the flow down the spillway, find the basin invert level required for a hydraulic jump to be formed within the basin.*

Solution

Let the basin invert be at a height z above the datum.

Further let y, y_2 and V_1, V_2 be the depth of flow and velocities of flow before and after the hydraulic jump respectively.

Then

$$V_1 = \frac{298}{14\times y_1} = \frac{21.286}{y_1}$$

and $$V_2 = \frac{298}{14\times y_2} = \frac{21.286}{y_2}$$

If q is the discharge per unit width of the spillway, then

$$q = \frac{298}{14} = 21.286 \text{ m}^3/\text{s per m}$$

Applying Bernoulli's equation between the section behind the spillway where the hydraulic jump begins, and assuming that no energy is dissipated in the flow down the spillway, we get

$$61 = z + y_1 + \frac{V_1^2}{2g}$$

or $$61 = z + y_1 + \frac{(21.286)^2}{2g\times y_1^2}$$

or $$61 = z + y_1 + \frac{23.093}{y_1^2} \quad \text{...(i)}$$

The loss of energy in a hydraulic jump is given by Eq. 16.25 as

$$\Delta E = \frac{(y_2-y_1)^3}{4y_1y_2}$$

Thus applying Bernoulli's equation between the section behind the spillway and the section downstream of the hydraulic jump, we get

$$61 = 29.2 + \frac{V_2^2}{2g} + \frac{(y_2-y_1)^3}{4y_1y_2}$$

or $$61 = 29.2 + \frac{(21.286)^2}{2g\times y_1^2} + \frac{(y_2-y_1)^3}{4y_1y_2}$$

or $$31.8 = \frac{23.0.93}{(y_2/y_1)^2\times y_1^2} + \frac{[(y_2/y_1)-1]^3\times y_1}{4(y_2/y_1)}$$

or $$31.8 = \frac{23.093}{r^2\times y_1^2} + \frac{(r-1)^3\times y_1}{4r} \quad \text{...(ii)}$$

where $(y_2/y_1) = r$

Further for hydraulic jump in a rectangular channel from Eq. 16.16, we have

$$\frac{2q^2}{g} = y_1y_2(y_1+y_2)$$

or $$\frac{2\times(21.286)^2}{g} = y_1y_2(y_1+y_2)$$

or $$\frac{92.374}{y_1^3} = (y_2/y_1)[(y_2/y_1)+1]$$

or $$\frac{92.374}{y_1^3} = (y_2/y_1)[(y_2/y_1)+1) \quad \text{...(iii)}$$

or $$\frac{92.374}{y_1^3} = r(r+1)$$

Equations (i), (ii) and (iii) are solved simultaneously by trial and error with the following steps.

1. Assume value of y_1 and obtain the value of α from Eq. (iii)
2. Repeat the trial till the values of y_1 and r obtained in step 1 satisfy Eq. (ii).
3. Compute the value of z from Eq. (i).
4. As an alternative check, from the values of y_1 and r obtained in step 2, compute the value of y_2, and $(y_2 + z) = 29.2$.

The results obtained in this case are as follows :

$y_1 = 0.7429$ m ; $r = 14.5183$; $y_2 = 10.786$ m

$z = 18.414$ m ; and $(y_2 + z) = (10.786 + 18.414) = 29.2$ m.

Thus the basin invert level should be at a height of 18.414 m above the datum.

Example 16.15. *The depth and velocity of flow in a rectangular channel are 0.9 m and 1.5 m/s respectively. If a gate at the downstream end of the channel is abruptly closed, what will be the height and absolute velocity of the resulting surge? If the channel is 900 m long, how much time will be required for the surge to reach the upstream end of the channel?*

Solution

By closing the gate at the downstream end of the channel, a positive surge of type *B* is created. Thus from Eq. 16.39 (*a*), we have

$$y_1(V_1 + V_w) = y_2 V_w$$

or $$0.9\,(1.5 + V_w) = y_2 V_w \qquad \text{...(i)}$$

Also from Eq. 16.40 (a), we have

$$\frac{1}{2}(y_2^2 - y_1^2) = \frac{y_2}{g} \times V_w V_1$$

or $$\frac{1}{2}\left(y_2^2 - 0.9^2\right) = \frac{y_2}{9.81} \times V_w \times 1.5 \qquad \text{...(ii)}$$

Eliminating V_w between Eqs (i) and (ii) and simplifying, we get

$$y_2^3 - 0.9y_2^2 - 1.223y_2 + 0.729 = 0$$

Solving this Eq. by trial and error, we get

$$y_2 = 1.401 \text{ m}$$

From Eq. (i), we have

$$V_w = \frac{1.35}{y_2 - 0.9} = \frac{1.35}{1.401 - 0.9}$$

$\therefore$ $$V_w = 2.695 \text{ m/s}$$

Time required for the surge to reach the upstream end of the channel

$$= \frac{900}{2.695} = 334 \text{ s} = 5.57 \text{ min.}$$

Example 16.16. *The depth and velocity of flow in a rectangular channel are 1 m and 1.5 m/s respectively. If the rate of inflow at the upstream end is suddenly doubled, what will be the height and absolute velocity of the resulting surge and the celerity of the wave?*

Solution

$$y_1 = 1 \text{ m} ; V_1 = 1.5 \text{ m/s}$$

$\therefore$ $$q1 = 1 \times 1.5 = 1.5 \text{ m}^3/\text{s per m}$$

When the discharge is doubled

$$q_2 = 1.5 \times 2 = 3.0 \text{ m}^3/\text{s per m}$$

But $$q_2 = y_2 \times V_2$$

$\therefore$ $$y_2 V_2 = 3$$

In this case a positive surge of type *A* will be created. Thus from Eq. 16.27, we have

$$V_w = \frac{V_1 y_1 V_2 y_2}{y_1 - y_2}$$

or $$V_w = \frac{(1 \times 1.5) - 3.0}{1 - y_2}$$

or $$V_w = \frac{1.5}{y_2 - 1}$$

Also from Eq. 16.31, we have

$$V_w = \left[\sqrt{\frac{gy_2}{2y_1}(y_2 + y_1)}\right] + V_1$$

or $$\frac{1.5}{y_2 - 1} = \left[\sqrt{\frac{9.81 \times y_2}{2 \times 1}(y_2 + 1)}\right] + 1.5$$

or $$\frac{1.5}{y_2 - 1} - 1.5 = \sqrt{4.905y_2(y_2 + 1)}$$

or $$\frac{3 - 1.5y_2}{y_2 - 1} = \sqrt{4.905y_2(y_2 + 1)}$$

Solving the above equation by trial and error, we get

$$y_2 = 1.284 \text{ m}$$

$\therefore$ Height of the surge

$$h = (y_2 - y_1)$$
$$= (1.284 - 1.0) = 0.284 \text{ m}$$

Absolute velocity of the surge

$$V_w = \frac{1.5}{1.284 - 1} = 5.282 \text{ m/s}$$

Celerity of the wave

$$C = (V_w - V_1)$$
$$= (5.282 - 1.5) = 3.782 \text{ m/s}$$

Example 16.17. *A horizontal rectangular channel 3 m wide and 2 m water depth conveys water at 18 m³/s. If the flow rate is reduced to 2/3 of its original value, compute the magnitude and speed of the upstream surge. Assume that the front of the surge is rectangular and friction in the channel is neglected.*

Solution

Due to reduction in the flow rate (which would be achieved by closing the gate) a positive surge of type B would be created.

Thus from Eq. 16.39, we have

$$y_1(V_1+V_w) = y_2(V_2+V_w)$$

$$y_1 = 2\,\text{m}; V_1 = \frac{18}{3\times 2} = 3\,\text{m/s}$$

$$V_2 = \frac{2}{3}\times\frac{18}{3\times y_2} = \frac{4}{y_2}$$

Thus by substitution, we get

$$2(3+V_w) = y_2\left(\frac{4}{y_2}+V_w\right)$$

or $$6+2V_w = 4+y_2V_w$$

or $$V_w(y_2-2) = 2 \qquad \text{...(i)}$$

Further from Eq. 16.40, we have

$$\frac{1}{2}w(y_2^2-y_1^2) = \frac{w}{g}y_2(V_2+V_w)[(V_1+V_w)-(V_2+V_w)]$$

or $$\frac{1}{2}(y_2^2-4) = \frac{1}{g}y_2\left(\frac{4}{y_2}+V_w\right)\left[(3+V_w)\left(\frac{4}{y_2}+V_w\right)\right]$$

or $$\frac{1}{2}(y_2^2-4) = \frac{1}{9.81}y_2\left(\frac{4}{y_2}+V_w\right)\left(3-\frac{4}{y_2}\right) \qquad \text{...(ii)}$$

Eliminating V_w between Eqs (i) and (ii) and simplifying we get

$$y_2^4-2y_2^3-7.669y_2^2+1.786\,y_2-6.524 = 0$$

Solving this equation by trial and error, we get

$$y_2 = 3.939\text{ m}$$

∴ Height of surge above the original water surface,

$$h = (3.939-2) = 1.939\text{ m}$$

Further from Eq. (i), we have

$$V_w = \frac{2}{(y_2-2)} = \frac{2}{(3.939-2)}$$

∴ $$V_w = 1.031\text{ m/s}$$

SUMMARY OF MAIN POINTS

1. The dynamic equation for gradually varied flow is given by

$$\frac{dy}{dx} = \frac{S_0-S_f}{1-\dfrac{Q^2T}{gA^3}} = \frac{S_0-S_f}{1-F_r^2}$$

where

$\frac{dy}{dx}$ = slope of the water surface with respect to the channel bottom;

S_0 = bed slope of the channel;

S_f = slope of the energy gradient; and

F_r = Froude number.

2. The dynamic equation for gradually varied flow in wide rectangular channel is given by

(i) For Manning's formula

$$\frac{dy}{dx} = \frac{1-\left(\dfrac{y_n}{y}\right)^{10/3}}{1-\left(\dfrac{y_c}{y}\right)^3}$$

(ii) For Chezy's formula

$$\frac{dy}{dx} = \frac{1-\left(\dfrac{y_n}{y}\right)^3}{1-\left(\dfrac{y_c}{y}\right)^3}$$

where

y_n = normal depth of flow or depth of flow for uniform flow for a given discharge Q;

y = actual depth of flow for non-uniform flow; and

y_c = critical depth of flow

3. The channel bottom slopes are classified as

(i) Critical slope—when $S_0 = S_c$ and $y_n = y_c$

(ii) Mild slope—when $S_0 < S_c$ and $y_n > y_c$

(iii) Steep slope—when $S_0 > S_c$ and $y_n < y_c$

(iv) Horizontal slope—when $S_0 = 0$ and $y_n = \infty$

(v) Adverse slope—when $S_0 < 0$ and y_n is imaginary or non-existent

where

S_0 = channel bottom slope; and

S_c = critical slope.

4. In a mild sloped channel there will be three zones viz., $y > y_n > y_c$ and $y_n > y > y_c$ and $y_n > y_c > y$ in which respectively M_1, M_2 and M_3 curves will be formed.

5. In a steep slopped channel also there will be three zones viz $y > y_c > y_n$, $y_c > y > y_n$ and $y_c > y_n > y$, in which respectively S_1, S_2 and S_3 curves will be formed.

6. In a critical slopped channel there will be only two zones viz., $y > y_n = y_c$ and $y < y_n = y_c$ in which C_1 and C_3 curves will be formed.

7. In a horizontal channel also there will be two zones viz., $y_n > y > y_c$ and $y_n > y_c > y$ in which H_2 and H_3 curves will be formed.

8. In an adverse slopped channel also there will be only two zones viz., $y > y_c$ and $y < y_c$ in which A_2 and A_3 curves will be formed.

9. The general characteristics of the various water surface profiles are

(i) The water surface profiles with subscript 1 and 3 i.e., M_1, M_3, S_1, S_3, C_1, C_3 H_3 and A_3 are back water or rising curves.

(ii) The water surface profiles with subscript 2 i.e., M_2, S_2, H_2 and A_2 are drawdown or falling curves.

(iii) The water surface profile meets the normal depth line asymptotically and the critical depth line normally.

10. The length of the water surface profile may be obtained by integrating the dynamic equation for gradually varied flow.

In the step method of integration the entire length of the channel is divided into short reaches and computation is carried out step by step from one end of the reach to the other. The length of the small reach of the channel dx is given by

$$dx = \frac{E_1-E_2}{S_0-S_f} = \frac{\Delta E}{S_0-S_f}$$

where

ΔE = change in specific energy between the two sections of the small reach of the channel. By adding the dx values for the different reaches the lenght of the water surface profile x is obtained.

11. The hydraulic jump is defined as the sudden and turbulent passage of water from a supercritical state to subcritical state. It is classified as rapidly

varied flow.

12. The depth of flow before the hydraulic jump is known as initial depth and the depth of flow after the hydraulic jump is known as sequent depth. The initial and sequent depths are commonly known as cojugate depths which indicate the same specific force.

13. For hydraulic jump in a rectangular channel the relation between the initial depth y_1 and the sequent depth y_2 is given as

$$y_2 = \frac{y_1}{2}\left[-1+\sqrt{1+\frac{8q^2}{gy_1^3}}\right]$$

or

$$y_2 = \frac{y_1}{2}\left[-1+\sqrt{1+8F_{r1}^2}\right]$$

where

q = discharge per unit width of the channel; and

Fr_1 = froude number before the hydraulic jump.

14. The loss of energy ΔE in a hydraulic jump in rectangular channel is given by

$$\Delta E = \frac{(y_2-y_1)^3}{4y_1y_2}$$

PROBLEMS

16.1 State and discuss the assumptions made in the derivation of the dynamic equation for gradually varied flow. Starting from first principles derive equations for the slope of the water surface in gradually varied flow with respect to (i) channel bed, (ii) horizontal.

16.2 Show that the water surface slope S_w of a gradually varied flow is equal to the sum of the energy slope S_f and the slope due to velocity change $\frac{d}{dx}\left(\frac{V^2}{2g}\right)$.

16.3 Show that the gradually varied flow equation for flow in a rectangular channel of variable width B may be expressed as

$$\frac{dy}{dx} = \frac{S_0 - S_f + \left(\frac{Q^2 y}{gA^3}\right)\frac{dB}{dx}}{1-\frac{Q^2 B}{gA^3}}$$

16.4 A trapezoidal channel having bottom width 6 m, side slopes 2 horizontal to 1 vertical and bottom slope 0.0016, carries a discharge of 10 m³/s. A weir placed across the channel backs up the water to a depth of 2 m. Calculate how far upstream the depth is 1.5 m. Take Manning's n = 0.025. [*Ans.* 345 m]

16.5 A rectangular channel 10 m wide carries a discharge of 40 m³/s. If at a section in this channel the depth is 1.5 m, how far (upstream and downstream) from this section will the depth be 2.0 m. Take S_0 = 0.00009 and n = 0.017. [*Ans.*367 m upstream]

16.6 A rectangular channel 10 m wide is laid with a break in its bottom slope from 0.01 to 0.0064. If it carries 125 m³/s, determine the nature of the surface profile and compute its length. Take n = 0.015. [*Ans.* S_3 profile, 440 m]

16.7 In a horizontal rectangular channel 1.5 m wide, if the observed depths before and after the jump are 0.2 m and 1.0 m respectively, determine the discharge flowing through the channel. [*Ans.* 1.63 m³/s]

16.8 A rectangular channel carries a discharge of 3 m³/s per metre width. If the loss of energy in the hydraulic jump is found to be 3.2 m, determine the conjugate depths before and after the jump. [*Ans.* 0.293 m, 2.36 m]

16.9 In a horizontal triangular channel with sides sloping at 1:1, the depths before and after the jump are observed as 1 m and 1.25 m respectively. Determine the discharge flowing through the channel. [*Ans.* 2.943 m³/s]

16.10 A discharge of 600 m³/s flows over spillway and then passes over a horizontal concrete apron (n = 0.015). The velocity of water at the bottom of the spillway is 10 m/s and the width of the apron is 40 m. A hydraulic jump forms on the apron due to a tail water depth of 4.5 m in the channel below. In order that the jump be contained on the apron, how long should the apron be built? Also determine the loss of energy from the foot of the spillway to the downstream side of the jump. [*Ans.* 81.5 m; 90.06 × 10⁵ W]

16.11 For a hydraulic jump in a horizontal triangular channel show that

$$3Fr_1^2 = \frac{r^2(r^3-1)}{r^2-1}$$

where $Fr_1^2 = \left(V_1^2/gy_1\right)$ and $r = (y_2/y_1)$

16.12 The normal depth of flow of water in a rectangular channel 1.5 m wide is 1 m. The bed slope of the channel is 0.0006 and Manning's n = 0.012. Find the critical depth.

At a certain section of the same channel the depth is 0.92 m while at second section the depth is 0.86 m. Find the distance between the two sections. Also find whether the second section is located upstream of the first section. [*Ans.* 0.516 m; 222 m downstream]

16.13 Determine the approximate water surface slope (m/km) at a point in a rectangular channel in which the discharge is 0.84 cumec, the bed-width is 3 m, the depth of flow is 0.75 m and the bed slope is 0.15 m/km. Assume C in Chezy's formula as 55 and take energy coeffiient α equal to unity. [*Ans.* 0.091 m/km]

16.14 The depth and velocity of flow in a rectangular channel are 1 m and 1.5 m/s respectively. If the rate of inflow at the upstream end of the channel is increased to the extent that the depth is doubled in magnitude, determine the absolute velocity of the resulting surge. [*Ans.* 6.925 m/s]

16.15 A gate is to be suddenly dropped into place closing a rectangular channel 2 m deep and 3 m wide in which 6 cumec of water is flowing at a depth of 1.2 m. Will the water spill over the sides? What will be the velocity and height of the surge produced?

[*Ans.* y_2 = 1.842 m, hence water will not spill over the sides ; 3.115 m/s ; 0.642 m]

16.16 Water flows from a lake into a steep rectangular channel 3 m wide. Determine the lake level above the channel bed at the outfall if the discharge is 40 m³/s. [*Ans.* 2.626 m]

16.17 A sluice acros a channel 6 m wide discharges a stream 1 m deep. What is the flow rate when the depth upstream of the sluice is 7 m? On downstream side, concrete blocks have been placed to create conditions for hydraulic jump to occur. Determine the force on the blocks if the downstream depth is 3 m.

[*Ans.* 65.77 m³/s; 24,996 kg(or 245.21 kN)]

16.18 Water emerges from a spillway with a velocity of 20 m/s and a depth 0.5 m at the toe. Estimate the distance of the hydraulic jump from the spillway toe if no specific measures are taken for the formation of jump at the toe. Assume for the downstream channel Manning's n = 0.020 and S_o = 0.001. Calculations may be performed by direct step method taking only one step. Assume channel to be a wide rectangular channel.

[**Hint :** y_c = 2.168 m; y_n = 3.026 m; Since $y_n > y_c$, it is mild slopped channel. For sequent depth y_2 = 3.26 m, the initial depth y_1 = 1.493 m. Thus M_3 type surface profile will be formed along which the depth of flow will increase from 0.5 m at the toe of the spillway to 1.493 m at the section where hydraulic jump will be formed. The distance of the hydraulic jump from the toe = 83.14 m which is equal to the length of the M_3 type surface profile.]

Dimensional Analysis, Hydraulic Similitude and Model Investigation

Chapter **17**

17.1 INTRODUCTION

Dimensional Analysis is a mathematical technique which makes use of the study of dimensions as an aid to the solution of several engineering problems. Each physical phenomenon can be expressed by an equation, composed of variable (or physical quantities) which may be dimensional and non-dimensional quantities. Dimensional analysis helps in determining a systematic arrangement of the variables in the physical relationship and combining dimensional variables to form non-dimensional parameters.

In the study of fluid mechanics the dimensional analysis has been found to be useful in both analytical and experimental investigations. Some of the uses of dimensional analysis are as narrated below:

(i) Testing the dimensional homogeneity of any equation of fluid motion.

(ii) Deriving equations expressed in terms of non-dimensional parameters to show the relative significance of each parameter.

(iii) Planning model tests and presenting experimental results in a systematic manner in terms of non-dimensional parameters; thus making it possible to analyse the complex fluid flow phenomenon.

In order to facilitate the study of dimensional analysis and its application to practical problems a preliminary discussion about dimensions is given below.

17.2 DIMENSIONS

The various physical quantities used by the engineers and the scientists, to describe a given phenomenon can be described by a set of quantities which are in a sense independent of each other. These quantities are known as *fundamental* or *primary quantities*. The primary quantities are mass, length, time and temperature, designated by the letters M, L, T and θ respectively. Temperature is specially useful in compressible flow. All other quantities such as area, volume, velocity, acceleration, force, energy, power, etc., are termed as *derived quantities* or *secondary quantities*, because they can be expressed in terms of the primary quantities. The expression for a derived quantity in terms of the primary quantities is called the *dimension* of the physical quantity. For example the dimension of force can be expressed as indicated below.

$$[\text{Force}] = [\text{Mass} \times \text{Acceleration}]$$

Since, $$[\text{Acceleration}] = \left[\frac{L}{T^2}\right]$$

$$[\text{Force}] = \left[\frac{ML}{T^2}\right] = [MLT^{-2}]$$

The rectangular bracket signifies that the dimensions of the quantity are being considered.

Some engineers prefer to use force instead of mass as fundamental quantity because the former is easy to measure. The system is then represented by symbols F-L-T instead of M-L-T. The dimensional form of any quantity is independent of the system of units (*SI*, metric or English) chosen and enables us to convert the units readily from one system of measurements to the other.

Table 17.1 gives the dimensions and units of measurement of measurement for the various physical quantities involved in the problems concerning fluid flow.

TABLE 17.1

Quantities	*Symbol*	*Dimensions in terms of*		*Units of Measurement*		
		M–L–T	*F–L–T*	*SI*	*Metric*	*English*
1	2	3	4	5	6	7
(A) Geometric						
Length (any linear measurement)	L, l	L	L	m	m	ft
Area	A	L^2	L^2	m^2	m^2	ft^2
Volume	V	L^3	L^3	m^3	m^3	ft^3
Curvature	C	1/L	1/L	m^{-1}	m^{-1}	ft^{-1}
Slope	S, *i*	$M^0L^0T^0$	$F^0L^0T^0$	m/m	m/m	ft/ft
Angle	α, θ	$M^0L^0T^0$	$M^0L^0T^0$	radians or degrees	radians or degrees	radians or degrees
Shape factor	η	$M^0L^0T^0$	$M^0L^0T^0$	—	—	—
(B) Kinematic						
Time	T, *t*	T	T	s	s	s
Velocity (linear)	V, v	L/T	L/T	m/s	m/s	ft/s
Angular velocity	ω	1/T	1/T	rad/s	rad/s	rad/s
Frequency	*n*	1/T	1/T	1/s	1/s	1/s
Acceleration (linear)	*a*	L/T^2	L/T^2	m/s^2	m/s^2	ft/s^2
Angular acceleration	α	$1/T^2$	$1/T^2$	rad/s^2	rad/s^2	rad/s^2
Gravitational acceleration	*g*	L/T^2	L/T^2	m/s^2	m/s^2	ft/s^2
Discharge	Q	L^3/T	L^3/T	m^3/s (cumec)	m^3/s (cumec)	ft^3/s (cusec)
Discharge per unit width	*q*	L^2/T	L^2/T	$[m^3/s/m]$	$[m^3/s/m]$	$[ft^3/s/ft]$
Kinematic Viscosity (μ/ρ)	μ	L^2/T	L^2/T	m^2/s	m^2/s	ft^2/s
Circulation	Γ	L^2/T	L^2/T	m^2/s	m^2/s	m^2/s

(Contd.)

Table 17.1 Contd.

1	2	3	4	5	6	7
(C) Dynamic						
Mass	M, m	M	FT^2/L	kg	metric slug(msl) or gm(mass)	slug or lb (mass)
Force	F	ML/T^2	F	N	kg(f)	lb(f)
Weight	W	ML/T^2	F	N	kg(wt.)	lb(wt)
Mass density	ρ	M/L^3	FT^2/L^4	kg/m^2	$ms\ell/m^3$ or gm/cm^3	$slug/ft^3$ or $lb(m)/ft^3$
Specific weight	ω, γ	M/L^2T^2	F/L^3	N/m^2	$kg(f)/m^3$	$lb(f)/m^3$
Specific gravity	s	$M^\circ L^\circ T^\circ$	$F^\circ L^\circ T^\circ$	—	—	—
Pressure intensity	p	M/LT^2	F/L^2	N/m^2 or pascal (Pa)	$kg(f)/m^2$ or $kg(f)\ cm^2$	$lb(f)/ft^2$ or $lb(f)\ in^2$
Shear stress	τ	M/LT^2	F/L^2	N/m^2 or pascal (Pa)	$kg(f)/m^2$ or $kg(f)\ cm^2$	$lb(f)/ft^2$ or $lb(f)\ in^2$
Dynamic viscosity	μ	M/LT	FT/L^2	$N.s/m^2$ or pa.s	$\frac{kg(f)\text{-}s}{m^2}$	$\frac{lb(f)\text{-}s}{ft^2}$
Surface Tension	σ	M/T^2	F/L	N/m	kg(f)/m	lb(f)/ft
Modulus of Elasticity	E	M/LT^2	F/L^2	N/m^2 or pascal(Pa)	$kg(f)/m^2$ or $kg(f)/cm^2$	$lb(f)/ft^2$ or $lb(f)/in^2$
Compressibility	1/E	LT^2/M	L^2/F	m^2/N or I/(Pa)	$m^2/kg(f)$ or $cm^2/kg(f)$	$ft^2/lb(f)$ or $in^2/lb(f)$

Contd.

Table 17.1 Contd.

1	2	3	4	5	6	7
Impulse, Momentum	I, M	ML/T	FT	N.s	kg(f)-s	lb(f)-s
Work, Energy	W, E	ML^2/T^2	FL	N.m or joule (J)	kg(f)-m	ft-lb(f)
Torque	T	ML^2/T^2	FL	N.m or joule (J)	kg(f)-m	ft-lb(f)
Power	P	ML^2/T^2	FL/T	N.m or watt(W)	$\frac{kg(f)\text{-}m}{s}$	$\frac{ft\text{-}lb(f)}{s}$
Weight rate of flow	W	ML/T^3	F/T	N/s	kg(f)/s	lb(f)/s
(D) General						
Moment of a force	$F\times l$	ML^2/T^2	FL	N.m	kg(f)-m	ft-lb(f)
Moment of Inertia of area	I	L^4	L^4	m^4	m^4	ft^4
Moment of Inertia of mass	Ml^2	ML^2	FLT^2	$kg.m^2$	$mal\text{-}m^2$ or $gm\text{-}cm^2$	$slug\text{-}ft^2$ or $lb(m)\text{-}ft^2$
Thermodynamic temperature	T	θ	θ	K	°C (abs) (degree or celsius absolute)	°F (abs) (degree absolute)
Temperature	t°	(kelvin) θ	centi-grade θ	Fahre-nheit °C (degree Celsius)	°C (degree centigrade or Celsius)	°F (degree Fahrenheit)
Heat	ϕ	ML^2/T^2	FL	joule(J)	kilo-calories	B.Th.U or C.H.U
Specific heat	ϕ_s	$L^2/T^2\theta$	$L^2/T^2\theta$ kilogram	joule per msl°C(abs) kelvin (J/kg.K)	kg(f)-m per slug °F(abs)	lb(f)-ft per

Contd.

Table 17.1 Contd.

1	2	3	4	5	6	7
Thermal conductivity or Heat conduction	ϕ_c	$ML/T^3\theta$	$F/T\theta$	watt per metre kelvin (W/m.K)	[kg(f)-m/s] per m °C (abs)	[ft-lb(f)/s] per ft °F(abs)
(E) Dimensionless						
Reynolds, number	Re,N_R	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Froude number	Fr,N_F	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Euler number	Eu,N_E	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Weber number	We,N_W	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Mach number	Ma,N_M	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Friction factor	f	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Kinetic Energy correction factor	α	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Momentum correction factor	β	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—
Efficiency	η	$M^0L^0T^0$	$F^0L^0T^0$	—	—	—

17.3 DIMENSIONAL HOMOGENEITY

Fourier's principle of dimensional homogeneity states that an equation which expresses a physical phenomenon of fluid flow must be algebraically correct and dimensionally homogeneous.

An equation is said to be dimensionally homogeneous, if the dimensions of the terms on its left hand side are same as the dimensions of the terms on its right hand side. To illustrate this consider the equation for the time of swing of a simple pendulum,

$$t = 2\pi\sqrt{\frac{l}{g}}$$

The dimensions of the left hand side are $[T]$. Since 2π is a pure number, i.e., it has no dimensions, the dimensions of $\sqrt{(l/g)}$ are $\left[\frac{L}{(L/T^2)}\right]^{1/2} = [T]$. Thus the dimensions of the terms on both sides of the equation are same and hence it is dimensionally homogeneous. Some of the other examples of the dimensionally homogeneous equations are as noted below:

$$Q = \frac{2}{3}C_d\sqrt{2g}\,LH^{3/2} \quad \text{... (Flow over rectangular weir)}$$

$$Q = \frac{8}{15}C_d\sqrt{2g}\tan\frac{\theta}{2}H^{5/2} \quad \text{... (Flow over triangular weir)}$$

$$\Delta p = \frac{32\mu VL}{D^2} \quad \text{... (Laminar flow in pipes)}$$

$$\Delta p = f\frac{l}{D}\frac{\rho V^2}{2} \quad \text{... (Turbulent flow in pipes)}$$

A dimensionally homogeneous equation has the unique characteristic of being independent of the units chosen for measurement i.e., if an equation is dimensionally homogeneous, it can be used without any modification with either system of units.

It is always possible to reduce a dimensionally homogeneous equation to a non-dimensional form as explaind below:

$$\left[\frac{t}{\sqrt{\frac{l}{g}}}\right] = 2\pi;\ \left[\frac{Q}{\sqrt{g}LH^{3/2}} = \frac{2\sqrt{2}}{3}C_d\right];\ \left[\frac{(\Delta p)D}{\mu V}\right] = 32\left[\frac{L}{D}\right]$$

and

$$\left[\frac{\Delta p}{\frac{\rho V^2}{2}} = f\frac{l}{D}\right]$$

Further from the study of these equations it will be observed that in general the number of dimensionless groups that can be made is equal to the number of variables constituting the equation minus the number of primary quantities involved.

In addition to these, there are several equations in hydraulics which are dimensionally non-homogeneous, but still well applicable to flow system within their limited ranges. For example

$$V = \frac{1}{n}R^{2/3}S^{1/2}$$

is Manning's equation (for SI or metric system of units) which relates mean velocity V, in channels to the hydraulic mean depth R, bed slope S and the roughness coefficient n. If n is assumed to be a dimensionless factor, then on analysing the dimensions of both sides of this equation it will be found that

$$\left[\frac{L}{T}\right] \neq \left[L^{2/3}\right]$$

Since the equation is dimensionally non-homogeneous, it can be used only in SI or metric system of units, with V in m/s and R in metres. However, if this equation is to be dimensionally homogeneous, then n must have the dimension $\left[\frac{T}{L^{1/3}}\right]$

Some of the other examples of the dimensionally non-homogeneous equations are

$$V = C\sqrt{RS} \quad \text{... (Chezy's formula)}$$

$$V = 0.84y^{0.64} \quad \text{... (Kennedy's equation)}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} \quad \text{...(Specific speed of turbines)}$$

$$Q = 1.84(L - 0.1nH)H^{3/2} \quad \text{...(Francis formula)}$$

17.4 METHODS OF DIMENSIONAL ANALYSIS

The following two methods of dimensional analysis are generally used:

(a) Rayleigh Method.

(b) Buckingham π-Method.

(a) Rayleigh Method. This method of dimensional analysis was proposed by Lord Rayleigh in 1899, for determining the effect of temperature on the viscosity of a gas. In this method a functional relationship of some variables is expressed in the form of an exponential equation which must be dimensionally homogeneous. Thus if X is some function of variables $X_1, X_2, X_3..X_n$; the functional equation can be written in the following general form

$$X = f(X_1, X_2, X_3....X_n) \quad \text{...(17.1)}$$

In this equation X is a dependent variable, while $X_1, X_2, X_3.....X_n$ are independent variables. A dependent variable is the one about which information is required while independent variables are those which govern the variation of dependent variable.

The above noted equation may be expressed as

$$X = C\left(X_1^a X_2^b X_3^c........X_n^n\right) \quad \text{...(17.2)}$$

in which C is a dimensionless constant which may be determined either from the physical characteristics of the problem or from experimental measurements. The exponents $a, b, c,....n$ are then evaluated on the basis that the equation is dimensionally homogeneous. The dimensionless parameters are then formed by grouping together the variables with like powers. The method is now illustrated by means of an example.

Consider the problem of fluid flow through a small orifice discharging freely into atmosphere under a constant head. Let Q be the discharge passing through a small orifice of diameter d, under a constant head H. Also let ρ be the mass density and μ the dynamic viscosity of the fluid flowing through the orifice. The discharge Q may be assumed to depend on these variables viz., d, H, ρ, μ and the gravitational acceleration g since the flow is freely into atmosphere. Therefore the functional relationship for Q may be written as

$$Q = f(\mu, \rho, d, H, g)$$

which by Rayleigh's method may be expressed in an exponential form

$$Q = C(\mu^a \rho^b d^c H^d g^e)$$

where C is a dimensionless constant.

Substituting the proper dimensions for each variable in this exponential equation in M-L-T system,

$$(L^3/T) = (M^\circ L^\circ T^\circ)(M/LT)^a (M/L^3)^b (L)^c (L)^d (L/T^2)^e$$

For dimensional homogeneity the exponents of each dimension on the both sides of the equation must be indentical. Thus

for M : $0 = a + b$

for L : $3 = -a - 3b + c + d + e$

for T : $-1 = -a - 2e$

Since there are five unknowns in three equations, three of the unknowns should be expressed in terms of the other two:

$$b = -a$$

$$e = \frac{1}{2} - \frac{a}{2}$$

$$c = \frac{5}{2} - \frac{3a}{2} - d$$

Therefore,

$$Q = C\left[\mu^a \rho^{-a} d^{\left(\frac{5}{2}-\frac{3a}{2}-d\right)} H^d g^{\left(\frac{1}{2}-\frac{a}{2}\right)}\right]$$

$$= C\left[\left(d^{\frac{5}{2}} g^{\frac{1}{2}}\right)\left(\mu^a \rho^{-a} d^{\frac{-3a}{2}} g^{\frac{-a}{2}}\right).\left(H^d d^{-d}\right)\right]$$

$$= C\left[\left(d^2 d^{\frac{1}{2}} g^{\frac{1}{2}}\right)\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^d\right]$$

$$\therefore \quad Q = C\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^{d-\frac{1}{2}} d^2 H^{1/2} g^{1/2}$$

$$= \frac{C}{\frac{\pi}{4}\sqrt{2}}\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right)^a \left(\frac{H}{d}\right)^{d-\frac{1}{2}} \frac{\pi}{4} d^2 \sqrt{2gH}$$

$$= \left(a\sqrt{2gH}\right) f_1\left[\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right), \left(\frac{H}{d}\right)\right]$$

This expression may be written in the usual form, viz.,

$$Q = C_d a \sqrt{2gH}$$

where C_d is the coefficient of discharge of the orifice which can be expressed as

$$C_d = f_1\left[\left(\frac{\mu}{\rho d^{3/2} g^{1/2}}\right), \left(\frac{H}{d}\right)\right]$$

It may be pointed out that both the terms in the bracket are dimensionless and C_d is also a non-dimensional factor.

(b) Buckingham π-Method. The Buckingham's π-theorem states that if there are n dimensional variables involved in a phenomenon, which can be completely described by m fundamental quantities or dimensions (such as mass, length, time etc.), and are related by a dimensionally homogeneous equation, then the relationship among the n quantities can always be expressed in terms of exactly $(n-m)$ dimensionless and independent π terms.

Mathematically, if any variable Q_1 depends on the independent variables, $Q_2, Q_3, Q_4 \ldots Q_n$; the functional equation may be written as

$$Q_1 = f(Q_2, Q_3, Q_4 \ldots Q_n) \quad \ldots(17.3)$$

which can be transformed to another functional relationship

$$f_1(Q_1, Q_2, Q_3, \ldots Q_n) = C \quad \ldots(17.4)$$

where C is a dimensionless constant. This is as if $x = f(y) = y^2 + C$; whence $x - y^2 = f_1(x, y) = C$. In accordance with the π-theorem, a non-dimensional equation can thus be obtained in the following form

$$f_2(\pi_1, \pi_2, \pi_3, \ldots \pi_{n-m}) = C_1 \quad \ldots(17.5)$$

wherein each dimensionless π-term is formed by combining m variables out of the total n variables with one of the remaining $(n-m)$ variables. These m variables which appear repeatedly in each of the π-terms, are consequently called *repeating variables* and are chosen from among the variables such that they together involve all the m fundamental quantities (or dimensions) and they themselves do not form a dimensionless parameter. Thus the different π-terms may be established as shown below:

$$\left.\begin{aligned}\pi_1 &= Q_1^{a_1} Q_2^{b_1} Q_3^{c_1} \ldots\ldots Q_m^{m_1} \quad Q_{m+1}\\ \pi_2 &= Q_1^{a_2} Q_2^{b_2} Q_3^{c_2} \ldots\ldots Q_m^{m_2} \quad Q_{m+2}\\ &\ldots\ldots\ldots\ldots\ldots\ldots\\ \pi_{n-m} &= Q_1^{a_{n-m}} Q_2^{b_{n-m}} Q_3^{c_{n-m}} \ldots Q_m^{m_{n-m}} \quad Q_n\end{aligned}\right\} \quad \ldots(17.6)$$

In Eq. 17.6 each individual equation is dimensionless and the exponents $a, b, c, d \ldots m$ etc., are determined by considering dimensional homogeneity for each equation so that each π-term is dimensionless.

The final general equation for the phenomenon may then be obtained by expressing any one of the π-terms as a function of the others as

$$\pi_1 = f_1(\pi_2, \pi_3, \pi_4, \ldots \pi_{n-m})$$
$$\pi_2 = f_2(\pi_1, \pi_3, \pi_4, \ldots \pi_{n-m})$$

or any other desired relationship may be obtained.

The use of Buckingham's π-method may now be illustrated by means of an example.

Considering the same problem of flow through small orifice (as discussed under Rayleigh's method) the following steps may be adopted to solve it by Buckingham's π-method.

Step 1. The physical quantities involved in the phenomenon are, Q, d, H, g, μ and ρ. The functional equation for the discharge Q may be expressed as

$$Q = f(d, H, g, \mu, \rho)$$

which in its most general form may be written as

$$f_1(Q, d, H, g, \mu, \rho) = C$$

Thus the total number of variables $n = 6$ and all these variables may be completely described by the three fundamental dimensions of either the M-L-T or F-L-T systems. Hence m equals 3. Therefore there are $(n-m) = 3$ dimensionless π-terms, so that

$$f_2(\pi_1, \pi_2, \pi_3) = C_1$$

Step 2. In order to form these π-terms, we have to choose 3 repeating variables, since in this case $m = 3$. As stated earlier these variables should be such that they, among them, contain all the three fundamental dimensions and they themselves do not form a dimensionless parameter. Thus let us choose, $\rho(M/L^3)$, $d(L)$ and $g(L/T^2)$ as repeating variables, since the above noted requirements are fulfilled by these.

Step 3. Since physical quantities of dissimilar dimensions can neither be added nor subtracted the terms are expressed as product as follows:

$$\left.\begin{aligned}\pi_1 &= \rho^{a_1} d^{b_1} g^{c_1} Q\\ \pi_2 &= \rho^{a_2} d^{b_2} g^{c_2} \mu\\ \pi_3 &= \rho^{a_3} d^{b_3} g^{c_3} H\end{aligned}\right\}$$

Step 4. Expressing π_1 dimensionally in terms of M-L-T system

$$\pi_1 = M^\circ L^\circ T^\circ = (M/L^3)^{a_1} (L)^{b_1} (L/T^2)^{c_1} (L^3/T)$$

Equating the exponents of M, L and T, we get

for $\quad M: 0 = a_1$

for $\quad L: 0 = -3a_1 + b_1 + c_1 + 3$

for $\quad T: 0 = -2c_1 - 1$

From these equations, we get

$$a_1 = 0; b_1 = -\frac{5}{2}; c_1 = -\frac{1}{2}$$

Hence $\quad \pi_1 = \left(\dfrac{Q}{g^{1/2} d^{5/2}}\right)$

Similarly, we have

$$\pi_2 = \rho^{a_2} d^{b_2} g^{c_2} \mu$$

or $\quad M^\circ L^\circ T^\circ = (M/L^3)^{a_2} (L)^{b_2} (L/T^2)^{c_2} (M/LT)$

Equating the exponents of M, L, and T, we get

for $\quad M: 0 = a_2 + 1$

for $\quad L: 0 = -3a_2 + b_2 + c_2 - 1$

for $\quad T: 0 = -2c_2 - 1$

from which

$$a_2 = -1; b_2 = -\frac{3}{2}; c_2 = -\frac{1}{2}$$

Hence $\quad \pi_2 = \left(\dfrac{\mu}{\rho g^{1/2} d^{3/2}}\right)$

Lastly, we have

$$\pi_3 = \rho^{a_3} d^{b_3} g^{c_3} H$$

or

$$M^0L^0T^0 = (M/L^3)^{a_3}(L)^{b_3}(L/T^2)^{c_3}(L)$$

Equating the exponents of M, L and T, we get

for $\quad M : 0 = a_3$

for $\quad L : 0 = -3a_3 + b_3 + c_3 + 1$

for $\quad T : 0 = -2c_3$

from which

$$a_3 = 0; b_3 = -1; c_3 = 0$$

Hence

$$\pi_3 = \frac{H}{d}$$

Step 5. Since

$$f_2(\pi_1, \pi_2, \pi_3) = C_1$$

By substitution, we have

$$f_2\left(\frac{Q}{g^{1/2}d^{5/2}}, \frac{\mu}{\rho g^{1/2}d^{3/2}}, \frac{H}{d}\right) = C_1$$

or

$$\frac{Q}{g^{1/2}d^{5/2}} = C_2 f_3\left(\frac{\mu}{\rho g^{1/2}d^{3/2}}, \frac{H}{d}\right)$$

from which the expression in its usual form may be obtained as explained earlier.

17.5 OUTLINE OF PROCEDURE FOR BUCKINGHAM METHOD

The following is the outline of the procedure to be followed when the dimensional analysis is carried out by the Buckingham π-method.

(1) List all the n physical quantities or variables involved in the phenomenon. Note their dimensions and the number m of the fundamental dimensions comprised in them. So that there will be $(n-m)$ π-terms.

(2) Select m variables out of these which are to serve as repeating variables. These variables should be such that none of them is dimensionless, no two variables have the same dimensions, they themselves do not form a dimensionless parameter and all the m fundamental dimensions are included collectively in them. Moreover, as far as possible the dependent variable should not be taken as a repeating variable as otherwise it will not be possible to obtain an explicit relationship. In fluid flow problems, usually a characteristic linear dimension, a characteristic velocity and a characteristic fluid property (usually fluid density) are chosen as repeating variables, so that familiar non-dimensional parameters as discussed later are formed.

(3) Write the general equations for different π-terms. These may be expressed as the product of the repeating variables each raised to an unknown exponent and one of the remaining variables, taken in turn, with a known power (usually taken as one).

(4) Write the dimensional equations for the equations of the π-terms obtained in the step (3) above. Compute the values of the unknown exponents by equating the exponents of the respective fundamental dimensions on both the sides of each of the dimensional equation and obtain the different dimensionless groups or π-terms.

(5) Write the final general equation for the phenomenon in terms of the π-terms.

In order to obtain the final expression in the desired manner the following useful suggestions may be noted:

(i) If a quantity is dimensionless, it is a π-term without going through the above procedure.

(ii) If any two physical quantities have the same dimensions, their ratio will be one of the π-terms. For example (H/d) is dimensionless and hence it is a π-term which may be directly written without going through the above procedure.

(iii) Any π-term may be replaced by any power of that term, including negative as well as fractional powers. For example π_1 may be replaced by π_1^{-1}, or π_2 may be replaced by π_2^2, π_3 may be replaced by $(1/\pi_3^{1/2})$ etc.

(iv) Any π-term may be replaced by multiplying it by a numerical constant. For example π_1 may be replaced by $3\pi_1$ or so.

(v) Any π-term may be replaced by another π-term obtained by adding or subtracting an absolute numeral from it.

(vi) Any π-terms may be replaced by multiplying or dividing it by another π-term. For example π_1 may be replaced by $(\pi_1 \times \pi_2)$ or (π_1/π_2), or π_1 and π_2 may be replaced by $(\pi_1 \times \pi_2)$ or (π_1/π_2).

A critical study of the methods of the dimensional analysis discussed above reveals that the two methods are essentially the same. But when the number of variables is considerably increased, the Rayleigh's method becomes tedious. This difficulty in case of more variables is however, avoided by using Buckingham's π-method instead of Rayleigh's method.

17.6 NUMBER OF DIMENSIONLESS GROUPS IN A COMPLETE SET OF VARIABLES

According to Buckingham's π-theorem if there are n variables involved in a phenomenon then these n variables can be grouped together into $(n-m)$ dimensionless groups (or π-terms) when m is the number of the fundamental dimensions required to express the n variables. This has been found to be true not only for the problems of fluid mechanics, but also for the problems relating to several other fields. However, in some cases the above rule does not hold good and therefore it cannot be considered as a general rule for determining the number of the dimensionless groups in a complete set of variables. As such in order to obtain the number of dimensionless groups into which a complete set of variables may be grouped, another method based on *dimensional matrix approach* may be adopted which is as follows:

(i) List all the variables involved in the phenomenon and prepare a *dimensional matrix* of the variables, which is nothing but display of the exponents of the dimensions of the variables by a tabular arrangement.

(ii) Find the rank of the dimensional matrix. The rank of a matrix is said to be r, if it contains a non-zero determinant of order r and if all determinants of order greater than r that the matrix contains are equal to zero.

(iii) The number of dimensionless groups of π-terms is equal to $(n-r)$, that is, total number of variables n minus the rank of the dimensional matrix r.

As an example let it be assumed that in a particular problem the variables involved are velocity V, length L, force F, mass density ρ, dynamic viscosity μ, and gravitational acceleration g. The dimensional matrix is formed by writing the exponents (or powers) of M, L and T which appear in the dimensional expression of the variable in the column below the variable itself. As such dimensional matrix of these variables is as given below:

	V	L	F	ρ	μ	g
M	0	0	1	1	1	0
L	1	1	1	–3	–1	1
T	–1	0	–2	0	–1	–2

In order to find the rank of the matrix, a third order determinant formed from any three columns of the matrix may be considered and evaluated. Thus the determinant formed from the last three columns in the above dimensional matrix is

$$\begin{vmatrix} 1 & 1 & 0 \\ -3 & -1 & 1 \\ 0 & -1 & -2 \end{vmatrix} = -3$$

Since in this case a third order determinant of the dimensional matrix is non-zero (i.e., it has a value different from zero) the rank of the matrix is 3. However, if this determinant would have been equal to zero, then other third order determinants should be considered and evaluated one after the other till any one of these is non-zero. Furthermore if none of the third order determinants of the dimensional matrix is non-zero, then the second order determinants should be considered and evaluated till a non-zero determinant of second order is found.

Since the rank of the above dimensional matrix is 3 and the number of variables is 6, the number of dimensionless groups in the present case is therefore $(6-3) = 3$. The actual dimensionless groups may however be formed by using Buckingham's method.

Another example that may be considered is that of capillary rise h which depends on the radius r of the tube, surface tension σ and specific weight w of the liquid. Thus in this case there are 4 variables which involve 3 fundamental dimensions. As such according to Buckingham's π-theorem these four variables can be grouped into $(4-3)$ i.e., only 1 dimensionless group. However, it is not true and by adopting the above method based on dimensional matrix approach, as indicated below, it may be shown that in this case the number of dimensionless groups will be 2.

The dimensional matrix of these four variables is as given below:

	h	r	σ	w
M	0	0	1	1
L	1	1	0	–2
T	0	0	–2	–2

In order to find the rank of the above matrix, a third order determinant formed from any three columns of the matrix may be considered and evaluated. But in this case it will be observed that none of the third order determinant is non-zero. As such the rank of the matrix is not three. Then a second order matrix as indicated below may be considered and evaluated:

$$\begin{vmatrix} 1 & 1 \\ 0 & -2 \end{vmatrix} = -2$$

Since in this case a second order determinant of the dimensional matrix is non-zero, the rank of the matrix is 2 and the number of variables is 4, the number of dimensionless groups is therefore $(4-2) = 2$. The actual dimensionless groups may however be formed by using Buckingham's method.

17.7 SUPERFLUOUS AND OMITTED VARIABLES

The first step in the dimensional analysis of a problem is to decide about the variables that govern the flow phenomenon. The various variables that may occur in the problems concerning fluid flow are categorically listed in Table 17.1. Any phenomenon under consideration may involve only a few of these variables. There is however, no definite rule which may govern the proper selection of variables to be included in any problem, and it has to be carried out purely as guess work though supported by logic. But even for this, one must have enough insight into the mechanism of the problem, so as to explain why and how any of the variables, otherwise considered suitable, influences the phenomenon.

Often variables may be included that really do not have any effect on the phenomenon. Such variables are known as *superfluous variables*. The inclusion of these variables will result in the appearance of too many terms in the final equation, thereby making the whole analysis unnecessarily complicated. These superfluous variables can be eliminated from the analysis only on the basis of the results obtained from the experimental investigation of the problems. This may be made clear by means of an example. In the problem of flow of fluid under pressure through a pipe, if the acceleration due to gravity, g, is taken as one of the variables in addition to those considered in the Illustrative Example 17.1 for the pressure drop Δp, one more π-term may be formed, as indicated below:

$$\pi_5 = V^{a_5} D^{b_5} \rho^{c_5} g$$

which by the usual procedure will be grouped as

$$\pi_5 = \left(\frac{gD}{V^2}\right) \text{ or } \left(\frac{V^2}{gD}\right)$$

Thus the final expression for Δp will have to be modified as

$$\left(\frac{\Delta p}{\rho V^2}\right) = \phi\left(\frac{l}{D}, \frac{k}{D}, \frac{\rho D V}{\mu}, \frac{V^2}{gD}\right)$$

A set of experiments can now be carried out to determine if g really affects the phenomenon. For this purpose the other parameters namely $\left(\frac{l}{D}\right)$, $\left(\frac{k}{D}\right)$ and $\left(\frac{\rho D V}{\mu}\right)$ must be kept constant while $\left(\frac{V^2}{gD}\right)$ is varied. A plot of $\left(\frac{\Delta p}{\rho V^2}\right)$ against $\left(\frac{V^2}{gD}\right)$ prepared from the experimental observations would then indicate a straight line parallel to the axis of $\left(\frac{V^2}{gD}\right)$ showing thereby that g is irrelevant to this problem.

Alternatively, a number of experiments are carried out and all the variables, V, D, ρ, π, l and k changed, then a plot of $\left(\frac{\Delta p}{\rho V^2}\right)$ against $\left(\frac{\rho DV}{\mu}\right)$ for constant values of $\left(\frac{l}{D}\right)$ and $\left(\frac{k}{D}\right)$, would show all the points on a single curve on graph, no matter how changes. Thus it can be concluded that the acceleration due to gravity g, considered additionally in the present problem is a superfluous variable. However if g would have been really significant, a plot of $\left(\frac{\Delta p}{\rho V^2}\right)$ against $\left(\frac{\rho DV}{\mu}\right)$ for constant value of $\left(\frac{l}{D}\right)$ and $\left(\frac{k}{D}\right)$, would show several curves on the graph, one curve for each value of $\left(\frac{V^2}{gD}\right)$.

If, on the contrary, any of the pertinent variable that may actually influence the phenomenon, is omitted at the beginning, the analysis may lead to an incomplete, or even erroneous conclusions. Thus for instance in the same problem of fluid flow through a pipe if viscosity of the flowing fluid is not included in the analysis, the π-term $\left(\frac{\rho DV}{\mu}\right)$ will be eliminated from the general relationship shown above. If the viscosity of the fluid has a significant effect on the phenomenon, the inferences drawn on the basis of the modified relationship viz., $\left(\frac{\Delta p}{\rho V^2}\right) = C_1 f_1\left(\frac{l}{D}, \frac{k}{D}\right)$ will be altogether incorrect. This will however be indicated by the final experimental plot of π-terms which will show a scatter of the points which is not due to experimental error. In such cases a search must then be made for the omitted variable, a new dimensional analysis made and a revised plotting carried out.

A common mistake may be on account of the omission of certain variables that have a practically constant value. These variables are sometimes essential because they combine with other active variables to form dimensionless parameters. The example of gravitational acceleration g, may again be quoted in this context. In combination with certain characteristic length dimension, this variable having almost a constant value, develops a parameter, which has the dimensions of those of (velocity)^2. In certain problems, wherein the effect of gravity prevails, this new dimensional parameter combines with other variables to form an important non-dimensional parameter.

It is thus evident that the method of dimensional analysis does not give any clue as to the correctness of the selection of the relevant variables of a particular phenomenon. It merely tells how the variables should be grouped so that from the experimental investigation it may be decided which of the variables are important ones. This is however, the main *limitation* of the dimensional analysis.

17.8 USE OF DIMENSIONAL ANALYSIS IN PRESENTING EXPERIMENTAL DATA

If there are only two variables involved in any problem, a single curve is sufficient to describe their functional relationship. An addition of one more variable to a problem will require a family of curves relating any two variables with constant values of the third variable as varying parameter. Likewise as the number of variables involved in any problem increases, more and more number of families of curves are required to describe their relative effect on the phenomenon. It is thus obvious that with a large number of variables involved in any problem, it may be rather very difficult to carry out the tests in an orderly manner and to present the experimental data in suitable form, unless the number of variables is reduced. Dimensional analysis is however, extremely useful in reducing the number of variables in a problem by formulating them into non-dimensional parameters. The reduction of the number of variables in a problem provides a systematic scheme for planning laboratory tests and also permits the presentation of experimental results in a more concise and useful form.

Suppose it has been found by the dimensional analysis that some of the variables A, B, C, D etc., influencing any phenomenon, are interconnected by say four non-dimensional groups π_1, π_2, π_3 and π_4 such that

$$\pi_1 = f(\pi_2, \pi_3, \pi_4)$$

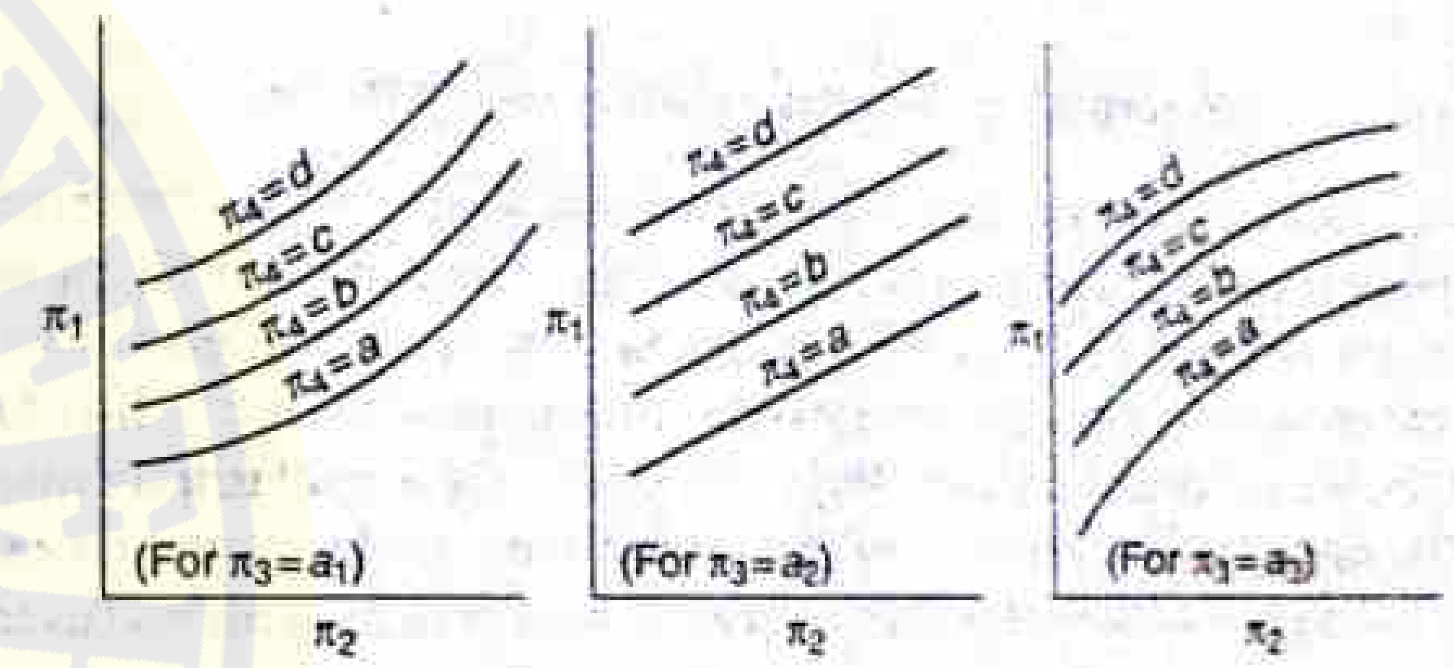


Figure 17.1 *Family of curves plotted between non-dimensional groups*

An experimental investigation will provide certain data which may be presented as shown in Fig.17.1, as a set of groups of π_1 against π_2 with a number of curves for different values of π_4 and each graph being for a certain value of π_3. In this way the effect of every group is presented separately. There will of course be many sets of graphs if the number of relevant non-dimensional π groups is large. If one group, say π_4 is irrelevant, then all the curves for different values of π_4 in each graph will coincide leaving only π_1, π_2 and π_3 as relevant groups.

17.9 MODEL INVESTIGATION

The engineers associated with the design, construction and efficient working of the various types of hydraulic structures (such as dams, spillways etc.) or hydraulic machines (such as turbines, pumps etc.) usually try to find out, in advance, how the structure or the machine would behave when it is actually constructed. For this purpose the engineers have to resort to experimental investigation. Such experiments are also necessitated in the case of the problems which cannot be solved completely simply by theoretical analysis. Obviously the experiments can not be carried out on the full size hydraulic structures or machines, which are proposed to be erected. It is then essential to construct a small scale replica of the structure or the machine and the tests are performed on it to obtain the desired information. The small scale replica of the actual structure or the machine is known as its *model* while the actual structure or machine is called *prototype*. Figure 17.2 shows a prototype of overflow spillway and its model. Mostly the models are much smaller than the corresponding prototypes, but in some cases the models may be larger than the prototypes.

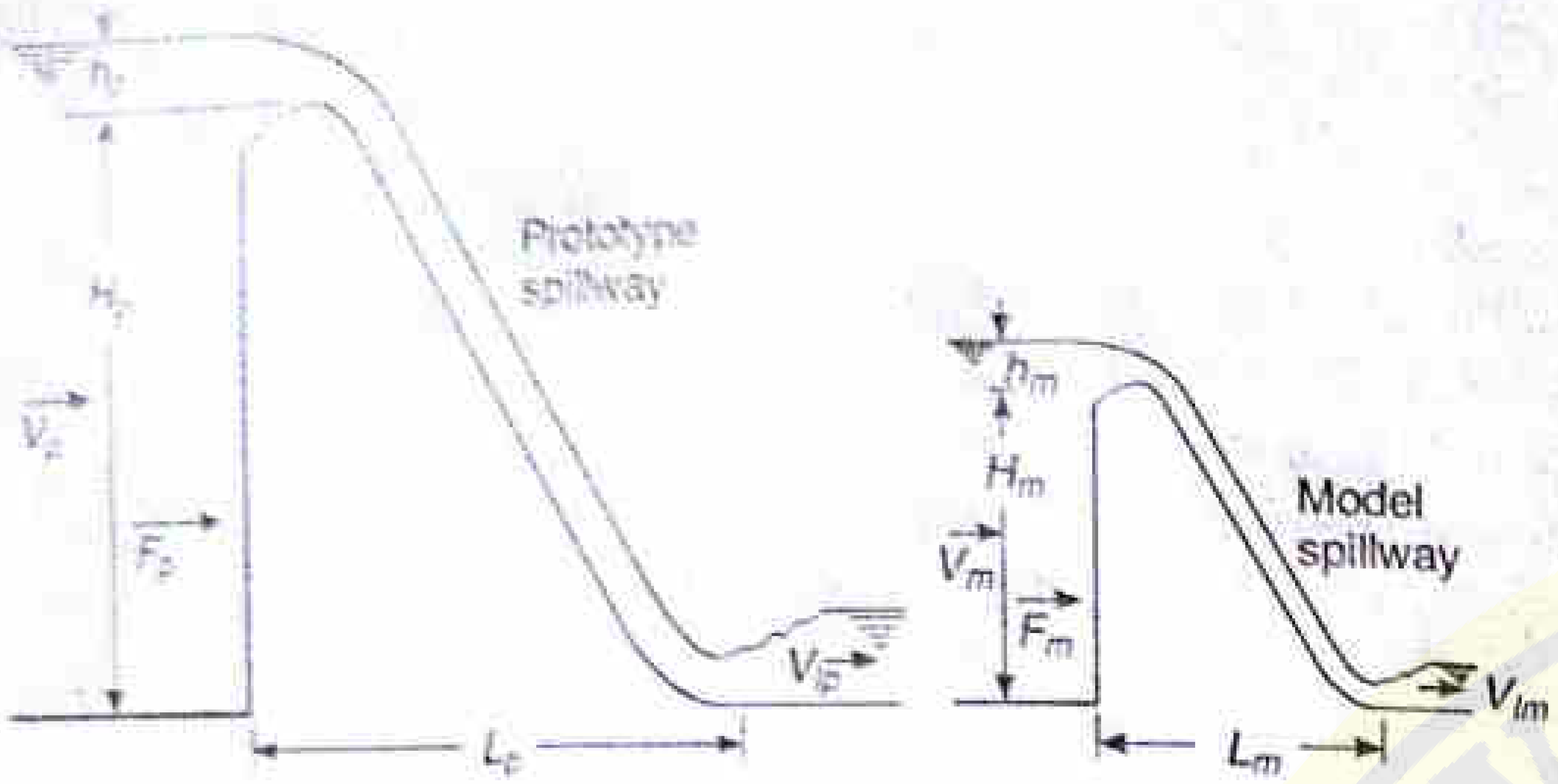


Figure 17.2 *Prototype spillway and its model*

The model tests are quite economical and convenient, because the design, construction and operation of the model may be altered several times if necessary (without incurring much expenditure), till all the defects of the model are eliminated and the most suitable design is obtained. On the basis of the final results obtained from the model tests the design of the prototype may be modified and also it may be possible to predict the behaviour of the prototype. However, the model test results can be utilized to obtain in advance the useful information about the performance of the prototype only if there exists a complete similarity between the model and prototype, which may be achieved as indicated below.

17.10 SIMILITUDE–TYPES OF SIMILARTIES

There are in general three types of similarities to be established for complete similarity to exist between the model and its prototype. These are:

(a) Geometric Similarity

(b) Kinematic Similarity

(c) Dynamic Similarity.

(a) Geometric Similarity. Geometric similarity exists between the model and the prototype if the ratios of corresponding length dimensions in the model and the prototype are equal. Such a ratio is defined as scale ratio and may be written as:

Length scale ratio $$= L_r = \frac{L_m}{L_p} = \frac{b_m}{b_p} = \frac{d_m}{d_p} \text{ etc.}$$

Area scale ratio $$= A_r = \frac{A_m}{A_p} = \left(\frac{L_m \times b_m}{L_p \times b_p}\right) = L_r^2$$

Volume scale ratio $$= V_r = \frac{V_m}{V_p} = \left(\frac{L_m \times b_m \times d_m}{L_p \times b_p \times d_p}\right) = L_r^3$$

in which subscripts m and p correspond to model and prototype respectively.

It will thus be observed that if the model and the prototype are geometrically similar, by mere change of scale they can be superimposed.

(b) Kinematic Similarity. Kinematic similarity exists between the model and the prototype if (1) the paths of the homologous moving particles are geometrically similar, and (2) if the ratios of the velocities as well as acceleration of the homologous particles are equal. Homologous point means corresponding point in the model and prototype. Since velocity and acceleration have magnitude and direction, kinematic similarity implies that velocities and accelerations at corresponding points are parallel and have a constant ratio of magnitude between all the corresponding sets of points. A few useful scale ratios are as follows:

Time scale ratio $$= T_r = \frac{T_m}{T_p}$$

Velocity scale ratio $$= V_r = \frac{V_m}{V_p} = \frac{\frac{L_m}{T_m}}{\frac{L_p}{T_p}} = \frac{L_r}{T_r}$$

Acceleration scale ratio $$= a_r = \frac{a_m}{a_p} = \frac{\frac{L_m}{(T_m)^2}}{\frac{L_p}{(T_p)^2}} = \frac{L_r}{T_r^2}$$

Discharge scale ratio $$= Q_r = \frac{Q_m}{Q_p} = \frac{\frac{(L_m)^3}{T_m}}{\frac{(L_p)^3}{T_p}} = \frac{L_r^3}{T_r}$$

Kinematic similarity can be attained if flownets for the model and the prototype are geometrically similar, which in turn means that by mere change in scale the two flownets—one for the model and other for the prototype—can be superimposed.

(c) Dynamic Similarity. Dynamic similarity exists between the model and the prototype which are geometrically and kinematically similar if the ratio of all the forces acting at homologous points in the two systems viz., the model and the prototype, are equal. Thus for flows to be dynamically similar, the ratios of the various forces acting on the fluid particles in one flow system should be equal to the ratios of similar forces at corresponding points in the other flow system.

In the problems concerning fluid flow, the forces acting may be any one, or a combination of the several of the following forces:

(i) Inertia forces, F_i

(ii) Friction or viscous forces, F_v

(iii) Gravity forces F_g

(iv) Pressure forces F_p

(v) Elastic forces F_e

(vi) Surface tension forces F_s

Inertia force is the force of resistance offered by an inert mass to acceleration. According to Newton's second law of motion, the magnitude of inertia force is equal to the product of the particle mass and particle acceleration and its direction is opposite to the direction of the acceleration of the particle. The conditions required for complete dynamic similarity are developed from the Newton's second law of motion. If in a certain system of flowing fluid, a fluid particle of mass M is subjected to acceleration a, the inertia force F_i of the particle equals Ma. Again, if all the above noted forces exist in the system under consideration, then the resultant force ΣF, acting on the particle, which is the vectorial sum of all the forces acting on the particle, will be equal to the inertia force of the particle i.e.,

$$\Sigma F = F_v + \to F_g + \to F_p + \to F_e + \to F_s = (Ma)$$

For complete dynamic similarity to exist between the model and its prototype, the ratio of inertia forces of the two systems must be equal to the ratio of the resultant forces. Thus the following relation between the forces acting on model and prototype develops:

$$\frac{(\Sigma F)_m}{(\Sigma F)_p} = \frac{(F_v + \to F_g + \to F_p + \to F_e + \to F_s)_m}{(F_v + \to F_g + \to F_p + \to F_e + \to F_s)_p} = \frac{(Ma)_m}{(Ma)_p} \quad \ldots(17.7)$$

In addition to the above noted condition for complete dynamic similarity, the ratio of the inertia forces of the two systems must also be equal to the ratio of individual component forces i.e., the following relationships will be developed:

(i) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_v)_m}{(F_v)_p} \text{ or } \left(\frac{Ma}{F_v}\right)_m = \left(\frac{Ma}{F_v}\right)_p \quad \ldots(17.8)$$

(ii) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_g)_m}{(F_g)_p} \text{ or } \left(\frac{Ma}{F_g}\right)_m = \left(\frac{Ma}{F_g}\right)_p \quad \ldots(17.9)$$

(iii) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_p)_m}{(F_p)_p} \text{ or } \left(\frac{Ma}{Fp}\right)_m = \left(\frac{Ma}{F_p}\right)_p \quad \ldots(17.10)$$

(iv) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_e)_m}{(F_e)_p} \text{ or } \left(\frac{Ma}{F_e}\right)_m = \left(\frac{Ma}{F_e}\right)_p \quad \ldots(17.11)$$

(v) $$\frac{(Ma)_m}{(Ma)_p} = \frac{(F_s)_m}{(F_s)_p} \text{ or } \left(\frac{Ma}{F_s}\right)_m = \left(\frac{Ma}{F_s}\right)_p \quad \ldots(17.12)$$

It may thus be mentioned that when the two systems are geometrically, kinematically and dynamically similar, then they are said to be *completely similar* or *complete similitude* exists between the

two systems. However, as stated earlier dynamic similarity implies geometric and kinematic similarities and hence if two systems are dynamically similar, they may be said to be completely similar. Moreover as indicated later, for complete similitude to exist between the two systems viz., model and prototype, the dimensionless or π terms, formed out of the complete set of variables involved in that phenomenon, must be equal.

17.11 FORCE RATIOS–DIMENSIONLESS NUMBERS

Since inertia force always exists when any mass is in motion, the conditions for dynamic similarity are developed by considering the ratio of the inertia force and any one of the remaining forces noted above. Each of these ratios will obviously be a non-dimensional factor. These ratios are discussed below:

(a) The Inertia Force Ratio

$$\begin{aligned}
\text{Inertia force} &= \text{mass} \times \text{acceleration} \\
&= \rho \times \text{volume} \times \frac{\text{velocity}}{\text{time}} \\
&= \rho \times \frac{\text{volume}}{\text{time}} \times \text{velocity} \\
&= \rho \times AV \times V \\
&= \rho L^2 V^2
\end{aligned}$$

Since area A has the dimensions as L^2

$$\therefore \quad (F_i)_r = \frac{(F_i)_m}{(F_i)_p} = \frac{\rho_m L_m^2 V_m^2}{\rho_p L_p^2 V_p^2} = \rho_r L_r^2 V_r^2$$

(b) Inertia-Viscous Force Ratio–Reynolds Number

Since $\quad F_i = \rho L^2 V^2$

and $\quad F_v = \tau A = \left(\mu \frac{dv}{dy}\right) A = \left(\mu \frac{V}{L}\right) L^2 = (\mu V L)$

where τ and μ are shear stress and absolute viscosity respectively.

$$\therefore \quad \frac{F_i}{F_v} = \frac{\rho L^2 V^2}{\mu V L} = \left[\frac{\rho V L}{\mu}\right] = \left[\frac{VL}{\upsilon}\right]$$

where $(\mu/\rho) = \upsilon$ the kinematic viscosity of the fluid.

This non-dimensional ratio $(\rho VL/\mu)$ or (VL/υ) is called 'Reynolds number' (Re or N_R) in honour of O. Reynolds, a British physicist. Thus Reynolds number signifies the relative predominance of the inertia to the viscous forces occurring in the flow system. The larger the Reynolds number, the greater will be the relative contribution of inertia effects. The smaller the Reynolds number, the greater will be the relative magnitude of viscous force.

(c) Inertia–Gravity Force Ratio–Froude Number

$$\begin{aligned}
F_i &= \rho L^2 V^2 \\
F_g &= \text{mass} \times \text{acceleration due to gravity} \\
&= (\rho \times \text{volume}) \times g
\end{aligned}$$

$$= \rho L^3 g$$

$$\therefore \quad \frac{F_i}{F_g} = \frac{\rho L^2 V^2}{\rho L^3 g} = \frac{V^2}{Lg}$$

The square root of this ratio i.e., $\left(\frac{V}{\sqrt{gL}}\right)$ is known as 'Froude number' (Fr or N_F) after W. Froude, a British naval architect who first applied it to the practical problem of investigating the resistance to ships. Froude number may also be interpreted as a ratio of mean velocity to the velocity of a small wave (or celerity) in quiet fluid. Its value equal to one is considered as critical value.

(d) Inertia–Pressure force Ratio–Euler Number

$$F_i = \rho L^2 V^2$$

$$F_p = \text{pressure intensity} \times \text{area}$$

$$= p \times A = p \times L^2$$

where p represents pressure intensity.

$$\therefore \quad \frac{F_i}{F_p} = \frac{\rho L^2 V^2}{p L^2} = \frac{\rho V^2}{p} = \frac{V^2}{p/\rho}$$

The square root of this ratio i.e., $\frac{V}{\sqrt{p/\rho}}$ is designated as 'Euler number', named after Swiss mathematician and is represented by the symbol Eu or N_E. The reciprocal of this number viz., $\frac{1}{Eu}$ or $\left(\frac{\sqrt{p/\rho}}{V}\right)$ is sometimes known as 'Newton number'. Further ratio of pressure force to inertial force is also known as 'pressure coefficient'.

(e) Inertia–Elasticity Force Ratio–Mach Number

$$F_i = \rho L^2 V^2$$

$$F_e = \text{Bulk modulus of elasticity} \times \text{Area}$$

$$= (K \times A) = K \times L^2$$

where K is the bulk (or volume) modulus of elasticity of the flowing fluid.

$$\frac{F_i}{F_e} = \frac{\rho L^2 V^2}{K L^2} = \frac{V^2}{K/\rho} = \frac{V^2}{C^2}$$

where $C = \sqrt{K/\rho}$, which represents the velocity of sound in that fluid medium whose K and ρ are being considered.

The ratio $\left(\frac{V^2}{C^2}\right)$ or $\frac{V^2}{K/\rho}$ is known as 'Cauchy number'. The square root of this ratio i.e, $\frac{V}{C}$ or $\frac{V}{\sqrt{K/\rho}}$ is known as 'Mach number' (Ma or N_M) in honour of E. Mach an Austrian philosopher. As Mach number represents a ratio of two velocities, its value equal to one, is considered as critical value. This is so because it amounts to the characteristic velocity of the phenomenon becoming equal to the velocity of sound in that fluid medium. It becomes still more significant when $Ma > 1$ i.e., $V > C$, in which case the flow is designated as *supersonic*. For $Ma < 1$ i.e., $V < C$ the flow is termed as *subsonic*, and if $Ma = 1$ i.e., $V = C$ flow is considered to be *sonic*. When $Ma >> 1$ i.e., $V >> C$ then the flow is sometimes termed as *hypersonic*. A higher Mach number signifies the predominance of the effect of compressibility of the fluid. However, when Mach number is relatively small say $Ma < 0.4$, the effect of compressibility of the fluid can be altogether neglected.

(f) Inertia–Surface Tension Force Ratio–Weber Number

$$F_i = \rho L^2 V^2$$

$$F_s = \sigma L$$

where σ represents surface tension

$$\therefore \quad \frac{F_i}{F_s} = \frac{\rho L^2 V^2}{\sigma L} = \frac{\rho V^2 L}{\sigma} = \frac{V^2}{\sigma/\rho L}$$

The square root of this ratio viz., $\frac{V}{\sqrt{\sigma/\rho L}}$ is known as 'Weber number' in honour of W.E. Weber, a German physicist. In analogy with other numbers, a smaller Weber number signifies larger predominance of surface tension force and vice versa. It is represented by We or N_W.

17.12 SIMILARITY LAWS OR MODEL LAWS

The results obtained from the model tests may be transferred to the prototype by the use of model laws which may be developed from the principles of dynamic similarity. The conditions for the existence of complete dynamic similarity between a model and its prototype are denoted by Eqs 17.7 to 17.12. It may, however, be pointed out that, in the case of almost all the hydraulic problems, for which model studies are required to be carried out, it is quite rare that all the above noted forces are simultaneously predominant in the phenomenon. Moreover, debarring certain exceptions, in most of the problems only one force in addition to the inertia force is relatively more significant than the rest of the forces, which may either not exist or may be of negligible magnitude. Under the circumstances the various model laws have been developed depending upon the significant influence of each of the forces on the different phenomena. In the derivation of these model laws, it has been assumed that for equal values of the dimensionless parameters the corresponding flow pattern in model and its prototype are similar.

(a) Reynolds Model Law. For the flows where in addition to inertia, viscous force is the only other predominant force, the similarity of flow in the model and its prototype can be estabilised if the Reynolds number is same for both the systems. This is known as Reynolds model law, according to which

$$(Re)_{model} = (Re)_{prototype} \qquad \text{...(17.13)}$$

or

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

or
$$\frac{\rho_r V_r L_r}{\mu_r} = 1 \quad \text{...(17.14)}$$

or
$$\frac{V_r L_r}{\upsilon_r} = 1 \quad \text{...(17.15)}$$

where the various quantities with subscript *r* represent the corresponding scale ratios.

Equation 17.14 or 17.15 may be used to obtain the scale ratios for various other physical quantities on the basis of Reynolds model law. Some of these scale ratios are shown in Table 17.2.

TABLE 17.2 *Scale Ratios for Models Governed by Reynolds and Froude Model Laws Scale Ratios*

Decription of Quantities	*Reynolds Law*	*Froude Law*
Length	L_r	L_r
Velocity	$m_r/L_r r_r$	$L_r^{1/2} g_r^{1/2}$
Time	$L_r^2 \rho_r/\mu_r$	$L_r^{1/2}/g_r^{1/2}$
Acceleration	$\mu_r^2/\rho_r^2 L_r^3$	g_r
Discharge	$L_r \mu_r/\rho_r$	$L_r^{5/2} g_r^{1/2}$
Force	μ_r^2/ρ_r	$\rho_r L_r^3 g_r$
Work, Energy and Torque	$\mu_r^2 L_r/\rho_r$	$\rho_r L_r^4 g_r$
Pressure intensity	$\mu_r^2/L_r^2 \rho_r$	$\rho_r g_r L_r$
Power	$\mu_r^3/\rho_r^2 L_r$	$\rho_r L_r^{7/2} g_r^{3/2}$

Some of the phenomena for which the Reynolds model law can be a sufficient criterion for similarity of flow in the model and the prototype are—flow of incompressible fluid in closed pipes, motion of submarines completely under water, motion of air planes, and flow around structures and other bodies immersed completely under moving fluids.

(b) Froude Model Law. When the force of gravity can be considered to be the only predominant force which controls the motion in addition to the force of inertia, the similarity of the flow in any two such systems can be established if the Froude number for both the systems is the same. This is known as Froude model law according to which

$$(Fr)_{\text{model}} = (Fr)_{\text{prototype}}$$

or
$$\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_p}{\sqrt{g_p L_p}} \quad \text{...(17.16)}$$

or
$$\frac{V_r}{\sqrt{g_r L_r}} = 1; \text{ or } V_r = \sqrt{g_r L_r} \quad \text{...(17.17)}$$

Since in most of the cases $g_r = 1$, as the value of *g* at the site of model testing will be practically the same as at the site of the proposed prototype, Eq. 17.17 then becomes

$$\frac{V_r}{\sqrt{L_r}} = 1; \text{ or } V_r = \sqrt{L_r} \quad \text{...(17.18)}$$

Equation 17.17 or 17.18 may be used to obtain the scale ratios for various other physical quantities on the basis of Froude model law. Some of these scale ratios are also shown in Table 17.2.

Some of the phenomena for which the Froude model law can be a sufficient criterion for dynamic similarity to be established in the model and the prototype are—free surface flows such as flow over spillways, sluices etc., in which gravity is a motivating force, flow of jet from an orifice or nozzle, problems in which waves are likely to be formed on the surface and problems in which fluids of different mass densities flow over one another.

(c) Euler Model Law. In a fluid system where supplied pressures are the controlling forces in addition to the inertia force and the other forces are either entirely absent or are insignificant, the dynamic similarity is obtained by equating the Euler number for both the model and its prototype. This is known as Euler model law according to which

$$(Eu)_{\text{model}} = (Eu)_{\text{prototype}}$$

or
$$\frac{V_m}{(p_m/\rho_m)^{1/2}} = \frac{V_p}{(p_p/\rho_p)^{1/2}} \quad \text{...(17.19)}$$

or
$$\frac{V_r}{(p_r/\rho_r)^{1/2}} = 1 \quad \text{...(17.20)}$$

Equation 17.20 represents the primary relationship for the Euler model law which may be used to evaluate the scale ratios for various other physical quantities in accordance with the Euler model law.

Euler model law may be contemplated as an essential requirement for establishing dynamic similarity in an enclosed fluid system where the turbulence is fully developed, so that the viscous forces are insignificant, and also the forces of gravity and surface tension are entirely absent.

(d) Mach Model Law. If in any phenomenon only the forces resulting from elastic compression are significant in addition to inertia and all other forces may be neglected, then the dynamic similarity between the model and its prototype may be achieved by equating the Mach number for both the systems. This is known as Mach model law, according to which

$$(Ma)_{\text{model}} = (Ma)_{\text{prototype}}$$

or
$$\frac{V_m}{\sqrt{K_m/\rho_m}} = \frac{V_p}{\sqrt{K_p/\rho_p}} \quad \text{...(17.21)}$$

or
$$\frac{V_r}{\sqrt{K_r/\rho_r}} = 1 \quad \text{...(17.22)}$$

The expression represented by Eq. 17.22 may be considered as the basic relationship for the Mach model law from which the scale ratios for the other physical quantities may be derived.

The similitude based on Mach model law finds extensive application in aerodynamic testing and in phenomena involving velocities exceeding the speed of sound. In addition to this, it is also applied in hydraulic model testing for the cases of unsteady flow, especially water hammer problems.

(e) Weber Model Law. When surface tension effects predominate in addition to inertia force the pertinent similitude law is obtained by equating the Weber number for the model and its prototype, which is known as Weber-model law. Thus according to this model law

$$(We)_{\text{model}} = (We)_{\text{prototype}} \quad \text{...(17.23)}$$

or
$$\frac{V_m}{\sqrt{(\sigma_m/\rho_m L_m)}} = \frac{V_p}{\sqrt{(\sigma_p/\rho_p L_p)}}$$

or
$$\frac{V_r}{\sqrt{\sigma_r / \rho_r L_r}} = 1 \quad \text{... (17.24)}$$

Equation 17.24 may be considered as the basic equation for the Weber model law, and the scale ratios for the various physical quantities may be derived with the help of this equation.

Some of the practical cases where surface tension forces dominate and accordingly the Weber model law can be the sufficient criterion for the dynamic similarity to be established between the model and the prototype are—flow over weirs involving very low heads, very thin sheet of liquid flowing over a surface, capillary waves in channels etc.

17.13 TYPES OF MODELS

In general hydraulic models can be classified under two broad categories:

(i) Undistorted Models

(ii) Distorted Models.

(i) Undistorted Models. An undistorted model is that which is geometrically similar to its prototype, that is, the scale ratios for corresponding linear dimensions of the model and its prototype are same. Since the basic condition of perfect similitude is satisfied, prediction in the case of such models is relatively easy and many of the results obtained from the model tests can be transferred directly to the prototype.

(ii) Distorted Models. Distorted models are those in which one or more terms of the model are not identical with their counterparts in the prototype. Since the basic condition of perfect similitude is not satisfied, the results obtained with the help of a distorted model are liable to distortion and have more qualitative value only.

A distorted model may have either geometrical distortion, or material distortion, or distortion of hydraulic quantities or a combination of these. In geometrical distortion, the distortion can be either of dimension or that of configuration. When different scale ratios are adopted for the longitudinal, transverse and vertical dimensions then it is said to be a distortion of dimensions. Distortion of dimensions is frequently adopted in river models where a different scale ratio for depth is adopted. Such models with vertical scales greater than horizontal scales are called 'vertically exaggerated' models. The distortion of configuration results when the general configuration of the model does not bear a resemblance with its prototype. For example a river model will have distortion of configuration if it is constructed with a bed-slope different from the one given by vertical exaggeration. The material distortion takes place when the physical properties of the corresponding materials in the model and in the prototype do not satisfy the similitude conditions. The material distortion may have to be adopted in the river models constructed for the studies of sediment transport. Also it may not be possible to obtain similitude in respect of certain uncontrollable hydraulic quantities such as time, discharge etc., which may lead to distortion of hydraulic quantities.

Some of the typical examples for which distorted models are required to be prepared are rivers, dams across very wide rivers, harbours, estuaries etc., for which the horizontal dimensions are large in proportion to the vertical ones. Obviously, it is on account of certain limitations imposed, it becomes necessary to design distorted models. The following are some of the reasons for adopting distorted models:

(1) To maintain accuracy in vertical measurements.

(2) To maintain turbulent flow.

(3) To obtain suitable bed material and its adequate movement.

(4) To obtain suitable roughness condition.

(5) To accommodate the available facilities such as space, money, water supply and time.

17.14 MERITS AND LIMITATIONS OF DISTORTED MODELS

The merits of distorted models may be summed up as follows:

(a) The vertical exaggeration results in steeper water surface slopes and magnification of wave heights in models, which can therefore be measured easily and accurately.

(b) Due to exaggerated slopes, the Reynolds numbers of a model is considerably increased and the surface resistance is lowered. This assists in simulation of the flow conditions in the model and the prototype.

(c) In case of distorted models sufficient tractive force can be developed to produce adequate bed movement with a reasonably small model.

(d) Model size can be sufficiently reduced by its distortion, thereby its operation is simplified and also cost is lowered considerably.

Besides the advantages accruing from distortion as indicated above, there are certain *limitations* of distorted models, which are as listed below:

(a) The magnitude and distribution of velocities are incorrectly reproduced because vertical exaggeration causes distortion of lateral distribution of velocity and kinetic energy.

(b) The pressures may not be correctly reproduced in magnitude and direction.

(c) Some of the flow details may not be correctly reproduced because distortion increases longitudinal slopes of model streams thus tending to upset flow regime at a point where artificial model roughness is required to restore it.

(d) Slopes of river bends, earth cuts and dikes are often so steep that they cannot be moulded satisfactorily in sand or other erodible material.

(e) A model wave may differ in type and possibly in action from that of the prototype.

(f) There is an unfavourable psychological effect on the observer.

Although distorted models have a number of limitations, yet if judicious allowances are made in the interpretation of the results obtained from such models, useful information can be obtained, which is not possible otherwise.

17.15 SCALE EFFECT IN MODELS

If complete similitude does not exist between a model and its prototype there will be some discrepancy between the results obtained from the model tests and those which will be indicated by the prototype after its construction. This discrepancy or disturbing influence is called *scale effect*. Some of the factors which may lead to the departure from complete similarity and hence result in the development of the scale effect are as indicated below.

In the case of certain problems if several forces have predominance, the complete similitude will be ensured only if all the pertinent model laws are simultaneously satisfied. However, as indicated below, it is quite difficult to satisfy all the model laws involved in the phenomenon and hence in such cases complete similarity cannot be achieved. Under such circumstances the variables which may be considered to have secondary influence on the phenomenon are neglected, so that the number of the model laws to be satisfied is reduced. But by neglecting these variables some discrepancy or scale effect would be developed between the results obtained from the model tests and those of the prototype. For instance if in a particular problem, both gravity and viscous forces are predominant, then for complete similarity to exist between the model and prototype, both Froude as well as Reynolds model laws should be satisfied simultaneously. For this the necessary condition to be satisfied may be obtained by combining Eqs 17.15 and 17.18 as

$$(L_r)^{3/2} = \nu_r \qquad ...(17.25)$$

Equation 17.25 indicates that once the model scale L_r is selected, a liquid of appropriate viscosity as required by Eq. 17.25 must be found for the model testing; or if a particular liquid is to be used for the model testing then the model scale L_r has to be in accordance with Eq. 17.25. Further if for the model testing the liquid same as that for the prototype is to be used then $L_r = 1$, i.e., the model must be as large as the prototype. It is, however, impossible to find liquids of any desired viscosity and also models with odd scales or full scale models are quite impractical. Under such circumstances the investigator is forced to construct and operate the model in accordance with only one of two model laws, say Froude model law, believed to be more significant. But it would however result in developing the scale effect, which will invalidate the transferring of the results of the model tests to the prototype, without correcting the same.

The scale effect may also be developed in cases where the forces which have practically no effect on the behaviour of the prototype, significantly affected the behaviour of its model. Such a situation may arise in the case of flow over weir models wherein it is apparent that if the head is less than about 15 mm, surface tension plays a proportionately greater part than would be the case in the full size weir having a larger head than in the model.

Often it may not be possible to correctly simulate all the conditions (e.g., roughness) in the model, as that of the prototype. This may also result in developing scale effect if any of these conditions has a pronounced effect on the phenomenon.

In order to detect the presence of such disturbing influences the proposed work may be tried in models with different scales and the resulting scale effects judged from the comparative results so obtained. Besides this the observations collected on models constructed to different scales will also provide an empirical relationship between scale effect and size of model, which may be utilized to correct the results of the model tests.

17.16 APPLICATION OF DYNAMIC SIMILARITY TO SPECIFIC MODEL INVESTIGATIONS

1. Submerged Objects. As an example consider the testing of an aeroplane model in a wind tunnel in order to find the drag F (that is the resistance to its motion) on the prototype. In the case of this phenomenon only viscous and inertia forces are involved. Gravity forces may be disregarded because the fluid concerned being air has very low density and the vertical movements of the air particles as they flow past the aeroplane are small, so that the work done by gravity is negligible. Surface tension is not involved because there is no interface between a liquid and another fluid. Further if it is assumed that it is a low-speed type aeroplane then the effects of compressibility are also negligible. Thus the following functional relationship is obtained for the drag F.

$$F = \phi(V, \rho, \mu, L)$$

where ρ and μ respectively represent the density and viscosity of the fluid used, V represents the relative velocity between the aeroplane (prototype or model) and the fluid and L is some characteristic length.

By adopting the usual method of dimensional analysis the above functional relationship can be expressed in terms of dimensionless parameters as

$$\frac{F}{\rho L^2 V^2} = \phi\left(\frac{\rho V L}{\mu}\right) = \phi(\text{Re}) \qquad ...(17.26)$$

Equation 17.26 is true for both model and prototype. For true dynamic similarity between model and prototype (which are of course geometrically similar) the ratio of force magnitudes at corresponding points must be the same for both, and so, for this case the Reynolds number must be same. Consequently the function ϕ (Re) has the same value for both prototype and model and so $\frac{F}{\rho L^2 V^2}$ is the same in each case. Thus using suffix m for the model and suffix p for the prototype, we may write

$$\frac{F_m}{\rho_m L_m^2 V_m^2} = \frac{F_p}{\rho_p L_p^2 V_p^2} \qquad ...(17.27)$$

This expression is valid only if the test on the model is carried out under such conditions that Reynolds number is the same as for the prototype. Then

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_m V_p L_p}{\mu_p}$$

or

$$V_m = V_p\left(\frac{L_p}{L_m}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{\mu_m}{\mu_p}\right) \qquad ...(17.28)$$

The velocity V_m represented by Eq. 17.28 is known as the *corresponding velocity*. Only when the model is tested at the corresponding velocity as given by Eq. 17.28, the drag F_p for the prototype aeroplane can be predicted from the measured drag F_m for the model aeroplane by using Eq. 17.27.

It may however be stated that if in such problems the effects of compressibility are also important for the prototype, then the Mach numbers must also be indentical in addition to the Reynolds numbers for complete similarity to be established between the model and the prototype. Thus, we have

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

and

$$\frac{V_m}{\sqrt{K_m/\rho_m}} = \frac{V_p}{\sqrt{K_p/\rho_p}}$$

For both the above noted conditions to be satisfied simultaneously

$$\left(\frac{L_p}{L_m}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{\mu_m}{\mu_p}\right) = \frac{V_m}{V_p} = \left(\frac{K_m/\rho_m}{K_p/\rho_p}\right)^{1/2}$$

However, on account of the range of ρ, μ and K for available fluids being limited it is not possible to satisfy the above noted condition. In such circumstances one of the forces is regarded as having secondary effect, so that it may be neglected and the model tests may be carried out in accordance with the remaining model law. This would however result in introducing the scale effect which will have to be accounted for before making any predictions about the prototype behaviour.

2. Partially Submerged Objects. Another interesting example of model testing which is also of wide engineering interest is to find the drag or the resistance experienced by a ship. The resistance experienced by a ship moving through water is generally due to three causes:

(i) Viscosity,

(ii) Surface waves, and

(iii) Eddies formed in the wake.

Due to the viscosity of the fluid viscous forces are set up between the fluid layers close to the ship and those farther away, which results in causing a frictional resistance to the motion of the ship. This part of the total drag is usually termed as *skin friction*. Further a ship being only partly immersed in water, its motion through water gives rise to waves on the surface. The formation of these waves requires energy and, since this energy must be derived from the motion, the ship experiences an increased resistance to its motion through water due to the surface waves. In addition, as the ship moves forward, part of the flow towards the rear of the ship breaks away from its surface to form a 'wake' of eddying motion. The eddies so formed considerably modify the distribution of pressure round the moving ship, which results in developing an additional resistance to its motion through water. Thus for a ship moving through water the total drag is the sum of friction drag, drag due to surface waves and pressure or form drag. Since the ships are generally streamlined, the pressure or form drag is relatively small and within the usual range of Reynolds number, it can be assumed to remain constant. Moreover, it is usual practice to consider this portion of the total drag along with the drag due to surface waves. The formation of surface waves is associated with gravitational action. As such the resistance to the motion of a ship is affected by viscosity as well as gravity.

The total resistance R encountered by a ship will thus be a function of the velocity V of the ship, the viscosity of water μ and its density ρ, some characteristic length L to specify the size of the ship and the gravitational acceleration g. In this problem the compressibility of liquid is however insignificant and the surface tension forces may also be neglected if the model is not too small. The functional relationship for R may thus be written as follows :

$$R = \phi(V, \mu, \rho, L, g)$$

which according to the method of dimensional analysis can be expressed in terms of dimensionless parameters as

$$\frac{R}{\rho L^2 V^2} = \phi_1\left[\left(\frac{\rho V L}{\mu}\right), \frac{V^2}{gL}\right]$$

that is

$$\frac{R}{\rho L^2 V^2} = \phi_1[\text{Re}, F_r^2] \qquad \text{...(17.29)}$$

The total resistance therefore, depends both on the Reynolds number and on the Froude number. Thus, in this case for complete similarity between the prototype and its model the Reynolds number as well as Froude number must be the same for each, that is

$$\frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_p V_p L_p}{\mu_p}$$

and

$$\frac{V_m}{\sqrt{g_m L_m}} = \frac{V_p}{\sqrt{g_p L_p}}$$

Since in practice $g_m = g_p$, the combination of the above equations gives the necessary conditions to be satisfied for complete similarity to be obtained in this case as $\left(\frac{L_m}{L_p}\right)^{3/2} = \left(\frac{\upsilon_m}{\upsilon_p}\right)$ or $L_r^{3/2} = \upsilon_r$. However, as stated earlier this condition cannot be satisfied in actual practice. Therefore, a method suggested by W. Froude may be adopted to overcome the difficulty.

The total resistance R experienced by a ship may be assumed to be consisting of two portions viz., (i) the wave resistance R_w due to the action of the waves; and (ii) the frictional resistance R_f due to frictional effects on the wetted surface of the ship. That is

$$R = R_w + R_f$$

A similar subdivision can also be made for the total resistance r of the model. Thus if r_w and r_f represent the wave and frictional resistance respectively, for the model, then

$$r = r_w + r_f$$

The values of the frictional resistances for the ship and the model can be computed by the expressions noted below :

$$R_f = f_p A_p (V_p)^n \qquad \text{...(17.30)}$$

$$r_f = f_m A_m (V_m)^n \qquad \text{...(17.31)}$$

where f_p and f_m represent the frictional resistance per unit area of a given surface at unit velocity for the ship and the model respectively, n is an arbitrary index, A_p and A_m are the wetted surface areas and V_p and V_m are velocities of the ship and the model respectively. The values of f_p and f_m can be found experimentally by Froude's experiment as discussed in Chapter on 'Flow Through Pipes'.

A geometrically similar model of the ship is then tested by towing it in water at a corresponding speed given by Froude model law viz., $V_m = V_p\left(\frac{L_m}{L_p}\right)^{1/2}$; and total drag r is measured. The wave drag of the model r_w is then found as

$$r_w = (r - r_f)$$

since r_f for the model can be obtained as indicated above. Now for dynamic similarity established between the model and the prototype for wave resistance only (since Froude number is same for both), we have

$$\frac{R_w}{\rho_p L_p^2 V_p^2} = \frac{r_w}{\rho_m L_m^2 V_m^2} \qquad \text{...(17.32)}$$

The wave drag R_w for the prototype ship can thus be obtained from Eq. 17.32 as

$$R_w = \frac{\rho_p}{\rho_m}\left(\frac{L_p}{L_m}\right)^2\left(\frac{V_p}{V_m}\right)^2 r_w$$

But from Froude model law

$$\frac{V_p}{V_m} = \left(\frac{L_p}{L_m}\right)^{1/2}$$

$$\therefore \quad R_w = \frac{\rho_p}{\rho_m}\left(\frac{L_p}{L_m}\right)^3 r_w \qquad \ldots(17.33)$$

Thus R_w for the ship may be computed by Eq. 17.33. Similarly R_f for the ship may be computed by Eq. 17.30. The total drag R for the ship may then be obtained by adding the two as

$$R = R_w + R_f$$

(c) Models of Rivers. River models are frequently used to predict the behaviour of the rivers when some works are proposed to be undertaken on the rivers. In such models the Froude law is the basic similitude criteria, but in addition to this the surface roughness must also be given careful consideration. Mostly the river models are of distorted type, which may have either vertical exaggeration or they may have slope exaggeration. In a model with vertical exaggeration the depth scale is exaggerated so that the model has a depth distortion or vertical distortion. On the other hand in a model with slope exaggeration the model is otherwise geometrically similar but it is provided with a relatively larger slope by tilting it. The various scale ratios for river models with such exaggeration may be obtained as indicated below.

The scale relationships for river models are usually based on Manning's formula, according to which

$$V_r = \frac{R_r^{2/3} S_r^{1/2}}{n_r}$$

For a model with vertical exaggeration since the depth scale is exaggerated, let L_r and D_r represent the horizontal length scale ratio and vertical length (or depth) scale ratio respectively. The hydraulic radius R is dependent upon both horizontal and vertical dimensions, so that R_r cannot be uniquely represented by a simple scale ratio. However, as an approximation for wide rivers $R_r = D_r$. Also the slope scale ratio $S_r = (D_r/L_r)$. Thus velocity scale ratio may be expressed as

$$V_r = \frac{D_r^{7/6}}{n_r L_r^{1/2}}$$

Further according to Froude law

$$V_r = D_r^{1/2}$$

Thus scale ratio for rugosity coefficient n_r becomes

$$n_r = \frac{D_r^{2/3}}{L_r^{1/2}}$$

The value of n_r can thus be controlled by suitably fixing the scale ratios D_r and L_r. However, for an undistorted model the above expression reduces to

$$n_r = L_r^{1/6}$$

which is rather difficult to be satisfied in actual practice.

The value of the scale ratio for rugosity coefficient n_r can also be controlled by tilting the model which is otherwise geometrically similar. Such models are known as tilted models or the models with slope distortion or slope exaggeration. For such models according to Froude Law and Manning's formula

$$V_r = L_r^{1/2} = \frac{R_r^{2/3} S_r^{1/2}}{n_r} = \frac{L_r^{2/3} S_r^{1/2}}{n_r}$$

or

$$n_r = L_r^{1/6} S_r^{1/2}$$

Thus by suitably selecting the slope and length scale ratios the corresponding value of n_r can be obtained from the above expression.

The river models may be classified as (i) *fixed bed models*; and (ii) *movable bed models*. The fixed bed models are those which are built with rigid beds and banks which cannot be eroded. Such models are useful in investigating the problems related to the stretches of rivers having relatively rigid bed and banks so that the movement of sediment is almost negligible or the same is not of much significance. For designing such models the above derived expressions may be used to compute the relevant dimensions of the models.

The movable bed models have both the beds and banks erodible. These models are useful in investigating the problems involving scour, transportation and deposition of the sediment through the action of flowing water. These are the most difficult type of models to design and operate. Moreover, in order to obtain satisfactory results from such models a lot of experience is needed.

The movement of the sediment depends on the tractive force exerted on the river bed by the flowing water. In an undistorted model the tractive force is extremely small. Therefore movable bed models are invariable distorted to develop steep water surface slopes, thereby providing the required tractive force. This requirement may also be accomplished by adopting an exaggerated vertical scale in conjunction with the exaggerated slope scale as indicated earlier. For this purpose different scales are examined in the model with a given material and the one which gives a general bed movement for the model discharge corresponding to which bed movement occurs in the prototype is adopted. The necessary tractive force may however be reduced by using light weight, easily transportable materials with specific gravity only slightly in excess of unity. Some of the substances used in movable bed models for this purpose are coal dust, lignite, saw dust, pumice stone, plexi glass or plastic sand, lac and lac products and powdered bakelite. The effect of these lighter materials is to lessen the need for extreme distortion of the model and also to decrease the time required to complete the experimental investigation.

In addition to this, since the flow in the prototype rivers is mostly turbulent the model scale should be such that in the model also the flow is turbulent even for the lowest discharge. This can be ensured if the Reynolds number for the flow in the model is sufficiently high. A simplified form of this criterion as specified by the U.S. Waterways Experiment Station, Vicksburg is that the flow in the model may be considered as turbulent as long as the value of the parameter VR is more than 0.007, where V is the velocity of flow in m/s and R is hydraulic mean depth in metres. The flow will however be laminar if VR is less than 0.002 and for the value of VR between 0.002 and 0.007 the flow will be in transition.

Moreover, the state of flow prevailing in a model can also be detected by means of another parameter called Karman number or shear (or friction or roughness) Reynolds number which is defined as

$$R_k = \frac{V_* k}{\nu} = \frac{k\sqrt{gRS}}{\nu}$$

where R is hydraulic mean radius, S is bed slope, $V_* = \sqrt{gRS}$ is shear velocity, k is height of equivalent sand grain roughness and ν is kinematic viscosity. If the value of Karman number is more than 50 or 60 for the sand grain type of roughness and more than 100 for any other type of roughness, rough turbulent flow will prevail in the model.

ILLUSTRATIVE EXAMPLES

Example 17.1. *The pressure drop 'Δp' in a pipe of diameter D and length l depends on mass density ρ and viscosity μ of the flowing fluid, mean velocity of flow V and average height k of roughness projections on the pipe surface. Obtain a dimensionless expression for Δp. Hence show that*

$$h_f = \frac{flV^2}{2gD}$$

where h_f is the head loss due to friction $\left(=\frac{\Delta p}{w}\right)$, w is the specific weight of the fluid and f is coefficient of friction.

Solution

The functional equation for the pressure drop Δp can be expressed as

$$\Delta p = \phi(V, l, D, \mu, k, \rho)$$

which in its general form may be written as

$$\phi(\Delta p, V, l, D, \mu, k, \rho) = C$$

Thus the number of variables $n = 7$, and all the variables may be completely described by 3 primary dimensions. Hence $m = 3$. Therefore, the functional relationship can be expressed in terms of $(7-3) = 4$ non-dimensional π terms, i.e.,

$$\phi_1(\pi_1, \pi_2, \pi_3, \pi_4) = C_1$$

Let V, D and ρ be the repeating variables chosen in this case, then the π-terms are grouped as follows:

$$\left.\begin{aligned} \pi_1 &= V^{a_1}\ D^{b_1}\ \rho^{c_1}\ \Delta p \\ \pi_2 &= V^{a_2}\ D^{b_2}\ \rho^{c_2}\ l \\ \pi_3 &= V^{a_3}\ D^{b_3}\ \rho^{c_3}\ k \\ \pi_4 &= V^{a_4}\ D^{b_4}\ \rho^{c_4}\ \mu \end{aligned}\right\}$$

Expressing dimensionally in terms of M-L-T system

$$\pi_1 = (M^\circ L^\circ T^\circ) = \left(\frac{L}{T}\right)^{a_1} (L)^{b_1} \left(\frac{M}{L^3}\right)^{c_1} \left(\frac{M}{LT^2}\right)$$

Equating the exponents of M, L and T on both sides, we have

for $\quad M: 0 = c_1 + 1$

for $\quad L: 0 = a_1 + b_1 - 3c_1 - 1$

for $\quad T: 0 = -a_1 - 2$

From these equations we get

$$a_1 = -2, b_1 = 0, c_1 = -1$$

Hence $\quad \pi_1 = \left(\frac{\Delta p}{\rho V^2}\right)$

By adopting the same procedure, we have

$$\pi_2 = \left(\frac{l}{D}\right)$$

$$\pi_3 = \left(\frac{k}{D}\right)$$

and $\quad \pi_4 = \left(\frac{\mu}{\rho VD}\right)$

Thus the following expression for Δp in terms of non-dimensional parameters is obtained

$$\frac{\Delta p}{\rho V^2} = \phi\left(\frac{l}{D}, \frac{k}{D}, \frac{\mu}{\rho VD}\right)$$

which may also be written as

$$\Delta p = (\rho V^2)\phi\left(\frac{l}{D}, \frac{k}{D}, \frac{\rho VD}{\mu}\right)$$

From experimental observations it is indicated that the drop in pressure Δp is proportional to the length l. Therefore

$$\Delta p = \left(\rho V^2 \times \frac{l}{D}\right)\phi\left(\frac{k}{D}, \frac{\rho VD}{\mu}\right)$$

or

$$\frac{\Delta p}{w} = \left(\frac{\rho V^2 l}{wD}\right)\phi\left(\frac{k}{D}, Re\right)$$

or

$$\frac{\Delta p}{w} = \left(\frac{\rho V^2 l}{2\rho g D}\right)\left[2\phi\left(\frac{k}{D}, Re\right)\right]$$

or

$$\frac{\Delta p}{w} = h_f = \frac{flV^2}{2gD}$$

where f is friction factor which may be expressed as

$$f = \phi\left(\frac{k}{D}, Re\right)$$

Example 17.2. *(a) Show by method of dimensional analysis that the resistance R to the motion of a sphere of diameter D moving with uniform velocity V through a fluid having density ρ and viscosity μ may be expressed as*

$$R = (\rho D^2V^2)\,\phi\left(\frac{\mu}{\rho VD}\right)$$

Also show that the above expression reduces to R = kμVD when the motion is through viscous fluid at low velocity, where k is a dimensionless constant.

(b) Find the viscosity in poise of a liquid through with a steel ball of diameter 1 mm falls, with a uniform velocity of 20 mm/s. The specific gravity of the liquid is 0.91 and that of the steel is 7.8. Given that k = 3π.

Solution

(a) The functional relationship for R may be expressed as

$$R = f(\mu, \rho, D, V)$$

which by Rayleigh's method may be expressed in an exponential form

$$R = k(\mu^a \rho^b D^c V^d)$$

where k is a dimensionless constant.

Now considering the dimensions for each variable on both the sides of the equation in M-L-T system, we get

$$\left[\frac{ML}{T^2}\right] = [M^\circ L^\circ T^\circ]\left[\frac{M}{LT}\right]^a \left[\frac{M}{L^3}\right]^b [L]^c \left[\frac{L}{T}\right]^d$$

For dimensional homogeneity the exponents of each dimension of both sides of the equation must be identical. Thus

for M : $1 = a + b$ $\quad\therefore b = (1-a)$

for T : $-2 = -a - d$ $\quad\therefore d = (2-a)$

for L : $1 = -a - 3b + c + d$ $\quad\therefore c = (2-a)$

Therefore by substitution, we get

$$R = k\left[\mu^a \rho^{(1-a)} D^{(2-a)} V^{(2-a)}\right]$$

$$= k(\rho D^2V^2)\left(\frac{\mu}{\rho DV}\right)^a$$

This may be expressed in the functional form as

$$R = (\rho D^2V^2)\,\phi\left(\frac{\mu}{\rho DV}\right)$$

For viscous flow, $R \sim V$, hence taking $a = 1$, we get

$$R = k(\rho D^2V^2)\left(\frac{\mu}{\rho DV}\right) = k\mu VD.$$

(b) Given $D = 1$ mm; $V = 20$ mm/s; $k = 3\pi$

For uniform velocity of the sphere, resistance R is equal to the gravitational force.

$\therefore$ By substitution, we get

$$3\pi \times \mu \times 20 \times 10^{-3} \times 1 \times 10^{-3} = \pi\left(\frac{1}{2}\times 10^{-3}\right)^3 (7.8 - 0.91)\,9810$$

$$\therefore \quad \mu = \frac{4(0.5)^3(6.89 \times 9.81)}{3 \times 3 \times 20}$$

$$= 0.1878\ \text{N.s/m}^2$$

$$= 1.878\ \text{poise.}$$

Example 17.3. *Find the form of the equation for discharge Q through a sharp-edged triangular notch assuming Q depends on the central angle α of the notch, head H, gravitational acceleration g and on the density ρ, viscosity μ and surface tension σ of the fluid.*

Solution

The functional relationship for the discharge Q may be written as

$$Q = \phi(\rho, H, g, \mu, \sigma, \alpha)$$

which in its most general form may be written as

$$\phi(Q, \rho, H, g, \mu, \sigma, \alpha) = C$$

Thus in the present case the number of variables $n = 7$ and these can be completely described by three fundamental dimensions. Hence $m = 3$ and $(n-m) = 4$.

Thus there are 4 dimensionless π-groups in terms of which the above functional relationship may be expressed.

Choosing ρ, g and H as repeating variables, we have

$$\pi_1 = \rho^{a_1}\ g^{b_1}\ H^{c_1}\ Q;\quad \pi_2 = \rho^{a_2}\ g^{b_2}\ H^{c_2}\ \mu$$

$$\pi_3 = \rho^{a_3}\ g^{b_3}\ H^{c_3}\ \sigma;\quad \pi_4 = \alpha$$

since α itself is a dimensionless quantity

$$\pi_1 = \rho^{a_1}\ g^{b_1}\ H^{c_1}\ Q$$

Inserting fundamental dimensions

$$[M^\circ L^\circ T^\circ] = \left[\frac{M}{L^3}\right]^{a_1}\left[\frac{L}{T^2}\right]^{b_1}[L]^{c_1}\left[\frac{L^3}{T}\right]$$

Equating the exponents of M, L and T, we get

for M : $0 = a_1$

for L : $0 = -3a_1 + b_1 + c_1 + 3$

for T : $0 = -2b_1 - 1$

from which $a_1 = 0,\ b_1 = -\dfrac{1}{2},\ c_1 = -\dfrac{5}{2}$

$$\therefore \quad \pi_1 = \left(\frac{Q}{g^{1/2}H^{5/2}}\right)$$

Similarly, we have

$$\pi_2 = \rho^{a_2} \; g^{b_2} \; H^{c_2} \; \mu$$

or

$$[M^{\circ} L^{\circ} T^{\circ}] = \left(\frac{M}{L^3}\right)^{a_2} \left(\frac{L}{T^2}\right)^{b_2} (L)^{c_2} \left(\frac{M}{LT}\right)$$

Equating the exponents of M, L, and T, we get

for $M: 0 = a_2 + 1$

for $L: 0 = -3a_2 + b_2 + c_2 - 1$

for $T: 0 = -2b_2 - 1$

from which $a_2 = -1, b_2 = -\frac{1}{2}, c_2 = -\frac{3}{2}$

$$\therefore \quad \pi_2 = \left(\frac{\mu}{\rho g^{1/2} H^{3/2}}\right)$$

Again, we have

$$\pi_3 = \rho^{a_3} g^{b_3} H^{c_3} \sigma$$

or

$$[M^{\circ} L^{\circ} T^{\circ}] = \left[\frac{M}{L^3}\right]^{a_3} \left[\frac{L}{T^2}\right]^{b_3} [L]^{c_3} \left[\frac{M}{T^2}\right]$$

Equating the exponents of M, L and T, we get

for $M: 0 = a_3 + 1$

for $L: 0 = -3a_3 + b_3 + c_3$

for $T: 0 = -2b_3 - 2$

from which $a_3 = -1, b_3 = -1, c_3 = -2$

$$\therefore \quad \pi_3 = \left(\frac{\sigma}{\rho g H^2}\right)$$

Thus, we have

$$\left(\frac{Q}{g^{1/2} H^{5/2}}\right) = \left[\left(\frac{\mu}{\rho g^{1/2} H^{3/2}}\right), \left(\frac{\sigma}{\rho g H^2}\right), \alpha\right]$$

or

$$Q = C H^{5/2}$$

where

$$C = (g^{1/2})\,\phi\left[\left(\frac{\mu}{\rho g^{1/2} H^{3/2}}\right), \left(\frac{\sigma}{\rho g H^2}\right), \alpha\right].$$

Example 17.4. *Assuming that the rate of discharge Q of a centrifugal pump is dependent upon the mass density ρ of fluid, pump speed N (rpm), the diameter of impeller D, the pressure p and the viscosity of fluid μ, show using the Buckingham's π-theorem that it can be represented by*

$$Q = (ND^3)\phi\left[\left(\frac{gH}{N^2 D^2}\right), \left(\frac{\upsilon}{ND^2}\right)\right]$$

where H = head and υ = kinematic viscosity of the fluid.

Solution

From the given data, we have

$$Q = \phi(\rho, N, D, p, \mu)$$

Thus there are 6 variables involved in the phenomenon which are expressible in terms of three fundamental dimensions. Hence the above fundamental relationship may be replaced by another one involving only 3 dimensionless π-terms.

Thus taking ρ, N and D as repeating variables, we have

$$\pi_1 = \rho^{a_1} N^{b_1} D^{c_1} Q$$

$$\pi_2 = \rho^{a_2} N^{b_2} D^{c_2} p$$

and

$$\pi_3 = L \quad {}^{2}_{r}$$

Now,

$$\pi_1 = \rho^{a_1} N^{b_1} D^{c_1} Q$$

Inserting fundamental dimensions

$$[M^{\circ} L^{\circ} T^{\circ}] = \left[\frac{M}{L^3}\right]^{a_1} \left[\frac{I}{T}\right]^{b_1} [L]^{c_1} \left[\frac{L^3}{T}\right]$$

Equating the exponents of M, L, and T, we obtain

for $M: 0 = a_1$

for $L: 0 = -3a_1 + c_1 + 3$

for $T: 0 = -b_1 - 1$

from which $a_1 = 0, b_1 = -1, c_1 = -3$

$$\therefore \quad \pi_1 = \left(\frac{Q}{ND^3}\right)$$

By adopting the same procedure, we obtain

$$\pi_2 = \left(\frac{p}{\rho N^2 D^2}\right)$$

and

$$\pi_3 = \left(\frac{\mu}{\rho N D^2}\right)$$

Therefore

$$\frac{Q}{ND^3} = \phi\left[\left(\frac{p}{\rho N^2 D^2}\right)\left(\frac{\mu}{\rho N D^2}\right)\right]$$

since $p = wH = \rho g H$ and $\frac{\mu}{\rho} = \upsilon$, by substituting these expressions, we get

$$Q = (ND^3)\,\phi\left[\left(\frac{gH}{N^2D^2}\right)\left(\frac{\upsilon}{ND^2}\right)\right]$$

Example 17.5. *The equation for specific speed for a turbine is given by*

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

By π-theorem and using variables such as power P, speed N, head H, diameter D of the turbine, density ρ of the fluid and acceleration due to gravity g, deduce the above expression for N_s.

Solution

From the data given, we have

$$N = f(\rho, gH, P, D)$$

wherein gH called shaft work is considered to be one variable only.

Thus in all there are 5 variables involved in the phenomenon which can be completely described by 3 fundamental dimensions. As such the above functional relationship may be replaced by another one involving only 2 dimensionless π-terms.

Choosing ρ, gH and P as repeating variables, we have

$$\pi_1 = \rho^{a_1}(gH)^{b_1}P^{c_1}N$$

and

$$\pi_2 = \rho^{a_2}(gH)^{b_2}P^{c_2}D$$

Now,

$$\pi_1 = \rho^{a_1}(gH)^{b_1}P^{c_1}N$$

Inserting fundamental dimensions

$$[M^0L^0T^0] = \left[\frac{M}{L^3}\right]^{a_1}\left[\frac{L^2}{T^2}\right]^{b_1}\left[\frac{ML^2}{T^3}\right]^{c_1}\left[\frac{1}{T}\right]$$

Equating the powers of M, L and T, we get

for $\quad M: 0 = a_1 + c_1$

for $\quad L: 0 = -3a_1 + 2b_1 + 2c_1$

for $\quad T: 0 = -2b_1 - 3c_1 - 1$

from which $\quad a = -\frac{1}{2}, b_1 = -\frac{5}{4}, c_1 = \frac{1}{2}$

$$\therefore \quad \pi_1 = \left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right]$$

By adopting the same procedure, we get

$$\pi_2 = \left[\frac{D(gH)^{3/4}\rho^{1/2}}{P^{1/2}}\right]$$

Thus, we have

$$\left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right] = \phi\left[\frac{D(gH)^{3/4}\rho^{1/2}}{P^{1/2}}\right]$$

Now if a geometrically similar model of the turbine is considered, then complete similarity between the two will exist when the value of function φ is same for both. According to which the value of the parameter on the L.H.S. must be same for the model and its prototype. That is

$$\left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right]_{model} = \left[\frac{NP^{1/2}}{(gH)^{5/4}\rho^{1/2}}\right]_{prototype}$$

The specific speed N_s is defined as the speed of a geometrically similar turbine which when working under a unit head develops unit power.

Thus substituting $P = 1$, $H = 1$ and $N = N_s$ for the model and assuming g and ρ to be same for the model and the prototype, we get

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

Example 17.6. *Water at 15°C flows at 4 m/s in a 150 mm pipe. At what velocity must oil at 30°C flow in a 75 mm pipe for the two flows to be dynamically similar? Take υ for water at 15°C as 1.145×10^{-6} m²/s and that for oil at 30°C as 3.0×10^{-6} m²/s.*

Solution

Since for the flow of fluids through pipes only viscous and inertia forces are predominent, Reynolds model law is the criterion for similarity. Thus

$$\left(\frac{Vd}{\upsilon}\right)_m = \left(\frac{Vd}{\upsilon}\right)_p$$

By substitution, we get

$$\frac{4\times150\times10^{-3}}{1.145\times10^{-6}} = \frac{V\times75\times10^{-3}}{3.0\times10^{-6}}$$

$$\therefore \quad V = 20.96 \text{ m/s}$$

Example 17.7. *(a) Show that the total resistance R of a floating ship is given by the expression*

$$R = \frac{1}{L_r^3}\left[r + r_f\left(\frac{f_s}{f_m} - 1\right)\right]$$

where L_r is the length scale ratio, r is total resistance for the model, r_f is the frictional resistance of the model, and f_s and f_m are frictional coefficients for the ship and its model respectively. It may be assumed that the velocity index is 2 and the ship and model are towed in the same water.

(b) Hence determine the power required to drive a ship 85 m long, having a wetted surface 1024 m² and moving at 15 m/s, if the resistance of its 1:16 model at the corresponding speed is 276.64 N and f_s = 3.53 and f_m = 3.92. It may also be assumed that both are towed in the same water and only 60% of power is available for propulsion.

Solution

We have

$$R = R_w + R_f$$

and

$$r = r_w + r_f$$

Also

$$R_f = f_s A_p V_p^2 \text{ and } r_f = f_m A_m V_m^2$$

since

$$n = 2.$$

Thus, we have

$$\frac{R_f}{r_f} = \frac{f_s}{f_m}\left(\frac{A_p}{A_m}\right)\left(\frac{V_p}{V_m}\right)^2$$

According to Froude model law

$$\frac{V_m}{V_p} = \sqrt{\frac{L_m}{L_p}} = (L_r)^{1/2} \text{ and } \frac{A_m}{A_p} = (L_r)^2$$

∴ By substitution, we get

$$\frac{R_f}{r_f} = \frac{f_s}{f_m} \times \frac{1}{L_r^3}; R_f = r_f\left(\frac{f_s}{f_m} \cdot \frac{1}{L_r^3}\right)$$

The wave drag of the ship R_w and that of the model r_w can thus be expressed as

$$R_w = R - R_f = R - \left(r_f \frac{f_s}{f_m} \cdot \frac{1}{L_r^3}\right)$$

and

$$r_w = (r - r_f)$$

From Eq. 17.32

$$\frac{R_w}{\rho_p L_p^2 V_p^2} = \frac{r_w}{\rho_m L_m^2 V_m^2}$$

∴

$$R_w = \frac{\rho_p}{\rho_m} \frac{L_p^2}{L_m^2} \frac{V_p^2}{V_m^2} r_w = \frac{r_w}{L_r^3} = \frac{r - r_f}{L_r^3}$$

since

$$\left(\frac{\rho_p}{\rho_m}\right) = 1$$

Thus equating both the values of R_w obtained above, we get

$$R - \left(r_f \frac{f_s}{f_m} \frac{1}{L_r^3}\right) = \frac{r - r_f}{L_r^3}$$

∴

$$R = \frac{1}{L_r^3}\left[r + r_f\left(\frac{f_s}{f_m} - 1\right)\right]$$

(b) Given:

$$L_p = 85 \text{ m}; A_p = 1024 \text{ m}^2; L_r = \frac{1}{16}$$

$$V_p = 15 \text{ m/s}; f_s = 3.53; f_m = 3.92$$

$$r = 276.64 \text{ N}$$

$$r_f = f_m A_m V_m^2$$

$$A_m = A_p L_r^2 = 1024 \times \frac{1}{(16)^2} = 4 \text{ m}^2$$

From Froude model law, we have

$$V_m = V_p (L_r)^{1/2} = 15 \times \left(\frac{1}{16}\right)^{1/2} = 3.75 \text{ m/s}$$

∴

$$r_f = 3.92 \times 4 \times (3.75)^2 = 220.5 \text{ N}$$

Thus by substitution in the above expression, we get

$$R = (16)^3 \left[276.64 + 220.5\left(\frac{3.53}{3.92} - 1\right)\right]$$

$$= 1.043 \times 10^6 \text{ N}$$

∴ Power required to drive the ship

$$= RV \times \frac{100}{60}$$

$$= \frac{1.043 \times 10^6 \times 15 \times 100}{60}$$

$$= 26.075 \times 10^6 \text{ W}$$

$$= 26.075 \times 10^3 \text{ kW}$$

Example 17.8. *A model with length scale ratio, model to prototype, equal to x, of a Mach 2 supersonic aircraft is tested in a wind tunnel, wherein air is maintained at atmospheric temperature and a pressure of y times atmospheric pressure. Determine the speed of the model in the tunnel. Given that the velocity of sound in atmospheric air = z.*

Solution

In the present case

$$Ma = 2 = \frac{V}{C} = \frac{V}{z}; \therefore V = 2z$$

where V is the velocity of the prototype aircraft.

Further according to Boyle's law

$$\frac{1}{\rho} = \frac{y}{\rho_m}; \text{or } \rho_m = y\rho$$

where ρ_m represents the mass density of the compressed air of the wind tunnel.

Dynamic similarity between the model and the prototype will be ensured if Reynolds number is same for both. Thus

$$\left(\frac{\rho VL}{\mu}\right)_m = \left(\frac{\rho VL}{\mu}\right)_p$$

Since absolute viscosity is not affected by pressure changes, $\mu_m = \mu_p = \mu$. Also $L_m = xL_p$; $\rho_p = \rho$ and $\rho_m = y\rho$.

Thus by substitution, we get

$$\frac{(y\rho)(V_m)(xL_p)}{\mu} = \frac{(\rho)(2z)L_p}{\mu}$$

$$\therefore \quad V_m = \left(\frac{2z}{xy}\right)$$

Example 17.9. *A solid sphere of diameter 100 mm moves in water at 5 m/s. It experiences a drag of magnitude 19.62 N (2 kg(f)). What would be the velocity of 5 m diameter sphere moving in air in order to ensure similarity? What will be the drag experienced by it? State which law governs the similarity.*

Take $\rho_{water} = 1000\ kg/m^3$; $\rho_{air} = 1.2\ kg/m^3$; $\upsilon_{air} = 13\upsilon_{water}$

Solution

Since the spheres in both the cases are moving in a submerged state, Reynolds model law governs the similarity, according to which

$$\left(\frac{VL}{\upsilon}\right)_m = \left(\frac{VL}{\upsilon}\right)_p$$

$$\therefore \quad V_m = V_p \times \frac{L_p}{L_m} \times \frac{\upsilon_m}{\upsilon_p}$$

$$= 5 \times \frac{5 \times 1000}{100} \times \frac{1}{13}$$

$$= 19.23\ \text{m/s}$$

From Eq. 17.27 the ratio of the drag for the model and the prototype is

$$\frac{F_m}{F_p} = \frac{\rho_m}{\rho_p} \times \left(\frac{L_m}{L_p}\right)^2 \times \left(\frac{V_m}{V_p}\right)^2$$

$$= \frac{1000}{1.2} \times \left(\frac{100}{5 \times 1000}\right)^2 \times \left(\frac{5}{19.23}\right)^2$$

$$= 0.0225$$

$$\therefore \quad F_p = \frac{19.62}{0.0225} = 872\ \text{N}$$

or

$$F_p = \frac{2}{0.0225} = 88.89\ \text{kg(f)}$$

Example 17.10. *A proposed ship is to have a length of 125 m at the water line and a wetted surface of 1600 m². Its speed is to be 36 km/hour. A 1:25 model of the proposed ship is tested in a towing tank at a velocity corresponding to the wave resistance. The total drag (resistance) of the model was found to be 27 N (2.75 kg(f)). Calculate the total drag of the prototype. The friction drag (resistance) may be computed by the relation*

$$R_f = \frac{1}{2} C_f \rho A V^2$$

where A is area of wetted surface, C_f is average friction drag coefficient given by the following relations

$$C_f = \frac{0.074}{(Re)^{1/5}} \text{ for } Re < 2 \times 10^7$$

and

$$C_f = \frac{0.01}{(Re)^{1/5}} \text{ for } Re > 2 \times 10^7$$

$$\rho_m = 1\,000\ kg/m^3\ (102\ msl/m^3)$$
$$\rho_p = 1027\ kg/m^3\ (104.7\ msl/m^3)$$
$$\upsilon_m = 1.115 \times 10^{-6}\ m^2/s;\ \upsilon_p = 1.121 \times 10^{-6}\ m^2/s$$

Solution

Since the model is tested at a velocity corresponding to the wave resistance, the similarity is based on Froude model law, according to which

$$\left(\frac{V}{\sqrt{gL}}\right)_m = \left(\frac{V}{\sqrt{gL}}\right)_p$$

But

$$V_p = \frac{36 \times 1000}{60 \times 60} = 10\ \text{m/s}$$

$$\therefore \quad V_m = V_p \sqrt{\left(\frac{L_m}{L_p}\right)}$$

$$= 10\sqrt{\left(\frac{1}{25}\right)} = 2\ \text{m/s}$$

The length of the model corresponding to 125 m of the prototype is

$$L_m = \frac{125}{25} = 5\ \text{m}$$

The Reynolds number for the model is

$$(\mathrm{Re})_m = \frac{V_m L_m}{\upsilon_m}$$

$$= \frac{2\times 5}{1.115\times 10^{-6}} = 8.97\times 10^6$$

Since $(\mathrm{Re})_m < 2\times 10^7$, the friction drag coefficient for the model is given by

$$(C_f)_m = \frac{0.074}{(Re_m)^{1/5}}$$

$$= \frac{0.074}{(8.97\times 10^5)^{1/5}} = 0.003\ 01$$

Similarly the Reynolds number for the prototype is

$$(\mathrm{Re})_p = \frac{V_p L_p}{\upsilon_p}$$

$$= \frac{10\times 125}{1.121\times 10^{-6}} = 1.115\times 10^9$$

Since $(\mathrm{Re})_p > 2\times 10^7$, the friction drag coefficient for the prototype is given by

$$(C_f)_p = \frac{0.01}{(Re_p)^{1/5}}$$

$$= \frac{0.01}{(1.115\times 10^9)^{1/5}} = 0.000\ 155$$

SI units

The friction drag for the model is given by

$$(\mathrm{Re})_m = \frac{1}{2}(C_f \rho A V^2)_m$$

$$= \frac{1}{2}\times 0.003\ 01\times 1000\times \frac{1600}{(25)^2}\times (2)^2$$

$$= 15.411\ \mathrm{N}$$

Therefore, wave resistance for the model is

$$(R_w)_m = (27 - 15.411) = 11.589\ \mathrm{N}$$

From Eq. 17.32, we have

$$\left(\frac{R_w}{\rho L^2 V^2}\right)_m = \left(\frac{R_w}{\rho L^2 V^2}\right)_p$$

$$\therefore \quad (R_w)_p = (R_w)_m \times \frac{\rho_p L_p^2 V_p^2}{\rho_m L_m^2 V_m^2}$$

$$= \frac{11.589\times 1027\times (25)^3}{1000}$$

$$= 18.597\times 10^4\ \mathrm{N}$$

The friction drag for the prototype is given by

$$(R_f)_p = \frac{1}{2}(C_f \rho A V^2)_p$$

$$= \frac{1}{2}\times 0.000\ 155\times 1027\times 1600\times (10)^2$$

$$= 1.273\times 10^4\ \mathrm{N}$$

Hence the total drag (or resistance) for the prototype is

$$R_p = (R_f)_p + (R_w)_p$$

$$= 1.273\times 10^4 + 18.597\times 10^4$$

$$= 19.87\times 10^4\ \mathrm{N}$$

Metric units

The friction drag for the model is given by

$$(R_f)_m = \frac{1}{2}(C_f \rho A V^2)_m$$

$$= \frac{1}{2}\times 0.003\ 01\times 102\times \frac{1600}{(25)^2}\times (2)^2$$

$$= 1.572\ \mathrm{kg(f)}$$

Therefore, wave resistance for the model is

$$(R_w)_m = (2.75 - 1.572) = 1.178\ \mathrm{kg(f)}$$

From Eq. 17.32, we have

$$\left(\frac{R_w}{\rho L^2 V^2}\right)_m = \left(\frac{R_w}{\rho L^2 V^2}\right)_p$$

$$\therefore \quad (R_w)_p = (R_w)_m \times \frac{\rho_p L_p^2 V_p^2}{\rho_m L_m^2 V_m^2}$$

$$= \frac{1.178\times 104.7\times (25)^3}{102}$$

$$= 1.889\times 10^4\ \mathrm{kg(f)}$$

The friction drag for the prototype is given by

$$(R_f)_p = \frac{1}{2}\left(C_f \rho A V^2\right)_p$$

$$= \frac{1}{2} \times 0.000\,155 \times 104.7 \times 1600 \times (10)^2$$

$$= 0.1298 \times 10^4 \text{ kg(f)}$$

Hence the total drag (or resistance) for the prototype is

$$R_p = (R_f)_p + (R_w)_p$$
$$= 0.1298 \times 10^4 + 1.889 \times 10^4$$
$$= 2.0188 \times 10^4 \text{ kg(f)}$$

Example 17.11. *A 1:10 scale model of a submarine moving far below the surface of water is tested in a water tunnel. If the speed of the prototype is 8 m/s, determine the corresponding velocity of water in the tunnel. Also determine the ratio of the drag for the model and the prototype. $\upsilon_{sea\ water} = 1.121 \times 10^{-6}$ m²/s; $\upsilon_{water} = 1.00 \times 10^{-6}$ m²/s; $\rho_{sea\ water} = 1027$ kg/m³; $\rho_{sea\ water} = 1000$ kg/m³.*

Would this result be true if the prototype were to move close to the surface? Explain.

Solution

Since the submarine is moving far below the surface of water, it is totally submerged and hence the similarity between the model and its prototype can be established on the basis of Reynolds model law according to which

$$\left(\frac{VL}{\upsilon}\right)_m = \left(\frac{VL}{\upsilon}\right)_p$$

$$\therefore \quad V_m = V_p \times \frac{L_p}{L_m} \times \frac{\upsilon_m}{\upsilon_p}$$

$$= 8 \times 10 \times \frac{1.00 \times 10^{-6}}{1.121 \times 10^{-6}}$$

$$= 71.365 \text{ m/s}$$

From Eq. 17.27 the ratio of the drag for the model and the prototype is

$$\frac{F_m}{F_p} = \frac{\rho_m}{\rho_p} \times \left(\frac{L_m}{L_p}\right)^2 \times \left(\frac{V_m}{V_p}\right)^2$$

$$= \frac{1000}{1027} \times \left(\frac{1}{10}\right) \times \left(\frac{71.365}{8}\right)^2$$

$$= 0.775$$

If the prototype were to move close to the surface then it will be subjected to both viscous and wave resistance. Hence for complete similarity between the model and the prototype, the model will have to be tested in accordance with both Reynolds as well as Froude model laws. As such the results obtained above on the basis of model testing done in accordance with only the Reynolds model law will not be true for this case.

Example 17.12. *In order to estimate the energy loss in a pipeline of 1 m diameter through which kerosene of specific gravity 0.80 and dynamic viscosity 0.02 poise is to be transported at the rate of 2000 litres per second, model tests were conducted on a 0.1 m diameter pipe using water at 20° C. Calculate the discharge required for the model pipe. If the energy head loss in 30 m length of the model pipe is measured as 4.8 m of water, determine the corresponding head loss in the prototype. Also determine the value of Darcy's friction factor for the prototype pipe. Take absolute viscosity of water at 20°C as 1.00×10^{-2} poise.*

Solution

In this case the dynamic similarity will be achieved if Reynolds number in the model and the prototype are equal. Thus

$$\left(\frac{\rho V D}{\mu}\right)_m = \left(\frac{\rho V D}{\mu}\right)_p$$

or

$$\frac{V_m}{V_p} = \frac{\rho_p}{\rho_m} \times \frac{\mu_m}{\mu_p} \times \frac{D_p}{D_m}$$

$$= \frac{1000 \times 0.8}{1000} \times \frac{1.0 \times 10^{-2}}{0.02} \times \frac{1}{0.1}$$

$$= 4.0$$

but

$$V_p = \frac{2000 \times 10^{-3}}{\frac{\pi}{4} \times (1)^2} = 2.546 \text{ m/s}$$

$$\therefore \quad V_m = (2.546 \times 4) = 10.184 \text{ m/s}$$

Therefore, model discharge

$$Q_m = A_m V_m$$

$$= \frac{\pi}{4} \times (0.1)^2 \times 10.184$$

$$= 0.08 \text{ m}^3\text{/s or } 80 \text{ l/s}$$

Resistance to flow

$$R = \rho L^2 V^2 \phi\left(\frac{\rho V D}{\mu}\right)$$

As the Reynolds numbers for the model and the prototype are equal the value of the function ϕ will be same for the both. Hence

$$\left(\frac{R}{\rho L^2 V^2}\right)_m = \left(\frac{R}{\rho L^2 V^2}\right)_p$$

But

$$R = \text{pressure drop due to head loss} \times \text{cross-sectional area}$$

$$= (\rho g h_f)\left(\frac{\pi D^2}{4}\right)$$

where h_f is the head loss. Thus, we have

$$\left(\frac{\rho g h_f \times \frac{\pi D^2}{4}}{\rho L^2 V^2}\right)_m = \left(\frac{\rho g h_f \times \frac{\pi D^2}{4}}{\rho L^2 V^2}\right)_p$$

or

$$\frac{(h_f)_m}{(h_f)_p} = \frac{D_p^2}{D_m^2} \times \frac{L_m^2}{L_p^2} \times \frac{V_m^2}{V_p^2}$$

As

$$\frac{L_m}{L_p} = \frac{D_m}{D_p} = \frac{0.1}{1}$$

$$\therefore \quad \frac{(h_f)_m}{(h_f)_p} = \left(\frac{1}{0.1}\right)^2 \times \left(\frac{0.1}{1}\right)^2 \times \left(\frac{10.184}{2.546}\right)^2$$

$$= 16.0$$

But $(h_f)_m = 4.8$ m of water for a length of 30 m

$$\therefore \quad (h_f)_p = \frac{4.8}{16} = 0.30 \text{ m of kerosene}$$

The corresponding length of the prototype pipe is

$$L_p = \frac{30 \times 1}{0.1} = 300 \text{ m}$$

$\therefore$ The head loss per metre length of the prototype pipe is

$$\frac{(h_f)_p}{L_p} = \frac{0.30}{300} = 0.001 \text{ m of kerosene per m}$$

According to Darcy's formula, the head loss due to friction for the prototype is given by

$$(h_f)_p = \frac{f_p L_p V_p^2}{2 g D_p}$$

$$\therefore \quad f_p = \frac{2g(h_f)_p D_p}{L_p V_p^2}$$

$$= \frac{2 \times 9.81 \times 0.30 \times 1}{300 \times (2.546)^2}$$

$$= 0.003\ 027$$

In the same manner the friction factor f_m for the model pipe may also be computed and it will be found that $f_m = f_p$. This is so because for smooth pipes (which is an implied assumption in this case) the friction factor depends only on Reynolds number, which is same for both the model and the prototype pipes.

Example 17.13. *An overflow structure 550 m long is designed to pass a flow of 4400 m³/s. A 1:20 model of the cross-section of the structure is built in a laboratory channel 0.5 m wide. Calculate the corresponding flow rate for the model if the action of viscosity and surface tension may be neglected. When the model is tested at this flow rate the pressure at a point on the model is observed to be 50 mm of mercury vacuum, how should this be interpreted.*

Solution

This model has been built in a laboratory channel which is 0.5 m wide and has a scale of 1:20 for its cross-section. Obviously in this case the model is a distorted one, with horizontal length scale ratio $L_r = \frac{0.5}{550} = \frac{1}{1100}$, and vertical and transverse length scale ratios being $\frac{1}{20}$ (as given).

Further in this case since gravity is the predominant force, according to Froude model law, we have

$$V_r = (D_r)^{1/2} = \left(\frac{1}{20}\right)^{1/2}$$

Now

$$Q_r = \frac{Q_m}{Q_p}$$

$$= A_r V_r$$

$$= (L_r D_r) \times (D_r)^{1/2}$$

$$= \frac{1}{1100} \times \left(\frac{1}{20}\right)^{3/2}$$

But

$$Q_p = 4400 \text{ m}^3/\text{s}$$

$$\therefore \quad Q_m = \frac{4400}{1100 \times (20)^{3/2}}$$

$$= 0.0447 \text{ m}^3/\text{s} = 44.7 \text{ l/s}$$

If h_m and h_p are the pressure heads at the corresponding points in the model and the prototype, then

$$\frac{h_m}{h_p} = \frac{1}{20}$$

Since at a certain point in the model, $h_m = 50$ mm of mercury vacuum, at the corresponding point in the prototype

$$h_p = (50 \times 20)$$

$$= 1000 \text{ mm of mercury vacuum}$$

From this it may be inferred that if a pressure of such a low magnitude will be allowed to be developed at any point of the prototype structure cavitation will be inevitable (as cavitation may occur even at 580 mm of mercury vacuum). Thus in order to avoid this happening the profile of the model overflow structure should be modified to such an extent that the corresponding pressure at any point on the prototype overflow structure will not drop below the permissible limit.

Example 17.14. *The performance of a spillway of a power project is to be studied by means of a model constructed to a scale of 1:9. Neglecting the viscous and surface tension effect, determine:*

(i) *rate of flow in the model for a prototype discharge of 1000 m³/s; and*

(ii) *the dissipation of energy in the prototype hydraulic jump, if the jump in the model studies dissipates 294.2 watts (0.4 metric horse power).*

Solution

In this case since gravity is the predominant force, according to Froude model law, we have

$$Q_r = L_r^{5/2} g_r^{1/2}$$

$$L_r = \frac{1}{9};\ \text{and } g_r = 1$$

$$\therefore \quad Q_r = \left(\frac{1}{9}\right)^{5/2} \times 1 = \frac{1}{243}$$

Since $Q_r = \dfrac{Q_m}{Q_p}$; and $Q_p = 1000\ \text{m}^3/\text{s}$,

$$\therefore \quad Q_r = \frac{1000}{243} = 4.115\ \text{m}^3/\text{s}$$

Again according to Froude model law the scale ratio for power is given by

$$P_r = \rho_r L_r^{7/2} g_r^{3/2}$$

Since $P_r = (P_m/P_p);\ \rho_r = 1;\ g_r = 1;$

and $P_m = 294.2$ watts

By substitution, we get

$$\frac{294.2}{P_p} = 1 \times \left(\frac{1}{9}\right)^{7/2} \times (1)^{3/2}$$

$$\therefore \quad P_p = 294.2 \times (9)^{7/2}\ \text{watts}$$

$$= 643\,415\ \text{watts}$$

$$= 643.415\ \text{kilo watts}$$

If $P_m = 0.4$ metric horse power, then

$$\frac{0.4}{P_p} = 1 \times \left(\frac{1}{9}\right)^{7/2} \times (1)^{3/2}$$

$$\therefore \quad P_p = 0.4 \times (9)^{7/2}$$

$$= 874.8\ \text{metric horse power}$$

Example 17.15. *A shallow river is 1500 m wide and the maximum depth of flow in it is 5 m. It carries a discharge of 3000 m³/s, the velocity of flow being 1.5 m/s. The model of river is constructed to the horizontal scale of 1:800 and the vertical scale of 1:40. If Manning's n for the bed material in the river is 0.025, find the value of n for the bed material of the model. Check whether the flow in the model is turbulent. The hydraulic mean depth may be assumed to be equal to mean depth of flow.*

Solution

In the present case we have $B_p = 1500$ m, maximum depth of flow = 5 m, $Q_p = 3000\ \text{m}^3/\text{s}$, $V_p = 1.5$ m/s, $n_p = 0.025$, $L_r = (1/800)$ and $D_r = (1/40)$.

$$Q_p = A_p V_p = 3000$$

$$\therefore \quad A_p = \frac{3000}{1.5} = 2000\ \text{m}^2$$

$\therefore$ Mean depth of flow

$$D_p = \frac{A_p}{B_p} = \frac{2000}{1500} = 1.33\ \text{m}$$

Thus $R_p = D_p = 1.33$ m

Further $B_m = \dfrac{B_p}{800} = \dfrac{1500}{800} = 1.875$ m

and $D_m = \dfrac{D_p}{40} = \dfrac{1.33}{40} = 0.033\ \text{m} = R_m$

According to Manning's formula

$$V_r = \frac{1}{n_r} R_r^{2/3} S_r^{1/2}$$

or

$$n_r = \frac{R_r^{2/3} S_r^{1/2}}{V_r}$$

Since $V_r = (D_r)^{1/2}$ in accordance of Froude model law, $R_r = D_r$ and $S_r = (D_r/L_r)$

$$n_r = \frac{D_r^{2/3}}{L_r^{1/2}} = \frac{(800)^{1/2}}{(40)^{2/3}} = 2.418$$

$$\therefore \quad n_m = n_p \times n_r$$

$$= (0.025 \times 2.418) = 0.0605$$

It may however be noted that on account of vertical exaggeration the model will be more rough as compared to the prototype.

Further $V_m = V_r \times V_p = \left(\dfrac{1}{40}\right)^{1/2} \times 1.5 = 0.237$ m/s

$\therefore$ $V_m R_m = (0.237 \times 0.033) = 0.0078$ which is greater than 0.007, and hence the flow in the model will be turbulent.

SUMMARY OF MAIN POINTS

1. Dimensional analysis is a mathematical technique which makes use of the study of dimensions as an aid to the solution of several engineering problems.
2. In the study of fluid mechanics the uses of dimensional analysis are :
   (i) Testing the dimensional homogeneity of any equation of fluid motion.
   (ii) Deriving equations expressed in terms of non-dimensional parameters to show the relative significance of each parameter .
   (iii) Planning model tests and presenting experimental results in a systematic manner in terms of non-dimensional paraeters, thus making it possible to analyse the complex fluid flow phenomenon.
3. The various physical quantities can be described by a set of quantities which are in a sense independent of each other. These quantities are known as *fundamental or primary quantities*. The primary quantities are mass, length, time and temperature designated by the letters M, L, T and θ respectively. All other quantities such as area, volume, velocity, acceleration, force, energy, power, etc. are termed as *derived quantities* or *secondary quantities*, because they can be expressed in terms of primary quantities. The expression for a derived quantity in terms of primary quantities is called the *dimension* of the physical quantity.
4. An equation is said to be dimensionally homogeneous if the dimensions of the terms on its left hand side are the same as the dimensions of the terms on its right hand side.
5. The two methods of dimensional analysis used are
   (i) Rayleigh Method
   (ii) Buckingham π-method.
   In general if there are n variables involved in a phenomenon then these n variables can be grouped together into $(n-m)$ dimensionless groups (or π-terms) when m is the number of the fundamental dimensions required to express the n variables.
6. A small scale replica of the actual structure or machine is known as the *model* while the actual structure or machine is called *prototype*.
   The model test results can be utilized to obtain in advance the useful information about the performance of the prototype only if there exists a complete similarity between the model and the prototype.
7. For complete similarity to exist between the model and its prototype the three types of similarities to be established are
   (i) Geometric similarity
   (ii) Kinematic similarity
   (iii) Dynamic similarity
   Geometric similarity exists between the model and the prototype if the ratios of corresponding length dimensions in the model and the prototype are equal. Such a ratio is defined as scale ratio.
   Kinematic similarity exists between the model and the prototype if (1) the paths of the homologous moving particles are geometrically similar, and (2) if the ratios of the velocities as well as accelerations of the homologous particles are equal. Homologous point means corresponding point in the model and prototype.
   Dynamic similarity exists between the model and the prototype which are geometrically and kinematically similar if the ratio of all the forces acting at homologous points in the model and the prototype are equal. It may be mentioned that when the two systems are geometrically, kinematically and dynamically similar, then they are said to be completely similar of complete similitude exists between the two systems. However, as stated earlier dynamic similarity implies geometric and kinematec similarities and hence if the two systems are dynamically similar, they may be said to be completely similar.
8. In the problems concerning fluid flow the forces acting may be any one or a combination of the several of the following forces
   (i) Inertia forces
   (ii) Friction of viscous forces
   (iii) Gravity forces
   (iv) Pressure forces
   (v) Elastic forces
   (vi) Surface tension forces

9. The various force ratios or dimensionless numbers are given as
   (i) Inertia-Viscous force ratio-Reynolds number
   $$Re \text{ or } N_R = \frac{\rho VL}{\mu}$$
   (ii) Inertia-Gravity force ratio-Froude number
   $$Fr \text{ or } N_F = \frac{V}{\sqrt{gL}}$$
   (iii) Inerita-Pressure force ratio-Euler number
   $$E_u \text{ or } N_E = \frac{V}{\sqrt{p/\rho}}$$
   (iv) Inerita-Elasticity force ratio-March number
   $$Ma \text{ or } N_M = \frac{V}{\sqrt{K/\rho}} = \frac{V}{C}$$
   (v) Inerita-Surface tension force ratio-Weber number We or $N_W = \dfrac{V}{\sqrt{\sigma/\rho L}}$
10. Depending on the significance of each of the forces of the different phenomenon the various model laws have been developed which are given as
   (i) Reynolds Model Law : $\dfrac{\rho_r V_r L_r}{\mu_r} = 1$
   (ii) Froude Model Law : $\dfrac{V_r}{\sqrt{g_r L_r}} = 1$; or $\dfrac{V_r}{\sqrt{L_r}} = 1$, if $g_r = 1$
   (iii) Euler Model Law : $\dfrac{V_r}{\sqrt{p_r/\rho_r}} = 1$
   (iv) Mach Model Law : $\dfrac{V_r}{\sqrt{K_r/\rho_r}} = 1$
   (v) Weber Model Law : $\dfrac{V_r}{\sqrt{\sigma_r/\rho_r L_r}} = 1$
   where
   V_r = velocity scale ratio ;
   L_r = Length scale ratio ;
   ρ_r = mass density scale ratio ;
   μ_r = viscosity scale ratio ;
   g_r = gravitational acceleration scale ratio ;
   p_r = pressure scale ratio ;
   K_r = bulk modulus of elasticity scale ratio ; and
   σ_r = surface tencion scale ratio .
11. Hydraulic models are classified as
   (i) Undistorted models
   (ii) Distorted models
   An undistorted model is that which is geometrically similar to its prototype. Distorted models are those in which one or more terms of the model are not identical with their counterparts in the prototype.
12. The scale-effect is the discrepancy caused if complete similitude does not exist between the model and its prototype.
13. The models of completely submerged objects such as aeroplanes are governed by Reynolds model law.
14. The models of partially submerged objects such as ships are governed by both Reynolds model law as well as Froude model law.
15. River models are generally of distorted type which may have either vertical exaggeration or may have slope exaggeration.

PROBLEMS

17.1 List all the variables that may influence the motion of a moving body fully submerged in a fluid, and by dimensional analysis derive an expression for resistance of its motion.

[*Ans.* $(R/\rho L^2 V^2) = \phi\,(\rho LV/\mu)$]

17.2 Obtain expressions for the resistance to motion of a motor boat, assuming it to be (a) entirely due to viscosity, (b) entirely due to waves, (c) due to viscosity and waves, and (d) due to viscosity and surface tension.

[*Ans.* (a) $(R/\rho L^2 V^2) = \phi\,(\rho LV/\mu)$ (b) $(R/\rho L^2 V^2) = \phi\,(V^2/gL)$ (c) $(R/\rho L^2 V^2) = \phi\,[(\rho LV/\mu, (V^2/gL)]$ (d) $(R/\rho L^2 V^2) = \phi\,[(\rho LV/\mu), (\sigma/\rho V^2 L)]$]

17.3 State and explain Buckingham's π-theorem. Show by p-theorem that a general equation for discharge Q over a weir of any shape is given by

$$Q = (H^{5/2} g^{1/2})\,\phi\left[\left(\frac{\upsilon}{H^{3/2} g^{1/2}}\right), \left(\frac{\sigma}{H^2 \rho g}\right)\right]$$

where

H = head over the weir, υ = kinematic viscosity of the liquid, ρ = mass density of the liquid, and σ = surface tension of the liquid. Hence show that discharge over a rectangular weir of crest length L is given by

$$Q = CdLH^{3/2}$$

17.4 By dimensional analysis show that the torque T on a shaft of diameter d, revolving at speed N in a fluid of viscosity μ and mass density ρ is given by the expression

$$T = (\rho d^5 N^2)\phi\left(\frac{\upsilon}{d^2 N}\right)$$

Use Rayleigh's method. Hence show that power P is given by

$$P = (\rho d^5 N^3)\,\phi\left(\frac{\upsilon}{d^2 N}\right)$$

17.5 The resistance R, to the motion of a supersonic aircraft of length L, moving with a velocity V in air of density ρ, depends on the viscosity μ and bulk modulus of elasticity K of air. Obtain using Buckingham's π-theorem, the following expression for the resistance R.

$$R = (\rho L^2 V^2)\,\phi\left[\left(\frac{\mu}{\rho L V}\right),\left(\frac{K}{\rho V^2}\right)\right]$$

17.6 Torque T exerted on the shaft of a turbine is found to be a function of diameter D, width B and speed N of the turbine runner, density ρ and viscosity μ of the water flowing in the turbine and difference of pressure p at inlet and outlet of turbine. If $p = wH$, where w is specific weight of water and H is the available head on the turbine, show that torque T may be expressed by the equation

$$T = (\rho N^2 D^5)\,\phi\left[\frac{B}{D},\frac{\mu}{\rho D^2 N},\frac{gH}{D^2 N^2}\right]$$

17.7 If the capillary rise h depends on specific weight w, surface tension σ of the fluid and the radius of the tube r, show that

$$\frac{h}{r} = \phi\left(\frac{\sigma}{wr^2}\right).$$

17.8 The efficiency η of a fan depends on density ρ and viscosity μ of fluid, angular velocity ω, diameter D and discharge Q. Obtain a functional relationship for η in terms of dimensionless parameters. [*Ans.* $\eta = \phi[(Q/\omega D^3), (\mu/\rho\omega D^2)]$]

17.9 The performance of a lubricating oil ring depends on the inside diameter of the ring d, shaft speed N, oil discharge Q, density ρ, viscosity μ, surface tension σ and specific weight w of the fluid. Find a functional relationship in terms of dimensionless parameters.

[*Ans.* $\phi[(Q/Nd^3), (\mu/\rho N d^2), (\sigma/\rho N^2 d^3), (w/\rho N^2 d)] = 0$]

17.10 Obtain an expression for the critical depth y_c in a triangular channel which depends on discharge Q, gravitational acceleration g and angle of channel θ. [*Ans.* $y_c^5 = (Q^2/g)\,\phi(\theta)$]

17.11 The discharge Q through a rotating machine such as pump, turbine or compressor depends on the shaft work (gH), power supplied P, speed of rotation N, characteristic length D, mass density ρ and viscosity μ of the fluid. Using Buckingham's π-theorem, obtain a set of dimensionless parameters.

[*Ans.* $(Q/ND^3) = \phi\,[(P/\rho D^5 N^3), (gH/N^2D^2), (\mu/\rho ND^2)]$]

17.12 For laminar flow in a pipe the drop in pressure Δp is a function of pipe length l, its diameter d, mean velocity of flow V, and viscosity of fluid m. Using Rayleigh's method obtain an expression for Δp. [*Ans.* $\Delta p = (\mu V/d)\,\phi\,(l/d)$]

17.13 The drop in pressure Δp due to an obstruction in a pipe depends on the pipe diameter D, average velocity in the pipe V, mass density ρ and viscosity μ of the flowing fluid; and characteristic length of obstruction d. Determine a set of dimensionless parameters.

[*Ans.* $(\Delta p/\rho V^2) = \phi[(d/D), (\rho VD/\mu)]$]

17.14 For small model and (small) prototype of surface ships and overflow structures, the actions of gravity, viscosity and surface tension may be of equal significance. For dynamic similarity between model and prototype, what relation must exist between viscosity, surface tension and model scale?

[*Ans.* $(\mu_r L_r^{1/2}/\sigma_r) = 1$]

17.15 Assuming that the same fluid at same temperature is to be used in the model as in the prototype, what is the force ratio, F_r for the various similarity criteria (a) Reynolds, (b) Froude, (c) Mach, (d) Weber ?

[*Ans.* (a) 1; (b) L_r^3; (c) L_r^2; (d) L_r]

17.16 A ship 150 m long moves in fresh water at 15°C at 36 km/hour. A 1:100 model of this ship is to be tested in a towing basin containing a liquid of sp. gr. 0.90. What viscosity must this liquid have for both Reynolds and Froude model laws to be satisfied? At what speed must the model be towed? If 117.7 watts is required to tow the model at this speed, what power is required by the ship? Dynamic viscosity of water at 15°C is 1.13×10^{-3} N.s/m².

[*Ans.* 1.017×10^{-6} N.s/m²; 1 m/s; 1.308×10^{5} kW]

17.17 A spillway 7.2 m high and 150 m long discharges 2150 m³/s under a head of 4 m. If a 1:16 model of the spillway is to be constructed, find the model dimensions, head over the model and the model discharge.

[*Ans.* (Height = 0.45 m; Length = 9.375 m; Head = 0.25 m; Discharge =2.1m³s]

17.18 A river model constructed with a horizontal scale of 1:5000 and a vertical scale of 1:256 has a discharge scale ratio of $1:2 \times 10^{7}$. Obtain the scale ratios for velocity and rugosity coefficient. If the value of Manning's n for the prototype is 0.025, find the corresponding value of n for the model.

[*Ans.* $V_r = 0.064$; $n_r = 1.713$; $n_m = 0.043$]

17.19 In order to estimate the frictional head loss in a pipe 1 m in diameter, through which castor oil of sp. gr. 0.96 and dynamic viscosity 9.9 poise, is to be transported at the rate of 5000 litres per second, a test was conducted on a pipe of diameter 50 mm using water at 15°C as the model fluid. Calculate the discharge required for the model pipe. If the head loss in 40 m length of the model pipe is measured as 13.6 mm of water, determine the corresponding head loss in the prototype. Also obtain the value of Darcy's friction factor for the prototype. Given absolute viscosily of water at 15°C = 0.0131 poise.

[*Ans.* 0.318 1/s; 26.35 mm of oil per metre length of pipe; 0.013]

17.20 A spillway model is to be made to a scale of $\frac{1}{25}$ across a flume which is 0.5 m wide. The prototype is 15 m high and the maximum head expected is 2 m.

(a) What height of model and what head on model should be used?

(b) If flow over the model at 80 mm head is 0.012 m³/s, what flow per metre length of prototype may be expected?

(c) If the model shows a measured hydraulic jump of 30 mm, how high is jump in prototype?

(d) If the energy dissipated in the model at the hydraulic jump is 110.34 watts, what would be the energy dissipated in the prototype?

[*Ans.* (a) 0.6 m, 80 mm; (b) 3 m³/s per m; (c) 0.75 m; (d) 3.862 × 10³ kW]

17.21 In an open channel having a rectangular section water flowing at 3 m/s with a depth of 0.6 m rises suddenly at a point in the flow to form a jump so that the depth increases to 1.5 m. If a 1:10 scale model of the jump is to be constructed in a laboratory channel, find the flow rate per unit width of the model channel. What are the Froude numbers upstream and downstream of the jump in model and prototype.

[*Ans.* 0.057 m³/s/m; Fr_1 =1.24, Fr_2 = 0.31 (same for the model and the prototype)]

17.22 A distorted model of a rectangular channel of best cross-section is made with horizontal and vertical scale ratios L_r and D_r respectively. Show that velocity ratio $V_r = [(D_r^{7/6})/(n_r L_r^{1/2})]$

17.23 A harbour model has a horizontal scale of 1/150 and a vertical scale of 1/60. Storm waves of amplitude 2 m and velocity 10 m/s strike against the breakwater of the harbour. Determine for the model the amplitude and velocity of the waves, neglecting the viscous effects. Further if the time between the successive high tides in the prototype is 12 hours, what should be the tidal period in the model ?

[*Ans.* 0.28 m ; 1.29 m/s ; 37.18 min]

Flow of Compressible Fluids

Chapter 19

19.1 INTRODUCTION

In general all the fluids are compressible, but in any fluid flow system if the density of the fluid does not change appreciably, then the fluid may be treated as incompressible. However, when the density changes are appreciable, the compressibility of the fluid needs to be taken into account in the analysis of such fluid flow phenomena. Significant density changes may usually be produced in a gas if the velocity (either of the gas itself or of a body moving through it) approaches or exceeds the speed of propagation of sound through the gas, if the gas is subjected to sudden acceleration; or if there are very large changes in elevation. The compressibility is also significant in certain problems involving the flow of water and one such example is the phenomenon of water hammer in pipes. In this chapter the discussion is however confined to only those problems of fluid flow which involve the variations of density of highly compressible fluids such as gases due to the changes of pressure and velocity.

The density of a gas is related to both the pressure and the temperature and all the changes of density are governed by certain laws of thermodynamics. As such in the next paragraph some of the basic relationships of thermodynamics are briefly reviewed.

19.2 BASIC RELATIONSHIP OF THERMODYNAMICS

(I) **Isothermal and Adiabatic Processes.** When the physical properties of a gas such as pressure, density and temperature are changed due to compression or expansion of the gas, it is said to undergo a *process*. A gas may be compressed or expanded either by *isothermal process* or by *adiabatic process*. An isothermal process is that in which the temperature is held constant and it is governed by Boyle's law according to which

$$\left.\begin{aligned} pv &= p_1v_1 = p_2v_2 = \text{constant} \\ (p/\rho) &= (p_1/\rho_1) = (p_2/\rho_2) = \text{constant} \\ (p/w) &= (p_1/w_1) = (p_2/w_2) = \text{constant} \end{aligned}\right\} \quad \text{or} \quad \ldots (19.1)$$

On the other hand if during the course of a process, the gas neither absorbs heat from, nor gives heat to, its surroundings then the process is said to be adiabatic. An adiabatic process is governed by the following law:

$$\left.\begin{aligned} pv^k &= p_1v_1^k = p_2v_2^k = \text{constant} \\ (p/\rho^k) &= (p_1/\rho_1^k) = (p_2/\rho_2^k) = \text{constant} \\ (p/w^k) &= (p_1/w_1^k) = (p_2/w_2^k) = \text{constant} \end{aligned}\right\} \quad \text{or} \quad \ldots (19.2)$$

In the above equation p is pressure, v is specific volume, ρ is mass density, w is weight density, and k is adiabatic index. The adiabatic index is given by $k = (C_p/C_v)$, the ratio of the specific heat at constant pressure C_p and the specific heat at constant volume C_v. For air and other diatomic gases in the usual ranges of temperature and pressure $k = 1.4$.

The process is said to be *reversible* if the gas and its surroundings could subsequently be completely restored to their initial conditions by adding to (or extracting from) the gas exactly the same amount of heat and work taken from (or added to) it during the process. A frictionless adiabatic process is a reversible process and it is often termed as *isentropic process* because as indicated later in this process the entropy does not change. However, a reversible process is ideal, i.e., it is never achieved in practice.

There are some actual processes which are not isentropic (so that pv^k is not constant) but in which the relation between p and v (or ρ) is given to a reasonable approximation by pv^n = constant (or p/ρ^n = constant) where n is a positive constant. Such processes are termed as polytropic processes.

(II) **Equation of State.** The density ρ of a particular gas is related to its absolute pressure p and absolute (or thermodynamic) temperature T by the *equation of state*, which for a perfect gas takes the form

$$p = \rho RT; \text{or } pV = mRT$$

in which R is *gas constant* and V is the volume occupied by the mass m of the gas. The absolute (or thermodynamic) temperature is expressed in 'kelvin' (K) when the temperature is measured in °C and it is given by

$$T° \text{ (abs)} = T° \text{K} = 273.15 + t°\text{C}$$

Further details of the equation of state including the dimensional formula of the gas constant R and its value for air are given in Chapter 1.

(III) **Internal Energy.** The energy possessed by the molecules of a fluid due to molecular activity is termed as *internal energy* or *microscopic energy*. It is to be distinguished from the external or macroscopic energies viz., kinetic and potential energies associated with the fluid mass. The internal energy is a property of the fluid which cannot be measured directly but manifests itself in terms of temperature, that is high or low temperature implies high or low internal energy respectively. If some quantity of heat is supplied to a certain mass of gas, a part of it may be stored in the gas as internal energy thus producing the rise in temperature of the gas and the remaining part of the heat supplied may be utilized in increasing the volume of the gas thus doing some external work.

(IV) **First Law of Thermodynamics.** The first law of thermodynamics states that the total heat H supplied to a gas must be equal to the increase in the internal energy $(I_2 - I_1)$ of the gas plus the external work W done by the gas in expanding. Thus

$$H = (I_2 - I_1) + W \quad \ldots (19.3)$$

However, if instead of heating, the gas is cooled then the heat will be rejected from the gas and hence H will be negative. The value of $(I_2 - I_1)$ may also be negative if the internal energy of the gas decreased. Further W is negative, if the gas is compressed and consequently contracts in volume, which means that work is done on the gas from some external source.

For isothermal process, since the temperature remains constant, $(I_2 - I_1) = 0$, and hence

$$H = W \quad \text{... (19.3 a)}$$

On the other hand, for adiabatic process there being neither addition nor removal of heat, $H = 0$, and hence

$$W + (I_2 - I_1) = 0$$

or

$$W = (I_1 - I_2) \quad \text{... (19.3 b)}$$

The work done by a gas in expanding (or on a gas in contracting) is given by

$$W = \int_{v_1}^{v_2} p\,dv \quad \text{... (19.4)}$$

For isothermal process introducing the value of p from Eq. 19.1 in Eq. 19.4 and integrating it we obtain

$$\left.\begin{aligned} W &= p_1 v_1 \log_e(v_2/v_1) = p_1 v_1 \log_e(p_1/p_2) \\ \text{or} \quad W &= K \log_e(v_2/v_1) = K \log_e(p_1/p_2) \end{aligned}\right] \quad \text{... (19.4 a)}$$

in which K is the constant of Eq. 19.1. Eq. 19.4 (a) gives the work done per unit mass or per unit weight of the gas depending on the specific volume being considered as the volume per unit mass or per unit weight of the gas respectively. Similarly for adiabatic process introducing the value of p from Eq. 19.2 in Eq. 19.4 and integrating it we obtain

$$W = \frac{1}{k-1}(p_1 v_1 - p_2 v_2) \quad \text{... (19.4 b)}$$

Equation 19.4 (b) gives the work done per unit mass of the gas if the specific volume is considered as the volume per unit mass of the gas. However, if the specific volume is considered as the volume per unit weight of the gas then Eq. 19.4 (b) gives the work done per unit weight of the gas and it may then be expressed as

$$W = \frac{1}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) \quad \text{... (19.4 c)}$$

(V) **Entropy.** *Entropy* of a gas may be defined as the measure of the maximum heat energy available for conversion into work. It is a property of the gas and it varies with its absolute temperature and its state. If ΔH is the heat transferred per unit weight of gas in a small interval of time and if T is the absolute temperature of the gas at that instant, then the change in the entropy $\Delta\phi$ is defined by the relation

$$\Delta\phi = \frac{\Delta H}{T}$$

or

$$(\phi_2 - \phi_1) = \int_{T_1}^{T_2} \frac{\Delta H}{T} \quad \text{... (19.5)}$$

For isothermal process since the temperature remains constant

$$(\phi_2 - \phi_1) = \frac{H}{T} \quad \text{... (19.5 a)}$$

But in adiabatic process since there is no transfer of heat, $\Delta H = 0$, hence

$$(\phi_2 - \phi_1) = 0; \ \phi = \text{constant} \quad \text{... (19.5 b)}$$

A process in which the entropy does not change is termed as *isentropic process*. However, the entropy does not change in an adiabatic process, only if it is frictionless. A frictionless adiabatic process is therefore isentropic, which is also reversible.

(VI) **Enthalpy.** The sum of the internal energy and pressure-specific volume product (or pressure divided by mass density; or pressure divided by weight density) (converted into heat units) is termed as *enthalpy*. It is a purely mathematical quantity, the consideration of which simplifies certain calculations, since in several problems of thermodynamics this sum occurs.

19.3 CONTINUITY EQUATION

The equation of continuity which is applicable for compressible as well as incompressible fluids has been developed in Chapter 6 which may be expressed as

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

Further as discussed in Chapter 6 for one dimensional steady flow of compressible fluids the equation of continuity reduces to

$$\rho AV = \text{constant}$$

where ρ is the mass density of the fluid at any section of the flow passage having its cross-sectional area A and the average velocity of flow V. Obviously for the flow of compressible fluid all these viz., ρ, A and V may vary along the flow passage. The one dimensional equation of continuity for the flow of a compressible fluid may also be expressed in differential form as

$$d(\rho AV) = 0$$

19.4 MOMENTUM EQUATION

The momentum equation for compressible fluids is similar to the one derived in Chapter 8 for incompressible fluid. This is because in momentum equation the change in momentum flux is equated to the force required to cause this change. The momentum flux is the product of the mass flux (ρAV) and the velocity V. By the continuity equation the mass flux (ρAV) is constant from section to section. As such the momentum equation is completely independent of the compressibility effects and hence for compressible fluids too the momentum equation (for say x direction) may be expressed as

$$\Sigma F_x = (\rho AVVx)_2 - (\rho AVVx)_1$$

19.5 ENERGY EQUATION

The total energy E_t at a point in a compressible fluid flow system comprises the following:

(1) Kinetic energy, E_k
(2) Potential energy, E_e
(3) Flow or pressure energy, E_p
(4) Molecular energy, E_m

This is may be expressed as

$$E_t = E_k + E_e + E_p + E_m \quad \text{...(19.6)}$$

According to the law of conservation of energy, the total energy at one point in a fluid flow system, plus the energy added E_a minus the energy subtracted E_s between two points, must be equal to the total energy at the second point in the system. That is

$$(E_k + E_e + E_p + E_m)_1 + E_a - E_s = (E_k + E_e + E_p + E_m)_2 \quad \text{...(19.7)}$$

Equation 19.7 shows that energy can be changed from one form to another on the two sides of the equation, but the total must be the same. In other words energy can neither be created nor destroyed.

For incompressible fluids, due to the resistance to the flow of fluid, a part of the energy is converted into heat energy which is considered indirectly as an 'energy loss'. However, for compressible fluids the heat energy produced or supplied may be stored as a molecular energy which is termed as internal energy and it must be included as a specific part of the energy equation. Since the molecular energy of a compressible fluid is developed due to activity of the molecules, it increases with the increase in the temperature of the fluid. Further as indicated earlier, in thermodynamics the sum of the pressure energy (p/w) and the internal energy I is known as enthalpy E_n, and hence $E_n = (p/w) + I$.

Equation 19.7 can thus be written in terms of the energy per unit weight of the flowing fluid as

$$\left(\frac{V^2}{2g} + z + E_n\right)_1 + (E_a - E_s) = \left(\frac{V^2}{2g} + z + E_n\right)_2 \quad \text{...(19.8)}$$

in which $(V^2/2g)$ is kinetic energy, z is potential energy and E_n is enthalpy, each of these being per unit weight of the fluid.

In the case of isothermal flow, since the temperature remains constant $I_1 = I_2$ and from Eq. 19.1 $(p_1/w_1) = (p_2/w_2)$, therefore $E_{n1} = E_{n2}$. Further E_a is usually the heat energy H, which for isothermal flow is equal to the work done and is given by Eq. 19.4 (a). Now if there is no loss of energy i.e., $E_s = 0$ then for a frictionless isothermal flow of fluid the energy equation becomes

$$K \log_e (p_1/p_2) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{...(19.9)}$$

In the case of adiabatic flow there being neither addition nor removal of heat energy $E_a = H = 0$. Also for this case from Eq. 19.3 (b) and 19.4 (c),

$$(I_1 - I_2) = W = \frac{1}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) \quad \text{...(19.10)}$$

Thus for this case

$$(E_{n1} - E_{n2}) = \frac{k}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) \quad \text{...(19.11)}$$

Again if there is no loss of energy i.e., $E_s = 0$, then for a frictionless adiabatic flow (or isentropic flow) of fluid the energy equation becomes

$$\frac{k}{k-1}\left(\frac{p_1}{w_1} - \frac{p_2}{w_2}\right) = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{...(19.12)}$$

Equations 19.9 and 19.12 are same as those derived in Chapter 7.

Further combining Eqs 19.2 and 19.12, we obtain

$$\frac{p_1}{w_1}\frac{k}{k-1}\left[1 - \left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right] = \frac{V_2^2}{2g} - \frac{V_1^2}{2g} + (z_2 - z_1) \quad \text{...(19.13)}$$

For the flow of gas the potential head difference term $(z_2 - z_1)$ is usually small as compared to the pressure head terms, and hence it may be neglected in Eqs 19.9, 19.12 and 19.13.

19.6 PROPAGATION OF ELASTIC WAVES DUE TO COMPRESSION OF FLUID, VELOCITY OF SOUND

If the pressure at a point in a fluid is changed, the new pressure is rapidly transmitted throughout the rest of the fluid. It so happens because any change in pressure causes an elastic or pressure wave reflecting this change to travel through the fluid at a certain velocity. In a completely incompressible fluid the transmission of new pressure would take place instantaneously and hence the elastic or pressure wave would be propagated with infinite velocity. However, in a compressible fluid the elastic or pressure wave is propagated with finite velocity. But even in a compressible fluid the changes of pressure are transmitted quite rapidly and as indicated below the velocity of propagation of the elastic wave in a compressible fluid is equal to the velocity of the sound in that fluid medium.

Consider a long rigid tube of uniform cross-sectional area A fitted with a piston at one end as shown in Fig. 19.1. The tube is filled with a compressible fluid initially at rest. If the piston is moved suddenly to the right with a velocity V_p, a pressure wave would be propagated through the fluid at a velocity C. During a small time interval dt, the piston moves through a distance dx $(= V_p dt)$ while the pressure wave travels ahead of the piston through a distance $dL = (C dt)$.

Before the sudden motion of the piston the fluid within the length dL has an initial density ρ, and its total mass is $(\rho dL\, A)$. After the piston is moved through a distance dx, the fluid density within the compressed region of length $(dL - dx)$ will be increased and becomes $(\rho + d\rho)$. Thus the total mass of fluid in the compressed region will be $[(\rho + d\rho)(dL - dx)A]$. Therefore, according to the principal of continuity

$$\rho (dL) A = (\rho + d\rho)(dL - dx) A$$

or

$$\rho (dL) = (\rho + d\rho)(dL - dx)$$

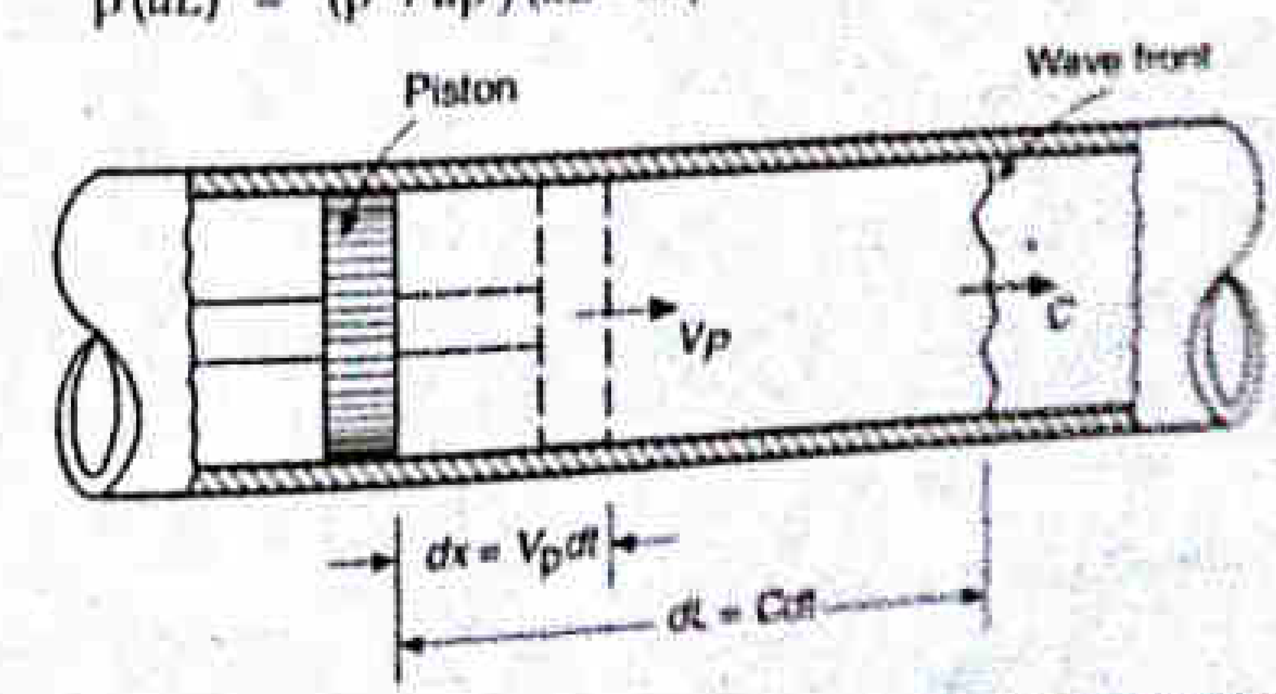


Figure 19.1 *A simplified model for one dimensional elastic or pressure wave propagation*

But $dL = Cdt$ and $dx = V_p dt$, the above equation becomes

$$\rho C dt = (\rho + d_\rho)(C - V_p)dt$$

or

$$0 = Cd\rho - \rho V_p - d\rho V_p$$

Since the velocity V_p of the piston is much smaller than the elastic wave velocity C, the term $(d\rho V_p)$ may be neglected and hence

$$C(d\rho) = \rho V_p;\ \text{or}\ C = \frac{\rho V_p}{d\rho} \quad ...(19.14)$$

Further in the region of compressed fluid, the fluid particles, have acquired velocity which is apparently equal to the velocity of the piston, accompanied by an increase in pressure d_p due to the sudden motion of the piston. Thus applying the impulsemomentum equation for the fluid in the compressed region during dt, we get

$$dpAdt = \rho dLA(V_p - 0)$$

which may be simplified as

$$dp = \rho \frac{dL}{dt} \times V_p = \rho C V_p$$

since $dL = C\,dt$

Thus $$C = \frac{dp}{\rho V_p} \quad ...(19.15)$$

By combining Eqs 19.14 and 19.15

$$C^2 = \frac{dp}{d\rho};\ \text{or}\ C = \sqrt{\frac{dp}{d\rho}} \quad ...(19.16)$$

Thus the velocity of propagation of the elastic or pressure wave in a compressible fluid is given by Eq. 19.16.

In Chapter 1 the bulk modulus of elasticity of fluid is defined as

$$K = -\frac{dp}{dv/v} \quad ...(1.5)$$

Since volume $v \alpha \frac{l}{\rho}$, or $v\rho$ = constant.

Differentiating both sides, we get

$$v d\rho + \rho dv = 0 \quad ...(19.17)$$

$$\therefore \quad \frac{dv}{v} = -\frac{d\rho}{\rho}$$

Combining Eqs 1.5 and 19.17, we have

$$K = \frac{dp}{d\rho/\rho} \quad ...(19.18)$$

From Eqs 19.16 and 19.18 the expression for C becomes

$$C = \sqrt{\frac{K}{\rho}} \quad ...(19.19)$$

Equation 19.19 shows that the velocity of the elastic or pressure wave depends on the bulk modulus of elasticity and the density of the fluid in which it is propagated. Furthermore the velocity of sound (which is also composed of elastic waves of this nature) in any fluid medium is also equal to $\sqrt{K/\rho}$, it may be mentioned that the elastic wave velocity C is same as the velocity of sound or sonic velocity in that fluid.

For isothermal process from Eq. 19.1

$$\frac{p}{\rho} = \text{constant}$$

or

$$\frac{dp}{\rho} - \frac{pd\rho}{\rho^2} = 0;\ \frac{dp}{d\rho} = \frac{p}{\rho} \quad ...(19.20)$$

Thus from Eqs 19.18 and 19.20 for isothermal process, $K = p$ and hence from Eq. 19.19

$$C = \sqrt{(p/\rho)} = \sqrt{RT} \quad ...(19.21)$$

since for perfect gas from the equation of state $p = \rho RT$.

Similarly for isentropic process from Eq. 19.2

$$\frac{p}{\rho^k} = \text{constant}$$

or

$$\frac{dp}{\rho^k} - \frac{kpd\rho}{\rho^{k+1}} = 0$$

or

$$\frac{dp}{d\rho} = \frac{kp}{\rho} \quad ...(19.22)$$

Thus from Eqs 19.18 and 19.22, $K = kp$ and hence from Eq. 19.19

$$C = \sqrt{(kp/\rho)} = \sqrt{kRT} \quad ...(19.23)$$

19.7 MACH NUMBER AND ITS SIGNIFICANCE

The *Mach Number Ma* (or N_M) at a point in the flow is defined as the ratio of velocity V of flow at the point to the velocity of sound or sonic velocity C in that fluid medium corresponding to the values of K and ρ existing at the point where V is measured. That is

$$Ma = \frac{V}{C} \quad ...(19.24)$$

The Mach number is a dimensionless parameter which is the square root of the ratio of inertia force of the flow to the elastic force of fluid. Therefore for two geometrically similar compressible flow systems the dynamic similarity can be obtained when the ratio V/C or Ma is a constant for corresponding points in the two systems. Further it may be mentioned that Mach number is important in those problems in which the velocity of flow is comparable with the velocity of sound or sonic velocity. It may happen in the case of airplanes travelling at very high speed, bullets, projectiles etc. However, if for any flow system the Mach number is less than about 0.4 the effect of compressibility may be neglected for that flow system. Depending on the value of the Mach number, the flow can be classified as follows:

Subsonic Flow: $Ma < 1$; *Sonic Flow:* $Ma = 1$

Supersonic Flow: $1 < Ma < 3$; *Hypersonic Flow:* $Ma > 4$

When the Mach number in flow region is slightly less to slightly greater than unity, the flow is termed as *transonic flow*.

19.8 PROPAGATION OF ELASTIC WAVES DUE TO DISTURBANCE IN FLUID

Elastic or pressure waves are also generated when some disturbance is created in a compressible fluid. Such disturbance may be created when an object moves in a relatively stationary compressible fluid or when a compressible fluid flows past a stationary object, resulting in the generation of the elastic or pressure waves, which are transmitted radially in all directions with a velocity equal to that of sound in that fluid medium. In order to study the pattern of such waves, consider a small object, such as a tiny projectile, moving in a straight line with velocity V through a compressible fluid which is stationary. Let at time $t = 0$ the object be at A, then in time t it will move through a distance AB equal to Vt. During this time the wave which originated from the object when it was at A will grow into the surface of sphere of radius Ct as shown in Fig. 19.2, which also shows the growth of the other waves which will originate from the object at every $t/4$ interval of time as the object moves from A to B. Now depending upon the magnitude of velocity V relative to the velocity of sound C, different patterns of waves will be developed for the following three cases:

(i) **V less than C.** In this case since $V < C$ the elastic waves will travel in the forward direction ahead of the object, and hence as shown in Fig. 19.2 a, the object at point B is inside the sphere of radius Ct and also inside the other spheres formed by the waves started at intermediate times.

(ii) **V equal to C.** In this case both the object and the waves originating from it will reach point B at the same instant of time. As such as shown in Fig. 19.2 b, the object at point B is on the periphery of the sphere of radius Ct and also that of the other spheres formed by the waves started at intermediate times.

(iii) **V greater than C.** In this case the object travels faster than the waves. Thus the distance AB which the object has travelled is more than Ct, and hence as shown in Fig. 19.2 c the object at point B is outside the spheres formed due to the formation and growth of the elastic waves at time $t = 0$ and at the intermediate times. If from point B common tangents such as BT are drawn to all the spheres formed by the waves, then a cone will be generated, having its axis in the direction of motion and its vertex at B, the instantaneous position of the object. In this case the cone therefore advances with the object in the undisturbed fluid. This cone is known as *Mach cone*. The semi-vertex angle α of the cone is known as *Mach angle* which is given by

$$\sin \alpha = \frac{Ct}{Vt} = \frac{C}{V} = \frac{1}{Ma}$$

where Ma represents the Mach number (V/C) as indicated in the preceding section. The limiting value of angle α corresponds to $V = C$ or $Ma = 1$ which is shown in Fig. 19.2 b. The line such as BT is known

as *Mach line*. It is obvious that in this case outside the cone the fluid is completely unaffected by the waves, and hence the region outside the cone is often known as *zone of silence*. The region inside the cone is termed as *zone of action or region of influence*. Across the surface of the Mach cone there is an abrupt change of pressure and density.

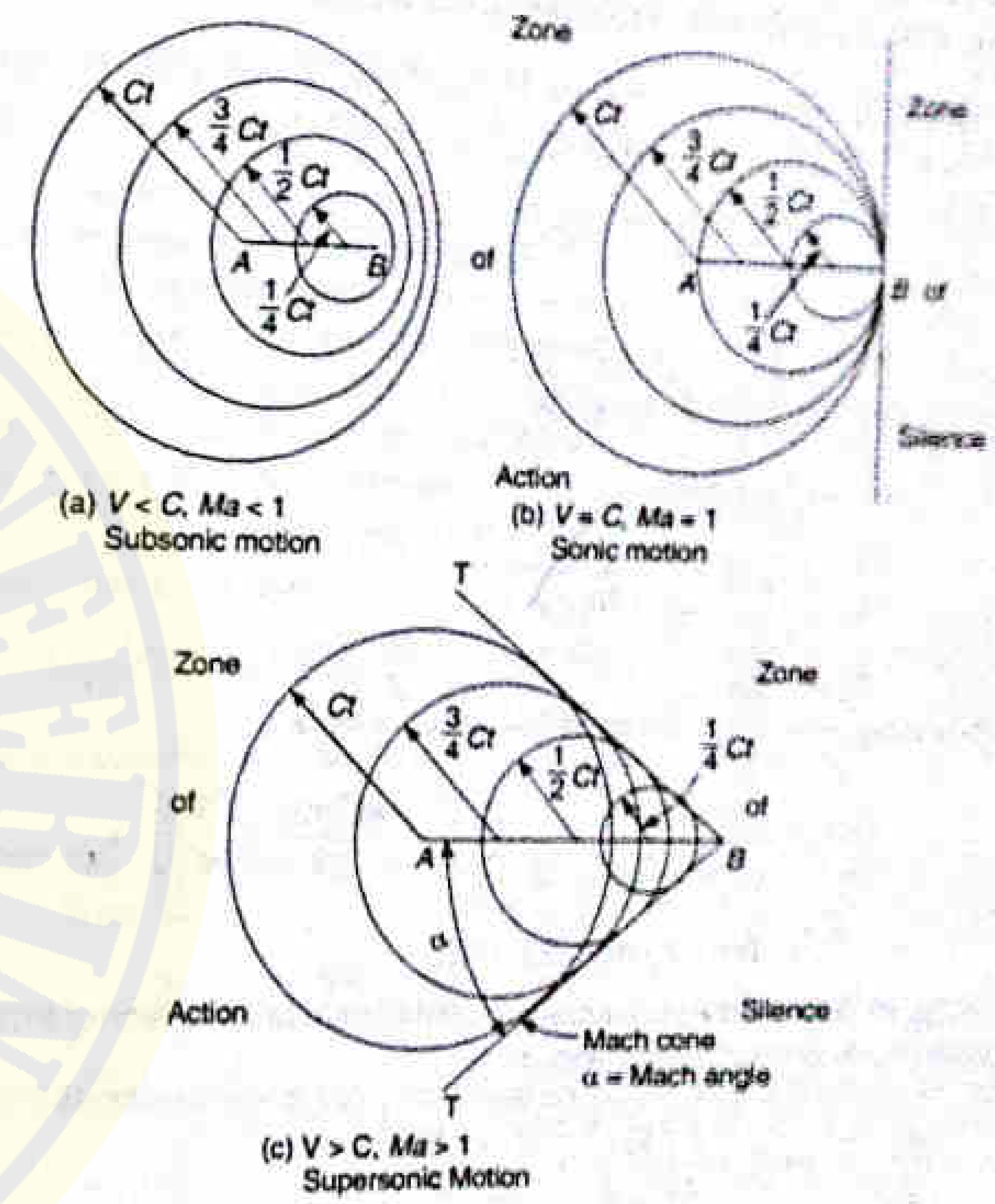


Figure 19.2 *Propagation of elastic or pressure waves by an object moving in a compressible fluid*

It may however be stated that although in the above discussion the object has been assumed to be very small such that is may be considered as a mere point and the resulting waves are also very small, yet the behaviour of the actual larger objects, such as aircraft, travelling at supersonic velocities is also very much similar to that discussed here, with the difference that in the case of larger objects the pressure waves developed are not very small and the same are propagated with a velocity greater than C. Further the Mach cone of an actual object travelling at supersonic velocity has a rounded apex.

19.9 STAGNATION PRESSURE IN CCOMPRESSIBLE FLOWS

Stagnation point is defined as a point in a fluid stream where the velocity of flow is reduced to zero and the kinetic energy is converted into pressure energy. The pressure at stagnation point is known as stagnation pressure. The foregoing principles can be applied to compute the stagnation pressure in a compressible flow. Thus consider the flow of a compressible fluid past an immersed object under frictionless adiabatic (or isentropic) conditions. Let V_1, p_1 and ρ_1 be the velocity, pressure and mass density of the approaching flow and V_2, p_2 and ρ_2 be the corresponding quantities at the point of stagnation. However, at the point of stagnation, the velocity head is converted into pressure head and therefore $V_2 = 0$. Thus from Eq. 19.13 if $(z_1 - z_2)$ is neglected, we have

$$\frac{p_1}{w_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right] = -\frac{V_1^2}{2g}$$

or

$$\frac{p_2}{p_1} = \left[1+\frac{k-1}{k}\left(\frac{V_1^2}{2g}\right)\frac{w_1}{p_1}\right]^{\frac{k}{k-1}}$$

or

$$\frac{p_2}{p_1} = \left[1+\frac{k-1}{2}\left(\frac{V_1^2}{kp_1/\rho_1}\right)\right]^{\frac{k}{k-1}} \qquad ...(19.25)$$

since $w_1 = \rho_1 g$.

Expanding the above expression by binomial theorem, we have

$$\frac{p_2}{p_1} = 1+\frac{k}{2}\left(\frac{V_1}{C_1}\right)^2+\frac{k}{8}\left(\frac{V_1}{C_1}\right)^4+\frac{k(2-k)}{48}\times\left(\frac{V_1}{C_1}\right)^6+.. \qquad ...(19.26)$$

where $C_1^2 = (kp_1/\rho_1)$ from Eq. 19.23

Rearrangement of Eq. 19.26 gives the pressure coefficient (also defined as ratio of pressure to dynamic pressure) or the dimensionless stagnation pressure as

$$\frac{p_2-p_1}{\frac{\rho_1 V_1^2}{2}} = 1+\frac{1}{4}\left(\frac{V_1}{C_1}\right)^2+\frac{2-k}{24}\left(\frac{V_1}{C_1}\right)^4+... \qquad ...(19.27)$$

It may be seen from Eq. 19.27 that when the approach velocity V_1 is small compared with the velocity of sound then it reduces to

$$\frac{p_2-p_1}{\frac{\rho_1 V_1^2}{2}} = 1 \qquad ...(19.28)$$

which is same as for an incompressible fluid. Further the ratio (V_1/C_1) is the Mach number Ma and hence Eq. 19.27 may also be expressed as

$$\frac{p_2-p_1}{\frac{\rho_1 V_1^2}{2}} = 1+\frac{1}{4}Ma^2+\frac{2-k}{24}Ma^4+... \qquad ...(19.29)$$

A comparison of Eq. 19.27 and 19.28 show that as the approach velocity becomes larger, the stagnation pressure becomes greater for compressible fluids with isentropic flow than for incompressible fluids. Thus, for example, when the approach velocity V_1 is half the speed of sound or $Ma = 0.5$, the stagnation pressure is about 6 per cent greater and when the approach velocity approaches the sonic velocity or $Ma = 1$, the stagnation pressure is about 28 per cent greater.

19.10 FLOW OF COMPRESSIBLE FLUID WITH NEGLIGIBLE FRICTION THROUGH A PIPE OF VARYING CROSS-SECTION

When an incompressible fluid flows through a pipe of varying cross-section then the velocity simply varies inversely with the area, since in this case $Q = A_1V_1 = A_2V_2$. But when a compressible fluid flows through a pipe of varying cross-section the variation of velocity is affected by area as well as the density, since in this case according to continuity equation $\rho_1 A_1 V_1 = \rho_2 A_2 V_2$. The effect of the variation of the area of the flow passage on the velocity of flow of a compressible fluid may also be analysed by applying the foregoing principles as dislussed below

For an incompressible flow the continuity equation may be expressed as

$$AV = \text{constant}$$

which can be differentiated to give

$$VdA + AdV = 0; \text{ or } \frac{dA}{A} = -\frac{dV}{V} \qquad ...(19.30)$$

Similarly for compressible flow the continuity equation may be expressed as

$$\rho AV = \text{constant}$$

which can be differentiated to give

$$AVd\rho + \rho VdA + \rho AdV = 0$$

or

$$\frac{dA}{A} = -\frac{dV}{V} - \frac{d\rho}{\rho} \qquad ...(19.31)$$

For an isentropic flow from energy Eq. 19.12 neglecting the elevation difference term $(z_2 - z_1)$, it can be seen that

$$\frac{V^2}{2g}+\frac{k}{k-1}\left(\frac{p}{w}\right) = \text{constant}$$

or

$$\frac{V^2}{2}+\frac{k}{k-1}\left(\frac{p}{\rho}\right) = \text{constant} \qquad ...(19.32)$$

since the acceleration due to gravity g may be treated as constant.

Equation 19.32 can also be differentiated to give

$$VdV+\frac{k}{k-1}\left(\frac{dp}{\rho}-\frac{pd\rho}{\rho^2}\right) = 0 \qquad ...(19.33)$$

Further for an isentropic flow from Eqs 19.22 and 19.23

$$\frac{dp}{d\rho} = \frac{kp}{\rho}; \text{ or } \frac{dp}{\rho} = \frac{kpd\rho}{\rho^2}$$

and
$$C = \sqrt{\frac{kp}{\rho}}$$

Introducing the above expressions in Eq. 19.33, we have

$$-VdV = \frac{kp}{\rho}\frac{d\rho}{\rho} = C^2\frac{d\rho}{\rho}$$

or
$$-\frac{V^2}{C^2}\frac{dV}{V} = \frac{d\rho}{\rho} \qquad ...(19.34)$$

Adding Eqs 19.31 and 19.34, we have

$$\frac{dA}{A} = \left[\frac{V^2}{C^2}-1\right]\frac{dV}{V} = \left[Ma^2-1\right]\frac{dV}{V} \qquad ...(19.35)$$

This important equation is due to Hugoniot. Equations 19.30 and 19.35 give variation of (dA/A) for the flow of incompressible and compressible fluids respectively. The ratios (dA/A) and (dV/V) are respectively the fractional variations in the values of area and flow velocity in the flow passage.

Further in order to study the variation of pressure with the change in flow area, an expression similar to Eq. 19.35 may be obtained for the change of pressure dp as indicated below.

Since
$$\frac{dp}{\rho} = \frac{kpdp}{\rho^2}$$

Introducing the above expression in Eq. 19.33 and simplifying it, the following expression may be obtained

$$\frac{dp}{\rho} = -VdV; \text{ or } \frac{dp}{\rho V^2} = -\frac{dV}{V}$$

Substituting for (dV/V) from Eq. 19.35, we have

$$dp = (\rho V^2)\frac{1}{1-Ma^2}\frac{dA}{A} \qquad ...(19.35\ a)$$

From Eqs 19.35 and 19.35 (a) it is possible to formulate the following conclusions of practical significance:

1. For *subsonic flows* (i.e., $Ma < 1$)

$$\frac{dV}{V} > 0;\ \frac{dA}{A} < 0;\ dp < 0 \text{ (convergent nozzle)}$$

$$\frac{dV}{V} < 0;\ \frac{dA}{A} > 0;\ dp > 0 \text{ (divergent diffuser)}$$

2. For *supersonic flows* (i.e., $Ma > 1$)

$$\frac{dV}{V} > 0;\ \frac{dA}{A} > 0;\ dp < 0 \text{ (divergent nozzle)}$$

$$\frac{dV}{V} < 0;\ \frac{dA}{A} < 0;\ dp > 0 \text{ (convergent diffuser)}$$

3. For *sonic flow* (i.e., $Ma = 1$)

$$\frac{dA}{A} = 0 \text{ (straight flow passage since } dA \text{ must be zero)}$$

and dp = (zero/zero) i.e., indeterminate, but when evaluated, the change of pressure $dp = 0$,

since $dA = 0$ and the flow is frictionless.

Thus the effect of area variation on subsonic flow of compressible fluids is same as that for the flow of incompressible fluids. But the effect of area variation on supersonic flow of compressible fluids is exactly opposite to that for subsonic flow of compressible fluids. In order to accelerate the flow of compressible fluids at supersonic velocity the flow passage must diverge. The divergence of the flow passage is necessary because in supersonic flow of compressible fluid the decrease in the density ρ of the fluid is greater than the increase in the flow velocity, and the area of flow passage must increase for the mass flow rate (ρVA) to remain constant.

When $V = C$ so that $Ma = 1$, dA must be zero and since the second derivative of A is positive, A must be minimum. It thus means that if the velocity of flow equals the sonic velocity anywhere it must therefore do so where the cross-section is of minimum area. However, dA could also be zero when $dV = 0$. As such at a position of minimum cross-section the velocity is either equal to the sonic velocity or it is a maximum velocity for subsonic flow or it is a minimum velocity for supersonic flow.

From the foregoing discussion it may be seen that in order to convert an initially subsonic flow of a compressible fluid in a nozzle into a supersonic flow, it is necessary to have a convergent passage followed by a divergent one. But this conversion will take place only if at the throat where the area is minimum the velocity of flow is equal to sonic velocity. Otherwise the divergent passage will have only the subsonic flow of compressible fluid and then it will simply function as a diffuser where the flow is decelerated.

19.11 FLOW OF COMPRESSIBLE FLUID IN CONVERGENT—DIVERGENT PASSAGES

The foregoing discussion may be applied to study the flow of compressible fluid through a convergent-divergent nozzle. Consider a convergent passage or nozzle as shown in Fig. 19.3 (a), through which a compressible fluid is flowing. Let the initial velocity V_1 of the flow in the approach section be small in comparison with the exit velocity V_2 at the throat and let the flow accelerate subsonically in the convergent passage. Thus for reversible adiabatic process with $V_1 = 0$, from energy Eq. 19.13, we obtain

$$\frac{V_2^2}{2g} = \frac{p_1}{w_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{(k-1)/k}\right] \qquad ...(19.36)$$

Since this is an adiabatic process for which from Eq. 19.2

$$\frac{p_1}{w_1^k} = \frac{p_2}{w_2^k}$$

Hence Eq. 19.36 can be written as

$$\frac{V_2^2}{2g} = \frac{p_2}{w_2}\frac{k}{k-1}\left[\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right]$$

or

$$\frac{V_2}{\sqrt{(kp_2/\rho_2)}} = \left[\frac{2}{k-1}\left\{\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right\}\right]^{1/2}$$

or

$$\frac{V_2^2}{2g} = \left[\frac{2}{k-1}\left\{\left(\frac{p_1}{p_2}\right)^{(k-1)/k} - 1\right\}\right]^{1/2} \quad ...(19.37)$$

in which $C_2 = \sqrt{(kp_2/\rho_2)}$ is the speed of sound in the jet. From Eq. 19.37 it can be seen that the velocity of the jet V_2 relative to the speed of sound C_2 is dependent only upon the pressure ratio (p_1/p_2).

From continuity equation the mass rate of flow G at the throat is given as

$$G = \rho_2 A_2 V_2$$

which by combining Eqs. 19.36 and 19.2 may be expressed as

$$\frac{G}{A_2} = \rho_2 V_2 = \sqrt{\frac{2k}{k-1}(p_1\rho_1)\left\{\left(\frac{p_2}{p_1}\right)^{2/k} - \left(\frac{p_2}{p_1}\right)^{(k+1)/k}\right\}} \quad ...(19.38)$$

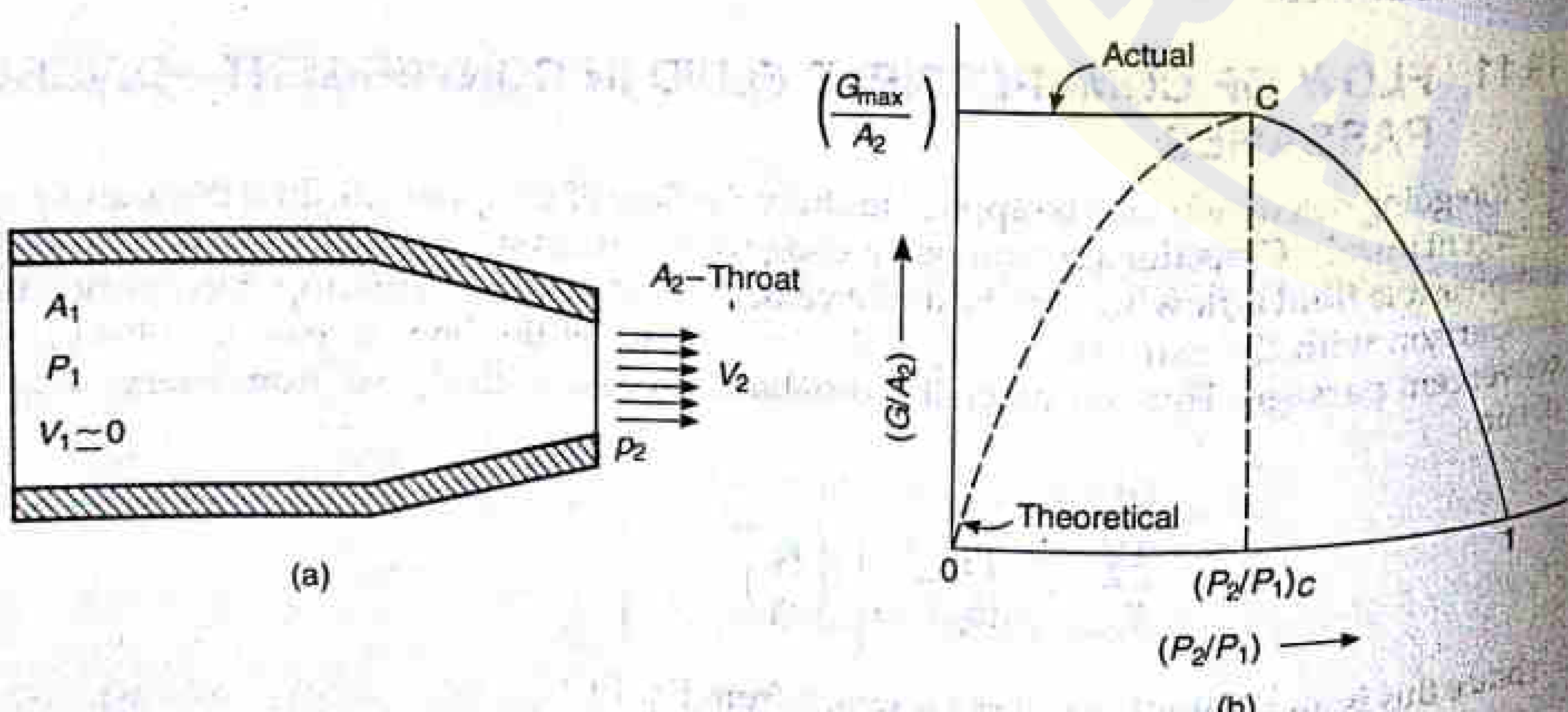


Figure 19.3 *(a) Flow of compressible fluid in a convergent nozzle; (b) Variation of mass rate of flow of compressible fluid* (G/A$_2$) *with pressure ratio* (p_2/p_1)

Equation 19.38 represents the mass rate of flow per unit area (G/A_2) at the throat, from which it may be seen that for any given nozzle with known conditions at the approach section 1 the mass rate of flow depends on the pressure ratio (p_2/p_1). A plot of (G/A_2) versus (p_2/p_1) is shown in Fig. 19.3 (b). It can be seen from the plot that $(G/A_2) = 0$ for $(p_2/p_1) = 0$ and $(p_2/p_1) = 1$; and as the throat pressure p_2 is reduced from $p_2 = p_1$ (i.e., $p_2/p_1 < 1$) the mass flow increases (as shown by the solid curve) until it reaches a maximum at a certain value of the pressure ratio. The pressure ratio at which the mass flow is a maximum is called the *critical pressure ratio*. The critical pressure ratio $(p_2/p_1)_c$ can be determined by differentiating both sides of Eq. 19.38 with respect to (p_2/p_1) and equating the resulting expression to zero. Thus, we obtain

$$\frac{2}{k}\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}-1} - \left(\frac{k+1}{k}\right)\left(\frac{p_2}{p_1}\right)^{\frac{k+1}{k}-1} = 0$$

or

$$\left(\frac{p_2}{p_1}\right)_c = \left(\frac{2}{k+1}\right)^{\frac{k}{k-1}} \quad ...(19.39)$$

Equation 19.39 shows that the critical pressure ratio depends only upon the adiabatic exponent k. For air at normal conditions since $k = 1.40$, the critical pressure ratio has the value $(p_2/p_1)_c = 0.528$. By substituting the value of (p_2/p_1) from Eq. 19.39 in Eq. 19.38 the maximum mass rate of flow is found to be

$$G_{max} = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}} \quad ...(19.40)$$

It can be seen from Eq. 19.40 that the maximum mass flow rate of a compressible fluid through a convergent nozzle depends only upon the pressure p_1 and mass density ρ_1 at the approach section 1.

Again from Eq.19.38 it is seen that with further decrease in the throat pressure below the critical value [i.e., $p_2/p_1 < (p_2/p_1)_c$] the mass flow should decrease as shown by the dotted portion of the curved line in Fig. 19.2 (b). However, this decrease in the mass flow rate is only theoretically true, because actually for any given value of p_1, for pressure ratios lower than the critical pressure ratio, the mass flow rate does not decrease but it remains constant at the maximum value for all the values of (p_2/p_1) less than the critical pressure ratio. This may be explained as follows.

By introducing Eq. 19.39 in Eq. 19.37 the value of the Mach number at the throat corresponding to the critical pressure ratio is obtained as

$$[(Ma)_2]_c = \frac{V_2}{C_2} = \left[\frac{2}{k-1}\left(\frac{k+1}{2} - 1\right)\right]^{1/2} = 1$$

That is, at the critical pressure ratio the velocity V_2 at the throat is equal to the speed of the sound. As explained in Sec. 19.10 for an accelerating flow of a compressible fluid in a convergent nozzle the velocity of flow within the nozzle is subsonic with a maximum velocity equal to the sonic velocity at the throat. Thus when once the velocity V_2 at the throat has reached the speed of sound at the critical pressure ratio, it remains at the same value for all the values of (p_2/p_1) less than critical pressure ratio since the flow in the nozzle is being continuously accelerated with the reduction in the throat pressure below the critical values and hence the velocity cannot reduce. Therefore, for any given value of p_1, the mass flow rate for all values of (p_2/p_1) less than critical pressure ratio, remains constant at the maximum alue as indicated by the solid horizontal line in Fig. 19.3 (b). This fact has been verified experimentally.

Now if a properly designed divergent section is attached to the throat of the convergent section as hown in Fig. 19.4, then as discussed in Sec. 19.10, the flow of the compressible fluid can be accelerated o supersonic velocities in the divergent section when the throat pressure p_2 corresponds to the critical alue given by Eq. 19.39. The application of these principles is found in the *de Laval nozzle*, developed y the Swedish engineer Carl Gustaf Patrik de Laval (1845–1913) for accelerating steam to supersonic elocity in his high speed steam turbine. Similar nozzles are used nowadays in jet propulsion units. The nozzle consists of converging and diverging sections (Fig. 19.4) so proportioned that at the throat

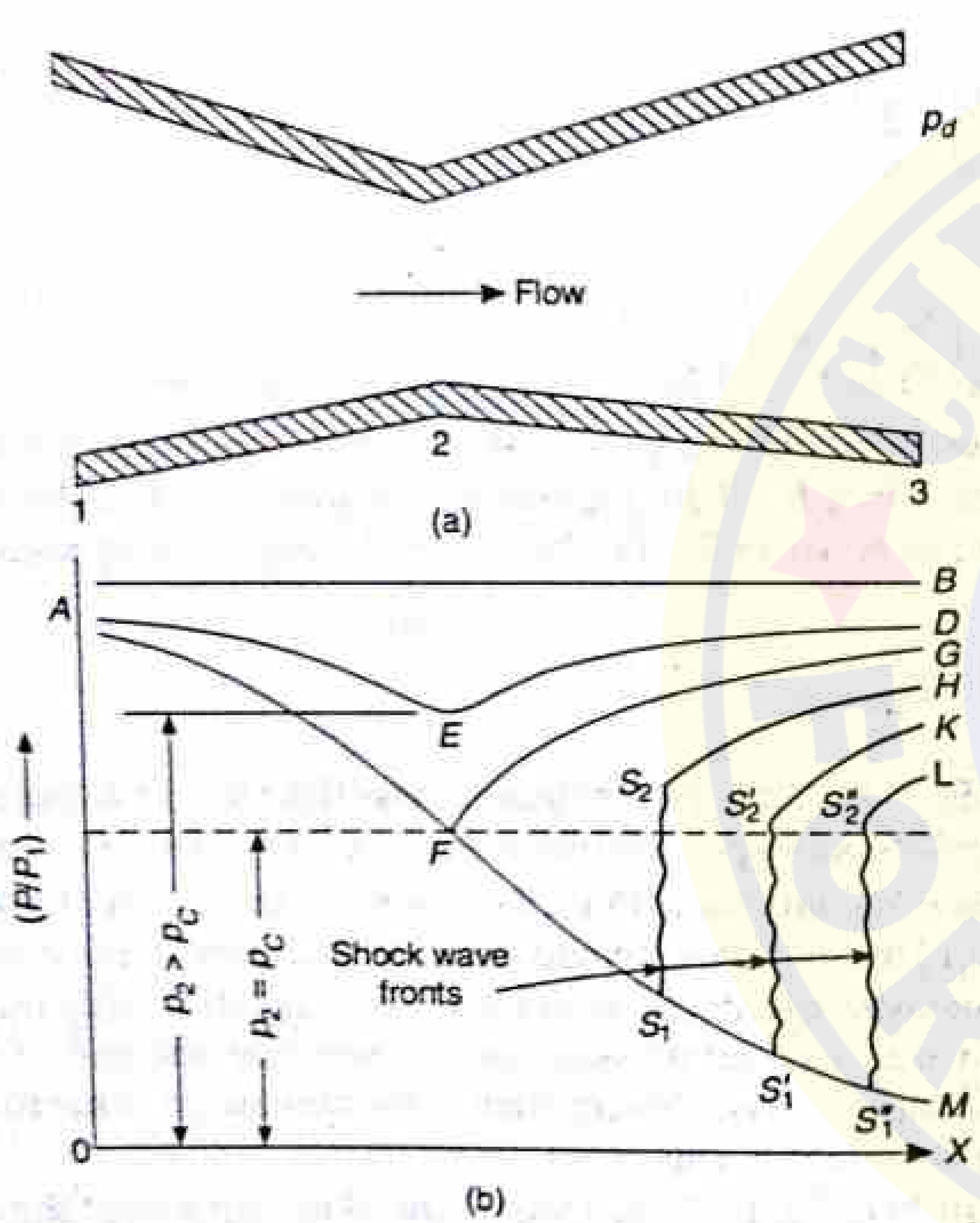


Figure 19.4 (a) *Convergent–divergent nozzle* (b) *Pressure distribution through a convergent– divergent nozzle with flow of compressible fluid*

section critical pressure and critical velocity (equal to sonic velocity) occurs. The various possible flow phenomena for a compressible fluid flowing in such a convergent-divergent passage may be analysed as follows.

Consider a compressible fluid flowing into a convergent–divergent nozzle from a large reservoir in which the approach velocity is negligible. The pressure in the reservoir may thus be assumed to be constant being equal to the pressure p_1 at the inlet section of the nozzle. It is, however, assumed that the pressure p_d at the downstream of the nozzle may be varied. Now if the pressure p_d is equal to the pressure p_1, then there can be no flow and the pressure throughout the nozzle, as well as downstream is same (i.e., $p/p_1 = 1.0$) as represented by the horizontal line AB in Fig. 19.4 (b). As the downstream

pressure p_d is gradually reduced, the pressure p_3 at section 3 of the nozzle is likewise reduced, but since the velocity has nowhere yet reached the sonic velocity, the influence of this reduction of pressure can move upstream, thereby causing the flow to take place with a greater drop in pressure at the throat section 2 of the nozzle. The pressure distribution as represented by the curve AED of Fig. 19.4 (b) is then obtained. This is seen to be similar in shape to that for the flow of a liquid in venturimeter. The mass rate of flow which depends on the values of pressure ratio (p_3/p_1) is given by Eq. 19.38 with A_3 and p_3 replacing A_2 and p_2 respectively. Thus

$$\frac{G}{A_3} = \left(\frac{2k}{k-1} p_1 \rho_1\right)^{\frac{1}{2}} \left\{\left(\frac{p_3}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_3}{p_1}\right)^{\frac{k+1}{k}}\right\}^{\frac{1}{2}} \quad ...(19.41)$$

When the downstream pressure p_d is further reduced to such a value as shown at G the pressure at the throat section 2 becomes equal to the critical pressure and the velocity of flow at the throat section is just equal to the sonic velocity. The pressure distribution is then represented by the curve AFG of Fig. 19.4 (b), which indicates that the flow is being accelerated in the convergent section upto the throat and it decelerates in the divergent section of the nozzle. This is due to the fact that immediately after the throat section the flow reverts to subsonic conditions in the divergent section which then acts as a diffuser. Under such conditions the nozzle is said to be *choked*. The mass rate of flow is then given by Eq. 19.40.

If the downstream pressure p_d is further reduced, pressure or shock waves moving in the upstream direction are developed in the divergent section, which affect the flow conditions there and hence new flow conditions are developed in the divergent section. But the flow conditions in the convergent part of the nozzle remain unchanged, the throat pressure remains at the critical value and the upstream pressure distribution follows the single curve AF.

Since the pressure or shock waves moving upstream in the divergent section encounter the sonic flow at the throat and hence these waves, which are also moving at the speed of sound, cannot penetrate into the flow upstream of the throat, thereby not affecting the flow in the convergent portion of the nozzle. However, the formation of the shock wave front as well as its location in the divergent portion of the nozzle depends on the magnitude of the downstream pressure p_d. Thus if the downstream pressure p_d corresponds to a value as shown at H such that $(p_d)H > p_c$ then the pressure distribution follows the curve AFS_1S_2H in which S_1S_2 represents a shock wave front formed in the divergent section. This formation of the shock wave front may be explained as follows.

The downstream pressure p_d being more than the critical pressure, subsonic flow conditions prevail in most of the downstream portion of the divergent section of the nozzle. However, the flow just downstream from the throat is accelerated to become slightly supersonic in the upstream portion of the divergent section, and the change of the approaching supersonic flow to the subsonic flow takes place through the formation of a shock wave front in the divergent section. Across the shock the velocity changes abruptly from supersonic to subsonic. There is also a finite pressure difference across the shock. The position of the shock wave front is fixed by the physical condition that the momentum difference across the shock is just balanced by the pressure difference between the supersonic flow on the upstream of the shock and the subsequent subsonic flow on the downstream of the shock. With further reduction in the downstream pressure p_d the shock wave front gets shifted and it will be established further downstream in the divergent section as indicated successively by the curves $AFS_1'S_2'K$ and $AFS_1''S_2''L$ of Fig. 19.4 (b). Now if the downstream pressure p_d is reduced to such a value as that at M, then the pressure distribution would follow the curve AFM. The flow thus accelerates steadily throughout the entire passage, being subsonic in the convergent section, sonic at the throat and supersonic in the subsequent divergent section. In this case the gas expands isentropically to critical pressure p_c at the throat and continues its isentropic expansion along the curve FM. As such in this case there is no shock wave front developed in the divergent section. The mass rate of flow of fluid may therefore be computed either by Eq. 19.40 or by Eq. 19.41. By equating the mass rate of flow given by Eqs. 19.40 and 19.41, the ratio of the throat area A_2, and the exit area A_3, for a convergent-divergent nozzle having isentropic supersonic flow without shock in the divergent section, may be obtained as

$$\frac{A_2}{A_3} = \left(\frac{k+1}{2}\right)^{\frac{k+1}{2(k-1)}}\left(\frac{2}{k-1}\right)^{1/2}\left[\left(\frac{p_3}{p_1}\right)^{\frac{1}{k}}-\left(\frac{p_3}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2} \quad ...(19.42)$$

It may be seen from Eq. 19.42 that for a convergent-divergent nozzle required to develop an isentropic supersonic flow without shock in the divergent section, the ratio of the throat area A_2 and the exit area A_3 depends on the pressure ratio (p_3/p_1). For a convergent-divergent nozzle designed to produce a supersonic flow such that smooth expansion (along the curve AFM) takes place, the pressure ratio $p_1/(p_d)_M$ is known as the *design pressure ratio* of the nozzle.

A further reduction of the downstream pressure p_d below $(p_d)_M$ (the value corresponding to point M) will however have no effect on the flow in the nozzle. As such the flow conditions throughout the nozzle will remain exactly the same as those corresponding to the downstream pressure equal to $(p_d)_M$.

19.12 NORMAL SHOCK WAVES

It has been indicated in the foregoing section that shock waves may occur in the diverging section of a convergent-divergent nozzle when the approaching supersonic flow is converted to a subsonic flow. Such shock waves may also occur in pipes as well as in front of a blunt-nosed body, when an approaching supersonic flow is converted to subsonic flow. The flow on upstream of the shock is supersonic having a relatively lower pressure (being less than the critical pressure p_c), while on the downstream of the shock it is subsonic having a higher pressure (being more than the critical pressure p_c). It therefore means that the conversion of supersonic flow to a subsonic flow is always accompanied by a sudden pressure rise as the flow passes the shock. Although in practice a shock is not absolutely abrupt, but the distance over which it occurs being very small, for most purposes it may be supposed to be abrupt. Accordingly across a shock wave abrupt and discontinuous (i.e., sudden and finite) changes of flow properties such as pressure, density, velocity etc., may be supposed to occur. The process of the formation of the shock waves in a compressible fluid is analogous to the formation of 'hydraulic jump' in open channels. In case of hydraulic jump the flow changes from supercritical to subcritical and is accompanied by an increase in depth and decrease in the velocity of flow. In shock waves the critical velocity being the velocity of sound (or sonic velocity), the flow changes from supersonic to subsonic and it is accompanied by an increase in the pressure and density and a decrease in the velocity.

The shock waves can be either normal or oblique with respect to the direction of flow. However, only the normal shock waves have been discussed here. As indicated below the various elements of a normal shock wave can be determined by the application of continuity, momentum and energy equations.

Consider a stationary normal shock wave developed in a pipe as shown in Fig. 19.5. Let ρ_u, V_u and ρ_d, V_d represent the mass density and velocity upstream and downstream of the shock.

Thus according to continuity equation,

$$\rho_u V_u = \rho_d V_d = \frac{G}{A} \quad ...(19.43)$$

where G/A is the mass rate of flow per unit area.

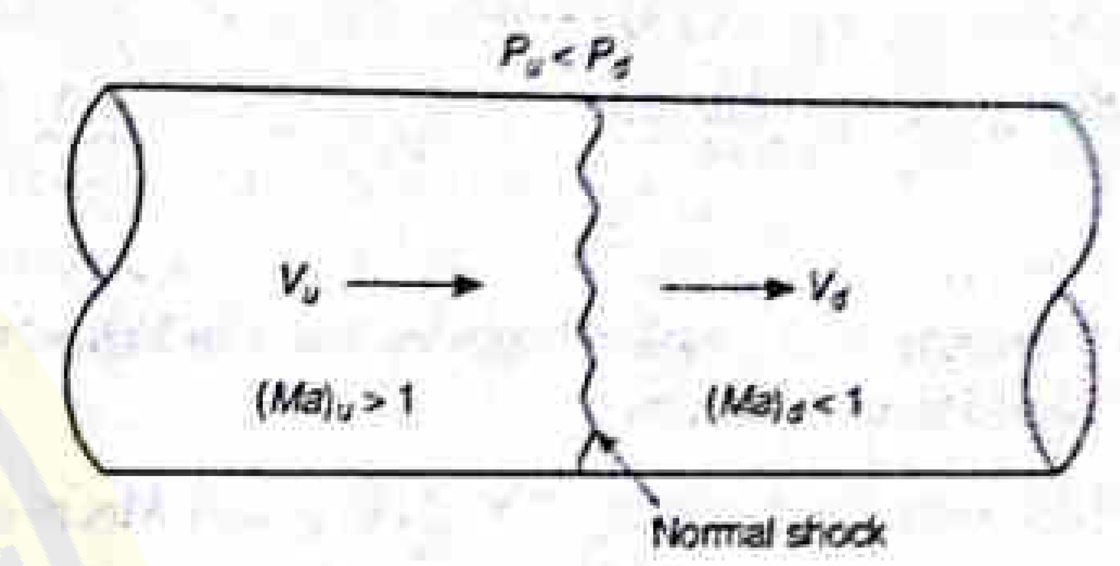


Figure 19.5 *Stationary normal shock wave in a pipe*

Further if p_u and p_d are the pressure intensities upstream and downstream of the shock and if the effects of boundary friction are negligible then applying the momentum equation across the shock, we have

$$(p_u - p_d)A = (\text{Mass flow})(V_d - V_u)$$

i.e.

or

$$\left.\begin{aligned} p_u - p_d &= \rho_d V_d^2 - \rho_u V_u^2 \\ p_u + \rho_u V_u^2 &= p_d + \rho_d V_d^2 \end{aligned}\right\} \quad ...(19.44)$$

If it is assumed that there is no net heat transfer to or from the pipe, then from Eq. 19.12 the energy equation for the flow across the shock may be obtained as

$$\frac{k}{k-1}\left(\frac{p_d}{\rho_d}-\frac{p_u}{\rho_u}\right) = \frac{V_u^2}{2}-\frac{V_d^2}{2} \quad ...(19.45)$$

By combining continuity Eq. 19.43 and momentum Eq. 19.44, we get

$$p_u + \frac{(G/A)^2}{\rho_u} = p_d + \frac{(G/A)^2}{\rho_d} \quad ...(19.46)$$

Equation 19.46 is the *equation of the Rayleigh line*.

Similarly by combining the continuity Eq. 19.43 and the energy Eq. 19.45, we get

$$\frac{(G/A)^2}{2\rho_u^2}+\frac{k}{k-1}\frac{p_u}{\rho_u} = \frac{(G/A)^2}{2\rho_d^2}+\frac{k}{k-1}\frac{p_d}{\rho_d} \quad ...(19.47)$$

Equation 19.47 is the *equation of the Fanno line*.

Further elimination of (G/A) from Eqs. 19.46 and 19.47 results in the following relationship between the pressure and densities of flow on both sides of a normal shock

$$\frac{p_d}{p_u} = \frac{\left(\frac{k+1}{k-1}\right)\left(\frac{\rho_d}{\rho_u}\right)-1}{\left(\frac{k+1}{k-1}\right)-\frac{\rho_d}{\rho_u}} \quad ...(19.48)$$

or

$$\frac{\rho_d}{\rho_u} = \frac{V_u}{V_d} = \frac{1+\left(\frac{k+1}{k-1}\right)\frac{p_d}{p_u}}{\left(\frac{k+1}{k-1}\right)+\frac{p_d}{p_u}} \quad ...(19.49)$$

Equations 19.48 and 19.49 are known as *Rankine–Hugoniot Equations* in honour of W. Rankine and H. Hugoniot who first developed these equations.

From Eqs 19.23 and 19.24 since sonic velocity $C = \sqrt{(pk/\rho)}$ and Mach number $Ma = (V/C)$, Eqs. 19.48 and 19.49 can be written in alternative form as

$$\frac{p_d}{p_u} = \frac{2k(Ma)_u^2-(k-1)}{(k+1)} \quad ...(19.50)$$

and

$$\frac{\rho_d}{\rho_u} = \frac{V_u}{V_d} = \frac{(k+1)(Ma)_u^2}{(k-1)(Ma)_u^2+2} \quad ...(19.51)$$

where $(Ma)_u$ is the Mach number just upstream of the shock wave.

Further by combining Eqs 19.50 and 19.51 the following expression for $(Ma)_d^2$, may also be obtained

$$(Ma)_d^2 = \frac{(k-1)(Ma)_u^2+2}{2k(Ma)_u^2-(k-1)} \quad ...(19.52)$$

It may however be noted that as the flow crosses a shock the energy of flow is conserved, but there is an increase in entropy in the flow process. Therefore the flow crossing a shock is an irreversible process.

19.13 MEASUREMENT OF COMPRESSIBLE FLUID FLOW

(a) Measurement of Discharge. For measuring the mass rate of flow of compressible fluids in pipe lines, venturimeter, orifice meter and nozzle meter are commonly used. In each of these meters the flow conditions resemble those in a convergent-divergent nozzle. Thus as the fluid passes through the throat or minimum section of the flow meter the fluid pressure is partly converted to kinetic energy, thereby developing a pressure difference between the inlet and the throat sections of the flow meter, which can be measured. The distance between inlet and throat sections is usually small so that friction and heat transfer between these sections can be neglected. Therefore as indicated below the expression for the mass rate of flow G through any of these flow meters can be obtained by applying the continuity equation, isentropic or polytropic process equation for pressure variation and the general energy equation for compressible fluid flow.

From the continuity equation for one dimensional flow

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2$$

Assuming the flow process in the flow meter to be polytropic, the relationship between the pressure and density may be expressed as

$$\frac{p_1}{\rho_1^n} = \frac{p_2}{\rho_2^n}$$

For a compressible fluid flow the general energy equation may be expressed as

$$\frac{1}{2}V_1^2 + \frac{n}{n-1}\frac{p_1}{\rho_1} = \frac{1}{2}V_2^2 + \frac{n}{n-1}\frac{p_2}{\rho_2}$$

However, for insentropic flow $n = k$ in the above expressions. By combining the above noted three expressions following expression for the rate of flow G may be readily obtained

$$G = CA_2\rho_2\left[\frac{\frac{2n}{n-1}\frac{p_1}{\rho_1}\left\{1-\left(\frac{p_2}{p_1}\right)^{\frac{n-1}{n}}\right\}}{1-\left(\frac{A_2}{A_1}\right)^2\left(\frac{p_2}{p_1}\right)^{2/n}}\right]^{\frac{1}{2}} \quad ...(19.53)$$

where C is the meter coefficient that takes care of the overall friction losses in the flow meter. Tests have shown that the numerical values of C for these meters are approximately the same as those of coefficient of discharge C_d (or K) for incompressible fluids flowing through them.

(b) Measurement of Velocity (i) *Pitot Tube*. For determining the velocity of flow of a compressible fluid *pitot tube* (as described in Chapter 7) is a commonly used device. When pitot tube is inserted in a stream of compressible fluid, its tip becomes a stagnation point since the velocity of flow there is reduced to zero and stagnation pressure is developed. By measuring the stagnation pressure p_s and the static pressure p_1 in the approaching stream, the velocity of flow can be determined as indicated below. The mathematical relationship between the stagnation pressure and the velocity of compressible flow depends on whether the flow is subsonic or supersonic.

If the flow is 'subsonic' it can be assumed to be isentropic and from Eq. 19.25, the ratio of the stagnation pressure p_s to the pressure p_1 in the approaching stream may be obtained as

$$\frac{p_s}{p_1} = \left[1+\frac{k-1}{2}\left(\frac{V_1^2}{kp_1/\rho_1}\right)\right]^{\frac{k}{k-1}}$$

or

$$\frac{p_s}{p_1} = \left[1+\frac{k-1}{2}(Ma_1)^2\right]^{\frac{k}{k-1}} \quad ...(19.54)$$

For small values of $(Ma_1)^2$, the right hand side of Eq. 19.54 may be expanded into a convergent series by means of the binomial theorem. Thus

$$\frac{p_s}{p_1} = 1+\frac{1}{2}k(Ma_1)^2\left[1+\frac{1}{4}(Ma_1)^2+\frac{2k}{24}(Ma_1)^4+\ldots\right]$$

The terms within the bracket in the above expression are usually referred to as the *compressibility factor* because the pressure ratio (p_s/p_1) for a flow of an incompressible fluid is

$$\frac{p_s}{p_1} = 1+\frac{1}{2}\frac{\rho_1 V_1^2}{p_1} = 1+\frac{1}{2}\;k\;\frac{V_1^2}{kp_1/\rho_1} = 1+\frac{1}{2}\;k\,(Ma_1)^2$$

When a pitot tube is introduced into a 'supersonic' stream, a shock will occur upstream of pitot tube. If p_2 is the pressure on the downstream of the shock then across the shock the pressure ratio (p_2/p_1) is given by Eq. 19.50 as

$$\frac{p_2}{p_1} = \frac{2k(Ma_1)^2-(k-1)}{(k+1)}$$

The Mach number Ma_2 of the subsonic flow downstream of the shock is given by Eq. 19.52 as

$$Ma_2 = \left[\frac{(k-1)(Ma_1)^2+2}{2k(Ma_1)^2-(k-1)}\right]^{\frac{1}{2}}$$

Further the pressure ratio (p_s/p_2) may be expressed by Eq. 19.54 as

$$\frac{p_s}{p_2} = \left[1+\frac{k-1}{2}(Ma_2)^2\right]^{\frac{k}{k-1}}$$

By combining the above expressions the following expression for (p_s/p_1) may be readily obtained

$$\frac{p_s}{p_1} = \frac{\dfrac{p_s}{p_2}}{\dfrac{p_1}{p_2}} = \left[\frac{\left\{\dfrac{k+1}{2}(Ma_1)^2\right\}^k}{\dfrac{2k(Ma_1)^2-(k-1)}{k+1}}\right]^{\frac{1}{k-1}} \qquad \ldots(19.55)$$

Equation 19.55 is the wellknown *Rayleigh–Pitot equation* for stagnation pressure in a supersonic stream with a shock wave front.

(ii) Hot-wire Anemometer. A *hot-wire anemometer* is another instrument which is commonly used for measuring the velocity of flow of a compressible fluid such as air or gas. The anemometer consists of a platinum, nickel or tungsten wire of about 5×10^{-3} to 8×10^{-3} mm diameter and 6 mm length. The wire is mounted on the ends of two pointed prongs and it is introduced in the flow field so as to be perpendicular to the direction of flow. A small electric current is passed through the wire, in order to heat it above the temperature of the surrounding air or gas. As the air or gas flows round the wire, the wire is cooled. The amount of heat lost from the heated wire to the air or gas flow is a function of the flow velocity past the wire. The amount of heat transfer from the wire to the surrounding air or gas results in altering the temperature of the wire which in turn results in the variation of the electrical resistance since the electrical resistance of the wire is a function of its temperature. Thus a relationship between the heat transfer or the resistance of the wire and the flow velocity can be obtained by calibration.

Figure 19.6 illustrates two circuits for a hot-wire anemometer either of which may be used for measuring the velocity of flow. R_1, R_2 and R_3 are electric resistances in the Wheatstone–bridge circuit and the hot-wire forms the remaining resistance in the bridge. *B* represents a variable electric resistance, and *A* represents a battery. In the arrangement shown in Fig. 19.6 (a) constant current is passed through the wire by keeping the voltage across the bridge (i.e., across points *CD*) constant after the circuit is adjusted so that the galvanometer indicates zero current when the heated wire is in air or gas at rest. As the air or gas flows the hotwire cools, its resistance changes and the galvanometer deflects. The galvanometer deflection is correlated with the velocity of flow of air or gas by calibration. It is then termed as *constant current hot-wire anemometer*. Because of its high sensitivity the constant-current hot-wire anemometer is especially suitable for flow in which velocity fluctuations are small.

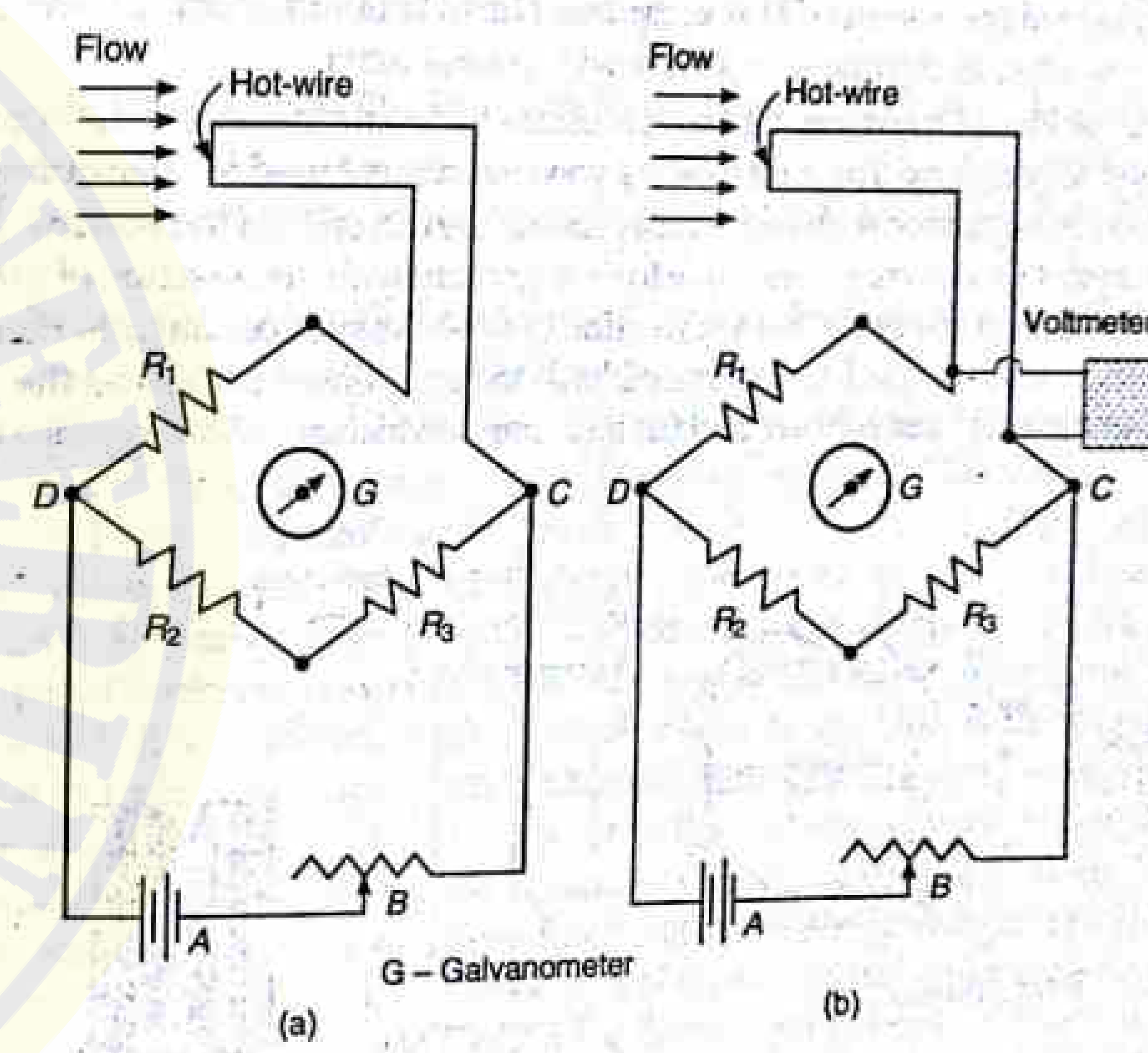


Figure 19.6 (a) *Constant-current hot-wire anemometer;* (b) *Constant-temperature hot-wire anemometer*

Figure 19.6 (b) illustrates another arrangement for hot wire anemometer which is termed as *constant temperature (or constant-resistance) hot-wire anemometer*. In this case the temperature of the hot-wire is held constant and hence its resistance remains constant. Initially when there is no flow and the hot-wire is in contact with air or gas at rest, a small current is passed through the hot-wire and the circuit is adjusted so that the galvanometer reads zero. As the air or gas flows past the hot-wire, its temperature and hence its resistance will vary, which will cause the galvanometer needle to deflect from zero reading. Now by adjusting the variable resistance *B* the current passing through the hot-wire is suitably adjusted so that its temperature and hence the resistance is maintained constant and the galvanometer reading is brought back to zero. The reading of the voltmeter connected across the hot-wire will then change which may be noted. By calibration in a stream of known velocity, the voltmeter reading is correlated with the velocity of flow of air or gas. The constant temperature hot-wire anemometer may be conveniently used for velocity measurement even when the velocity fluctuations are large.

A hot-wire anemometer requires frequent calibration against a known velocity. The calibration should be carried out in the same fluid as that whose velocity is to be measured, because heat transfer depends on the fluid density also. Moreover it is a delicate device being subjected to breakage. However, despite these limitations it has widespread use in air or gas flow studies. Since it responds very rapidly to fluctuations of velocity it is widely used in conjunction with oscilloscopes and similar electronic instruments for measuring the intensity of turbulence.

A hot-wire anemometer is mainly used for measuring the velocity of flow of air or gas. It has not been possible to use it successfully with liquids because bubbles and some small solid particles present in the liquid tend to collect on the wire and spoil the calibration.

Recently a *hot film anemometer* has been developed, in which a thin platinum film fused to a glass support is used in place of the hot-wire. This instrument has a better mechanical strength but slightly inferior frequency response as compared with hot-wire anemometer.

(c) Measurement of Flow Direction. Often it is desired to determine both the magnitude as well as the direction of the velocity of flow. The pitot tube is a good instrument used for measuring the magnitude of the velocity of flow, but it cannot be used for determining the direction of the velocity. Moreover it has been observed that a pitot tube is very insensitive to its alignment with the direction of flow, as it has only about 1 per cent error at an angle of 20° to the direction of flow. As such certain other instruments have been developed which may be used to determine both the magnitude as well as the direction of the velocity of flow. These are (*i*) *Pitot cylinder* and (*ii*) *Pitot sphere* which are shown in Fig. 19.7.

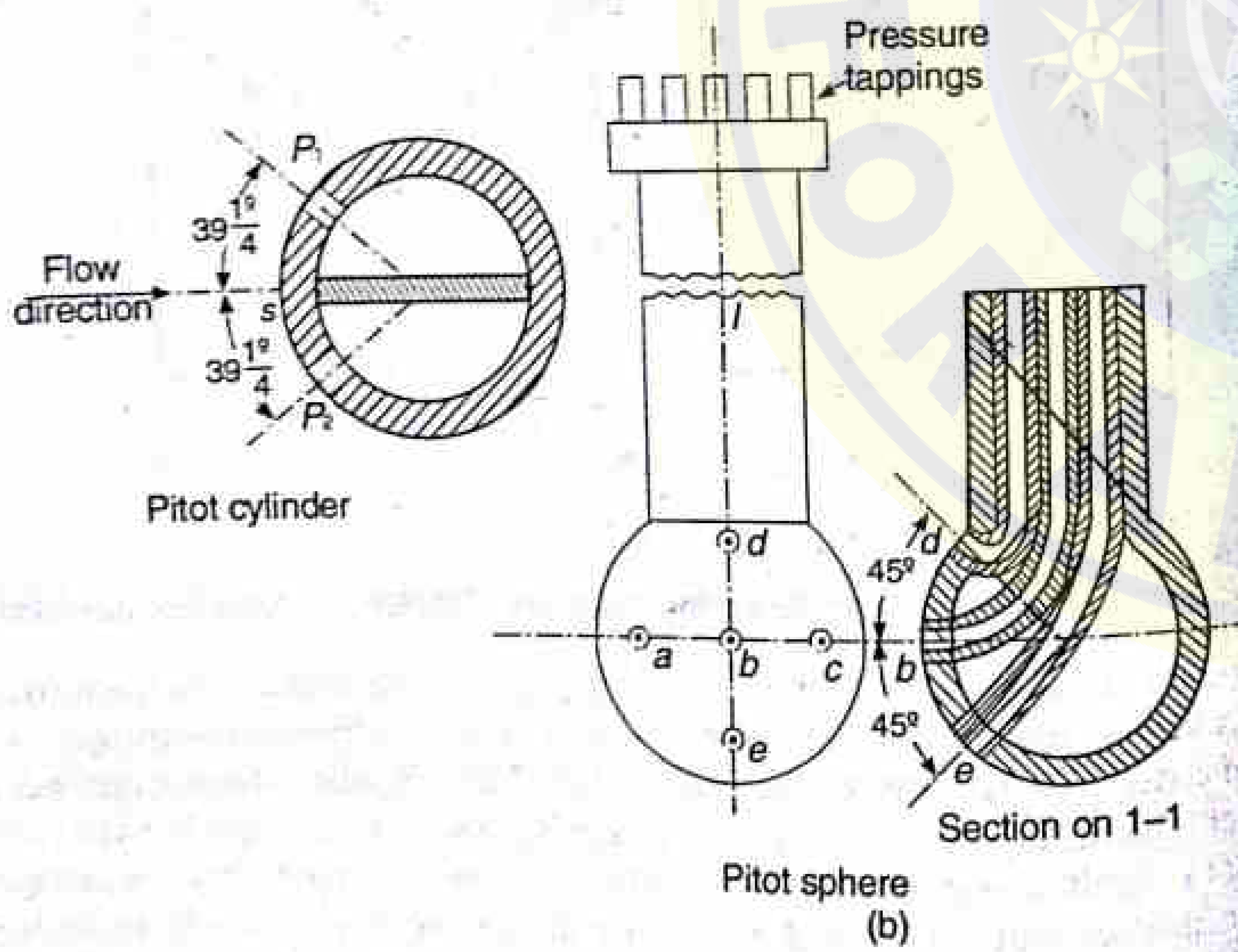


Figure 19.7 *(a) Pitot cylinder; (b) Pitot sphere*

A *Pitot cylinder* consists of a cylindrical tube, with two small pressure taps P_1 and P_2. The axis of the cylinder is perpendicular to the direction of flow. Each pressure tap is connected to one side of differential pressure gage. For determining the direction of flow the cylinder is rotated about its axis in the stream, until the pressures at P_1 and P_2 are equal as indicated by a null reading on the differential

pressure gage. In this null position the stagnation point S is mid-way between P_1 and P_2 and therefore the bisector of the angle P_1OP_2 gives the direction of flow. If the pitot cylinder is to be used for the determination of the direction of flow only, then the angle P_1OP_2 may have any value. But if it is used for determining the magnitude of the velocity also, then for an ideal fluid each of the pressure taps P_1 and P_2 should be at an angle of 30° from S, so that in null position pressure recorded at P_1 or P_2 would be the static pressure of the undisturbed stream. However, for the fluids in actual practice this angle between P_1 or P_2 and S is to be about $39\frac{1}{4}°$. Thus if the angle P_1OP_2 is $78\frac{1}{2}°$, then in the null position the pressure reading from the tap P_1 or P_2 gives the static pressure. Now if the cylinder is rotated about its axis through $39\frac{1}{4}°$, then one of the pressure taps will be in line with the direction of flow. In this position the pressure reading at this tap gives the stagnation pressure. The difference between the stagnation and the static pressure heads is equal to the velocity head $\frac{V^2}{2g}$ (Sec. 7.14) from which the velocity of flow V can be computed. A Pitot cylinder is suitable for determination of both magnitude and direction of velocity in a two dimensional flow.

A *Pitot sphere* is another instrument which may be used to determine the magnitude and direction of velocity of flow in a three-dimensional flow. As shown in Fig. 19.7 (b) it consists of a small hollow sphere 8 to 15 mm in diameter having five pressure taps *a*, *b*, *c*, *d* and *e*. The three taps *a*, *b* and *c* are located on the horizontal great circle, each being inclined at 50° to the other. The taps *d*, *b* and *e* are located on the vertical great circle passing through tap *b* and each of these taps is inclined at 45° to the other. All the taps are connected to manometers. The sphere is fixed on the end of a hollow stem which can be rotated about its axis and the angle of rotation may be read off on fixed horizontal graduated disc by means of an indicating pointer attached to the stem. The stem can also be rotated in a vertical plane and the angle of tilt may be read off on a vertical graduated arc attached to the stem.

For measuring the velocity of flow the sphere is introduced in the flowing fluid with its stem perpendicular to the direction of the longitudinal axis of the flow passage. It is then so adjusted that the plane containing the taps *d*, *b* and *e*; and the zero of the horizontal graduated disc are parallel to the longitudinal axis of the flow passage. The stem is then rotated about its axis until the manometers connected to the taps *a* and *c* indicate the same reading. In this position, the velocity vector lies in the vertical plane passing through the taps *d*, *b* and *e*. The angle of rotation is obtained from the horizontal graduated disc. The stem is then tilted by rotating it in the vertical plane, until the manometers connected to the taps *d* and *e* indicate the same reading. In this position the velocity vector would be along the radius of the sphere passing through the central tap *b*. The angle of tilt is obtained from the vertical graduated arc. Thus the direction of the velocity vector is known. In order to determine the magnitude of the velocity, the pressure reading at the tap *b* is recorded when the instrument is in tilted position, which gives the stagnation pressure. By subtracting the static pressure head from the stagnation pressure head the velocity head can be known from which the velocity of flow can be computed.

Another type of Pitot sphere which is useful for the determination of the direction of flow only, has also been developed in which only three pressure taps are provided on the sphere which are situated at the vertices of an equilateral triangle. It is held in the flowing stream and so adjusted that the pressure at all the three taps are equal. The velocity of flow is then parallel to the radius of the sphere passing through the centroid of the triangle.

Both the Pitot cylinder and the Pitot sphere should be properly calibrated in a stream of known velocity prior to these being used for the measurement of the velocity of flow.

ILLUSTRATIVE EXAMPLES

Example 19.1. *A test plane is described as having attained a flight speed of Ma = 2 at an attitude of 16 km where the temperature is approximately –56.5°C (or 216.65 K). Assuming k = 1.4 and R = 287 J/kg.K, determine the speed of the air plane.*

Solution

From Eq. 19.23 sonic velocity is given by

$$C = \sqrt{kRT}$$

$$= \sqrt{1.4 \times 287 \times 216.65}$$

$$= 295 \text{ m/s}$$

The speed of the test plane is

$$V = 2 \times 295 \times \frac{3600}{1000} = 2124 \text{ km/h}$$

Example 19.2. *To what change in pressure a liquid of specific gravity 1.2 should be subjected to cause reduction in volume by one percent? The velocity of sound in the liquid is 1500 m/s.*

Solution

From Eq. 19.16, we have

$$C^2 = \frac{dp}{d\rho} \quad ...(i)$$

and from Eq. 19.17, we have

$$\frac{dv}{v} = -\frac{d\rho}{\rho} \quad ...(ii)$$

Multiplying Eqs (i) and (ii), the change in pressure dp is given by

$$dp = -\rho C^2 \frac{dv}{v}$$

$$\rho = 1.2 \times 1000 = 1200 \text{ kg/m}^3;\ C = 1500 \text{ m/s; and } \frac{dv}{V} = -0.99$$

Thus by substituting, we get

$$dp = \left[-1200 \times (1500)^2 \times (-0.99)\right] \text{N/m}^2$$

$$= 2.673 \times 10^9 \text{ N/m}^2 = 2.673 \times 10^6 \text{ kN/m}^2$$

Example 19.3. *Air flows isentropically around a submerged object. At section 1 in the approaching stream the pressure p_1 = 101.043 kN/m², the density ρ_1 = 1.226 kg/m³ and the velocity V_1 = 135 m/s. At point 2 near the object, the pressure p_2 is observed to be 39.24 kN/m². Calculate (a) the temperature ratio between these two points and the Mach number at each point. Take k = 1.4 and R = 287 J/kg. K.*

Solution

By combining Eq. 19.2 and the equation of state the following expression can be readily obtained for such a flow

$$\frac{T_1}{T_2} = \left(\frac{p_1}{p_2}\right)^{\frac{k-1}{k}} = \left(\frac{101.043}{39.24}\right)^{\frac{1.4-1}{1.4}} = 1.31$$

Since $\quad p_1 = \rho_1 RT_1$

$$T_1 = \frac{101.043 \times 10^3}{1.226 \times 287} = 287.17 \text{ K}$$

$$\therefore \quad T_2 = \frac{287.17}{1.31} = 219.21 \text{ K}$$

Also $\quad \left(\frac{\rho_1}{\rho_2}\right) = \left(\frac{p_1}{p_2}\right)^{\frac{1}{k}} = \left(\frac{101.043}{39.24}\right)^{\frac{1}{1.4}} = 1.965$

$$\therefore \quad \rho_2 = \frac{1.226}{1.965} = 0.624 \text{ kg/m}^3$$

Applying Eq. 19.12 between the sections 1 and 2 with $z_1 = z_2$ and substituting the known values, we get

$$\frac{V_2^2 - (135)^2}{2 \times 9.81} = \frac{1.4}{1.4-1}\left(\frac{101.043}{1.226} - \frac{39.24}{0.624}\right)\frac{10^3}{9.81}$$

$$\therefore \quad V_2 = 393.64 \text{ m/s}$$

$$C_1 = \sqrt{kRT_1}$$

$$= \sqrt{1.4 \times 287 \times 287.17} = 339.68 \text{ m/s}$$

$$\therefore \quad Ma_1 = \frac{V_1}{C_1} = \frac{135}{339.68} = 0.397$$

$$C_2 = \sqrt{kRT_2}$$

$$= \sqrt{1.4 \times 287 \times 219.21} = 296.78 \text{ m/s}$$

$$\therefore \quad Ma_2 = V_2 C_2 = \frac{393.64}{296.78} = 1.326$$

Example 19.3. *Air discharges from a large tank, in which the pressure is 700 kN/m² and temperature 40°C, through a convergent nozzle of 25 mm tip diameter. Calculate the flow rates when the pressure outside the jet is (a) 200 kN/m², and (b) 550 kN/m² and the barometric pressure is 101 kN/m². Also calculate the pressure, temperature, velocity and sonic velocity at the nozzle tip for these flow rates. Take k = 1.4 and R = 286.8 J/kg.K.*

Solution

From the equation of state the mass density ρ_1 for the air in the tank is obtained as

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{(700+101)\times 10^3}{286.8\times(273.15+40)} = 8.919 \text{ kg/m}^3$$

From Eq. 19.39, critical pressure ratio is

$$\left(\frac{p_2}{p_1}\right)_c = \left(\frac{2}{2.4}\right)^{3.5} = 0.528$$

(a)
$$\left(\frac{p_2}{p_1}\right) = \frac{301}{801} = 0.376 < 0.528$$

Therefore in this case the flow rate is given by Eq. 19.40 as

$$G = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

$$= \frac{\pi}{4}\times(0.025)^2\times(1.4\times 801\times 103\times 8.919)^{\frac{1}{2}}\left(\frac{2}{2.4}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 0.898 \text{ kg/s}$$

As explained in Sec. 19.11 for this case

$$\frac{p_2}{p_1} = 0.528$$

$$\therefore \quad p_2 = (0.528\times 801) = 422.93 \text{ kN/m}^2 \text{ (abs)}$$

From Eq. 19.36, we have

$$\frac{V_2^2}{2} = \frac{p_1}{\rho_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]$$

By substituting, we get

$$\frac{V_2^2}{2} = \frac{801\times 10^3}{8.919}\times\frac{1.4}{0.4}\left[1-(0.528)^{\frac{0.4}{1.4}}\right]$$

$$\therefore \quad V_2 = 323.82 \text{ m/s}$$

which is also the sonic velocity at the nozzle tip in this case.

From Eq. 19.2 and the equation of state

$$T_2 = T_1\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}$$

$$= 313.15(0.528)^{\frac{0.4}{1.4}} = 261 \text{ K}$$

(b)
$$\left(\frac{p_2}{p_1}\right) = \frac{651}{801} = 0.813 > 0.528$$

Therefore in this case the flow rate is given by Eq. 19.38 as

$$G = A_2\left(\frac{2k}{k-1}p_1\rho_1\right)^{1/2}\left[\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}}-\left(\frac{p_2}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2}$$

By substituting, we get

$$G = \frac{\pi}{4}\times(0.025)^2\left(\frac{2\times 1.4}{1.4-1}\times 801\times 10^3\times 8.919\right)^{1/2}\times\left[(0.813)^{\frac{2}{1.4}}-(0.813)^{\frac{1.4+1}{1.4}}\right]^{1/2}$$

$$= 0.718 \text{ kg/s}$$

In this case since

$$\frac{p_2}{p_1} = 0.813$$

$$\therefore \quad p_2 = (0.813\times 801) = 651.21 \text{ kN/m}^2 \text{ (abs)}$$

From Eq. 19.36, we have

$$\frac{V_2^2}{2} = \frac{p_1}{\rho_1}\frac{k}{k-1}\left[1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]$$

By substitution, we get

$$\frac{V_2^2}{2} = \frac{801\times 10^3}{8.919}\times\frac{1.4}{0.4}\left[1-(0.813)^{\frac{0.4}{1.4}}\right]$$

$$\therefore \quad V_2 = 190 \text{ m/s}$$

From Eq. 19.37, we have

$$\frac{V_2}{C_2} = \left[\frac{2}{k-1}\left\{\left(\frac{p_1}{p_2}\right)^{\frac{k-1}{k}} - 1\right\}\right]^{1/2}$$

By substitution, we get

$$\frac{190}{C_2} = \left[\frac{2}{1.4-1}\left\{\left(\frac{1}{0.813}\right)^{\frac{0.4}{1.4}} - 1\right\}\right]^{1/2}$$

∴ Sonic velocity at the nozzle tip is

$$C_2 = 344.22 \text{ m/s}$$

From Eq. 19.2 and the equation of state

$$T_2 = T_1\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}$$

$$= 313.15(0.813)^{\frac{0.4}{1.4}} = 295.16 \text{ K}$$

Example 19.4. *A tank contains air at a gage pressure of 34 kN/m² and a temperature of 32.2°C. The local barometric pressure is 101 kN/m². Air discharges out of the tank into atmosphere through a convergent nozzle. The exit opening of the nozzle is 25 mm in diameter. The coefficient of discharge C_d for the nozzle may be taken as unity. Assuming insentropic flow, what is the mass rate of air flow through the nozzle. Take k = 1.4 and R = 287 J/kg.K.*

Solution

From the equation of state the mass density ρ_1 for the air in the tank is obtained as

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{(101+34)\times 10^3}{287\times(273.15+32.2)} = 1.54 \text{ kg/m}^3$$

The pressure ratio

$$\frac{p_2}{p_1} = \frac{101}{135} = 0.748$$

which is greater than the critical pressure ratio, and hence the mass rate of flow of air is given by Eq. 19.38 as

$$G = A_2\left(\frac{2k}{k-1}p_1\rho_1\right)^{1/2}\left[\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_2}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2}$$

By substituting, we get

$$G = \frac{\pi}{4}(0.025)^2\left(\frac{2\times1.4}{1.4-1}\times135\times10^3\times1.54\right)^{1/2} \times\left[\left(\frac{101}{135}\right)^{\frac{2}{1.4}} - \left(\frac{101}{135}\right)^{\frac{1.4+1}{1.4}}\right]^{1/2}$$

$$= 0.136 \text{ kg/s.}$$

Example 19.5. *What will be the mass rate of flow if the pressure in the tank of Example 19.4 is 343.5 kN/m²? Other data remain the same.*

Solution

The density of air corresponding to a pressure of 343.5 kN/m² and a temperature of 32.2°C is

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{(101+343.5)\times10^3}{287\times(273.15+32.2)} = 5.072 \text{ kg/m}^3$$

The pressure ratio

$$\frac{p_2}{p_1} = \frac{101}{444.5} = 0.227$$

which is less than the critical pressure ratio. Therefore the velocity of air flow at the nozzle exit is sonic and the mass rate of flow is given by Eq. 19.40 as

$$G = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

By substitution, we get

$$G = \frac{\pi}{4}(0.025)^2 \times (1.4\times444.5\times10^3\times5.072)^{1/2}\times\left(\frac{2}{1.4+1}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 0.505 \text{ kg/m}^3$$

Example 19.6. *The exit area to throat area ratio for a convergent-divergent nozzle is 2.5, the throat being 12.5 × 10⁻⁴ m². Air enters this nozzle from a reservoir where the absolute pressure is 686.7 kN/m² and the temperature is 24°C. Air discharges from the exit area into the atmosphere where the local barometric pressure is 101 kN/m². Assume the flow to be isentropic with R = 287 J/kg.K and k = 1.4. Calculate (a) the mass rate of air flow through the nozzle, (b) the exit pressure, (c) the exit temperature, (d) the exit Mach number, and (e) the exit velocity for the following two conditions:*

(i) Sonic velocity at the throat, with divergent section acting as a nozzle;

(ii) Sonic velocity at the throat, with divergent section acting as a diffuser.

Solution

Applying energy equation between the throat (2) and the exit section (3) of the nozzle, we have

$$\frac{p_2}{\rho_2}\frac{k}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = \frac{V_3^2}{2}-\frac{V_2^2}{2}$$

or $$\frac{p_2}{\rho_2}\frac{2}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = R\left[\frac{V_3^2T_3}{kRT_3}-\frac{V_2^2T_2}{kRT_2}\right]$$

or $$\frac{p_2}{\rho_2}\frac{2}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = \frac{p_3}{\rho_3}\left(Ma^2\right)_3-\frac{p_2}{\rho_2}\left(Ma^2\right)_2$$

since $$p = \rho RT \text{ and } \frac{V^2}{kRT} = Ma^2$$

or $$\frac{2}{k-1}\left[1-\left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\right] = \left(\frac{p_3}{p_2}\right)^{\frac{k-1}{k}}\left(Ma^2\right)_3-\left(Ma^2\right)_2$$

since $$\left(\frac{\rho_2}{\rho_3}\right) = \left(\frac{p_2}{p_3}\right)^{\frac{1}{k}}$$

Solving for (p_3/p_2), we get

$$\frac{p_3}{p_2} = \left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_3}\right]^{\frac{k}{k-1}} \quad \text{...(i)}$$

By combining the above equation with Eq. 19.2 and the equation of state, the following equation for the temperature ratio may be readily obtained.

$$\frac{T_3}{T_2} = \left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_3}\right] \quad \text{...(ii)}$$

Again by introducing Eq. (i), the equation of state and Eq. 19.2 in the above noted energy equation the following equation for the velocity ratio may be obtained.

$$\frac{V_3}{V_2} = \frac{(Ma)_3}{(Ma)_2}\left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_3}\right]^{1/2} \quad \text{...(iii)}$$

By combining Eq. (iii) with the continuity equation the following equation for the area ratio may be obtained.

$$\frac{A_3}{A_2} = \frac{(Ma)_2}{(Ma)_3}\left[\frac{1+\frac{k-1}{2}(Ma^2)_3}{1+\frac{k-1}{2}(Ma^2)_2}\right]^{\frac{k+1}{2(k-1)}} \quad \text{...(iv)}$$

Now using Eq. (iv) to relate conditions at the throat and at the nozzle exit by setting $Ma_2 = 1$ and $(A_3/A_2) = 2.5$, we get

$$2.5 = \frac{1}{(Ma)_3}\left[\frac{1+\frac{1.4-1}{2}(Ma^2)_3}{1+\frac{1.4-1}{2}(1)^2}\right]^{\frac{1.4+1}{2(1.4-1)}}$$

Solving the above equation by trial and error, we get

$$(Ma)_3 = 2.44 \text{ or } (Ma)_3 = 0.24$$

The former value is for the divergent section acting as a nozzle and the later value corresponds to the condition when the divergent section acts as a diffuser.

(a) When sonic velocity occurs at the throat, the mass rate of flow is the same for both conditions and is determined by using Eq. 19.40.

Thus

$$G = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

in which

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{686.7\times10^3}{287\times(273.15+24)} = 8.052 \text{ kg/m}^3$$

$$\therefore \quad G = 12.5\times10^{-4}\sqrt{1.4\times686.7\times10^3\times8.052}\left(\frac{2}{1.4+1}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 2.013 \text{ kg/m}^3$$

(i) When $(Ma)_3 = 2.44$

(b) By applying Eq. (i) between points 1 in the reservoir and 3 at the exit of the nozzle with $(Ma)_1 = 0$, we get

$$\frac{p_3}{686.7\times10^3} = \left[\frac{1}{1+\frac{1.4-1}{2}(2.44)^2}\right]^{\frac{1.4}{1.4-1}}$$

$$\therefore \quad p_3 = 44.128\ \text{kN/m}^2 \text{ absolute}$$

(c) Again by applying Eq. (i) between points 1 and 3 the exit temperature may be calculated by considering

$$(Ma)_1 = 0 \text{ and } T_1 = (273.15 + 24) = 297.15\ \text{K}$$

Thus
$$\frac{T_3}{297.15} = \frac{1}{1+\frac{1.4-1}{2}(2.44)^2}$$

$$\therefore \quad T_3 = 135.6\ \text{K}$$

(d) The exit velocity is given by

$$V_3 = (Ma)_3\sqrt{kRT_3}$$

$$\therefore \quad V_3 = (2.44)\sqrt{1.4\times287\times135.6} = 569.5\ \text{m/s}$$

(ii) When $(Ma)_3 = 0.24$

(b) The exit pressure:

$$p_3 = \frac{686.7\times10^3}{10^3}\left[\frac{1}{1+\frac{1.4-1}{2}(0.24)^2}\right]^{\frac{1.4}{1.4-1}}$$

$$= 659.7\ \text{kN/m}^2$$

(c) The exit temperature:

$$T_3 = \frac{297.15}{1+\frac{1.4-1}{2}(0.24)^2} = 293.8\ \text{K}$$

(d) The exit velocity:

$$V_3 = 0.24\sqrt{1.4\times287\times293.8} = 82.5\ \text{m/s}$$

Example 19.7. *Conditions immediately upstream from a compression shock in an air flow are given as follows: $p_u = 34.3$ kN/m² absolute, $T_u = 0°$ C and $V_u = 1000$ m/s. Calculate the conditions immediately downstream from the shock.*

Solution

Assuming $k = 1.4$ and $R = 287$ J/kg.K

$$C_u = \sqrt{kRT_u}$$

$$= \sqrt{1.4\times287\times273.15} = 331.3\ \text{m/s}$$

$$(Ma)_u = \frac{1000}{331.3} = 3.02$$

From Eq. 19.50, we have

$$\frac{p_d}{34.3} = \frac{2\times1.4\times(3.02)^2-(1.4-1)}{(1.4+1)}$$

$$\therefore \quad p_d = 359.3\ \text{kN/m}^3 \text{ absolute}$$

From Eq. 19.51, we have

$$\frac{1000}{V_d} = \frac{(1.4+1)(3.02)^2}{(1.4-1)(3.02)^2+2}$$

$$\therefore \quad V_d = 258\ \text{m/s}$$

From Eq. 19.52, we have

$$(Ma)_d^2 = \frac{(1.4-1)(3.02)^2+2}{2\times1.4(3.02)^2-(1.4-1)}$$

$$\therefore \quad (Ma)_d = 0.474$$

By combining Eqs 19.50 and 19.51, we obtain

$$\frac{T_d}{T_u} = \frac{p_d\rho_u}{p_u\rho_d} = \frac{2k(Ma)_u^2-(k-1)}{(k+1)}\times\frac{(k-1)(Ma)_u^2+2}{(k+1)(Ma)_u^2}$$

$$\therefore \quad T_d = (273.15)\times\frac{359.3}{34.3}\times\frac{258}{1000} = 738\ \text{K}$$

Example 19.8. *Air at 30°C at a pressure of 294.3 kN/m² absolute flows into another reservoir through an orifice 50×10^{-4} m² in area. If the pressure in the receiver is 147.15 N/m² absolute, what is the discharge? Given $R = 287.14$ J/kg.K, $k = 1.4$ and coefficient of discharge $C_d = 0.65$.*

Solution

The density of air corresponding to a pressure of 294.3 kN/m² and a temperature of 30°C is given by equation of state as

$$\rho_1 = \frac{p_1}{RT_1}$$

$$= \frac{294.3\times10^3}{287.14\times(273.15+30)} = 3.38\ \text{kg/m}^3$$

The pressure ratio

$$\frac{p_2}{p_1} = \frac{147.15}{294.3} = 0.5$$

which is less than the critical pressure ratio. Therefore the mass rate of flow is given by Eq. 19.40 as

$$G = C_d A_2 \sqrt{k p_1 \rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2(k-1)}}$$

By substituting, we get

$$G = 0.65 \times 50 \times 10^{-4} \times (1.4 \times 294.3 \times 10^3 \times 3.38)^{1/2} \times \left(\frac{2}{1.4+1}\right)^{\frac{1.4+1}{2(1.4-1)}}$$

$$= 2.22 \text{ kg/m}^3$$

SUMMARY OF MAIN POINTS

1. When the density changes are appreciable the compressibility of the fluid needs to be taken into account in the analysis of such fluid flow phenomenon.
2. The density ρ of a particular gas is related to its absolute pressure p and absolute temperature T by the *equation of state*, which for a perfect gas is given as

$$p = \rho RT$$

where

R = gas constant

The absolute temperature is expressed in kelvin (K) and it is given by

$$T^\circ\text{(abs)} = TK = 273.15 + t^\circ C$$

3. A gas may be compressed or expanded by either *isothermal process* or *adiabatic process*.

An isothermal procss is that in which the temperature is held constant and it is governed by Boyles, law according to which

$$\frac{p}{\rho} = \text{constant}$$

An adiabatic process is governed by the following law

$$\frac{p}{\rho^k} = \text{constant}$$

where

k = adiabatic index.

The adiabatic index is given by

$$k = \frac{C_p}{C_v}$$

where

C_p = specific heat at constant pressure; and

C_v = specific heat at constant volume.

For air and other diatomic gases in the most ranges of temperature and pressure $k = 1.4$.

4. For ne dimensional steady flow of compressible fluid the equation of continuity is given as

$$\rho AV = \text{constant}$$

and in diffrential form continuity equation is given as

$$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dV}{V} = \text{constant}$$

5. The energy equation for flow of compressible fluid under isothermal condition is given as

$$\frac{p}{\rho g}\log_e p + \frac{V^2}{2g} + z = \text{constant}$$

The energy equation for flow of compressible fluid under adiabatic condition is given as

$$\frac{p}{\rho g}\frac{k}{k-1} + \frac{V^2}{2g} + z = \text{consant}$$

6. The velocity of sound wave is given by

(i) In terms of Bulk modulus

$$C = \sqrt{\frac{dp}{d\rho}} = \sqrt{\frac{K}{\rho}}$$

(ii) For isothermal process

$$C = \sqrt{\frac{p}{\rho}} = \sqrt{RT}$$

(iii) For adiabatic process

$$C = \sqrt{\frac{kp}{\rho}} = \sqrt{kRT}$$

7. Mach number M_a (or N_M) is given by

$$M_a = \frac{V}{C}$$

For $M_a < 1$ the flow is sub-sonic flow

For $M_a > 1$ the flow is super-sonic flow

For $M_a = 1$ the flow is sonic flow

8. In a sub-sonic flow the elastic waves will travel in the forward direction ahead of the object.

In a sonic flow both the object and the elastic waves will travel side by side.

In a super-sonic flow the object moves ahead of the elastic waves.

9. Mach angle α is given by

$$\sin\alpha = \frac{C}{V} = \frac{1}{M_a}$$

10. Stagnation point is defined as the point in a fluid stream where the velocity of flow is reduced to zero. The pressure, density and temperature at the stagnation point are called stagnation pressure, stagnation density and stagnation temperature which are given by

$$p_s = p_2 = p_1\left[1 + \frac{k-1}{2}M_{a_1}^2\right]^{\frac{k}{k-1}}$$

$$\rho_s = \rho_2 = \rho_1\left[1 + \frac{k-1}{2}M_{a_1}^2\right]^{\frac{k}{k-1}}$$

$$T_s = T_2 = T_1\left[1 + \frac{k-1}{2}M_{a_1}^2\right]$$

Also $$T_s = \frac{1}{R}\frac{p_s}{\rho_1}$$

11. The dimension less stagnation pressure is given by

$$\frac{p_s - p_1}{\frac{\rho_1 V_1^2}{2}} = \frac{p_2 - p_1}{\frac{\rho_1 V_1^2}{2}} = 1 + \frac{1}{4}M_{a_1}^2 + \frac{2-k}{24}M_{a_1}^4 + \ldots$$

12. Area velocity relationship for compressible fluid flow through a pipe of varying cross-section is given by

$$\frac{dA}{A} = \frac{dV}{V}(M_a^2 - 1)$$

(i) For sub-sonic flow, i.e., $M_a < 1$

$\frac{dV}{V} > 0$; $\frac{dA}{A} < 0$ (convergent nozzle)

$\frac{dV}{V} < 0$; $\frac{dA}{A} > 0$ (divergent diffuser)

(ii) For super-sonic flow, i.e., $M_a > 1$

$\frac{dV}{V} > 0$; $\frac{dA}{A} > 0$ (divergent nozzle)

$\frac{dV}{V} < 0$; $\frac{dA}{A} < 0$ (convergent diffuser)

(iii) For sonic flow, i.e., $M_a = 1$

$\frac{dA}{A} = 0$ (straight flow passage since ΔA must be zero)

13. The velocity of flow of compressible fluid through a convergent nozzle is given by

$$V_2 = \sqrt{\frac{2k}{k-1}\frac{p_1}{\rho_1}\left[1 - \left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right]}$$

14. The maximum mass flow rate G_{max} of a compressible fluid through a convergent nozzle is given by

$$G_{max} = A_2\sqrt{kp_1\rho_1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{2k-2}}$$

15. The mass rate of flow G of a compressible fluid through a convergent divergent nozzle is given by

$$G = \left(\frac{2k}{k-1}p_1\rho_1\right)^{1/2} A_3\left[\left(\frac{p_3}{p_1}\right)^{\frac{2}{k}} - \left(\frac{p_3}{p_1}\right)^{\frac{k+1}{k}}\right]^{1/2}$$

16. When a super-sonic flow is converted to a subsonic flow normal shock waves may occur.

The relation between the pressures on the upstream and downstream of normal shock waves developed in a pipe is given by the following equations:

(i) Equation of Rayleigh line

$$p_x + \frac{(G/A)^2}{\rho_x} = p_y + \frac{(G/A)^2}{\rho_y}$$

(ii) Equation of Fanno line

$$\frac{(G/A)^2}{2\rho_u^2}+\frac{k}{k-1}\frac{p_u}{\rho_u}=\frac{(G/A)^2}{2\rho_d^2}+\frac{k}{k-1}\frac{p_d}{\rho_d}$$

(iii) Rankine-Hugoniot Equations

$$\frac{p_d}{p_u}=\frac{(k+1)\frac{\rho_d}{\rho_u}-(k-1)}{(k+1)-(k-1)\frac{\rho_d}{\rho_u}}$$

$$\frac{\rho_d}{\rho_u}=\frac{V_u}{V_d}=\frac{(k-1)+(k+1)\frac{p_d}{p_u}}{(k+1)+(k-1)\frac{p_d}{p_u}}$$

17. The mass rate of flow of comprssible fluid G through a venturimeter or orifice meter is given by

$$G = CA_2\rho_2\sqrt{\frac{\frac{2k}{k-1}\frac{p_1}{\rho_1}\left\{1-\left(\frac{p_2}{p_1}\right)^{\frac{k-1}{k}}\right\}}{1-\left(\frac{A_2}{A_1}\right)^2\left(\frac{p_2}{p_1}\right)^{\frac{2}{k}}}}$$

where
C = coefficient of the meter;
A_1 = area at inlet; and
A_2 = area at the throat.

PROBLEMS

19.1 A volume of air at an atmospheric pressure of 101.04 kN/m² [1.03 kg(f)/cm²] absolute and 24°C is compressed adiabatically to a gage pressure of 3.09 × 10⁷ N/m² [315 kg(f)/cm²]. Calculate the final volume.

[*Ans.* $\left(\frac{1}{59.75}\right)$ of original volume]

19.2 Calculate the pressure on the nose of a modern airplane moving at 800 km/h through still air and the Mach number, when atmospheric pressure and air temperature are 68.7 kN/m² (0.7 kg(f)/cm²] absolute and –1°C respectively. Take mass density of air as 1.295 kg/m³.

[*Ans.* 106.34 kN/m²; 0.82]

19.3 Oxygen flows through a 300 mm circular duct at the rate of 7.22 kN per minute [736 kg(f) per minute]. Gages at points 1 and 2 indicate pressures of 480.7 kN/m² [4.9 kg(f)/cm²] and 48.07 kN/m² [0.49 kg(f)/cm²] respectively. Calculate the Mach number of the flow at points 1 and 2. Assume t = 25°C, R = 263 J/kg.K and k = 1.4. [*Ans.* 0.085; 0.85]

19.4 Air at an absolute pressure of 101.04 kN/m² [1.03 kg(f)/cm²] and 15°C with zero velocity expands isentropically to Ma = 0.50. Calculate (a) the final pressure, (b) the final density, and (c) the temperature. Take R = 287 J/kg.K.

[*Ans.* (a) 85.18 kN/m²; (b) 1.08 kg/m³; (c) 1.28°C]

19.5 Solve Problem 19.4 if the air at an absolute pressure of 101.04 kN/m² and 15°C, with an initial Mach number of 0.50 expands isentropically to a final Mach number of 1.0.

[*Ans.* (a) 63.32 kN/m², (b) 0.875 kg/m³; (c) –21°C]

19.6 Air flows isentropically in a variable area duct. At section 1, V_1 = 64 km/hour, p_1 = 101.04 kN/m² absolute and t_1 =22°C. At section 2, p_2 = 68.7 kN/m² absolute. Determine (a) the velocity and temperature at section 2 and (b) the area ratio of these two sections. Take R = 287 J/kg K

[*Ans.* (a) 249.4 m/s, – 8.8° C; (b) 10.65]

19.7 Derive the relationship

$$\frac{T_2}{T_1}=\left(\frac{p_2}{p_1}\right)^{\frac{(k-1)}{k}}$$

and specifically state the conditions for which the equation is valid.

19.8 Air flows isentropically in a flow passage with increasing velocity in the direction of flow. At section 1, p_1 = 274.68 kN/m² [2.8 kg(f)/cm²] absolute, t_1 = 180°C and V_1 = 150 m/s. At section 2, the air flow just attains the sonic velocity. Calculate for section 2, (a) the pressure, (b) the temperature, (c) the velocity, and (d) the density of air. Take R = 287 J/kg.K.

[*Ans.* (a) 145.11 kN/m²; (b) 104.45°C (c) 389.53 m/s; (d) 1.339 kg/m³]

19.9 Considering the variation of velocity with area, prove that

$$\frac{dA}{dV}=\frac{A}{V}(Ma^2-1)$$

and explain variation of velocity with change in area for the subsonic and supersonic velocities.

19.10 Nitrogen at an absolute pressure of 123.61 kN/m² and 13°C flows in a 300 mm diameter pipe at a velocity of 12 m/s when it enters a 300 mm × 200 mm gradual reducer. Assuming isentropic flow, calculate for the 200 mm section, (a) the velocity, (b) the pressure, and (c) the temperature of nitrogen. Take R = 296.26 J/kg.K [30.2 kg(f)–m/kg(m) deg C abs] and k = 1.4.

[*Ans.* (a) 27.12 m/s; (b) and (c) pressure and temperature are almost the same as that at 300 mm section]

[**Hint.** $(Ma)_1$ = 0.035,

$$\frac{A_2}{A_1}=\frac{(Ma)_1}{(Ma)_2}\left[\frac{1+\frac{k-1}{2}(Ma^2)_2}{1+\frac{k-1}{2}(Ma^2)_1}\right]^{\frac{k+1}{2(k-1)}}$$

from which $(Ma)_2$ = 0.0791. Since $(Ma)_2$ is very small, no appreciable change in pressure and temperature at 200 mm section occurs]

19.11 An ideal gas with k = 2 and R = 225.63 J/kg.K [23 kg(f)-m/kg (m) deg C abs] flows through an isentropic convergent-divergent nozzle. The pressure and temperature at the nozzle inlet are respectively 412.02 kN/m² [4.2 kg(f)/cm²] absolute and 27°C. Air discharges into the atmosphere where the barometric pressure is 740 mm of mercury. For the condition of complete expansion of air as it leaves the exit, calculate (a) the temperature in K at the throat and exit, and (b) the sonic velocities at the inlet, throat and exit sections of the nozzle.

[*Ans.* (a) 200 K, 146.9 K; (b) 368 m/s, 300.4 m/s, 257.5 m/s]

19.12 A pitot tube is used to measure the air velocity in the test section of a wind tunnel. During a test run, the manometer connected to the stagnation tube indicates a total gage pressure of 112 mm of water when the static gage pressure reads 22 mm of water. The barometric pressure is 101 kN/m² and the temperature of the atmosphere is 24°C. Calculate the air velocity in the test section. [*Ans.* 4.28 m/s]

19.13 Show that the celerity C of a sound wave in a fluid is given by

$$C^2=\left(\frac{dp}{d\rho}\right)$$

Hence show that in a perfect gas $C=\sqrt{kRT}$ where k is the adiabatic index, R the gas constant and T the absolute temperature. Also show that for a liquid $C=\left(\sqrt{K/\rho}\right)$ where K is the bulk modulus of elasticity and ρ the density of the fluid.

19.14 Define pressure coefficient. A Prandtl pitot tube attached to the nose of an aircraft gave a pressure coefficient of 1.04. Determine the Mach number of the flying aircraft.

[*Ans.* 0.4]

Impact of Free Jets

Chapter 20

20.1 INTRODUCTION

A jet of fluid emerging from a nozzle has some velocity and hence it possesses a certain amount of kinetic energy. If this jet strikes an obstruction placed in its path, it will exert a force on the obstruction. This impressed force is known as impact of the jet and it is designated as hydrodynamic force, in order to distinguish it from the forces due to hydrostatic pressure. Since a dynamic force is exerted by virtue of fluid motion, it always involves a change of momentum, unlike a force due to hydrostatic pressure that implies no motion. Hence the impluse-momentum principle may be utilized to evaluate the hydrodynamic force exerted on a body by a fluid jet. In this chapter, different cases of forces exerted by free jets on stationary and moving plates or vanes of different shapes have been discussed, which forms an important application of the impulse-momentum principle as mentioned in Chapter 8.

20.2 FORCE EXERTED BY FLUID JET ON STATIONARY FLAT PLATE

(a) Flat Plate Normal to the Jet. Let a jet of diameter d and velocity V issue from a nozzle and strike a flat plate as shown in Fig. 20.1 (a). The plate is held stationary and perpendicular to the centre line

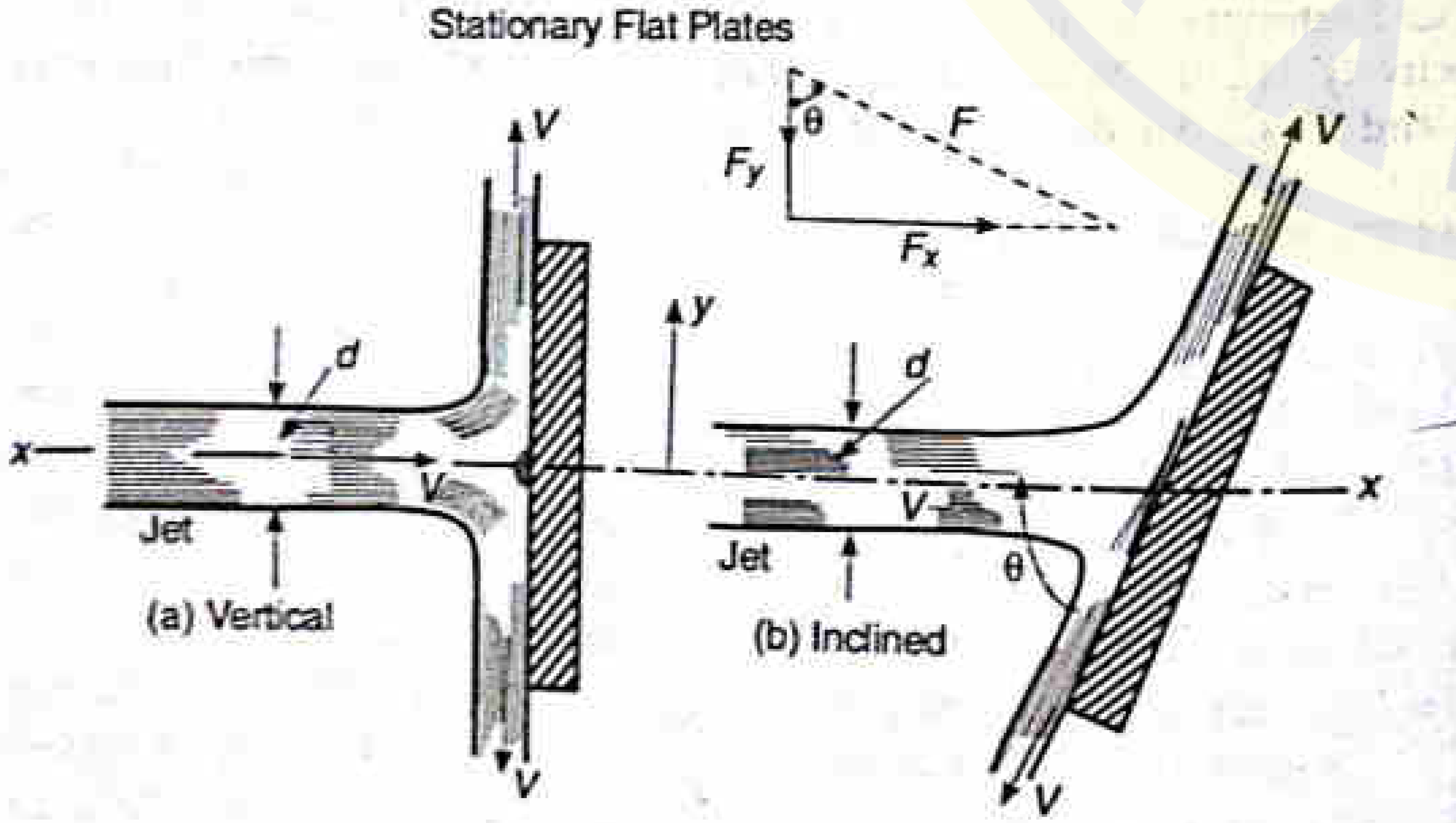


Figure 20.1 *Fluid jet striking stationary flat plate*

of the jet. The jet after striking the plate will leave it tangentially i.e., the jet will get deflected through 90°. If the plate is quite smooth the friction between the jet and the plate may be neglected. Further if there is no energy loss in the flow because of impact of the fluid jet, and the difference in elevation between the incoming and outgoing jets is neglected; the application of Bernoulli's equation indicates that the jet will move on and off the plate with the same velocity V. However, if some energy loss occurs the velocity of the fluid leaving the plate will be slightly less than V.

The quantity of fluid striking the plate $Q = (\pi d^2/4) \times V = aV$, where a is the area of cross-section of the jet. Thus the mass of fluid issued by the jet per second is $m = \rho Q = \rho a V$; where ρ represents the mass density of the fluid. Since $\rho = (w/g)$, where w is the specific weight of the fluid, the mass m may also be expressed as $m = (waV/g)$.

After striking the plate since the jet gets deflected through 90°, the component of the velocity of the jet leaving the plate, in the original direction of the striking jet will be zero. Therefore by applying the impulse-momentum equation, the force F exerted by the stationary plate on the jet of fluid in the direction normal to the plate may be determined as

$$-F = \frac{waV}{g}(0 - V)$$

or

$$F = \frac{waV^2}{g} \qquad \text{...(20.1)}$$

The sign for the force F has been considered as negative because the force exerted by the plate on the jet is in the negative x-direction. Further the force which the jet exerts on the plate is equal and opposite to the force exerted by the plate on the jet, hence it is equal to F acting in the positive x direction and its magnitude is given by Eq. 20.1.

(b) Flat Plate Inclined at Angle θ to the Jet. Let a stationary smooth plate be held inclined at an angle θ to the direction of flow of jet of diameter d and velocity V, as shown in Fig. 20.1 (b). After impact the fluid will slide away in all directions over the plate and it will leave the plate tangentially. It is assumed that there exists no frictional resistance at the plate and there are no impact losses. Further if the small difference in elevation between the incoming and outgoing jets is neglected, the application of Bernoulli's equation then yields that velocity of the divided jets developed after impact will be the same as the velocity of jet before impact i.e., after impact the velocity of jet parallel to the surface of the plate will also be equal to V.

The mass of fluid striking the plate per second is

$$m = \rho Q = \rho a V = (waV/g)$$

where all the notations are same as defined earlier.

The component of the velocity of the jet before impact in the direction normal to the plate = V (sin θ).

Further since the jet after impact leaves the plate tangentially, the component of the velocity of the leaving jet in the direction normal to the plate becomes zero.

Therefore by the applications of the impulse-momentum equation, the thrust exerted by the jet in the direction normal to the plate may be expressed as

$$F = \frac{waV}{g}[(V \sin \theta) - 0]$$

or
$$F = \frac{waV^2}{g} \sin\theta \qquad ...(20.2)$$

There being no resultant thrust acting in the direction tangential to the plate the thrust F given by Eq. 20.2 is the resultant dynamic thrust exerted by the jet.

The dynamic thrust F can be resolved into two components viz., component F_x parallel to the direction of the jet and component F_y normal to the direction of the jet. Thus

$$F_x = F\sin\theta = \frac{waV^2}{g}\sin^2\theta \qquad ...(20.3)$$

$$F_y = F\cos\theta = \frac{waV^2}{g}\sin\theta\cos\theta \qquad ...(20.4)$$

The division of discharge that results in this case may be computed by applying the condition that there is no resultant force acting in the direction parallel to the plate, because the pressure everywhere is atmospheric and there is no frictional resistance.

Thus if Q_1 and Q_2 represent the discharge flowing in the two directions then according to momentum equation

$$\left(\frac{wQ_1V}{g} - \frac{wQ_2V}{g}\right) - \frac{w}{g}QV\cos\theta = 0$$

or
$$(Q_1 - Q_2) - Q\cos\theta = 0 \qquad ...(i)$$

Also
$$Q_1 + Q_2 = Q \qquad ...(ii)$$

Solving Eqs (i) and (ii) the values of Q_1 and Q_2 are obtained as

$$Q_1 = \frac{Q}{2}(1+\cos\theta) \qquad ...(20.5)$$

$$Q_2 = \frac{Q}{2}(1-\cos\theta) \qquad ...(20.6)$$

Alternatively the expression for the resultant force F may also be obtained by applying the momentum equation along the x and y directions. Thus if F is the force exerted by the plate on the jet and F_x and F_y are its components along the x and y directions respectively then according to the momentum equation

$$-F_x = [\rho Q_1 V\cos\theta - \rho Q_2 V\cos\theta] - \rho QV$$

or
$$F_x = \rho V[Q - (Q_1 - Q_2)\cos\theta]$$

and
$$F_y = [\rho Q_1 V\sin\theta - \rho Q_2 V\sin\theta]$$

or
$$F_y = \rho V(Q_1 - Q_2)\sin\theta$$

For F_x the negative sign has been considered because this component of the force exerted by the plate on the jet is in the negative x-direction.

$\therefore$
$$F = \sqrt{F_x^2 + F_y^2}$$

$$= \sqrt{(\rho V)^2[(Q_1 - Q_2)^2 + Q^2 - 2Q(Q_1 - Q_2)\cos\theta]}$$

Substituting for Q_1 and Q_2 from Eqs. 20.5 and 20.6 in the above expression, it becomes

$$F = \rho QV\sin\theta = \frac{waV^2}{g}\sin\theta$$

which is same as Eq. 20.2.

Further if α is the angle of inclination of F with x-direction then

$$\tan\alpha = \frac{F_y}{F_x} = \frac{\rho V(Q_1 - Q_2)\sin\theta}{\rho V[Q - (Q_1 - Q_2)\cos\theta]}$$

Again substituting for Q_1 and Q_2 from Eqs. 20.5 and 20.6 in the above expression, it becomes

$$\tan\alpha = \cos\theta = \tan(90° - \theta)$$

which means that the resultant force F is acting in the direction normal to the plate.

It may however be noted that for a given inclination of the plate the magnitude of the resultant force F as well as its inclination α will depend on the magnitudes of Q_1 and Q_2. Thus if

$$Q_1 = Q_2, \text{ then}$$
$$F = F_x = \rho QV;\ F_y = 0; \text{ and } \alpha = 0$$

i.e. for equal division of discharge the resultant force F will be acting in the direction of the jet.

In both the cases discussed above, since the plate is held stationary, the work done by the force exerted by the jet on the plate is zero.

20.3 FORCE EXERTED BY FLUID JET ON MOVING FLAT PLATE

(a) Flat Plate Normal to Jet. As shown in Fig. 20.2 (a) a flat plate held normal to the jet is assumed to move with a velocity u in the same direction as the jet. In this case absolute velocity of the jet V, will not be the effective velocity with which the jet strikes the plate, because as the jet is about to strike

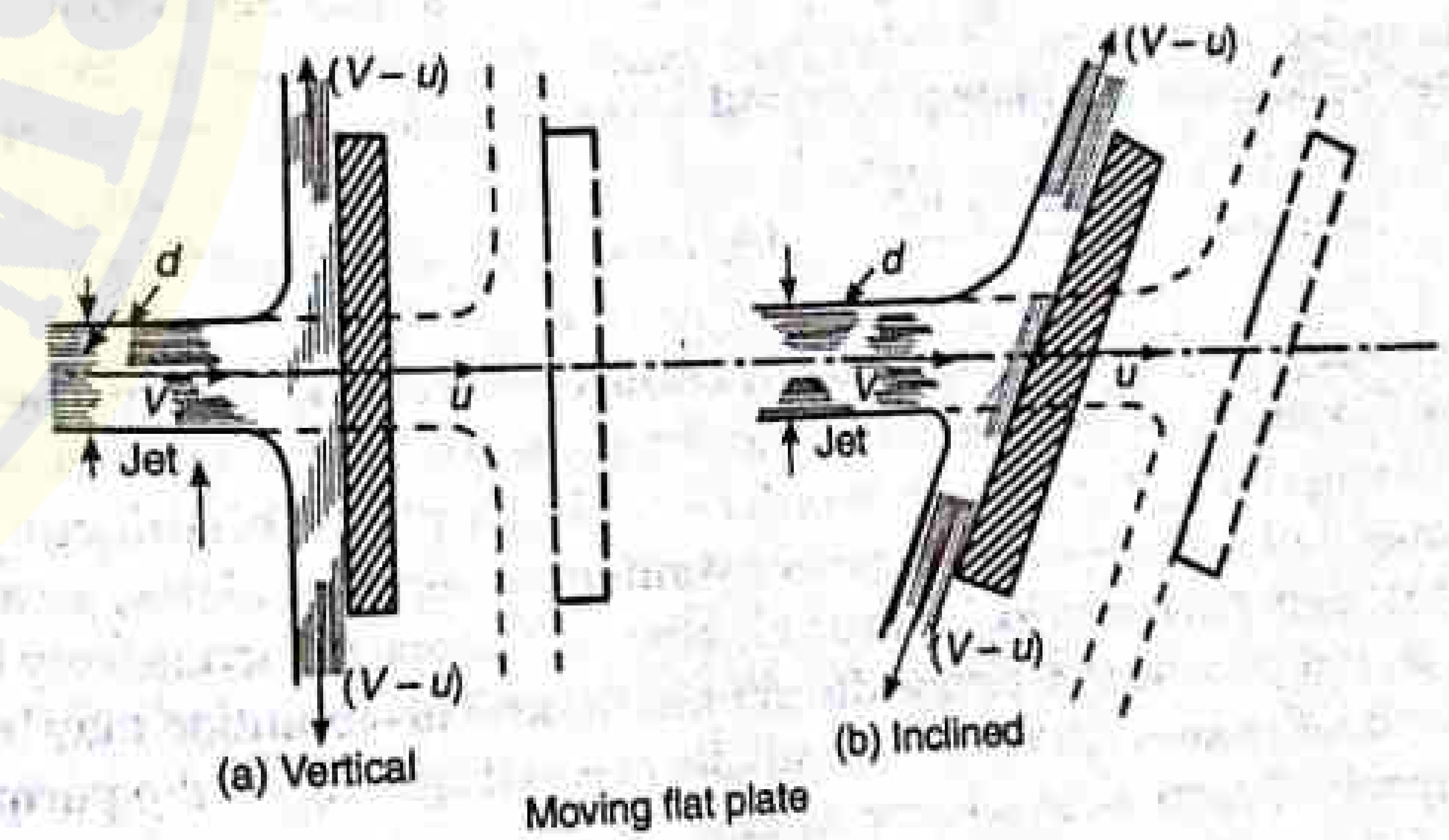


Figure 20.2 *Fluid jet striking moving flat plate*

the plate, the plate has also moved away from the jet with a velocity u, and only that mass of fluid which really overtakes the plate will be striking it to cause the impinging action.

This type of problem may therefore be analyzed by applying the principle of relative motion to the whole system. This is done by bringing the moving plate into a stationary state before the impact momentum equation is applied. For this a velocity equal in magnitude to that of the plate but in opposite direction, i.e., $-u$, is applied to the whole system, thereby bringing the plate to rest and making the effective velocity with which the jet strikes the plate equal to $(V-u)$, which is the velocity of the jet relative to the plate.

Thus the quantity of fluid striking the plate per second is

$$Q\left(\frac{\pi d^2}{4}\right)(V-u) = a\,(V-u)$$

and the mass of fluid striking the plate is

$$m = \rho Q = \rho a\,(V-u) = \left[\frac{wa(V-u)}{g}\right]$$

Again if there is no frictional resistance offered by the plate to the jet, as also there are no impact losses and the small difference in elevation between the incoming and outgoing jets is neglected, the application of Bernoulli's equation indicates that the jet leaving the plate will have the velocity equal to $(V-u)$. Further since the jet, after impact, gets deflected through 90°, the component of the velocity of the leaving jet in the direction normal to the plate is equal to zero. Therefore by applying the impulse-momentum equation the dynamic thrust exerted by the jet on the plate in the direction normal to the plate is obtained as

$$F = \left[\frac{wa(V-u)}{g}\right][(V-u)-0]$$

or

$$F = \frac{wa(V-u)^2}{g} \qquad \text{...(20.7)}$$

The work done by the jet on the plate per second

$$= (F \times u) = \frac{wa(V-u)^2 u}{g}$$

The above discussed case will not be practically feasible because the distance between the plate and nozzle will be constantly increasing at the rate of u per second, which in turn would require a continuous lengthening of the jet at this rate in order to strike the plate. This difficulty may however be overcome, if instead of a single plate, there is a continuous series of plates, arranged at a fixed distance apart so that each plate moving with a velocity u, may appear successively before the jet in the same position, and all moving in the same direction as the jet. This condition may be accomplished if a number of flat smooth plates are mounted radially at equal spacing on the periphery of a large wheel, which is capable of rotating in a vertical plane.

Accordingly as shown in Fig. 20.3, let a jet of diameter d and velocity V strike the wheel tangentially at its lower edge, such that the bottom-most vertical plate is impinged at its centre. The impact of jet will push the plate in the same direction as the jet and its position will be occupied by another plate in succession. Likewise the wheel will commence revolving at a constant angular speed. Corresponding to this angular speed, let u be the tangential velocity of the wheel along the centre of the plates, which will be the velocity of the plates as well.

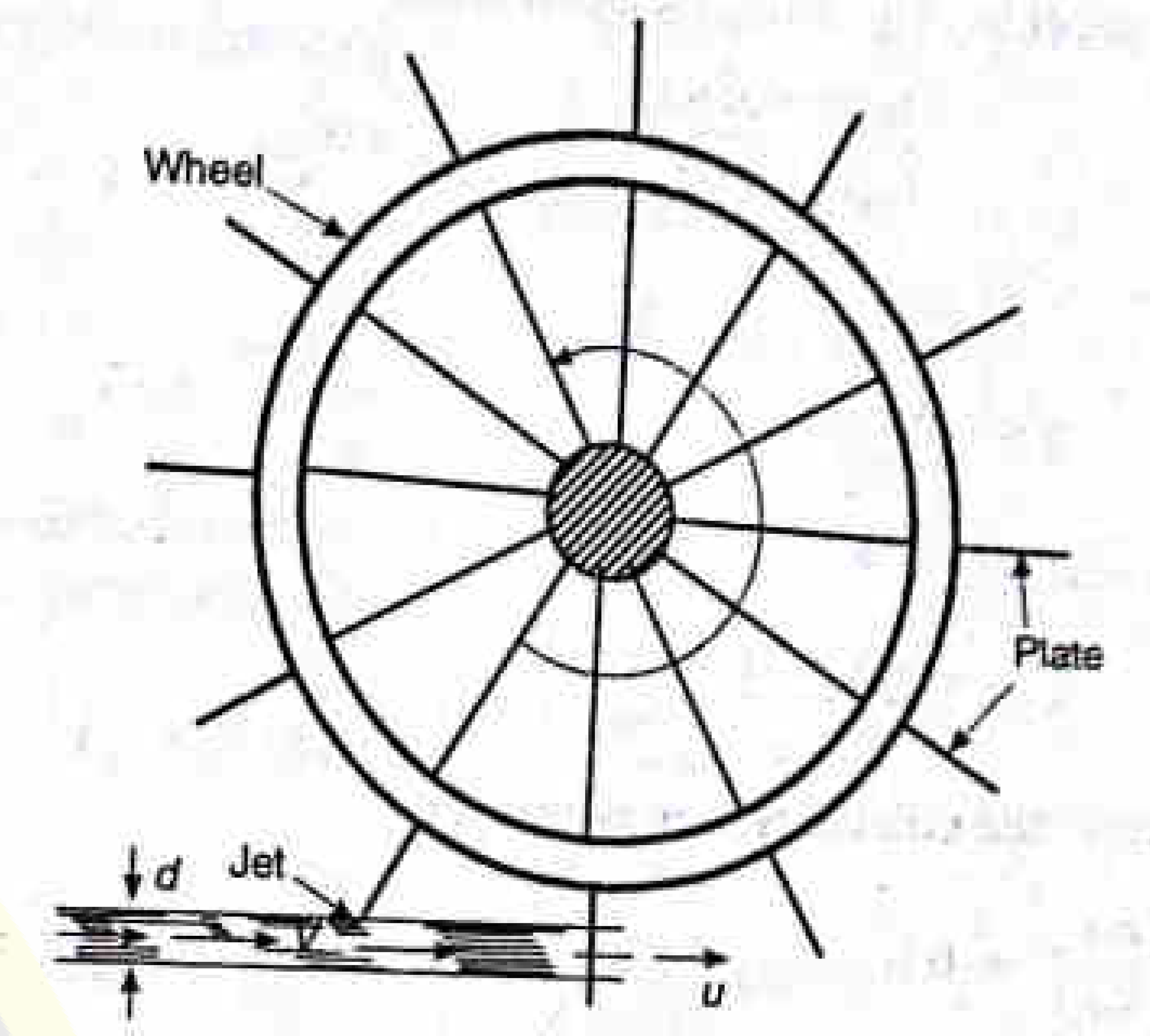


Figure 20.3 *Flat plates mounted on the periphery of a wheel*

In this case since the plates face the jet normally one after the other it is as if one plate is always in the vertical position to intercept the jet. Hence the entire fluid issuing from the nozzle will strike the plates when all the plates are considered. Therefore the mass of fluid striking the plates is

$$m = \rho\left(\frac{\pi d^2}{4}\right)V = \left(\frac{waV}{g}\right)$$

Again the effective velocity of the jet before impact will be $(V-u)$ which is the relative velocity of the jet with respect to the plate. Further since the jet will leave the plate tangentially, the component of the velocity of the jet leaving the plate, in the direction normal to the plate is equal to zero. Therefore by applying the impulse-momentum equation to the flow system, the force exerted by the jet in the direction normal to the plate is given by

$$F = \frac{waV}{g}[(V-u)-0]$$

or

$$F = \frac{waV}{g}(V-u) \qquad \text{...(20.8)}$$

The work done by the jet per second will be the output of the jet.

Thus, output of jet $= (F \times u)$

$$= \frac{waV(V-u)u}{g}$$

The kinetic energy supplied by the jet per second will be input of the jet to the wheel.

Thus, input of jet $$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

∴ Efficiency of the wheel is

$$\eta = \frac{\text{Output of jet}}{\text{Input of jet}}$$

or

$$\eta = \frac{\frac{waV}{g}(V-u)u}{\frac{1}{2}\frac{waV}{g}(V^2)}$$

or

$$\eta = \frac{2u(V-u)}{V^2} \qquad ...(20.9)$$

For a given jet velocity V, the efficiency is maximum if

$$\left(\frac{d\eta}{du}\right) = 0$$

Thus

$$\frac{d\eta}{du} = \frac{2(V-2u)}{V^2} = 0$$

or

$$u = \frac{V}{2}, \text{ since } \frac{2}{V^2} \neq 0$$

Hence for the efficiency of the wheel to be maximum,

$$u = \left(\frac{V}{2}\right)$$

and the corresponding maximum efficiency is

$$\eta_{max} = \frac{2u(2u-u)}{4u^2} = 0.5 \text{ or } 50\%.$$

(b) Flat Plate Inclined at Angle θ to Jet. Let a smooth plate held inclined at an angle θ to the direction of flow of jet, be moving with velocity u in the same direction as the jet, as shown in Fig. 20.2 (b). As stated earlier the effective velocity with which the jet strikes the plate will be the velocity of the jet relative to the plate i.e., $(V-u)$.

Thus the mass of fluid striking the plate per second is

$$m = \rho\left(\frac{\pi d^2}{4}\right)(V-u) = \left[\frac{wa(V-u)}{g}\right]$$

where all the notations are as defined earlier. Further the component of the velocity $(V-u)$ in the direction normal to the plate is $[(V-u)\sin\theta]$.

Again if there are no losses due to friction on the plate as well as due to impact and if the small difference in elevation is neglected the jet will come off the plate with velocity $(V-u)$. But as it leaves the plate tangentially, the component of this velocity in the direction normal to the plate will be zero.

Therefore by applying the impulse-momentum equation the force exerted on the plate in the direction normal to the plate is given by

$$F = \frac{wa(V-u)}{g}[(V-u)\sin\theta - 0]$$

or

$$F = \frac{wa(V-u)^2}{g}\sin\theta \qquad ...(20.10)$$

The normal thrust F can be resolved in two components viz., F_x parallel to the direction of the jet and F_y normal to the direction of jet. Thus,

$$F_x = F\sin\theta = \frac{wa(V-u)^2}{g}\sin^2\theta \qquad ...(20.11)$$

$$F_y = F\cos\theta = \frac{wa(V-u)^2}{g}\sin\theta\cos\theta \qquad ...(20.12)$$

The work done by the jet per second

$$= (F_x \times u) = \frac{wa(V-u)^2}{g}u\sin^2\theta$$

It is obvious from the above expression that for $\theta = 90°$ the same result as for a flat pate normal to the jet will be obtained.

20.4 FORCE EXERTED BY A FLUID JET ON STATIONARY CURVED VANE

On the basis of the direction of the jet striking the curved vane or plate, two distinct cases as discussed below may be considered.

(a) Jet Striking a Symmetrical Stationary Curved Vane at the Centre. As shown in Fig. 20.4, let a horizontal jet of cross-sectional area a and velocity V be striking a smooth symmetrical curved vane at its centre on the concave side. After striking the vane the jet will divide itself and the divided jets will glide over the vane and will leave the vane at its both the ends. The vane being smooth the velocity of the leaving jets will also be V. Let 2θ be the angle between the two tangents drawn to the vane at its outlet tips. Thus the jet after striking, will be deflected on each side through the angle of (180°– θ). Accordingly the component of the velocity of the leaving jet in the direction of flow of incoming jet = $(V\cos(180° - \theta)] = (-V\cos\theta)$. Therefore the application of the impulse-momentum equation gives the force exerted on the plate in the direction of flow of jet as

$$F = \frac{waV}{g}[V - (-V\cos\theta)]$$

or

$$F = \frac{waV^2}{g}(1+\cos\theta) \qquad ...(20.13)$$

When θ = 90°(or 180° – θ = 90°) the curved vane will be reduced to a flat plate, and the expression for F then becomes

$$F = \frac{waV^2}{g} \qquad \ldots(20.14)$$

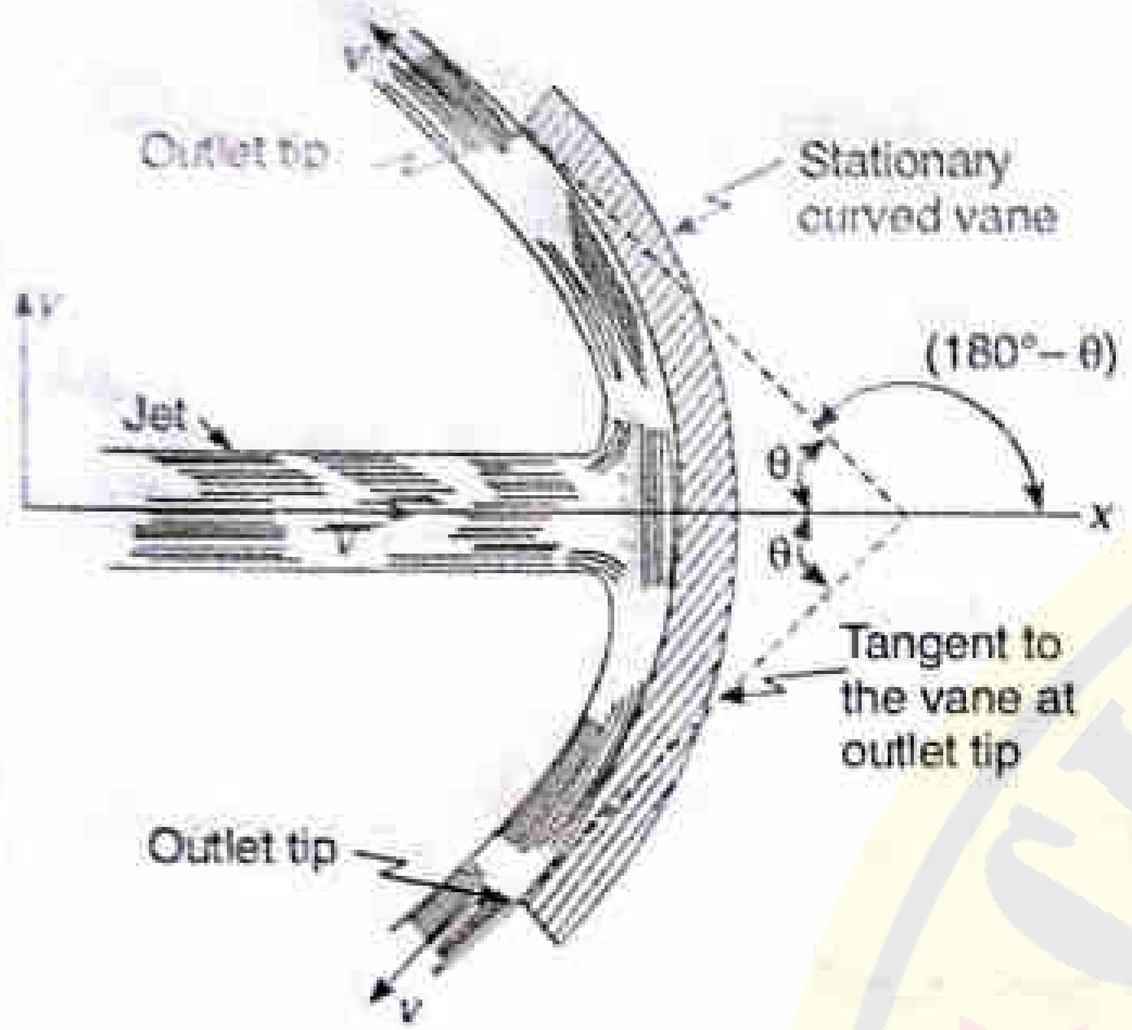


Figure 20.4 *Jet striking a stationary symmetrical curved vane at its centre*

which is same as previously obtained expression in the case of a flat plate held fixed in position. By comparing Eqs 20.13 and 20.14 it will be observed that the force exerted by a jet in its direction of flow, on a curved vane is always greater than that exerted on a flat plate, because

$$\frac{waV^2}{g}(1+\cos\theta) > \frac{waV^2}{g}$$

When $\theta = 0$ (or $180° - \theta = 180$), the vane becomes semicircular and the jets leaving at the outlet tips of the vane will be parallel to the incoming jet, but their directions will be opposite to the direction of the incoming or striking jet. The expression for F then becomes

$$F = \frac{2waV^2}{g} \qquad \ldots(20.15)$$

which indicates that the force exerted by a jet in its direction of flow, on a semicircular vane is twice that exerted on a flat plate, both plates being fixed in position.

From the foregoing discussion it is thus observed that as the angle of deflection of jet increases from 90° (for a flat plate) to 180 (for a semicircular plate) the value of F also increases from $\left(\frac{waV^2}{g}\right)$ to $\left(\frac{2waV^2}{g}\right)$.

Obviously on account of the symmerty the net force acting on the vane in the direction normal to the direction of flow of jet will be equal to zero. Hence in this case the resultant thrust acting on the vane will be equal to F.

(b) Jet Striking an Unsymmetrical Stationary Curved Vane Tangentially at one of the Tips Let a jet of cross-sectional area a and velocity V strike tangentially at one of the tips, called inlet tip, of a smooth curved vane as shown in Fig. 20.5. The jet after gliding over the inner surface of the vane will leave it tangentially at its other tip called outlet tip. Let the tangent at the inlet tip make an angle θ with the horizontal x-axis and that at the outlet tip make an angle ϕ with this axis. The jet will therefore get deflected through an angle $[180 - (\theta + \phi)]$. The components of the velocity of jet at the inlet and the outlet tips of the vane in the x-direction will be $(V \cos\theta)$ and $(-V\cos\phi)$ respectively. Therefore by applying the impulse-momentum equation, the force exerted by the jet on the vane in the x-direction is obtained as

$$F_x = \frac{waV}{g}[V\cos\theta - (-V\cos\phi)]$$

or

$$F_x = \frac{waV^2}{g}(\cos\theta + \cos\phi) \qquad \ldots(20.16)$$

If the curvature of the vane at the outlet tip is such that angle ϕ is more than 90°, then the component of the velocity of the jet at the outlet tip will be $(V \cos 180° - \phi)$. Accordingly the expression for the force F_x is again same as given by Eq. 20.16.

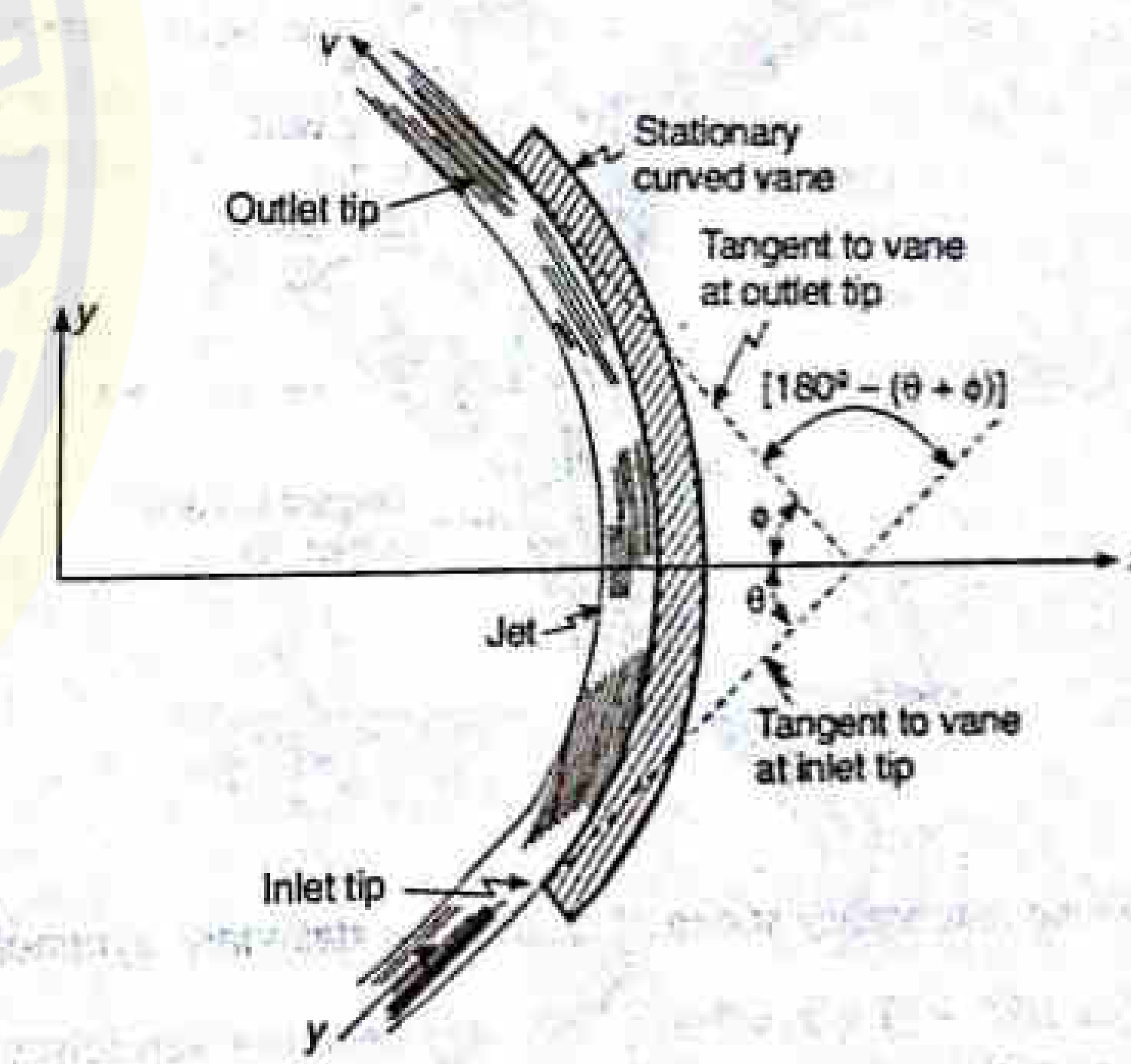


Figure 20.5 *Jet striking tangentially at one of the tips of a stationary unsymmetrical curved vane*

The components of the velocity of jet at the inlet and the outlet tips of the vane in the y-direction will be $(V \sin\theta)$ and $(V \sin\phi)$ respectively. The application of impulse-momentum equation then gives the force exerted by the jet on the vane in the y-direction as

$$F_y = \frac{waV}{g}(V\sin\theta - V\sin\phi)$$

or
$$F_y = \frac{waV^2}{g}(\sin\theta - \sin\phi) \qquad ...(20.17)$$

The resultant thrust exerted by the jet on the vane will be $F = \sqrt{F_x^2 + F_y^2}$ and its inclination with the x-axis is

$$\alpha = \tan^{-1}\left(\frac{F_y}{F_x}\right) \qquad ...(20.18)$$

If the curvature of the vane is same at both inlet and outlet tips i.e., $\theta = \phi$, as shown in Fig. 20.6, then the expression for F_x is reduced to

$$F_x = \frac{2waV^2}{g}\cos\theta \qquad ...(20.19)$$

and the value of F_y becomes zero. As such in this case F_x represents the resultant thrust on the vane.

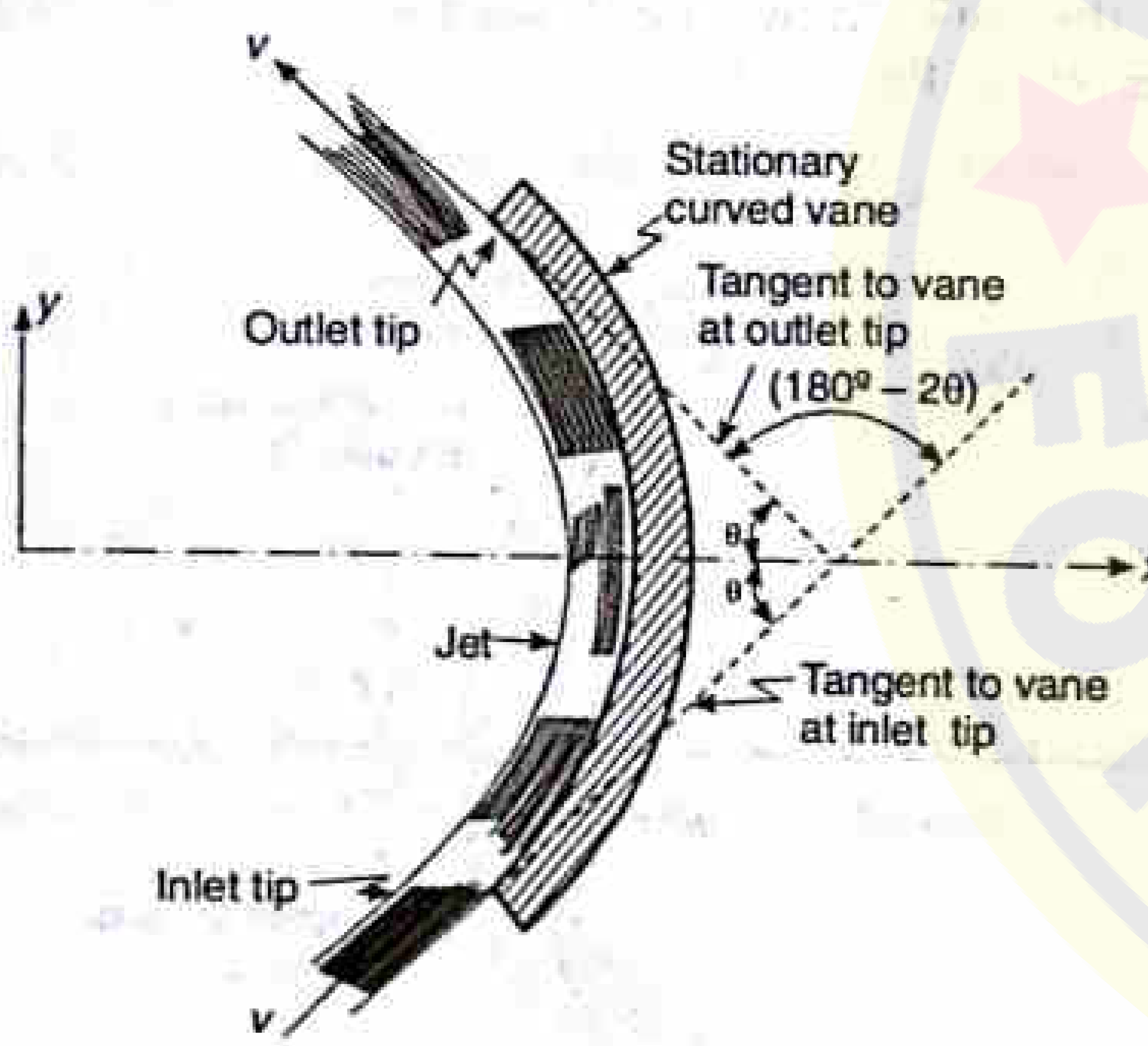


Figure 20.6 *Jet striking tangentially at one of the tips of a stationary symmetrical curved vane*

Further if $\theta = \phi = 0$ [or $180° - (\theta + \phi) = 180°$], the vane becomes semicircular and therefore the incoming and the outgoing jets will be parallel to the horizontal axis of symmetry but the two jets will be in opposite direction. The expression for F_x then becomes

$$F_x = \frac{2waV^2}{g} \qquad ...(20.20)$$

which is same as obtained previously in section 20.4 for a semicircular vane.

In the expressions derived above the velocity of jet has been assumed to be same at both inlet and outlet tips of the vane. It will hold good if there is no friction on the vane, there are no impact losses and the small difference in elevation between the inlet and the outlet ends of the vane is neglected.

However, if some frictional resistance is offered to the jet as it glides over the vane and/or if there occurs an impact loss, the velocity of jet at outlet end will not be the same as that at the inlet end. Under these circumstances

$$F_x = \left(\frac{waV_1}{g}\right)(V_1\cos\theta + V_2\cos\phi) \qquad ...(20.21)$$

$$F_y = \left(\frac{waV_1}{g}\right)(V_1\sin\theta - V_2\sin\phi) \qquad ...(20.22)$$

where V_1 and V_2 represent the velocities of the jet at the inlet and the outlet ends of the vane respectively, a is the cross-sectional area of the jet at inlet and the other notations are same as defined earlier.

In both these cases since the vane is held stationary the work done is equal to zero.

20.5 FORCE EXERTED BY A FLUID JET ON MOVING CURVED VANE

(a) Jet Striking Symmetrical Moving Curved Vane at the Centre. Let the curved vane of section 20.4 (a) be allowed to move with a velocity u in the direction of jet as shown in Fig. 20.7. Thus as stated earlier the effective velocity with which the jet strikes the vane will be the velocity of jet relative to the plate i.e., $(V-u)$. Accordingly the mass of fluid striking the vane per second will be $m = [\rho a\,(V-u)] = [wa\,(V-u)/g]$.

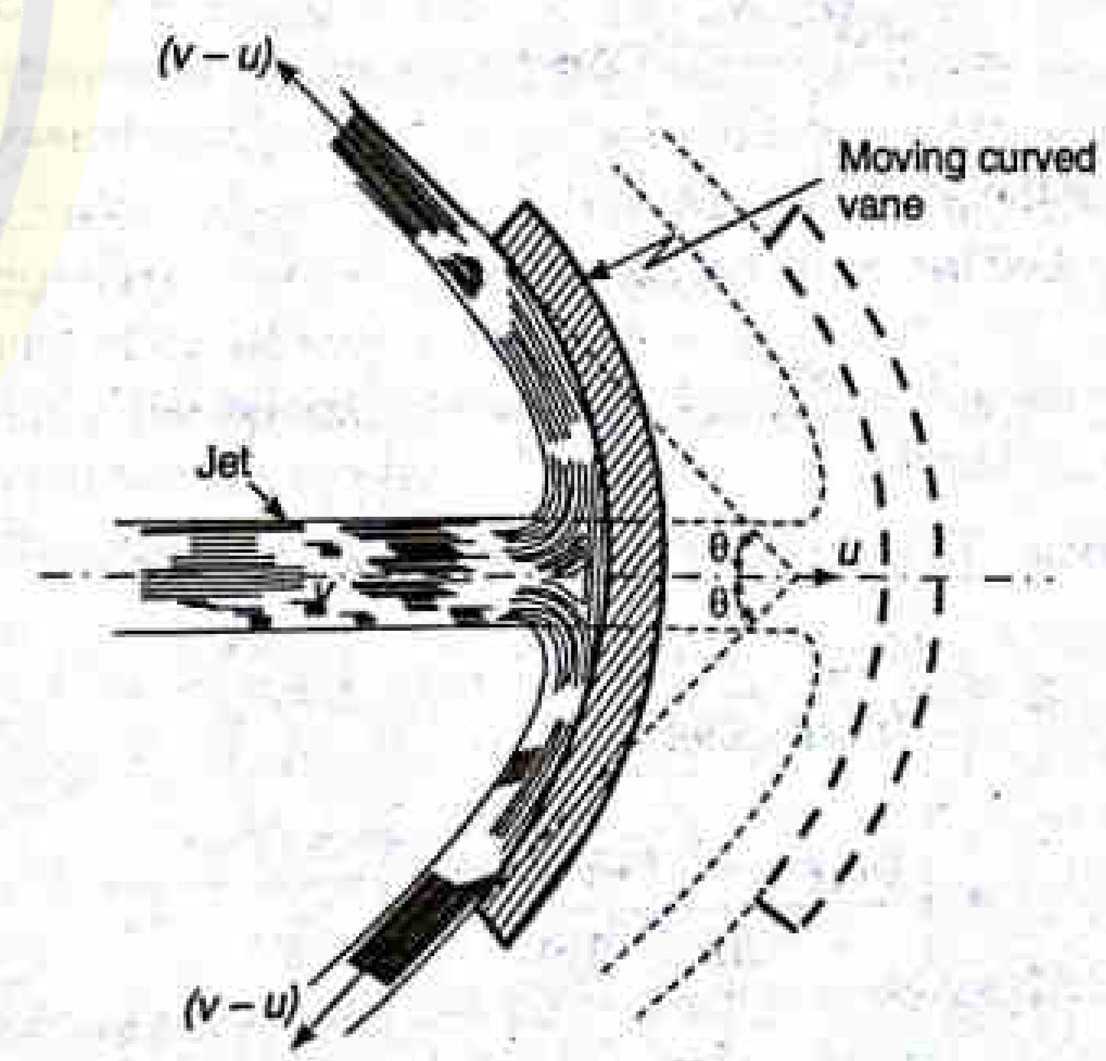


Figure 20.7 *Jet striking moving symmetrical curved vane at its centre*

Further in accordance with the assumptions stated earlier the jet leaving the vane will have its velocity equal to the effective velocity of the incoming jet i.e., $(V-u)$. Thus the component of the velocity of the jet leaving the vane in the direction of the incoming jet is $[(V-u)\cos(180-\theta)]$ or $[-(V-u)\cos\theta]$. Therefore by applying the impulse-momentum equation the force exerted by the jet on the plate in the direction of flow of incoming jet is given by

$$F = \frac{wa(V-u)}{g}[(V-u)-(V-u)\cos(180°-\theta)]$$

or
$$F = \frac{wa(V-u)^2}{g}(1+\cos\theta) \quad \ldots(20.23)$$

When $\theta = 90°$ (or $180° - \theta = 90°$), the above obtained expressions is reduced to

$$F = \frac{wa(V-u)^2}{g} \quad \ldots(20.24)$$

which is the expression for the force exerted by a jet on a moving flat plate as derived in Sec. 20.3 (a). A comparison of Eqs. 20.23 and 20.24 indicates that the force exerted by a jet on a moving curved vane is greater than that on a moving flat plate.

When $\theta = 0$ (for $180° - \theta = 180$), the vane becomes semicircular and the expression for the force exerted by the jet in its direction of flow becomes

$$F = \frac{2wa(V-u)^2}{g} \quad \ldots(20.25)$$

The work done by the jet on the vane per second

$$= (F \times u)$$

$$= \frac{wa(V-u)^2}{g}(1+\cos\theta)\,u$$

which will be the output of the jet in this case.

The kinetic energy supplied by the jet per second

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2 = \frac{waV^3}{2g}$$

which will be the input of the jet. Therefore efficiency of the jet is

$$\eta = \frac{\text{Output of jet}}{\text{Input of jet}}$$

$$= \left[\frac{wa(V-u)^2(1+\cos\theta)u/g}{(waV^3/2g)}\right]$$

$$= \frac{2u(V-u)^2(1+\cos\theta)}{V^3}$$

For a jet of constant velocity V, striking a vane of constant angle θ, the efficiency is maximum if

$$(d\eta/du) = 0.$$

Thus
$$\frac{d\eta}{du} = \frac{2(V^2-4uV+3u^2)(1+\cos\theta)}{V^3} = 0$$

Since $\frac{2(1+\cos\theta)}{V^3} \neq 0$ being constant for a given system,

$$V^2 - 4uV + 3u^2 = 0$$

or
$$(V-u)(V-3u) = 0$$

i.e.,
$$V = u \text{ or } V = 3u$$

Evidently if $V = u$ the work done by the jet will be equal to zero, hence for maximum η, $V = 3u$ and then

$$\eta_{max} = \frac{2u[(3u)^2 - 2u(3u) + u^2](1+\cos\theta)}{(3u)^3}$$

or
$$\eta_{max} = \frac{8}{27}(1+\cos\theta) = \frac{16}{27}\cos^2\frac{\theta}{2}$$

For a semicircular vane since $\theta = 0$ the maximum efficiency becomes $\eta_{max} = 0.592$ which is slightly less than 0.6.

Obviously the case discussed above will not be practically feasible because the distance between the vane and the nozzle issuing the jet will be constantly increasing at the rate of u per second, which would required a continuous lengthening of the jet. This difficulty may however be overcome if instead of a single curved vane, there is a series of such vanes arranged at equal spacing on the periphery of a large wheel which is capable of rotating in a vertical plane. Thus as the wheel rotates, each vane will become normal to the jet in turn, so that the entire fluid of the incoming jet issued from the nozzle will be utilized in striking the vanes. Thus for a jet of cross-sectional area a and velocity V, the mass of water striking the wheel per second is $m = (\rho aV) = (waV/g)$.

Further if u is the tangential velocity of the wheel along the centre of the vanes, the effective velocity of the jet before impact will be the relative velocity of the jet with respect to that of the vane, i.e., $(V-u)$. Since the velocity of the jet leaving the vane is also equal to $(V-u)$ and it leaves the vane tangentially, the component of the velocity of the leaving jet in the direction of the incoming jet will be $[(V-u)\cos(180°-\theta)]$. Therefore by applying the impulse-momentum equation the force exerted by the jet on the vanes in the direction of flow of jet is given by

$$F = \frac{waV}{g}[(V-u)-(V-u)\cos(180°-\theta)]$$

or
$$F = \frac{waV(V-u)}{g}(1+\cos\theta) \quad \ldots(20.26)$$

The work done on the wheel per second will be the output of the jet. Thus output of the jet

$$= (F \times u)$$

$$= \frac{waV(V-u)}{g}(1+\cos\theta)\,u$$

Again the kinetic energy of the jet per second will be the input of the jet. Thus input of the jet

$$= \frac{1}{2}\frac{waV}{g}(V^2) = \frac{waV^3}{2g}$$

Therefore the efficiency of the wheel is

$$\eta = \frac{\text{Work done per second}}{\text{Kinetic energy of jet per second}}$$

or

$$\eta = \frac{2u(V-u)(1+\cos\theta)}{V^2}$$

For a jet of constant velocity V, striking a given wheel, the efficiency will be maximum when $(d\eta/du) = 0$. Thus

$$\frac{d\eta}{du} = \frac{2(V-2u)(1+\cos\theta)}{V^2} = 0$$

Since $\frac{2(1+\cos\theta)}{V^2} \neq 0$, being constant for a given system

$$V - 2u = 0, \quad \text{or } V = 2u.$$

Hence for maximum efficiency $V = 2u$ and by substitution the value of maximum efficiency is obtained as

$$\eta_{max} = \frac{2u(2u-u)(1+\cos\theta)}{(2u)^2}$$

or

$$\eta_{max} = \frac{1}{2}(1+\cos\theta)$$

For $\theta = 90°$, the curved vanes are reduced to flat plates mounted on the periphery of a wheel. In this case $\eta_{max} = 0.5$ or 50%, which is same as obtained previously.

For $\theta = 0$ the curved vanes will become semicircular. Then the value of $\eta_{max} = 1$ or 100%. This is however a theoretical value of maximum efficiency for a wheel provided with semicircular curved vanes mounted on its periphery. Similar is the principle employed in the case of bucket of a Pelton wheel. The bucket of a Pelton wheel is double semi-ellipsoidal in shape. The jet of water impinges at the centre of the bucket and deflects through 160° to 170° depending upon the particular design.

In the above derived expressions it has been assumed that no loss of energy either due to impact or due to frictional resistance takes places as the jet strikes the vane and it glides over the same. However, if some loss of energy occurs, the relative velocity at the outlet end (or tip) of the vane will be less than that at the inlet end. Thus, if the relative velocity at the outlet end is $k(V-u)$, where k is a coefficient having a value less than 1, Eq. 20.23 becomes

$$F = \frac{wa(V-u)^2}{g}(1+k\cos\theta) \quad \text{...(20.27)}$$

Similarly Eq. 20.26 may also be written as

$$F = \frac{waV(V-u)}{g}(1+k\cos\theta) \quad \text{...(20.28)}$$

The other expressions may also be modified accordingly.

(b) Jet Striking an Unsymmetrical Moving Curved Vane Tangentially at One of the Tips.
Let the curved vane of section 20.4 (b) be moving with a velocity u in the x-direction as shown

in Fig. 20.8 when a jet of cross-sectional area a and velocity V impinges on the vane as before. Since in this case the vane is also moving, as stated earlier, the effective velocity of the jet entering the vane will be equal to the relative velocity of the jet with respect to the vane. Thus, if V_r represents the relative velocity between the jet and the vane at the entrance, it is the velocity with which the jet actually enters the vane at its inlet tip and its magnitude may be found by subtracting the vectors V and u. Referring to Fig. 20.8 draw ab to represent the velocity V in magnitude and direction to some convenient scale. Next draw ac to represent the velocity u in magnitude and direction to the same

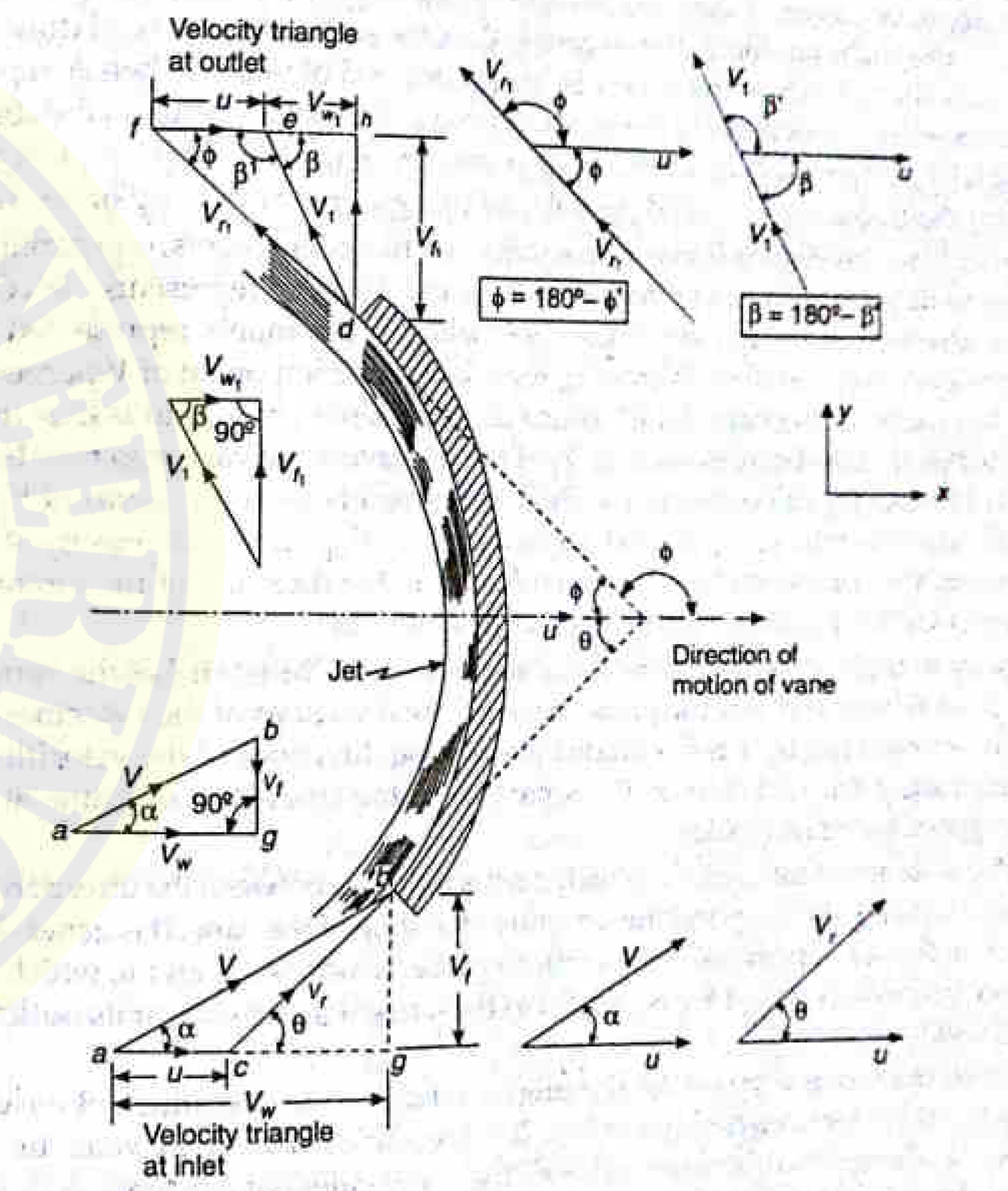


Figure 20.8 *Jet striking tangentially at one of the tips of a moving curved vane*

scale. Then cb represents the relative velocity V_r between the jet and the vane in magnitude and direction to the same scale. In order that the jet may enter the vane smoothly without shock, cb must be parallel to the tangent to the vane at its inlet tip. The triangle abc is known as velocity triangle at inlet tip, in which the absolute velocity of the jet V is the resultant of the velocity of the vane u and the relative velocity V_r.

Since the jet is assumed to impinge on the vane tangentially, there is no loss of energy because of impact. Further the friction loss of the flow passing along the smooth surface of the vane is almost equal to zero, after entry the jet will glide on the inside surface of the vane with velocity V_r. If the small difference in elevation between the two ends of the vane is neglected. Thus the jet will leave the vane at the outlet tip d, with relative velocity V_{r1} which will be equal to V_r. However, if the loss of energy due to friction cannot be neglected, V_{r1} will be less than V_r. The absolute velocity of the leaving jet may be found by drawing the triangle of velocities outlet tip of the vane as indicated below.

Referring to Fig. 20.8, draw df to represent the relative velocity V_{r1} in magnitude and direction to some convenient scale. Again if the jet is to leave the vane without shock (or without causing impact loss), V_{r1} should be parallel to the tangent to the vane at its outlet tip. Next draw fe to represent to the same scale, the velocity of the vane u in magnitude and direction. Then de represents (to the same scale) the absolute velocity V_1 of the jet coming out at the outlet tip. Again the absolute velocity of the leaving jet V_1 is the resultant of the relative velocity of the leaving jet V_{r1} and the velocity of the vane u; and the triangle def is called the velocity triangle at the outlet tip of the vane.

The velocity of the entering jet V may be resolved into two components, one parallel to and other perpendicular to the direction of the motion of the vane. Thus, ag represents the component of V parallel to the direction of the motion of the vane, which is commonly represented by V_w, and is known as velocity of whirl at inlet. Further bg represents the component of V perpendicular to the direction of the motion of the vane, which is usually represented by V_f and is known as velocity of flow at inlet. Similarly the absolute velocity V_1 of the jet leaving the vane also may be resolved into two components along the same directions. Thus, eh represents the components of V_1 parallel to the direction of the motion of the vane, which is represented by V_{w1}, known as velocity of whirl at outlet and dh represents the component of V_1 perpendicular to the direction of the motion of the vane which is represented by V_{f1}, known as velocity of flow at outlet.

In the velocity triangle abc at inlet $\angle bac$ is the angle at which the jet strikes the vane. This angle is usually represented by α and it is the angle between the directions of the velocities V and u. Also $\angle bcg$ generally represented by θ is the angle between the directions of the velocities V_r and u. As indicated earlier since the direction of V_r is parallel to the tangent to the vane at its inlet tip, θ denotes the angle of the vane at inlet.

Similarly in the velocity triangle def at outlet, $\angle deh$ is the angle between the direction of the absolute velocity of the jet leaving the vane and the direction of motion of the vane. It is generally represented by β. Also $\angle dfe$ is the angle between the directions of the velocities V_{r1} and u, which is represented by ϕ. Again, since the direction of V_{r1} is parallel to the tangent to the vane at its outlet tip, ϕ denotes the angle of the vane at outlet.

Actually from the usual convention the angles measured in the anticlockwise direction are considered as positive. Hence as shown in Fig. 20.8 the angle between the velocities u and V_1 is β', where $\beta = (180° - \beta')$ but for the sake of convenience the angle between these velocities is considered as β, where $\beta = (180° - \beta')$. Similarly the angle between the velocities u and V_{r1} is ϕ' but again the angle of the vane at outlet tip is considered as ϕ, where $\phi = (180° - \phi')$.

This problem may also be analyzed by applying the principle of relative motion to the whole system. For this the moving vane is brought to stationary state by applying a velocity equal in magnitude to that of the vane but in opposite direction, i.e., $-u$ on the whole system. This will reduce the problem to that of a stationary curved vane subjected to an impinging action of a jet entering tangentially at one of the tips with an effective velocity equal to V_r and leaving tangentially at the other tip with an effective velocity V_{r1}.

Thus the mass of fluid striking the vane per second is

$$m = (\rho a V_r) = (w a V_r / g) = (W/g)$$

The component of velocity V_r at inlet in the direction of motion of the vane

$$= (V_r \cos\theta) = (V_w - u)$$ [From Δabg]

Similarly the component of velocity V_{r1} at outlet in the direction of the motion of the vane

$$= (-V_{r1} \cos\phi) = -(u + V_{w1})$$ [From Δdfh]

The negative sign has been introduced in the above expression because the direction of this component is opposite to that of the motion of the vane. Moreover this expression will hold good as long as β is an acute angle. However, when $\beta = 90°$, $V_{w1} = 0$, in which case $(-V_{r1} \cos\phi) = -(u)$; and when β is an obtuse angle

$$(-V_{r1} \cos\phi) = -(u - V_{w1}).$$

Therefore by applying the impulse-momentum equation the force exerted on the vane in the direction of motion of the vane is obtained for different values of β as noted below.

When β is an acute angle

$$F_x = \frac{waV_r}{g}[(V_w - u) - \{-(u + V_{w1})\}]$$

or

$$F_x = \frac{waV_r}{g}(V_w + V_{w1}) \quad \text{...(20.29)}$$

when $\beta = 90°$

$$F_x = \frac{waV_r}{g}[(V_w - u) - (-u)]$$

or

$$F_x = \frac{waV_r}{g}(V_w) \quad \text{...(20.30)}$$

When β is an obtuse angle

$$F_x = \frac{waV_r}{g}[(V_w - u) - \{-(u - V_{w1})\}]$$

or

$$F_x = \frac{waV_r}{g}(V_w - V_{w1}) \quad \text{...(20.31)}$$

Therefore, the general expression for F_x becomes

$$F_x = \frac{waV_r}{g}(V_w \pm V_{w1}) \quad \text{...(20.32)}$$

The work done per second by the jet on the vane

$$= (F_x \times u) = \frac{waV_r}{g}(V_w \pm V_{w1})\, u \quad \text{...(20.33)}$$

The work done per second is also equal to the change in kinetic energy of the jet per second.

Hence, the work done per second

$$= \left[\frac{1}{2}\frac{waV_r}{g}(V^2) - \frac{1}{2}\frac{waV_r}{g}(V_1^2)\right]$$

$$= \left(\frac{waV_r}{2g}\right)(V^2 - V_1^2) \quad ...(20.34)$$

Equating the work done represented by Eqs. 20.33 and 20.34, we obtain

$$\frac{waV_r}{g}(V_w \pm V_{w1})\,u = \frac{waV_r}{2g}(V^2 - V_1^2)$$

or

$$2(V_w \pm V_{w1})u = (V^2 - V_1^2)$$

It may, however, be stated that the above expression will hold good only if $V_r = V_{r1}$ (i.e., when there is no loss of energy).

The work done by the jet on the vane will be the output of the jet, and the initial kinetic energy of the jet will be the input. Thus, the efficiency of the jet η may be expressed as

$$\eta = \frac{\left(\frac{waV_r}{g}\right)(V_w \pm V_{w1})u}{\frac{1}{2}\left(\frac{waV}{g}\right)(V^2)} \quad ...(20.35)$$

Similarly the efficiency of the jet may also be expressed as

$$\eta = \frac{\frac{1}{2}\left(\frac{waV_r}{g}\right)(V^2 - V_1^2)}{\frac{1}{2}\left(\frac{waV}{g}\right)V^2} \quad ...(20.36)$$

As explained earlier the case discussed above is also not practically feasible. But the corresponding practical case may be that of a series of such vanes fixed radially to the rim of a wheel. Such types of wheels are usually provided for radial flow hydraulic turbines.

A *radial flow* is one in which a fluid particle during its flow through the vane of a rotating wheel, remains in a plane normal to the axis of rotation and its distance from the axis of rotation is continuously changing. On the other hand in case of *tangential flow*, considered in sections 20.3 and 20.5 ; the distance of a fluid particle from the axis of rotation remains same, as it flows through the vanes of rotating wheel. As such in the case of tangential flow the action of jet on the vanes can be determined by applying impulse-momentum equation as indicated earlier. But in the case of radial flow, since the distance of a fluid particle from the axis of rotation varies along its path of flow, the same has to be taken into account while computing the action of jet on the vanes. Hence instead of finding the force by impulse-momentum equation, the action of jet on the vanes for radial flow is determined by evaluating the torque or turning moment produced on the vanes. The torque may be evaluated on the basis of angular momentum principle indicated in Sec. 8.8 of Chapter 8.

20.6 TORQUE EXERTED ON A WHEEL WITH RADIAL CURVED VANES

Figure 20.9 shows a part of the wheel of a radial flow hydraulic turbine. For the sake of convenience only one vane out of the series of the vanes is shown. Due to the impact of the jet on the vanes, the wheel rotates. Now if R and R_1 are the radii of the wheel at the inlet and the outlet ends of the vane respectively, then since the entire wheel rotates at the same angular velocity ω, the tangential velocities at the inlet and the outlet tips of the vane will be different. Thus if u and u_1 are the tangential velocities at the inlet and the outlet tips of the vane then

$$u = \omega R = \frac{2\pi N}{60}R \text{ and } u_1 = \omega R_1 = \frac{2\pi N}{60}R_1$$

where N represents the speed of the wheel in r.p.m. The velocity triangles at the inlet and the outlet tips of the vane may thus be obtained in the same manner as discussed earlier in section 20.5 (b).

The tangential component or the component of absolute velocity V of the incoming jet in the direction of motion u is

$$V\cos\alpha = V_w$$

Similarly the tangential component of the absolute velocity V_1 of the leaving jet in the direction of motion u_1 is

$$-V_1\cos\beta = (-V_{w1})$$

The negative sign has been introduced because as shown in Fig. 20.9 the jet leaves the vane in the direction opposite to the motion of the wheel.

Further if W is the weight of fluid striking the vanes per second then the mass of fluid striking the vane per second is

$$m = \rho Q = \left(\frac{W}{g}\right)$$

The tangential momentum of the fluid striking the vanes at inlet

$$= \left(\frac{WV_w}{g}\right)$$

As such the angular momentum at inlet

$$= \left(\frac{WV_w}{g}\right)R$$

Similarly the tangential momentum of the fluid at outlet of the vanes

$$= \left(\frac{WV_{w_1}}{g}\right)$$

and hence the angular momentum at outlet

$$= \left(-\frac{WV_{w_1}}{g}\right)R_1$$

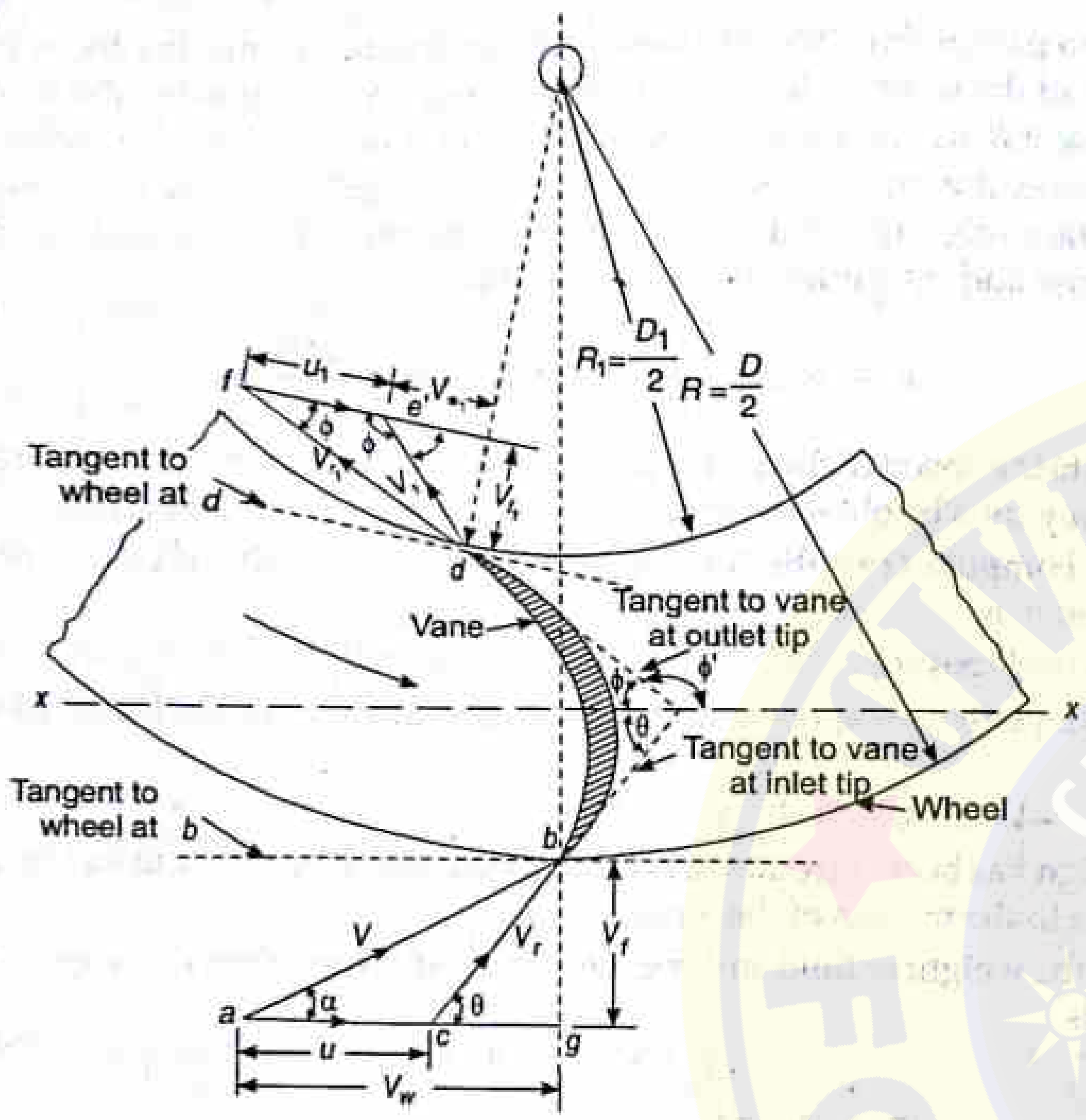


Figure 20.9 *Curved vane mounted radially on a wheel*

Therefore, applying Eq. 8.29 (Chapter 8) the torque T exerted by the fluid on the wheel to revolve it is obtained as

$$T = \left[\frac{WV_w}{g}R - \left(-\frac{WV_{w1}}{g}R_1\right)\right]$$

or

$$T = \frac{W}{g}(V_w R + V_{w1} R_1)$$

Further the work done on the wheel per second or the power delivered to the wheel

$$= \text{Torque} \times \text{angular velocity} = T \times \omega$$

$$= \frac{W}{g}(V_w R + V_{w1} R_1)\,\omega$$

But $u = \omega R$ and $u_1 = \omega R_1$

$\therefore$ Work done on the wheel per second or the power delivered to the wheel

$$= \frac{W}{g}(V_w u + V_{w1} u_1) \qquad \text{...(20.37)}$$

If the jet leaves in the direction as that of the motion of the wheel, V_{w1} will be positive (which corresponds to β being an obtuse angle) and then Eq. 20.37 becomes:

Work done on the wheel per second

$$= \frac{W}{g}(V_w u - V_{w1} u_1) \qquad \text{...(20.38)}$$

By combining Eqs 20.37 and 20.38 the general expression for the work done on the wheel per second may be written as

Work done on wheel per second

$$= \frac{W}{g}(V_w u \pm V_{w1} u_1) \qquad \text{...(20.39)}$$

For radial discharge at the outlet tip of the vane $\beta = 90°$ and $V_{w1} = 0$. Then from Eq. 20.39, work done on wheel per second

$$= \frac{W}{g}(V_w u) \qquad \text{...(20.40)}$$

Further from Eq. 20.39, work done on the wheel per second per unit weight of fluid

$$= \frac{1}{g}(V_w u \pm V_{w1} u_1) \qquad \text{...(20.41)}$$

If there is no loss of energy, the work done on the wheel per second is also equal to the change in kinetic energy of the jet per second. Hence, the work done per second

$$= \frac{W}{2g}(V^2 - V_1^2) \qquad \text{...(20.42)}$$

Equating the work done represented by Eqs 20.39 and 20.42, we obtain

$$\frac{W}{g}(V_w u \pm V_{w1} u_1) = \frac{W}{2g}(V^2 - V_1^2)$$

or

$$2(V_w u \pm V_{w1} u_1) = (V_2^2 - V_1^2) \qquad \text{...(20.43)}$$

The work done by the jet on the wheel will be the output of the system and the initial kinetic energy of the jet will be the input. Thus the efficiency of the system may be expressed as

$$\eta = \frac{W(V_w u \pm V_{w1} u_1)/g}{WV^2/2g}$$

or

$$\eta = \frac{2(V_w u \pm V_{w1} u_1)}{V^2} \qquad \text{...(20.44)}$$

Similarly the efficiency of the system may also be expressed as

$$\eta = \frac{W(V^2 - V_1^2)2g}{WV^2/2g}$$

or

$$\eta = \frac{V^2 - V_1^2}{V^2} = \left[1 - \left(\frac{V_1}{V}\right)^2\right] \quad ...(20.45)$$

It follows from Eq. 20.45 that for a given angle α and the initial jet velocity V, the efficiency will be maximum when V_1 has a minimum value. However, V_1 cannot be zero as in that case the incoming jet will not move out of the vane. Further Eq. 20.44 indicates that the efficiency will be maximum when β is an acute angle so that there will be a positive sign between V_w and V_{w1}. In other words, it means that for the efficiency to be maximum, V_1 should be in the direction opposite to that of u_1. Thus, for maximum efficiency the velocities V_1, V_{r1} and u_1 should be colinear, so that $V_{r1} = (V_1 - u_1)$ and V_{r1} is in the direction of V_1. But this condition requires that $\phi = 0$ and $V_{w1} = V_1$, which is, however, not possible to accomplish in actual practice, because in practice ϕ will have some value other than zero. Hence, the above indicated condition for maximum efficiency is only theoretical, but it may, however, be stated that smaller is the value of ϕ, higher will be the efficiency.

ILLUSTRATIVE EXAMPLES

Example 20.1. *A jet of water 75 mm diameter having a velocity of 20 m/s, strikes normally a flat smooth plate. Determine the thrust on the plate (a) if the plate is at rest, (b) if the plate is moving in the same direction as the jet with a velocity of 5 m/s. Also find the work done per second on the plate in each case and the efficiency of the jet when the plate is moving.*

Solution

(a) From Eq. 20.1 the normal thrust on a stationary flat plate is given as

$$F = \left(\frac{waV^2}{g}\right)$$

Area of jet $\quad a = \frac{\pi}{4}\left(\frac{75}{1000}\right)^2 = 0.004\,42 \text{ m}^2$

Velocity of jet $V = 20$ m/s

$$\therefore \quad F = \frac{9810 \times (0.00442) \times (20)^2}{9.81}$$

$$= 1\,768 \text{ N}$$

Since in this case plate is at rest, $u = 0$

$\therefore$ Work done $= (F \times u) = 0$

(b) From Eq. 20.7 the normal thrust on a moving flat plate is given as

$$F = \frac{wa(V-u)^2}{g}$$

$$= \frac{9810 \times (0.00442) \times (20-5)^2}{9.81} = 994.5 \text{ N}$$

Work done per second by the jet on the plate

$= (F \times u)$

$= (994.5 \times 5) = 4\,972.5$ N.m

Thus in this case, output

$= 4\,972.5$ N. m/s

Input $=$ Kinetic energy of the issuing jet

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

$$= \frac{1}{2}\left(\frac{9810 \times 0.00442}{9.81}\right)(20)^3$$

$$= 17\,680 \text{ N.m/s}$$

$$\therefore \quad \eta \text{ of jet} = \frac{4972.5}{17680} \times 100 = 28.13\%$$

Example 20.2. *A 50 mm diameter jet having a velocity of 25 m/s, strikes a flat plate, the normal of which is inclined at 30° to the axis of the jet. Calculate the normal force exerted on the plate (a) when the plate is stationary, (b) when the plate is moving with a velocity of 10 m/s in the direction of the jet, (c) when the plate is moving with a velocity of 8 m/s parallel to itself and in the direction of the normal to its surface. Find also the work done and the efficiency of the jet when the plate is moving.*

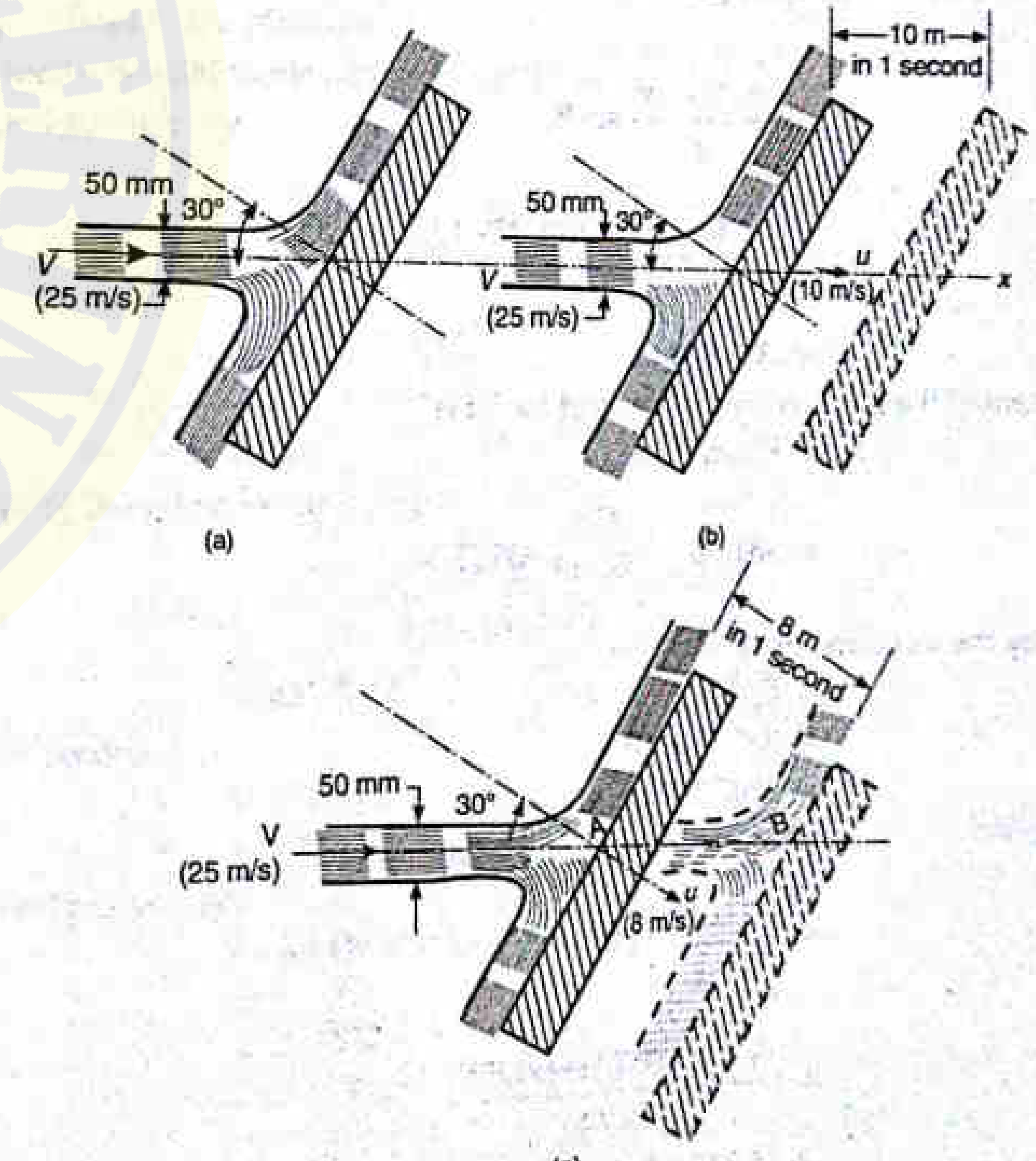


Figure Ex. 20.2

Solution

(a) From Eq. 20.2 the normal force on an inclined stationary flat plate is given as

$$F = \left(\frac{waV^2}{g}\right)\sin\theta$$

Area of jet $a = \frac{\pi}{4}\left(\frac{50}{1000}\right)^2 = 0.00196\ \text{m}^2$

Velocity of jet $V = 25\ \text{m/s}$

Angle of inclination

$$\theta = (90° - 30°) = 60°$$

Thus $F = \frac{9810\times(0.00196)\times(25)^2}{9.81}\times\left(\frac{\sqrt{3}}{2}\right)$

$$= 1060.88\ \text{N}$$

(b) From Eq. 20.10 for an inclined plate moving in the same direction as the jet, the normal force is given as

$$F = \frac{wa(V-u)^2}{g}\sin\theta$$

$$= \frac{9810\times(0.00196)\times(25-10)^2}{9.81}\times\left(\frac{\sqrt{3}}{2}\right)$$

$$= 381.92\ \text{N}$$

The component of the force in the direction of the jet is

$$F_x = F(\sin\theta)$$

$$= 381.92\times\left(\frac{\sqrt{3}}{2}\right) = 330.75\ \text{N}$$

Work done by the jet on the plate

$$= F_x \times u$$
$$= 330.75\times 10$$
$$= 3307.5\ \text{N.m/s}$$

Energy supplied = kinetic energy of the issuing jet

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

$$= \frac{9810\times(0.00196)\times(25)^3}{2\times 9.81}$$

$$= 15\,312.5\ \text{N.m/s}$$

$\therefore$ η of the jet $= \frac{3307.5}{15312.5}\times 100 = 21.6\%$

(c) Figure Ex. 20.2 shows the arrangement of the plate for this case.

Mass of fluid issued by the jet per second

$$= \frac{waV}{g}$$

Mass of fluid used to extend the jet per second

$$= \frac{wa(AB)}{g} = \frac{wa}{g}\left(\frac{u}{\cos\theta}\right)$$

$\therefore$ Mass of fluid actually striking the plate per second

$$= \frac{wa}{g}\left(V - \frac{u}{\cos\theta}\right)$$

Initial component of velocity normal to the plate = $V\cos\theta$

Final velocity normal to the plate = u

Change of velocity normal to the plate = $(V\cos\theta - u)$

$\therefore$ Force normal to the plate

$$F = \frac{wa}{g}\left(V - \frac{u}{\cos\theta}\right)(V\cos\theta - u)$$

or

$$F = \frac{wa(V\cos\theta - u)^2}{g\cos\theta}$$

In this case $\theta = 30°$

Thus substituting the given values, we get

$$F = \frac{9810\times(0.00196)(25\cos 30° - 8)^2}{9.81\times(\cos 30°)}$$

$$= 421.73\ \text{N}$$

Work done per second

$$= (F\times u)$$
$$= (421.73\times 8) = 3373.84\ \text{N.m}$$

Energy supplied per second

$$= \text{kinetic energy of the jet}$$

$$= \frac{1}{2}\left(\frac{waV}{g}\right)V^2$$

$$= \frac{9810\times(0.00196)\times(25)^3}{2\times 9.81}$$

$$= 15\,312.5 \text{ N.m}$$

$$\therefore \quad \eta \text{ of the jet } = \frac{3373.84}{15312.5} = 22.03\ \%$$

Note. The problem can also be solved by adopting the principle of relative motion, in which case a velocity equal to ($u/\cos\theta$) may be applied to the entire system in the direction opposite to that of the jet so that the plate becomes stationary.

Example 20.3. *A jet of water 75 mm in diameter having velocity of 20 m/s strikes a series of the flat plates arranged around the periphery of a wheel such that each plate appears successively before the jet. If the plates are moving at a velocity of 5 m/s, compute the force exerted by the jet on the plate, the work done per second on the plate and the efficiency of the jet.*

Solution

From Eq. 20.8 the force exerted on the plate is given as

$$F = \frac{waV}{g}(V-u)$$

Area of jet $\quad a = \frac{\pi}{4}\times\left(\frac{75}{1000}\right)^2 = 0.00442 \text{ m}^2$

Velocity of jet $\quad V = 20 \text{ m/s}$

Thus $\quad F = \frac{9810\times(0.00442)\times 20\times(20-5)}{9.81}$

$$= 1\,326 \text{ N}$$

Work done $\quad = (F\times u)$

$$= (1\,326\times 5) = 6\,630 \text{ N.m/s}$$

From Eq. 20.9 the efficiency of the jet is given as

$$\eta = \frac{2u(V-u)}{V^2}$$

$$= \frac{2\times5(20-5)}{(20)^2}\times 100 = 37.5\%$$

Example 20.4. *A metal plate of 10 mm thickness and 200 mm square is hung so that it can swing freely about the upper horizontal edge. A horizontal jet of water of 20 mm diameter impinges with its axis perpendicular and 50 mm below the edge of the hinge, and keeps it steadily inclined at 30° to the vertical. Find the velocity of the jet if the specific weight of the metal is 75.54 kN/m³.*

Solution

Weight of the plate

$$= \frac{(10\times200\times200)\times75.54\times10^3}{(1000)^3}$$

$$= 30.22 \text{ N}$$

Area of the jet,

$$a = \frac{\pi}{4}\times\left(\frac{20}{1000}\right)^2$$

$$= 0.000\,314 \text{ m}^2$$

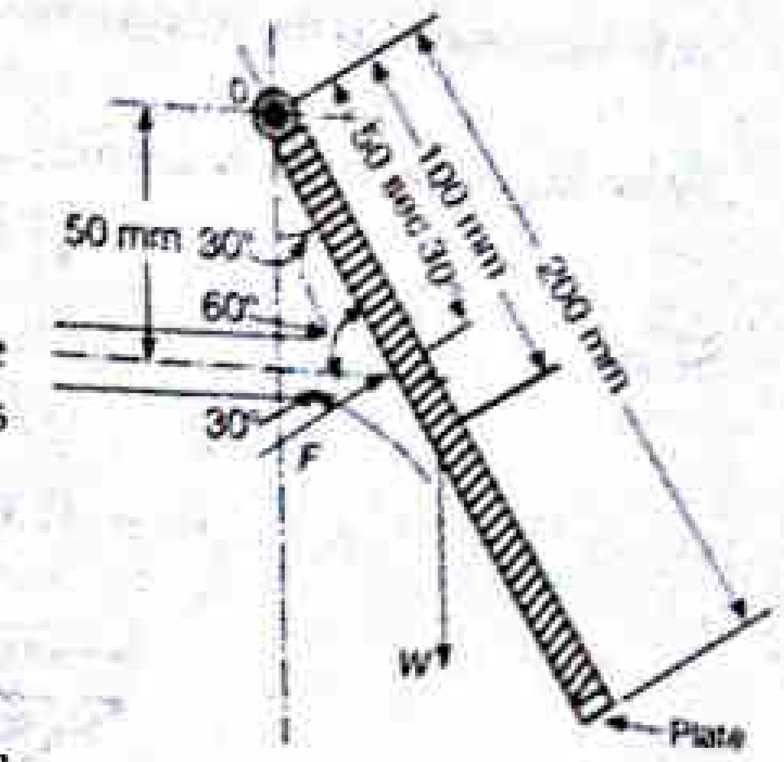


Figure Ex. 20.4

Let the velocity of the jet be V m/s, then from Eq. 20.2 the force exerted by the jet in the direction normal to the plate is given as

$$F = \left(\frac{waV^2}{g}\right)\sin\theta$$

As shown in Fig. Ex. 20.4 $\theta = 60°$, thus by substitution, we get

$$F = \frac{9810\times(0.000314)\times(\sqrt{3})}{9.81\times2}V^2$$

$$= 0.272\ V^2$$

Taking moments of all the forces about the hinge 0, we have

$$30.22\times\left(\frac{100}{1000}\right)(\sin 30°) = 0.272\ V^2\left(\frac{50}{1000}\right)(\sec 30°)$$

$\therefore$ Velocity of jet $\quad V = 9.81 \text{ m/s}$

Example 20.5. *A jet of water moving at 20 m/s impinges on a symmetrical curved vane shaped to deflect the jet through 120° (that is the vane angles θ and ϕ are each equal to 30°). If the vane is moving at 5 m/s, find the angle of the jet so that there is no shock at inlet. Also determine the absolute velocity of exit in magnitude and direction, and the work done.*

Solution

From the velocity triangle at inlet, as shown in Fig. Ex. 20.5, applying the sine rule, we have

$$\frac{V}{\sin(180°-30°)} = \frac{u}{\sin(30°-\alpha)} = \frac{V_r}{\sin\alpha}$$

Also $\quad V = 20 \text{ m/s}, u = 5 \text{ m/s}$

Thus considering the first two terms of the above equation, we have

$$\frac{20}{\sin 150°} = \frac{5}{\sin(30°-\alpha)}$$

or $\quad \sin(30°-\alpha) = 0.125$

or $\quad \alpha = 22°49'$

Now considering the first and the third terms of the above equation, we have

$$\frac{20}{\sin 150°} = \frac{V_r}{\sin(22°49')}$$

$$\therefore \quad V_r = 15.51 \text{ m/s}$$

It is assumed that the relative velocity of flow remains constant, that is,

$$V_{r1} = V_r = 15.51 \text{ m/s}$$

From the outlet velocity triangle applying the cosine rule, we have

$$V_1^2 = u_1^2 + V_{r1}^2 - 2u_1 V_{r1} (\cos 30°)$$

Since $u_1 = u = 5$ m/s

$$V_1^2 = (5)^2 + (15.51)^2 - 2(5 \times 15.51)(0.866)$$

$\therefore$ $V_1 = 11.46$ m/s

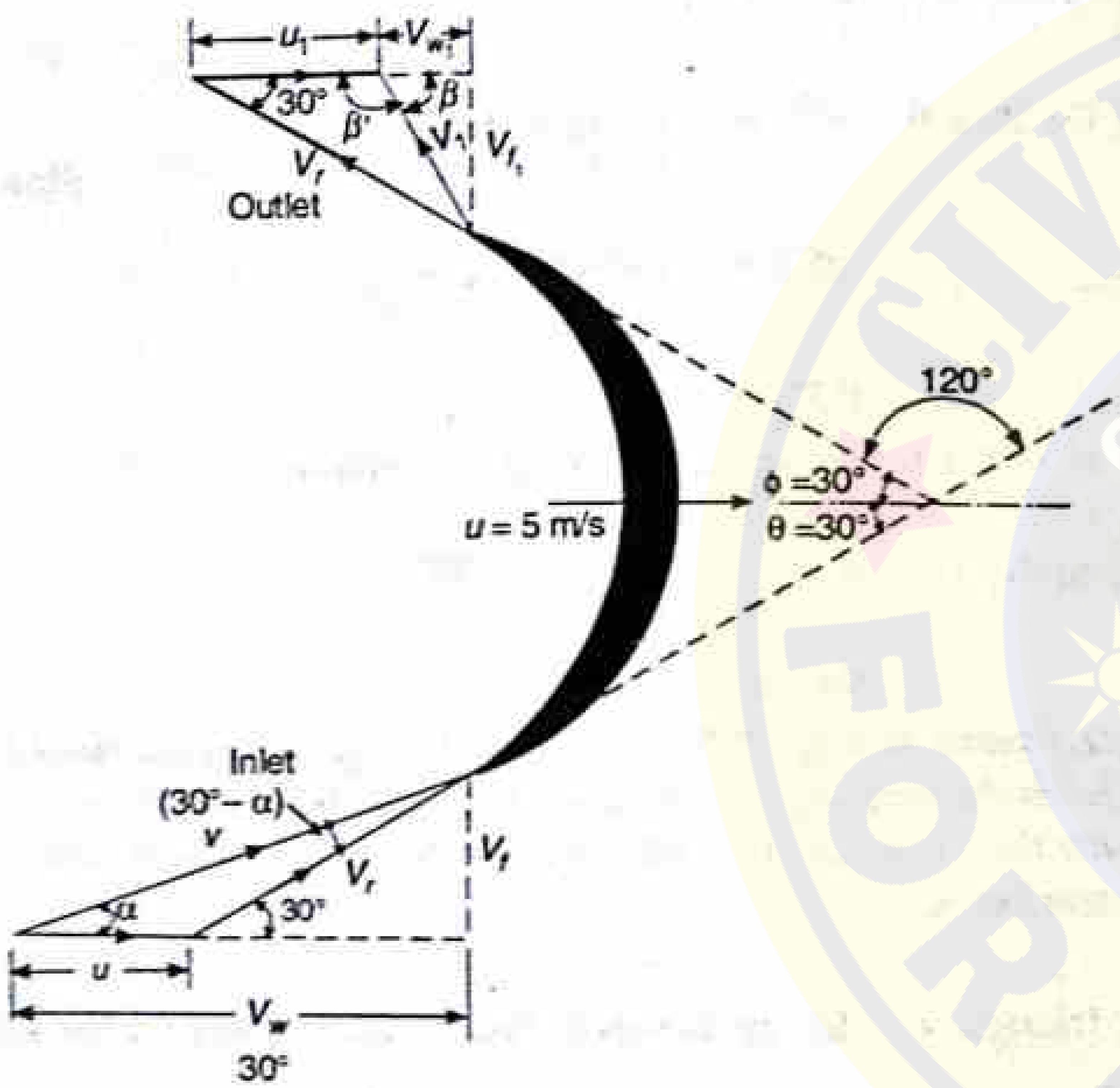


Figure Ex. 20.5

Again from the outlet velocity triangle

$$\tan \beta = \frac{V_{r1} (\sin 30°)}{V_{r1} (\cos 30°) - u_1}$$

or

$$\tan \beta = \frac{15.51 \times 0.5}{(15.51 \times 0.866) - 5} = 0.9197$$

$\therefore$ $\beta = 42° 36'$

and $\beta' = (180° - \beta) = 137° 24'$

From Eq. 20.41, work done per unit weight of fluid

$$= \frac{1}{g} [V_{w} \pm V_{w1} u_1]$$

$$V_w = V (\cos \alpha)$$

$$= 20 (\cos 22°49') = 18.44 \text{ m/s}$$

$$V_{w1} = V_1 (\cos \beta)$$

$$= 11.45 (\cos 42°36') = 8.43 \text{ m}$$

Since the direction of V_{w1} is opposite to that of the motion of vane and $u = u_1 = 5$ m/s. Thus work done

$$= \frac{1}{g} [V_w + V_{w1}] u$$

$$= \frac{5}{9.81} [18.44 + 8.43]$$

$$= 13.70 \text{ N.m/N}$$

Example 20.6. *A jet of water 50 mm in diameter having a velocity of 20 m/s, impinges tangentially on a series of vanes, which when stationary deflect the jet through an angle of 120°. Calculate the magnitude of the resultant force on the vanes when they are (a) stationary, (b) moving with a velocity of 10 m/s in the same direction as the jet. Also determine the work done per second and the efficiency in case (b).*

Solution

In this case the vanes may be set in either of the two ways as shown in Fig. Ex. 20.6.

(a) (i) Let the axis of symmetry of the vane be horizontal as shown in Fig. Ex. 20.6 (a). In this case the tangents at inlet and outlet will make an angle of 30° with the horizontal. Then from Eq. 20.19, we have

$$F_x = 2\left(\frac{wav^2}{g}\right) \cos \theta$$

Area of jet

$$a = \frac{\pi}{4}\left(\frac{50}{1000}\right)^2 = 0.00196 \text{ m}^2$$

Thus

$$F_x = \frac{2 \times 9810 \times 0.00196 \times (20)^2}{9.81} \times \left(\frac{\sqrt{3}}{2}\right)$$

$$= 1357.93 \text{ N}$$

Further $F_y = 0$

Thus the resultant force acting on the vane is

$$F = F_x = 1357.93 \text{ N}$$

which is acting in the direction of horizontal axis or it is inclined at an angle of 30° to the direction of the jet.

(ii) Let the tangent at the inlet tip of the vane be horizontal, that is, let the axis of symmetry be inclined at an angle of 30° to the horizontal as shown in Fig. Ex. 20.6 (b).

Thus in this case the tangent at outlet tip of the vane will make an angle of 60° with the horizontal. Hence in this case θ = 0° and φ = 60°.

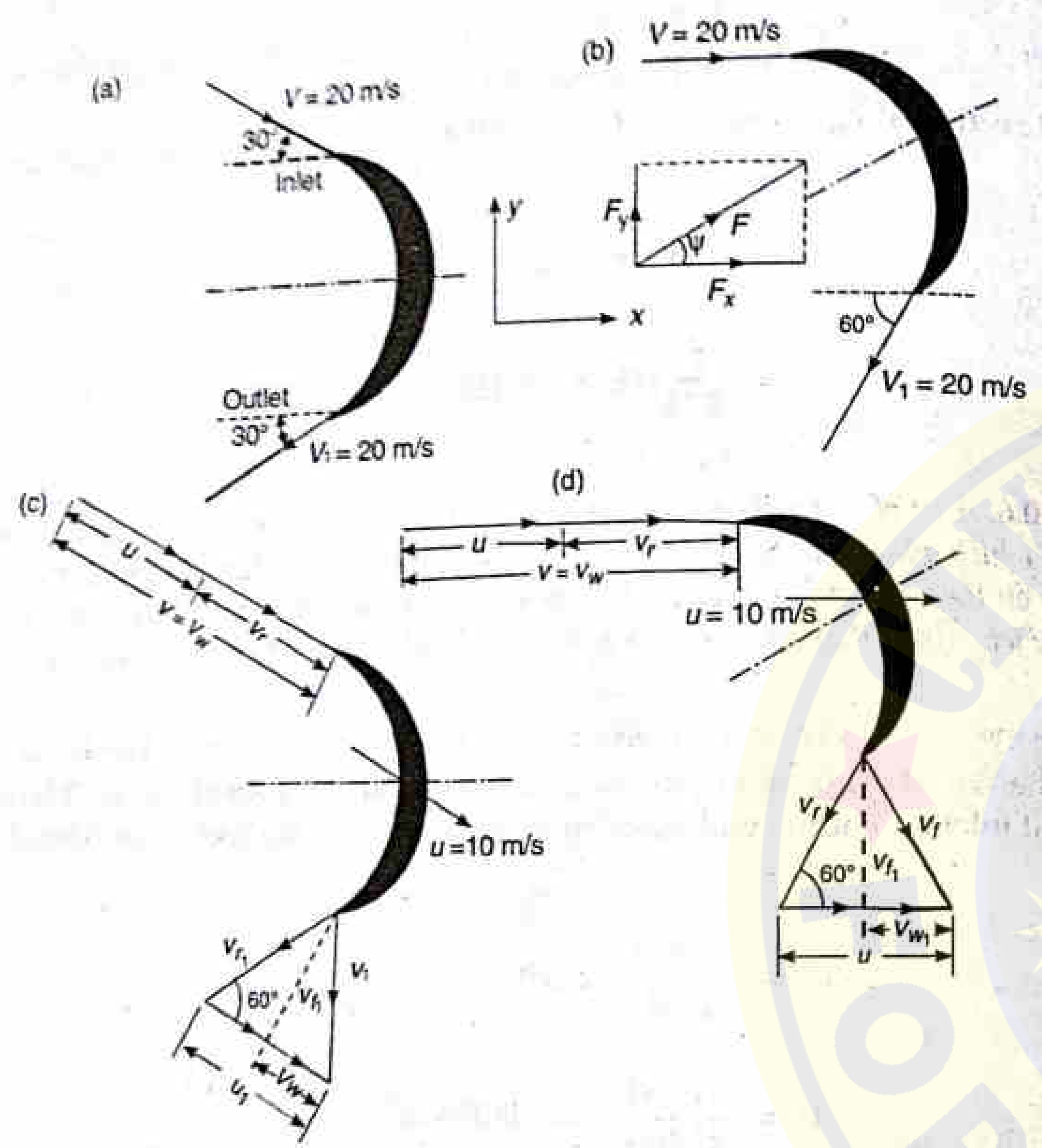


Figure Ex. 20.6

From Eq. 20.16, we have

$$F_x = \frac{waV^2}{g}[\cos\theta + \cos\phi]$$

$$= \frac{9810 \times (0.00196) \times (20)^2}{9.81} \times (1 + \cos 60°)$$

$$= 1176 \text{ N}$$

Again from Eq. 20.17, we have

$$F_y = \frac{waV^2}{g}[\sin\theta - \sin\phi]$$

$$= \frac{waV^2}{g}[0 - \sin 60°]$$

$$= \frac{9810 \times (0.00196) \times (20)^2}{9.81} \times \left(-\frac{\sqrt{3}}{2}\right)$$

$$= -678.96 \text{ N}$$

The negative sign for F_y is, however, not significant and the same may be ignored.

The resultant force exerted by the jet on the vane is

$$F = \sqrt{F_x^2 + F_y^2}$$

$$= \sqrt{(1176)^2 + (678.96)^2} = 1357.93 \text{ N}$$

If ψ is the angle made by the resultant force with the direction of the jet, then

$$\tan\psi = \frac{F_y}{F_x}$$

$$= \frac{678.96}{1176} = 0.5773$$

$$\therefore \quad \psi = 30°$$

(b) When the vanes are moving in the same direction as the jet then the velocity triangles at the inlet and outlet tips of the vanes for both the settings of vanes are shown in Fig. Ex. 20.6 (c) and (d). Since in both the settings of vanes the velocity triangles are exactly same, the case shown in Fig. Ex. 20.6 (d) only is discussed. From the velocity triangle at inlet (which is a straight line), we have

$$V_r = (V - u) = (20 - 10) = 10 \text{ m/s}$$

$$V_w = V = 20 \text{ m/s}$$

Assuming that the relative velocity remains constant, we have

$$V_r = V_{r1} = 10 \text{ m/s}$$

Moreover, at the outlet tip of the vane

$$\phi = 60°, u_1 = u = 10 \text{ m/s}$$

and $\quad V_{r1} = 10$ m/s

hence the outlet velocity triangle is an equilateral triangle.

Thus $\quad V_1 = 10$ m/s

and $\quad V_{w1} = V_1 (\cos 60°) = 5$ m/s

It is assumed that the vanes are so arranged that the vanes appear successively before the jet, hence the mass of water striking the vanes per second

$$= \frac{waV}{g} = \frac{9810 \times (0.00196) \times (20)}{9.81} = 39.2 \text{ kg}$$

Thus the component of the force in the x-direction is

$$F_x = \frac{waV}{g}[V_w - V_{w1}]$$

$$= 39.2\,(20 - 5) = 588 \text{ N}$$

Similarly the component of force in the y-direction is

$$F_y = \frac{waV}{g}[0 - V_1 \sin 60°]$$

$$= 39.2\left(-10 \times \frac{\sqrt{3}}{2}\right) = -339.48 \text{ N}$$

The negative sign for F_y is, however, not significant and the same may be ignored.

The resultant force exerted by the jet on the vane is

$$F = \sqrt{F_x^2 + F_y^2}$$

$$= \sqrt{(588)^2 + (339.48)^2} = 678.96 \text{ N}$$

If ψ is the angle between the resultant force and the direction of jet, then

$$\tan\psi = \frac{F_y}{F_x} = \frac{339.48}{588} = 0.5773$$

$\therefore$ $$\psi = 30°$$

Hence the resultant force is inclined at an angle of 30° to the direction of the jet.

From Eq. 20.39 the work done per second

$$= \frac{waV}{g}(V_w u - V_{w1} u_1)$$

$$= 39.2\,(20 - 5) \times 10 = 5\,880 \text{ N.m}$$

From Eq. 20.44 the efficiency is given as

$$\eta = \frac{2(V_w u - V_{w1} u_1)}{V^2}$$

$$= \frac{2(20-5)\times 10}{(20)^2} \times 100 = 75\%$$

Example 20.7. *A jet of water having a velocity of 45 m/s impinges without shock a series of vanes moving at 15 m/s, the direction of motion of the vanes being inclined at 20° to that of the jet. The relative velocity at outlet is 0.9 of that at inlet, and the absolute velocity of the water at exit is to be normal to motion of the vanes. Find: (a) vane angles at entrance and exit; (b) work done on vanes per unit weight of water supplied by the jet and (c) the hydraulic efficiency.*

Solution

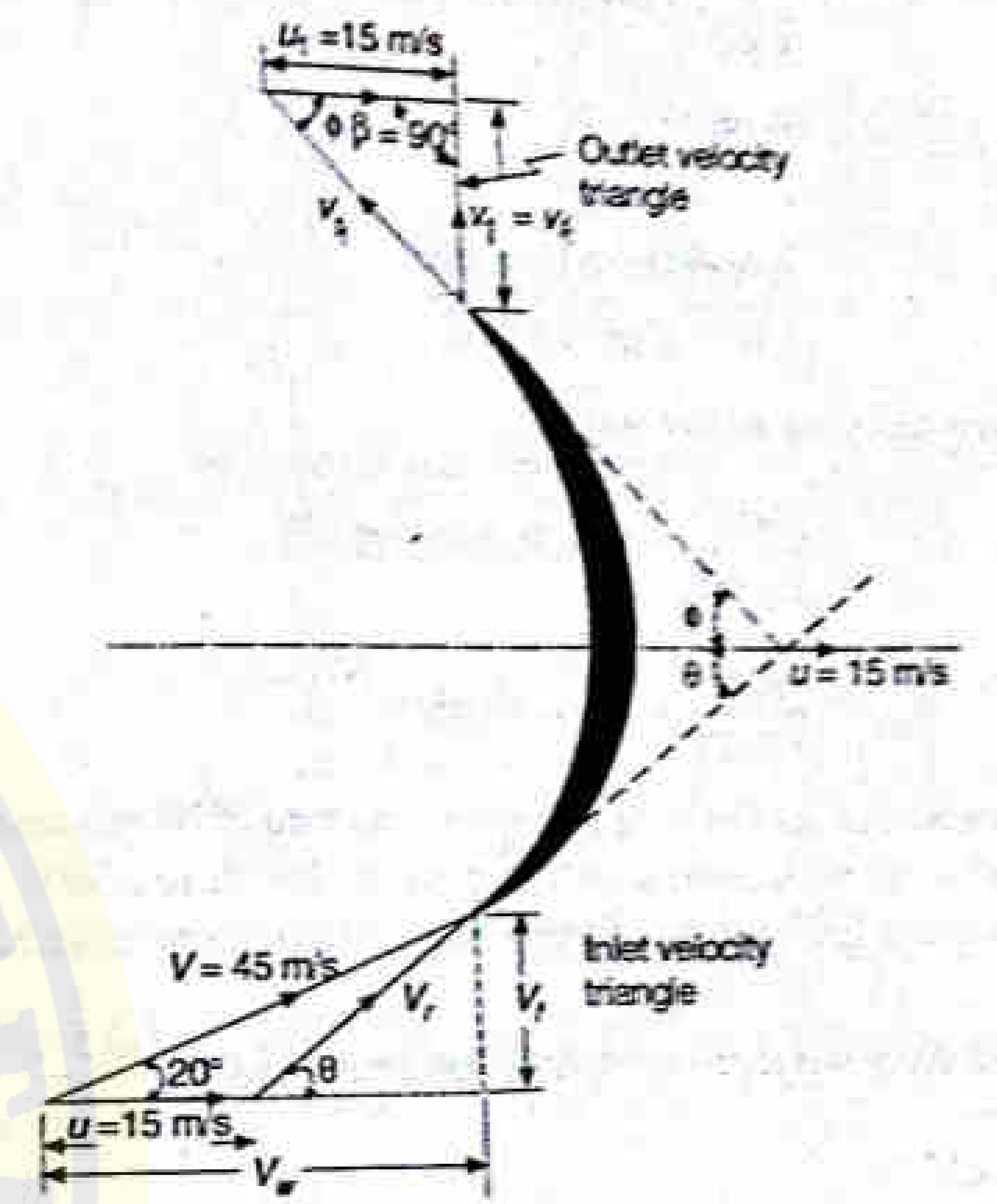


Figure Ex. 20.7

Applying the cosine rule at the inlet velocity triangle, we have

$$V_r^2 = (45)^2 + (15)^2 - 2(45 \times 15)(\cos 20°)$$

$\therefore$ $$V_r = 31.33 \text{ m/s}$$

Thus $$V_{r1} = 0.9\,V_r = (0.9 \times 31.33) = 28.2 \text{ m/s}$$

From inlet velocity triangle

$$\tan\theta = \frac{45(\sin 20°)}{45(\cos 20°) - 15} = 0.5641$$

$\therefore$ $$\theta = 29°\,26'$$

From outlet velocity triangle since

$$\beta = 90°$$

$$\cos\phi = \frac{u_1}{V_{r_1}} = \frac{15}{28.2} = 0.531\,9$$

$\therefore$ $$\phi = 57°\,52'$$

i.e., the vane angles at entrance and exit are 29°26′ and 57°52′ respectively.

Work done on the vane per unit weight of water (since $V_{w1} = 0$)

$$= \frac{1}{g}(V_w u)$$

$$= \frac{1}{9.81} \times (45 \cos 20°) \times 15$$

$$= 64.66 \text{ N.m/N}$$

Hydraulic efficiency

$$\eta_h = \frac{\text{Work done/second}}{\text{K.E of jet/second}}$$

K.E. of jet per unit weight of water per second

$$= \frac{(45)^2}{2 \times 9.81} = 103.21 \text{ N.m/N}$$

$$\therefore \quad \eta_h = \frac{64.66}{103.21} \times 100 = 62.65\ \%$$

Example 20.8. *A jet discharges 0.15 m³/s of water with velocity of 70 m/s impinges without shock on a series of curved vanes which move in the same direction as the jet. The shape of each vane is such that it would deflect the jet through an angle of 150°. Surface friction reduces the relative velocity by 8 percent as the water passes across the vanes and there is a further windage loss equivalent to* $\left(\frac{0.5\, u^2}{2g}\right)$ *N.m per N of water, u being the vane velocity. Find:*

(a) the velocity of the vanes corresponding to maximum efficiency;

(b) the value of this efficiency;

(c) the corresponding force on the vanes in, and at right angles to the direction of their motion ; and

(d) the power of this arrangement.

Solution

Let u be the required velocity of the vanes, then from Fig. Ex. 20.8,

$$V_r = (V - u) = (70 - u)$$

and $$V_{r1} = 0.92\, V_r = 0.92\,(V - u)$$

Further $$V_w = V = 70 \text{ m/s}$$

and $$V_{w1} = [u - V_{r1} (\cos 30°)]$$

$$= \left[u - 0.92(V-u)\left(\sqrt{3}/2\right)\right]$$

$\therefore$ Work done by water per second

$$= \frac{W}{g}[V_w u - V_{w1} u_1]$$

$$= \frac{W}{g}\left[V - \left\{u - 0.92(V-u)\left(\sqrt{3/2}\right)\right\}\right]u$$

$$= \frac{W}{g}[1.797\,(V-u)u\,]$$

Therefore useful work done per second

$$= \frac{W}{g}[1.797(V-u)u] - (0.5)\frac{Wu^2}{2g}$$

Kinetic energy of jet

$$= \frac{WV^2}{2g}$$

$$\therefore \text{ Efficiency} \quad \eta = \frac{\text{Useful work done/second}}{\text{Kinetic energy of jet}}$$

$$= \frac{2[1.797(V-u)u] - 0.5u^2}{V^2}$$

$$= 3.594\left(\frac{u}{V} - \frac{u^2}{V^2}\right) - 0.5\frac{u^2}{V^2}$$

$$= 3.594\left(\frac{u}{V}\right) - 4.094\left(\frac{u}{V}\right)^2$$

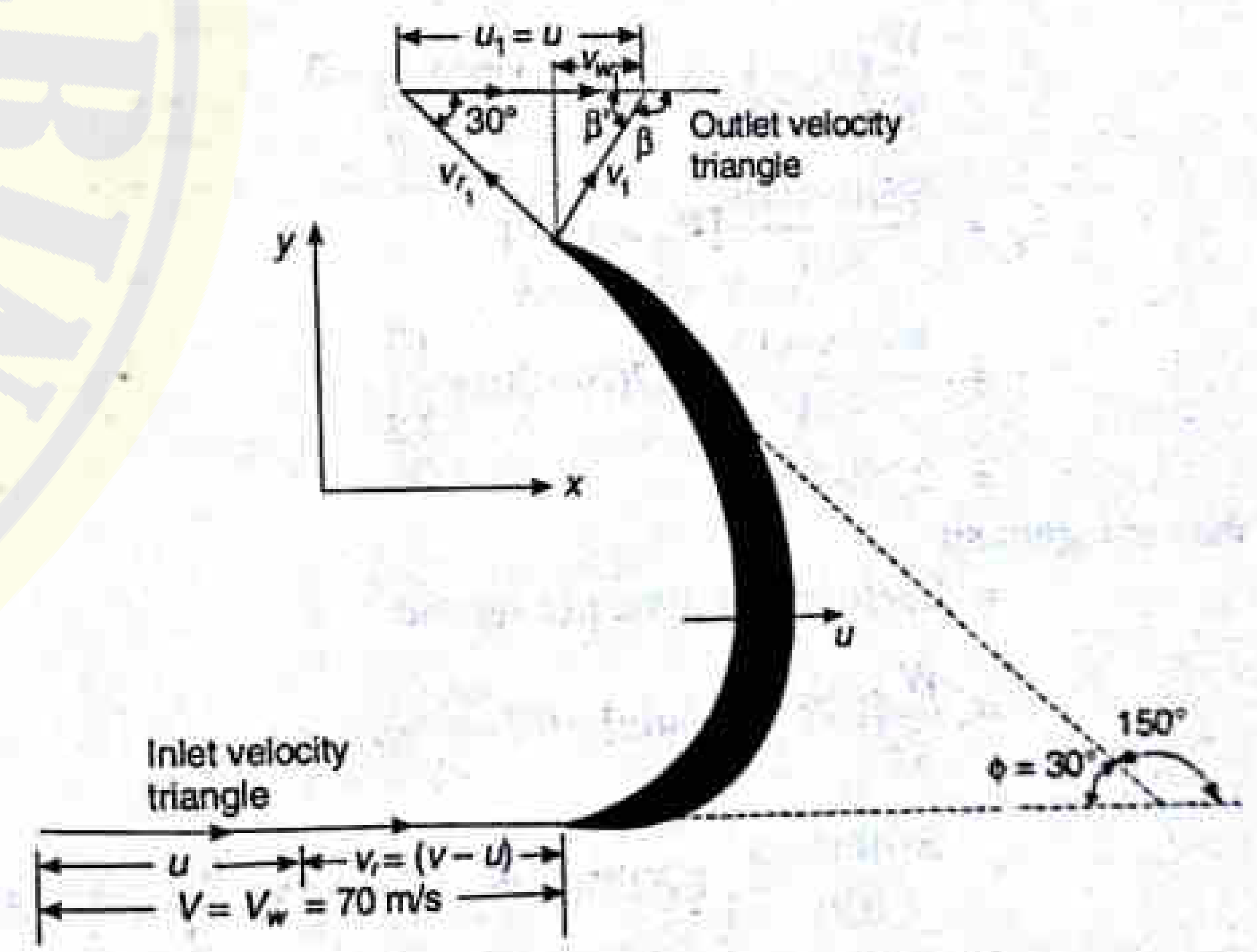


Figure Ex. 20.8

Now if $\frac{u}{V} = n$, then

$$\eta = n\,(3.594 - 4.094\,n)$$

For maximum efficiency

$$\frac{d\eta}{dn} = 0$$

Thus $\quad 3.594 - 2(4.094)n = 0$

or $$n = \frac{u}{V} = \frac{3.594}{8.188} = 0.439$$

$\therefore$ For maximum efficiency

$$u = (0.439 \times 70) = 30.73 \text{ m/s}$$

Maximum efficiency

$$\eta_{max} = 0.439\,[3.594 - (4.094)\,(0.439)]$$
$$= 0.789 \text{ or } 78.9\%$$

Force on the vane in the direction of motion of the vane is

$$F_x = \frac{W}{g}[V_w - V_{w1}]$$

$$= \frac{9810\times0.15}{9.81}[70 - 30.73 + 0.92\,(70 - 30.73)\left(\frac{\sqrt{3}}{2}\right)$$

$$= 10\,583.7 \text{ N}$$

Force on the vane at right angles to the direction of the motion of the vane is

$$F_y = \frac{W}{g}(V_{f1} - V_f) = \frac{W}{g}V_{f1}, \text{ since } V_f = 0$$

or $$F_y = \frac{9810\times0.15}{9.81}[V_{r1}\sin 30°]$$

$$= \frac{9810\times0.15}{9.81}\left[0.92(70-30)\times\frac{1}{2}\right]$$

$$= 2\,709.6 \text{ N}$$

The power of the arrangement

$$= \text{Useful work done per second}$$

$$= \frac{W}{g}[1.797\,(V-u)u] - 0.5\frac{Wu^2}{2g}$$

$$= \frac{9810\times0.15}{9.81}\left[1.797(70-30.73)\times30.73 - \frac{0.5}{2}\times(30.73)^2\right]$$

$$= 289\,872 \text{ W} = 289.872 \text{ kW}$$

Example 20.9. *The rotor of an inward flow hydraulic turbine has a diameter over the tips of the blades of 1m. The diameter at the bottom of the blades is 0.6 m. The speed is 300 r.p.m. The water is supplied through fixed vanes at 10° to the tangent to the outer circumference of the rotor, the velocity of water being 10 m/s. Find the blade angles at entry and exit, so that the water may enter and leave the moving blades without shock, the water leaves the blades with the velocity entirely radial and equal to 3.5 m/s. Also find the velocity of the water relative to the blades at the exit.*

Solution

With reference to Fig. Ex. 20.9 the following data are given

$$\alpha = 10°, R = 0.5 \text{ m}, R_1 = 0.3 \text{ m}$$
$$V = 10 \text{ m/s}, \beta = 90°, V_1 = V_{f1} = 3.5 \text{ m/s}$$
$$N = 300 \text{ r.p.m.}$$

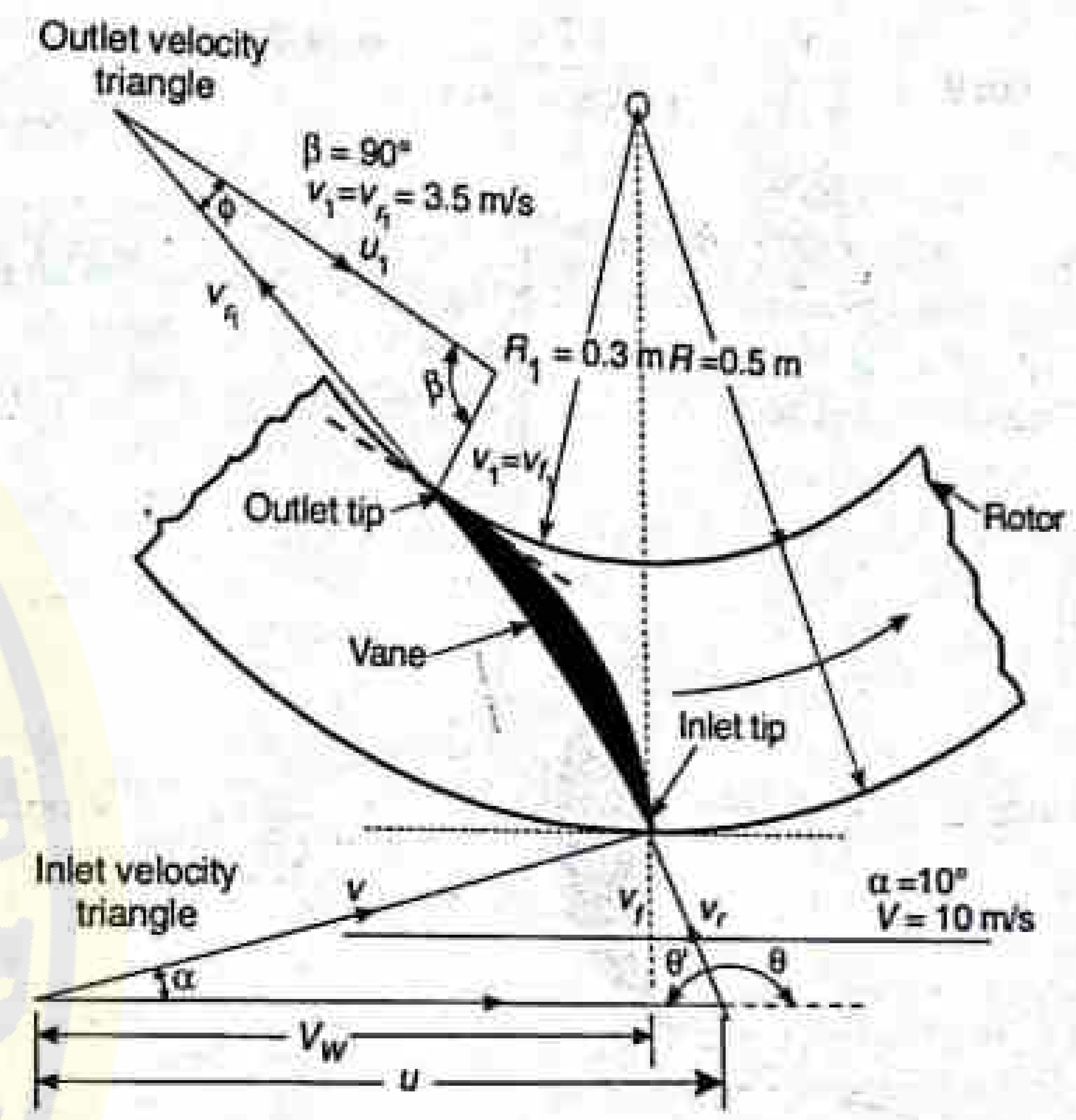


Figure Ex. 20.9

Now $$u = \frac{2\pi NR}{60}$$

$$= \frac{2\times\pi\times300\times0.5}{60} = 15.708 \text{ m/s}$$

and $$u_1 = \frac{2\pi NR_1}{60}$$

$$= \frac{2\times\pi\times300\times0.3}{60} = 9.425 \text{ m/s}$$

From the outlet velocity triangle, we have

$$\tan\phi = \frac{V_1}{u_1} = \frac{3.5}{9.425} = 0.371\,4$$

$$\therefore \quad \phi = 20.38°$$

Again $\quad V_{r1} = V_1 \operatorname{cosec} \phi$

$= 3.5 (\operatorname{cosec} 20.38°) = 10.05$ m/s

From the velocity triangle at inlet tip of the blade, we have

$$V_w = V\cos\alpha = 10(\cos 10°) = 9.848 \text{ m/s}$$

$$V_f = V\sin\alpha = 10(\sin 10°) = 1.736 \text{ m/s}$$

Now $\quad \tan\theta' = \dfrac{V_f}{u - V_w} = \dfrac{1.736}{15.708 - 9.848} = 0.2962$

or $\quad \theta' = 16.50°$

$\therefore \quad \theta = (180° - \theta') = (180° - 16.50°) = 163.50°$

Example 20.10. *A jet of water 15 mm in diameter impinges on a fixed cup which deflects the jet by 165° as shown in Fig. Ex. 20.10. If the reaction of the cup was found to be 26.5 N, when the discharge from the jet was 980 N per minute, calculate the ratio of (a) actual to theoretical force of the jet, (b) velocity at outlet to velocity at inlet.*

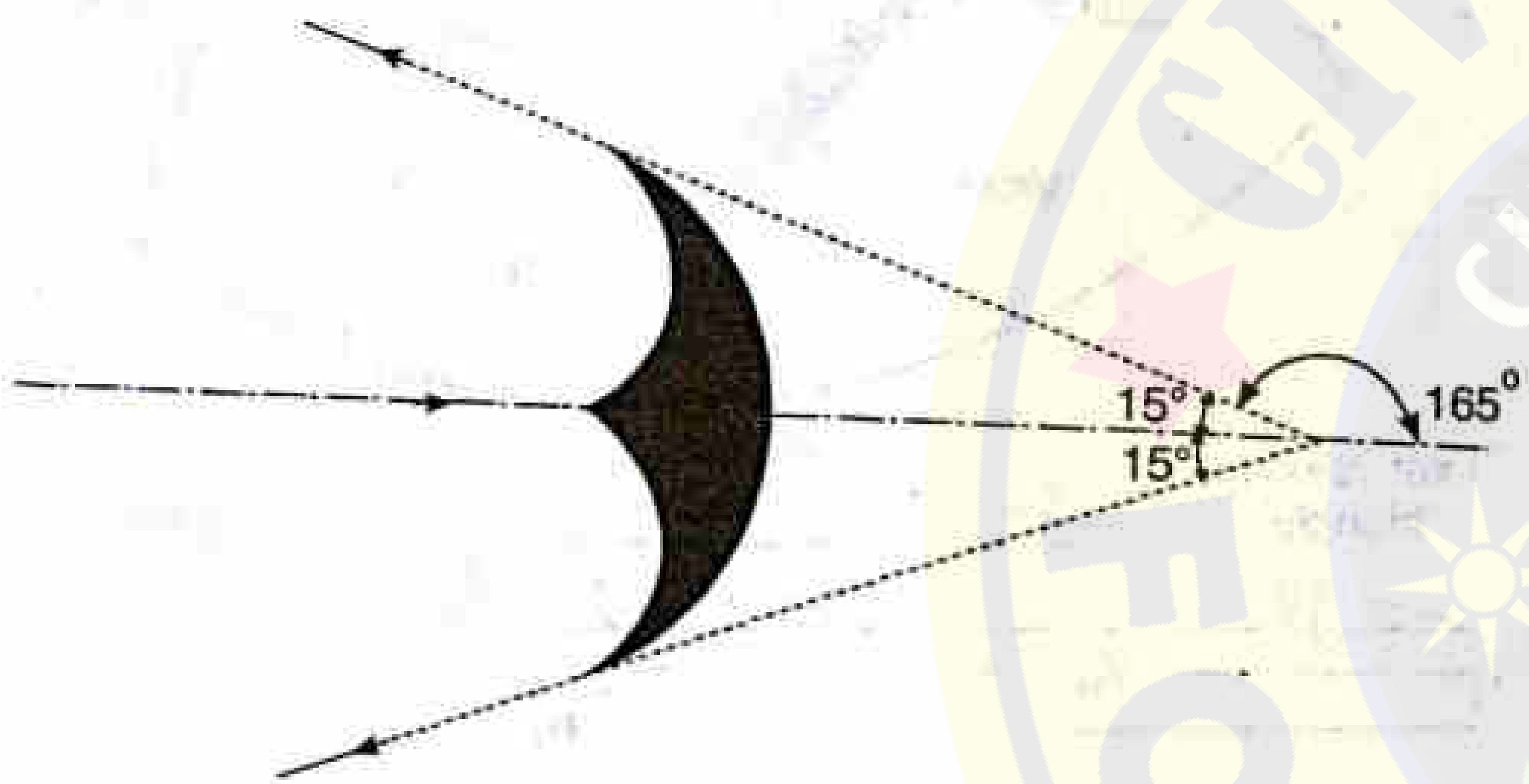


Figure Ex. 20.10

Solution

$$\text{Discharge from jet} = \frac{980}{9810 \times 60} = 0.001\,66 \text{ m}^3/\text{s}$$

$$\therefore \quad \text{Velocity of jet} = \frac{0.00166 \times 4 \times (1000)^2}{\pi \times (15)^2} = 9.39 \text{ m/s}$$

For theoretical force it is assumed that the velocity is not reduced as the water passes round each cup. Change of velocity in direction of jet = 9.39 (1 – cos 165°) = 18.46 m/s

Hence theoretical force

$= \text{(mass flowing/second)} \times \text{(change of velocity)}$

$$= \frac{980}{9.81 \times 60} \times 18.46 = 30.74 \text{ N}$$

$$\therefore \quad \frac{\text{Actual force}}{\text{Theoretical force}} = \frac{26.5}{30.74} = 0.862$$

If V_1 is the actual velocity at the outlet, then

Actual force = (Mass flowing/second) × (change of velocity)

or $\quad 26.5 = \dfrac{980}{9.81 \times 60}(9.39 - V_1 \cos 165°)$

Solving for V_1, we get

$$V_1 = 6.76 \text{ m/s}$$

$$\frac{\text{Outlet Velocity}}{\text{Inlet Velocity}} = \frac{6.76}{9.39} = 0.72$$

SUMMARY OF MAIN POINTS

1. The force exerted by a jet of water of cross-sectional area a and velocity V on stationary flat plate in the direction of the jet is given by

(i) For the plate held normal to the jet

$$F = \frac{w\,aV^2}{g} = \frac{wQV}{g} \quad (\text{since } Q = aV)$$

(ii) For the plate held inclined at an angle θ to the jet

$$F = \frac{w\,aV^2}{g}\sin\theta = \frac{wQV}{g}\sin\theta$$

The division of discharge Q that results in this case is given as

$$Q_1 = \frac{Q}{2}(1 + \cos\theta)$$

$$Q_2 = \frac{Q}{2}(1 - \cos\theta)$$

2. The force exerted by a jet of water of cross-sectional area a and velocity V on a flat plate moving with velocity u in the dirction of the jet is given by

(i) For the plate held normal to the jet

$$F = \frac{wa(V-u)^2}{g}$$

(ii) For the plate held inctined at an angle θ to the jet

F = 968a jet of water of cross-sectional area a and velocity V, striking the wheel tangentially at its lower edge is given by

$$F = \frac{waV}{g}(V - u)$$

The work done by the jet per second will be the output of the jet. Thus output of the jet = $\dfrac{waV}{g}(V-u)u$

The efficiency of the wheel is given by

$$\eta = \frac{2u(V-u)}{V^2}$$

For maximum efficiency $u = \dfrac{V}{2}$, and $\eta_{max} = 0.5$ or 50%.

4. The force exerted by a jet of water of cross-sectional area a and velocity V on a stationary curved vane in the direction of the jet is given by

(i) For the jet striking a symmetrical vane at the centre

$$F = \frac{waV^2}{g}(1 + \cos\theta)$$

where

θ = angle of the vane tips.

(ii) For the jet striking an unsymmetrical vane tangentially at one of the tips

$$F_x = \frac{waV^2}{g}(\cos\theta + \cos\phi)$$

$$F_y = \frac{waV^2}{g}(\sin\theta - \sin\phi)$$

The resultant thrust extered by the jet will be $F = \sqrt{F_x^2 + F_y^2}$ and its inclination with the x-axis is

$$\alpha = \tan^{-1}\left(\frac{F_y}{F_x}\right)$$

where

θ = angle of the vane at the inlet tip; and

ϕ = angle of the vane at the outlet tip.

For semicircular vane $\theta = \phi = 0$, and hence $F = F_x$

$$= \frac{2wa V^2}{g} \quad \text{and } F_y = 0$$

5. The force exerted by a jet of water of cross-sectional area a and velocity V on a curved vane moving with velocity u in the direction of the jet is given by
   (i) For the jet striking a symmetrical vane at the centre

$$F = \frac{wa(V-u)^2}{g}(1+\cos\theta)$$

The work done by the jet on the vane per second

$$= F \times u$$

$$= \frac{wa(V-u)^2}{g}(1+\cos\theta)u$$

The efficiency of the jet is given by

$$\eta = \frac{2u(V-u)^2(1+\cos\theta)}{V^3}$$

$$\eta_{max} = \frac{8}{27}(1+\cos\theta)$$

For a series of symmetrical curved vanes mounted on the periphery of a wheel the force exerted by the jet is given by

$$F = \frac{waV(V-u)}{g}(1+\cos\theta)$$

The work done by the jet per second will be the output of the jet. Thus output of the jet = $\frac{waV(V-u)}{g}(1+\cos\theta)u$. The efficiency of the wheel is given by

$$\eta = \frac{2u(V-u)(1+\cos\theta)}{V^2}$$

$$\eta_{max} = \frac{1}{2}(1+\cos\theta)$$

(ii) For the jet striking an unsymmetrical vane tangentially at one of the tips

$$F_x = \frac{waV_r}{g}(V_w \pm V_{w_1})$$

$$\text{Work done per second} = \frac{waV_r}{g}(V_w \pm V_{w_1})u$$

6. For a series of unsymmetrical curved vanes mounted radially on the periphery of a wheel the work done by the jet of water per second

$$= \frac{W}{g}(V_w u \pm V_{w_1} u_1)$$

The efficiency of the system in this case is given by

$$\eta = \frac{2(V_w u \pm V_{w_1} u_1)}{V^2}$$

where

V_w = velocity of whirl at inlet;
V_{w1} = velocity of whirl at outlet;
u = tangential velocity of vane at inlet;
u_1 = tangential velocity of vane at outlet and
V = absolute velocity of jet at inlet.

PROBLEMS

20.1 A jet of water 50 mm in diameter having a velocity of 20 m/s, strikes normally a flat smooth plate. Determine the thrust on the plate (a) if the plate is at rest; (b) if the plate is moving in the same direction as the jet with a velocity of 8 m/s. Also find the work done per second on the plate and the efficiency of the jet when the plate is moving.
[*Ans.* (a) 784 N [79.92 kg (f)]; (b) 282.24 N [28.77 kg(f)], 2 257.92 N.m [230.16 kg (f) – m], 28.80%]

20.2 Show that the efficiency of a free jet striking normally as series of flat plates mounted on the periphery of a wheel never exceeds 50%.

20.3 A 75 mm diameter jet having a velocity of 30 m/s strikes a flat plate, the normal of which is inclined at 45° to the axis of the jet. Find the normal pressure on the plate (a) when the plate is stationary; (b) when the plate is moving with a velocity of 15 m/s in the direction of the jet and away from the jet. Also determine the power and the efficiency of the jet when the plate is moving.
[*Ans.* (a) 2812.87 N [286.7 kg(f)]; (b) 703.22 N [71.68 kg (f)], 7 458.77 W (10.14 h.p.), 12.5%]

20.4 A rectangular plate, weight 60 N is suspended vertically by a hinge on the top horizontal edge. The c.g. of the plate is 100 mm from the hinge. A horizontal jet of water 20 mm diameter, whose axis is 150 mm below the hinge impinges normally on the plate with velocity 5 m/s. Find the horizontal force, applied at the c.g. to maintain the plate in its vertical position. Find the alteration of the velocity of the jet if the plate is deflected through 30° and the same force continues to act at the c.g. of the plate.
[*Ans.* 11.78 N[1.2 kg (f)], 4.24 m/s]

20.5 A jet of water 75 mm diameter and with velocity of 20 m/s flows tangentially on to a stationary vane which deflects the water through 120. What is the magnitude and direction of the resultant force on the vane? If the jet flows on to a series of vanes moving in the direction of the jet with velocity 12 m/s, find: (a) the force on the system of vanes in the direction of motion; (b) the work done per second; and (c) the efficiency.
[*Ans.* (a) 3 062.27 N [312.16 kg (f)] at 30° to jet; (b) 1060.8 N [108.13 kg (f)]; 12 729.6 N.m [1297.6 kg (f) – m]; (c) 72%]

20.6 A jet of water moving at 15 m/s impinges on a symmetrical concave vane shaped to deflect the jet through 140°. If the vane is moving at 6 m/s, find the angle of the jet so that there is no shock at inlet. Also determine the absolute velocity of exit in magnitude and direction and the work done per unit weight of water.
[*Ans.* 12.14°; 4.13 m/s, 49.8°; 104 N.m/N [10.6 kg(f) - m/kg (f)]]

20.7 A jet of water having a velocity of 60 m/s impinges without shock on a series of vanes moving at 30 m/s. The direction of motion of vanes being inclined at 20° to that of the jet. The angle between the inlet and outlet relative velocities is 120° and the ratio between these velocities is 0°9. Find: (a) the angle θ at the entry edge of the vane; (b) the work done by the vanes in N.m/N of water supplied by the jet ; (c) the hydraulic efficiency.
[*Ans.* (a) 37.9°; (b) 1627.55 N.m/N ; (c) 90.42%]

20.8 A jet having a velocity V strikes a single curved vane moving in the same direction as the jet with velocity u, so that the velocity of the jet relative to the vane is $(V - u)$. The vane causes the jet to be reversed in direction. Show that the maximum efficiency is obtained when $V = 3u$ and that this maximum efficiency is slightly less than 0.6.

20.9 Water enters a wheel consisting of curved vanes, outside diameter 1 m, inside diameter 0.5 m. The flow is inwards from the circumference. The supply jet is at 30° to the tangent to the outside circle with a velocity of 40 m/s. The water is to leave the wheel at 3.5 m/s at 120° to the tangent to the inner circle. Draw the velocity triangles for the inlet and outlet and find suitable blade angles if the wheel runs at 360 r.p.m.
[*Ans.* 51.7°; 15.2°]

20.10 A wheel having radial blades is of 0.6 m radius at the outer tip of the blades and 0.3 m at the inner tip. Water enters the blades at the outer tip with a velocity of 30 m/s at the angle of 30° to tangent and leaves the blades with a velocity of flow of 4 m/s. The blades have an angle of 40° at entrance and 35° at exit. Find the work done per newton of water entering the wheel, the speed of the wheel, and the efficiency.
[*Ans.* 22 N.m/N ; 129 r.p.m ; 48%]

20.11 Water impinges on a smooth moving curved vane under the following conditions: Velocity of vane = 15 m/s; direction of jet at entry to vane = 30° to the direction of the vane; velocity of jet= 30 m/s. At the exit side the vane makes an angle of 170° with the direction of motion of the vane (i.e., it is curved backwards). Find: (a) the vane angle at entry so that the water strikes the vanes tangentially; (b) the absolute velocity of the water after leaving the vane and its direction ; (c) the work done on the vanes per unit weight of water supplied.
[*Ans.* (a) 53° 48′ ; (b) 4.62 m/s at 135° 42′ to tangent at exit ; (c) 44.79 N.m/N]

20.12 A free jet of water of velocity V strikes against a series of curved semi-circular vanes tangentially. The vanes are moving in the direction of the jet with a velocity equal to 0.6V. Assuming the relative velocity of water is reduced by 10% by moving over the vanes, show that the vanes have an efficiency of 91.20%.

20.13 The following data refer to a water wheel comprising a series of flat plates mounted on the periphery of the wheel radially. Diameter of wheel (upto centre of plates) = 1.22 m, diameter of free jet = 168 mm. Head at the beginning of the nozzle = 18.3 m, coefficient of velocity of the nozzle = 0.97, and speed of the wheel = 120 r.p.m. Determine (a) efficiency of the water wheel, and (b) power delivered to the water wheel.

[*Ans.* (a) 48.6%; (b) 68.82 kW (93.54 h.p.)]

20.14 A water wheel has a number of hemispherical vanes equally spaced on its periphery. A jet of diameter 300 mm having a velocity of 16 m/s impinges on the vanes which have a tangential velocity of 6 m/s. Determine the work done on the wheel.

[*Ans.* 135.717 kN. m/s {13 835 kg(f)-m/s}]

20.15 Water flows over series of curved vanes of a hydraulic turbine wheel, the diameter of which between inlet tips of vanes is 2 m and that between outlet tips is 1 m. The wheel rotates at 240 r.p.m. Jet of water enters at an angle of 30° to the tangent to wheel at inlet with a velocity of 40 m/s and leaves with a velocity of 10 m/s at an angle of 30° to the tangent to wheel at outlet tip. Find (a) vane angles at entry and exit; (b) work done on the wheel per newton of water; (c) hydraulic efficiency of wheel; (d) power developed by the wheel when the discharge flowing through it is 0.3 m³/s.

[*Ans.* (a) 64° 34′, 52° ; (b) 77.65 N.m/N; (c) 95.2%; (d) 228.52 kW (310.6 h.p.)]

20.16 A 50 mm diameter jet having a discharge of 50 litres per second impinges without shock on a series of vanes which move in the same direction as the jet at 10 m/s. The shape of each vane is such that if stationary it would deflect the jet through an angle of 135°. Due to frictional resistance the relative velocity at outlet is 0.85 of that at inlet. Determine : (a) the magnitude and direction of the resultant force on vanes; (b) the work done on the vanes; and (c) the efficiency of the system.

[*Ans.* (a) 1326.2 N {135.2 kg (f)} at 20° 34′; (b) 12 416 N.m/s {1 265.6 kg (f) - m/s}; (c) 76.32%]

20.17 A 20 mm diameter jet having a velocity of 60 m/s impinges without shock on a series of vanes which move in the same direction as the jet. The shape of each vane is such that, if stationary, it would deflect the jet through an angle of 150°. Friction reduces the relative velocity by 10% as water flows across the vanes and there is a further windage loss given as $(0.4u^2/2g)$ N.m per N of water, where u is velocity of vanes. Find the velocity of vanes corresponding to an efficiency of 80%. Also determine the force on the vanes in, and at right angles, to the direction of their motion.

[*Ans.* 27 m/s ; 1106.3 N{112.77 kg(f)}, 279.8 N {28.52 kg (f)}]

Hydraulic Turbines

Chapter 21

21.1 INTRODUCTION

Hydraulic (or water) turbines are the machines which use the energy of water (hydro-power) and convert it into mechanical energy. As such these may be considered as hydraulic motors or primemovers. The mechanical energy developed by a turbine is used in running an electric generator which is directly coupled to the shaft of the turbine. The electric generator thus develops electric power, which is known as *hydroelectric power.* Since the generation of hydro-electric power is relatively cheaper than the power generated by other sources such as coal, oil, etc., nowadays a number of hydro-electric and multipurpose projects have been undertaken in our country in order to harness more and more power from the available water potential.

The idea of utilising hydraulic energy to develop mechanical energy has been in existence for more than 2000 years. In the earlier days of water-power development, water wheels made of wood, were widely applied which used either the energy of falling water (i.e., potential energy) or the kinetic energy of the flowing stream of water. One of the types of water wheels formerly used was the *overshot wheel.* It consisted of a series of buckets attached to the periphery of a wheel, the diameter of which was equal to the available head. Water was permitted to enter the buckets at the top, and the imbalance created by the weight of the water caused the wheel to rotate. The buckets were designed to empty themselves when they reached to bottom of the wheel. The overshot wheel, when properly designed, had very good efficiency, but it could not be built to handle large quantity of water. Another type of water wheel formerly used was the *undershot wheel,* which used the kinetic energy of the water. An earlier type of undershot wheel consisted of a series of a straight blades attached to the periphery of a wheel and so placed that a swiftly moving stream of water used to strike the blades on the underside of the wheel. The efficiency of this type of wheel was low. As such an improvement on the straight blade type of undershot wheel was suggested by Poncelet, who instead of straight blades designed curved blades so that water strikes the blades of the wheel, practically without shock. This type of wheels were called Poncelet wheels.

However, these water wheels utilized small heads and were capable of producing small power. Moreover, these wheels had a low efficiency and they used to run very slowly and hence these could not be directly coupled to the modern fast running electric generators for the purpose of power generation. As such the water wheels have been completely replaced by the modern type of hydraulic (or water) turbines, which may operate under any head and practically any desired speed thereby enabling the generator to be coupled directly.

In general a water turbine consists of a wheel called *runner (or rotor)* having a number of specially designed *vanes or blades or buckets*. The water possessing a large amount of hydraulic energy when strikes the runner, it does work on the runner and causes it to rotate. The mechanical energy so developed is supplied to the generator coupled to the runner, which then generates electrical energy.

21.2 ELEMENTS OF HYDROELECTRIC POWER PLANTS

One of the essential requirements of the hydroelectric power generation is the availability of a continuous source of water with a large amount of hydraulic energy. Such a source of water may be made available if a natural lake or a reservoir may be found at a higher elevation or an artificial reservoir may be formed by constructing a dam across a river. Figure 21.1 shows a general layout of a hydroelectric power plant, in which an artificial storage reservoir formed by constructing a dam has

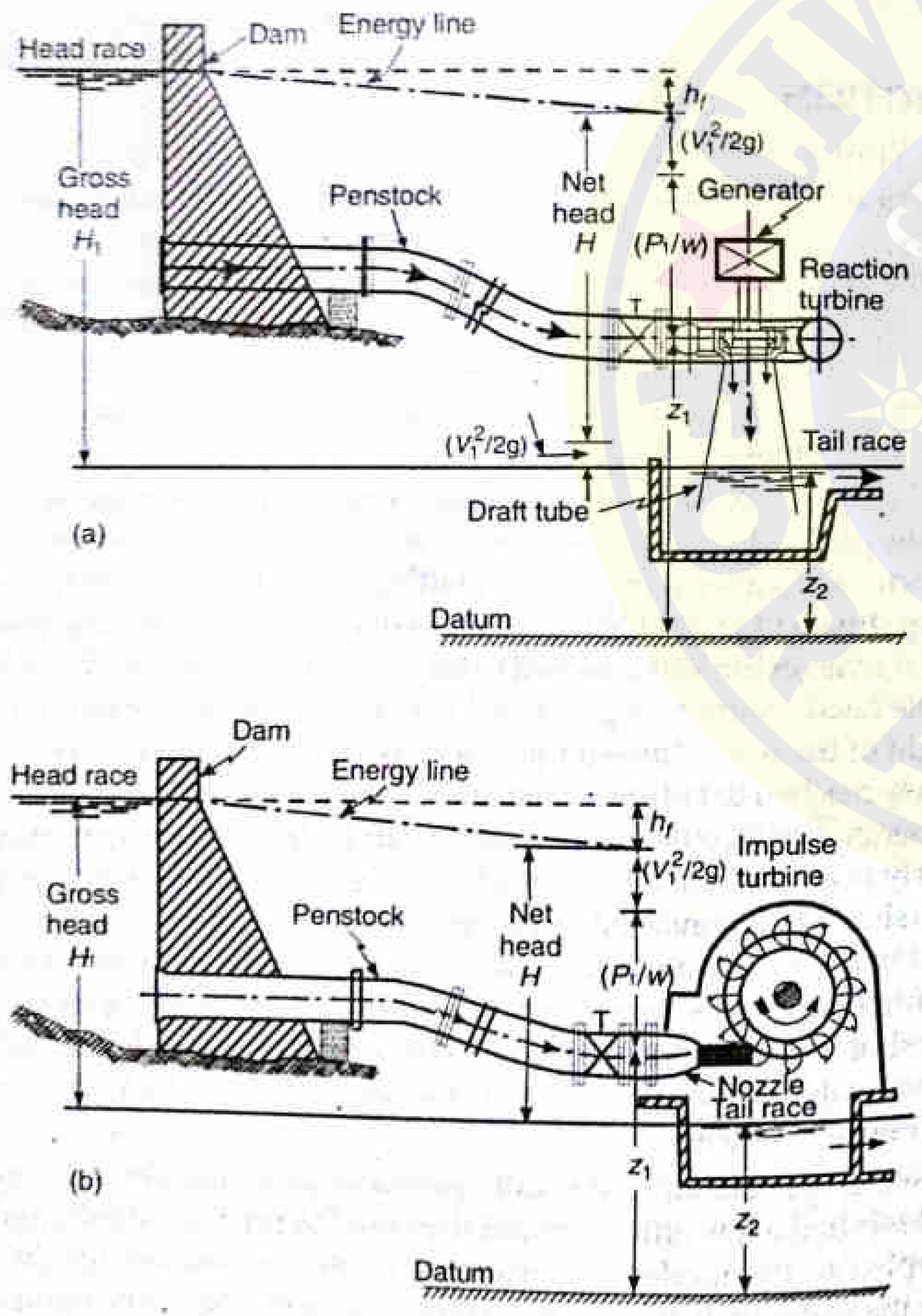


Figure 21.1 *General layout of a hydro-electric power plant*

been shown. The water surface in the storage reservoir is known as *head race level* or simply *head race*. Water from the storage reservoir is carried through penstock or canals to the power house. Penstocks are the pipes of large diameter, usually made of steel, wood or reinforced concrete, which carry water under pressure from the storage reservoir to the turbine. In some installations smaller reservoirs known as forebays are also provided. A *forebay* is essentially a storage reservoir at the head of the penstocks. The purpose of a forebay is to temporarily store water when it is not required by the turbine and supply the same when required. When the power house is located just at the base of the dam no forebay is required to be provided since the reservoir itself serves the same purpose. However, if the power house is situated away from the storage reservoir and water is carried to the power house through a canal, then a forebay may be provided. In that case water from the reservoir is first led into forebay which in turn distributes it to penstocks through which it is supplied to the turbines. Furthermore, where the power house is located across a canal, a forebay may be provided by enlarging the canal just ahead of the power house.

The water passing through the turbine is discharged to the tail race. The *tail race* is the channel which carries water (known as tail water) away from the power house after it has passed through the turbine. It may be a natural stream channel or a specially excavated channel entering the natural stream at some point downstream from the power house. The water surface in the tail race channel is known as tail race level or simply tail race.

21.3 HEAD AND EFFICIENCIES OF HYDRAULIC TURBINES

(a) **Heads.** The head acting on a turbine may be defined in two ways as follows:

(i) **Gross head** is defined as the difference between the head race level and the tail race level when no water is flowing. As such the gross head is often termed as static head or total head and it may be represented by H_1 as shown in Fig. 21.1.

(ii) **Net or effective head** is the head available at the entrance to the turbine. It is obtained by subtracting from the gross head all the losses of head that may occur as water flows from the head race to the entrace of the turbine. The losses of head are mainly due to friction occurring in penstocks, canal etc. Thus if H represents the net head and h_f is the total loss of head between the head race and the entrance of the turbine then

$$H = H_1 - h_f \qquad ...(21.1)$$

For a *reaction (or encased) turbine* as shown in Fig. 21.1 (a) the net head is equal to the difference between (1) the pressure head at the entrance to the turbine plus the velocity head in the penstock at this point plus the elevation of this point above the assumed datum, and (2) the elevation of the tail water plus the velocity head in the draft tube at its exit. Thus

$$H = \left(\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1\right) - \left(Z_2 + \frac{V_2^2}{2g}\right) \qquad ...(21.2)$$

For an *impulse turbine* as shown in Fig. 21.1 (b) the net head is equal to the difference between (1) the pressure head at the entrance to the nozzle plus the velocity head in the penstock at this point plus the elevation of this point above the assumed datum and (2) the elevation of the tail water. Thus

$$H = \left(\frac{p_1}{w} + \frac{V_1^2}{2g} + Z_1\right) - Z_2 \qquad ...(21.3)$$

Hydroelectric power plants are usually classified according to the heads under which they w... high head, medium head and low head plants. High head plants are those which are working ... heads more than about 250 m, while low head plants are those which are working under head... than about 60 m, and medium head plants are those which are working under heads ranging fr... 60 m to 250 m.

(b) Efficiencies. The various energy (or head) losses that may occur in a hydroelectric power pl... with reaction and impulse turbine units are shown in Fig. 21.2. Accordingly the various efficienc... of the turbines may be expressed as following.

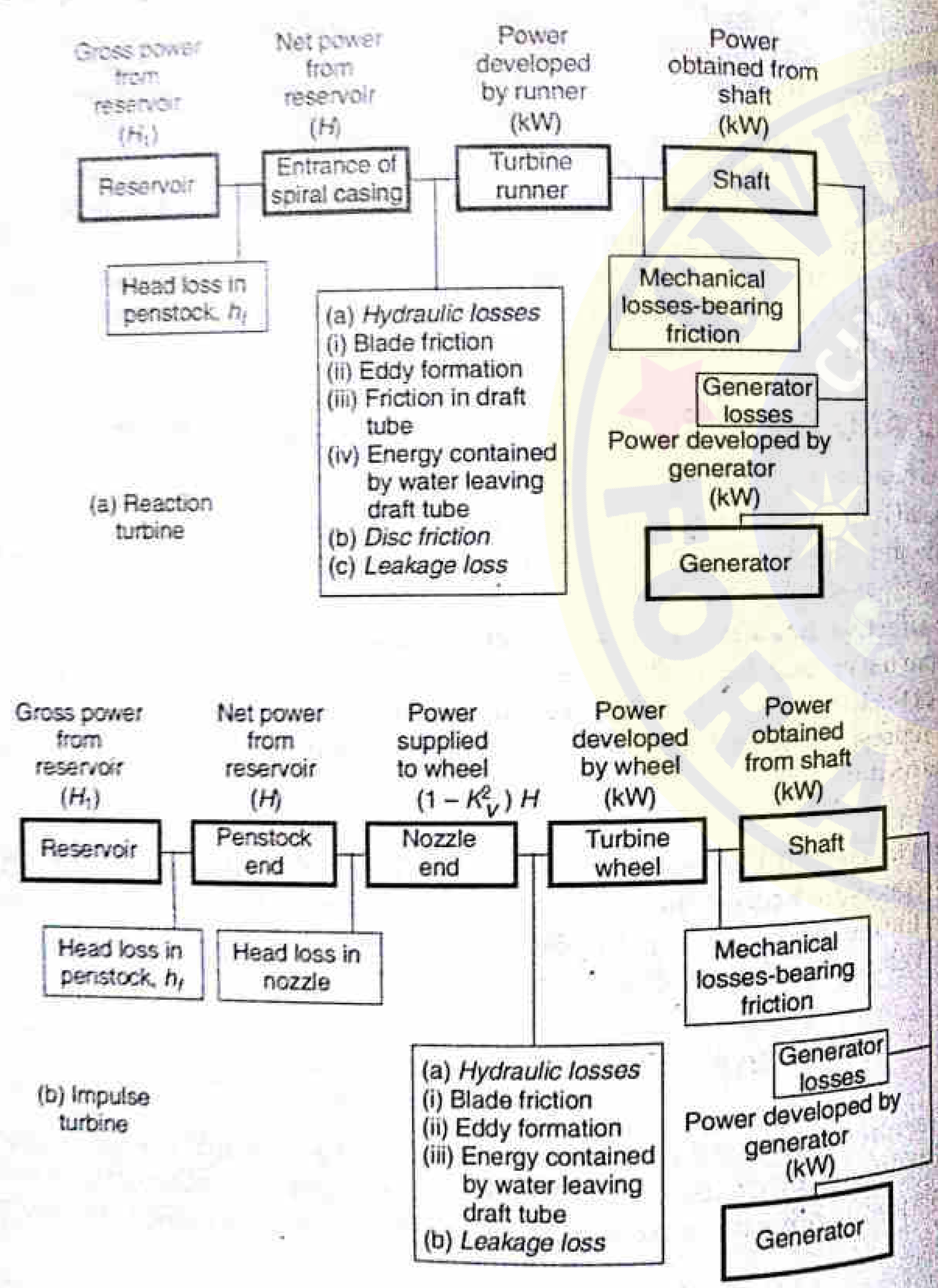


Figure 21.2 *Losses of energy in hydroelectric installations*

(i) ***Hydraulic efficiency*** η_h. The hydraulic efficiency of the turbine is the ratio of the power developed by the runner to the net power supplied by the water at the entrance to the turbine. These two powers differ by the amount of the hydraulic losses. That is

$$\eta_h = \frac{\text{Power developed by the runner}}{\text{Net power supplied at the turbine entrance}}$$

or

$$\eta_h = \frac{\text{Power developed by the runner}}{[w(Q+\Delta Q)H]} \quad ...(21.4)$$

where Q is the quantity of water actually striking the runner, ΔQ is the quantity of water that is discharged directly to the tail race without striking the runner, and H is the net head available at the entrance to the turbine. However, if ΔQ is negligibly small, Eq. 21.4 becomes

$$\eta_h = \frac{\text{Power developed by the runner}}{(wQH)} \quad ...(21.5)$$

In SI units the powers are usually expressed in kilowatts (kW). However, in metric units the powers are expressed in metric horse power. In the later case the power supplied at the turbine entrance is termed as water horse power (W.H.P.).

(ii) ***Mechanical efficiency*** η_m. The mechanical efficiency of the turbine is the ratio of the power available at the turbine shaft to the power developed by the runner. These two powers differ by the amount of the mechanical losses viz., bearing friction. That is

$$\eta_m = \frac{\text{Power available at the turbine shaft}}{\text{Power developed by the runner}}$$

or

$$\eta_m = \frac{P}{\text{Power developed by the runner}} \quad ...(21.6)$$

where P is the power available at the turbine shaft. In metric units since the power is expressed in horse power (metric), the power available at the turbine shaft is termed as Shaft Horse Power, S.H.P., or Brake Horse Power, B.H.P.

(iii) ***Volumetric efficiency*** h_v. The volumetric efficiency is the ratio of the quantity of water actually striking the runner and the quantity of water supplied to the turbine. These two quantities differ by the amount of water that slips directly to the tail race without striking the runner. That is

$$\eta_v = \frac{Q}{(Q+\Delta Q)} \quad ...(21.7)$$

(iv) ***Overall efficiency*** η_o. The overall efficiency of the turbine is the ratio of the power available at the turbine shaft to the power supplied by the water at the entrance to the turbine. That is

$$\eta_o = \frac{\text{Power available at the turbine shaft}}{\text{Net power supplied at the turbine entrance}}$$

or

$$\eta_o = \frac{P}{[w(Q+\Delta Q)H]} \quad ...(21.8)$$

It is evident from Eq. 21.8 that overall efficiency of the turbine is

$$\eta_o = \eta_h \times \eta_m$$

if η_h is given by Eq. 21.4 in which the volumetric efficiency is implied ; and

$$\eta_o = \eta_h \times \eta_v \times \eta_m$$

if η_h is given by Eq. 21.5.

21.4 CLASSIFICATION OF TURBINES

Hydraulic turbines may be classified according to several considerations as discussed below.

According to the action of the water flowing through the turbine runners the turbines may be classified as impulse turbines and reaction turbines. In an *impulse turbine*, all the available energy of water is converted into kinetic energy or velocity head by passing it through a contracting nozzle provided at the end of the penstock. The water coming out of the nozzle is formed into a free jet which impinges on a series of buckets of the runner thus causing it to revolve. The runner revolves freely in air. The water is in contact with only a part of the runner at a time, and throughout its action on the runner and in its subsequent flow to the tail race, the water is at atmospheric pressure. A casing is however provided on the runner to prevent splashing and to guide the water discharged from the buckets to the tail race. Some of the impulse turbines are Pelton wheel, Turgo-impulse wheel, Girard turbine, Banki turbine, Jonval turbine etc. Out of these turbines only *Pelton wheel* is predominantly used at present, which has been described later.

In a *reaction turbine*, at the entrance to the runner, only a part of the available energy of water is converted into kinetic energy and a substantial part remains in the form of pressure energy. As water flows through the runner the change from pressure to kinetic energy takes place gradually. As such the pressure at the inlet to the turbine is much higher than the pressure at the outlet and it varies throughout the passage of water through the turbine. For this gradual change of pressure to be possible the runner in this case must be completely enclosed in an air-tight casing and the passage in entirely full of water throughout the operation of the turbine. The difference of pressure (or pressure difference) between the inlet and the outlet of the runner is called *reaction pressure*, and hence these turbines are known as reaction turbines. Some of the reaction turbines are Fourneyron, Thomson, Francis, Propeller, Kaplan, etc. Out of these the Francis and the Kaplan turbines are predominantly used at present which have been described later.

The turbines may also be classified according to the main direction of flow of water in the runner as (i) tangential flow turbine, (ii) radial flow turbine, (iii) axial flow turbine, and (iv) mixed flow turbine.

In a *tangential flow* turbine the water flows along the tangent to the path of rotation of the runner. Pelton wheel is a tangential flow turbine.

In a *radial flow* turbine the water flows along the radial direction and remains wholly and mainly in the plane normal to the axis of rotation, as it passes through the runner. A radial flow turbine may be either inward radial flow type or outward radial flow type. In an *inward radial flow* turbine the water enters at the outer circumference and flows radially inwards towards the centre of the runner. Old Francis turbine, Thomson turbine, Girard radial flow turbine etc., are some of the examples of inward radial flow turbine. In an *outward radial flow* turbine water enters at the centre and flows radially outwards towards the outer periphery of the runner. Fourneyron turbine is an example of outward radial flow turbine.

In an *axial flow* turbine the flow of water through the runner is wholly and mainly along the direction parallel to the axis of rotation of the runner. Jonval turbine, Girard axial flow turbine, Propeller turbine, Kaplan turbine etc., are some of the examples of axial flow turbines.

In *mixed flow* turbine, water enters the runner at the outer periphery in the radial direction and leaves it at the centre in the direction parallel to the axis of rotation of the runner. Modern Francis turbine is an example of the mixed flow type turbine.

On the basis of the head and quantity of water required, the turbines may be classified as (i) high head turbine, (ii) medium head turbine, and (iii) low head turbine.

High head turbines are those which are capable of working under very high heads ranging from several hundred metres to few thousand metres. These turbines thus require relatively less quantity of water. In general impulse turbines are high head turbines. In particular Pelton wheel has so far been used under a highest head of about 1770 m (5800 ft.).

Medium head turbines are those which are capable of working under medium heads ranging from about 60 m to 250 m. These turbines require relatively large quantity of water. Modern Francis turbines may be classified as medium head turbines.

Low head turbines are those which are capable of working under the heads less than 60 m. These turbines thus require a large quantity of water. Kaplan and other propeller turbines may be classified as low head turbines.

The turbines may also be classified according to their specific speed. The *specific speed* of a turbine is the speed of a geometrically similar turbine that would develop one kilowatt power when working under a head of one metre. However, in metric units the *specific speed* of a turbine is defined as the speed of a geometrically similar turbine that would develop one metric horse power when working under a head of one metre. On the basis of the specific speed the various turbines may be considered in the following three groups in which the values given in the brackets represent the range of specific speed in metric units.

(i) Specific speed varying from 8.5 to 30 (10 to 35) — Pelton wheel with single jet and upto 43 (50) for Pelton wheel with double jet.

(ii) Specific speed varying from 50 to 340 (60 to 400) —Francis turbine.

(iii) Specific speed varying from 255 to 860 (300 to 1000) — Kaplan and other propeller turbines.

The turbines may also be classified according to the disposition of their shafts. The turbines may be disposed with either vertical or horizontal shafts and hence these may be classified as turbines with *vertical disposition of shaft* and turbines with *horizontal disposition of shaft*. Out of the two types the turbines with vertical disposition of shaft are commonly adopted.

21.5 PELTON WHEEL

This is the only impulse type of hydraulic turbine now in common use. It is named after Lester A. Pelton (1829–1908), the American engineer who contributed much to its development in about 1880. It is well suited for operating under high heads.

Figure 21.3 shows the elements of a typical Pelton wheel installation. The runner consists of a circular disc with a number of buckets evenly spaced round its periphery. The buckets have a shape of double semi-ellipsoidal cups. Each bucket is divided into two symmetrical parts by a sharp-edged ridge known as splitter. One or more nozzles are mounted so that each directs a jet along a tangent to the circle through the centres of the buckets called the pitch circle. The jet of water impinges on the splitter, which divides the jet into two equal portions, each of which after flowing round the smooth inner surface of the bucket leaves it at its outer edge. The buckets are so shaped that the angle at the outlet tip varies from 10° to 20° (usually kept as 15°) so that the jet of water gets deflected through 160° to 170°. The advantage of having a double cup-shaped buckets is that the axial thrusts neutralise each other, being equal and opposite, and hence the bearings supporting the wheel shaft are not sub... to any axial or end thrust. The back of the bucket is so shaped that as it swings downward into the ... no water is wasted by splashing. Further at the lower tip of the bucket a notch is cut which prevents ... jet striking the preceding bucket being intercepted by the next bucket very soon, and it also avoids ... deflection of water towards the centre of the wheel as the bucket first meets the jet. For low heads ... buckets are made of cast iron, but for higher heads they are made of cast steel, bronze or stainless ...

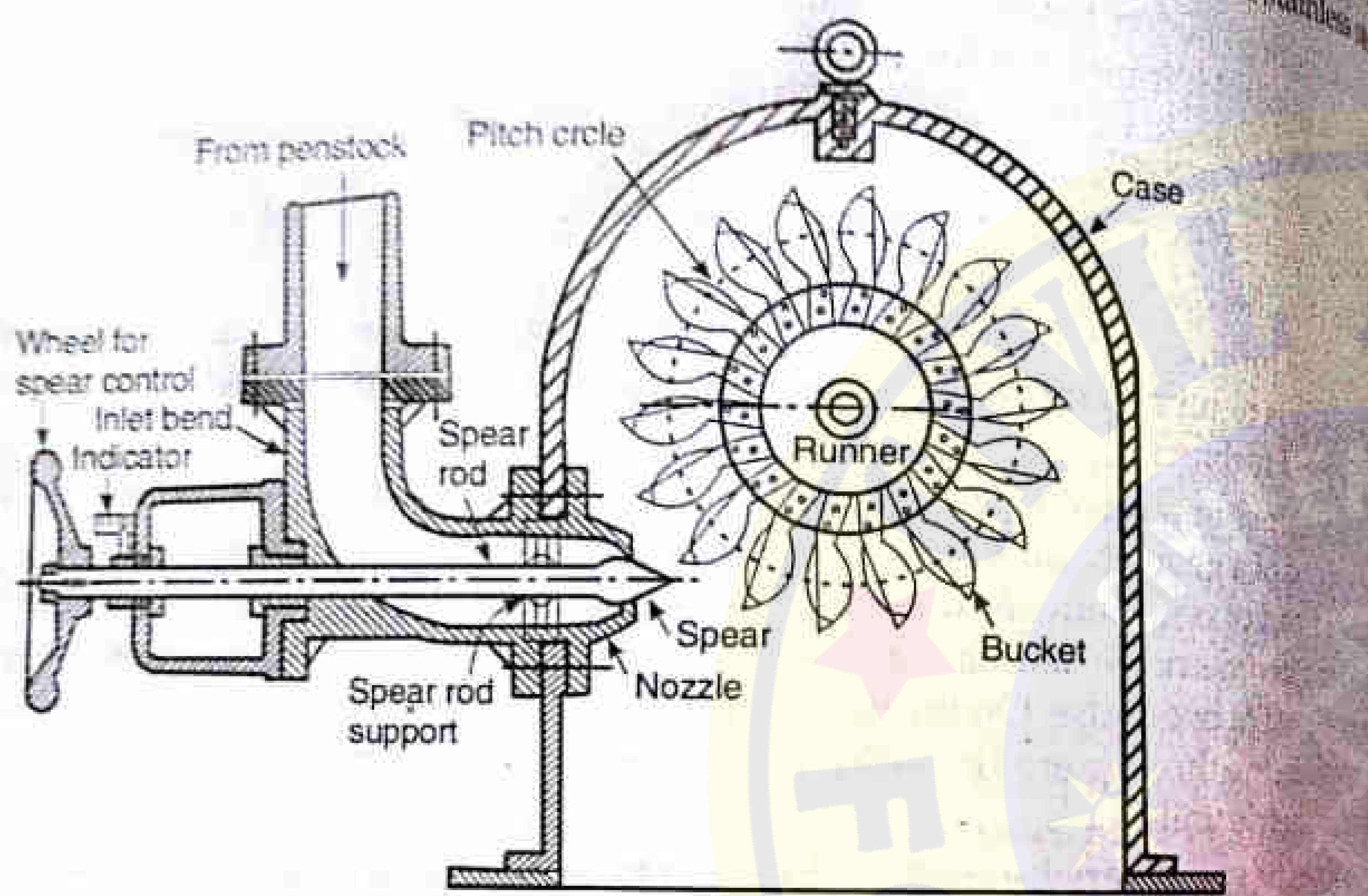


Figure 21.3 *Single jet Pelton wheel*

In order to control the quantity of water striking the runner, the nozzle fitted at the end of the penstock is provided with a spear or needle having a streamlined head which is fixed to the end of the rod as shown in Fig. 21.3. The spear may be operated either by a wheel (Fig. 21.3) in case of very small units or automatically by a governor (described later) in case of almost all the bigger units. When the shaft of the Pelton wheel is horizontal then not more than two jets are used. But if the wheel is mounted on a vertical shaft a larger number of jets (upto six) is possible.

A casing made of cast iron or fabricated steel plates is usually provided for a Pelton wheel as shown in Fig. 21.3. It has no hydraulic function to perform. It is provided only to prevent splashing of water, to lead water to the tail race and also to act as a safeguard against accidents.

Larger Pelton wheels are usually equipped with a small brake nozzle which when opened directs a jet of water on the back of the buckets, thereby bringing the wheel quickly to rest after it is shut down (as otherwise it would go on revolving by inertia for a considerable time).

21.6 WORK DONE AND EFFICIENCIES OF PELTON WHEEL

The transfer of work from the jet of water to the buckets may be determined by applying the momentum equation as indicated in Chapter 20. Figure 21.4 shows velocity triangles at the tips of the bucket of a Pelton wheel. Let

V = absolute velocity of jet before striking the bucket

V_1 = absolute velocity of jet leaving the bucket

u = absolute velocity of bucket considered along the direction tangential to the pitch circle

V_r = velocity of the incoming jet relative to the bucket

V_{r1} = velocity of the jet leaving the bucket relative to the bucket

V_w = velocity of whirl at inlet tip of the bucket

V_{w1} = velocity of whirl at outlet tip of the bucket

θ = angle through which the jet is deflected by the bucket $(=180° - \phi)$, where ϕ is the angle of the bucket at the outlet tip.

Since the velocities V and u are colinear, the velocity triangle at the inlet tip of the bucket is a straight line and thus

$$V_r = (V-u) \text{ and } V_w = V$$

At the outlet tip any one of the three velocity triangles as shown in Fig. 21.4 is possible depending upon the magnitude of u, corresponding to which it is a slow, medium or fast runner. As the inlet and

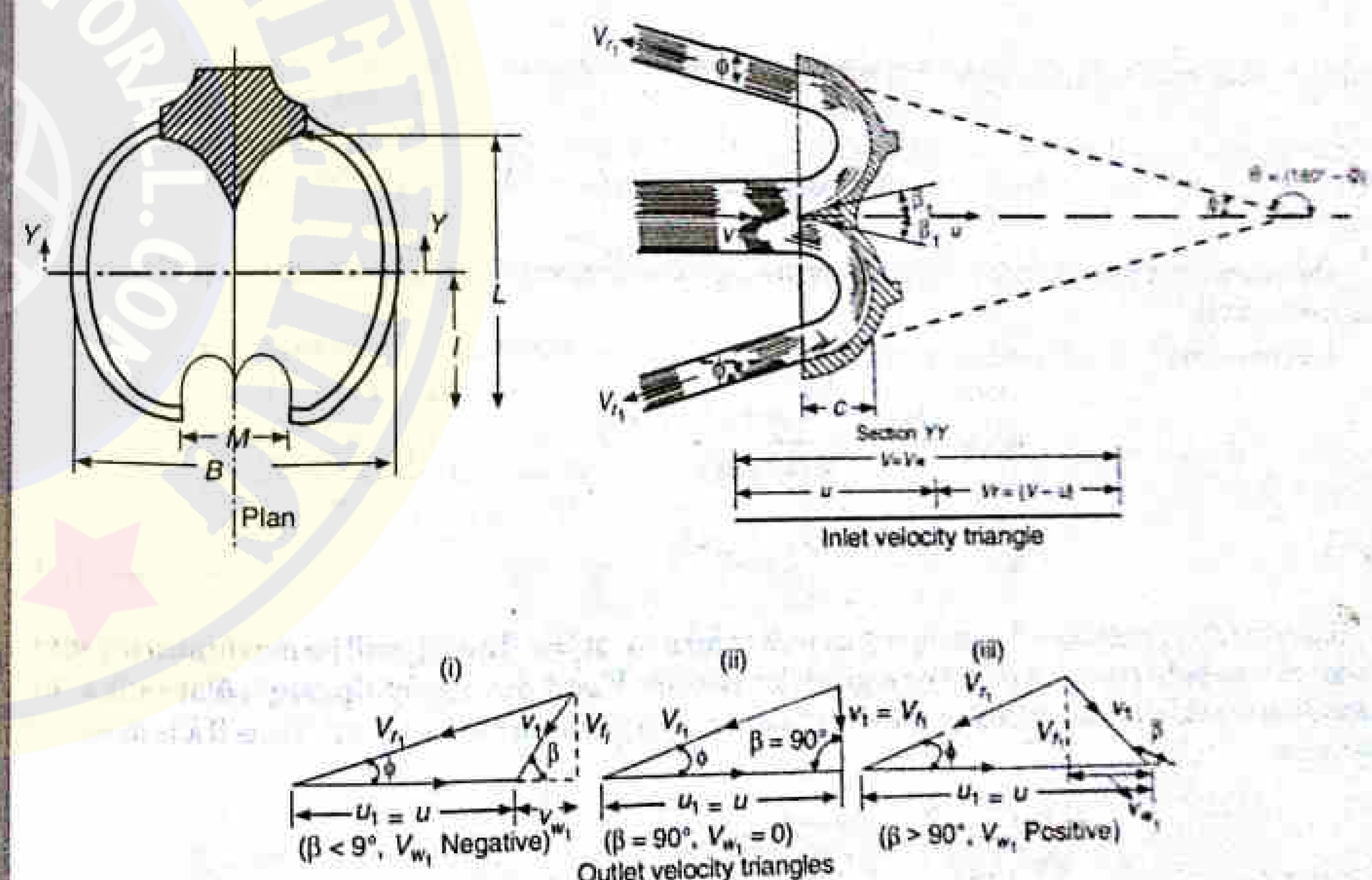


Figure 21.4 *Pelton wheel bucket and velocity-triangles*

the outlet tips of the bucket are at the same radial distance, the tangential velocity of the bucket at both the tips is same, i.e., $u = u_1$. Further the relative velocity V_{r1} with which the jet leaves the bucket will be somewhat less than the initial relative velocity V_r at the inlet tip. This is because, although the inner faces of the buckets are polished so as to minimize frictional losses as water flows over them, such losses cannot be completely eliminated. In addition to this some loss of energy will also take place as the jet strikes the splitter. These losses of energy reduce the relative velocity between the jet and the bucket, and hence

$$V_{r1} = k(V_r) = k(V-u)$$

where k is a fraction slightly less than unity.

Now from the outlet velocity triangle (i) of Fig. 21.4

$$V_{w1} = (V_{r1}\cos\phi - u_1) = (kV_r\cos\phi - u)$$

and from the outlet velocity triangle (iii) of Fig. 21.4

$$V_{w1} = (u_1 - V_{r1}\cos\phi) = (u - kV_r\cos\phi)$$

where $\phi = (180° - \theta)$ is the angle of the bucket at the outlet tip.

However, if the losses are neglected then $k = 1$. Also for outlet velocity triangle (ii) $V_{w1} = 0$.

If W is the weight of water per second which strikes the buckets then, as shown in Chapter 20, the work done per second on the wheel is given as

$$\text{Work done} = \frac{W}{g}\left[V_w u \pm V_{w_1} u_1\right]$$

$$= \frac{W}{g}[V + kV_r\cos\phi - u]\,u$$

$$= \frac{W}{g}[(V-u)(1+k\cos\phi)]\,u \qquad ...(21.10)$$

The energy supplied to the wheel is in the form of kinetic energy of the jet which is equal to $(WV^2/2g)$ per second.

Therefore the hydraulic efficiency of Pelton wheel is given by

$$\eta_h = \frac{(W/g)[(V-u)(1+k\cos\phi)u]}{(WV^2/2g)}$$

or

$$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2} \qquad ...(21.11)$$

Equation 21.11 indicates that for $u = 0$, as well as for $u = V$, $\eta_h = 0$. Thus η_h will be maximum for some value of u in between O and V. For a given jet velocity V and the bucket tip angle ϕ at outlet, the condition for maximum efficiency may be obtained by considering $(d\eta_h/du) = 0$. Thus if k is assumed constant

$$\frac{d(\eta_h)}{du} = \frac{2(1+k\cos\phi)}{V^2}(V-2u) = 0$$

or

$$u = \left(\frac{V}{2}\right) = 0.5\,V \qquad ...(21.12)$$

Thus the hydraulic efficiency is maximum when the bucket speed is equal to half the velocity of the jet. Substituting this condition in Eq. 21.11, the maximum hydraulic efficiency is obtained as

$$(\eta_h)_{max} = \frac{1}{2}(1+k\cos\phi) \qquad ...(21.13)$$

Now if $k = 1$, then Eq. 21.13 indicates that the maximum value of η_h will be equal to 1 (or 100%) when $\phi = 0$ or $\theta = 180°$ i.e., the buckets are so shaped that the jet gets deflected through 180°. This is however theoretical maximum value of η_h. The actual maximum value of η_h will be slightly less and it varies from 0.9 to 0.94 (or 90 to 94%). This is so because the actual value of k is not equal to one but is slightly less. Further in actual practice θ can not be made equal to 180°, because in that case the jet leaving the bucket will strike the back of the bucket just following it, thus exerting a retarding force on it. Hence in order that the outgoing jet keeps clear of the following bucket, the bucket tip angle ϕ at outlet is usually kept ranging from 10° to 20°, (the average value being 15°) so that the jet gets deflected through 160° to 170°. The angle ϕ is also known as side clearance angle.

If there is no loss of energy as the water flows over the buckets then the work done per second on the Pelton wheel (or the power output of the Pelton wheel) may also be expressed as

$$\text{Work done} = \frac{W}{2g}(V^2 - V_1^2) \qquad ...(21.14)$$

Accordingly the hydraulic efficiency of Pelton wheel may also be expressed as

$$\eta_h = \frac{[(W/2g)(V^2 - V_1^2)]}{(W/2g)V^2} = \frac{V^2 - V_1^2}{V^2} \qquad ...(21.15)$$

By substituting the value of V_1 obtained from the outlet velocity triangle, Eq. 21.15 becomes exactly similar to Eq. 21.11 with $k = 1$.

The loss of head as the water flows through the buckets of the Pelton wheel may be obtained by applying the Bernoulli's equation between the inlet and the outlet tips of the bucket. Thus

$$\frac{V^2}{2g} = \frac{V_w u \pm V_{w_1} u_1}{g} + \frac{V_1^2}{2g} + h_L$$

where h_L is the loss of head in the buckets.

From the velocity triangles at the inlet and the outlet tips of the bucket

$$\frac{V_w u \pm V_{w_1} u_1}{g} = \left[\frac{V^2 - V_1^2}{2g} + \frac{u^2 - u_1^2}{2g} + \frac{V_{r1}^2 - V_r^2}{2g}\right]$$

$$= \left[\frac{V^2 - V_1^2}{2g} + \frac{V_{r1}^2 - V_r^2}{2g}\right] \text{(since } u = u_1\text{)}$$

Thus by substitution, we get

$$h_L = \frac{V_r^2 - V_{r1}^2}{2g} = \frac{V_r^2}{2g}(1-k^2)$$

$$= \frac{(V-u)^2}{2g}(1-k^2) \qquad ...(21.16)$$

If P is the power available at the shaft then the mechanical efficiency of Pelton wheel is given by

$$\eta_m = \frac{P}{W\left[\dfrac{(V-u)u(1+k\cos\phi)}{g}\right]} \qquad ...(21.17)$$

The volumetric efficiency is given by Eq. 21.7 and its value for Pelton wheel ranges from 97 to 99%.

The overall efficiency is given by Eq. 21.8 and for large Pelton wheels the overall efficiency of 85 to 90% may usually be achieved.

21.7 WORKING PROPORTIONS OF PELTON WHEEL

(i) The ideal velocity of jet usually known as spouting velocity = $(\sqrt{2gH})$ where H is the net head. However, the actual velocity of the jet is slightly less, due to friction loss in the nozzle. Thus

$$V = (K_v \text{ or } C_v)\sqrt{2gH} \qquad ...(21.18)$$

where K_v or C_v is the coefficient of velocity for the nozzle with its value ranging from 0.97 to 0.99.

(ii) As obtained earlier for maximum η_h the velocity of wheel u at pitch circle is equal to 0.5V. However, in actual practice the maximum efficiency occurs when the value of u is about 0.46V. Moreover, it is convenient to express u in terms of H, in the form of an expression $u = K_u(\sqrt{2gH})$, where K_u is known as *speed ratio*. Thus considering $V = 0.98(\sqrt{2gH})$, we obtain

$$u = 0.46\,V = 0.45(\sqrt{2gH}) \qquad ...(21.19)$$

In practice the value of K_u ranges from 0.43 to 0.47.

(iii) Angle through which jet of water gets deflected in buckets = 165°, unless otherwise stated.

(iv) Least diameter d of the jet is given by

$$d = \left[\frac{4Q}{\pi K_v(\sqrt{2gH})}\right]^{1/2} \qquad ...(21.20)$$

where Q is the discharge through the jet in m³/s.

Taking $K_v = 0.98$, we obtain

$$d = 0.542\left(\frac{Q}{\sqrt{H}}\right)^{1/2} \text{ metres} \qquad ...(21.20\,a)$$

(v) *Mean diameter* or the pitch diameter D of the Pelton wheel may be obtained as follows. If the wheel rotates at N r.p.m., then $u = (\pi DN/60)$. Thus

$$D = \frac{60u}{\pi N} = \frac{60(K_u\sqrt{2gH})}{\pi N} \qquad ...(21.21)$$

(vi) The ratio of pitch diameter D of the wheel to the jet diameter d is known as *jet ratio* and is represented by m i.e., $m = (D/d)$. The jet diameter is an important parameter in the design of a Pelton wheel. For maximum efficiency the jet ratio should be from 11 to 14 and normally a jet ratio of 12 is adopted in practice. A smaller value of m results in either too close a spacing of the buckets or too few buckets for the whole jet to be used. A larger value of m results in a more bulky installation. However, in extreme cases a value of m as low as 7 and as high as 110 has been used.

(vii) Some of the main dimensions of the bucket of Pelton wheel as shown in Fig. 21.4 expressed in terms of the jet diameter are as noted below:

$B = (4 \text{ to } 5)\,d$; $\qquad L = (2.4 \text{ to } 3.2)\,d$;

$C = (0.81 \text{ to } 1.05)\,d$; $\qquad l = (1.2 \text{ to } 1.9)\,d$;

$M = (1.1 \text{ to } 1.25)\,d$; $\qquad$ Angle $\phi = 10°$ to $20°$

Angle $\beta_1 = 5°$ to $8°$.

(viii) The number of buckets for a Pelton wheel should be such that the jet is always completely intercepted by the buckets so that volumetric efficiency of the turbine is very close to unity. The number of buckets is usually more than 15. Certain empirical formulae have been developed for determining the number of buckets. One such formula which is widely used has been given by Taygun according to which the number of buckets Z is approximately given by

$$Z = \left(\frac{D}{2d} + 15\right) = (0.5\,m + 15) \qquad ...(21.22)$$

This equation has been found to hold good for all values of m ranging from 6 to 35.

21.8 DESIGN OF PELTON TURBINE RUNNER

A Pelton turbine runner is required to be designed to develop a known power P, when running at a known speed N r.p.m. under a known head H. The probable values of η_o, K_v, K_u and m are assumed. The various steps involved in the design are as follows:

(i) Determine the required discharge Q from the relation

$$P = \eta_o(WH) = \eta_o(wQH)$$

(ii) From Eq. 21.18 calculate the velocity V of the jet.

(iii) Calculate the total area of the jets required by using the relation $a = (Q/V)$.

(iv) Calculate the pitch circle diameter D from Eq. 21.21.

(v) Calculate the required diameter d of the jet from the relation $m = (D/d)$ and also calculate the corresponding area of the jet.

(vi) Obtain the number of jets required by dividing the total area of jets obtained in step (iii) by the area of each jet obtained in step (v).

(vii) The fractional number of jets obtained in step (vi) may be rounded up to the appropriate integral number and the corresponding diameter of each jet to be actually provided may be calculated. Care should however be taken to have the actual value of m very nearly equal to the assumed value.

(viii) Calculate the number of buckets to be provided from Eq. 21.22 and the bucket dimensions from the relevant equations.

21.9 MULTIPLE JET PELTON WHEEL

The power developed by a Pelton wheel provided with a single jet is usually quite low. This is so because on account of the restrictions of the jet velocity, wheel speed and the jet ratio, a single jet cannot be made big enough to develop any desired power. The amount of power developed by a single runner of a Pelton wheel turbine may however be increased by providing more than one jet spaced evenly around the same runner. The nozzles must never be spaced so closely that water issued from one jet after striking the runner interferes with another jet. As such the maximum number of jets so far used with a single runner of some large units is six. A Pelton wheel having more than one jet spaced around its runner is called *multiple jet Pelton wheel*. If P is the power developed by a Pelton wheel when working under head H and having one jet only, then the power developed by the same Pelton wheel will be (nP), if n jets are used for its working under the same head.

Sometimes even if by using more number of jets for a single runner, the required power is not developed then a number of runners mounted on a common shaft may be used. In some cases a combination of the above two systems may be used, i.e., a number of multiple jet wheels may be mounted on the same shaft.

21.10 RADIAL FLOW IMPULSE TURBINE

For a radial flow impulse turbine the inlet velocity triangle is not a straight line and hence

$$\text{Work done} = \frac{W}{g}(V_w u \pm V_{w_1} u_1)$$

Further if there is no loss of energy in the runner vanes then the work done may also be expressed by Eq. 21.14. Thus equating the two, we get

$$\frac{W}{g}(V_w u \pm V_{w_1} u_1) = \frac{W}{2g}(V^2 - V_1^2)$$

From the velocity triangles at the inlet and the outlet tips of a radial vane it can be shown that

$$[V_w u \pm Vw_1 u_1] = \left[\frac{V^2 - V_1^2}{2} + \frac{u^2 - u_1^2}{2} + \frac{V_{r1}^2 - V_r^2}{2}\right]$$

Thus by substituting this value in the above expression it becomes

$$\frac{u^2 - u_1^2}{2g} + \frac{V_{r1}^2 - V_r^2}{2g} = 0$$

or

$$\frac{V_{r_1}^2}{2g} = \frac{V_r^2}{2g} - \left(\frac{u^2 - u_1^2}{2g}\right) \quad \text{...(21.23)}$$

The second term on the right hand side of Eq. 21.23 represents the centrifugal head impressed on the water as it flows through the runner of a radial flow impulse turbine. For an outward flow turbine $u_1 > u$, then from Eq. 21.23, $V_{r1} > V_r$; and for an inward flow turbine $u_1 < u$ and hence $V_{r1} < V_r$. That is the centrifugal head increases the relative velocity of water in an outward flow turbine and decreases it in an inward flow turbine. As such a better control of speed can be enforced in the case of an inward flow turbine.

21.11 REACTION TURBINES

As stated earlier the principal distinguishing features of a reaction turbine are that only part of the total head of water is coverted into velocity head before it reaches the runner, and that the water completely fills all the passage in the runner. The pressure of water changes gradually as it passes through the runner. The two reaction type of turbines which are predominantly used these days are Francis turbine and Kaplan turbine, which are described below.

21.12 FRANCIS TURBINE

Figure 21.5 illustrates a Francis turbine which is a mixed flow type of reaction turbine. It is named in honour of James B. Francis (1815–92), an American Engineer, who was the first to develop an inward radial flow type of reaction turbine in 1849. Later on it was modified and the modern Francis turbine is a mixed flow type, in which water enters the runner radially at its outer periphery and leaves axially at its centre.

The water from the penstock enters a *scroll casing* (also called *spiral casing*) which completely surrounds the runner. The purpose of the casing is to provide an even distribution of water around the circumference of the turbine runner, maintaining an approximately constant velocity for the water so distributed. In order to keep the velocity of water constant throughout its path around the runner, the cross-sectional area of the casing is gradually decreased. The casing is made of a cast steel, plate steel, concrete or concrete and steel depending upon the pressure to which it is subjected. Out of these a plate steel scroll casing is commonly provided for turbines operating under 30 m or higher heads.

From the scroll casing the water passes through a *speed ring* or *stay ring* (see Fig. 21.5). The speed ring consists of an upper and a lower ring held together by series of fixed vanes called *stay vanes*. The number of stay vanes is usually taken as half the number of guide vanes. The speed ring has two functions to perform. It directs the water from the scroll casing to the guide vanes or wicket gates. Further it resists the load imposed upon it by the internal pressure of water and the weight of the turbine and the electrical generator and transmits the same to the foundation. The speed ring may be either of cast iron or cast steel or fabricated steel.

From the speed ring the water passes through a series of *guide vanes* or *wicket gates* provided all around the periphery of the turbine runner. The function of guide vanes is to regulate the quantity of water supplied to the runner and to direct water on to the runner at an angle appropriate to the design. The guide vanes are airfoil shaped and they may be made of cast steel, stainless steel or plate steel. Each guide vane is provided with two stems, the upper stem passes through the head cover and the lower stem seats in a bottom ring. By a system of levers and links, all the guide vanes may be turned about their stems, so as to alter the width of the passage between the adjacent guide vanes, thereby allowing a variable quantity of water to strike the runner. The guide vanes are operated either by means of a wheel (for very small units) or automatically by a governor.

The main purpose of the various components so far described is to lead the water to the runner with a minimum loss of energy. The runner of a Francis turbine consists of a series of a curved vanes (about 16 to 24 in number) evenly arranged around the circumference in the annular space between two plates. The vanes are so shaped that water enters the runner radially at the outer periphery and leaves it axially at the inner periphery. The change in the direction of flow of water, from radial to axial, as it passes through the runner, produces a circumferential force on the runner which makes the runner to rotate and thus contributes to the useful output of the runner. The runners are usually made up of cast iron, cast steel, mild steel or stainless steel. Often instead of making the complete runner of stainless steel, only those portions of the runner blades, which may be subjected to cavitation erosion, are made of stainless steel. This reduces the cost of the runner and at the same time ensures the operation of runner with a minimum amount of maintenance. The runner is keyed to a shaft which is usually of forged steel. The torque produced by the runner is transmitted to the generator through the shaft which is usually connected to the generator shaft by a bolted flange connection.

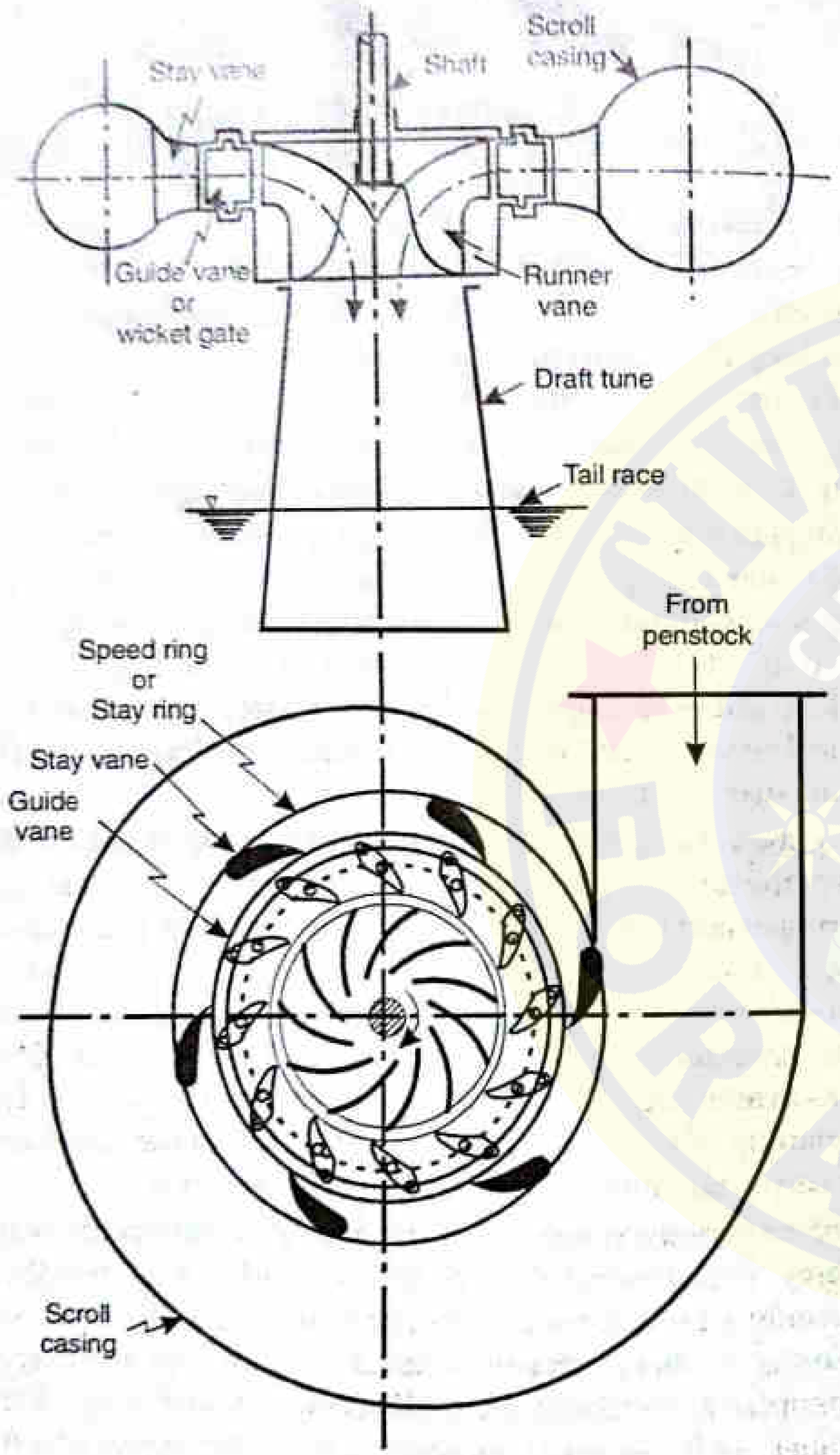
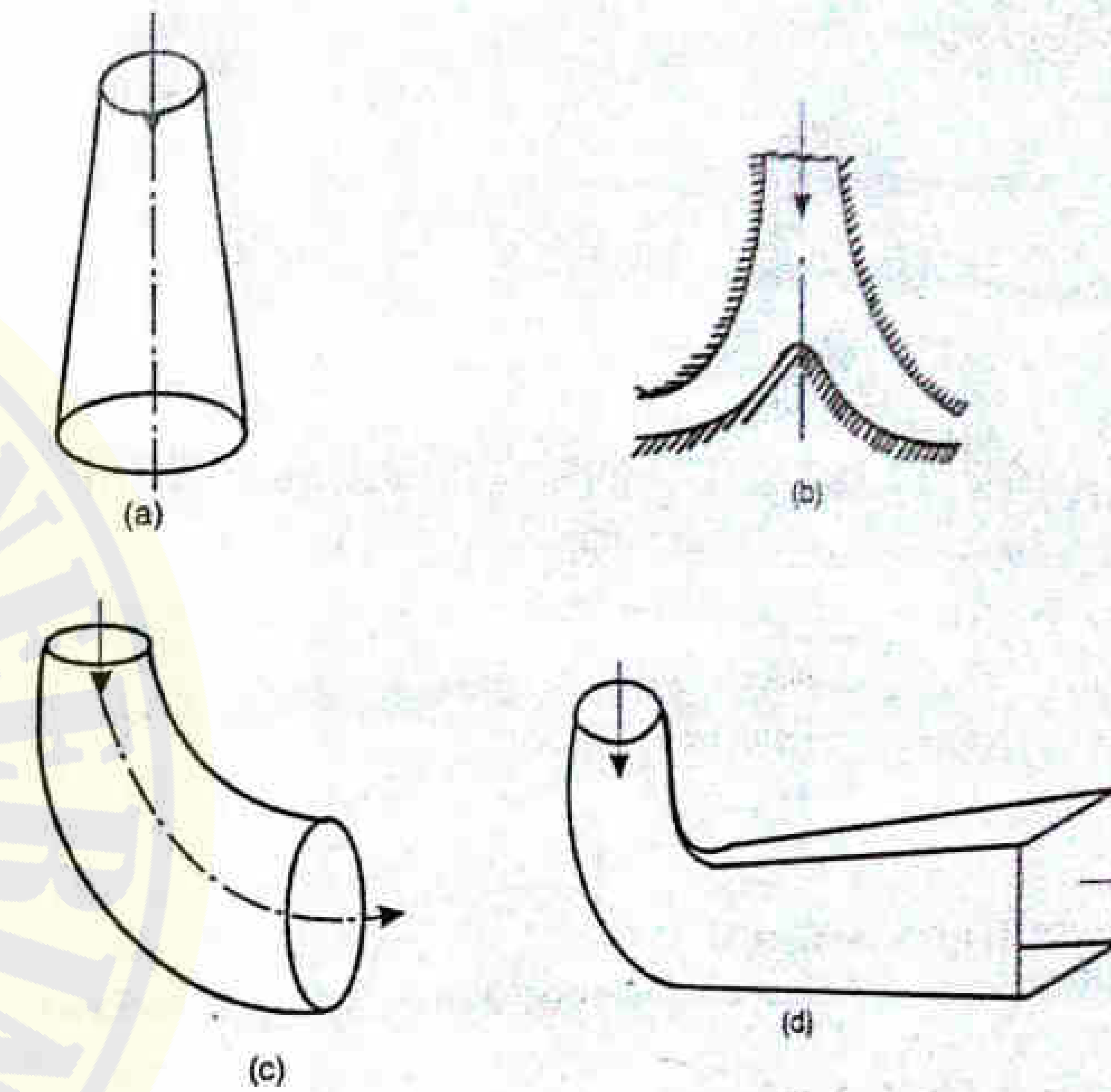


Figure 21.5 *Sectional arrangement of Francis turbine*

The water after passing through the runner flows to the tail race through a draft tube. A *draft tube* is a pipe or passage of gradually increasing cross-sectional area which connects the runner exit to the tail race. It may be made of cast or plate steel or concrete. It must be airtight and under all conditions of operation its lower end must be submerged below the level of water in the tail race. The draft tube has two purposes as follows:

(i) It permits a negative or suction head to be established at the runner exit, thus making it possible to instal the turbine above the tail race level without loss of head.

(ii) It converts a large proportion of velocity energy rejected from the runner into useful pressure energy i.e., it acts as a recuperator of pressure energy.



(a) Straight divergent tube
(b) Moody spreading tube (or hydraucone tube)
(c) Simple elbow tube .
(d) Elbow tube Having circular cross section at inlet and rectangular at outlet

Figure 21.6 *Different types of draft tubes*

Figure 21.6 shows the different types of draft tubes which are employed in the field to suit particular conditions of installation. Of these the types (a) and (b) are the most efficient, but the types (c) and (d) have an advantage that they require lesser excavation for their installation. It has been observed that for straight divergent type draft tube the central cone angle should not be more than 8°. This is so because if this angle is more than 8° the water flowing through the draft tube will not remain in contact with its inner surface, with the result that eddies are formed and the efficiency of the draft tube is reduced.

21.13 WORK DONE AND EFFICIENCIES OF FRANCIS TURBINE

If W is the weight of water per second which strikes the runner then as derived in Chapter 20 the work done per second on the runner may be expressed as

$$\text{Work done} = \frac{W}{g}[V_w u - Vw_1 u_1]$$

Evidently the maximum output under specified conditions is obtained by making the velocity of whirl at exit V_{w1} equal to zero.

Then

$$\text{Work done} = \frac{W}{g}(V_w u)$$

or Work done per unit weight of water

$$= \left(\frac{V_w u}{g}\right)$$

Now if H is the net head then the input energy per second for the runner = (WH).

Therefore hydraulic efficiency of the Francis turbine is given by

$$\eta_h = \frac{V_w u}{gH} \quad ...(21.24)$$

However, if V_{w1} is not equal to zero, then

$$\eta_h = \frac{V_w u - V_{w_1} u_1}{gH} \quad ...(21.24\,a)$$

The value of η_h ranges from 85 to 95%.

Again if P is the power available at the runner shaft then the mechanical efficiency is given by

$$\eta_m = \frac{P}{W\left(\frac{V_w u}{g}\right)} \quad ...(21.25)$$

However, if V_{w1} is not equal to zero, then

$$\eta_m = \frac{P}{W\left(\frac{V_w u - V_{w1} u_1}{g}\right)} \quad ...(21.25\,a)$$

Further the overall efficiency is given by Eq. 21.9 as

$$\eta_0 = \eta_h \times \eta_m = \frac{P}{(WH)}$$

The overall efficiency of a Francis turbine ranges from 80 to 90%.

Degree of reaction

Degree of reaction ρ, is defined as the ratio of pressure drop in the runner to the hydraulic work done on the runner. Thus is p and p_1 are the pressures at the inlet and the outlet of the runner, then

$$\rho = \frac{\left(\frac{p}{w} - \frac{p_1}{w}\right)}{\frac{V_w u - V_{w_1} u_1}{g}}$$

and if $V_{w_1} = 0$, then

$$\rho = \frac{\left(\frac{p}{w} - \frac{p_1}{w}\right)}{\frac{V_w u}{g}}$$

As indicated in Example 21.17 for no loss of head in the runner the degree of reaction can be expressed in terms of guide vane and runner vane angles. Thus for a given runner and set of guide vanes the degree of reaction is more or less constant if there is no loss of head in the runner. However, in actual practice because of the head loss in the runner the degree of reaction is not constant.

21.14 WORKING PROPORTIONS OF FRANCIS TURBINE

(i) The ratio of the width B of the runner to the diameter D of the runner (see Fig. 21.8) is represented by n, that is

$$n = (B/D) \quad ...(21.26)$$

The value of n ranges from 0.10 to 0.45.

(ii) The ratio of the velocity of flow V_f at the inlet tip of the vane to the spouting velocity $\sqrt{2gH}$ is known as *flow ratio* ψ, that is

$$\psi = \frac{V_f}{\sqrt{2gH}}\,;\ \text{or}\ V_f = \psi\sqrt{2gH} \quad ...(21.27)$$

The value of ψ ranges from 0.15 to 0.3.

(iii) In this case also the speed ratio K_u is defined as

$$K_u = \frac{u}{\sqrt{2gH}}\,;\ \text{or}\ u = K_u\sqrt{2gH} \quad ...(21.27\,a)$$

The value of K_u ranges from 0.60 to 0.90.

21.15 DESIGN OF FRANCIS TURBINE RUNNER

A Francis turbine runner is required to be designed to develop a known power P, when running at a known speed N r.p.m. under a known head H. The probable values of η_h, η_0, n and ψ are assumed. The design of the runner which involves the determination of the size and the vane angles is carried out as follows:

(i) Determine the required discharge Q from the relation

$$P = \eta_0 (WH) = \eta_0 (wQH)$$

(ii) If Z is the number of vanes in the runner, t is the thickness of the vane at inlet and B is the width of the wheel at inlet, then the area of flow section at the wheel inlet $= (\pi D - Zt)\, B = k\pi BD$, where k is a factor which allows for the thickness of the vanes. Then

$$Q = k\pi BDV_f = k\pi n D^2 V_f$$

Since $$B = nD. \qquad ...(21.28)$$

Thus assuming a suitable value of k, the diameter D and the width B of the runner can be determined. For the first approximation the vane thickness may be neglected in which case k may be assumed to be equal to unity.

(iii) The tangential velocity of the runner at inlet may be determined from the equation $u = (\pi DN/60)$.

(vi) The velocity of whirl V_w at inlet of the runner can be determined from the expression $\eta_h = (V_w u/gH)$.

(v) From the inlet velocity triangle, the guide vane angle α and the runner vane angle θ at inlet can be calculated from the expression $\tan\alpha = (V_f / V_w)$ and $\tan\theta = V_f/(V_w - u)$.

(vi) The runner diameter D_1 at the outlet end varies from $(1/3)\,D$ to $(2/3)\,D$ and usually it is taken equal to $(1/2)\,D$. Thus the tangential velocity of the runner at outlet may be determined from the expression $u_1 = (\pi D_1 N/60)$.

(vii) If t_1 and B_1 are respectively the thickness of the vane and the width of the runner at outlet, then

$$Q = (\pi D_1 - Zt_1)\, B_1 \times V_{f1} = k_1 \pi D_1 B_1 V_{f1} \qquad ...(21.29)$$

From Eq. 21.28 and 21.29

$$\frac{V_f}{V_{f_1}} = \frac{k_1 \pi B_1 D_1}{k\pi BD} \qquad ...(21.30)$$

Normally it is assumed that $V_f = V_{f1}$ and $k = k_1$, then $B_1 = 2B$ since $D_1 = (1/2)D$.

(viii) Generally the runner is designed to have the velocity of whirl V_{w1} at outlet equal to zero, i.e., $V_{w1} = 0$ and $\beta = 90°$. Then from outlet velocity triangle the runner vane angle ϕ at outlet may be determined from the expression $\tan\phi = (V_{f1}/u_1)$.

(ix) The number of runner vanes should be either one more or one less than the number of guide vanes, in order to avoid setting up of periodic impulse.

21.16 DRAFT TUBE THEORY

Refer Fig. 21.7 in which points 1, 2 and 3 have been considered at the runner entrance, runner exit and at the outlet end of the draft tube respectively. By applying Bernoulli's equation between points 1 and 2 the pressure and velocity heads at the inlet and the outlet ends of the runner may be obtained.

Applying Bernoulli's equation between points 2 and 3, we get

$$\frac{p_2}{w} + \frac{V_2^2}{2g} + z_2 = \frac{p_3}{w} + \frac{V_3^2}{2g} + z_3 + h_f \qquad ...(21.31)$$

where p_2, p_3 and V_2, V_3 are the pressures and velocities at points 2 and 3 respectively, and h_f is the loss of head in the draft tube. From Eq. 21.31, we have

$$\frac{p_2}{w} = \frac{p_3}{w} - (z_2 - z_3) - \left(\frac{V_2^2 - V_3^2}{2g}\right) + h_f$$

But $$\frac{p_3}{w} = \frac{p_a}{w} + h$$

$$\therefore \quad \frac{p_2}{w} = \frac{p_a}{w} - (z_2 - z_3 - h) - \left(\frac{V_2^2 - V_3^2}{2g}\right) + h_f$$

Since $(z_2 - z_3 - h) = H_s$, the height of runner exit above tail race level, thus

$$\frac{p_2}{w} = \frac{p_a}{w} - \left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + h_f \qquad ...(21.32)$$

It may be seen from Eq. 21.32 that the pressure at the runner exit is suction pressure, that is below atmospheric. H_s is known as static suction head and $[(V_2^2 - V_3^2)/2g]$ is known as dynamic suction head. Generally h_f is expressed as $h_f = k\,(V_2^2 - V_3^2)/2g$, then

$$\frac{p_2}{w} = \frac{p_a}{w} - \left[H_s + (1-k)\frac{V_2^2 - V_3^2}{2g}\right] \qquad ...(21.33)$$

Now if the draft tube is made cylindrical then $V_2 = V_3$; and if the frictional loss is neglected, then Eq. 21.32 indicates that the pressure head at the runner exit would be less than atmospheric pressure by an amount H_s, the height of the runner exit above the tail race. Hence with such a draft tube the turbine would not loose head H_s because of equal reduction in pressure head at the runner exit. On the other hand if a draft tube of gradually increasing cross-section is used, then the value of pressure at runner

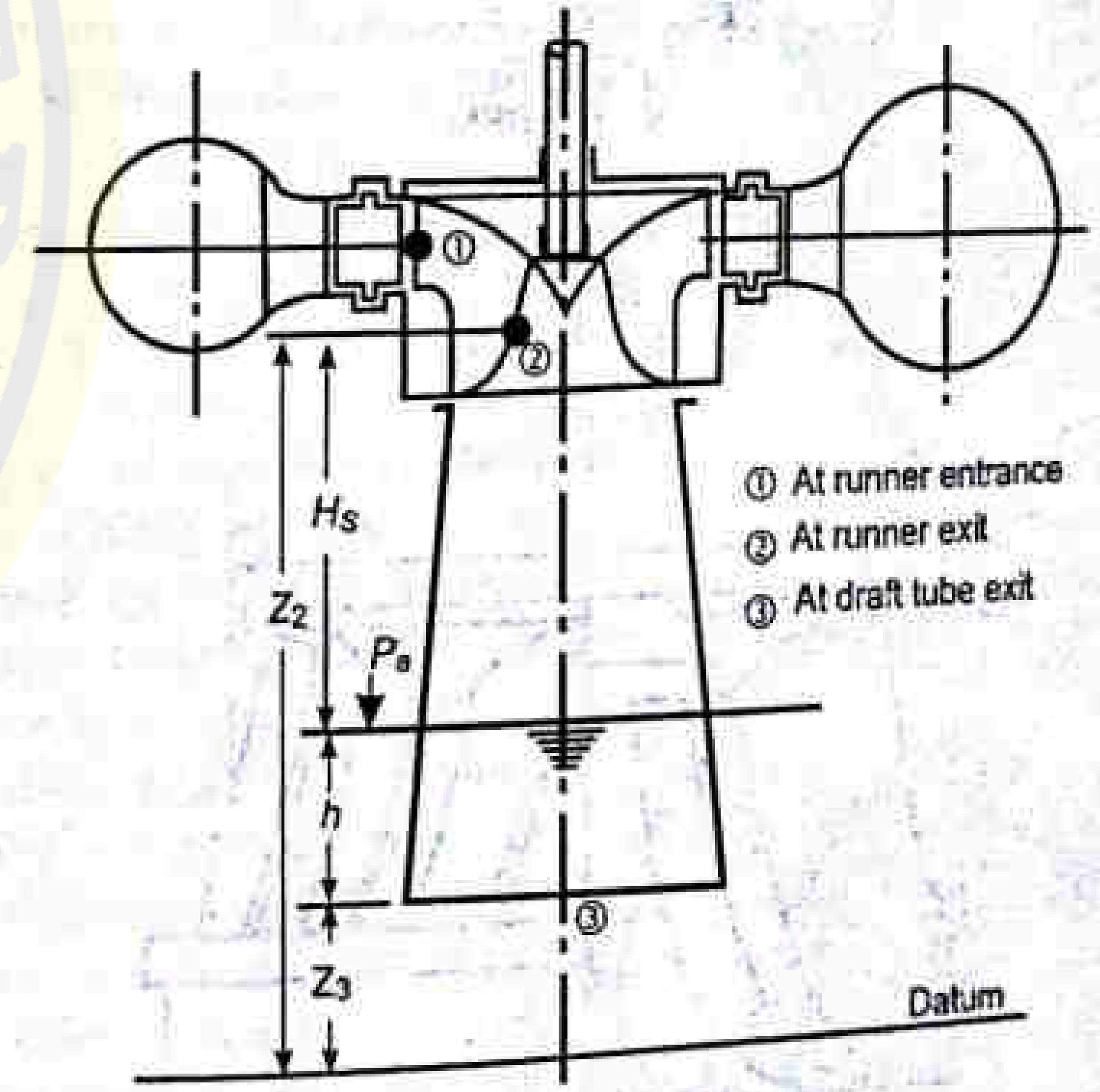


Figure 21.7 *Draft tube theory*

exit is further reduced by an amount equal to $(1-k)\,[(V_2^2 - V_3^2)/2g]$. In other words such a draft tube causes a large portion of kinetic head to be converted into pressure head.

The efficiency of a draft tube is defined as

$$\eta_d = \frac{\text{Actual regain of pressure head}}{\text{Velocity head at entrance to draft tube}}$$

The actual regain of pressure head

$$= \left[\frac{V_2^2 - V_3^2}{2g}\right] - h_f = (1-k)\left(\frac{V_2^2 - V_3^2}{2g}\right)$$

$$\therefore \quad \eta_d = \frac{[(V_2^2 - V_3^2)/2g] - h_f}{(V_2^2/2g)} = \frac{(1-k)[(V_2^2 - V_3^2)/2g]}{(V_2^2/2g)} \qquad ...(21.34)$$

21.17 SHAPE OF FRANCIS TURBINE RUNNER AND DEVELOPMENT OF KAPLAN TURBINE RUNNER

A Francis turbine runner of given diameter, when required to develop certain power P, under a low head, should be so designed that it admits comparatively large quantity of water. This can be done either by increasing the flow ratio ψ or the ratio n (= B/D). An increase in the value of ψ means a higher

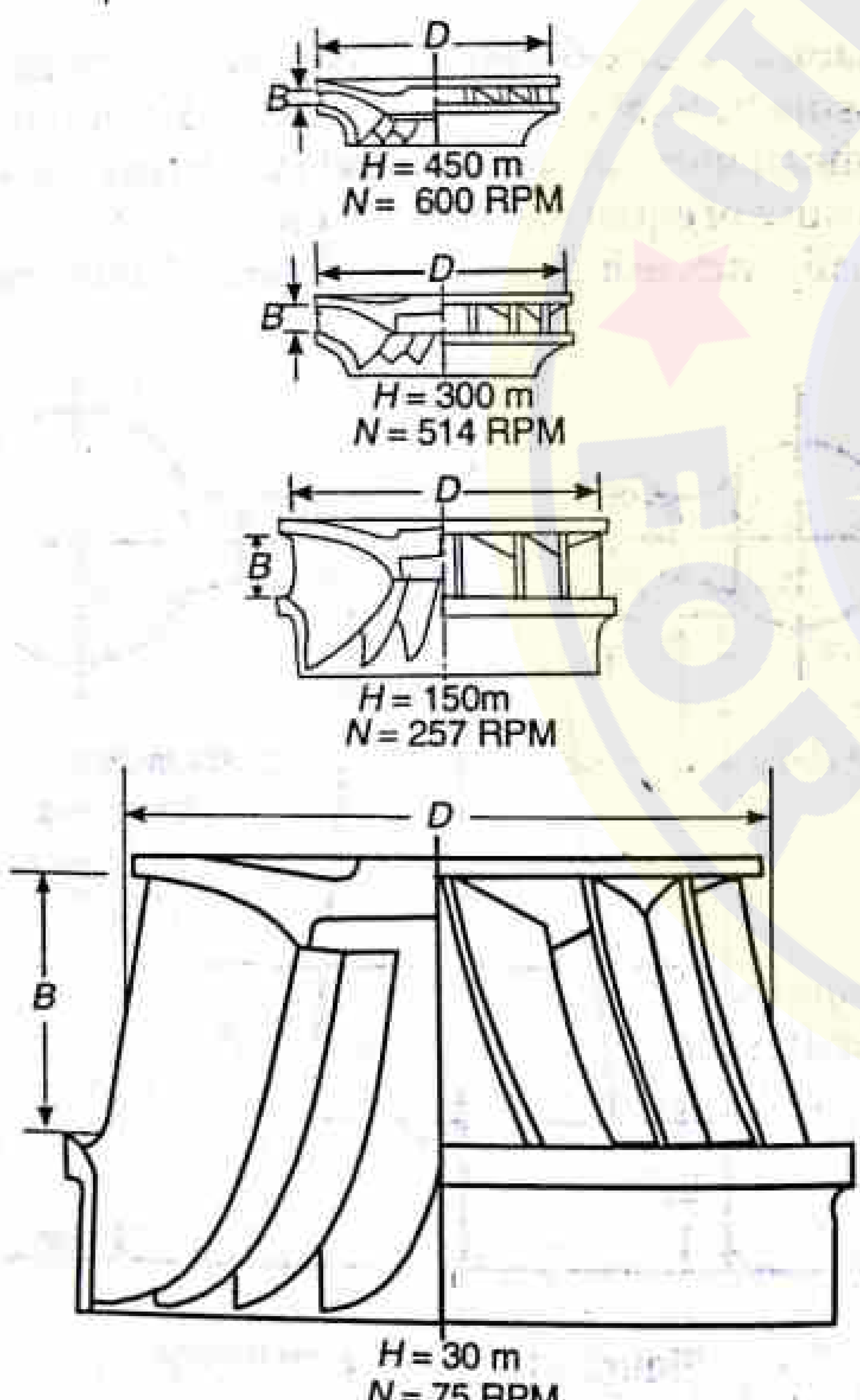


Figure 21.8 *Relative sizes of Francis turbine to produce 73550 W (105 h.p.)*

value of V_f which will results in excessive losses in the form of velocity energy rejected from the runner. As such ψ cannot be increased to any amount. On the other hand an increase in the value of n results in a higher value of B (for a given value of D), which will yield a larger inlet area. But a larger inlet area must be accompanied by a larger outlet area for all the water to escape freely. A larger outlet area is, however, obtained by making the discharge at the runner exit to be axial and hence the modern Francis turbine runners are of 'mixed flow type'. Figure 21.8 shows the relative sizes and shapes of the Francis turbine runners required to develop 73550 W (10^5 h.p.) under varying heads. It will be observed that as the head for the turbine is reduced and hence the discharge is increased (in order to develop the required power) the runner size considerably increases and also the shape of the runner has to be such that the flow is more and more axial right from the entrance to the runner. Thus as the quantity of water flowing through the runner increases it has to be so designed that it has more and more axial flow. Since in the case of a Kaplan turbine a large quantity of water is required to be passed, it is so designed that the flow is purely axial right from the inlet section to the outlet section of the runner.

21.18 KAPLAN TURBINE

A Kaplan turbine is a type of propeller turbine which was developed by the Austrian engineer V. Kaplan (1876–1934). It is an axial flow turbine, which is suitable for relatively low heads, and hence requires a large quantity of water to develop large amount of power. It is also a reaction type of turbine and hence it operates in an entirely closed conduit from the head race to the tail race.

From Fig. 21.9 it will be seen that the main components of a Kaplan turbine such as scroll casing, stay ring, arrangement of guide vanes, and the draft tube are similar to those of a Francis turbine. Between the guide vanes and the runner the water in a Kaplan (or propeller) turbine turns through a right-angle into the axial direction and then passes through the runner. The runner of a Kaplan (or propeller) turbine has four or six (or eight in some exceptional cases) blades and it closely resembles a ship's propeller. The blades (or vanes) attached to a hub or boss are so shaped that water flows axially through the runner. Ordinarily the runner blades of a propeller turbine are fixed, but the Kaplan turbine runner blades can be turned about their own axis, so that their angle of inclination may be adjusted while the turbine is in motion. This adjustment of the runner blades is usually carried out automatically by means of a servomotor operating inside the hollow coupling of turbine and generator shaft. When both guide-vane angle and runner-blade angle may thus be varied, a high efficiency can be maintained over a wide range of operating conditions. In other words even at part load, when a lower discharge is flowing through the runner, a high efficiency can be attained in the case of a Kaplan turbine. It may be explained with the help of Fig. 21.10, in which inlet and outlet velocity triangles for a Kaplan turbine runner working at constant

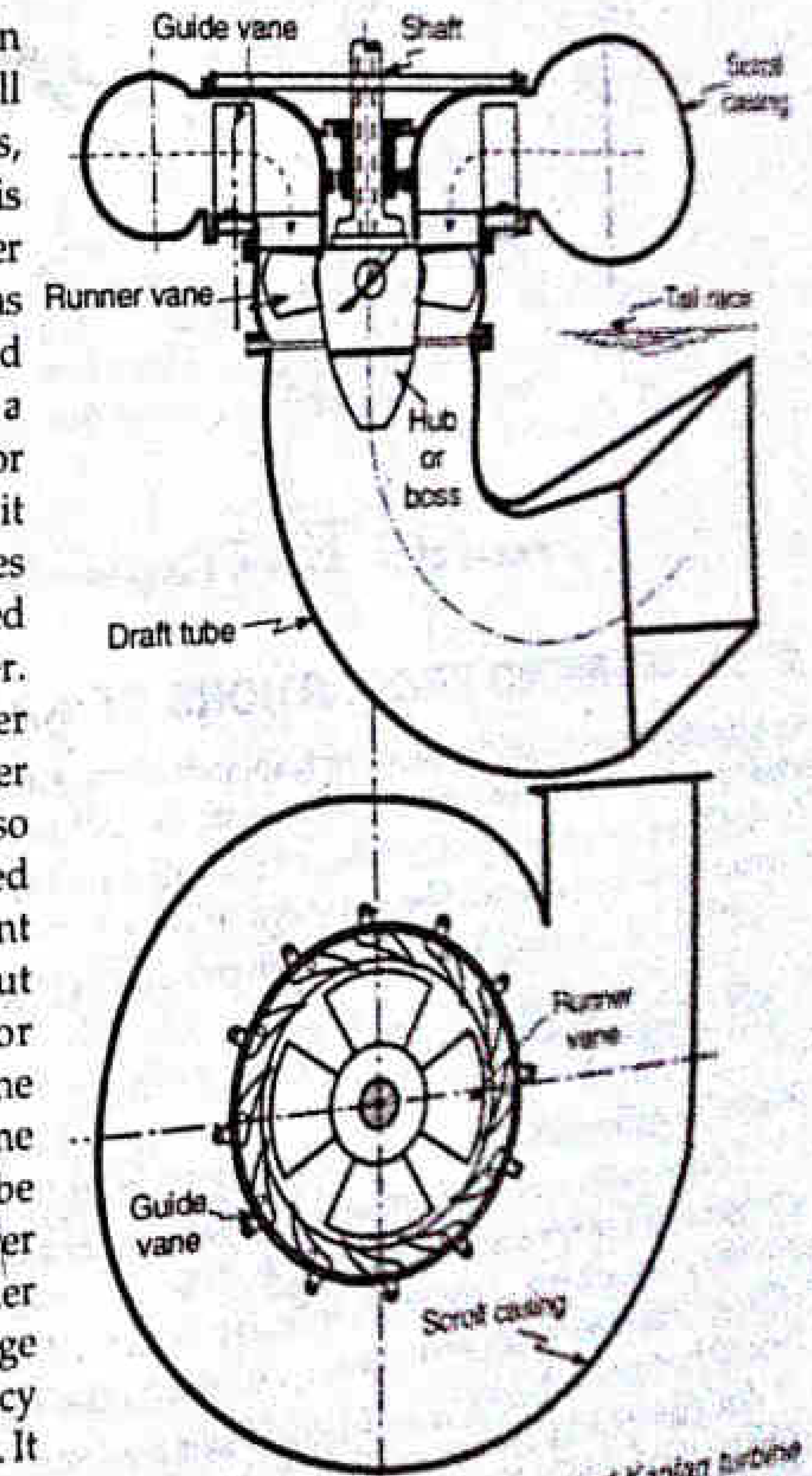


Figure 21.9 *Sectional arrangement of Kaplan turbine*

speed under constant head at *full load* and at *part load* are shown. It will be observed that although the corresponding changes in the flow through the turbine runner does affect the shape of the velocity triangles, yet as the blade angles are simultaneously adjusted, the water under all the working conditions flows through the runner blades without shock. As such the eddy losses which are inevitable in Francis and propeller turbines are almost completely eliminated in a Kaplan turbine.

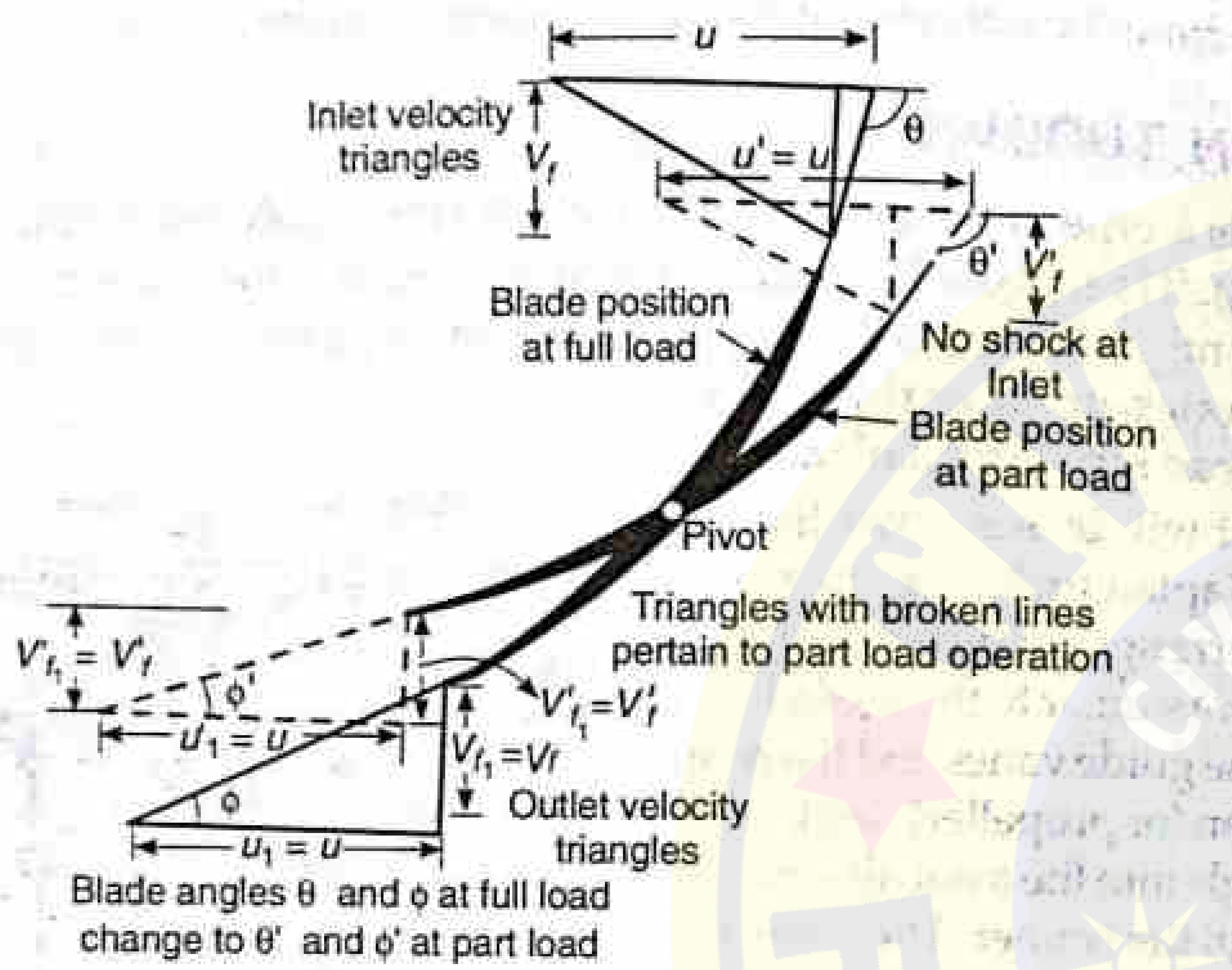


Figure 21.10 *Velocity triangles for a Kaplan turbine runner blade*

21.19 WORKING PROPORTIONS OF KAPLAN TURBINE

In general the main dimensions of Kaplan turbine runners are established by a procedure similar to that for a Francis turbine runner. However, the following are the main deviations:

(i) Choose an appropriate value of the ratio $n = (d/D)$, where d is hub or boss diameter and D is runner outside diameter. The value of n usually varies from 0.35 to 0.60.

(ii) The discharge Q flowing through the runner is given by

$$Q = \frac{\pi}{4}(D^2 - d^2)V_f = \frac{\pi}{4}(D^2 - d^2)\psi\sqrt{2gH} \quad ...(21.35)$$

The value of flow ratio ψ for a Kaplan turbine is around 0.70.

(iii) The runner blades of Kaplan turbine runner are warped or twisted, the blade angle being greater at the outer tip than at the hub. This is because the peripheral velocity of the blades being directly proportional to radius, it will vary from section to section along the blade, and hence in order to have shock free entry and exit of water the blades with angles varying from section to section will have to be designed.

The expressions for the work done and the efficiencies of Kaplan turbine are same as those for Francis turbine.

21.20 NEW TYPES OF TURBINES

A brief description of some of the new types of turbines is given below.

1. Deriaz or Diagonal Turbine. Figure 21.11 illustrates a Deriaz (or Diagonal) Turbine which is a reaction turbine. It is named in honour of its inventor P. Deriaz. This type of turbine is intermediate between the mixed-flow and the axial-flow turbines, because the flow of water as it passes through the

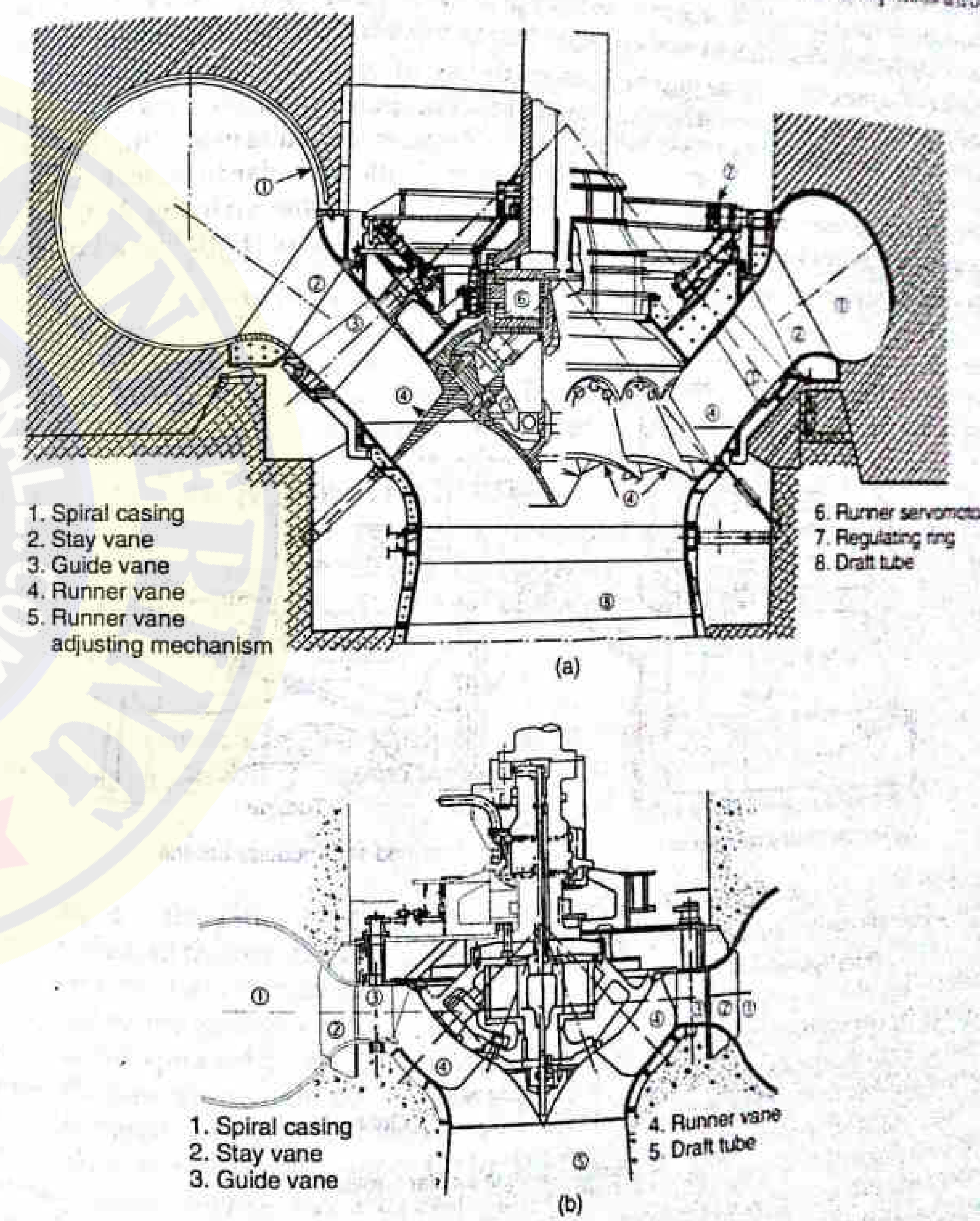
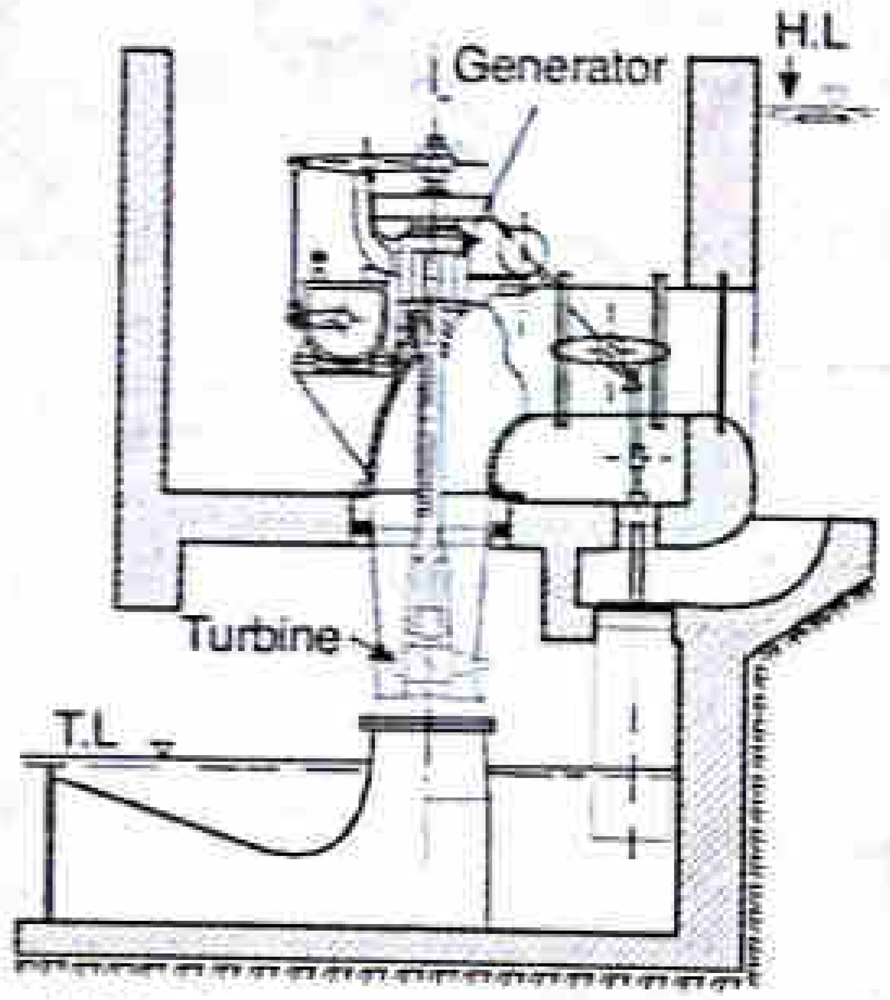


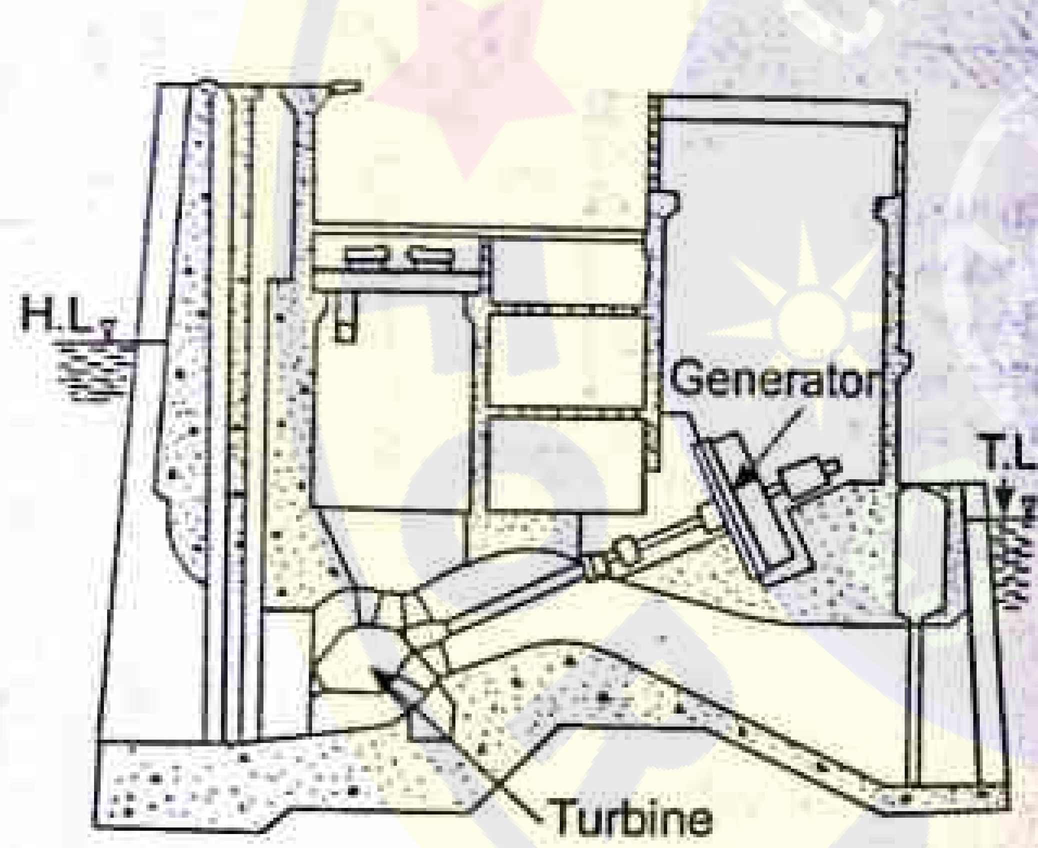
Figure 21.11 *Deriaz (or Diagonal) turbine*

runner is at an angle of 45° to the axis and hence it is also known as Diagonal turbine. It can be used under heads ranging from 30 m to 150 m. Alike Kaplan turbines the runner blades of these turbines are also adjustable. The other components such as scroll (or spiral) casing, stay vanes, guide vanes and draft tube are same as those in the case of other reaction turbines. As shown in Fig. 21.11 (a) generally in the case of a Deriaz turbine the stay vanes as well as the guide vanes are also inclined as a result of which passages, unlike in conventional Kaplan turbines, are straight. However, some Deriaz turbines are also provided with stay vanes and guide vanes similar to those provided for conventional Kaplan turbines as shown in Fig. 21.11 (b). The Deriaz runner is so shaped that it can be used both as a turbine as well as a pump and hence it may be classified as a reversible type turbine. As such Deriaz turbines are quite suitable for pumped storage hydropower plants (See Chapter 27). In some of the Western countries these turbines have been installed but in India no Deriaz turbine has been installed so far.

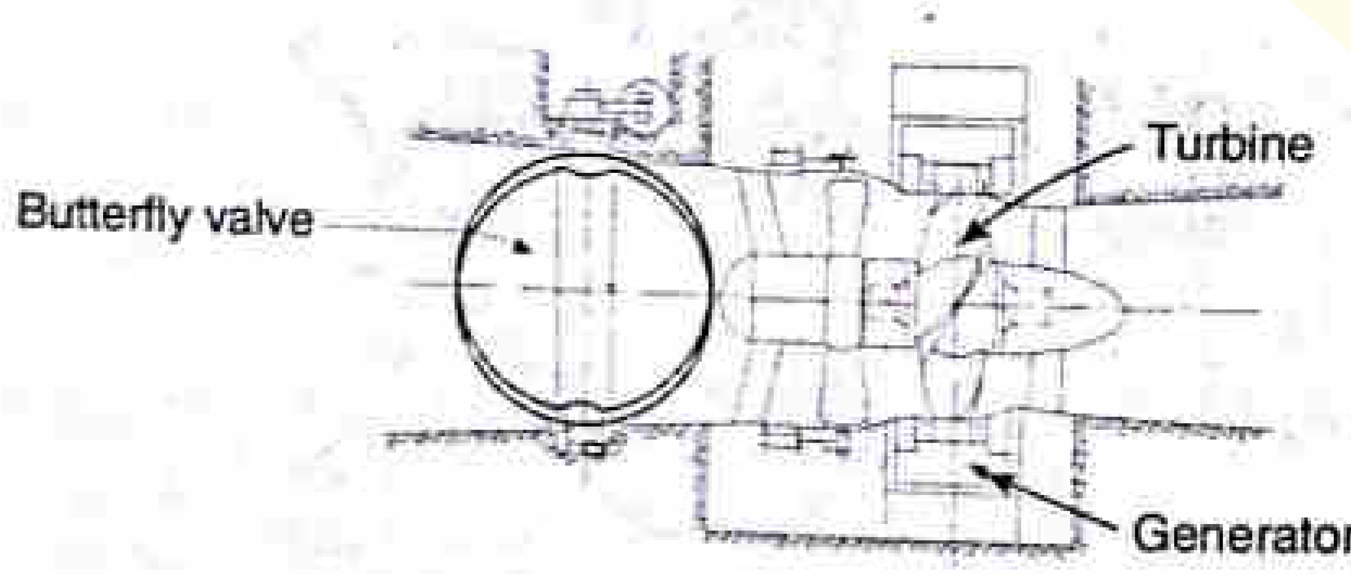
2. Tubular Turbine. A Tubular turbine is an axial flow turbine with either adjustable or non-adjustable runner vanes and hence it is similar to Kaplan or propeller turbines. However, in a tubular turbine the scroll casing is not provided but the runner is placed in a tube extending from the head water to the tail water as shown in Fig. 21.12, and hence it is called tubular turbine. It is a low head turbine which can be used under heads ranging from 3 m to 15 m. As shown in Fig. 21.12 the tubular turbines may have either vertical or inclined or horizontal disposition of shaft. The tubular turbines are quite suitable for tidal power plants (See Chapter 27).



(a) Vertical-shaft tubular turbine



(b) Inclined-shaft tubular turbine



(c) Horizontal-shaft tubular turbine

Figure 21.12 *Tubular turbine*

3. Bulb Turbine. It is same as tubular turbine with horizontal disposition of shaft in which the generator is housed in a bulb-shaped watertight casing as shown in Fig. 21.13 and hence it is called bulb turbine. The bulb together with the turbine runner form an integral unit. The outer surface of the bulb is streamlined so as to form a passage for the flow of water from the head pond to the turbine with minimum loss of energy. The bulb could be either upstream or downstream of the runner but in most of the cases it is on the upstream of the runner. The bulb turbines are also quite suitable for tidal power plants.

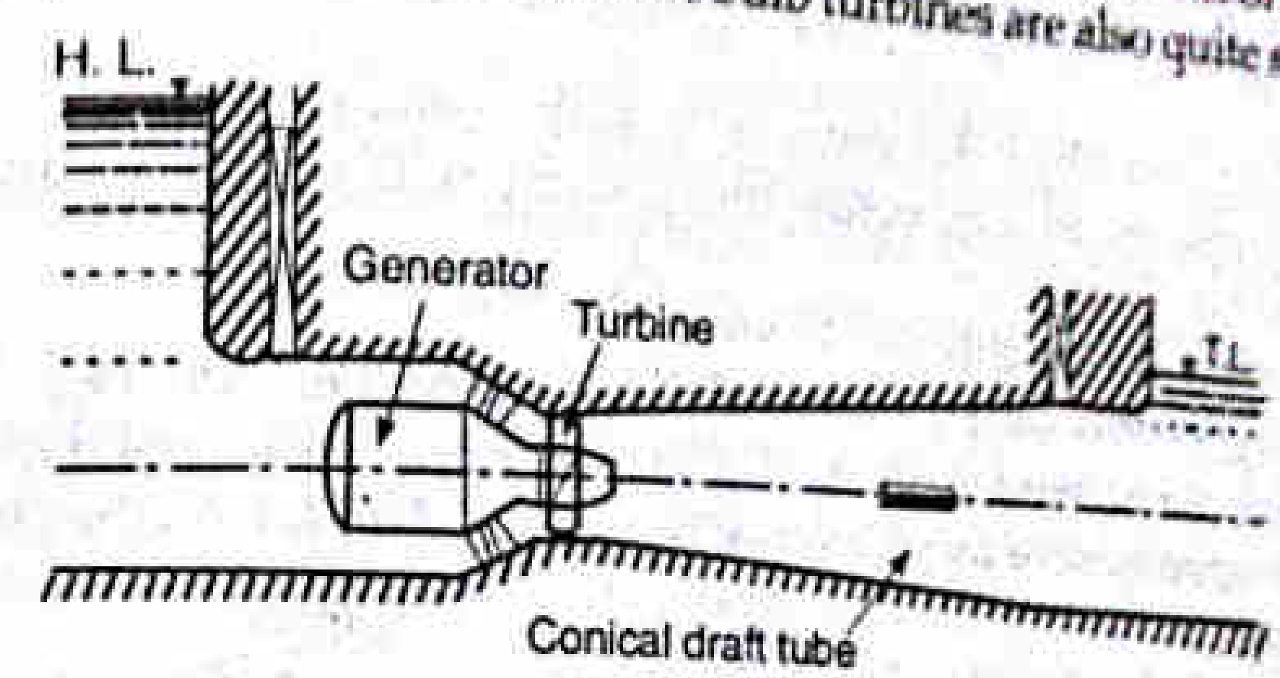


Figure 21.13 *Bulb turbine*

A number of bulb turbines are operating successfully all over the world. In India at present there are only two power stations where bulb turbines have been installed. At Gandak Western Canal Power House three bulb turbine units each of 5.5 MW are operating under a head of 6.1 m. The runner diameter is 4.1 m and it runs at 107.2 r.p.m. The total discharge rating is about 103.8 m^3/sec per unit. The Kosi East Canal Power House has four bulb turbine units each of 5.6 MW operating under a head of 6.1 m and has discharge rating about 104.5 m^3/sec per unit. The runner diameter in this case is 4.5 m.

21.21 GOVERNING OF TURBINES

All the modern hydraulic turbines are directly coupled to the electric generators. The generators are always required to run at constant speed irrespective of the variations in the load. This constant speed N r.p.m. of the generator is given by the expression

$$f = \frac{pN}{60} \tag{21.36}$$

where f is the frequency for power generated in cycles per second and p is the number of pair of poles for the generator. Usually $f = 50$ and hence

$$N = \frac{60f}{p} = \frac{3000}{p} \tag{21.37}$$

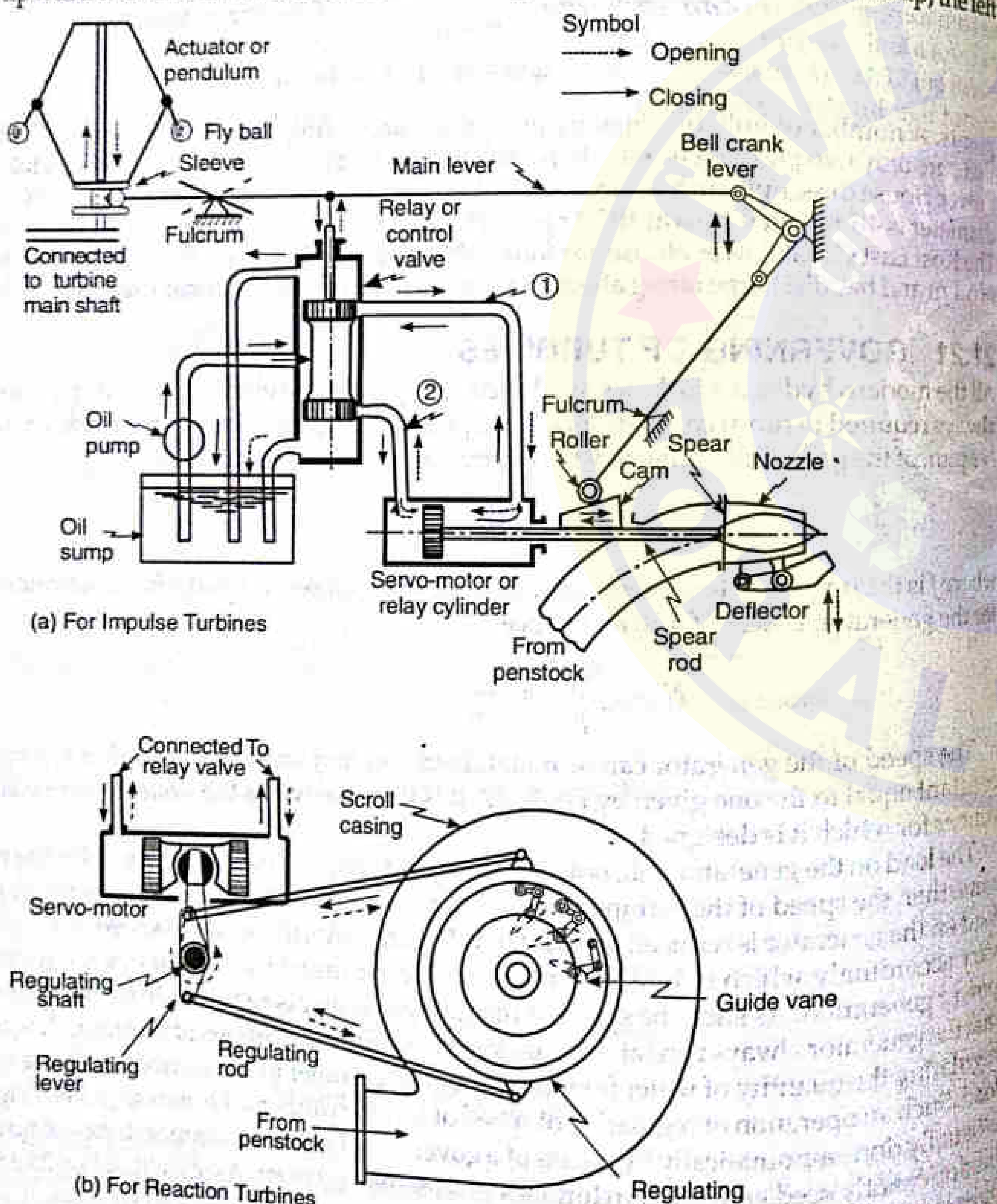
The speed of the generator can be maintained constant only if the speed of the turbine runner is constant equal to the one given by Eq. 21.37. It is then known as the *synchronous speed of the turbine* runner for which it is designed.

The load on the generator will, however, go on varying and now if the input for the turbine remains same then the speed of the turbine runner will tend to either increase or decrease depending on the load on the generator is reduced or increased. This in turn will cause the speed of the generator also to vary accordingly, which is, however, not desirable because it may result in varying the frequency of power generation. As such the speed of the turbine runner is required to be maintained constant so that the generator always runs at constant speed under all conditions of working. It is usually done by regulating the quantity of water flowing through the runner in accordance with the variations in the load. Such an operation of regulation of speed of turbine runner is known as governing of turbine which is usually done automatically by means of a governor. One of the common types of governor which is predominantly used with modern turbines is *oil pressure governor*. As shown in Fig. 21.14 its component parts are as follows:

(i) Servomotor also known as relay cylinder.
(ii) Relay valve also known as control valve or distribution valve.
(iii) Actuator or pendulum which is belt or gear driven from the turbine main shaft.
(iv) Oil sump.
(v) Oil pump which is driven by belt connected to turbine main shaft.
(vi) A system of oil supply pipes connecting the oil sump with the relay valve and the relay valve with the servomotor.

The working of the governor is explained below:

When the load on the generator drops, the speed of turbine runner increases. Now since the actuator or pendulum is driven by the turbine main shaft, due to increase in the speed the balls move outward, resulting in an upward movement of the sleeve shown in Fig. 21.14 (a). As the sleeve moves up, the left



(a) For Impulse Turbines

(b) For Reaction Turbines

Figure 21.14 *Governing mechanism of turbines*

hand end of the main lever is raised, which causes the bell crank lever to move downward and simultaneously pushes the piston of the control valve down in its cylinder. In the case of a Pelton turbine the downward motion of the bell crank lever brings the *deflector* in front of the jet thereby diverting a portion of the jet away from the buckets. On the other hand in the case of reaction turbine such as Francis or Kaplan turbine a *relief valve* shown in Fig. 21.15 is provided. A relief valve consists of a spear, which under normal condition of operation of turbine, is held by oil or water under pressure in such a position that it keeps the opening of the bye-pass from the spiral casing to the tail race closed. The downward motion of the bell crank lever (or other suitable arrangement) opens the pilot valve of the pressure chamber so that the pressure on the spear is reduced, thereby permitting the spear to be lifted up and allowing a portion of water to flow directly from the spiral casing to the tail race through the bye-pass without striking the turbine runner. Thus both these devices, viz., deflector and relief valve have similar function to perform. These eliminate the necessity of rapid closure of the nozzle opening or the guide vanes, at the same time the quantity of water striking the runner is reduced. The rapid closure of the nozzle opening or the guide vanes is not desirable because a sudden reduction of the rate of flow in the penstock may result in serious water hammer problems.

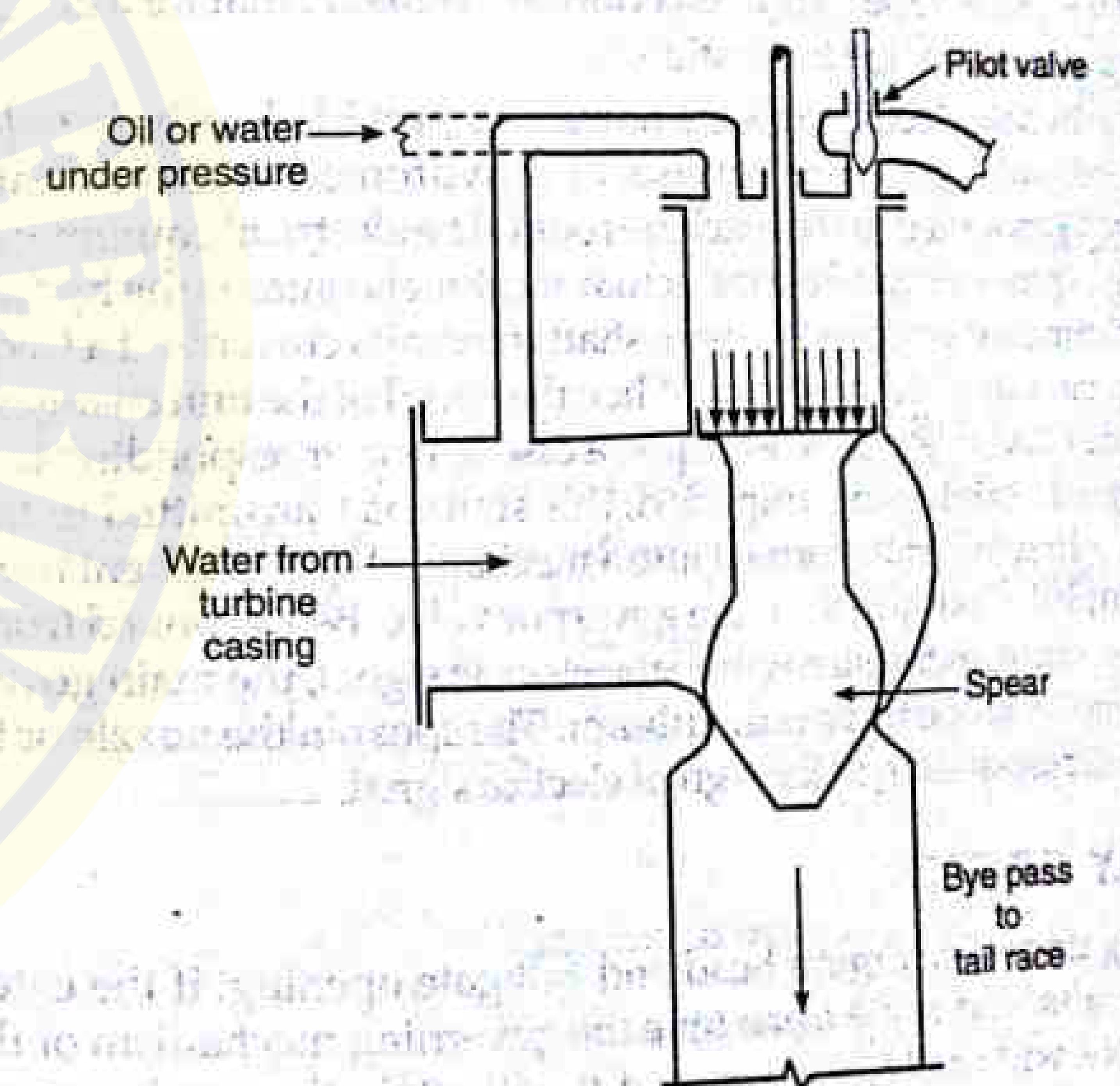


Figure 21.15 *Relief valve*

With the downward motion of the piston of the control valve the passage for pipeline 2 opens and oil under pressure is admitted from the control valve cylinder to the servomotor on the left side of the piston. The servomotor piston, therefore, moves to the right. In the case of Pelton wheel the servomotor piston being connected to the spear rod, thus causes the spear to move forward. The forward motion of the spear reduces the area of the nozzle outlet and thus decreases the rate of flow (as required for the decreased load) whereby the normal turbine speed is restored. However, in the case of a reaction turbine the forward motion of the servomotor piston is transmitted to the regulating ring as shown in Fig. 21.14 (b) which causes all the guide vanes to move simultaneously in one direction, and thus the area of flow passage between the adjacent guide vanes is reduced and the rate of flow of water striking the runner is also reduced. When once the normal turbine speed is restored the main lever attains its initial normal position and through a suitable cam arrangement the deflector is brought back to its original position or the relief valve stops byepassing of water to the tail race.

When the load on the generator increases, the speed of the turbine runner decreases. Due to this the balls move inward resulting in the downward movement of the sleeve. The left hand end of the main lever is lowered which pulls the piston of the control valve up in the cylinder. With the upward motion of the piston of the control valve the passage for pipeline 1 opens and oil under pressure rushes from the control valve cylinder to the servomotor on the right side of the piston. The servomotor piston then moves to the left. This increases the area of the nozzle outlet or the passage between the adjacent guide vanes, thereby allowing a large quantity of water to strike the runner (as required for the increased load) and the normal speed for the turbine runner is thus restored.

In the case of Kaplan turbine since in addition to guide vanes the runner vanes are also adjustable, the governor is required to operate both sets of vanes simultaneously. The runner vanes are also operated by a separate servomotor and a control valve, and the servomotors and the control valves for both, the runner and the guide vanes, are interconnected to ensure that for a given guide vane opening there shall be a definite runner vane inclination.

However, the large hydro-electric units are nowadays provided with *electrohydraulic governers*. In general, an electrohydraulic governor consists of a hydromechanical actuator and an electrical equipment cabinet, both mounted in the machine room. The electrical equipment cabinet contains the electrical part of the governor only, while the actuator cabinet contains the hydromechanical devices. A special techogenerator connected to the main shaft transmits currents at a frequency of 50 cycles to the electric measuring circuits of the governor. When the speed of the unit changes, the frequency of the techogenerator supplies also changes, whereupon a discriminator responding to frequency variations emits an electrical signal. After being amplified, this signal is transmitted to the electromechanical final control element, where it is transformed into a mechanical displacement transmitted through the lever system of the amplifier valve to the main governor valve. Being moved from its mid (or normal) position in direct proportion to the strength of the electric signal, the main governor feeds oil under pressure to the servomotor of the turbine distributor. The spear in the nozzle or the guide vanes thus turn in the direction corresponding to the sign of electric signal.

21.22 RUNAWAY SPEED

For a turbine working under maximum head and full gate opening, if the external load suddenly drops to almost zero value and at the same time the governing mechanism of the turbine also fails, then the turbine runner will tend to race up and it will attain the maximum possible speed. This maximum or limiting speed of the turbine runner is known as *runaway speed*. Obviously for safe design the various rotating components are designed for the runaway speed. For a Pelton wheel the runaway speed normally ranges from 1.8 to 1.9 times its normal speed, for a Francis turbine it normally ranges from 2 to 2.2 times its normal speed and for a Kaplan turbine it normally ranges from 2.5 to 3 times its normal speed.

21.23 SURGE TANKS

As indicated earlier when the load on the generator decreases the governor reduces the rate of flow of water striking the runner in order to maintain the constant speed for the runner. But the sudden reduction of the rate of flow in the penstock may lead to setting up of water hammer in the pipe, which may cause excessive inertia pressure in the pipeline due to which the pipe may burst. Two devices viz., deflector and relief valve as described earlier are thus provided to avoid the sudden reductions of the rate of flow in the penstock. But neither of these devices are of any assistance when the load on the generator increases and the turbine is in need of more water. Thus in order to fulfil both the above noted requirements, in addition to the above noted devices certain other devices such as surge tank and *forebay* are usually employed. Surge tanks are employed in the case of high and medium head hydro-power plants where the penstock is very long, and forebays are suitable for medium and low head hydro-power plants where the length of the penstock is short.

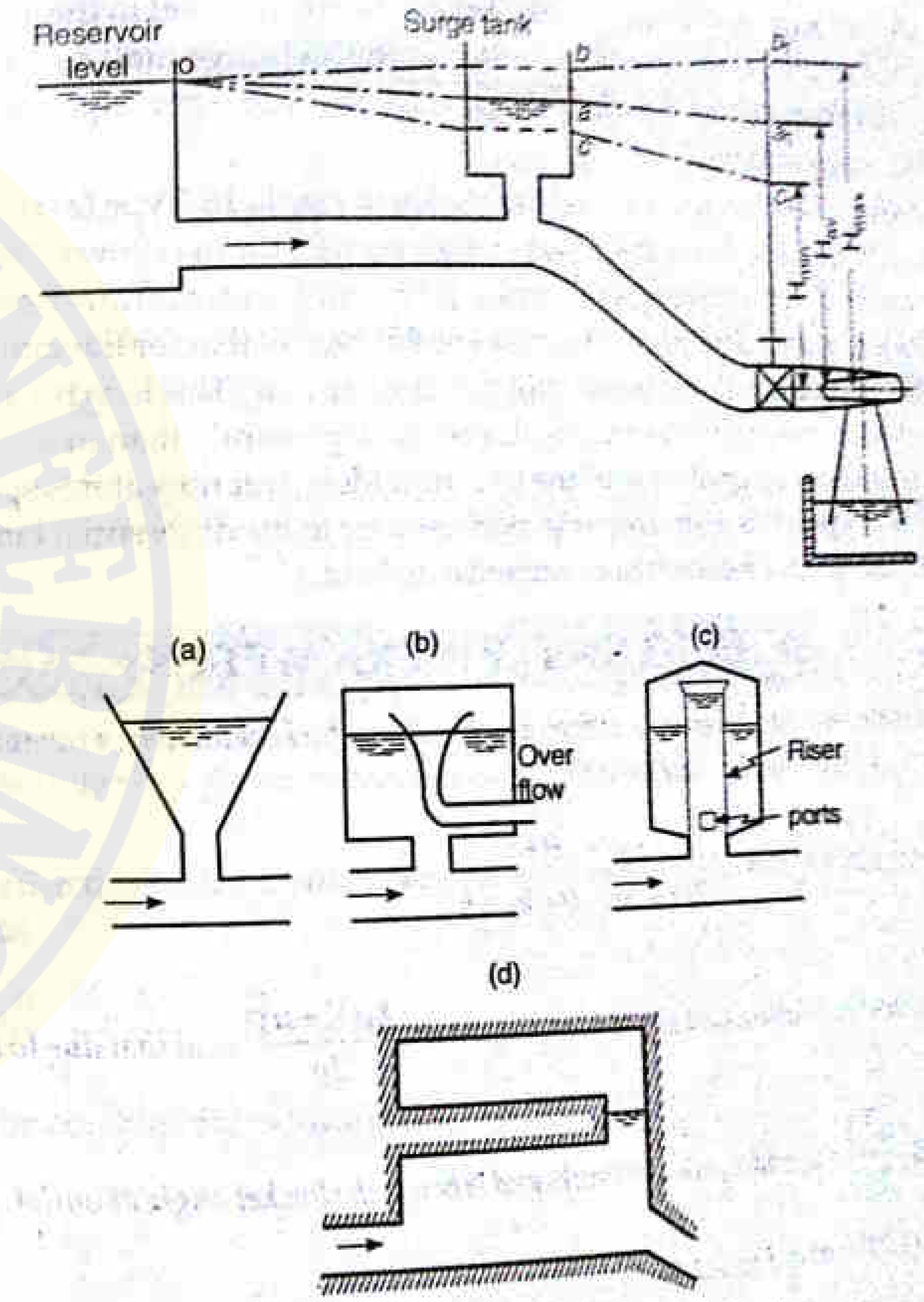


Figure 21.16 *Different types of surge tanks*

An ordinary surge tank is a cylindrical open-topped storage reservoir, as shown in Fig. 21.16. The upper lip of tank which is connected to the penstock at a point as close as possible to the turbine. The upper lip of tank is kept well above the maximum water level in the supply reservoir. When the load on the turbine is steady and normal and there are no velocity variations in the pipeline there will be a normal pressure gradient oaa_1. The water surface in the surge tank will be lower than the reservoir surface by an amount equal to the friction head loss in the pipe connecting the reservoir and the surge tank. When the load on the generator is reduced, turbine gates are closed and the water moving towards the turbine has to move backward. The rejected water is then stored in the surge tank in the space between levels *a* and *b* and a rising pressure gradient obb_1 is developed. The retarding head so built up in the surge tank reduces the velocity of flow in the pipeline corresponding to the reduced discharge required by the turbine.

When the load on the generator increases, the governor opens the turbine gates to increase the rate of flow entering the runner. The increased demand of water by the turbine is partly met by the water stored between levels *a* and *c* in the surge tank. As such the water level in the surge tank falls and a falling pressure gradient occ_1 is developed. In other words, the surge tank develops an accelerating head which increases the velocity of flow in the pipeline to a value corresponding to the increased discharge required by the turbine.

The various other types of surge tanks are also shown in Fig. 21.16. Type (a) is a conical type surge tank, type (b) has an internal bell mouth spillway which permits the overflow to be easily disposed of. Type (c) is known as the *differential surge tank*, which is provided with a central riser pipe having small ports or holes at its lower end. The main advantage of this type is that for the same stabilising effects its capacity may be less than that of a simple cylindrical surge tank. This is so because in a differential tank retarding and accelerating heads are developed more promptly than in a simple surge tank in which the heads only built-up gradually as the tank fills. Moreover no water is spilled to waste from the differential tank. Type (d) is also similar in performance to the differential tank, but it is suitable when appropriate earth or rock excavation can be carried out.

ILLUSTRATIVE EXAMPLES

Example 21.1. *Prove that the maximum efficiency of Pelton wheel occurs when the ratio of bucket velocity u to the jet velocity V is given by the expression*

$$\rho = \frac{u}{V} = \frac{1-\cos\ \theta+k_1}{2(1-\cos\ \theta+k_1+k_2}$$

where the loss due to bucket friction and shock is expressed as $\frac{k_1(V-u)^2}{2g}$ *and that due to bearing friction and windage losses as* $\left(k_2\frac{u^2}{2g}\right)$; *$k_1$ and k_2 are constants and θ being the bucket angle at outlet. Neglect volumetric losses. [Note: $\angle 6 = (180° - \phi)$ of Fig. 21.4].*

Solution

Considering the losses given in the problem the net amount of work done per unit weight of water

$$= \frac{1}{g}\ [(V-u)\,(1-\cos\theta\,)]\,u - k_1\frac{(V-u)^2}{2g} - k_2\frac{u^2}{2g}$$

Thus

$$\eta = \frac{\frac{1}{g}[(V-u)(1-\cos\theta)]u - k_1\frac{(V-u)^2}{2g} - k_2\frac{u^2}{2g}}{\frac{V^2}{2g}}$$

For η to be maximum $\left(\frac{d\eta}{du}\right) = 0$, thus by differentiating with respect to u, we get

$$\frac{d\eta}{du} = \frac{2(V-2u)(1-\cos\theta) + 2k_1(V-u) - 2k_2u}{V^2} = 0$$

$$(V-2u)\,(1-\cos\theta) + k_1\,(V-u) - k_2u = 0$$

or

$$V\,(1-\cos\theta + k_1) = u[2(1-\cos\theta) + k_1 + k_2]$$

or

$$\therefore \quad \frac{u}{V} = \frac{1-\cos\theta + k_1}{2(1-\cos\theta) + k_1 + k_2}$$

Example 21.2. *In a Francis turbine, prove that hydraulic efficiency η_h of the turbine can be expressed as*

$$\eta_h = \frac{2}{2 + \frac{k_1+k_2+k_3+k_4}{(\cot\alpha - \cot\theta)[\cot\alpha(1+n^2) - n(\cot\phi + n\cot\theta)]}}$$

where k_1, k_2, k_3 and k_4 represent the fraction of the losses in the guide vanes, runner vanes, draft tube and at exit respectively expressed in terms of the velocity of flow head; α, θ and ϕ are the guide vane angle, the runner vane angle at inlet and the runner vane angle at outlet respectively and n is the ratio of the inner to outer diameter of the runner. Assume the velocity of flow to remain constant in the runner.

Solution

If H is the net head acting on the turbine then the work done by the runner per unit weight of water may be expressed as

$$\frac{V_w u - V_{w_1} u_1}{g} = H - \text{losses}$$

The losses may be combinedly expressed as

$$(k_1+k_2+k_3+k_4)\,\frac{V_f^2}{2g}$$

also $V_f = V_{f1}$ being constant.

Thus

$$\frac{V_w u - V_{w_1} u_1}{g} = H - (k_1+k_2+k_3+k_4)\,\frac{V_f^2}{2g} \qquad ...(i)$$

From inlet and outlet velocity triangles

$$V_w = V_f \cot\alpha ; \quad \frac{V_f}{V_w - u} = \tan\theta$$

or
$$\frac{V_f}{V_f \cot\alpha - u} = \tan\theta ;$$

$\therefore$
$$u = V_f(\cot\alpha - \cot\theta)$$

$$\frac{V_{f_1}}{v_{w_1} + u_1} = \tan\phi ;$$

or
$$\frac{V_f}{V_{w_1} + u_1} = \tan\phi$$

Also
$$u = \frac{\pi DN}{60} ; u_1 = \frac{\pi D_1 N}{60}$$

$\therefore$
$$u_1 = \left(\frac{D_1}{D}\right) u$$

$$= nu = n\, V_f\,(\cot\alpha - \cot\theta)$$

Introducing this value of u_1, we get

$$\frac{v_f}{V_{w_1} + nV_f(\cot\alpha - \cot\theta)} = \tan\phi = \frac{1}{\cot\phi}$$

or
$$V_{w1} = V_f[\cot\phi - n(\cot\alpha - \cot\theta)]$$

The hydraulic efficiency is given by

$$\eta_h = \frac{V_w u - V_{w_1} u_1}{gH}$$

From Eq. (i), we have

$$gH = (v_w u - v_{w1} u_1) + (k_1 + k_2 + k_3 + k_4)\frac{V_f^2}{2}$$

Thus

$$\eta_h = \frac{(V_w u - V_{w_1} u_1)}{(V_w u - V_{w_1} u_1) + (k_1 + k_2 + k_3 + k_4)\dfrac{V_f^2}{2}}$$

$$= \frac{2}{2 + \dfrac{(k_1 + k_2 + k_3 + k_4)V_f^2}{(V_w u - V_{w_1} u_1)}}$$

Substituting the values of V_w, u, V_{w1} and u_1, we get

$$\eta_h = \frac{2}{2 + \dfrac{(k_1 + k_2 + k_3 + k_4)}{(\cot\alpha - \cot\theta)\{\cot\alpha(1 + n^2) - n(\cot\phi + n\cot\theta)\}}}$$

Example 21.3. *A Pelton wheel has to be designed for the following data. Power to be developed = 6 000 kW. Net head available = 300 m; Speed = 550 r.p.m.; Ratio of jet diameter to wheel diameter = 1/10; and overall efficiency = 85%.*

Find the number of jets; diameter of the jet; diameter of the wheel; and the quantity of water required.

Solution

Assume
$$K_v = 0.98; \text{ speed ratio } K_u = 0.46$$

Thus
$$V = K_v \sqrt{2gH}$$
$$= 0.98\sqrt{2 \times 9.81 \times 300} = 75.19 \text{ m/s}$$
$$u = K_u \sqrt{2gH}$$
$$= 0.46\sqrt{2 \times 9.81 \times 300} = 35.29 \text{ m/s}$$
$$\eta_0 = \frac{P}{wQH}$$

or
$$0.85 = \frac{6000 \times 10^3}{9810 \times Q \times 300}$$

$\therefore$
$$Q = 2.399 \text{ m}^3/\text{s}$$

$$u = \frac{\pi DN}{60} = \frac{\pi \times D \times 550}{60} = 35.29 \text{ m/s}$$

$\therefore$
$$D = 1.225 \text{ m}$$

Since
$$\frac{d}{D} = \frac{1}{10}$$

$\therefore$
$$d = \frac{1.225}{10} = 0.1225 \text{ m} = 122.5 \text{ mm}$$

Area of jet
$$a = \frac{\pi}{4} \times (0.1225)^2 = 0.0118 \text{ m}^2$$

Total jet area required
$$= \frac{Q}{V} = \frac{2.399}{75.19} = 0.0.319 \text{ m}^2$$

$\therefore$ No. of jets required
$$= \frac{0.0319}{0.0118} = 2.7 \approx 3$$

Thus three jets are required each having a diameter

$$d = \left(\frac{0.0319}{3(\pi/4)}\right)^{1/2} = 0.1164 \text{ m} = 116.4 \text{ mm}$$

$$\frac{d}{D} = \frac{0.1164}{1.225} = \frac{1}{10.52}$$

which is close to the given value.

Example 21.4. *A Pelton wheel has a mean bucket speed of 12 m/s and is supplied with water at a rate of 750 litres per second under a head of 35 m. If the bucket deflects the jet through an angle of 160°, find the power developed by the turbine and its hydraulic efficiency. Take the coefficient of velocity as 0.98. Neglect friction in the bucket. Also determine the overall efficiency of the turbine if its mechanical efficiency is 80%.*

Solution

From Eq. 21.10 the power developed by the turbine is given as

$$P = \frac{W}{g}[(V-u)(1+k\cos\phi)]u$$

$$W = wQ$$

$$w = 9810 \text{ N/m}^3;$$

and

$$Q = 750 \text{ l/s} = \frac{750}{1000} = 0.75 \text{ m}^3/\text{s}$$

$\therefore$

$$W = 9810 \times 0.75 = 7357.5 \text{ N/s}$$

$$V = 0.98\sqrt{2\times 9.81\times 35} = 25.68 \text{ m/s}$$

$$u = 12 \text{ m/s}$$

$$k = 1 \text{(for neglecting the friction in the buckets)}$$

$$\phi = (180° - \theta)$$

$$= (180° - 160°) = 20°$$

$\therefore$

$$\cos\phi = \cos 20° = 0.9397$$

Thus by substitution, we get

$$P = \frac{7357.5}{9.81}[(25.68-12)(1+0.9397)]\times 12 \text{ W}$$

$$= 238816 \text{ W}$$

$$= \frac{238816}{1000} \text{ kW}$$

$$= 238.816 \text{ kW}$$

Since 1 metric h.p. = 735.5 W, the power developed by the turbine in metric h.p. is

$$P = \frac{238816}{735.5} = 324.699 \text{ metric h.p.}$$

From Eq. 21.11 the hydraulic efficiency of the turbine is given as

$$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2}$$

$$= \frac{2\times 12(25.68-12)(1+0.9397)}{(25.68)^2}$$

$$= 0.966 \text{ or } 96.6\%$$

The overall efficiency of the turbine is given by Eq. 21.9 as

$$\eta_0 = \eta_h \times \eta_m$$

$$\eta_m = 80\% \text{ or } 0.80$$

$\therefore$

$$\eta_0 = 0.966 \times 0.80$$

$$= 0.773 \text{ or } 77.3\%$$

Example 21.5. *The following are the design particulars of a large Pelton turbine:*

(i) Head at the distributor = 630 m
(ii) Discharge = 12.5 m³/s
(iii) Power developed = 65 MW
(iv) Speed of rotation = 500 r.p.m.
(v) Runner diameter = 1.96 m
(vi) Number of jets = 4
(vii) Jet diameter = 0.192 m
(viii) Angle through which the jet is deflected by the bucket = 165°
(ix) Mechanical efficiency of the turbine = 96%

(a) Determine the hydraulic power losses in the distributor-nozzle assembly and the buckets.

(b) If the loss in the buckets is given to be proportional to V_r^2, where V_r is the relative velocity at inlet, determine the best speed of rotation for this head and discharge.

Solution

(a) From Eq. 21.17, mechanical efficiency of Pelton turbine is given as

$$\eta_m = \frac{P}{W\left[\frac{(V-u)u(1+k\cos\phi)}{g}\right]}$$

$\eta_m = 96\%$ or 0.96; and $P = 65 \text{ MW} = 65 \times 10^6$ W

Thus by substitution, we get

$$0.96 = \frac{65\times 10^6}{W\left[\frac{(V-u)u(1+k\cos\phi)}{g}\right]}$$

or

$$W\left[\frac{(V-u)u(1+k\cos\phi)}{g}\right] = \frac{65\times 10^6}{0.96} = 67.71\times 10^6 \text{ W}$$

which is the power developed by the turbine runner.

Power supplied to the turbine runner at the distributor is

$$P_1 = wQH$$
$$w = 9810 \text{ N/m}^3;\ Q = 12.5 \text{ m}^3\text{/s; and } H = 630 \text{ m}$$

Thus by substitution, we get

$$P_1 = 9810 \times 12.5 \times 630 \text{ W} = 77.25 \times 10^6 \text{ W}$$

∴ Power losses in the distributor-nozzle assembly and the buckets

$$= (P_1 - P)$$
$$= (77.25 \times 10^6 - 67.71 \times 10^6) \text{ W}$$
$$= 9.54 \times 10^6 \text{ W}$$

(b) The head loss in the buckets is given to be proportional to V_r^2, i.e.,

$$h_L = CV_r^2 = C(V-u)^2$$

where C is constant of proportionality

Total disharge $= 12.5 \text{ m}^3\text{/s}$

No. of jets $= 4$

∴ Discharge through each jet $= \dfrac{12.5}{4} \text{ m}^3\text{/s}$

Diameter of jet $= 0.192$ m

∴ $$V = \frac{12.5}{4 \times \frac{\pi}{4} \times (0.192)^2} = 107.93 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$
$$D = 1.96 \text{ m; and } N = 500 \text{ r.p.m}$$

∴ $$u = \frac{\pi \times 1.96 \times 500}{60} = 51.31 \text{ m/s}$$

Thus by substitution, we get

$$h_L = C(107.93\text{-}51.31)^2 = 3206\ C \qquad ...(i)$$

The best speed of rotation is the one which would provide maximum hydraulic efficieny, for which the condition is given by Eq. 21.12 as

$$u = \frac{V}{2};\ \text{or } V = 2u$$

∴ $$h_L = C(2u-u)^2 = Cu^2 \qquad ...(ii)$$

From Eqs (i) and (ii), we have

$$Cu^2 = 3206\ C$$

or $$u = 56.62 \text{ m/s}$$

Thus if N is the best speed of rotation then

$$u = \frac{\pi DN}{60} = 56.62$$

or $$\frac{\pi \times 1.96 \times N}{60} = 56.62$$

∴ $$N = 552 \text{ r.p.m.}$$

Example 21.6. *A Pelton wheel produces 1000 hp under a gross head of 200 m. Its nozzle has a diameter of 10 cm and the losses in pipe line due to friction amount to 90 Q^2 where Q is the disharge in m³/s. Assuming the gross head and efficiency of the wheel to be constant and C_v for the nozzle as 0.98, find the disharge and overall efficiency.*

If the power produced is reduced to 800 hp (i) by operating the needle in the nozzle and (ii) by closing the valve provided in the main, determine the discharge in either case. What is the additional head loss in case (ii)? Comment on the results.

Solution

Net head $\quad H = (200 - 90\,Q^2)$

∴ $$V = 0.98\sqrt{2 \times 9.81 \times (200 - 90\,Q^2)}$$

Assuming jet diameter to be same as nozzle diameter, we have

$$Q = \frac{\pi}{4} \times (0.1)^2 \times 0.98\sqrt{2 \times 9.81 \times (200 - 90\,Q^2)}$$

Solving for Q, we get

$$Q = 0.465 \text{ m}^3\text{/s}$$

Power $\quad P = 1000 \times 746$ W (since 1 hp = 746 W)

Net head $\quad H = [200 - 90 \times (0.465)^2] = 180.54$ m

$$\eta_0 = \frac{P}{wQH}$$
$$= \frac{1000 \times 746}{9810 \times 0.465 \times 180.54}$$
$$= 0.906 \text{ or } 90.6\%$$

(i) When the power produced is reduced to 800 hp by operating the needle in the nozzle, then since power developed $P \propto Q \propto d^2$, where d is the diameter of the jet, the required jet diameter

$$d = 10 \times \left(\frac{800}{1200}\right)^{1/2}$$
$$= 8.16 \text{ cm} = 0.0816 \text{ m}$$

It is assumed that there is no loss of head in the nozzle, and if Q is the required discharge in this case then, we have

$$Q = \frac{\pi}{4} \times (0.0816)^2 \times 0.98\sqrt{2 \times 9.81 \times (200 - 90\,Q^2)}$$

Solving for Q, we get

$$Q = 0.314 \text{ m}^3/\text{s}$$

(ii) When the power produced is reduced to 800 hp by closing the valve provided in the main then the jet diameter remains the same as 10 cm, but there will be additional head loss at the valve. Thus if h_L is the loss of head at the valve then

Net head $$H = (200 - 90\,Q^2 - h_L)$$

$\therefore$ $$V = 0.98\sqrt{2\times 9.81\times(200-90Q^2-h_L)}$$

and $$Q = \frac{\pi}{4}\times(0.1)^2\times 0.98\sqrt{2\times 9.81\times(200-90\,Q^2-h_L)} \quad \text{...(i)}$$

Assuming the efficiency to remain constant, we have

$$0.906 = \frac{800\times 746}{9810\times Q\times(200-90\,Q^2-h_L)}$$

or $$\left(200-90Q^2-h_L\right) = \frac{800\times 746}{0.906\times 9810\times Q} = \frac{67.148}{Q} \quad \text{...(ii)}$$

From Eqs (i) and (ii), we get

$$Q = \frac{\pi}{4}\times(0.1)^2\times 0.98\sqrt{\frac{2\times 9.81\times 67.148}{Q}}$$

Solving for Q, we get

$$Q = 0.427 \text{ m}^3/\text{s}$$

Introducing this value of Q in Eq. (ii), we get

$$200-90(0.427)^2-h_L = \frac{67.148}{0.427}$$

or $$h_L = 26.34 \text{ m}$$

i.e., the additional head loss in case (ii) is 26.34 m

The provision of a valve in the main to reduce the discharge when the power produced is reduced results in additional head loss and also requires more discharge, and hence it is not a satisfactory arrangement. On the other hand the provision of a needle in the nozzle and its operation to reduce the discharge when the power produced is reduced, results in negligible head loss and requires less discharge, and hence it is a satisfactory arrangement which is commonly adopted.

Example 21.7 *A pipeline 1200 m long supplies water to 3 single jet Pelton wheels. The head above the nozzle is 360 m. The velocity coefficient for the nozzle is 0.98 and the coefficient of friction for the pipeline is 0.02. The turbine efficiency based on the head at the nozzle is 0.85. The specific speed of each turbine is 15.3 (in m, kW, r.p.m., units) and the head lost due to friction in the pipeline is 12 m of water. If the operating speed of each turbine is 560 r.p.m., determine:*

(i) the total power developed.

(ii) the diameter of each nozzle.

(iii) the diameter of the pipeline.

(iv) volume of water used per second.

Solution

Net head available at the base of the nozzle

$$= (360 - 12) = 348 \text{ m}$$

As indicated in Chapter 22

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

By substitution, we have

$$15.3 = \frac{560\sqrt{P}}{(348)^{5/4}}$$

$\therefore$ Power developed by each turbine is

$$P = \left[\frac{15.3\times(348)^{5/4}}{560}\right]^2 = 1686.37 \text{ kW}$$

$\therefore$ Total power developed by 3 turbines

$$= (3\times 1686.37) = 5\,059.11 \text{ kW}$$

Also $$\eta_0 = \frac{P}{wQH}$$

By substitution, we have

$$0.85 = \frac{1686.37\times 10^3}{9810\times Q\times 348}$$

$\therefore$ Discharge required by each turbine is

$$Q = 0.581 \text{ m}^3/\text{s}$$

$\therefore$ Total volume of water used per second by all the three turbines = $(3\times 0.581) = 1.743$ m^3, i.e., total

$$Q = 1.743 \text{ m}^3/\text{s}$$

$$V = K_v\sqrt{2gH}$$

$$= 0.98\sqrt{2\times 9.81\times 348} = 80.98 \text{ m/s}$$

Thus if diameter of the nozzle is d then

$$0.581 = \frac{\pi}{4}d^2\times 80.98$$

$$d = 0.0956 \text{ m} = 95.6 \text{ mm}$$

Further $$h_f = \frac{flv^2}{2gD} = \frac{flQ^2}{2g(\pi/4)^2D^5}$$

or $$12 = \frac{0.02\times 1200\times(1.743)^2}{2\times 9.81\times(\pi/4)^2D^5}$$

$\therefore$ Diameter of pipe $D = 0.871 \text{ m} = 871 \text{ mm}$

Example 21.8 *The following data were obtained from a test on a Pelton wheel:*

(a) Head at the base of the nozzle = 32 m

(b) Discharge of the nozzle = 0.18 m³/s

(c) Area of the jet = 7500 sq. mm

(d) Power available at the shaft = 44 kW

(e) Mechanical efficiency = 94 %

Calculate the power lost (i) in the nozzle; (ii) in the runner; (iii) in mechanical friction.

Solution

Power at the base of the nozzle

$$= wQH$$
$$= (9\,810 \times 0.18 \times 32)$$
$$= 56\,510 \text{ W} = 56.51 \text{ kW}$$

Velocity of flow through the nozzle

$$V = \frac{0.18}{7500 \times 10^{-6}} = 24 \text{ m/s}$$

Power at the nozzle exit (i.e., Kinetic energy of the jet)

$$= \frac{waV^3}{2g}$$

$$= \left[\frac{9810 \times 7500 \times 10^{-6} \times (24)^3}{2 \times 9.81}\right]$$

$$= 51\,840 \text{ W} = 51.84 \text{ kW}$$

$\therefore$ Power lost in the nozzle

$$= (56.51 - 51.84) = 4.67 \text{ kW}$$

Power supplied to the runner is equal to the kinetic energy of the jet

$$= 51.84 \text{ kW}$$

Power developed by the runner

$$= \frac{44}{0.94} = 46.81 \text{ kW}$$

$\therefore$ Power lost in the runner

$$= (51.84 - 46.81) = 5.03 \text{ kW}$$

Power lost in mechanical friction

$$= (46.81 - 44) = 2.81 \text{ kW}$$

As a check on computation, the difference of power at the base of the nozzle and the power available at the shaft must be equal to the sum of the power lost in the nozzle, in the runner and in mechanical friction.

Thus, we have

$$(56.51 - 44) = 12.51 \text{ kW}$$

and $$(4.67 + 5.03 + 2.81) = 12.51 \text{ kW}$$

Example 21.9 *An inward flow turbine (reaction type with radial discharge) with an overall efficiency of 80% is required to develop 150 kW. The head is 8 m; peripheral velocity of the wheel is $0.96\sqrt{2gH}$; the radial velocity of the flow is $0.36\sqrt{2gH}$. The wheel is to make 150 r.p.m., and the hydraulic losses in the turbine are 22% of the available energy. Determine:*

(i) the angle of the guide blade at inlet

(ii) the wheel vane angle at inlet

(iii) the diameter of the wheel

(iv) the width of the wheel at inlet

Solution

$$\eta_o = 80\%, H = 8 \text{ m}, P = 150 \text{ kW}, N = 150 \text{ r.p.m.}$$

Peripheral velocity $u = 0.96\sqrt{2gH}$

$$= 0.96\sqrt{2 \times 9.81 \times 8} = 12.03 \text{ m/s}$$

Velocity of flow at inlet

$$V_f = 0.36\sqrt{2gH}$$
$$= 0.36\sqrt{2 \times 9.81 \times 8} = 4.51 \text{ m/s}$$

Hydraulic losses $= 22\%$

$\therefore$ $\eta_h = (100 - 22) = 78\%$

For radial discharge

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.78 = \frac{V_w \times 12.03}{9.81 \times 8}$$

$\therefore$ $V_w = 5.09 \text{ m/s}$

From inlet velocity triangle, we have

$$\tan\alpha = \frac{V_f}{V_w} = \frac{4.51}{5.09} = 0.8861$$

$\therefore$ $\alpha = 41° \, 33'$

Also $$\tan(180 - \theta) = \frac{V_f}{u - V_w} = \frac{4.51}{12.03 - 5.09} = 0.6499$$

$\therefore$ $\theta = 147°$

$$u = \frac{\pi D N}{60}$$

or $$12.03 = \frac{\pi \times D \times 150}{60}$$

$$\therefore \quad D = 1.532 \text{ m}$$

Also
$$\eta_0 = \frac{P}{wQH}$$

or
$$0.80 = \frac{150 \times 10^3}{9810 \times Q \times 8}$$

$$\therefore \quad Q = 2.389 \text{ m}^3/\text{s}$$

Neglecting the vane thickness, we have

$$Q = \pi D B V_f$$

or
$$2.389 = \pi \times 1.532 \times B \times 4.51$$

$$\therefore \quad B = \frac{2.389}{\pi \times 1.532 \times 4.51} = 0.110 \text{ m} = 110 \text{ mm}$$

Example 21.10. *An inward flow reaction turbine discharges radially and the velocity of flow is constant and equal to the velocity of discharge from the turbine. Show that the hydraulic efficiency can be expressed by*

$$\eta_h = \frac{1}{1 + \dfrac{\frac{1}{2}(\tan^2\alpha)}{\left(1 - \dfrac{\tan\alpha}{\tan\theta}\right)}}$$

where α and θ are respectively the guide vane angle and wheel vane angle at inlet.

Solution

From the inlet velocity triangle

$$V_f = V_w \tan\alpha$$

$$u = V_w - \frac{V_f}{\tan\theta}$$

$$= V_w - \frac{V_w \tan\alpha}{\tan\theta} = V_w\left(1 - \frac{\tan\alpha}{\tan\theta}\right)$$

For radial discharge at outlet $V_{w1} = 0$ and $V_1 = V_{f1}$

Thus
$$\frac{V_w u}{g} = H - \frac{V_1^2}{2g}$$

or
$$H = \frac{V_w u}{g} + \frac{V_1^2}{2g}$$

$$= \frac{V_w u}{g} + \frac{V_{f1}^2}{2g} = \frac{V_w u}{g} + \frac{V_f^2}{2g}$$

Substituting the value of V_f, we get

$$H = \frac{V_w u}{g} + \frac{V_w^2 \tan^2\alpha}{2g}$$

Also substituting the value of u from above, we get

$$H = \frac{V_w^2}{g}\left(1 - \frac{\tan\alpha}{\tan\theta}\right) + \frac{V_w^2 \tan^2\alpha}{2g}$$

$$= \frac{V_w^2}{g}\left(1 - \frac{\tan\alpha}{\tan\theta} + \frac{\tan^2\alpha}{2}\right)$$

Now
$$\eta_h = \frac{V_w u}{gH} = \frac{V_w^2\left(1 - \dfrac{\tan\alpha}{\tan\theta}\right)}{gH}$$

or
$$\eta_h = \frac{\left(1 - \dfrac{\tan\alpha}{\tan\theta}\right)}{\left(1 - \dfrac{\tan\alpha}{\tan\theta} + \dfrac{\tan^2\alpha}{2}\right)}$$

or
$$\eta_h = \frac{1}{1 + \dfrac{\frac{1}{2}(\tan^2\alpha)}{1 - \dfrac{\tan\alpha}{\tan\theta}}}$$

Example 21.11. *Design a Francis turbine runner with the following data: Net head $H = 68$ m; speed $N = 750$ r.p.m; output power $P = 330$ kW; $\eta_h = 94\%$; $\eta_o = 85\%$; flow ratio $\psi = 0.15$; breadth ratio $n = 0.1$; inner diameter of runner is $\left(\frac{1}{2}\right)$ outer diameter. Also assume 6% of circumferential area of the runner to be occupied by the thickness of the vanes. Velocity of flow remains constant throughout and flow is radial at exit.*

Solution

$$\eta_0 = \frac{P}{wQH}$$

or
$$0.85 = \frac{330 \times 10^3}{9810 \times Q \times 68}$$

$$\therefore \quad Q = 0.582 \text{ m}^3/\text{s}$$

$$V_f = \psi\sqrt{2gH}$$

$$= 0.15 \times \sqrt{2 \times 9.81 \times 68} = 5.48 \text{ m/s}$$

Also $$Q = (k\pi n D^2)\, V_f$$

or $$0.582 = 0.94 \times \pi \times 0.10 \times D^2 \times 5.48$$

∴ $$D = 0.60 \text{ m} = 600 \text{ mm}$$

Since $$n = (B/D);\ B = (0.10 \times 600) = 60 \text{ mm}$$

$$u = \frac{\pi DN}{60} = \frac{\pi \times 0.60 \times 750}{60} = 23.56 \text{ m/s}$$

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.94 = \frac{V_w \times 23.56}{9.81 \times 68}$$

∴ $$V_w = 26.62 \text{ m/s}$$

From inlet velocity triangle, we have

$$\tan\alpha = \frac{V_f}{V_w} = \frac{5.48}{26.62} = 0.2059$$

∴ $$\alpha = 11°38'$$

$$\tan\theta = \frac{V_f}{V_w - u} = \frac{5.48}{(26.62 - 23.56)} = 1.7908$$

∴ $$\theta = 60°\,49'$$

Now $$D_1 = \frac{D}{2} = 0.30 \text{ m} = 300 \text{ mm}$$

$$\frac{u}{D} = \frac{u_1}{D_1};\ u_1 = 11.78 \text{ m/s}$$

Since the flow is radial at exit and $V_f = V_{f1}$, from outlet velocity triangle

$$\tan\phi = \frac{V_{f_1}}{u_1} = \frac{5.48}{11.78} = 0.4652$$

∴ $$\phi = 24°\,57'$$

Example 21.12. *A Kaplan turbine produces 60 000 kW under a net head of 25 m with an overall efficiency of 90%. Taking the value of speed ratio Ku as 1.6, flow ratio ψ as 0.5 and the hub diameter as 0.35 times the outer diameter, find the diameter and speed of the turbine.*

Solution

$$\eta_0 = \frac{P}{wQH}$$

or $$0.90 = \frac{60000 \times 10^3}{9810 \times Q \times 25}$$

$$Q = 271.83 \text{ m}^3/\text{s}$$

∴ $$Q = \frac{\pi}{4}(D^2 - d^2)\,\psi\sqrt{2gH}$$

$$271.83 = \frac{\pi}{4} \times D^2[1-(0.35)^2] \times 0.5 \times \sqrt{2 \times 9.81 \times 25}$$

or $$D = 5.97 \text{ m}$$

∴ $$u = K_u\sqrt{2gH}$$

$$= 1.6 \times \sqrt{2 \times 9.81 \times 25} = 35.44 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$

or $$35.44 = \frac{\pi \times 5.97 \times N}{60}$$

∴ $$N = 113.38 \text{ r.p.m.}$$

Example 21.13. *Determine the efficiency of a Kaplan turbine developing 3000 kW under a net head of 5 m. It is provided with a draft tube with its inlet (diameter 3 m) set 1.6 m above the tail race level. A vacuum gage connected to the draft tube indicates a reading of 5 m of water. Assume draft tube efficiency as 78%.*

Solution

Neglecting the head loss in the draft tube, from Eq. 21.32, we have

$$\frac{P_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right]$$

or $$-5 = -\left[1.6 + \frac{V_2^2 - V_3^2}{2g}\right]$$

or $$\frac{V_2^2 - V_3^2}{2g} = 3.4$$

From Eq. 21.34, we have

$$\eta_d = \frac{(V_2^2 - V_3^2)/2g}{V_2^2/2g}$$

or $$0.78 = \frac{3.4}{V_2^2/2g}$$

∴ $$V_2 = 9.25 \text{ m/s}$$

$$Q = \left[\frac{\pi}{4} \times (3)^2 \times 9.25\right] = 65.38 \text{ m}^3/\text{s}$$

$$\eta_0 = \frac{P}{wQH}$$

$$= \frac{3000\times10^3}{9810\times65.38\times5}$$

$$= 0.9355 = 93.55\%$$

Example 21.11. *A propeller turbine runner has outer diameter of 4.5 m, and the diameter of the hub 2 m. It is required to develop 20 600 kW when running at 150 r.p.m., under a head of 21 m. Assuming hydraulic efficiency of 94% and overall efficiency of 88%, determine the runner vane angles at inlet and exit at the mean diameter of the vanes. Also determine the runner vane angles at inlet and exit at two sections-near the hub and the outer periphery.*

Solution

$$\eta_0 = \frac{P}{wQH}$$

or $$0.88 = \frac{20600\times10^3}{9810\times Q\times21}$$

∴ $$Q = 113.63\ \text{m}^3/\text{s}$$

From Eq. 21.35, we have

$$Q = \frac{\pi}{4}(D^2-d^2)V_f$$

or $$113.63 = \frac{\pi}{4}\left[(4.5)^2-(2)^2\right]V_f$$

or $$V_f = 8.90\ \text{m/s}$$

(i) Computations for mean diameter.

Mean diameter $$D_m = \left(\frac{4.5+2}{2}\right) = 3.25\ \text{m}$$

$$u_m = \frac{\pi D_m N}{60}$$

$$= \frac{\pi\times3.25\times150}{60} = 25.53\ \text{m/s}$$

$$\eta_h = \frac{V_w u}{gH}$$

or $$0.94 = \frac{V_w\times25.53}{9.81\times21}$$

or $$V_w = 7.59\ \text{m/s}$$

From inlet velocity triangle, we have

$$\tan(180°-\theta) = \frac{V_f}{u_m - V_w}$$

$$= \frac{8.90}{(25.53-7.59)} = 0.4961$$

∴ $$\theta = 153°37'$$

From outlet velocity triangle, since

$$V_{f1} = V_f \text{ and } u_1 = u,$$

$$\tan\phi = \frac{V_{f_1}}{u_1}$$

$$= \frac{8.90}{25.53} = 0.348\,6$$

∴ $$\phi = 19°\,13'$$

(ii) Computations for section near the hub.

$$u_i = \frac{\pi D_i N}{60}$$

$$= \frac{\pi\times2\times150}{60} = 15.71\ \text{m/s}$$

$$\eta_h = 0.94 = \frac{V_w\times15.71}{9.81\times21}$$

or $$V_w = 12.33\ \text{m/s}$$

From inlet velocity triangle, we have

$$\tan(180°-\theta) = \frac{V_f}{u_i - V_w}$$

$$= \frac{8.90}{(15.71-12.33)} = 2.6331$$

∴ $$\theta = 110°\,48'$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f1}}{u_i}$$

$$= \frac{8.90}{15.71} = 0.5665$$

∴ $$\phi = 29°\,32'$$

(iii) Computations for section at the outer periphery.

$$u_0 = \frac{\pi D_0 N}{60}$$

$$= \frac{\pi \times 4.5 \times 150}{60} = 35.34 \text{ m/s}$$

$$\eta_h = 0.94 = \frac{V_w \times 35.34}{9.81 \times 21}$$

or $$V_w = 5.48 \text{ m/s}$$

From inlet velocity triangle, we have

$$\tan(180° - \theta) = \frac{V_f}{u_0 - V_w}$$

$$= \frac{8.90}{(35.34 - 5.48)} = 0.2981$$

$\therefore$ $$\theta = 163° 24'$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{u_0}$$

$$= \frac{8.90}{35.34} = 0.2518$$

$\therefore$ $$\phi = 14° 8'$$

Example 21.15. *The velocity of whirl at inlet to the runner of an inward flow reaction turbine is $(3.15\sqrt{H})$ m/s and the velocity of flow at inlet is $(1.05\sqrt{H})$ m/s. The velocity of whirl at exit is $(0.22\sqrt{H})$ m/s in the same direction as at inlet and the velocity of flow at exit is $(0.83\sqrt{H})$ m/s, where H is the head in metres. The inner diameter of the runner is 0.6 times the outer diameter. Assuming hydraulic efficiency of 80%, compute the angles of the runner vanes at inlet and exit.*

Solution

$$\eta_h = \frac{(V_w u - V_{w1} u_1)}{gH}; \left(\frac{u}{D}\right) = \left(\frac{u_1}{D_1}\right)$$

or $$0.8 = \frac{[(3.15\sqrt{H})u - (0.22\sqrt{H})0.6u]}{9.81H}$$

$\therefore$ $$u = (2.60\sqrt{H}) \text{ and } u_1 = (1.56\sqrt{H})$$

From inlet velocity triangle, we have

$$\tan\theta = \frac{V_f}{(V_w - u)}$$

$$= \frac{1.05\sqrt{H}}{(3.15\sqrt{H}) - (2.60\sqrt{H})} = 1.9091$$

$\therefore$ $$\theta = 62° 21'$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{(u_1 - V_{w1})}$$

$$= \frac{0.83\sqrt{H}}{(1.56\sqrt{H}) - (0.22\sqrt{H})} = 0.6194$$

$\therefore$ $$\phi = 31° 46'$$

Example 21.16. *A straight conical draft tube attached to a Francis turbine has an inlet of diameter 3 m and its outlet area is 20 m². The velocity of water at inlet is 5 m/s. The inlet is set 5 m above the tail race level. Assuming the loss of head in the draft tube equals half the velocity head at its outlet, determine (i) the pressure head at the top of the draft tube, (ii) total head at the top taking tail race level as datum, (iii) power of water at outlet of runner, (iv) power of water at the end of the draft tube, (v) the power lost in the draft tube.*

Solution

From Eq. 21.32, we have

$$\frac{p_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + h_f$$

$$h_f = \frac{1}{2}\frac{V_3^2}{2g}$$

Thus $$\frac{p_2}{w} = -\left[H_s + \frac{V_2^2 - V_3^2}{2g}\right] + \frac{1}{2}\frac{V_3^2}{2g}$$

By continuity, we have

$$\frac{\pi \times (3)^2}{4} \times V_2 = 20 \times V_3$$

or $$V_3 = 0.3534\, V_2$$
$$= (0.3534 \times 5) = 1.767 \text{ m/s}$$

$\therefore$ $$\frac{p_2}{w} = -\left[5 + \frac{(5)^2 - (1.767)^2}{2 \times 9.81}\right] + \frac{1}{2}\frac{(1.767)^2}{2 \times 9.81}$$

$$= -6.0355 \text{ m}$$

Total head at the top of the draft tube is equal to

$$\frac{p_2}{w}+\frac{V_2^{2'}}{2g}+z_2 = \left[-6.0355+\frac{(5)^2}{2\times 9.81}+5\right]$$

$$= 0.2387 \text{ m}$$

$$Q = \left[\frac{\pi\times(3)^2}{4}\times 5\right] = 35.34 \text{ m}^3/\text{s}$$

∴ Power at the outlet of the runner

$$= (9\,810\times 35.34\times 0.2387)$$

$$= 82\,754 \text{ W} = 82.754 \text{ kW}$$

Total head at the end of the draft tube is equal to

$$\frac{V_3^2}{2g} = \frac{(1.767)^2}{2\times 9.81} = 0.1591 \text{ m}$$

∴ Power at the end of the draft tube

$$= (9810\times 35.34\times 0.1591)$$

$$= 55\,158 \text{ W} = 55.158 \text{ kW}$$

Head lost in the draft tube is equal to

$$\frac{1}{2}\frac{V_3^2}{2g} = \frac{1}{2}\frac{(1.767)^2}{2\times 9.81} = 0.079\,6 \text{ m}$$

∴ Power lost in the draft tube

$$= (9810\times 35.34\times 0.0796)$$

$$= 27\,596 \text{ W} = 27.596 \text{ kW}$$

Note. Power lost in the draft tube can also be determined by subtracting the power at the end of the draft tube from the power at the outlet of the runner.

Example 21.17. *In a Francis turbine of very low specific speed, the velocity of flow from inlet to exit of the runner remains constant. If the turbine discharges radially, show that the degree of reaction ρ can be expressed as*

$$\rho = \frac{1}{2}-\frac{1}{2}\left[\frac{\cot\theta}{(\cot\alpha-\cot\theta)}\right]$$

where α and θ are the guide and runner vane angles respectively and the degree of reaction ρ is equal to the ratio of pressure drop to the hydraulic work done in the runner, assuming that the losses in the runner are negligible.

Solution

Applying Bernoulli's equation between the inlet and exit of the runner and neglecting the potential difference, we get

$$\frac{p}{w}+\frac{V^2}{2g} = \frac{p_1}{w}+\frac{V_1^2}{2g}+\frac{V_w u}{g} \qquad \text{(for radial discharge)}$$

where $\frac{p}{w}$ and $\frac{p_1}{w}$ are the pressure heads at the inlet and the exit of the runner respectively.

Thus pressure head drop due to hydraulic work done in the runner is given by

$$\frac{p}{w}-\frac{p_1}{w} = \frac{V_1^2}{2g}-\frac{V^2}{2g}+\frac{V_w u}{g}$$

Now

$$\rho = \frac{\left(\frac{p}{w}-\frac{p_1}{w}\right)}{\frac{V_w u}{g}}$$

or

$$\rho = \frac{\frac{V_1^2}{2g}-\frac{V^2}{2g}+\frac{V_w u}{g}}{\frac{V_w u}{g}}$$

or

$$\rho = 1+\frac{1}{2}\left[\frac{V_1^2-V^2}{V_w u}\right] \qquad \text{...(i)}$$

For radial discharge

$$V_1 = V_{f1} = V_f$$

also

$$V_w = V\cos\alpha$$

$$V_f = V\sin\alpha$$

$$u = V_w - V_f\cot\theta$$

or

$$u = V[\cos\alpha-\sin\alpha\cot\theta]$$

and

$$V_1 = V_f = V\sin\alpha$$

Thus, introducing these values in Eq. (i) above and simplifying it, we get

$$\rho = \frac{1}{2}-\frac{1}{2}\left[\frac{\cot\theta}{\cot\alpha-\cot\theta}\right]$$

Example 21.18. *The inlet area of a draft tube provided for a Francis turbine is X times the exit area of the tube. Head loss in the tube = (K V²), where V is equal to square root of the product of velocities at inlet and exit of the draft tube. Show that draft tube efficiency is given by the expression* $\eta_d = [1-X^2-2g\,KX]$.

Solution

From Eq. 21.34, we have

$$\eta_d = \frac{[(V_2^2-V_3^2)/2g]-h_f}{[V_2^2/2g]}$$

$$V_3 = XV_2; h_f = K(V_2 \times V_3) = KXV_2^2$$

∴ By substitution, we get

$$\eta_d = [1 - X^2 - 2gKX]$$

Example 21.19. *Full load is supplied by the turbine shaft when the diameter of jet issuing from the nozzle is 150 mm. If the load suddenly drops to 36% of the full load what diameter of the jet should be attained by regulating the spear rod?*

Solution

Power developed, $P \propto Q \propto d^2$; where d is the diameter of the jet.

∴ Required jet diameter

$$= 150\left(\frac{36}{100}\right)^{1/2} = 90 \text{ mm}$$

Example 21.20. *A reaction turbine works at 450 r.p.m under a head of 120 m. Its diameter at inlet is 1.2 m and the flow area is 0.4 m². The angles made by the absolute and relative velocities at inlet are 20° and 60° respectively with the tangential velocity. Determine the volume flow rate, the power developed and the hydraulic efficiency. Assume whirl at outlet to be zero.*

Solution

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 1.2 \times 450}{60} = 28.27 \text{ m/s}$$

From inlet velocity triangle, we have

$$V_f \cot\alpha = u + V_f \cot\theta$$

Thus by substitution, we get

$$V_f \cot 20° = 28.27 + V_f \cot 60°$$

$$\therefore \quad V_f = 13.03 \text{ m/s}$$

$$\therefore \quad Q = (0.4 \times 13.03) = 5.212 \text{ m}^3/\text{s}$$

Work done by the runner per second is equal to the power developed P, which is thus given by

$$P = wQ\left(\frac{V_w u}{g}\right)$$

$$V_w = V_f \cot 20°$$
$$= (13.03 \times 2.7475) = 35.8 \text{ m/s}$$

$$\therefore \quad P = (9810 \times 5.212) \times \left(\frac{35.8 \times 28.27}{9.81}\right)$$
$$= 5\,275 \times 10^3 \text{ W}$$
$$= 5\,275 \text{ kW}$$

Hydraulic efficiency is given by

$$\eta_h = \frac{V_w u}{gH}$$

$$= \frac{35.8 \times 28.27}{9.81 \times 120}$$
$$= 0.86 \text{ or } 86\%$$

Example 21.21. *The casing diameter of a propeller turbine is 5 m and its hub is of 1.7 m diameter. When the turbine is operating at a speed of 140 r.p.m., under a head of 20.5 m and discharging 160 m³/s, the power developed is 25800 kW. At this regime the axial flow velocity is constant everywhere. At the inlet of the runner blade cascade the tangential velocity varies with radius and is equal to $\frac{12.4}{R}$ m/s where R is radius in m. At the exit of the blade cascade the tangential velocity is (0.3 R – 0.25) m/s.*

Determine, based on one-dimensional theory, the blade angles at inlet and exit at the hub and the tip. Determine the head utilized by the runner, the hydraulic and mechanical efficiencies.

Solution

The discharge Q through the runner is given by Eq. 21.35 as

$$Q = \frac{\pi}{4}\left(D^2 - d^2\right)V_f$$

$$Q = 160 \text{ m}^3/\text{s}; D = 5 \text{ m and } d = 1.7 \text{ m}$$

Thus by substitution, we get

$$160 = \frac{\pi}{4}\left[(5)^2 - (1.7)^2\right]V_f$$

$$\therefore \quad V_f = 9.21 \text{ m/s}$$

(i) Computations for section near the hub

$$u_i = \frac{\pi D_i N}{60}$$

$$= \frac{\pi \times 1.7 \times 140}{60} = 12.46 \text{ m/s}$$

$$\therefore \quad \theta = 76°59'$$

Tangential velocity V_w at inlet is given by

$$V_w = \frac{12.4}{R}$$

$$R = R_i = \frac{1.7}{2} = 0.85 \text{ m}$$

$$\therefore \quad V_w = \frac{12.4}{0.85} = 14.59 \text{ m/s}$$

From inlet velocity triangle, we have

$$\therefore \quad \tan\theta = \frac{V_f}{(V_w - u_i)}$$

$$= \frac{9.21}{(14.59-12.46)} = 4.3239$$

$\therefore$ $\theta = 76° 59'$

Tangential velocity V_{w_1} at exit is given by

$$V_{w_1} = (0.3\,R - 0.25)$$

$$= [(0.3 \times 0.85) - 0.25] = 0.005 \text{ m/s}$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{u_1 + V_{w_1}}$$

$$V_{f_1} = 9.21 \text{ m/s};\ u_1 = u_i = 12.46 \text{ m/s}$$

Thus

$\therefore$ $$\tan\phi = \frac{9.21}{(12.46+0.005)} = 0.7389$$

$\therefore$ $\theta = 36° 27'$

(ii) Computations for section at the outer periphery

$$u_0 = \frac{\pi D_0 N}{60}$$

$$= \frac{\pi \times 5 \times 140}{60} = 36.65 \text{ m/s}$$

Tangential velocity at inlet is given by

$$V_w = \frac{12.4}{R}$$

$$R = R_0 = \frac{5}{2} = 2.5 \text{ m}$$

$\therefore$ $$V_w = \frac{12.4}{2.5} = 4.96 \text{ m/s}$$

From inlet velocity triangle, we have

$$\tan(180-\theta) = \frac{V_f}{u_0 - V_w}$$

$$= \frac{9.21}{(36.65-4.96)} = 0.2906$$

$\therefore$ $\theta = 163° 48'$

Tangential velocity V_{w_1} at exit is given by

$$V_{w_1} = (0.3R - 0.25)$$

$$= [(0.3 \times 2.5) - 0.25] = 0.5 \text{ m/s}$$

From outlet velocity triangle, we have

$$\tan\phi = \frac{V_{f_1}}{u_1 - V_{w_1}}$$

$$V_{f_1} = V_f = 9.21 \text{ m/s};\ u_1 = u_0 = 36.65 \text{ m/s}$$

Thus $$\tan\phi = \frac{9.21}{(36.65-0/5)} = 0.2548$$

$\therefore$ $\phi = 14°18'$

(iii) Computations for mean diameter

Mean diameter $$D_m = \left(\frac{5+1.7}{2}\right) = 3.35 \text{ m}$$

$$u_m = \frac{\pi D_m N}{60}$$

$$= \frac{\pi \times 3.35 \times 140}{60} = 24.56 \text{ m/s}$$

Hydraulic efficiency $$\eta_h = \frac{V_w u - V_{w_1} u_1}{gH}$$

Tangential velocity V_w at inlet is given by

$$V_w = \frac{12.4}{R}$$

$$R = R_m = \frac{3.35}{2} = 1.675 \text{ m}$$

$\therefore$ $$V_w = \frac{12.4}{1.675} = 7.40 \text{ m/s}$$

Tangential velocity V_{w_1} at exit is given by

$$V_{w_1} = (0.3R - 0.25)$$

$$= [(0.3 \times 1.675) - 0.25] = 0.2525 \text{ m/s}$$

$$u = u_1 = u_m = 24.56 \text{ m/s};\ H = 20.5 \text{ m}$$

Thus by substitution, we get

$$\eta_h = \frac{(7.40-0.2525)\times 24.56}{9.81 \times 20.5}$$

$$= 0.873 \text{ or } 87.3\%$$

$\therefore$ Head utilized by the runner = $(0.873 \times 20.5) = 17.9$ m

Mechanical efficiency

$$\eta_m = \frac{P}{wQ\left(\frac{V_w u - V_{w_1} u_1}{g}\right)}$$

$P = 25800$ kW $= 25800 \times 10^3$ W; $w = 9810$ N/m^3; $Q = 160$ m^3/s

$$\left(\frac{V_w u - V_{w_1} u_1}{g}\right) = \eta_h \times H = (0.873 \times 20.5)$$

Thus by substitution, we get

$$\eta_m = \frac{25800 \times 10^3}{9810 \times 160 \times 0.873 \times 20.5}$$
$$= 0.918 \text{ or } 91.8\%$$

Example 21.22. *The draft tube of a Kaplan turbine has inlet diameter 2.6 m and inlet is set at 2.9 m above the tail race. When the turbine develops 2100 metric horse power under a net head of 6.5 m, it is found that the vacuum gage fitted at inlet to draft tube indicates a negative head of 4 m. If the turbine efficiency is 86%, determine the draft tube efficiency. If the turbine output is reduced to half with the same head, speed and draft tube efficiency, what would be the reading of the vacuum gage? Atmospheric pressure is 10.3 m of water and specific weight of water is 1000 kg/m^3.*

Solution

$$\eta_0 = \frac{P}{\frac{wQH}{75}}$$

or

$$0.86 = \frac{2100 \times 75}{1000 \times Q \times 6.5}$$

$\therefore$

$$Q = 28.175 \text{ m}^3/\text{s}$$

The velocity at inlet to draft tube is

$$V_2 = \frac{28.175}{(\pi/4) \times (2.6)^2} = 5.307 \text{ m/s}$$

From Eq. 21.33, we have

$$\frac{p_2}{w} = \frac{p_a}{w} - \left[H_s + (1-k)\frac{V_2^2 - V_3^2}{2g}\right]$$

$\frac{p_2}{w} = (10.3 - 4) = 6.3$ m; $\frac{p_a}{w} = 10.3$ m; and $H_s = 2.9$ m

Thus by substitution, we get

$$6.3 = 10.3 - \left[2.9 + (1-k)\frac{V_2^2 - V_3^2}{2g}\right]$$

or

$$(1-k)\frac{V_2^2 - V_3^2}{2g} = 1.1$$

From Eq. 21.34, efficiency of draft tube is given as

$$\eta_d = \frac{(1-k)\left[(V_2^2 - V_3^2)/2g\right]}{\left(V_2^2/2g\right)}$$

$$\frac{V_2^2}{2g} = \frac{(5.307)^2}{2 \times 9.81} = 1.435 \text{ m}$$

$\therefore$

$$\eta_d = \frac{1.1}{1.435} = 0.767 \text{ or } 76.7\%$$

When the turbine output is reduced to half,

$$P = \frac{2100}{2} = 1050 \text{ metric horse power}$$

Thus with the same head and the efficiency of the turbine, we have

$$0.86 = \frac{1050 \times 75}{1000 \times Q \times 6.5}$$

$\therefore$

$$Q = 14.088 \text{ m}^3/\text{s}$$

and

$$V_2 = \frac{14.088}{(\pi/4) \times (2.6)^2} = 2.653 \text{ m/s}$$

$\therefore$

$$\frac{V_2^2}{2g} = \frac{(2.653)^2}{2 \times 9.81} = 0.359 \text{ m}$$

For the same draft tube efficiency, we have

$$0.767 = \frac{(1-k)\left[(V_2^2 - V_3^2)/2g\right]}{0.359}$$

or

$$(1-k)\left[(V_2^2 - V_3^2)/2g\right] = 0.275$$

Thus from Eq. 21.33, we have

$$\frac{p_2}{w} = 10.3 - [2.9 + 0.275] = 7.125 \text{ m (abs.)}$$

$\therefore$ reading of vacuum gage would be

$$(7.125 - 10.3) = -3.175 \text{ m}$$

i.e., negative head of 3.175 m

Example 21.23. *A Francis turbine supplied through a 6 m diameter penstock has the following particulars:*

Output of installation	*63500 kW*
Flow	*117 m³/s*
Speed	*150 r.p.m*
Hydraulic efficiency	*92%*
Mean diameter of turbine at entry	*4 m*
Mean blade height at entry	*1 m*
Entry diameter of draft tube	*4.2 m*
Velocity in tail race	*2.4 m/s*

The static pressure head in the penstock measured just before entry to the runner is 57.4 m. The point of measurement is 3 m above the level of the tail race. The loss in the draft tube is equivalent to 30% of the velocity head at entry to it. The exit plane of the runner is 2 m above the tail race and the flow leaves the runner without swirl. Determine:

(a) The overall efficiency,

(b) The direction of flow relative to the runner at inlet,

(c) The pressure head at entry to the draft tube.

Solution

(a) The net head H for the turbine is given by Eq. 21.2 as

$$H = \left(\frac{p_1}{w}+\frac{V_1^2}{2g}+Z_1\right)-\left(Z_2+\frac{V_2^2}{2g}\right)$$

$$\frac{p_1}{w} = 57.4 \text{ m}; V_1 = \frac{117}{(\pi/4)\times(6)^2} = 4.14 \text{ m/s}; (Z_1 - Z_2) = 3 \text{ m};$$

and $V_2 = 2.4$ m/s.

Thus by substitution, we get

$$H = \left[57.4+\frac{(4.14)^2}{2\times 9.81}+3-\frac{(2.4)^2}{2\times 9.81}\right] = 60.98 \text{ m}$$

The overall efficiency is given by

$$\eta_0 = \frac{P}{wQH}$$

$$= \frac{63500\times 10^3}{9810\times 117\times 60.98}$$

$$= 0.907 \text{ or } 90.7\%$$

(b) Neglecting the vane thickness, the velocity of flow at inlet is given by Eq. 21.28 as

$$V_f = \frac{Q}{\pi BD}$$

$$Q = 117 \text{ m}^3/\text{s}; B = 1 \text{ m}; \text{ and } D = 4 \text{ m}$$

Thus by substitution, we get

$$V_f = \frac{117}{\pi\times 1\times 4} = 9.31 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi\times 4\times 150}{60} = 31.42 \text{ m/s}$$

Hydraulic efficiency

$$\eta_h = \frac{V_w u}{gH}$$

or

$$0.92 = \frac{V_w\times 31.42}{9.81\times 60.98}$$

$\therefore$

$$V_w = 17.52 \text{ m/s}$$

The direction of flow relative to the runner at inlet is given by

$$\tan\theta = \frac{V_f}{u-V_w}$$

$$= \frac{9.31}{(31.42-17.52)} = 0.6698$$

$\therefore$

$$\theta = 146°11'$$

(c) The pressure head at entry to the draft tube is given by Eq. 21.32 as

$$\frac{p_2}{w} = -\left[H_s+\frac{V_2^2-V_3^2}{2g}\right]+h_f$$

$$H_s = 2 \text{ m}; V_2 = \frac{117}{(\pi/4)\times(4.2)^2} = 8.44 \text{ m/s}; V_3 = 2.4 \text{ m/s}$$

and

$$h_f = 0.30\left(\frac{V_2^2}{2g}\right) = 0.30\times\frac{(8.44)^2}{2\times 9.81} = 1.09 \text{ m}$$

Thus by substitution, we get

$$\frac{p_2}{w} = -\left[2.0+\frac{(8.44)^2-(2.4)^2}{2\times 9.81}\right]+1.09$$

$$= -4.25 \text{ m}$$

SUMMARY OF MAIN POINTS

1. Hydraulic turbines are the machines which use the energy of water (hydro-power) and convert it into electrical power which is known as hydroelectric power.
2. Hydraulic turbine consists of a wheel called runner having a number of specially designed vanes or blades or buckets. The water possessing a large amount of hydraulic energy when strikes the runner, it does work on the runner and causes it to rotate. The mechanical energy so developed is supplied to the generator coupled to the runner, which then generates electrical energy.
3. The water surface in the storage reservoir is known as *head race*, and that in the tail race channel is known as *tail race*.
4. The difference between the head race level and the tail race level is known as gross head H_1.

   The head available at the entrance to the turbine is known as net head H which is given by

   $$H = H_1 - h_f$$

   where

   h_f = total loss of head between the head race and the entrance of the turbine.
5. The various efficiencies of the turbines may be expressed as (i) Hydraulic efficiency, (ii) Machanical efficiency, (iii) Volumetric efficiency and (iv) Overall efficiency.

   Hydraulic efficiency η_h is the ratio of the power developed by the runner to the net power supplied by the water at the entrance to the turbine.

   Mechanical efficiency η_m is the ratio of the power available at the turbine shaft to the power developed by the runner.

   Volumetric efficiency η_v is the ratio of the quantity of water actually striking the runner and the quantity of water supplied to the turbine.

   Overall efficiency η_o is the ratio of the power available at the turbine shaft to the power supplied by the water at the entrance to the turbine.

   The relationship between the various efficiencies is given as

   $$\eta_o = \eta_h \times \eta_v \times \eta_m$$
6. The hydraulic turbines may be classified as (i) Impulse turbines and (ii) Reaction turbines.

   In an *impulse turbine* all the available energy of water is converted into kinetic energy by passing it through a contracting nozzle provided at the end of the penstock which carry water from the storage reservoir to the turbine. The water coming out of the nozzle is formed into a free jet which impingess on a series of buckets of the runner thus causing it to revolve.

   Pelton wheel is the most commonly used impulse turbine.

   In a *reaction turbine* at the entrance to the runner only a part of the available energy of water is converted into kinetic energy and a substantial part remains in the form of pressure energy.

   Francis turbine and *Kaplan turbine* are the two most commonly used reaction turbines.

   The impulse turbines may be classified as high head turbines, while the reaction turbines are the medium to low head turbines.
7. A Pelton wheel is tangential flow turbine in which the runner consists of circular disc with a number of bucket evenly spaced round its periphery. One or more nozzles are mounted so that each directs a jet along a tangent to the circle through the centres of the buckets called the pitch circle.

   For a Pelton wheel turbine

   (i) The velocity of the jet V is given by

   $$V = K_v \sqrt{2gH}$$

   where

   K_v = coefficient of velocity for the nozzle which ranges from 0.97 to 0.99; and

   H = net head

   (ii) The velocity of the wheel u at pitch circle is given by

   $$u = K_u \sqrt{2gH}$$

   where

   K_u = speed ratio which ranges from 0.43 to 0.47

   (iii) Least diameter d of the jet is given by

   $$d = \left[\frac{4Q}{\pi K_v \sqrt{2gH}}\right]^{1/2}$$

   where

   Q = discharge through the jet.

   (iv) Pitch diameter D is given by

   $$D = \frac{60 K_u \sqrt{2gH}}{\pi N}$$

   where

   N = speed of rotation of wheel in r.p.m

   (v) The number of buckets Z is given by

   $$Z = \left(\frac{D}{2d} + 15\right) = (0.5\, m + 15)$$

   where

   $m = (D/d)$
8. The hydraulic efficiency of a Pelton wheel is given by

   $$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2}$$

   where

   ϕ = angle of the bucket at the outlet tip; and

   k = fraction by which the relative velocity at exit tip of the bucket is reduced.

   For the maximum hydraulic efficiency the condition is $u = \dfrac{V}{2}$; and

   $$(\eta_h)_{max} = \frac{1}{2}(1 + k\cos\phi)$$
9. Francis turbine is a mixed flow turbine in which water enters the runner at the outer pariphery in the radial direction and leaves it at the centre in the direction parallel to the axis of rotation of the runner.

   For a Francis turbine

   (i) The ratio of the width B and the diameter D of the runner is represented by n, i.e., $n = (B/D)$ the value of which ranges from 0.10 to 0.45.

   (ii) The ratio of the velocity of flow V_f at the inlet tip of vane to the spouting velocity $\left(\sqrt{2gH}\right)$ is known as *flow ratio* ψ, i. e., $\psi = \left(V_f / \sqrt{2gH}\right)$ the value of which ranges from 0.15 to 0.3.

   (iii) In this case also the speed ratio K_u is defined as

   $$K_u = \frac{u}{\sqrt{2gH}}$$

   The value of K_u range from 0.60 to 0.90.

   (iv) The discharge through a Francis turbine is given by

   $$Q = k\pi BD\, V_f = k\pi\, n\, D^2\, V_f$$

   where

   k = a factor which allows for the thickness of the vanes.
10. The hydraulic efficiency of a Francis turbine is given by

    $$\eta_h = \frac{V_w u}{gH}$$

    where

    V_w = velocity of whirl at inlet; and

    u = tangential velocity at inlet
11. A Kaplan turbine is an axial flow turbine which is suitable for low heads, and hence requires a large quantity of water to develop large amount of power.

    For a Kaplan turbine

    (i) The ratio of the hub diameter d and runner outside diameter D is represented by n, i.e., n = (d/D), the value of which varies from 0.35 to 0.60.

    (ii) The discharge through a Kaplan turbine runner is given by

    $$Q = \frac{\pi}{4}\left(D^2 - d^2\right)V_f = \frac{\pi}{4}\left(D^2 - d^2\right)\psi\sqrt{2gH}$$

    The value of flow ratio for a Kaplan turbine is around 0.70.

    (iii) The runner of a Kaplan turbine has four or six blades which are warped or twisted. The runner blades can be turned about their own axis so that even at part load when a lower discharge is flowing through the runner, a high efficiency can be attained.
12. A draft tube is a pipe of gradually increasing cross-sectional area used for discharging water from the exit of the runner of a reaction turbine to the tail race.

    The efficiency of draft tube is given by

    $$\eta_d = \frac{\left[\left(V_2^2 - V_3^2\right)/2g\right] - h_f}{V_2^2/2g}$$

    where

    V_2 = velocity at the inlet of the draft tube

    V_3 = velocity at the outlet of the draft tube

    and

    h_f = loss of head in the draft tube
13. A governor is a device used to regulate the discharge through the runner of a turbine so that under varying loads the speed of rotation of the runner remains constant.

    Oil pressure governor is commonly used with modern turbines.

14. The runaway speed of a turbine runner is the maximum speed that may be attained by it if the external load suddenly drops to almost zero value and at the same time the governing mechanism of the turbine fails.

15. A surge tank is provided to store water when the discharge through the runner is reduced by the governing operation due to reduction in the load and supply the same when due to increase in load more discharge is required through the runner.

PROBLEMS

21.1 Show from first principles that the peripheral coefficient of a Pelton wheel is 0.5.

21.2 For the kinetic energy at the exit from buckets of Pelton turbine to be minimum, prove that

$$\rho = \frac{u}{V} = \frac{(k-\cos\phi)k}{(k-1)^2 + 2k(1-\cos\phi)}$$

where u = bucket velocity, V = jet velocity, k = ratio of relative velocity at exit to that at inlet and ϕ stands for the bucket vane angle at exit [which is same as angle $(180° - \phi)$ in Fig. 21.4]. Hence, show that the hydraulic efficiency is given by

$$\eta_h = \frac{2C_v^2 k}{(1+k)} \quad \text{where } \phi = 180°$$

21.3 Design a Pelton wheel which is required to develop 1 500 kW, when working under a head of 160 m at a speed of 420 r.p.m. The overall efficiency may be taken as 85% and assume other data required.

[*Ans.* D = 1.17 m; No. of jets = 3; d = 93.21 mm; No. of buckets = 21]

21.4 A double jet Pelton wheel is required to generate 7500 kilowatts when the available head at the base of the nozzle is 400 m. The jet is deflected through 165° and the relative velocity of the jet is reduced by 15% in passing over the buckets. Determine (i) the diameter of each jet, (ii) total flow, (iii) force exerted by the jet on buckets in tangential direction. Assume generator efficiency of 95%, overall efficiency of 80%, K_v = 0.97, and K_u = 0.46.

[*Ans.* (i) 136.5 mm; (ii) 2.515 m³/s; (iii) 103.42 kN]

21.5 The Pykara Power House is equipped with inpulse turbine of Pelton type. Each turbine delivers a maximum power of 14 000 kW when working under a head of 840 m and running at 600 r.p.m. Find the least diameter of the jet and the mean diameter of the wheel. What would be the approximate diameter of orifice at the nozzle tip? Determine the value of the jet ratio and state if it is within the limits. Specify the number of buckets for the wheel. Take the overall efficiency of the turbine as 89.2 per cent. Assume C_v = 0.988 and K_u = 0.45.

[*Ans.* 138.3 mm; 1839 mm; 173 mm (= 1.25 × 138.3); 13.3 hence it is within safe limits; 22]

21.6 A reaction turbine works under a head of 6 m. The guide blades are inclined at 30° to the tangent at periphery and the runner vanes make 110° to the forward tangent at the periphery at inlet. If the discharge is radial and if all the exit velocity is wasted, find the hydraulic efficiency of the turbine. Assume velocity of flow to be constant. Find also the velocity of flow.

[Hint. For η_h use the expression derived in Illustrative Example 21.7]

[*Ans.* 87.89%; 3.77 m/s]

21.7 An inward flow reaction turbine works under a head of 22.5 m. The external and internal diameters of the runner are 1.35 m and 1 m respectively. The angle of guide vanes is 15° and the moving vanes are radial at inlet. Radial velocity of flow through runner is constant and there is no velocity of whirl at outlet. Determine the speed of the runner in r.p.m., and the angle of vane at outlet. If the turbine develops 275 kW, find the specific speed. Neglect friction losses.

[*Ans.* 206.5 r.p.m; 19° 53°; 69.88 r.p.m]

21.8 An inward flow reaction turbine is required to develop 300 kW at 200 r.p.m. The active head at the turbine is 18 m. Determine the outside and inside diameters, the inlet and exit angles for the vanes and the exit angle for the guide vanes. Assume the inlet diameter equal to twice the outlet diameter, the hydraulic efficiency as 80%, the constant radial velocity of flow of 3.6 m/s through the runner, the mechanical efficiency as 95% and the width ratio at 0.10. Water leaves the runner radially.

[*Ans.* 1.406 m; 0.703 m; 144° 56′; 26° 4′; 20° 34′]

21.9 Show that in a turbine with radial vanes at inlet and outlet and operating under head h, if the velocity of flow at outlet is C times that at inlet, the peripheral velocity u and the hydraulic efficiency η_h are given by

$$u = \sqrt{\frac{2gh}{2 + C^2\tan^2\alpha}}\,;\ \eta_h = \frac{2}{2 + C^2\tan^2\alpha}$$

where α is guide blade angle.

21.10 Two inward flow reaction turbines have same runner diameter viz., 0.60 m and the same efficiency, they work under the same head and they have the same velocity of flow viz., 6 m/s. One of the runners A revolves at 520 r.p.m., and has an inlet vane angle of 65°. If the other runner B has an inlet vane angle of 110°, at what speed should it run? [*Ans.* 600 r.p.m.]

21.11 Find the leading dimensions of the runner of an inward flow Francis turbine producing 625 kW, at 1000 r.p.m., under a head of 100 m. Assume the guide vane angle of 16°, the axial length of the blade at inlet 0.1 times the outer diameter, inner radius 0.6 times the outer radius, radial velocity of flow constant, final discharge radial, hydraulic efficiency 87%, mechanical efficiency 97%, allowance for the blade thickness 5%.

[*Ans.* 541.3 mm; 324.8 mm; 54.13 mm; 90.22 mm; 78°26′; 26° 55′]

21.12 A reaction turbine runner has inlet diameter 3.5 m, exit diameter 2.1 m, width 0.5 m both at inlet and exit, and vane angles with the direction of motion 60° and 150°at inlet and exit respectively. If the guide blade angle is 15° and the rate of flow of water is 15 m³/s, calculate the required speed of the runner for smooth flow at the inlet. For this condition, also compute the torque exerted on the runner, the power developed, the energy extracted per unit weight of water, and the pressure drop through the runner.

[*Ans.* 47 r.p.m.; 309.903 kN.m; 1 525.3 kW; 10.366 N.m/N; 60.15 kN/m²]

21.13 Explain the purpose of providing (i) scroll casing; (ii) stay vanes; (iii) guide vanes, for a reaction turbine.

21.14 What are the uses of a draft tube? Describe with neat sketches different types of draft tubes.

21.15 (a) Explain briefly the principles on which a Kaplan turbine works.

(b) A Kaplan turbine runner develops 9 300 kW under a net head of 7.4 m. Mechanical efficiency of the wheel is 86%, speed ratio based on outer diameter is 2.2 and the flow ratio is 0.66. Diameter of the boss is 0.35 times the external diameter of the wheel. Determine (i) diameter of the runner; (ii) its synchronous speed; (iii) the specific speed of the runner. Assume mechanical efficiency equal to overall efficiency.

[*Ans.* (i) 5.21 m; (ii) 97.12 r.p.m.; (iii) 767.4 r.p.m.]

21.16 A Kaplan turbine develops 2 100 kW under a net head of 7.2 m with an overall efficiency of 87%. It is to be fitted with an elbow-type draft tube (draft tube efficiency 88%) having its inlet 1.8 m diameter. Determine how much above or below the tail race level should the draft tube inlet be set so that vacuum pressure there does not exceed 460 mm of mercury.

[*Ans.* 1.83 m below the tail race]

21.17 A Kaplan turbine develops 2575 kW under a net head of 7.5 m. It is provided with an elbow type draft tube with a circular inlet of 2.5 m diameter. The inlet is set at a height of 1.5 m above the tail race level. A vacuum gage connected to draft tube inlet records a reading of 36.3 kN/m². If the efficiency of the draft tube is 78%, calculate the efficiency of the Kaplan turbine. If the ratio of area of circular inlet and rectangular exit of the draft tube is 1 : 5, determine the power lost due to friction in the tube. If the turbine output were reduced to 1 250 kW, head and speed remaining unchanged, determine the vacuum gage reading.

[*Ans.* 96%; 181.9 kW; 19.78 kN/m²]

21.18 What is a surge tank and a forebay and what are their functions? Describe with neat sketches different types of surge tanks.

21.19 What do you understand by governing of hydraulic turbines? Explain with sketches the working of an oil pressure governor.

21.20 Write short notes on: Scroll casing, draft tube, wicket gates, speed ratio, flow ratio, turbine efficiencies, relief valve, oil pressure governor, electro-hydraulic governor.

21.21 A Kaplan turbine produces 44 000 kW under a head of 25 m with an overall efficiency of 90%. Taking the valve of speed ratio K_s as 1.6, flow ratio ψ as 0.5 and the hub diameter as 0.35 times the outer diameter, find the diameter and speed of the turbine. [*Ans.* 5.11 m; 132.4 r.p.m.]

21.22 A reaction turbine 0.5 m diameter when running at 600 r.p.m., develops 265 kW, the flow through the turbine being 0.75 m³/s. Under the above operating condition the pressure head at the entrance to the turbine is 28 m, the elevation of the turbine above the tail water level is 1.3 m and the velocity of flow at entrance to the turbine runner is 3.7 m/s. Assuming the runner vane angle at inlet as 90°, calculate the effective head on the turbine and the efficiency. [*Ans.* 42.58 m; 84.6%]

Performance of Turbines

Chapter 22

22.1 INTRODUCTION

Turbines are often required to work under varying conditions of head, speed, output and gate opening. As such, in order to predict their behaviour, it is essential to study the performance of the turbines under the varying conditions. The variations in the conditions for working may be as follows:

(i) The head and hence the output of the turbine may change, the speed being correspondingly adjusted so that no appreciable change in efficiency occurs, the gate opening remaining constant.

(ii) The output may be varied by the movements of the gates or the spear (or needle); the head and speed remaining constant. These are normal operating conditions for most of the turbines.

(iii) The head and speed may vary. Such variations are common particularly in low head units. It may, however, be stated that although the speed is permitted to vary within very narrow limits, the head may vary by even 50% or more.

(iv) The speed may be allowed to vary by adjusting the load on the turbine; the head and gate opening remaining constant. These conditions can be developed only for laboratory turbines or those in the test plant and are otherwise uncommon.

In order to predict the behaviour of a turbine working under varying conditions and to facilitate comparison between the performances of the turbines of the same type but having different outputs and speeds and working under different heads, it is often convenient to express the test results in terms of certain *unit quantities*. Similarly to facilitate the comparison between the turbines of different types certain *specific quantities* are also required to be considered. Both these quantities are explained in the following sections.

22.2 PERFORMANCE UNDRE UNIT HEAD—UNIT QUANTITIES

From the output of a turbine corresponding to different working heads it is possible to compute the output which would be developed if the head was reduced to unity (say 1 metre); the speed being adjusted so that the efficiency remains unaffected. The efficiency of a given turbine will remain unchanged if the velocity triangle under working head H and under unit head are geometrically similar so that shock at entry is avoided.

Figure 22.1 shows two inlet velocity triangles for Francis turbine under its working head H and under unit head, the subscript u being used to denote unit conditions. Both these triangles are similar if their corresponding sides are parallel. It then follows that

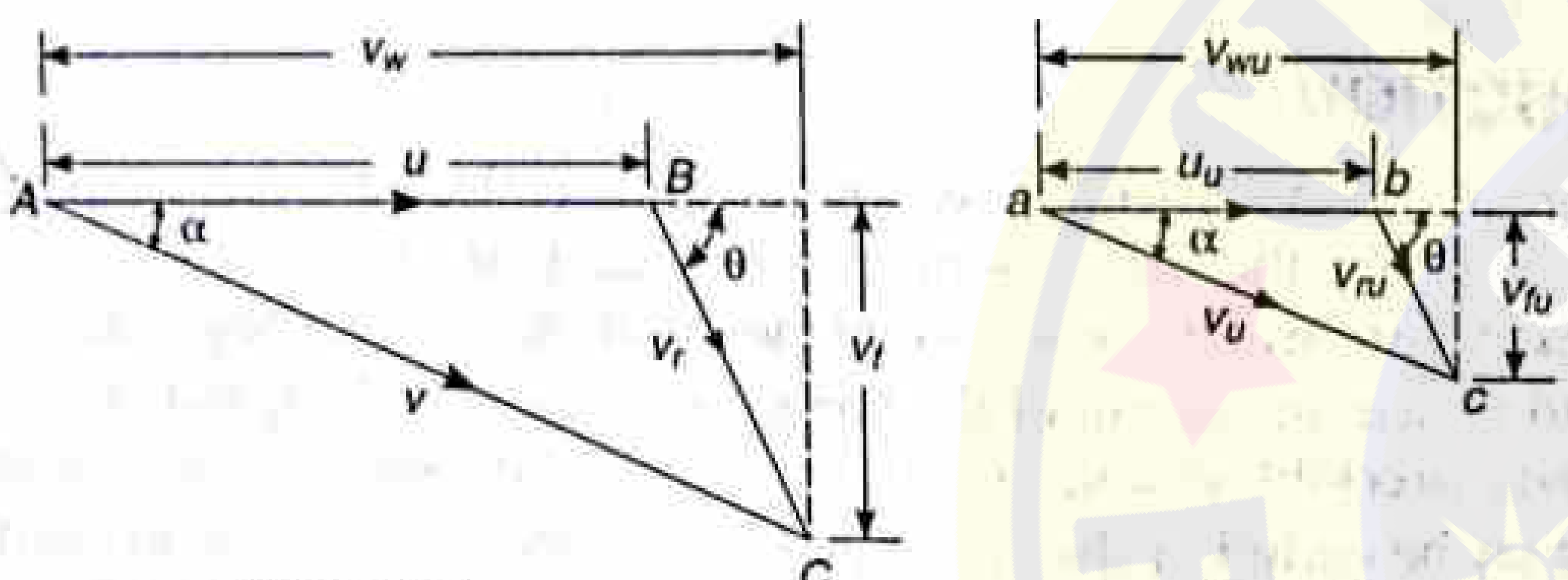
$$\frac{u}{V_w} = \frac{u_u}{(V_{wu})} \text{ and } \frac{u}{V_f} = \frac{u_u}{(V_{fu})} \quad ...(22.1)$$

Further it may be shown that

$$\frac{(V_w u/g)}{(V_f^2/2g)} = \frac{(V_{wu})(u_u)/g}{(V_{fu}^2/2g)} \; ; \; \frac{(V_w u/g)}{(V_{f1}^2/2g)} = \frac{(V_{wu})(u_u)/g}{(V_{1u}^2/2g)}$$

The above expression indicates that the ratio of the useful energy to wasted velocity energy is same under the working head and the unit head conditions. In other words the required condition of unchanged efficiency is fulfilled. Thus

$$\eta_h = \eta_{hu} ; \text{or } \frac{V_w u}{gH} = \frac{V_{wu} u_u}{g \times 1} \quad ...(22.2)$$



For head H — For unit head

Figure 22.1 *Inlet velocity triangles of head H and unit head*

Eliminating V_w and V_{wu} from Eqs. 22.1 and 22.2 we obtain

$$u_u^2 = \frac{u^2}{H} \; ; \quad \text{or } u_u = \frac{u}{\sqrt{H}} \quad ...(22.3)$$

and

$$V_{fu}^2 = \frac{V_f^2}{H} \; ; \quad \text{or } V_{fu} = \frac{V_f}{\sqrt{H}} \quad ...(22.4)$$

Since $u = \frac{\pi DN}{60}$ and $u_u = \frac{\pi D N_u}{60}$ it follows that

$$N_u = \frac{N}{\sqrt{H}} \quad ...(22.5)$$

where N_u represents the speed of the turbines under unit head and it is known as *unit speed*.

Further it can be shown that both the speed ratio K_u and the flow ratio ψ remain constant under the head H and the unit head.

The rate of flow $Q = (k\pi DB)V_f$ and $Q_u = (k\pi DB)V_{fu}$, hence

$$\frac{Q_u}{Q} = \frac{V_{fu}}{V_f} = \frac{1}{\sqrt{H}} ; \; \therefore Q_u = \frac{Q}{\sqrt{H}} \quad ...(22.6)$$

where Q_u represents the discharge flowing through the turbine under a unit head and is known as *unit discharge*.

The power developed

$$P = (wQH)\eta_0$$

and

$$P_u = (wQ_u \times 1)\eta_c, \text{ hence}$$

$$\frac{P_u}{P} = \frac{Q}{Q_u \times H} = \frac{1}{H^{3/2}} ; \; \therefore P_u = \frac{P}{H^{3/2}} \quad ...(22.7)$$

where P_u represents the output of the turbine under unit head and is known as *unit power*.

Although the above expressions have been deduced from the velocity triangles of a Francis turbine, the same are applicable to all types of turbines. The advantage of obtaining the performance at unit head is that from the known values of discharge Q, speed N and power P of a turbine working under head H, the value Q_1, N_1 and P_1 corresponding to some other head H_1, may be computed as indicated below:

Since Q_u, N_u and P_u will be same for both the heads, we have

$$Q_u = (Q/\sqrt{H}) = (Q_1/\sqrt{H_1}); \qquad \therefore Q_1 = (Q\sqrt{H_1})/\sqrt{H}$$

$$N_u = (N/\sqrt{H}) = (N_1/\sqrt{H_1}); \qquad \therefore N_1 = (N\sqrt{H_1})/\sqrt{H}$$

$$P_u = (P/H^{3/2}) = (P_1/H_1^{3/2}); \qquad \therefore P_1 = (PH_1^{3/2})/H^{3/2}$$

The above derived expressions are based on the assumption that the efficiencies remain constant at all the heads, which is however not correct. In practice it has been found that the efficiency varies with the head which will cause the scale effect.

The expressions for the various quantities as derived above can be used for comparing the performance of any one turbine only under different conditions of operation. However, according to another definition for the unit quantities as indicated below more general expressions for the unit quantities may be obtained which would facilitate a comparison between the performance of different turbines of the same type. According to this definition the unit speed, unit discharge and unit power are defined respectively as the speed, discharge and power of a turbine having a runner diameter of 1 m and operating under a head of 1 m (or unit head).

Thus from Eq. 22.3

$$\frac{\pi \times 1 \times N_u}{60} = \frac{\pi ND}{60} \times \frac{1}{\sqrt{H}} \; ; \text{or } N_u = \frac{ND}{\sqrt{H}} \quad ...(22.5\text{ a})$$

Similarly from Eqs. 22.6 and 22.7 the following expressions for Q_u and P_u may be obtained

$$Q_u = \frac{Q}{D^2\sqrt{H}} \quad ...(22.6\text{ a})$$

and

$$P_u = \frac{P}{D^2 H^{3/2}} \quad ...(22.7\text{ a})$$

By using these expressions the values of the unit quantities may be determined for any one turbine and the same applies to all the similar turbines of the same type. It may however be stated that the values of the unit quantities for similar turbines will be equal only if their efficiencies are equal. Since the efficiencies of even the similar turbines are not equal the values of the unit quantities will also differ.

22.3 PERFORMANCE UNDER SPECIFIC CONDITIONS

The performance of turbines under unit head facilitate the comparison of turbines of the same type. However, the turbines of different types may be compared by considering an imaginary turbine called *specific turbine*. The specific turbine is an imaginary turbine which is identical in shape, geometrical proportions, blade angles, gate setting etc., with the actual turbine but reduced to such a size that it will develop one kilowatt power (or one metric horse power) under unit head. The suffix *s* is used to denote the symbols expressing the performance of the specific turbine.

Figure 22.2 shows the actual and the specific runners for two types of turbines. Since the actual turbine under unit conditions and the specific turbine both work under a head of 1 metre it is evident that the velocity triangle *abc* in Fig. 22.1 will hold good for the specific turbine as well. In other words $u_s = u_u$, $V_{fs} = V_{fu}$ etc. Now $Q_u = \pi D(nD)V_{fu}$ and $Q_s = \pi D_s(nD_s)V_{fs}$

$$\therefore \quad \frac{Q_u}{Q_s} = \left(\frac{D}{D_s}\right)^2 \qquad \text{...(22.8)}$$

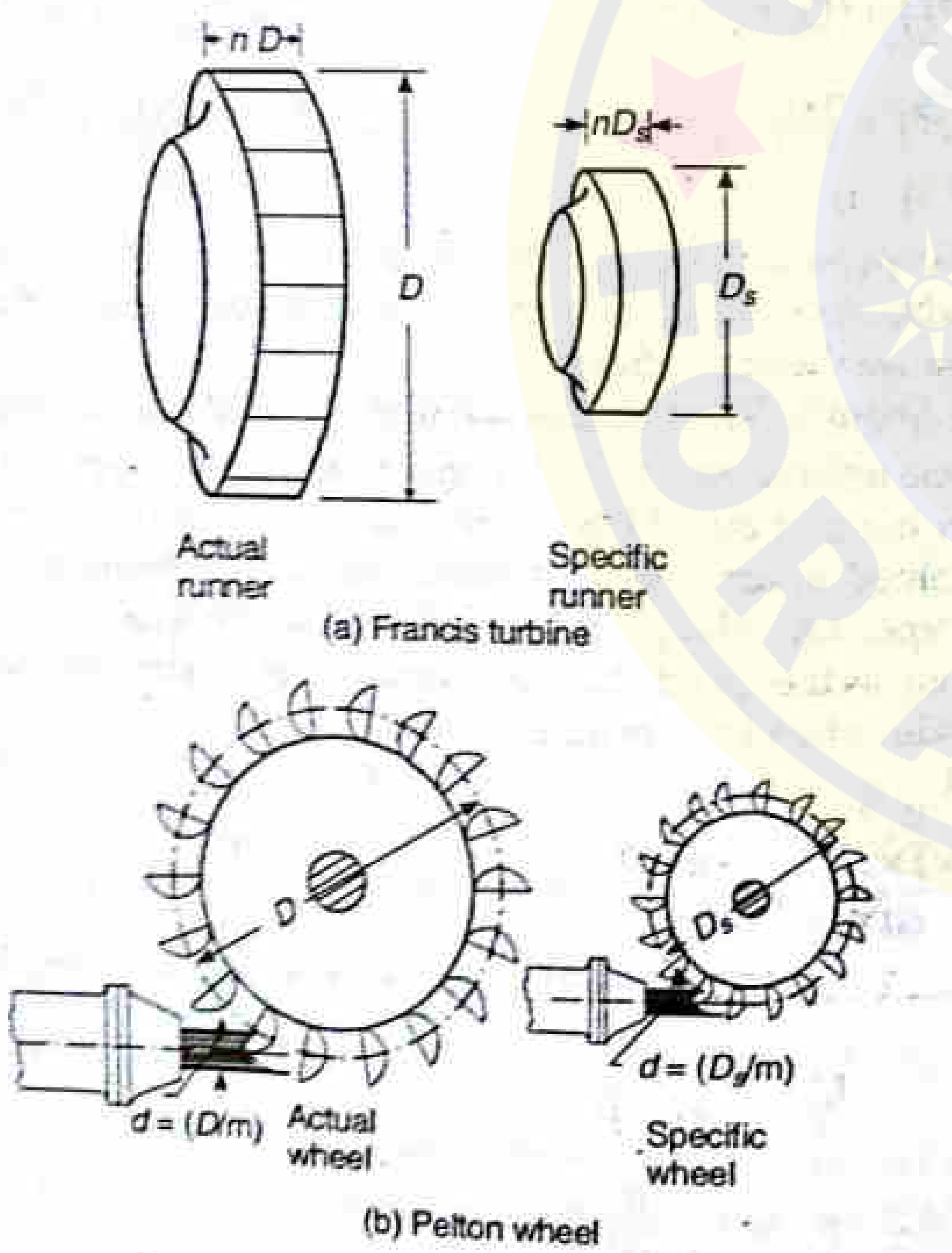


Figure 22.2 *Actual and specific runners*

Again $P_u = wQ_u \times 1 \times \eta_0$; and $P_s = wQ_s \times 1 \times \eta_0$

Assuming η_o to be same for both

$$\frac{P_u}{P_s} = \frac{Q_u}{Q_s} = \left(\frac{D}{D_s}\right)^2 \qquad \text{...(22.9)}$$

Equation 22.9 shows that the power of geometrically similar turbines working under the same head vary as the square of the runner diameters.

Since $\quad P_s = 1$ kW (or 1 metric h.p.) (by definition) and $P_u = \frac{P}{H^{3/2}}$

$$D_s = \frac{D}{\sqrt{P_u}} = \frac{DH^{3/4}}{\sqrt{P}}$$

Further $\quad u_u = \frac{\pi D N_u}{60}$

and $\quad u_s = \frac{\pi D_s N_s}{60}$

but $\quad u_u = u_s$

hence $\quad \frac{\pi D N_u}{60} = \frac{\pi D_s N_s}{60}$

$$\therefore \quad N_s = N_u \frac{D}{D_s} = N_u\sqrt{P_u} \qquad \text{...(22.10)}$$

but $\quad N_u = \frac{N}{\sqrt{H}}$

and $\quad P_u = \frac{P}{H^{3/2}}$

$$\therefore \quad N_s = \frac{N\sqrt{P}}{H^{5/4}} \qquad \text{...(22.11)}$$

This value of N_s, the speed of specific runner is known as *specific speed*. For any other turbine also exactly same relationship for N_s may be derived.

A significant point about the above noted expression for the specific speed is that it is independent of the dimensions or size, both of the actual turbine and of the specific turbine. It therefore means that all turbines of the same geometrical shape, working under the same value of K_u and ψ and thus having the same efficiency, will have the same specific speed, no matter what their sizes be and what powers they develop under what heads. As such it may be stated that N_s represents the specific speed of the actual turbine, as well as of the specific turbine. Therefore, the following general definition for the specific speed may be given.

The specific speed of any turbine is the speed in r.p.m. of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 kilowatt power when working under unit head (i.e., 1 metre).

In metric units the power is expressed in metric horse power and hence in metric units the following definition for the specific speed may be given.

The specific speed of any turbine is the speed in r.p.m. of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 metric horse power when working under unit head (i.e., 1 metre).

The specific speed is usually computed for the operating conditions corresponding to the maximum efficiency.

It may be seen from Eq. 22.11, which is dimensionally non-homogeneous, that the specific speed is not a pure number. By considering the dimensions of N, P and H it can be shown that the value of N_s in SI units is equal to about 0.858 times its value in metric units and the value of N_s in metric units is equal to about 4.44 times its value in F.P.S. units.

There exists a general trend to select a turbine of higher specific speed because higher specific speed of a turbine results in the reduction of the runner diameter as well as the overall size of the runner, due to which the weight and the cost of the runner are reduced. This may be explained from Eq. 22.11 which shows that for a given power output and head, an increase in the specific speed permits a higher speed of rotation for the turbine runner, corresponding to which even for a runner of smaller diameter a higher peripheral velocity may be developed. As such by increasing the specific speed the runner diameter as well as its overall size may be reduced. Further as shown in Illustrative Example 22.7, the specific speed of a turbine may be expressed in terms of unit speed N_u, unit discharge Q_u and overall efficiency η_o as

$$N_s = 3.13\, N_u \sqrt{Q_u \eta_o}$$

From the above expression it is seen that the specific speed may be increased by increasing the unit speed N_u or unit discharge Q_u. It is, however, more convenient to increase the specific speed by increasing unit discharge Q_u since by this as seen from Eq. 22.6 (a) for a given discharge Q and head H, the runner diameter is reduced proportionally to the square root of the unit discharge Q_u and the rotational speed N is increased proportionally to the specific speed. The unit discharge may be increased if the head H acting on the turbine is reduced and the discharge Q flowing through the runner is increased. As indicated in Chapter 21 a turbine operating under a low head and consequently required to allow larger discharge to flow through the runner must have a larger area of flow passage which may be obtained by designing the runner with axial flow right from its inlet end. Thus, axial flow turbines are high specific speed turbines. Further corresponding to high specific speed of an axial flow turbine the speed of rotation of its runner is also high, and hence for the same operating conditions, viz., output power (or discharge) and head, the diameter and the overall size of a runner of an axial flow turbine is much smaller than that for the runner of other types of turbines having lower specific speeds. This fact has been illustrated in Fig. 22.3 which shows relative sizes of different types of runners designed to develop same power and operating under the same head, for different specific speeds. It may be seen from Fig. 22.3 that for low specific speed radial flow runner a large diameter and relatively narrow fluid passage is required to be provided, and for increased specific speeds the runners of smaller diameters and wider flow passages are required to be provided.

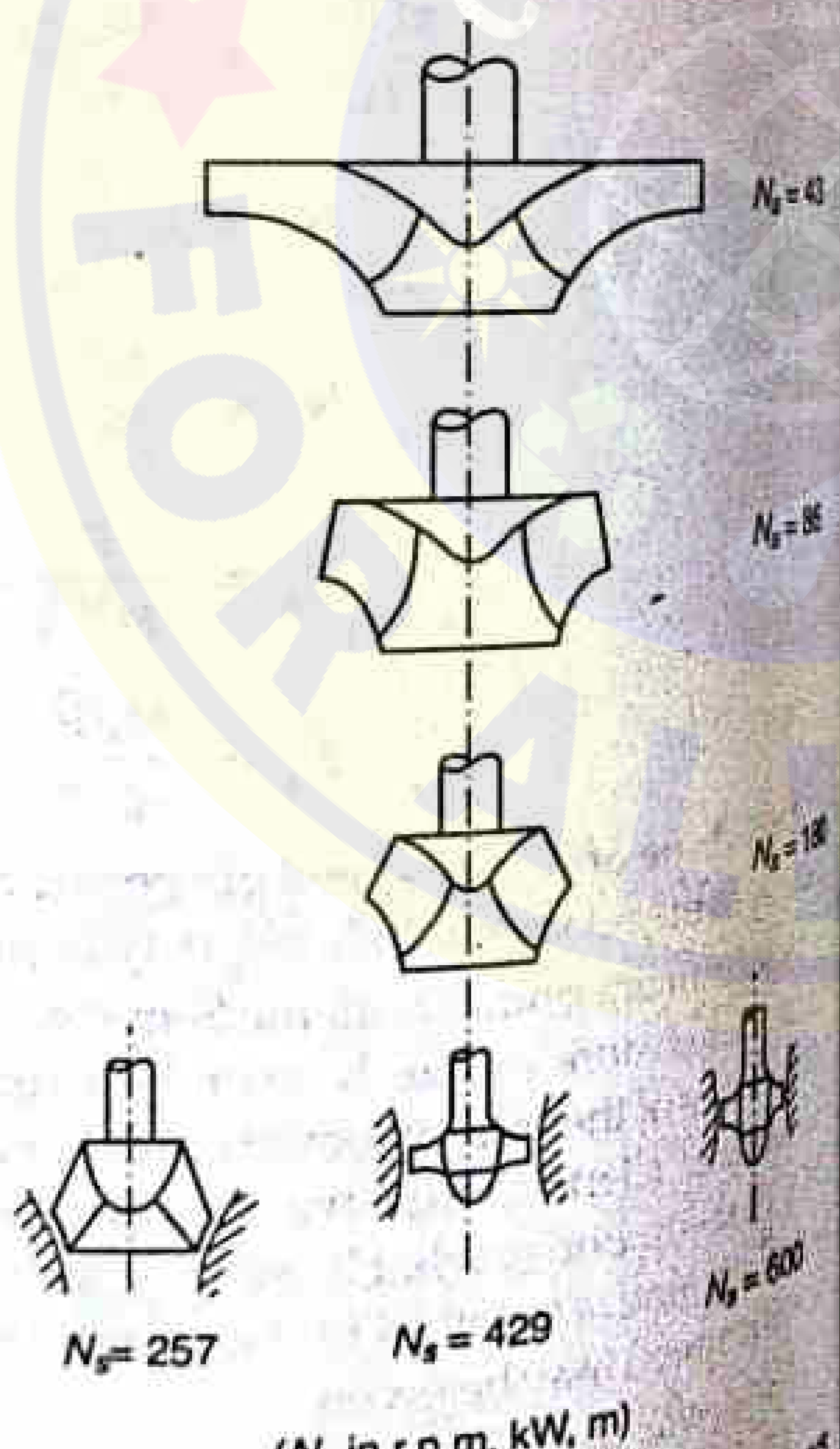


Figure 22.3 *Relative sizes of turbine runners developing same power under same head for different specific speeds*

It may thus be stated that an increase in the specific speed of a turbine results in the reduction of the diameter as well as the overall size of the turbine runner. Since this reduction in size of a turbine runner will lead to a corresponding reduction in the cost, it is evident that for economic reasons a turbine runner with highest specific speed possible should be selected.

22.4 EXPRESSIONS FOR SPECIFIC SPEEDS IN TERMS OF KNOWN COEFFICIENTS FOR DIFFERENT TURBINES

From the general expression for the specific speed represented by Eq. 22.11, it is possible to obtain the expressions for specific speed in terms of known coefficients for different turbines as discussed below.

1. Specific Speed of Pelton Wheel. An expression for the specific speed of Pelton wheel may be obtained in terms of K_v (or C_v), K_u, η_o and m as below:

$$u = K_u\sqrt{2gH} = \frac{\pi DN}{60}$$

$$\therefore \quad N = \left[\frac{60K_u\sqrt{2gH}}{\pi D}\right] = \left[\frac{84.6K_u\sqrt{H}}{D}\right]$$

$$P = wQH \times \eta_o$$

$$= \frac{9810}{1000}\left(\frac{\pi d^2}{4}\right) \times K_v\sqrt{2gH} \times H \times \eta_o$$

$$= 34.13\,(K_v d^2 \eta_o H^{3/2})\ \text{kW}$$

Since $\quad w = 9810\ \text{N/m}^3$ and $g = 9.81\ \text{m/s}^2$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$\therefore \quad N_s = \frac{(84.6)K_u(\sqrt{H})(34.13)^{1/2}(K_v d^2 \eta_o H^{3/2})^{1/2}}{D(H^{5/4})}$$

or

$$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{(D/d)}$$

or

$$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{m} \qquad ...(22.12)$$

Taking $K_u = 0.46$, $K_v = 0.98$ and $\eta_o = 85\%$, which are the usual values, Eq. 22.12 gives

$$N_s = \frac{207.5}{m} \quad ...(22.12 a)$$

In metric units since power P is expressed in metric horse power, we have

$$P = \frac{wQH}{75} \times \eta_o$$

$$= \frac{w(\pi/4)d^2 \times K_v\sqrt{2gH} \times H \times \eta_o}{75}$$

$$= 46.39\,(K_v d^2 H^{3/2}\eta_o)\text{ h.p.}$$

Since $w = 1000$ kg (f)/m^3 and $g = 9.81$ m/s^2

Thus $$N_s = \frac{(84.6)K_u(\sqrt{H})(46.39)^{1/2}(K_v d^2 \eta_o H^{3/2})^{1/2}}{D(H^{5/4})}$$

or $$N_s = \frac{(576.2)K_u(K_v\eta_o)^{1/2}}{m} \quad ...(22.13)$$

For the above noted values of K_u, K_v and η_o Eq. 22.12 (a) gives

$$N_s = \frac{242}{m} \quad ...(22.13 a)$$

Equation 22.13 gives a relationship between the specific speed N_s and jet ratio m for a single jet Pelton wheel. Since the value of m is more or less constant and in most of the cases it is equal to 12, almost all the Pelton wheels have the same specific speed. However, in some exceptional cases an abnormally low value of m equal to 7 has also been used. As such from Eq. 22.13 it is found that for m varying from 7 to 12 the value of N_s in SI units varies from about 30 to 17. Similarly from Eq. 22.13(a) for m varying from 7 to 12 the value of N_s in metric units varies from about 35 to 20. It is thus observed that for Pelton wheel the specific speed has quite a narrow range of variation.

For a multiple jet Pelton wheel having n number of jets, the expression for specific speed may be obtained in the same manner. In this case the discharge will be (nQ) and hence the power will be (nP). Therefore in this case the specific speed will be $\sqrt{n}$ times the specific speed for a single jet Pelton Wheel.

2. Specific Speed of Francis Turbine. For Francis turbines an expression for the specific speed may be obtained in terms of K_u, ψ and n as indicated below:

$$u = K_u\sqrt{2gH} = \frac{\pi DN}{60}$$

$$\therefore \quad N = \frac{84.6K_u\sqrt{H}}{D}$$

$$P = wQH \times \eta_o$$

$$= [w(k\pi n D^2)\psi\sqrt{2gH} \times (H \times \eta_o)]$$

$$= 136.51\,(k n D^2 \psi H^{3/2} \eta_o)\text{ kW}$$

Since $w = 9810$ N/m^3 and $g = 9.81$ m/s^2

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$\therefore \quad N_s = \frac{(84.6)K_u\sqrt{H} \times (136.51)^{1/2}(k\pi D^2\psi H^{3/2}\eta_o)^{1/2}}{DH^{5/4}}$$

or $$N_s = 988.45\left[K_u(\sqrt{k\, n\, \psi\, \eta_o})\right] \quad ...(22.14)$$

In metric units since power is expressed in metric horse power, we have

$$P = \frac{wQH}{75} \times \eta_o$$

$$= \frac{[w(k\,\pi\, n\, D^2)\psi\sqrt{2gH} \times (H \times \eta_o)]}{75}$$

$$= 185.54\,(k\, n\, D^2\, \psi\, H^{3/2}\, \eta_o)$$

since $w = 1000$ kg(f)/m^3 and $g = 9.81$ m/s^2

Thus $$N_s = \frac{(84.6)K_u\sqrt{H}(185.54)^{1/2}(k\pi D^2\psi H^{3/2}\eta_o)^{1/2}}{D(H^{5/4})}$$

or $$N_s = 1152.4\left[K_u(\sqrt{kn\psi\eta_o})\right] \quad ...(22.14 a)$$

Assuming k and η_o to be constant, Eq. 22.14 indicate that specific speed N_s for Francis turbines depends on speed ratio K_u, flow ratio ψ and breadth ratio n. Generally K_u ranges from 0.6 to 0.9, ψ ranges from 0.15 to 0.30 and n ranges from 0.10 to 0.45. The variation of an ψ or all of these will alter N_s, and hence a much greater range of N_s is available for Francis turbines as compared to that for Pelton wheel.

Francis turbine runners are classified as *slow runners* (N_s = 51 to 103), *normal or medium runners* (N_s = 103 to 154) and *fast runners* (N_s = 154 to 257). The corresponding ranges of N_s in metric units are for *slow runners* (N_s = 60 to120), for *normal* or *medium runners* (N_s = 120 to 180) and for *fast runners* (N_s = 180 to 300). It has been observed that the increase in N_s from 51 to 154 (or 60 to 180) is obtained by increasing n and ψ, but to attain the higher value, it would be necessary to increase simultaneously K_u, ψ and n. It may, however, be pointed out that in order to increase N_s the values of K_u and ψ cannot be increased to any value, because unduly higher values will result in increased hydraulic losses and consequently lower efficiency. It has been explained in Chapter 21 that as the value of n increases the runner shapes are also altered. For slow runners the flow is predominantly radial and exit is axial. For fast runners the flow is more or less axial both at entry and exit.

3. Specific Speed of Kaplan and Propeller Turbines. Since these axial flow turbines work under low heads, it is evident from Eq. 22.11 for N_s that these turbines have very high specific speed ranging from 257 to 858 (or 300 to 1000 in metric units). As for the Francis turbine in this case also it can be shown that

$$N_s = C_1\left(\sqrt{C_2}\right)\left[K_u\sqrt{\psi(1-n^2)\eta_o}\right] \quad ...(22.15)$$

where

$$C_1 = \frac{60\sqrt{2g}}{\pi} = 84.6\text{ ; and}$$

$$C_2 = \frac{w\pi\sqrt{2g}}{4} = 34.13$$

However, in metric units the value of C_2 is obtained as

$$C_2 = \frac{w\pi\sqrt{2g}}{4\times 75} = 46.39$$

Now if $\eta_o = 90\%;\ n = (d/D) = 0.35$ then

in SI units

$$N_s = 439.22\left(K_u\sqrt{\psi}\right) \quad ...(22.15\ a)$$

and in metric units

$$N_s = 512.07\left(K_u\sqrt{\psi}\right) \quad ...(22.15\ b)$$

22.5 PERFORMANCE CHARACTERISTIC CURVES

The turbines are generally designed to work at particular values of H, Q, P, N and η_o which are known as the designed conditions. But often the turbines are required to work at conditions different from those for which they have been designed. Therefore, it is essential to determine the exact behaviour of the turbines under the varying conditions by carrying out tests either on the actual turbines or on their small scale models. The results of these tests are usually graphically represented and the resulting curves are known as *characteristic curves*. For the sake of convenience the characteristic curves are plotted in terms of unit quantities. These characteristic curves are of the following three types:

(i) Constant head characteristic curves (or Main characteristic curves).

(ii) Constant speed characteristic curves (or Operating characteristic curves).

(iii) Constant efficiency curves

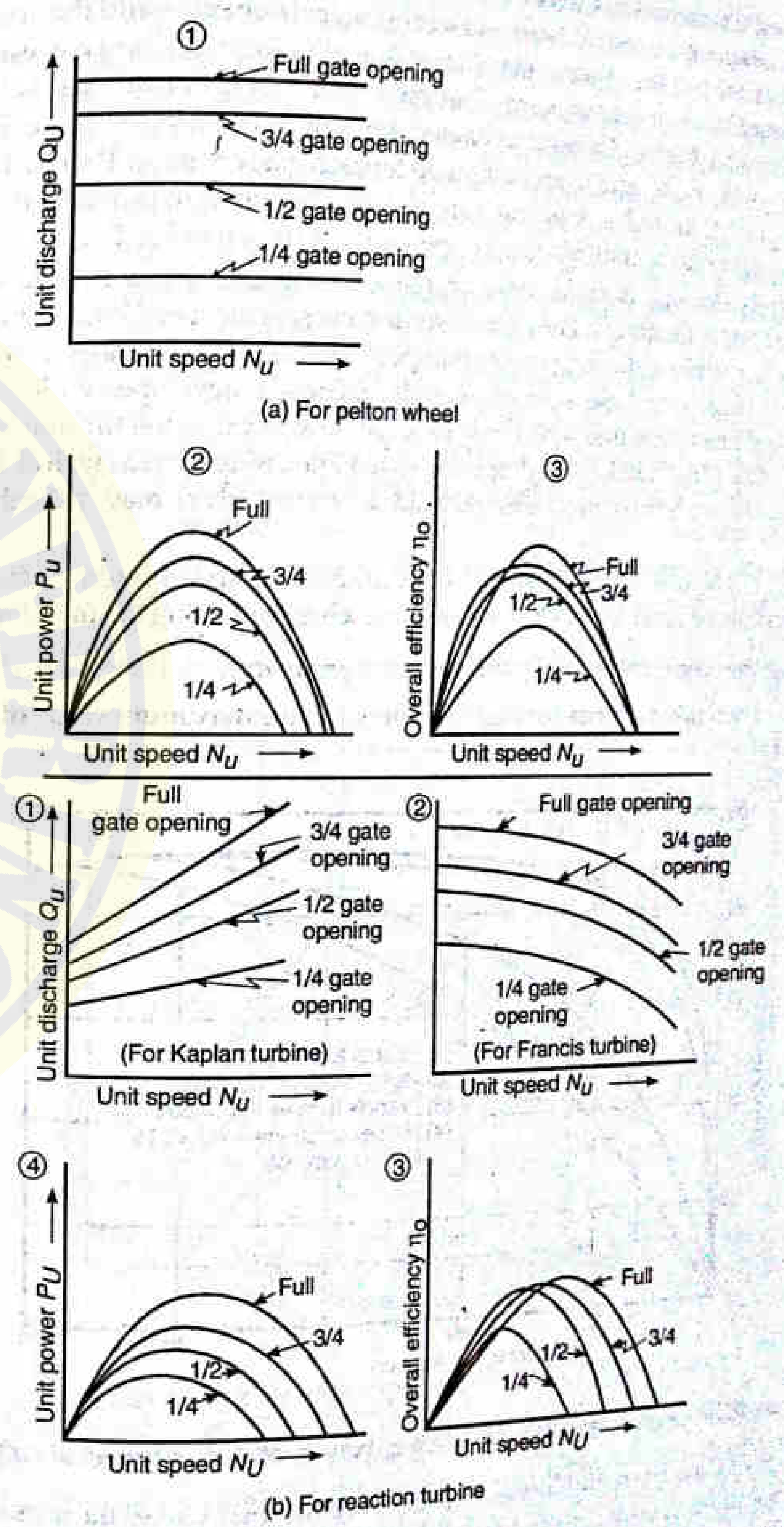


Figure 22.4 *Constant head characteristics for Pelton wheel and reaction turbines*

1. Constant Head Characteristic Curves. In order to obtain these curves the tests are performed on the turbine by maintaining a constant head and a constant gate opening and the speed is varied by changing the load on the turbine. A series of values of N are thus obtained and corresponding to each value of N, discharge Q and the output power P are measured. A series of such tests are performed by varying the gate opening, the head being maintained constant at the previous value. From the data of the tests the values of Q_u, P_u, N_u and η_o are computed for each gate opening. Then with N_u as abscissa the values of Q_u, P_u and η_o for each gate opening are plotted. The curves thus obtained for Pelton wheel and the reaction turbines for four different gate openings are shown in Fig. 22.4.

For Pelton wheels since Q_u depends only on the gate opening and is independent of N_u, the Q_u *v/s* N_u plots are horizontal straight lines. However, for low specific speed Francis turbines Q_u *v/s* N_u are drooping curves, thereby indicating that as the speed increases the discharge through the turbine decreases. This is so because in these turbines a centrifugal head is developed which retards the flow. Since the centrifugal head increases with the speed, the flow through the turbine is reduced as the speed increases. On the other hand, for high specific speed Francis turbines as well as Kaplan turbines, since the flow is axial there is no such centrifugal head developed which may cause the retardation of the flow.

The curves of P_u *v/s* N_u and η_o *v/s* N_u are parabolic in shape for the different turbines as shown in Fig. 22.4. It will be observed that for a Pelton wheel, for each gate opening the maximum value of η_o is attained at almost the same value of N_u which corresponds to $K_u = 0.46$ $[K_u = (N_u \pi D)/60\sqrt{2g}]$. However, in case of reaction turbines for each gate opening the maximum value of η_o is attained at different values of N.

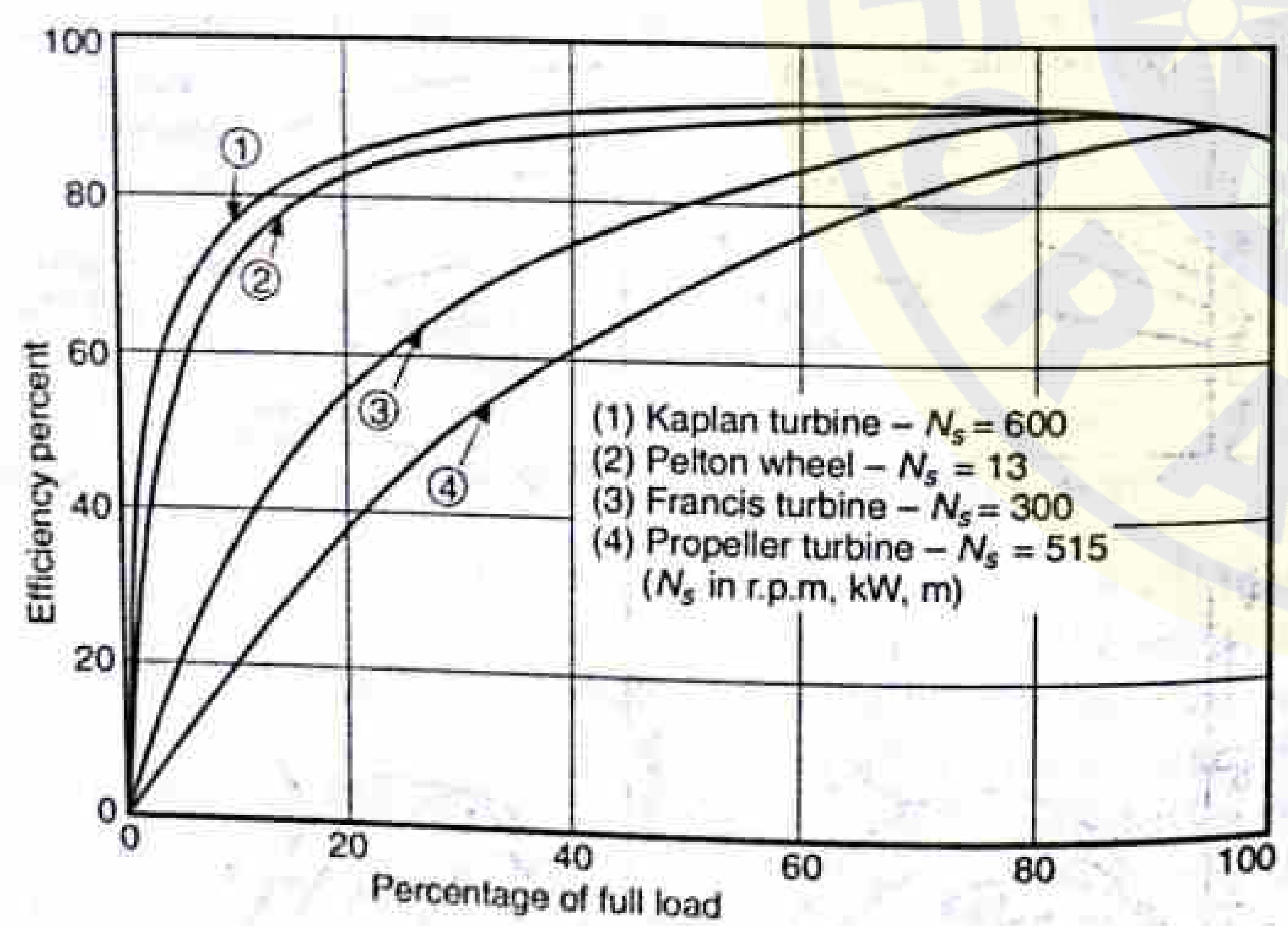


Figure 22.5 *Percentage of full load v/s η_o curves for different types of turbines*

2. Constant Speed Characteristic Curves. In order to obtain these curves the tests are performed on the turbines by operating them at constant speed. The constant speed is attained by regulating the gate opening thereby varying the discharge flowing through the turbine as the load varies. The head may or may not remain constant. The power developed corresponding to each setting of the gate opening is measured and the corresponding values of η_o are computed. Further knowing the total load capacity of the turbine the percentage of full load may be computed from the measured power, and a plot of η_o *v/s* percentage of full load is prepared. The curves thus obtained for the four different types of turbines working at constant speed under constant head at varying gate openings are shown in Fig. 22.5, from which it may be seen that as the % full load increases η_o also increases. In other words at reduced loads η_o is also less. At 100% full load η_o is near about the maximum efficiency in all cases. Further it may be observed that the Kaplan turbine and the Pelton wheel maintain a high efficiency over a longer range of the part load as compared with either the Francis or the fixed blade propeller turbine. Therefore this is an important factor in the selection of turbines.

In addition to the above graph, output power *v/s* discharge and η_o *v/s* discharge graphs are also plotted which are shown in Fig. 22.6. Both these graphs are plotted from a certain minimum discharge Q_o which is required to initiate the motion of the turbine runner from its state of rest. Further since output power is directly proportional to discharge if head is constant, output power *v/s* discharge graph is a straight line. On the other hand η_o *v/s* discharge graph is curvilinear and as shown in Fig. 22.6 the overall efficiency η_o increases with discharge and remains more or less constant beyond a particular value of discharge.

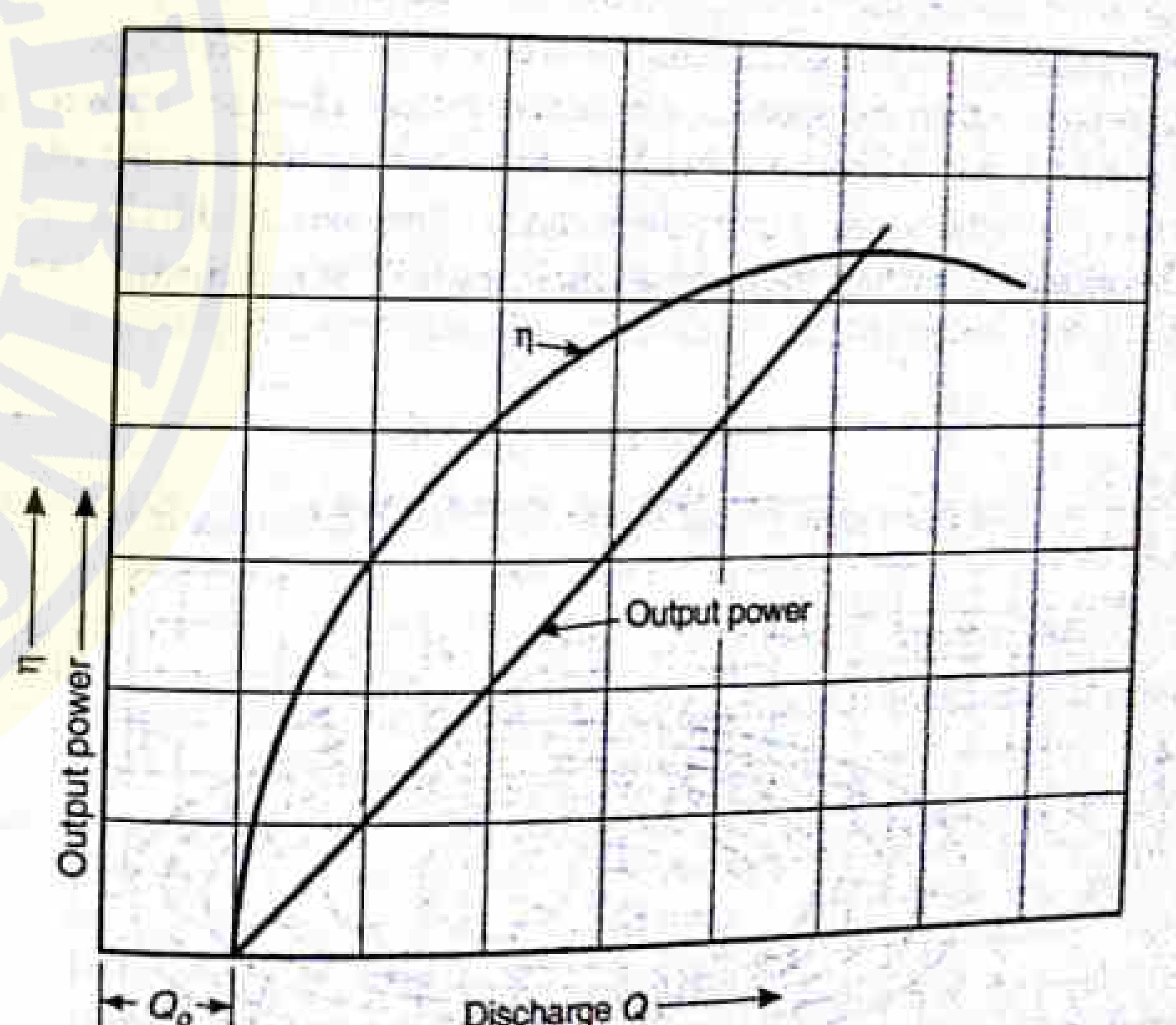


Figure 22.6 *η_o and output power v/s discharge curves*

3. Constant Efficiency (or Iso-efficiency) Curves. Figure 22.7 shows the constant efficiency or iso-efficiency curves. These curves show the efficiencies of the turbine for all conditions of running and hence these are also known as universal characteristic curves of the turbine. In order to draw these curves the following procedure is adopted. By operating the turbine at about 8 to 10 gate openings, the corresponding number of η_o *v/s* N_u and N_u *v/s* Q_u (or P_u) curves are plotted. On the η_o *v/s* N_u curves set of horizontal lines (each line representing the same efficiency) are drawn which will cut the curves corresponding to each gate opening at different points. These points are projected on the corresponding N_u v/s Q_u (or P_u) curves for each gate opening and the points of the same efficiency are joined by smooth curves which are the iso-efficiency curves. It is, however, clear from the graph that the innermost iso-efficiency curve represents the highest efficiency of the turbine and the outer curves represent the lower efficiencies. Further it will be seen from Fig. 22.7 that for a given unit discharge (or unit power) if a vertical line is drawn it will cut some iso-efficiency curve at two points and it will also touch some other inner iso-efficiency curve of higher efficiency, which may not have been drawn, just at one point only. Thus for a given unit discharge (or unit power) the vertical line touches the curve of maximum efficiency at only one point. It, therefore, means that if the peak points of the various iso-efficiency curves are joined together by a smooth curve then we obtain the best performance curve for the turbine, because the peak points of each iso-efficiency curve are the points of maximum efficiency for some value of the unit discharge (or unit power). With the help of these characteristic curves it is possible to predict the best performance of the turbine. Thus if the gate opening is known then the point of intersection of the N_u v/s Q_u (or P_u) curve for this gate opening and the best performance curve will represent the conditions for the best performance for that particular gate opening. From this point unit discharge (or unit power) can be known by drawing a vertical line and unit speed by drawing a horizontal line. Now if the head on the turbine is known the discharge Q, the power output P and the speed N for maximum efficiency at this gate opening can be computed. On the other hand if the head H and the speed N are known, then unit speed N_u can be computed. Then by drawing a horizontal line for this value of N_u, to cut the best performance line, the point of maximum efficiency is known corresponding to which unit discharge or unit power can be obtained, and as before the discharge Q and power P can be calculated at which the turbine efficiency will be maximum for the given head H and the speed N.

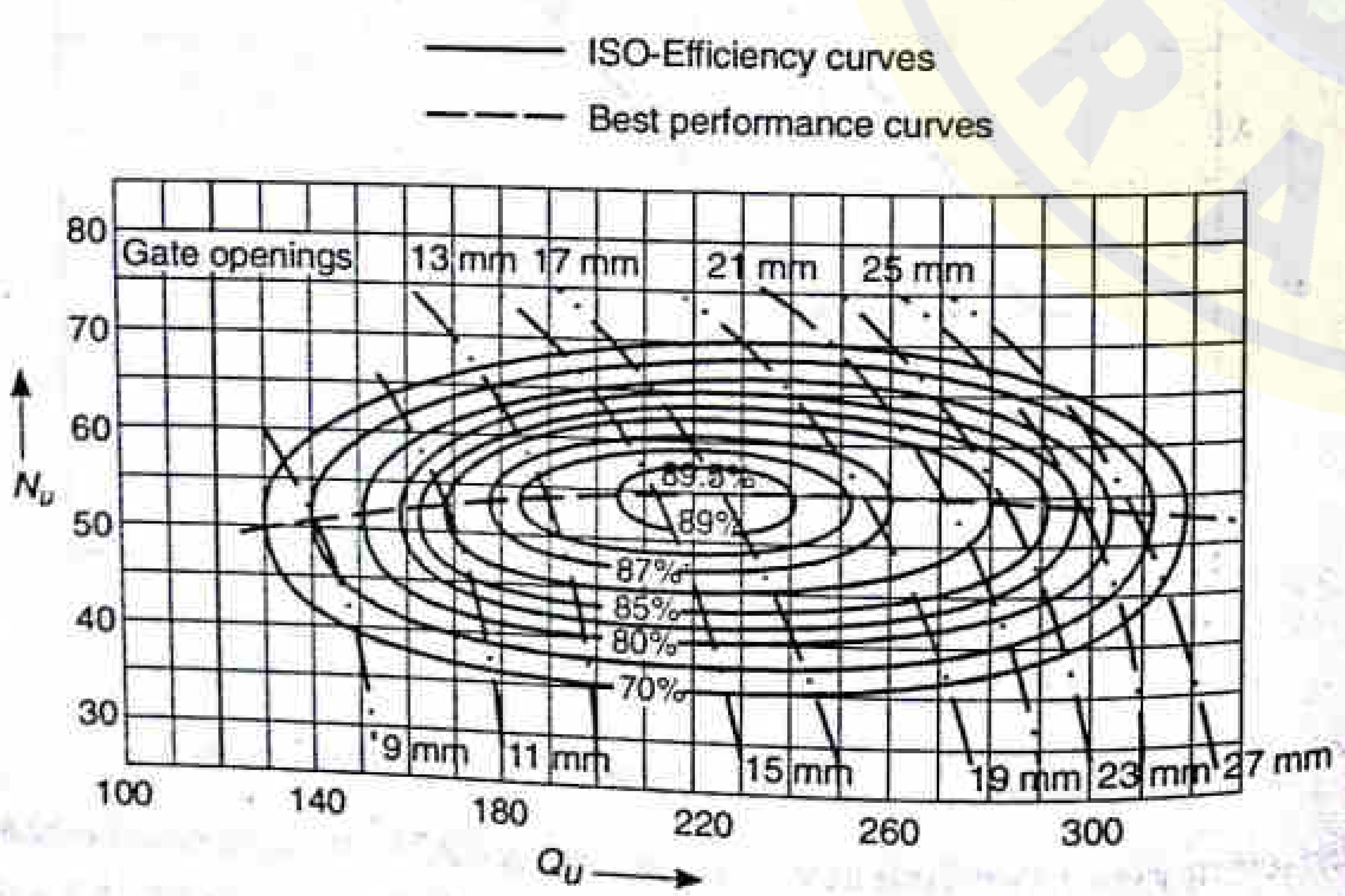


Figure 22.7 *Constant efficiency curve or Universal characteristics of a Francis turbine*

22.6 MODEL TESTING OF TURBINES

In order to have an idea about the performance of the actual turbine in advance, a small scale model of the turbine which is geometrically similar to the actual turbine is first prepared. The various linear dimensions of the model turbine bear the same proportion to their corresponding dimensions of the actual turbine. The model turbine is then tested under a known head, speed and flow rate and its output as well as the efficiency are determined. From these test results it will be possible to predict the performance of the actual turbine. Moreover on the basis of the test results if the design is to be modified it may be so done in the model turbine without incurring much expenditure. Thus model testing of turbines assists in obtaining a perfect design for the actual turbines, as well as in the development of the new types of turbines with higher specific speed and better efficiency.

The various variables involved in this case are discharge Q, head H, speed of rotation of runner N, runner diameter D, output power P, mass destiny ρ and viscosity μ of the flowing fluid. It may however be stated that in the problems of turbomachines generally shaft work gH is used as one of the variables instead of head H. By adopting the usual methods of dimensional analysis these variables may be grouped into following dimensionless parameters:

$$\left(\frac{Q}{ND^3}\right), \left(\frac{gH}{N^2D^2}\right), \left(\frac{P}{\rho gHND^3}\right), \left(\frac{\mu}{\rho ND^2}\right)$$

The parameter $\left(\frac{Q}{ND^3}\right)$ is known as *discharge number* or *flow number*, $\left(\frac{gH}{N^2D^2}\right)$ is known as *head number*, and $\left(\frac{P}{\rho gHND^3}\right)$ is known as *power number*. By combining these parameters, their alternative expressions may be obtained. Thus the discharge number may also be expressed as $\left(\frac{Q}{D^2\sqrt{gH}}\right)$ and the power number as $\left(\frac{P}{\rho g^{3/2}H^{3/2}D^2}\right)$. Further by dividing the square root of the power number by the head number raised to power (3/4) the following dimensionless parameter n_s is obtained

$$n_s = \frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}$$

The parameter n_s is known as *dimensionless specific speed* or *shape number* of the turbine.

It may, however, be noted that in the above expressions the output power P has been considered in terms of (N.m/s) [or kg(f)m/s in metric units]. But if the output power P is considered in terms of kW then in the above expressions P will have to be replaced by $(P \times 10^3)$. Further for water since $\rho = 1000$ kg/m³, the dimensionless specific speed n_s becomes

$$n_s = \frac{N\sqrt{P \times 10^3}}{(1000)^{1/2}(9.81)^{5/4}H^{5/4}} = \frac{N_s}{17.362}$$

or $$N_s = 17.362\, n_s$$

In metric units if the output power is considered in terms of horse power (metric) then in the above expressions P will have to be replaced by $(P \times 75)$. Further for water since $\rho = 102$ msl/m³, the dimensionless specific speed n_s becomes

$$n_s = \frac{N\sqrt{P \times 75}}{(102)^{1/2}(9.81)^{5/4} H^{5/4}} = \frac{N_s}{20.247}$$

or $$N_s = 20.247 n_s$$

These expressions give relationship between the specific speed (Eq. 22.11) and the dimensionless specific speed.

For complete similarity to exist between the model and the actual (or prototype) turbines the above noted parameters must have the same values for the model and the prototype turbines. The parameter $\left(\frac{\mu}{\rho N D^2}\right)$ represents Reynolds number which should also be equal in the model and the prototype turbines for complete similitude. But in actual practice, it is not possible to have the same value of the Reynolds number in the model and the prototype turbines, on account of considerable difference in their sizes. However, since the flow in the prototype turbine is turbulent, it has been observed that if the flow in the model turbine is also turbulent, then even if the Reynolds numbers are not equal for the model and the prototype turbines the similarity between them can be ensured.

Thus for the complete similarity to exist between the model and the prototype turbines, the following conditions may be required to be satisfied:

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$$

or
$$\left(\frac{Q}{D^2\sqrt{gH}}\right)_m = \left(\frac{Q}{D^2\sqrt{gH}}\right)_p \qquad \text{...(22.16)}$$

$$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p \qquad \text{...(22.17)}$$

$$\left(\frac{P}{\rho g H N D^3}\right)_m = \left(\frac{P}{\rho g H N D^3}\right)_p$$

or
$$\left(\frac{P}{\rho g^{3/2} H^{3/2} D^2}\right)_m = \left(\frac{P}{\rho g^{3/2} H^{3/2} D^2}\right)_p \qquad \text{...(22.18)}$$

$$\left(\frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}\right)_p \qquad \text{...(22.19)}$$

Since in most of the cases $g_m = g_p$ and also $\rho_m = \rho_p$ the above conditions may be simplified as follows:

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$$

or
$$\left(\frac{Q}{D^2\sqrt{H}}\right)_m = \left(\frac{Q}{D^2\sqrt{H}}\right)_p \qquad \text{...(22.16 a)}$$

$$\left(\frac{H}{N^2D^2}\right)_m = \left(\frac{H}{N^2D^2}\right)_p \qquad \text{...(22.17 a)}$$

$$\left(\frac{P}{HND^3}\right)_m = \left(\frac{P}{HND^3}\right)_p$$

or
$$\left(\frac{P}{H^{3/2}D^2}\right)_m = \left(\frac{P}{H^{3/2}D^2}\right)_p \qquad \text{...(22.18 a)}$$

$$\left(\frac{N\sqrt{P}}{H^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{H^{5/4}}\right)_p \qquad \text{...(22.19 a)}$$

where the subscripts m and p refer to model and prototype turbines respectively. By determining the values of P, H, N and D from the model test the values of the above noted parameters are determined, which will be same for both the model and the prototype turbines. For the prototype turbine since its diameter D and head H are known, its speed N, power P and discharge Q can thus be calculated with the help of the known values of these parameters.

A little consideration will show that the above noted conditions may be achieved if, K_u, ψ and N_s have the same values for the model and the prototype turbines. Further it may be seen that the various terms in the above expressions represent the unit quantities and hence for similarity between the model and the prototype turbines the values of the various unit quantities must be equal for the both.

The above noted conditions for establishing the similarity between the model and the prototype turbines are based on the assumption that the efficiency of the model is equal to that of the prototype. However, the efficiencies of the model and the prototype turbines are not equal. This is so because the energy losses are proportionately more in the model turbine than that in the prototype turbine. As such the efficiency of the model turbine is lower than that of the prototype turbine. On account of difference in the efficiencies of the model and the prototype turbines the scale effect would be developed and it would therefore be possible source of error in predicting the performance of the prototype turbine on the basis of the model test results. However, by modifying the above noted conditions for the similarity the error due to such a scale effect may be eliminated. The modified expressions for the similarity conditions may be obtained as indicated below.

If η_{om} and η_{op} are the overall efficiencies of the model and the prototype turbines respectively, then

$$\eta_{om} = \left(\frac{H - h_L}{H}\right)_m \text{; and } \eta_{op} = \left(\frac{H - h_L}{H}\right)_p$$

where H and h_L are respectively the head acting on the turbine and the loss of head in the turbine and the subscript m and p refer to model and prototype respectively. From the above expressions the net effective heads available for the model and the prototype turbines may be obtained as

$$(H-h_L)_m = \eta_{om}(H)_m$$

and

$$(H-h_L)_m = \eta_{op}(H)_p$$

By considering these values of the head in place of H in Eq. 22.16 (a) to 22.19 (a) the following expressions are obtained:

$$\left(\frac{Q}{D^2\sqrt{\eta_o H}}\right)_m = \left(\frac{Q}{D^2\sqrt{\eta_o H}}\right)_p \quad \text{...(22.16 b)}$$

$$\left(\frac{\eta_o H}{N^2 D^2}\right)_m = \left(\frac{\eta_o H}{N^2 D^2}\right)_p \quad \text{...(22.17 b)}$$

$$\left(\frac{P}{\eta_o^{3/2} H^{3/2} D^2}\right)_m = \left(\frac{P}{\eta_o^{3/2} H^{3/2} D^2}\right)_p \quad \text{...(22.18 b)}$$

$$\left(\frac{N\sqrt{P}}{\eta_o^{5/4} H^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{\eta_o^{5/4} H^{5/4}}\right)_p \quad \text{...(22.19 b)}$$

These equations thus represent the conditions for similarity between the model and the prototype turbines with due allowance for the difference in their efficiencies.

From Eq. 22.19 (b) the ratio of the specific speeds for the model and prototype turbines may be expressed as

$$\frac{(N_s)_m}{(N_s)_p} = \left(\frac{\eta_{om}}{\eta_{op}}\right)^{5/4}$$

For determining the efficiency of a prototype turbine from the efficiency obtained for its model, the following general expression has been given:

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{\alpha}\left(\frac{H_m}{H_p}\right)^{\beta} \quad \text{...(22.20)}$$

where η_{om} and η_{op} are the overall efficiencies of the model and the prototype respectively, D_m and D_p are the respective diameters of their runners, H_m and H_p are the heads acting on the model and the prototype turbines respectively and α and β are the numerical exponents. Different values have been suggested for the exponents α and β by different investigators, but the most commonly adopted values are those recommended by L.F. Moody which are $\alpha = 0.20$ and $\beta = 0$. Thus Eq. 22.20 becomes

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.20} \quad \text{...(22.20 a)}$$

Moody's Eq. 22.20 (a) is however applicable for head $H < 150$ m. For head $H > 150$ m the following relationship has been recommended:

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.25}\left(\frac{H_m}{H_p}\right)^{0.10} \quad \text{...(22.20 b)}$$

It may however be stated that different investigators have recommended different values for the exponent of (H_m/H_p) varying from 0.04 to 0.10 in Eq. 22.20 (b).

22.7 CAVITATION IN TURBINES

When the pressure in any part of the flow passage reaches the vapour pressure of the flowing liquid, it starts vaporizing and small bubbles of vapour form in large numbers. These bubbles (or vapour-filled pockets or cavities) are carried along by the flow, and on reaching the high pressure zones these bubbles suddenly collapse as the vapour condenses to liquid again. Due to sudden collapsing of the bubbles or cavities the surrounding liquid rushes in to fill them. The liquid moving from all directions collides at the centre of the cavity, thus giving rise to very high local pressure, which may be as high as 686.7 MN/m^2 [7000 kg(f)/cm^2]. Any solid surface in the vicinity is also subjected to these intense pressures. The alternate formation and collapse of vapour bubbles may cause severe damage to the surface which ultimately fails by fatigue and the surface becomes badly scored and pitted. This phenomenon is known as *cavitation* which is found to occur in turbines as well as in various hydraulic structures such as penstocks, gates, valves, spillways etc.

In reaction turbines the cavitation may occur at the runner exit or the inlet to the draft tube where the pressure is considerably reduced. Due to cavitation the metal of the runner vanes and the draft tube is gradually eaten away in these zones, which results in lowering the efficiency of the turbine. As such the turbine components should be so designed that as far as possible cavitation is eliminated. In order to determine whether cavitation will occur in any portion of the turbine, D. Thoma of Germany has developed a dimensionless parameter called *Thoma's cavitation factor* σ which is expressed as

$$\sigma = \frac{H_a - H_v - H_s}{H} \quad \text{...(22.21)}$$

where H_{a0} is atmospheric pressure head; H_v is vapour pressure head; H_s is suction pressure head (or height of runner outlet above tail race); and H is working head of turbine. Complete similarity in respect of cavitation can be ensured if the value of σ is same in both the model and the prototype. Moreover it has been found that σ depends on N_s of the turbine and for a turbine of particular N_s the factor σ can be reduced upto a certain value upto which its efficiency η_0 remains constant. A further decrease in the value of σ results in a sharp fall in η_0. The value of σ at this turning point is called *critical cavitation factor* σ_c. The value for σ_c for different turbines may be determined with the help of the following empirical relationships:

For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{380.78}\right)^2 \quad \text{...(22.22)}$$

For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{380.78}\right)^3\right] \quad ...(22.23)$$

For Kaplan turbines, values of σ_c obtained by Eq. 22.23 should be increased by 10 per cent.

In the above expressions N_s is in (r.p.m, kW, m) units. However, if N_s is in (r.p.m, h.p, m) units, the expressions for σ_c would be as follows:

For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{444}\right)^2 \quad ...(22.22a)$$

For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{444}\right)^3\right] \quad ...(22.23a)$$

Again for Kaplan turbines, values of σ_c obtained by Eq. 22.23 (a) should be increased by 10 per cent.

(*a*) *Suction Specific Speed*. In addition to Thoma's criterion the consideration of suction specific speed provides another very useful criterion for establishing similarity in respect of cavitation in the turbines. The *suction specific speed* S may be defined as the speed of a geometrically similar turbine such that when it is developing a power equal to 1 kilowatt (or in metric units equal to 1 metric horse power) the total suction head H_{sv} is equal to 1m (in absolute units). According to this definition the expression for suction specific speed may be obtained by replacing the total head H in Eq. 22.11 for the specific speed by the total suction head H_{sv}. Thus

$$S = \frac{N\sqrt{P}}{H_{sv}^{5/4}} \quad ...(22.24)$$

By having the same value of the suction specific speed for the model and the prototype turbines the similarity in respect of cavitation can be established.

The total suction head H_{sv} can be expressed as

$$H_{sv} = H_a - H_v - H_s$$

and hence from Eq. 22.21, we have

$$H_{sv} = \sigma H \quad ...(22.25)$$

By substituting the value of H_{sv} in Eq. 22.24, we get

$$S = \frac{N\sqrt{P}}{(\sigma H)^{5/4}}$$

or

$$\sigma = \left(\frac{N_s}{S}\right)^{4/5} \quad ...(22.26)$$

Equation 22.26 represents the relation between the two parameters σ and S, both of which are useful for establishing a similarity in respect of cavitation in the model and prototype turbines. However, the concept of suction specific speed is more commonly used in the case of pumps.

22.8 SELECTION OF TURBINES

The selection of a suitable type of turbine is usually governed by the following factors:

(i) Head and Specific Speed. It has been found that there is a range of head and specific speed for which each type of turbine is most suitable which is given in Table 22.1.

TABLE 22.1

S. No.	Head in metres	Type of turbine	Specific speed
1.	300 or more	Pelton wheel Single or Multiple jet	8.5 to 47 (in SI units) 10 to 55 (in metric units)
2.	150 to 300	Pelton or Francis	30 to 85 (in SI units) 35 to 100 (in metric units)
3.	60 to 150	Francis or Deriaz (or Diagonal)	85 to 188 (in SI units) 100 to 220 (in metric units)
4.	Less than 60	Kaplan or Propeller or Deriaz or Tubular	188 to 860 (in SI units) 220 to 1000 (in metric units)

However, as a general rule, it may be stated that as far as possible a turbine with highest permissible specific speed should be chosen, which will not only be the cheapest in itself but its relatively small size and high rotational speed will reduce the size of the generator as well as the power house. But the specific speed cannot be increased indefinitely, because higher specific speed turbine is generally more liable to cavitation. The cavitation may, however, be avoided by installing the turbine at a lower level with respect to the tail race.

(ii) Part Load Operation. The turbines may be required to work with considerable load variations. As the load deviates from the normal working load, the efficiency would also vary. In Fig. 22.5 a plot between η_0 and % of full load has been shown. At part load the performance of Kaplan and Pelton turbines is better in comparison to that of Francis and Propeller turbines. The variability of load will influence the choice of type of turbine if the head lies between 150 m to 300 m or lies below 30 m. For higher range of heads Pelton wheel is preferable for part load operation in comparison to Francis turbine, though the former involves higher initial cost. For heads below 30 m, Kaplan turbine is preferable for part load operation in comparison to Propeller turbine.

In addition to above mentioned factors there are certain other factors to be considered for the selection of the suitable type of a turbine. The *overall cost* which includes the initial cost and the running cost should be considered. The *cavitation* characteristics of the turbine should also be considered since it affects the installation of a reaction turbine.

ILLUSTRATIVE EXAMPLES

Example 22.1. *A Francis turbine working under a head of 5 m at a speed of 210 r.p.m., develops 75 kW when the rate of flow of water is 1.8 m³/s. The runner diameter is 1 m. If the head on this turbine is increased to 16 m, determine its new speed, discharge and power.*

Solution

$$N_1 = \left[\frac{N\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \frac{210\times\sqrt{16}}{\sqrt{5}} = 375.7 \text{ r.p.m.}$$

$$Q_1 = \left[\frac{Q\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \frac{1.8\times\sqrt{16}}{\sqrt{5}} = 3.22 \text{ m}^3/\text{s}$$

$$P_1 = \left[\frac{PH_1^{3/2}}{H^{3/2}}\right]$$

$$= \left[\frac{75\times(16)^{3/2}}{(5)^{3/2}}\right] = 429.33 \text{ kW}$$

Example 22.2. *A hydraulic turbine has an output of 6 600 kW when it works under a head of 25 m and runs at 100 r.p.m. What is the type of turbine? What would be its speed and what power will it develop when working under a head of 16 m?*

Solution

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{100\sqrt{6600}}{(25)^{5/4}} = 145.33$$

In this case since $85 < N_s < 188$, it can be a Francis turbine.

$$N_1 = \left[\frac{N\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \left[\frac{100\sqrt{16}}{\sqrt{25}}\right] = 80 \text{ r.p.m.}$$

$$P_1 = \left[\frac{P(H_1)^{3/2}}{(H)^{3/2}}\right]$$

$$= \left[\frac{6600(16)^{3/2}}{(25)^{3/2}}\right] = 3379.2 \text{ kW}$$

Example 22.3. *A turbine develops 7 460 kW under a head of 24.7 m at 135 r.p.m. What is the specific speed? What would be its normal speed and output under a head of 19.5 m?*

Solution

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{135\times\sqrt{7460}}{(24.7)^{5/4}} = 211.75$$

If N_1 is the speed under a head of 19.5 m, then

$$N_1 = N\left(\frac{H_1}{H}\right)^{1/2}$$

$$= 135\left(\frac{19.5}{24.7}\right)^{1/2} = 119.95 \text{ r.p.m.}$$

Similarly, if P_1 is the output under a head of 19.5 m, then

$$P_1 = P\left(\frac{H_1}{H}\right)^{3/2}$$

$$= 7460\left(\frac{19.5}{24.7}\right)^{3/2} = 5\,232.94 \text{ kW}$$

Examples 22.4. *A Pelton wheel develops 5520 kW under a head of 240 m at an overall efficiency of 80% when revolving at a speed of 200 r.p.m. Find the unit discharge, unit power and unit speed. Assume peripheral coefficient = 0.46. If the head on the same turbine falls during the summer season to 150 m, find the discharge, power and speed for this head.*

Solution

$$\eta_o = \frac{P}{wQH}$$

or

$$0.80 = \frac{5520\times10^3}{9810\times Q\times240}$$

$$\therefore \quad Q = 2.931 \text{ m}^3/\text{s}$$

Unit discharge $\quad Q_u = \frac{Q}{\sqrt{H}} = \frac{2.931}{\sqrt{240}} = 0.189$

Unit power $\quad P_u = \frac{P}{H^{3/2}} = \frac{5520}{(240)^{3/2}} = 1.485$

Unit speed $\quad N_u = \frac{N}{\sqrt{H}} = \frac{200}{\sqrt{240}} = 12.91$

Since peripheral coefficient is assumed to be constant equal to 0.46, the efficiency may be assumed to be constant under different heads and hence the unit quantities will be same under different heads. Thus

$Q_u = 0.189 = \frac{Q_1}{\sqrt{150}}$; $\quad \therefore\ Q_1 = 2.315 \text{ m}^3/\text{s}$

$P_u = 1.489 = \frac{P_1}{(150)^{3/2}}$; $\quad \therefore\ P_1 = 2\,735.47 \text{ kW}$

$N_u = 12.91 = \frac{N_1}{\sqrt{150}}$; $\quad \therefore\ N_1 = 158.1 \text{ r.p.m.}$

Example 22.5 *A model of Francis turbine one-fifth of full size, develops 3 kW at 306 r.p.m. under a head of 1.77 m. Find the speed and power of full size turbine operating under a head of 5.7 m, if (a) the efficiency of the model and the full size turbine are same, (b) the efficiency of the model turbine is 76% and the scale effect is considered.*

Solution

(a) For the same efficiency of the model and the prototype, from Eq. 22.17 (a)

$$\frac{D_m N_m}{\sqrt{H_m}} = \frac{D_p N_p}{\sqrt{H_p}}$$

or
$$\frac{306}{5\times\sqrt{1.77}} = \frac{N_p}{\sqrt{5.7}}; \qquad \therefore N_p = 109.8 \text{ r.p.m.}$$

Further from Eq. 22.18 (a)

$$\frac{P_m}{D_{m^2} H_m^{3/2}} = \frac{P_p}{D_p^2 H_p^{3/2}}$$

or
$$\frac{3}{(1.77)^{3/2}} = \frac{P_p}{(5)^2\times(5.7)^{3/2}}; \qquad \therefore P_p = 433.43 \text{ kW}$$

(b) According to Moody's Eq. 22.20 (a)

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.2}$$

or
$$\frac{1-\eta_{op}}{1-0.76} = \left(\frac{1}{5}\right)^{0.2}; \qquad \therefore \eta_{op} = 0.826$$

From Eq. 22.17 (b), we have

$$\frac{D_m N_m}{\sqrt{\eta_{om} H_m}} = \frac{D_p N_p}{\sqrt{\eta_{op} H_p}}$$

or
$$\frac{306}{5\times\sqrt{0.76\times1.77}} = \frac{N_p}{\sqrt{0.826\times5.7}}$$

$$\therefore \qquad N_p = 114.5 \text{ r.p.m.}$$

From Eq. 22.18 (b), we have

$$\frac{P_m}{\eta_{om}^{3/2} H_m^{3/2} D_m^2} = \frac{P_p}{\eta_{op}^{3/2} H_p^{3/2} D_p^2}$$

or
$$\frac{3}{(0.76)^{3/2}(1.77)^{3/2}} = \frac{P_p}{(0.826)^{3/2}(5.7)^{3/2}(5)^2}$$

$$\therefore \qquad P_p = 491.09 \text{ kW}$$

Example 22.6. *Estimate the maximum height of straight conical draft tube of 13240 kW Francis turbine running at 150 r.p.m., under a net head of 27 m. The turbine is installed at a station where the effective atmospheric pressure is 10.6 m of water. The draft tube must sink at least 0.77 m below the tail race.*

Solution

Specific speed of the turbine

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = \frac{150(13240)^{1/2}}{(27)^{5/4}} = 280.43$$

From Eq. 22.22, we have

$$\sigma_c = 0.625\left(\frac{N_s}{380.78}\right)^2 = 0.625\left(\frac{280.43}{380.78}\right)^2 = 0.339$$

But
$$\sigma_c = \frac{H_a - H_v - H_s}{H}$$

$$(H_a - H_v) = 10.6 \text{ m}; H = 27 \text{ m}$$

Thus by substitution, we get

$$0.339 = \frac{10.6 - H_s}{27}; \therefore H_s = 1.45 \text{ m}$$

Hence the maximum length of the draft tube

$$= (1.45 + 0.77) = 2.22 \text{ m}$$

Example 22.7 *(a) Define 'unit torque' and derive expression for it.*

(b) Prove that specific speed can be expressed as

$$N_s = 3.13\, N_u \sqrt{Q_u \eta_o}$$

where N_u and Q_u are the unit speed and unit discharge and η_0 is the overall efficiency.

Solution

(a) *Unit torque* may be defined as the torque per unit head acting on the turbine. Thus if T is torque and H is the head acting then unit torque $T_u = \frac{T}{H}$

i.e.,
$$\text{Torque} = \text{Force} \times \text{distance}$$
$$\text{Torque } T \propto (\text{Force } F)$$

and
$$F = \frac{wQ}{g}(V_w - V_{w_1})$$

i.e.,
$$F \propto QV_w$$

but
$$Q \propto \sqrt{H} \qquad \text{and } V_w \propto \sqrt{H}$$

Thus
$$F \propto H$$

$\therefore$
$$F \propto H; \text{or } T = \text{constant}\,(H)$$

'Unit torque' T_u is the torque developed due to head equal to 1 m (or unit head)

Thus
$$T_u = \text{Constant} \times 1$$

$$\therefore \quad T_u = \frac{T}{H}$$

(b)
$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$P = wQH \times \eta_o$$
$$= 9\,810\,QH\eta_o\, W = 9.81\,QH\,\eta_o \text{ kW}$$

Thus by substitution, we get

$$N_s = \frac{N}{H^{5/4}} \times (9.81\,QH\eta_o)^{1/2}$$

or
$$N_s = 3.13 \times \frac{N}{\sqrt{H}} \times \sqrt{\frac{Q}{\sqrt{H}} \times \eta_o} = 3.13\,N_u\,\sqrt{Q_u\eta_o}$$

Example 22.8. *In a Kaplan turbine the discharge is 25 m³/s under a net head of 10 m at the design speed of 250 r.p.m. The runner and hub diameters are 2 m and 1 m respectively. The vapour and barometric pressures are 1.6 m and 10 m respectively. The pressure recorded at inlet to the draft tube is 3 m vacuum, where the diameter of the draft tube is 2 m. The exit diameter of the draft tube is 3 m and the efficiency of the draft tube is 0.80. The Thoma's cavitation parameter for the turbine is given by the relation $\sigma = \eta_d\,\psi^2 + \lambda K_u^2$, in which η_d is the efficiency of the draft tube, ψ is flow ratio, K_u is speed ratio and λ is a dimensionless factor defined by $\lambda = \frac{(p_1/w)-(p_{min}/w)}{(u^2/2g)}$ in which u is the tangential velocity, (p_1/w) is the pressure head at inlet to draft tube, and (p_{min}/w) is the minimum pressure head at a point on the blade. Calculate the minimum pressure on the blade. Also calculate N, if $\eta_o = 0.90$.*

Solution

$$Q = \frac{\pi}{4}(D^2-d^2)\,\psi\sqrt{2gH}$$

or
$$25 = \frac{\pi}{4}(2^2-1^2)\,\psi\,\sqrt{2\times9.81\times10}$$

$$\therefore \quad \psi = 0.757$$

$$u = \frac{\pi DN}{60} = K_u\sqrt{2gH}$$

or
$$\frac{\pi\times2\times250}{60} = K_u = \sqrt{2\times9.81\times10}$$

$$\therefore \quad K_u = 1.87$$

Also
$$u = \frac{\pi DN}{60} = \frac{\pi\times2\times250}{60} = 26.18 \text{ m/s}$$

Applying Bernoulli's equation between the inlet of the draft tube and the water surface of the tail race channel, we get

$$\frac{p_1}{w} + H_s + \frac{V_1^2}{2g} = \frac{p_2}{w} + \frac{V_2^2}{2g} + h_f$$

where h_f is the loss of head in the draft tube which is given as

$$h_f = \frac{V_1^2 - V_2^2}{2g} - \eta_d\,\frac{V_1^2}{2g}$$

and
$$\frac{p_2}{w} = \frac{p_{atm}}{w}$$

$$\therefore \quad H_s = \frac{p_{atm}}{w} - \frac{p_1}{w} - \eta_d\,\frac{V_2^2}{2g}$$

$$V_1 = \frac{25}{\pi(2)^{2/4}} = 7.96 \text{ m/s}$$

$$\frac{p_1}{w} = (10-3) = 7 \text{ m absolute}$$

By substitution, we obtain

$$H_s = 10 - 7 - 0.80 \times \frac{(7.96)^2}{2\times9.81}$$
$$= 0.42 \text{ m}$$

Thoma's cavitation parameter is also given by

$$\sigma = \frac{H_a - H_v - H_s}{H}$$

Thus
$$\sigma = \frac{10-1.6-0.42}{10} = 0.798$$

As given
$$\sigma = \eta_d\,\psi^2 + \lambda K_u^2$$

Thus by substitution, we have

$$0.798 = 0.80(0.757)^2 + \lambda\,(1.87)^2$$

$$\therefore \quad \lambda = 0.097$$

but
$$\lambda = \frac{(p_1/w)-(p_{min}/w)}{(u^2/2g)}$$

or
$$(0.097) \times \frac{(26.18)^2}{2\times9.81} = -3 - \frac{p_{atm}}{w}$$

or
$$\frac{p_{min}}{w} = 6.389 \text{ m vacuum}$$

or
$$p_{min} = 62.68 \text{ kN/m}^2$$

Further from the Example 22.7, we have

$$N_s = 3.13\, N_u \sqrt{Q_u \eta_o}$$

or

$$N_s = 3.13 \times \frac{250}{\sqrt{10}} \times \sqrt{\frac{25}{\sqrt{10}} \times 0.90} = 660$$

Example 22.9. *Calculate the value of Thoma's cavitation parameter for a Kaplan turbine having non-dimensional specific speed of 19.36 and suction specific speed of 17.6.*

Solution

The non-dimensional specific speed n_s is given by

$$n_s = \frac{N_s}{17.362}$$

$$\therefore \quad N_s = (17.362 \times n_s) = (17.362 \times 19.36)$$

Similarly the non-dimensional suction specific speed is given by

$$\sigma = \frac{S}{17.362}$$

$$\therefore \quad S = (17.362 \times s) = (17.362 \times 17.6)$$

From Eq. 22.26, we have

$$\sigma = \left(\frac{N_s}{S}\right)^{4/5}$$

$$\therefore \quad \sigma = \left(\frac{17.362 \times 19.36}{17.362 \times 17.6}\right)^{4/5} = 1.08$$

Example 22.10. *A model of a turbine 0.3 m diameter when running at 900 r.p.m., under a head of 12 m gave the following results:*

Percentage gate opening	40	60	80	100
Output power kW	8.38	13.24	17.50	19.71.

The actual turbine is required to develop 7360 kW at 85% gate opening under a head of 35 m. Assuming same efficiency at corresponding gate opening for the model and actual turbines, find (i) diameter and speed of the actual turbine, (ii) output of the actual turbine at 50% gate opening.

Solution

From the plot of percentage gate opening *v/s* output power for the given data, for 85% gate opening the output power of the model is obtained as

$$P_m = 18.20 \text{ kW}$$

Thus from Eq. 20.18 (a), we have

$$\left(\frac{P}{H^{3/2}D^2}\right)_m = \left(\frac{P}{H^{3/2}D^2}\right)_p$$

or

$$\left[\frac{18.20}{(12)^{3/2}(0.3)^2}\right]_m = \left[\frac{7360}{(35)^{3/2}D^2}\right]_p$$

$$\therefore \quad D_p = 2.70 \text{ m}$$

Further from Eq. 22.17 (a), we have

$$\left(\frac{H}{N^2D^2}\right)_m = \left(\frac{H}{N^2D^2}\right)_p$$

or

$$\left[\frac{12}{(900)^2(0.3)^2}\right]_m = \left[\frac{35}{N^2(2.7)^2}\right]_p$$

$$\therefore \quad N_p = 171 \text{ r.p.m.}$$

From the same plot for 50% gate opening the output power of the model is

$$P_m = 11.03 \text{ kW}$$

Thus from Eq. 20.18 (a), we have

$$\left(\frac{P}{H^{3/2}D^2}\right)_m = \left(\frac{P}{H^{3/2}D^2}\right)_p$$

or

$$\left[\frac{11.03}{(12)^{3/2}(0.3)^2}\right]_m = \left[\frac{P}{(35)^{3/2}(2.7)^2}\right]_p$$

$$\therefore \quad P_p = 4\,450.3 \text{ kW}$$

Example 22.11. *For a large Pelton turbine the design values of K_v, K_u, k and m are respectively 0.97, 0.46, 0.98 and 10.2 (K_v is the ratio of jet velocity to the spouting velocity $\sqrt{2gH}$, K_u is the ratio of the circumferential velocity of the wheel to the spouting velocity, k is the ratio of the relative velocities at exit and inlet of the bucket and m is the ratio of the wheel diameter to the jet diameter).*

Determine the specific speed of the turbine if the bucket angle at exit is 10° and the mechanical efficiency of the turbine is 90.5%.

Solution

Specific speed of Pelton turbine is given by Eq. 22.12 as

$$N_s = \frac{(494.24)K_u(K_v\eta_o)^{1/2}}{m}$$

The overall efficiency η_o is given by

$$\eta_o = \eta_h \times \eta_m$$

where η_h is hydraulic efficiency and η_m is mechanical efficiency of Pelton turbine.

The hydraulic efficiency of Pelton turbine is given by

$$\eta_h = \frac{2u(V-u)(1+k\cos\phi)}{V^2}$$

in which by introducing $u = K_u\sqrt{2gH}$; and

$V = K_v\sqrt{2gH}$, we get

$$\eta_h = \frac{2K_u(K_v - K_u)(1 + k\cos\phi)}{K_v^2}$$

$$K_u = 0.46;\ K_v = 0.97;\ k = 0.98;\ \text{and}\ \ \phi = 10^\circ$$

Thus by substitution, we get

$$\eta_h = \frac{2\times0.46\times(0.97-0.46)\times(1+0.98\cos 10^\circ)}{(0.97)^2}$$

$$= \frac{2\times0.46\times(0.97-0.46)\times(1+0.98\times0.9848)}{(0.97)^2}$$

$$= 0.98 \text{ or } 98\%$$

$$\eta_m = 90.5\% \text{ or } 0.905 \text{ (given)}$$

$$\therefore \quad \eta_t = 0.98 \times 0.905 = 0.887$$

and $$m = 10.2$$

Thus by substitution, we get

$$N_s = \frac{494.24\times0.46\times(0.97)\times0.887)^{1/2}}{10.2}$$

$$= 20.67$$

Example 22.12. *A vertical shaft Kaplan turbine operating under a head of 9.8 m has a runner diameter of 9.3 m. At the optimum regime the turbine runs at 51.7 r.p.m and develops 45000 kW power discharging 535 m³/s water. Determine the values of unit speed, unit discharge, unit power and specific speed of the turbine.*

If the critical value of cavitation coefficient for the runner is 0.95, determine the location of the runner with respect to the tail race water level. The atmospheric and saturated vapour pressures may be taken as 0.98 kg(f)/cm² and 0.25 kg(f)/cm² respectively.

Solution

$H = 9.8$ m; $D = 9.3$ m; $N = 51.7$ r.p.m; $P = 45\,000$ kW; and $Q = 535\ \text{m}^3/\text{s}$

From Eq. 22.5(a), unit speed is given as

$$N_u = \frac{ND}{\sqrt{H}}$$

$$= \frac{51.7\times9.3}{\sqrt{9.8}} = 153.59$$

From Eq. 22.6 (a), unit discharge is given as

$$Q_u = \frac{Q}{D^2\sqrt{H}}$$

$$\frac{535}{(9.3)^2\sqrt{9.8}} = 1.98$$

From Eq. 22.7(a), unit power is given as

$$P_u = \frac{P}{D^2 H^{3/2}}$$

$$= \frac{45000}{(9.3)^2(9.8)^{3/2}} = 16.96$$

From Eq. 22.11, specific speed is given as

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$= \frac{51.7\sqrt{45000}}{(9.8)^{5/4}} = 632.51$$

From Eq. 22.21, we have

$$\sigma = \frac{H_a - H_v - H_s}{H}$$

$$\sigma = \sigma_c = 0.95; H_a = \frac{0.98\times10^4}{1000} = 9.8\text{ m; and}$$

$$H_v = \frac{0.25\times10^4}{1000} = 2.5\text{ m}$$

Thus by substitution, we get

$$0.95 = \frac{9.8 - 2.5 - H_s}{9.8}$$

$$\therefore \quad H_s = -2.01\text{ m}$$

The suction head H_s is negative which means that the turbine runner must be located 2.01 m below the tail race water level.

Example 22.13 *The model of a water turbine tested in the laboratory at a head of 20 m just began to cavitate when the static suction head was raised to 7 m. Its optimum hydraulic efficiency was 82%. The prototype turbine is required to work at a head of 50 m. Determine the maximum safe setting for the turbine above the tail race. Atmospheric conditions in the laboratory and at the site are given below:*

	Atmospheric pressure	*Vapour pressure corresponding To water temperature*
At site	1×10^5 N/m²	0.035×10^5 N/m²
In laboratory	1.012×10^5 N/m²	0.033×10^5 N/m²

The Reynolds number of model test was $Re = 6 \times 10^5$. The prototype Reynolds number is $Re = 1.5 \times 10^8$. Estimate the prototype efficiency explaining the reasoning behind any formula used.

Solution

The similarity between the model and the prototype in respect of cavitation characteristics can be ensured if Thoma's cavitation number is same in the model and the prototype. That is

$$(\sigma)_m = (\sigma)_p$$

or
$$\left(\frac{H_a - H_v - H_s}{H}\right)_m = \left(\frac{H_a - H_v - H_s}{H}\right)_p$$

$$(H_a)_m = \frac{1.012\times10^5}{9810} = 10.316 \text{ m}; \ (H_v)_m = \frac{0.033\times10^5}{9810} = 0.336 \text{ m};$$

$$(H_s)_m = 7 \text{ m; and } (H)_m = 20 \text{ m}$$

$$(H_a)_p = \frac{1\times10^5}{9810} = 10.194 \text{ m}; \ (H_v)_p = \frac{0.035\times10^5}{9810} = 0.357 \text{ m}$$

and
$$(H)_p = 50 \text{ m}$$

Thus by substitution, we get

$$\frac{(10.316-0.336-7)}{20} = \frac{(10.194-0.357-H_s)}{50}$$

or
$$\frac{2.98}{20} = \frac{9.837-H_s}{50}$$

$\therefore$
$$H_s = 2.387 \text{ m}$$

Thus the maximum safe setting for the turbine above the tail race is 2.387 m.

The Reynolds number may be expressed as

$$R_e = \frac{\rho N D^2}{\mu}$$

For the model turbine

$$(\text{Re})_m = \left(\frac{\rho N D^2}{\mu}\right)_m = 6\times10^5$$

or
$$(ND)_m = \left(\frac{6\times10^5\times\mu}{\rho D}\right)_m \quad \text{...(i)}$$

For the prototype turbine

$$(\text{Re})_p = \left(\frac{\rho N D^2}{\mu}\right)_p = 1.5\times10^8$$

or
$$(ND)_p = \left(\frac{1.5\times10^8\times\mu}{\rho D}\right)_p \quad \text{...(ii)}$$

For similarity between the model and the prototype turbines from Eq. 22.17(a), we have

$$\left(\frac{\sqrt{\eta_o H}}{ND}\right)_m = \left(\frac{\sqrt{\eta_o H}}{ND}\right)_p \quad \text{...(iii)}$$

Introducing the values of (ND) for the model and the prototype turbines from Eqs (i) and (ii) in Eq. (iii), we get

$$\left(\frac{\sqrt{\eta_o H}\times\rho D}{6\times10^5\times\mu}\right)_m = \left(\frac{\sqrt{\eta_o H}\times\rho D}{1.5\times10^8\times\mu}\right)_p$$

Assuming ρ and μ to be same for the model and the prototype turbines, we get

$$\left(\frac{\sqrt{\eta_o H}\times D}{6\times10^5}\right)_m = \left(\frac{\sqrt{\eta_o H}\times D}{1.5\times10^8}\right)_p$$

or
$$\frac{D_m}{D_p} = \frac{\sqrt{\eta_{op} H_p}}{\sqrt{\eta_{om} H_m}}\times\frac{6\times10^5}{1.5\times10^8} = \frac{\sqrt{\eta_{op} H_p}}{250\times\sqrt{\eta_{om} H_m}} \quad \text{...(iv)}$$

From Eq. 22.20 (a), we have

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{0.2}$$

Introducing the value of (D_m/D_p) from Eq. (iv), we get

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{\sqrt{\eta_{op} H_p}}{250\times\sqrt{\eta_{om} H_m}}\right)^{0.2}$$

$$\eta_{om} = 82\% = 0.82; H_m = 20 \text{ m; and } H_p = 50 \text{ m}$$

Thus by substitution, we get

$$\frac{1-\eta_{op}}{1-0.82} = \left(\frac{\sqrt{\eta_{op}\times50}}{250\times\sqrt{0.82\times20}}\right)^{0.2}$$

or
$$\frac{1-\eta_{op}}{0.18} = \frac{(\eta_{op})^{0.1}}{2.699}$$

Solving for η_{op} by trial and error, we get

$$\eta_{op} = 0.9337 \text{ or } 93.37\%$$

Thus efficiency of prototype turbine is 93.37%

Equation 22.20 (a) which correlates the efficiencies of the model and the prototype turbines has been used because it is applicable for head less than 150 m.

Example 22.14. *Show that the expressions for the non-dimensional numbers (i) specific speed, (ii) specific diameter, (iii) flow number, and (iv) head number for Pelton wheel are functions of only the diameter ratio (d/D) where d is the jet diameter and D is Pelton wheel diameter. It is assumed that friction effect is neglected and 180° reversal of flow is achieved by the Pelton wheel buckets.*

Find out the values of the above dimensionless numbers if diameter ratio d/D is 0.2.

Solution

Assuming $K_v = 1$ and $K_u = 0.5$, then for the given conditions the overall efficiency $\eta_0 = 1$.

(i) The non-dimensional specific speed is given by

$$n_s = \frac{N\sqrt{P}}{\rho^{1/2}(gH)^{5/4}}$$

or
$$n_s = \frac{N\sqrt{wQH\eta_o}}{(\rho g)^{1/2} g^{3/4} H^{5/4}}$$

or
$$n_s = \frac{N\sqrt{Q}}{(gH)^{3/4}}$$

Since $w = \rho_g$ and $\eta_o = 1$

Further $Q = \frac{\pi d^2}{4} \times K_v\sqrt{2gH} = \frac{\pi d^2}{4} \times \sqrt{2gH}$

Since $K_v = 1$

Also $u = \frac{\pi DN}{60} = K_u\sqrt{2gH}$

or
$$N = \frac{60K_u\sqrt{2gH}}{\pi D} = \frac{30\sqrt{2gH}}{\pi D}$$

since $K_u = 0.5$

Thus by substitution, we get

$$n_s = \frac{30\sqrt{2gH} \times \sqrt{(\pi d^{2/4})\sqrt{2gH}}}{\pi D \times (gH)^{3/4}}$$

or $n_s = 14.23\,(d/D)$

For $(d/D) = 0.2$

$n_s = (14.23 \times 0.2) = 2.846$

(ii) The non-dimensional specific diameter is given by Eq. 22.9 as

$$D_s' = \left(\frac{D_s}{D}\right) = \left(\frac{P_s}{P_u}\right)^{1/2} = \frac{1}{\sqrt{P_u}}\text{; since } P_s = 1 \text{ kW}$$

$$P_u = \left(\frac{wQ_u \times 1}{1000}\right)\eta_o \text{ kW}$$

$$Q_u = \frac{Q}{D^2\sqrt{H}} = \frac{(\pi/4)d^2 \times \sqrt{2gH}}{D^2\sqrt{H}}$$

Thus
$$D_s' = \frac{1}{\sqrt{\frac{9810}{1000} \times \frac{\pi}{4} \times \left(\frac{d}{D}\right)^2 \times \sqrt{2g}}}$$

or
$$D_s' = \frac{1}{5.84(d/D)}$$

For $(d/D) = 0.2$

$$D_s' = \frac{1}{5.84 \times 0.2} = 0.856$$

(iii) The non-dimensional flow number is given by

$$Q' = \frac{Q}{ND^3} = \frac{(\pi/4)d^2 \times \sqrt{2gH}}{\frac{30 \times \sqrt{2gH}}{\pi D} \times D^3}$$

or $Q' = 0.082\,(d/D)^2$

For $(d/D) = 0.2$

$Q' = 0.082 \times (0.2)^2 = 0.003\,28$

(iv) The non-dimensional head number is given by

$$H' = \frac{gH}{N^2D^2}$$

or
$$H' = \frac{\pi^2 N^2 D^2}{(30\sqrt{2})^2 N^2 D^2} = 0.005\,48$$

which is however independent of the ratio (d/D).

Example 22.15. *Calculate the diameter, the speed and N_s of a propeller turbine runner to develop 6250 kW under a head of 5 m, having given that speed ratio K_u based on outer diameter = 2.10, flow ratio ψ = 0.65; diameter of boss = 0.35 times external diameter of the runner; overall efficiency η_o = 85%.*

Solution

$$\eta_o = 0.85 = \frac{6250 \times 10^3}{(9810 \times Q \times 5)}$$

$\therefore$ $Q = 149.91 \text{ m}^3/\text{s}$

$$\psi = 0.65 = \frac{V_f}{\sqrt{2 \times 9.81 \times 5}}$$

$\therefore$ $V_f = 6.44 \text{ m/s}$

$$Q = 149.91 = \frac{\pi}{4}\left[D^2 - (0.35D)^2\right] \times 6.44$$

$\therefore \quad D = 5.81$ m

$$K_u = 2.10 = \frac{u}{\sqrt{2\times 9.81\times 5}}$$

$\therefore \quad u = 20.8$ m/s

$$u = 20.8 = \frac{\pi DN}{60} = \frac{\pi\times 5.81\times N}{60}$$

$\therefore \quad N = 68.37$ r.p.m.

$$N_s = \left(\frac{N\sqrt{P}}{H^{5/4}}\right)$$

$$= \frac{68.37\times(6250)^{1/2}}{(5)^{5/4}} = 722.93$$

Example 22.16 *The following data were obtained from the main characteristics of a Kaplan turbine of runner diameter 1 m: $P_u = 30.695$; $Q_u = 108.6$; $N_u = 63.6$. Estimate the runner diameter, the discharge and the speed of a similar runner working under a head of 30 m and developing 2000 kW. Determine the specific speed of the runner.*

Solution

The two runners will be similar if

$$\left(\frac{Q_1}{N_1 D_1^3}\right) = \left(\frac{Q_2}{N_2 D_2^3}\right);\ \left(\frac{N_1 D_1}{\sqrt{H_1}}\right) = \left(\frac{N_2 D_2}{\sqrt{H_2}}\right);$$

and
$$\left(\frac{N_1\sqrt{P_1}}{H_1^{5/4}}\right) = \left(\frac{N_2\sqrt{P_2}}{H_2^{5/4}}\right)$$

From these conditions the following identity may be obtained

$$\frac{P_1}{D_1^2 H_1^{3/2}} = \frac{P_2}{D_2^2 H_2^{3/2}}$$

or
$$\frac{30.695}{(1)^2} = \frac{2000}{D_2^2\times(30)^{3/2}};\ \therefore D_2 = 0.6297 \text{ m}$$

$$\frac{N_1 D_1}{\sqrt{H_1}} = \frac{N_2 D_2}{\sqrt{H_2}}$$

or
$$(63.6\times 1) = \frac{N_2\times 0.6297}{(30)^{1/2}};\ \therefore N_2 = 553.20 \text{ r.p.m.}$$

Again from the above noted conditions the following identity may be obtained

$$\frac{Q_1}{D_1^2\sqrt{H_1}} = \frac{Q_2}{D_2^2\sqrt{H_2}}$$

or
$$\frac{108.6}{(1)^2} = \frac{Q_2}{(0.6297)^2\times(30)^{1/2}}$$

$\therefore \quad Q_2 = 235.86$ m^3/s

$$N_s = \frac{N\sqrt{P}}{H^{5/4}} = N_u\sqrt{P_u}$$

For the first runner

$$N_s = 63.6\sqrt{30.695} = 352.36$$

For the second runner

$$N_s = \frac{553.20\sqrt{2000}}{(30)^{5/4}} = 352.36$$

The two values agree, as they ought to do.

Example 22.17 *A water turbine develops 130 kW at 230 r.p.m., under a head of 16 m. Determine the scale ratio and the speed of a similar machine which will generate 660 kW when working under a head of 25 m.*

Solution

Since
$$\frac{P_1}{D_1^2 H_1^{3/2}} = \frac{P_2}{D_2^2 H_2^{3/2}}$$

or
$$\frac{130}{D_1^2(16)^{3/2}} = \frac{660}{D_2^2(25)^{3/2}}$$

$\therefore$ Scale ratio $\dfrac{D_2}{D_1} = 1.612$

Also
$$\frac{N_1 D_1}{\sqrt{H_1}} = \frac{N_2 D_2}{\sqrt{H_2}}$$

or
$$\frac{230\times D_1}{\sqrt{16}} = \frac{N_2\times(1.612 D_1)}{\sqrt{25}}$$

$\therefore \quad N_2 = 178.35$ r.p.m.

Example 22.18 *The quantity of water available for a hydroelectric station is 275 m³/s under a head of 18 m. Assuming the speed of the turbines to be 150 r.p.m., and their efficiency 82%, determine the least number of machines, all of the same size, that will be needed if (i) Francis turbines whose N_s must not exceed 395, (ii) Kaplan turbines whose N_s must not exceed 690, are chosen. What would be the individual output of units in the two cases?*

Solution

Total power to be developed by the turbines in kW

$$= \left(\frac{wQH}{10^3} \times \eta_o\right)$$

$$= \frac{9810 \times 275 \times 18 \times 0.82}{10^3} = 39\,819 \text{ kW}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

(i) For Francis turbines considering the limiting value of N_s as 395, we have

$$395 = \frac{150\sqrt{P}}{(18)^{5/4}}$$

$$\therefore \quad P = 9\,532 \text{ kW}$$

$\therefore$ Number of turbines needed

$$= \frac{39819}{9532} \simeq 5$$

(ii) For Kaplan turbines taking N_s = 690, we have

$$690 = \frac{150\sqrt{P}}{(18)^{5/4}}$$

$$\therefore \quad P = 29\,087 \text{ kW}$$

$\therefore$ Number of turbines needed

$$= \frac{39819}{29087} \simeq 2.$$

Example 22.19. *A 90 MW hydroelectric plant is installed with reaction turbines with head = 26.5 m, overall efficiency = 85%, speed ratio (based on outer diameter) = 2.0, flow ratio = 0.65, diameter of boss = 0.35 × external diameter of runner. The turbine runner is of propeller turbine running at 192 r.p.m. Determine the specific speed of turbine and number of turbines needed.*

Solution

For a propeller turbine from Eq. 22.15, we have

$$N_s = C_1\sqrt{C_2}\left[K_u\sqrt{\psi(1-n^2)\eta_0}\right]$$

where

$$C_1 = \frac{60\sqrt{2g}}{\pi} = 84.6; \text{ and}$$

$$C_2 = \frac{w\pi\sqrt{2g}}{4} = 34.13$$

Also

$$K_u = 2.0;\ \psi = 0.65;\ n = 0.35; \text{ and } \eta_0 = 0.85$$

Thus by substitution, we get

$$N_s = 84.6\sqrt{34.13}\left[2.0 \times \sqrt{0.65[1-(0.35)^2] \times 0.85}\right]$$

$$\therefore \quad N_s = 688.27$$

From Eq. 22.11, we have

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$$N = 192 \text{ r.p.m; and } H = 26.5 \text{ m}$$

Thus by substitution, we get

$$688.27 = \frac{192\sqrt{P}}{(26.5)^{5/4}}$$

$$\therefore \quad P = 46\,454.7 \text{ kW}$$

Total installed capacity at the plant

$$= 90 \text{ MW} = 90\,000 \text{ kW}$$

$\therefore$ Number of turbines needed

$$= \frac{90000}{46454.7} \simeq 2$$

Alternative Method

$$u = K_u\sqrt{2gH} = \frac{\pi DN}{60}$$

Thus by substitution, we get

$$2.0 \times \sqrt{2 \times 9.81 \times 26.5} = \frac{\pi D \times 192}{60}$$

$$\therefore \quad D = 4.536 \text{ m}$$

$$V_f = \psi\sqrt{2gH}$$

or

$$V_f = 0.65 \times \sqrt{2 \times 9.81 \times 26.5} = 14.821 \text{ m/s}$$

$$Q = \frac{\pi}{4} D^2(1-n^2) \times V_f$$

or

$$Q = \frac{\pi}{4} \times (4.536)^2 \times [1-(0.35)^2] \times 14.821$$

or

$$Q = 210.165 \text{ m}^3/\text{s}$$

$$\eta_0 = \frac{P}{wQH}$$

or
$$0.85 = \frac{P}{9810 \times 210.165 \times 26.5}$$

$\therefore$
$$P = 0.85 \times 9810 \times 210.165 \times 26.5 \text{ W}$$
$$= 46.44 \times 10^6 \text{ W}$$
$$= 46.44 \times 10^3 \text{ kW}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

$\therefore$
$$N_s = \frac{192 \times \sqrt{46.44 \times 10^3}}{(26.5)^{5/4}} = 688.16$$

Number of turbines needed

$$= \frac{90000}{46.44 \times 10^3} \simeq 2.$$

Example 22.20. *A turbine is to operate under a head of 25 m at 200 r.p.m. If the discharge is 9 m³/s and turbine efficiency is 90%, calculate power generated by the turbine, specific speed of the turbine and performance of the turbine under a head of 20 m. Also state the type of turbine.*

Solution

$$\eta_o = \frac{P}{wQH}$$

or
$$P = wQH \times \eta_o$$

Thus by substitution, we get

$$P = 9\,810 \times 9 \times 25 \times 0.9 \text{ W}$$
$$= 198.65 \times 10^4 \text{ W}$$
$$= 1986.5 \text{ kW}$$

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

Thus by substitution, we get

$$N_s = \frac{200\sqrt{1986.5}}{(25)^{5/4}} = 159.46$$

For this value of N_s this can be a Francis turbine.

The performance of the turbine under a head of 20 m is indicated by the following expressions.

$$N_1 = \left[\frac{N\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \left[\frac{200 \times \sqrt{20}}{\sqrt{25}}\right] = 178.89 \text{ r.p.m.}$$

$$Q_1 = \left[\frac{Q\sqrt{H_1}}{\sqrt{H}}\right]$$

$$= \left[\frac{9 \times \sqrt{20}}{\sqrt{25}}\right] = 8.05 \text{ m}^3/\text{s}$$

$$P_1 = \left[\frac{PH_1^{3/2}}{H^{3/2}}\right]$$

$$= \left[\frac{1986.5 \times (20)^{3/2}}{(25)^{3/2}}\right] = 1421.4 \text{ kW}$$

SUMMARY OF MAIN POINTS

1. The speed, discharge and power developed for a turbine when operating under a unit head are known as unit speed, unit discharge and unit power which may be expressed in terms of the corresponding quantities for the same turbine operating under a head H as indicated below.

Unit speed $N_u = \dfrac{N}{\sqrt{H}}$

Unit speed $u_u = \dfrac{u}{\sqrt{H}}$

Unit discharge $Q_u = \dfrac{Q}{\sqrt{H}}$

Unit power $P_u = \dfrac{P}{H^{3/2}}$

The unit quantities facilitate the comparison between the performances of the turbines of the same type but having different outputs and speeds and working under different heads.

2. The turbines of different types may be compared by considering an imaginary turbine called specific turbine. The *specific turbine* is an imaginary turbine which is identical in shape, geometrical proportions, blade angles, gate setting, etc., with the actual turbine but reduced to such a size that it will develop one kilowatt power (or one metric horse power) under unit head.

The speed of specific runner is known as *specific speed* N_s which may be defined as follows:

The specific speed of any turbine is the speed in r.m.p of a turbine geometrically similar to the actual turbine but of such a size that under corresponding conditions it will develop 1 kilowatt power (or 1 metric horse power) when working under unit head (i.e., 1 metre). It is given by

$$N_s = \frac{N\sqrt{P}}{H^{5/4}}$$

3. The expressions for all the specific speed in terms of known coefficients for different turbines are as given below.

(i) For Pelton wheel

(a) In SI units

$$N_s = \frac{(494.24)K_u(K_p\eta_o)^{1/2}}{m}$$

(b) In metric units

$$N_s = \frac{(576.2)K_u(K_p\eta_o)^{1/2}}{m}$$

(ii) For Francis turbine

(a) In SI units

$$N_s = 988.45\left[K_u\left(\sqrt{km\,\psi\,\eta_o}\right)\right]$$

(b) In metric units

$$N_s = 1152.4\left[K_u\left(\sqrt{kn\,\psi\,\eta_o}\right)\right]$$

(iii) For Kaplan turbine

(a) In SI units

$$N_s = 439.22\,K_u\left(\sqrt{\psi}\right)$$

(b) In metric units

$$N_s = 512.07\,K_u\left(\sqrt{\psi}\right)$$

4. The performance characteristic curves for the turbines are of the following three types;
   (i) Constant head characteristic curves, (or Main characteristic curves)
   (ii) Constant speed characteristic curves, (or operating characteristic curves)
   (iii) Constant efficiency curves, (or Iso-efficiency curves)
5. For the complete similarity to exist between the model and the prototype turbines the following conditions are required to be satisfied.

(i) $$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$$

(ii) $$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p$$

(iii) $$\left(\frac{P}{\rho gHND^3}\right)_m = \left(\frac{P}{\rho gHND^3}\right)_p$$

(iv) $$\left(\frac{N\sqrt{P}}{H^{5/4}}\right)_m = \left(\frac{N\sqrt{P}}{H^{5/4}}\right)_p$$

6. The expression relating the efficiencies of the model and the prototype turbines is given as

$$\frac{1-\eta_{op}}{1-\eta_{om}} = \left(\frac{D_m}{D_p}\right)^{\alpha}\left(\frac{H_m}{H_p}\right)^{\beta}$$

where

α = 0.20 and β = 0 for $H < 150$ m; and

α = 0.25 and β = 0.10 for $H > 150$ m

7. In reaction turbines cavitation may occur at the runner exit or the inlet to the draft tube where the pressure is considerably reduced.

In order to avoid cavitation, the value of Thoma's cavitation factor should not be reduced below the critical cavitation factor σ_c which for different turbines is given as

(i) When specific speed N_s is in (r.p.m., kW, m) units

(a) For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{380.78}\right)^2$$

(b) For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{380.78}\right)^2\right]$$

For Kaplan turbines the values of σ_c obtained by above equation should be increased by 10 per cent.

(ii) When specific speed N_s is in (r.p.m, h.p., m)

(a) For Francis turbines

$$\sigma_c = 0.625\left(\frac{N_s}{444}\right)^2$$

(b) For Propeller turbines

$$\sigma_c = 0.28 + \left[\frac{1}{7.5}\left(\frac{N_s}{444}\right)^2\right]$$

Again for Kaplan turbines the values of σ_c obtained by the above equation should be increased by 10 per cent.

PROBLEMS

22.1 Explain (a) Unit speed, (b) Unit discharge, (c) Unit Power of a hydraulic turbine. Derive expressions for each of them.

22.2 (a) Explain the terms specific speed, unit speed and unit power as applied to hydraulic turbines. Deduce expressions to indicate their values.
(b) A turbine develops 7355 kW under a head of 24.7 m at 210 r.p.m. What is its specific speed? Indicate the type of turbine suitable for the purpose. If this turbine is tested in the laboratory where the head of water available is only 7.5 m, what power will it develop and at what speed?
[*Ans.* 327 r.p.m; Kaplan or Deriaz; 1231 kW; 115.7 r.p.m.]

22.3 A Francis turbine runner having a diameter of 2.92 m operates at 163.5 r.p.m., under 54 m head, and develops 19 900 kW at an efficiency of 87%. Find the other characteristics if this turbine is operated under 60 m head.
[*Ans.* 172.3 r.p.m.; 45.51 m³/s; 23 307 kW]

22.4 An impulse turbine at best speed produces 92 kW under a head of 63 m. By what per cent should the speed be increased for a head of 87 m? [*Ans.* 17.5%]

22.5 In an inward flow reaction turbine the speed ratio is K_u, the flow ratio is ψ, and the breadth ratio is n. If the overall efficiency of the turbine is 85% and the area taken up by the vanes at inlet is 5% of the peripheral area at inlet, then show that $N_s = 888.23\,K_u\left(\sqrt{\psi n}\right)$ (in SI units).
A runner of the above type having $N_s = 185$ is required to develop 6620 kW unde a head of 90 m. Take $\psi = 0.18$, $n = 0.2$, calculate speed and diameter of the runner.
[*Ans.* 630.3 r.p.m., 1.4 m]

22.6 What are the characteristic curves of a hydraulic turbine? How are they useful to practical Engineer? How are small scale models useful in obtaining these curves for a proposed turbine of a hydroelectric installation?

22.7 Explain how the 'best performance curve' can be obtained with the help of characteristic curves of a reaction turbine? If the turbine has to work at a given head and develop given power at maximum efficiency, how this curve can be used to determine the running speed, gate opening, discharge and efficiency

22.8 What is meant by 'cavitation'? What is Thoma's cavitation factor, and what its significance for water turbines?

22.9 Calculate the specific speed of a Francis turbine runner for which K_u = 1.05, ψ = 0.25 and the width of the runner at the outer periphery is 0.25 times the outer diameter. Take k_o as 0.80 and area occupied by vanes as negligible.
[*Ans.* 232.08 r.p.m.]

22.10 The following observations were obtained from a test carried on a turbine working under its designed head of 8.5 m:

Unit speed N_u	56	65	74	84	93	102
Unit power, P_u	8.83	9.19	9.49	9.49	9.34	8.97
Weight flow W(kN/s)	38.85	38.16	37.38	36.54	35.61	34.53

Plot a curve of η_o against N_u. Calculate the turbine N at maximum η_o and hence determine N_s of the turbine. If the head is changed to 10 m, estimate the power developed and find from the graph η_o at a turbine speed of 250 r.p.m.
[*Ans.* 271 r.p.m; 284 r.p.m; 295.4 kW; 75%]

22.11 A model turbine has a runner of diameter 0.61 m. It develops 50 kW under a head of 30 m at a speed of 4000 r.p.m. Compute N_s and N_u for this model. It is required to build a similar turbine to develop 155 kW under a head of 36 m. Calculate the required diameter.
[*Ans.* 402.85 r.p.m.; 730.3 r.p.m.; 0.937 m]

22.12 A 1/4 scale turbine model is tested under a head of 12 m. The prototype turbine is required to work under a head of 30 m at a speed of 425 r.p.m. At what speed the model be run and if it develops 100 kW and uses 1.14 m³/s of water at this speed, what power will be obtained from the prototype turbine, assuming that its efficiency is 3% better than that of the model? Also indicate the type of runner used in this turbine.
[*Ans.* 1054 r.p.m., 6710 kW; Kaplan (N_s = 496)]

22.13 Briefly explain the various considerations in the selection of a proper type of turbine for a hydroelectric station, indicating also the conditions where a particular type of turbine is suitable.

22.14 Estimate the maximum height of a conical draft tube for a Kaplan turbine developing 14 710 kW under a net head of 20 m and running at 145 r.p.m. The turbine is installed at a station where the effective atmospheric pressure is 10.4 m of water. The draft tube must sink at least 1 m below the tail race level. [*Ans.* 1.4 m]

22.15 A model having scale ratio $\frac{1}{10}$ is constructed to determine the best design of Kaplan turbine to develop 7355 kW under a net head of 10 m at a speed of 100 r.p.m. If the head available at the laboratory is 6 m and the model efficiency is 88% find: (i) running speed of the model; (ii) the flow required in the laboratory; (iii) the output power of the model; (iv) the specific speed in each case. Assume same efficiency for the model and prototype turbine.

[*Ans.* (i) 774.6 r.p.m.; (ii) 0.66 m^3/s; (iii) 34.19 kW (iv) 482.3 r.p.m., for both]

22.16 Solve Problem 22.15 assuming that the efficiency of the prototype turbine is 4% better than that of the model turbine.

[*Ans.* (i) 757.6 r.p.m.; (ii) 0.617 m^3/s; (iii) 31.96 kW; (iv) $(N_s)_p$ = 482.3 r.p.m.; $(N_s)_m$ = 456.1 r.p.m.]

Reciprocating Pumps

Chapter 23

23.1 INTRODUCTION

In general a pump may be defined as a mechanical device which when interposed in a pipeline, converts the mechanical energy supplied to it from some external source into hydraulic energy and transfers the same to the liquid through the pipeline, thereby increasing the energy of the flowing liquid. Almost all the pumps increase the pressure energy of the liquid which is subsequently converted into potential energy as the liquid is lifted from a lower level to a higher level. The various pumps may broadly be classified into the following two types:

1. Positive-displacement pumps.
2. Rotodynamic pumps (or dynamic-pressure pumps).

The 'positive displacement' pumps are those pumps in which the liquid is sucked and then it is actually pushed or displaced due to the thrust exerted on it by a moving member, which results in lifting the liquid to the required height. These pumps usually have one or more chambers which are alternately filled with the liquid to be pumped and then emptied again. As such the discharge of liquid pumped by these pumps almost wholly depends on the speed of the pump. The most common example of the positive displacement type of pumps is that of reciprocating pumps.

The 'rotodynamic pumps' have a rotating element, called impeller, through which as the liquid passes its angular momentum changes, due to which the pressure energy of the liquid is increased. As such a rotodynamic pump does not push the liquid as in the case of a positive displacement pump. The centrifugal pumps are the most common examples of rotodynamic pumps.

In this chapter only reciprocating pumps have been dealt with. The centrifugal pumps have been discussed in Chapter 24.

23.2 MAIN COMPONENTS AND WORKING OF A RECIPROCATING PUMP

A reciprocating pump essentially consists of a piston or plunger which moves to and fro in a close fitting cylinder, as shown in Fig. 23.1. The cylinder is connected to suction and delivery pipes, each of which is provided with a non-return or one way valve called suction valve and delivery valve respectively. The function of non-return or one way valve is to admit liquid in one direction only. Thus the suction valve allows the liquid only to enter the cylinder and the delivery valve permits only its discharge from the cylinder. The piston or the plunger is connected to a crank by means of a connecting rod. As the crank is rotated at uniform speed by a driving engine or motor, the piston or plunger moves to and fro (or backward and forward) in the cylinder. When the crank rotates from $\theta = 0°$ to $\theta = 180°$ (see Fig. 23.1), the piston or plunger which is initially at its extreme left position (that is, it is completely inside the cylinder), move to its extreme right position, (that is, it moves outwardly in the cylinder). During the outward movement of the piston or plunger a partial vacuum (pressure below atmospheric) is created in the cylinder, which enables the atmospheric pressure acting on the liquid surface in the

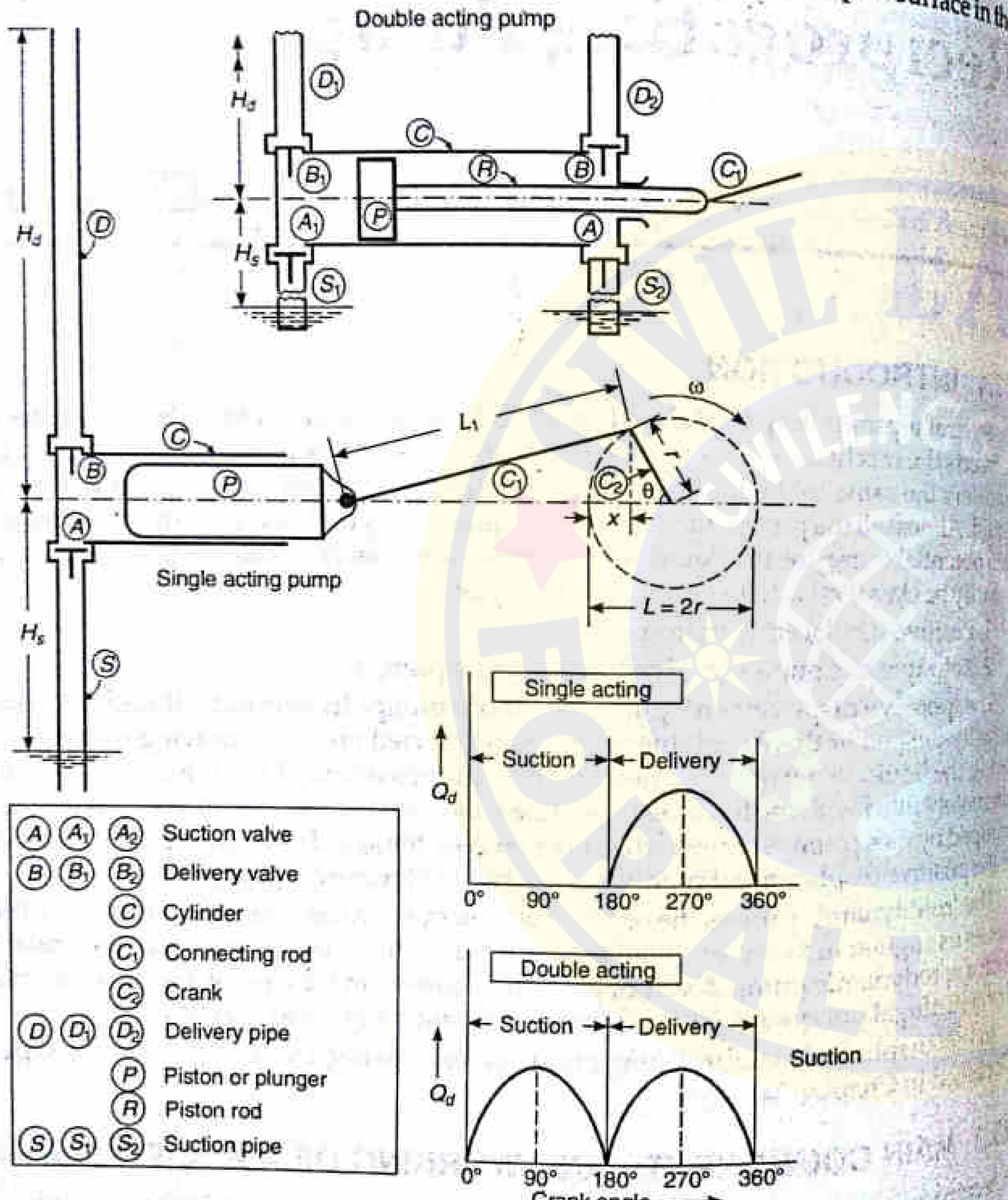


Figure 23.1 *Single acting and double acting reciprocating pumps*

well or sump below, to force the liquid up the suction pipe and fill the cylinder by forcing open the suction valve. Since during this operation of the pump the liquid is sucked from below it is known as its *suction stroke*. Thus at the end of the suction stroke the piston or plunger is at its extreme right position, the crank is at $\theta = 180°$ (i.e., at its outer dead centre), the cylinder is full of liquid, the suction valve is closed and the delivery valve is just at the point of opening.

When the crank rotates from $\theta = 180°$ to $\theta = 360°$ the piston or plunger moves inwardly from its extreme right position towards left. The inward movement of the piston or plunger causes the pressure of the liquid in the cylinder to rise above atmospheric, due to which the suction valve closes and the delivery valve opens. The liquid is then forced up the delivery pipe and raised to the required height. Since during this operation of the pump the liquid is actually delivered to the required height, it is known as its *delivery stroke*. At the end of the delivery stroke the piston or plunger is at extreme left position, the crank is at $\theta = 0°$ or 360° (i.e., at its inner dead centre) so that it has completed one full revolution, and both the suction and the delivery valves are closed. The same cycle is repeated as the crank rotates. This is in general the working principle of a reciprocating pump.

23.3 TYPES OF RECIPROCATING PUMPS

The reciprocating pumps can be classified according to the liquid being in contact with one side or both the sides of the piston or plunger; and according to the number of cylinders provided.

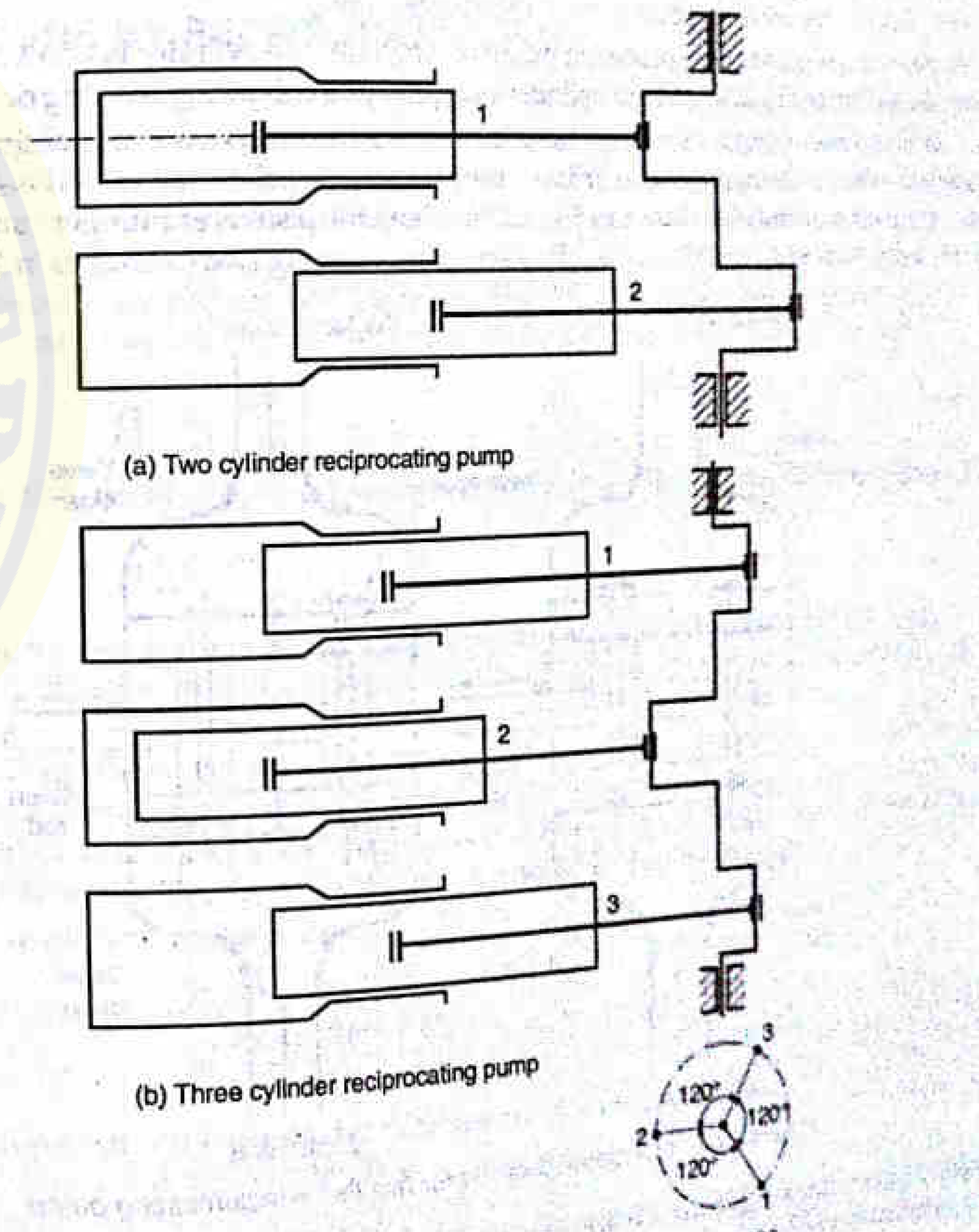


Figure 23.2 *Two and three cylinder reciprocating pumps*

According to the first basis of classification the reciprocating pumps may be classified as:

(i) Single acting pump. (ii) Double acting pump.

If the liquid is in contact with one side of the piston or plunger only, it is known as *single acting pump*. Thus, as shown in Fig. 23.1 a single acting pump has one suction and one delivery pipe and in one complete revolution of crank there are only two strokes—one suction and one delivery stroke. On the other hand if the liquid is in contact with both the sides of the piston or plunger, it is known as *double acting pump*. As shown in Fig. 23.1 a double acting pump has two suction and two delivery pipes with appropriate valves, so that during each stroke when suction takes place on one side of the piston, the other side delivers the liquid. In this way in the case of a double acting pump in one complete revolution of the crank there are two suction strokes and two delivery strokes.

According to the number of cylinders provided the reciprocating pumps may be classified as:

(i) Single cylinder pump, (ii) Double cylinder pump, (iii) Triple cylinder pump, (iv) Duplex double acting pump, (v) Quintuplex pump.

As the name suggests a reciprocating pump having only one cylinder is known as *single cylinder pump*. As shown in Fig. 23.1, a single cylinder pump may be either single acting or double acting.

A *double cylinder pump or two throw pump* is the one which has two single acting cylinders, each equipped with one suction and one delivery pipe with appropriate valves and a separate piston or plunger for each cylinder as shown in Fig. 23.2 (a). Both the pistons or plungers are simultaneously driven from cranks set at 180°. A double acting pump having two cylinders in line but only one

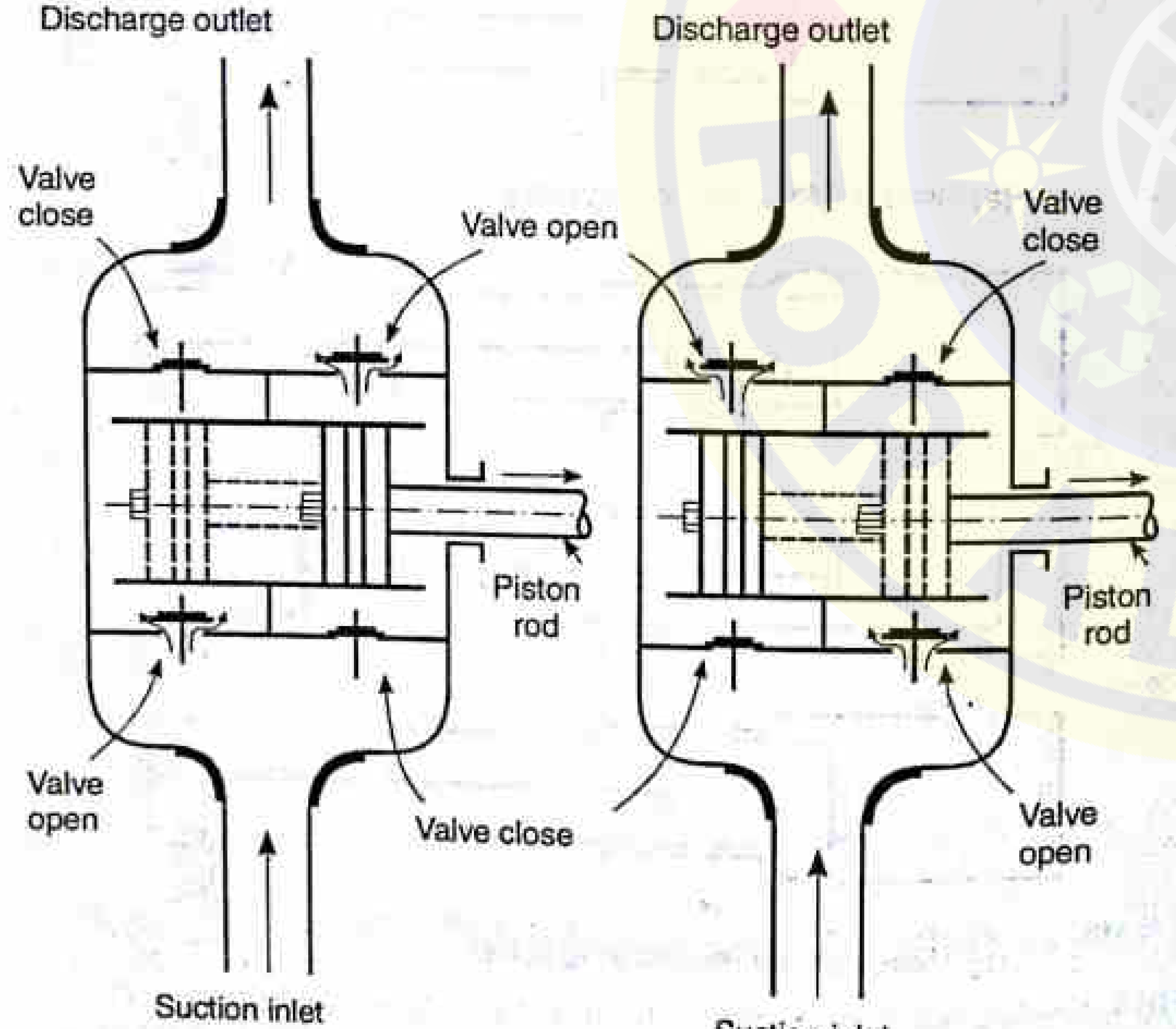


Figure 23.3 *Operation of valves for two throw reciprocating pump*

plunger driven from a single crank is also sometimes manufactured. Another alternative arrangement for a two throw pump may be as shown in Fig. 23.3, in which there is only one suction pipe, one delivery pipe and one piston for both the cylinders. The operation of the valves for such a pump is also shown in Fig. 23.3. The working of a double cylinder pump is similar to that of a double acting single cylinder pump.

A *triple cylinder pump or three throw pump* is the one which has three single acting cylinders each equipped with one suction and one delivery pipe with appropriate valves and a separate piston or plunger as shown in Fig. 23.2 (b). All the three pistons or plungers are simultaneously driven from cranks set at 120°.

A *duplex double-acting pump* is formed by combining in line either two double-acting single cylinder pumps or two double-acting double-cylinder pumps, each driven by a crank and the two cranks being set at 90°. The combination is thus equivalent to a *four-throw pump*.

A *quintuplex or five throw pump* has five single acting cylinders driven from cranks set at 72°.

In general the reciprocating pumps having more than one cylinder are known as *multi-cylinder pumps*.

23.4 WORK DONE BY RECIPROCATING PUMP

The expression for the work done by different types of reciprocating pumps are obtained below.

1. Single-Acting Pump. As explained earlier a single-acting reciprocating pump has only one suction stroke and one delivery stroke for one complete revolution of the crank. Further the liquid is delivered by the pump only during the delivery stroke. Thus if the speed of the crank is N r.p.m., then the number of delivery strokes will be N per minute or $(N/60)$ per second. Also if A is the cross-sectional area of the piston or plunger and L is the length of the stroke, then the theoretical volume of liquid pumped per stroke = (AL); and the theoretical discharge Q_{th} of the pump is given by

$$Q_{th} = \frac{ALN}{60} \quad ...(23.1)$$

As shown in Fig. 23.1, since $L = 2r$, where r is the radius of the crank, Eq. 23.1 may also be expressed as

$$Q_{th} = \frac{2ArN}{60} \quad ...(23.2)$$

However, owing to some leakage and time lag in the closure of the valves the actual discharge Q_a of the pump is slightly different from the theoretical discharge Q_{th}, and in almost all the cases Q_a is less than Q_{th}.

Now if H_s is the height of the centre of the cylinder above the liquid surface in the sump below, and H_d is the height to which the liquid is raised above the centre of the cylinder, then H_s and H_d are termed as suction head and delivery head of the pump respectively and $(H_s + H_d)$ is known as its static head, which is the net total vertical height through which the liquid is lifted by the pump.

The theoretical work done by the pump

$$= wQ_{th}(H_s + H_d) \quad ...(23.3)$$

$$= \frac{w(ALN)(H_s + H_d)}{60}$$

Hence the theoretical power P required to drive the pump is given by

$$P = wQ_{th}(H_s + H_d) \quad ...(23.4)$$

$$= \frac{w(ALN)(H_s + H_d)}{60}$$

here w is the specific weight of the liquid.

However, on account of frictional resistance and leakage the actual power required to drive the pump will be more than that given by Eq. 23.4.

The force acting on the piston or plunger during the suction stroke = (wH_sA). ...(23.5)

The force acting on the piston or plunger during the delivery stroke = (wH_dA). ...(23.6)

2. Double Acting Pump. In the case of a double acting reciprocating pump, when there is a suction stroke on one side of the piston or plunger, its other side has a delivery stroke. Thus for one complete revolution of the crank there are two delivery strokes, during which the liquid is delivered by the pump. Now if the speed of the crank is N r.p.m., then the number of delivery strokes will be $2N$ per minute or $(N/30)$ per second. However, due to the presence of the piston rod on one side, the volume of liquid delivered from both the sides of the piston or plunger will not be equal. As such if A_p is the area of cross-section of piston rod then the theoretical volume of liquid pumped per stroke

$$= [AL + (A - A_p)L];$$

and the theoretical discharge Q_{th} of the pump is given by

$$Q_{th} = \left[\frac{ALN}{60} + \frac{(A-A_p)LN}{60}\right] = \frac{(2A-A_p)LN}{60} \quad ...(23.7)$$

However, if the effect of A_p is neglected then the theoretical discharge Q_{th} is obtained as

$$Q_{th} = \frac{2ALN}{60} \quad ...(23.8)$$

which is equal to twice the theoretical discharge of a single acting pump of same dimensions and same speed.

Again for these pumps also, due to leakage and the imperfect operation of the valves, the actual discharge Q_a of the pump slightly differs from Q_{th} and in almost all the cases Q_a is less than Q_{th}.

The theoretical work done by the pump

$$= wQ_{th}(H_s + H_d) = \frac{w(2A-A_p)LN(H_s+H_d)}{60} \quad ...(23.9)$$

$$\approx \frac{2wALN(H_s+H_d)}{60}$$

Hence the theoretical power P required to drive the pump is given by

$$P = wQ_{th}(H_s+H_d) = \frac{w(2A-A_p)LN(H_s+H_d)}{60} \quad ...(23.10)$$

$$\approx \frac{2w(ALN)(H_s+H_a)}{60}$$

Again on account of frictional resistance and leakage the actual power required to drive the pump will be more than that given by Eq. 23.10.

The force acting on the piston or plunger during outward (left to right) stroke ...(23.11)

$$= [wH_sA + wH_d(A-A_p)].$$

The force acting on the piston or plunger during inward (right to left) stroke ...(23.12)

$$= [wH_dA + wH_s(A-A_p)].$$

23.5 COEFFICIENT OF DISCHARGE, SLIP, PERCENTAGE SLIP AND NEGATIVE SLIP OF RECIPROCATING PUMP

1. Coefficient of Discharge. As explained in section 23.4, due to leakage and the imperfect operation of the valve the actual discharge Q_a of a reciprocating pump is always slightly different from the theoretical discharge Q_{th}. The ratio between the actual discharge Q_a and the theoretical discharge Q_{th} is defined as the *coefficient of discharge* C_d of the pump. That is

$$C_d = \frac{\text{Actual discharge}}{\text{Theoretical discharge}} = \frac{Q_a}{Q_{th}} \quad ...(23.13)$$

The value of C_d when expressed in terms of percentage is also known as *volumetric efficiency* of the pump.

2. Slip and Percentage Slip. The difference between the theoretical discharge and the actual discharge is known as the *slip* of the pump. That is

$$\text{Slip} = (Q_{th} - Q_a) \quad ...(23.14)$$

Often the slip is expressed in terms of percentage which is then called *percentage slip* and it is given by

$$\text{Percentage slip} = \frac{Q_{th} - Q_a}{Q_{th}} \times 100 = (1 - C_d) \times 100 \quad ...(23.15)$$

For the pumps maintained in good condition the percentage slip is of the order of 2 per cent or even less.

3. Negative Slip. For most of the reciprocating pumps the actual discharge Q_a is less than the theoretical discharge Q_{th}, in which case C_d of the pump is less than one and the slip of the pump is positive. However, in some cases the actual discharge of the pump may be more than the theoretical discharge, in which case C_d will be more than one and the slip will be negative, which is then known as *negative slip*.

This may happen in the case of pumps having long suction pipe and low delivery head, especially when these are running at high speed. This is so because for such pumps the inertia pressure in the suction pipe will be large in comparison to the pressure on the outside of the delivery valve, which may cause the delivery valve to open before the suction stroke is completed. Some liquid is thus pushed directly into the delivery pipe even before the delivery stroke is commenced, which results in making the actual discharge more than the theoretical discharge.

23.6 EFFECT OF ACCELERATION OF PISTON ON VELOCITY AND PRESSURE IN THE SUCTION AND DELIVERY PIPES

During the reciprocating motion of the piston or plunger it does not move with uniform velocity for the entire stroke, but as indicated later it has an acceleration developed for the first half of each stroke and for the later half of the stroke there is retardation developed for it. Since the liquid flowing through the pump closely follows the piston or plunger, its acceleration and retarding effects will be correspondingly transmitted to the liquid flowing in the suction and delivery pipes. In other words the velocity of flow of liquid in the suction and delivery pipes will not be uniform but during the same stroke the velocity of flow in these pipes will vary with the crank position. These variations in the velocities of flow of liquid in the suction and delivery pipes give rise to inertia pressures which will cause a variation of pressure in the cylinder. The magnitudes of the inertia pressures so developed can be calculated as indicated below.

If the connecting rod is very long as compared with the length of the crank, then it can be assumed that the piston or plunger moves with simple harmonic motion. The assumption considerably simplifies the problem. Let the crank be rotating with an angular velocity ω radians per second and in time t seconds let it turn through an angle θ, from its inner dead centre. Then

$$\theta = \omega t = \frac{2\pi N t}{60}$$

If x is the linear displacement of the piston from the end of the stroke in t seconds then from Fig. 23.1, we have

$$x = r - r\cos\theta = r - r\cos(\omega t)$$

Further if v is the velocity of the piston and f is the acceleration of the piston then, we have

$$v = \frac{dx}{dt} = \omega r \sin(\omega t) = \omega r \sin\theta$$

and

$$f = \frac{dv}{dt} = \omega^2 r\cos(\omega t) = \omega^2 r\cos\theta$$

Now if A represents the cross-sectional area of the piston or plunger and a is the area of the pipe (either suction or delivery pipe), then since the volume of liquid flowing from the pipe per second equals the volume of liquid flowing into the cylinder per second, then the velocity V of flow of liquid in the pipe is given by

$$V = \frac{A}{a}v = \frac{A}{a}\omega r\sin(\omega t) = \frac{A}{a}\omega r\sin\theta \qquad \text{...(23.16)}$$

Similarly the acceleration of the liquid in the pipe is given by acceleration $= f\frac{A}{a} = \frac{A}{a}(\omega^2 r)\cos\theta$.

Now if l is the length of the pipe, then the weight of the liquid which is subjected to the above noted acceleration is (wal) and the corresponding mass of the liquid is $\left(\frac{wal}{g}\right)$.

Further if p_a is the intensity of pressure due to acceleration of liquid in the pipe then the accelerating force $= (p_a)\,a$.

But according to Newton's second law of motion

Accelerating force = mass × acceleration

Thus

$$(p_a)\,a = \frac{wal}{g}\times\frac{A}{a}(\omega^2 r)\cos\theta$$

or

$$p_a = \frac{wl}{g}\times\frac{A}{a}(\omega^2 r)\cos\theta \qquad \text{...(23.17)}$$

From Eq. 23.17 the pressure head H_a due to ...

$$H_a = \frac{p_a}{w} = \frac{l}{g}\frac{A}{a}(\omega^2 r)\cos\theta \qquad \text{...(23.18)}$$

It may thus be seen from Eq. 23.18 that the pressure head developed due to acceleration acting on the piston or plunger will vary with angle θ.

At the beginning of the stroke when $\theta = 0°$, $\cos\theta = 1$, i.e., when piston or plunger is at its inner dead centre, then

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r) \qquad \text{...(23.19)}$$

At the middle of the stroke when $\theta = 90°$, $\cos\theta = 0$, then

$$H_a = 0 \qquad \text{...(23.20)}$$

At the end of the stroke when $\theta = 180°$, $\cos\theta = -1$, i.e., when piston of plunger is at its outer dead centre, then

$$H_a = -\frac{l}{g}\frac{A}{a}(\omega^2 r) \qquad \text{...(23.21)}$$

Further for $0° < \theta < 90°$, H_a has positive values and for $90° < \theta < 180°$, H_a has negative values, thereby indicating that for the first half of the stroke there is acceleration head development and for the later half of the stroke there is retardation head developed.

The above derived equations for the pressure head H_a may be applied to both the suction and delivery pipes by using either the length l_s and area a_s of the suction pipe; or length l_d and area a_d of the delivery pipe.

However, if the connecting rod is not very long as compared with the length of the crank then simple harmonic motion cannot be assumed for the piston. The pressure head H_a is then given by

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r)\cos\theta\left(\cos\theta + \frac{\cos 2\theta}{n}\right) \qquad \text{...(23.22)}$$

when n is the ratio of the length of connecting rod to the length of crank.

From Eq. 23.22 at the beginning of the stroke

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r)\left(1+\frac{1}{n}\right) \qquad \text{...(23.23)}$$

and at the end of the stroke

$$H_a = \frac{l}{g}\frac{A}{a}(\omega^2 r)\left(1-\frac{1}{n}\right) \qquad \text{...(23.24)}$$

(a) Effect of Variation of Velocity on Friction in Pipes. There will be frictional resistance offered to the liquid flowing through suction and delivery pipes which will cause the loss of head. The loss of head due to friction is given by Darcy-Weisbach equation as

$$h_f = f\frac{l}{d}\frac{V^2}{2g}$$

where f is friction factor, and d is diameter of the pipe.

By substituting for the instantaneous velocity V of flow in the pipe at any point of the stroke from Eq. 23.16 the head loss due to friction becomes

$$h_f = \frac{fl}{2gd}\left(\frac{A}{a}\omega r\sin\theta\right)^2 \quad ...(23.25)$$

From Eq. 23.25 it is obtained that due to the fluctuation in the velocity of flow in pipes the loss of head due to friction also varies with the crank position. Thus at the begnning of the stroke, $\theta = 0°$, $\sin\theta = 0$ and hence $h_f = 0$. Similarly at the end of the stroke, $\theta = 180°$, $\sin\theta = 0$ and hence $h_f = 0$. At the middle of the stroke, $\theta = 90°$, $\sin\theta = 1$ and hence h_f has its maximum value as

$$(h_f)_{max} = \frac{fl}{2gd}\left(\frac{A}{a}\omega r\right)^2 \quad ...(23.26)$$

Further it may be seen from Eq. 23.25 that the variation of h_f with θ is parabolic. Again Eq. 23.25 is applicable to both the suction and delivery pipes and hence by using l_s, d_s, a_s for the suction pipe and l_d, d_d, a_d for the delivery pipe the corresponding values of h_f for each of these pipes may be obtained.

(b) Instantaneous and Mean Rates of Discharge. The instantaneous velocity V_d in the delivery pipe may be obtained from Eq. 23.16 by writing subscript d as

$$V_d = \frac{A}{a_d}(\omega r\sin\theta)$$

$$= \frac{\left(\frac{\pi}{4}\right)D^2}{\left(\frac{\pi}{4}\right)d_d^2}(\omega r\sin\theta)$$

or

$$V_d = \left(\frac{D}{d_d}\right)^2(\omega r\sin\theta)$$

$$= \left(\frac{D}{d_d}\right)^2\left(\frac{2\pi Nr}{60}\sin\theta\right) \quad ...(23.27)$$

where D is the diameter of piston or plunger.

From Eq. 23.27 a curve between V_d and θ can be plotted which will be a sine curve.

For a single acting pump since for one complete revolution of the crank there is only one delivery stroke during which the liquid is delivered, the mean velocity in the delivery pipe can be obtained by integrating Eq. 23.27 as follows

$$(V_d)_{mean} = \frac{1}{2\pi}\left[\int_0^\pi\left(\frac{D}{d_d}\right)^2(\omega r)\sin\theta\, d\theta\right]$$

$$= \left(\frac{D}{d_d}\right)^2\frac{\omega r}{\pi} \quad ...(23.28)$$

However, the above expression for the mean velocity may also be obtained by dividing the theoretical discharge Q_{th} of the pump given by Eq. 23.2, by the area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

From Eq. 23.27 when $\theta = 90°$, $\sin\theta = 1$; the velocity V_d has a maximum value given by

$$(V_d)_{max} = \left(\frac{D}{d_d}\right)^2\omega r \quad ...(23.29)$$

By dividing Eq. 23.28 by Eq. 23.29 we obtain the ratio between the mean velocity and maximum velocity in the delivery pipe as

$$\frac{(V_d)_{mean}}{(V_d)_{max}} = \frac{\left(\frac{D}{d_d}\right)^2\left(\frac{\omega r}{\pi}\right)}{\left(\frac{D}{d_d}\right)^2(\omega r)} = \frac{1}{\pi} \quad ...(23.30)$$

The instantaneous rate of discharge Q_d in the delivery pipe may be obtained by using Eq. 23.27 as

$$Q_d = (V_d a_d) = A\,\omega r\sin\theta \quad ...(23.31)$$

From Eq. 23.31 a plot of Q_d versus θ can be obtained, which will be a sine curve as shown in Fig. 23.1. For a single acting pump during the first half revolution of the crank i.e., for $\theta = 0°$ to $180°$, there is only suction and no delivery and during the second half revolution of the crank i.e., for $\theta = 180°$ to $360°$ there is delivery of liquid. The same cycle is repeated afterwards. Thus the part of the sine curve below the axis will represent suction and that above it will represent delivery. However, in Fig. 23.1 only that part of the curve is shown which represents the delivery of the liquid.

The mean discharge $(Q_d)_{mean}$ for a single acting pump can be obtained by integrating Eq. 23.31 as follows:

$$(Q_d)_{mean} = \frac{1}{2\pi}\left[\int_0^\pi A\omega r\sin\theta\, d\theta\right] = \frac{A\omega r}{\pi} \quad ...(23.32)$$

The above expression for the mean discharge may also be obtained by multiplying $(V_d)_{mean}$ given by Eq. 23.38 by the area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

Again from Eq. 23.31 for $\theta = 90°$, $\sin\theta = 1$, the discharge Q_d has a maximum value given by

$$(Q_d)_{max} = A\omega r \quad ...(23.33)$$

Further by dividing Eq. 23.32 by Eq. 23.33, we obtain

$$\frac{(V_d)_{mean}}{(V_d)_{max}} = \frac{1}{\pi} \quad ...(23.34)$$

For a double acting pump since there are two delivery strokes, for one complete revolution of the crank, the mean velocity of flow of liquid in the delivery pipe may be obtained by integrating Eq. 23.27 as follows

$$(V_d)_{mean} = 2 \times \frac{1}{2\pi}\left[\int_0^{\pi}\left(\frac{D}{d_d}\right)^2 (\omega r)\sin\theta\, d\theta\right]$$

or
$$(V_d)_{mean} = 2 \times \left(\frac{D}{d_d}\right)^2 \frac{\omega r}{\pi} \qquad ...(23.35)$$

Again the above expression for the mean velocity may also be obtained by dividing the theoretical discharge Q_{th} given by Eq. 23.8 by area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

By dividing Eq. 23.35 by Eq. 23.29 we obtain for a double acting pump

$$\frac{(V_d)_{mean}}{(V_d)_{max}} = \frac{2}{\pi} \qquad ...(23.36)$$

The instantaneous rate of discharge Q_d for a double acting pump is also given by Eq. 23.31. As such in this case also a plot of Q_d versus θ will be a sine curve. But for a double acting pump during one complete revolution of the crank i.e., for $\theta = 0°$ to $360°$, there being two delivery strokes, the Q_d versus θ will be a resultant of two sine curves drawn at a phase difference of 180°, as shown in Fig. 23.1, in which only the curves corresponding to the delivery of the liquid are shown. Thus the mean discharge $(Q_d)_{mean}$ for a double acting pump can also be obtained by integrating Eq. 23.31 as follows:

$$(Q_d)_{mean} = 2 \times \frac{1}{2\pi}\left[\int_0^{\pi} A\omega r \sin\theta\, d\theta\right]$$

$$= \frac{2A\omega r}{\pi} \qquad ...(23.37)$$

However the above expression for the mean discharge may also be obtained by multiplying $(V_d)_{mean}$ given by Eq. 23.35 by the area of the pipe $\left(\frac{\pi d_d^2}{4}\right)$.

Again by dividing Eq. 23.37 by Eq. 23.33 we obtain for double acting pump

$$\frac{(Q_d)_{mean}}{(Q_d)_{max}} = \frac{2}{\pi} \qquad ...(23.38)$$

However, if the area of piston rod is taken into account then it can be shown that the instantaneous discharges Q_{d1} and Q_{d2} for the two delivery pipes on either side of piston or plunger will be

$$\left.\begin{aligned} Q_{d_1} &= A\omega r \sin\theta \\ Q_{d_2} &= (A - A_p)\omega r[\sin(\pi-\theta)] \end{aligned}\right\} \qquad ...(23.39)$$

23.7 INDICATOR DIAGRAMS

1. Theoretical Indicator Diagram. Figure 23.4 shows the theoretical indicator diagram of a reciprocating pump which represents the work done by a single acting pump during one complete cycle as given by Eq. 23.3. In this diagram the pressure head on the piston or plunger is plotted along the vertical ordinate and the length of the stroke is represented by the abscissa. The horizontal line *fe* represents the atmospheric pressure. The line *dc* represents the pressure head in the cylinder during the suction stroke, which is below atmospheric pressure by an amount equal to suction head H_s. The line *ab* represents the pressure in the cylinder during the delivery stroke, which is above atmospheric pressure by an amount equal to delivery head H_d. The area *dcef* represents to some scale the work done by the piston or plunger during the suction stroke and the area *abef* represents to the same scale the work done by the piston or plunger during the delivery stroke. The total work done by the piston or plunger during the complete revolution of the crank is then represented to the same scale by area *abcd* which is equal to $(H_s + H_d)L$.

If the pump is double acting, this diagram will represent the pressure head on one side of the piston or plunger only. The work done per revolution will then be represented by twice the area of this diagram, if the area of the piston rod is neglected.

Such a diagram may be obtained automatically by means of an indicator placed on the cylinder and hence it is called an indicator diagram.

2. Effect of Acceleration in Suction Pipe on Indicator Diagram. Due to the acceleration head developed in the suction pipe during the suction stroke, the theoretical indicator diagram will be modified as indicated below. If l_s, d_s, a_s are respectively the length, diameter and the cross-sectional area of the suction pipe, then the acceleration head H_{as} in the pipe is given by Eq. 23.18 as

$$H_{as} = \frac{l_s}{g}\frac{A}{a_s}\,\omega^2 r \cos\theta \qquad ...(23.40)$$

At the beginning of the suction stroke since $\theta = 0°$, $\cos\theta = 1$, the acceleration head is

$$H_{as} = +\frac{l_s}{g} \times \frac{A}{a_s}\,\omega^2 r \qquad ...(23.41)$$

From Eq. 23.41 it is seen that at the beginning of the suction stroke since H_{as} is positive the liquid in the suction pipe has to be accelerated, for which an additional drop in pressure in the cylinder is required. As a result the pressure head in the cylinder will further drop by an amount equal to H_{as}, so that at the beginning of the suction stroke the pressure head will be $(H_s + H_{as})$ below the atmospheric pressure head.

At the middle of the suction stroke since $\theta = 90°$, $\cos\theta = 0$, the acceleration head $H_{as} = 0$. Thus at the middle of the suction stroke the pressure head will be only H_s below the atmospheric pressure head.

At the end of the suction stroke since $\theta = 180°$, $\cos\theta = -1$, the acceleration head is

$$H_{as} = -\frac{l_s}{g} \times \frac{A}{a_s}\,\omega^2 r \qquad ...(23.42)$$

From Eq. 23.42 it is seen that at the end of the suction stroke the liquid is to be retarded, corresponding to which a pressure rise in the cylinder is needed. As a result the pressure head in the cylinder will rise by an amount H_{as}, so that at the end of the suction stroke the pressure head will be $(H_s - H_{as})$ below the atmospheric pressure head.

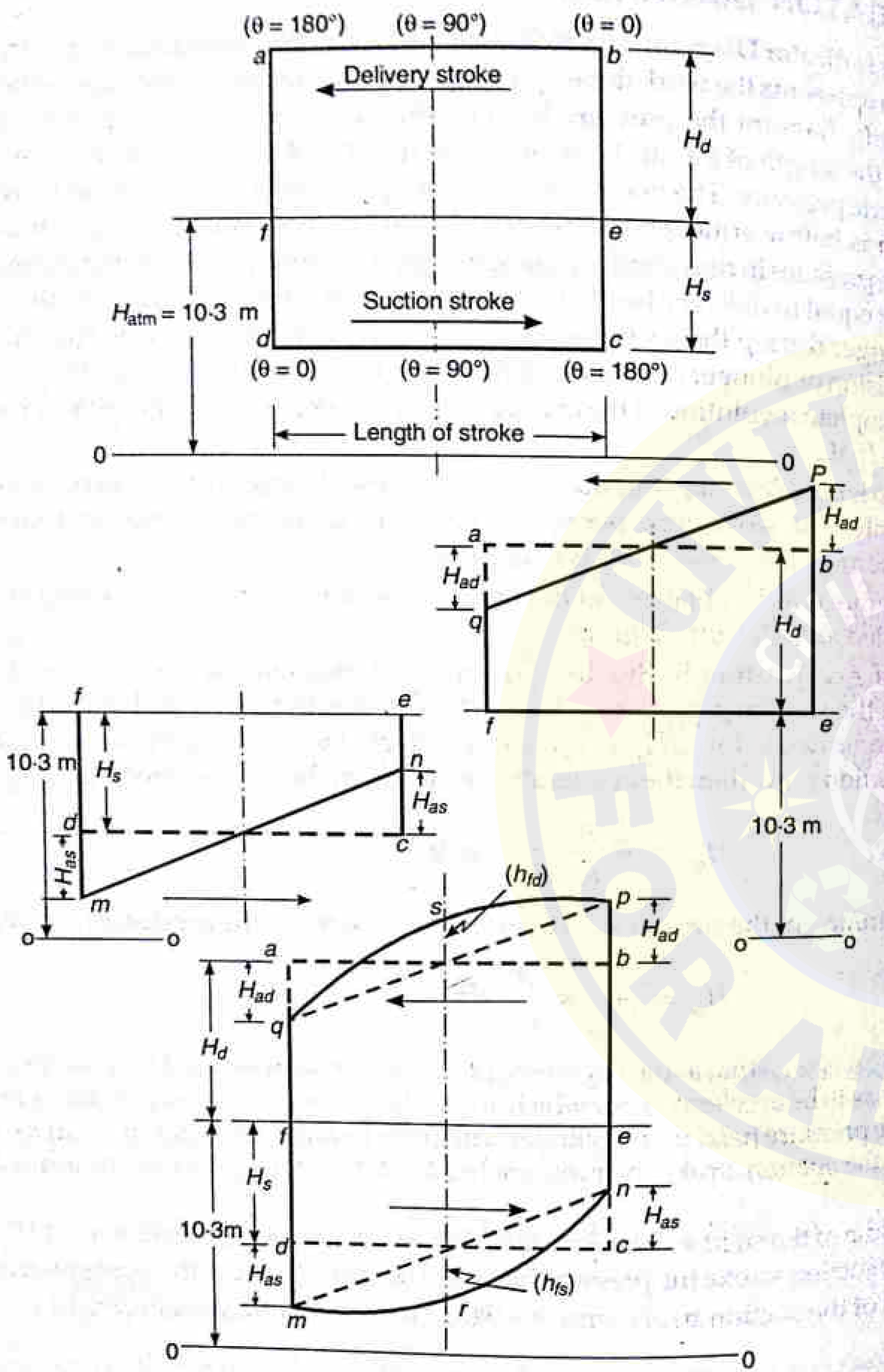


Figure 23.4 *Indicator diagram*

The indicator diagram for the suction stroke will thus be modified as shown in Fig. 23.4. The base of the diagram is now changed from *dc* to *mn*. Also the work done during the suction stroke is now represented by the area *mnef*. However, area *mnef* being equal to the area *dcef*, the net work done during the suction stroke is not altered on account of the accelerating effects in the suction pipe. It is thus

observed that the inertia pressure developed in the suction pipe does not affect the net work done, but merely causes a variation of pressure in the cylinder.

3. Maximum Speed of a Reciprocating Pump. The speed of a reciprocating pump is usually restricted by the pressure corresponding to the point *m* of the indicator diagram. This is so because higher is the speed, the greater will be the acceleration head and lower will be the pressure in the cylinder at the beginning of the suction stroke. The pressure in the cylinder must not be allowed to fall to the value at which dissolved gases are liberated from the liquid, since under these conditions the cavitation may occur and the continuity of the flow may not exist. For water the value of this limiting pressure is about 2.6 m of water absolute or if atmospheric pressure is 10.3 m of water $(H_s + H_{as})$ at any point during the suction stroke must not be greater than $(10.3 - 2.6) = 7.7$ m of water. The maximum permissible speed of a reciprocating pump may thus be computed from the following expression

$$H_{sep} = H_s + H_{as}$$

or

$$H_{sep} = H_s + \frac{l_s}{g} \times \frac{A}{a_s} \omega^2 r$$

or

$$H_{sep} = H_s + \frac{l_s}{g}\frac{A}{a_s}\left(\frac{2\pi N}{60}\right)^2 r \qquad ...(23.43)$$

where H_{sep} represents the pressure head below atmospheric pressure head at which the separation of dissolved gases and cavitation may occur.

It may, however, be seen from Eq. 23.43 that the drop of pressure in the cylinder during the beginning of the suction stroke may also be limited by suitably adjusting H_s, l_s and (A/a_s) for the pump.

4. Effect of Acceleration in Delivery Pipe on Indicator Diagram. As in the case of suction pipe the acceleration head will also be developed for the liquid flowing in the delivery pipe, due to which the indicator diagram will be modified. If l_d, d_d and a_d are respectively the length, diameter and the cross-sectional area of the delivery pipe then the acceleration head in the delivery pipe is given by Eq. 23.18 as

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \omega^2 r \cos\theta \qquad ...(23.44)$$

At the beginning of the delivery stroke $\theta = 0°$, $\cos\theta = 1$, the acceleration head is

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \omega^2 r \qquad ...(23.45)$$

Again at the beginning of the delivery stroke since H_{ad} is positive, the liquid in the delivery pipe is to be accelerated, for which an additional pressure head is required to be developed in the cylinder. As a result at the beginning of the delivery stroke the pressure head in the cylinder will be $(H_d + H_{ad})$ above the atmospheric pressure head.

At the middle of the delivery stroke since $\theta = 90°$, $\cos\theta = 0$, the acceleration head $H_{ad} = 0$. Thus at this point of the delivery stroke the pressure head in the cylinder will be only H_d above the atmospheric pressure head.

At the end of the delivery stroke since $\theta = 180°$, $\cos\theta = -1$, the acceleration head is

$$H_{ad} = -\frac{l_d}{g} \times \frac{A}{a_d} \omega^2 r \qquad ...(23.46)$$

Thus at the end of the delivery stroke the liquid is to be retarded, corresponding to which a drop of pressure in the cylinder is needed. As a result the pressure head in the cylinder will drop by an amount H_{ad}, so that at the end of the delivery stroke the pressure head will be $(H_d - H_{ad})$ above the atmospheric pressure head.

The indicator diagram for the delivery stroke will thus be modified as shown in Fig. 23.4. The top of the diagram is now changed from *ab* to *pq*. Again the work done during the delivery stroke is now represented by the area *pqfe* which is however equal to the area *abef*. As such during the delivery stroke also the net work done is not altered on account of the accelerating effects in the delivery pipe, but these merely cause a variation of pressure in the cylinder.

During the delivery stroke the minimum pressure head in the cylinder is at the end of the stroke, which is represented by the pressure head at point *q* of the indicator diagram and it is equal to $(H_{atm} + H_d - H_{ad})$ absolute. Again this pressure head must not be less than the absolute separation pressure head of the liquid, otherwise cavitation may occur at the end of the delivery stroke. For water the separation pressure head is about 2.6 m of water absolute, and hence in the limiting condition

$$10.3 + H_d - H_{ad} = 2.6$$

or

$$H_{ad} = 7.7 + H_d \qquad ...(23.47)$$

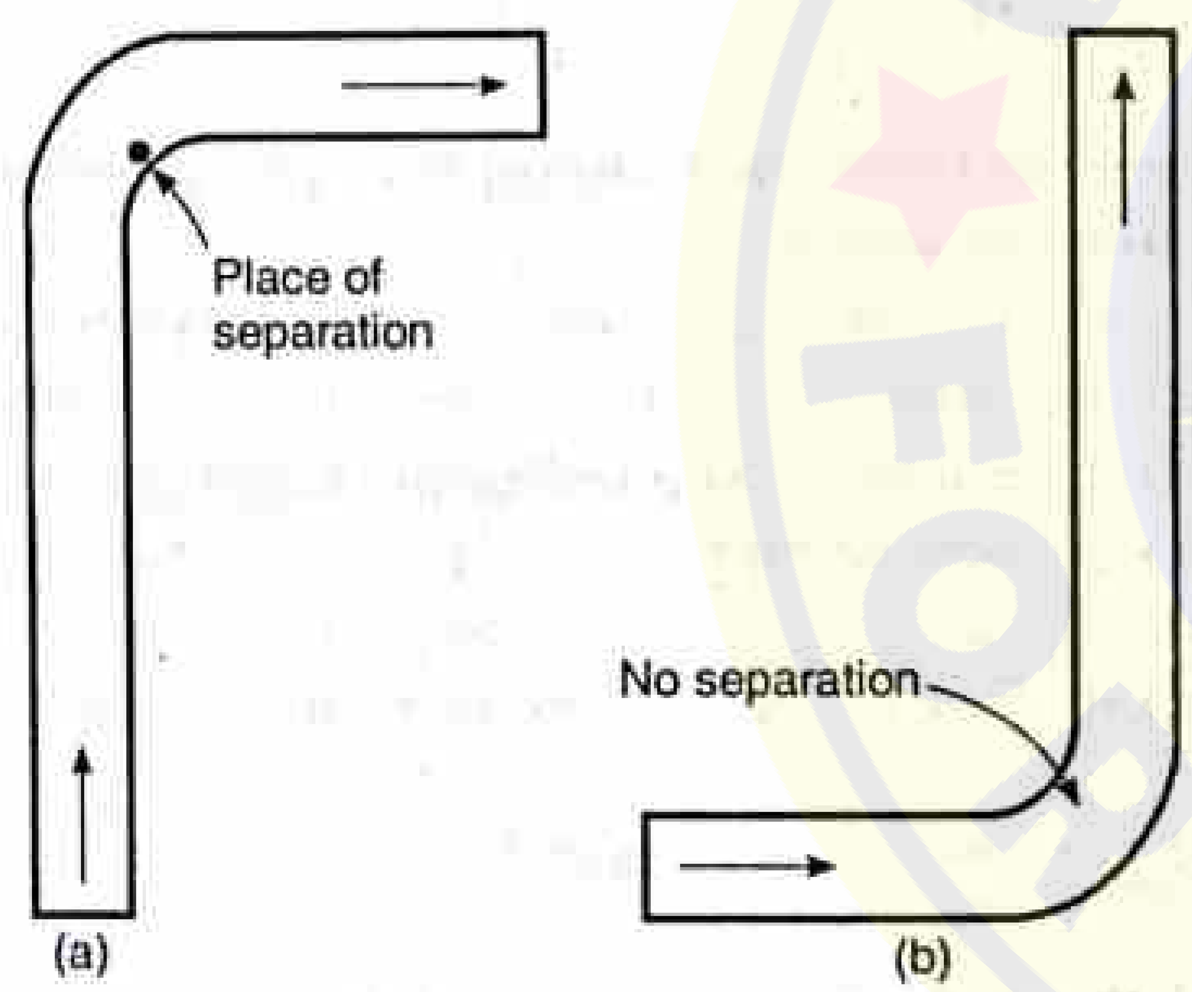


Figure 23.5 *Different arrangements of delivery pipe of a reciprocating pump*

The delivery pipe of a reciprocating pump may have either of the two arrangements as shown in Fig. 23.5. In the arrangement shown in Fig. 23.5(a) the delivery pipe is first vertical and then horizontal. In this case the delivery head H_d will become zero at the bend, after which there is still a long horizontal pipe which will have a considerable value of accelerating head H_{ad}. Therefore according to Eq. 23.47 the separation may take place at the bend in this case. On the other hand in the arrangement shown in Fig. 23.5 (b) the pipe is horizontal first and then it is vertical. As such there is still considerable H_d available at the bend in this case. Therefore there is no possibility of separation to occur at the bend in this case.

5. Effect of Friction in Suction and Delivery Pipes on Indicator Diagram. A further modification of the indicator diagram results from the effect of friction losses in the suction and delivery pipes. The loss of head due to friction h_f in these pipes is given by Eq. 23.25, which shows that the variation of h_f with θ is parabolic. At the beginning and the end of the suction as well as delivery strokes since $\theta = 0°$ and $\theta = 180°$, $\sin\theta = 0$, the head loss due to friction h_{fs} and h_{fd} in the suction and delivery pipes respectively will be equal to zero. At the middle of the stroke since $\theta = 90°$, $\sin\theta = 1$, the head loss due to friction h_{fs} and h_{fd} will be

$$h_{fs} = \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r\right)^2 \qquad ...(23.48)$$

and

$$h_{fd} = \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r\right)^2 \qquad ...(23.49)$$

Now if the effect of the loss of head due to friction in the suction pipe is considered then as shown in Fig. 23.4 the base of the diagram will change from *mn* to *mrn*, the parabola *mrn* being the work done against friction in the suction pipe. Similarly by considering the effect of the loss of head due to friction in the delivery pipe the top of the diagram will change from *pq* to *psq* as shown in Fig. 23.4, the parabola *psq* being the work done against friction in the delivery pipe.

Thus the total work done during suction stroke is represented by area (*efmrn*) which is equal to [area (*efdc*) + area (*mrn*)]. Similarly the total work done during delivery stroke is represented by area (*efqsp*) which is equal to [area (*abef*) + area (*psq*)].

As the mean ordinate of a parabola equals two-thirds of its maximum ordinate the total area of the indicator diagram *mrnpsq* is equal to

$$L\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right]$$

which to some scale represents the total work done during one complete revolution for a single acting pump. The total work done per second for a single acting pump is given as

$$\text{Work done} = \frac{wALN}{60}\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right] \qquad ...(23.50)$$

The absolute pressure head on the piston or plunger during suction stroke for any position of the crank i.e., for any crank angle θ may now be expressed as

$$H_{ts} = H_{atm} - H_s - H_{as} - h_{fs}$$

or

$$H_{ts} = H_{atm} - H_s - \left(\frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r\cos\theta\right) - \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r\sin\theta\right)^2 \qquad ...(23.51)$$

From Eq. 23.51 it can be shown that the minimum value of the absolute pressure H_{ts} will occur for $\theta = 0°$ i.e., at the beginning of the suction stroke. Therefore if cavitation occurs during the suction stroke, it will do so at the beginning of the stroke, at the junction of suction pipe with the cylinder.

Again the absolute pressure head on the piston or plunger during delivery stroke for any position of the crank i.e., for any crank angle θ may be expressed as

$$H_{td} = H_{atm} + H_d + H_{ad} + h_{fd}$$

or
$$H_{td} = H_{atm} + H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right) + \frac{fl_d}{2gd_d}\left(\frac{A}{a_d} \times \omega r \sin\theta\right)^2 \quad ...(23.52)$$

From Eqs 23.51 and 23.52 it is evident that the acceleration heads H_{as} and H_{ad} have maximum values at the ends of the stroke i.e., when $\theta = 0°$ and $\theta = 180°$, and are equal to zero at the middle of the stroke i.e., when $\theta = 90°$. On the other hand the frictional head losses h_{fs} and h_{fd} are equal to zero at the ends of the stroke and have a maximum value at the middle of the stroke.

23.8 AIR VESSELS

For a single acting pump, the flow rate in the delivery pipe at any instant varies considerably, having a value of zero during the whole of the suction stroke and rising to a maximum during the delivery stroke. In other words the supply of liquid by a single-acting pump will be quite intermittent. To some extent this difficulty may be overcome if the pump is double acting or if it is a multi-cylinder pump so that a continuous supply of liquid may be obtained. But in the case of almost all the pumps the velocity of flow in both suction and delivery pipes vary with the crank position, which results in developing a non-uniform rate of flow of liquid in these pipes. As such in order to obtain a continuous supply of liquid at a uniform rate from a single-acting pump and also to obtain a uniform rate of flow of liquid in both suction and delivery pipes of a reciprocating pump, a large air vessel is fitted to the suction as well as the delivery pipe at a point close to the cylinder. An *air vessel* is a close chamber having an opening at its base through which the liquid may flow into the vessel or it may flow out from the vessel. The top portion of the air-vessel contains compressed air, which will be further compressed when the liquid enters the vessel and it will expand when the liquid flows out from the vessel. Figure 23.6 shows air-vessels fitted to suction and delivery pipes of a single-acting pump, the working of each of these is explained below.

Consider an air vessel fitted to the suction pipe of the pump. During the first half of the suction stroke the piston or plunger moves with acceleration and hence the liquid in the suction pipe has also to be accelerated. In other words the liquid in the suction pipe must have a velocity of flow more than the mean velocity, so that the corresponding flow rate of liquid entering the cylinder may be more than the mean discharge. This excess quantity of liquid required on account of accelerating effects will however be supplied from the air vessel, so that the velocity of flow of liquid in the portion of the suction pipe below the point at which the air vessel is connected to it will be equal to the mean velocity of flow only. During the second half of the suction stroke the piston or plunger moves with retardation and hence the liquid in the suction pipe is also required to be retarded. In other words the liquid in the suction pipe must have a velocity of flow less than the mean velocity, so that the corresponding flow rate of liquid entering the cylinder is less than the mean discharge. Since the velocity of flow of liquid in the portion of the suction pipe below the point at which the air vessel is connected to it is equal to the mean velocity of flow, the excess quantity of liquid flowing in the suction pipe will flow into the air vessel thus compressing the air inside the vessel. In this way a relatively small quantity of liquid will only be entering the cylinder from the suction pipe. The liquid thus stored in the air-vessel will be supplied during the first half of the next suction stroke and the same cycle will thus be repeated.

From the foregoing discussion it is thus clear that by fitting an air vessel to the suction pipe, throughout the suction stroke a uniform velocity of flow occurs in the portion of the suction pipe below the point at which the air vessel is connected and the fluctuations in the velocity of flow due to the accelerating effects will be limited only to the portion of the suction pipe between the cylinder and the

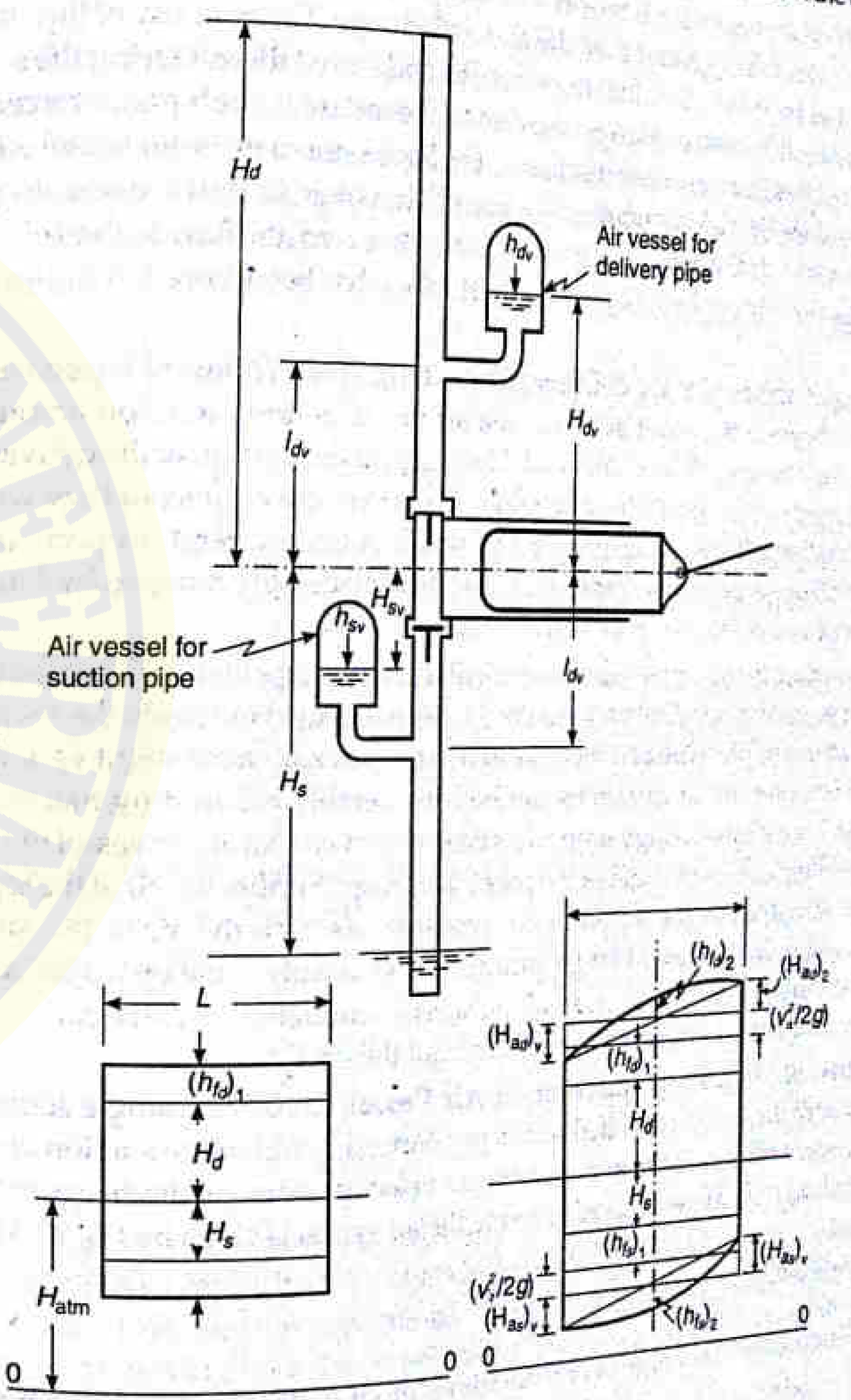


Figure 23.6 *Reciprocating pump with air vessels*

point at which the air vessel is connected to the suction pipe. Since the velocity fluctuations are eliminated for most of the length of the suction pipe, the acceleration pressure heads will also be eliminated and only very small acceleration pressure heads corresponding to the length of the suction pipe between the cylinder and the point at which the air vessel is connected to it, will be developed. As such the acceleration pressure heads may be considerably reduced by fitting the air vessel to the suction pipe at a point very close to the cylinder.

Now consider an air vessel fitted to the delivery pipe of the pump. Again during the first half of the delivery stroke the piston or plunger moves with acceleration and hence the liquid is forced into the delivery pipe at a rate of flow greater than the mean discharge. The quantity of liquid in excess of the mean discharge will, however, flow into the air vessel and stored there. During the second half of the delivery stroke, since the piston or plunger moves with retardation, the liquid is forced in the delivery pipe at a rate of flow less than the mean discharge. The liquid stored in the air vessel will then flow into the delivery pipe, thereby making up the deficiency. Moreover, in the case of a single acting pump during the next suction stroke, when the pump discharge is zero, the flow in the delivery pipe will be maintained by the liquid supplied by the air vessel which has been stored in it during the previous delivery stroke.

Thus, by fitting an air vessel to the delivery pipe a uniform rate of flow of liquid will be obtained in the delivery pipe beyond the point at which the air vessel is fitted to it, on account of which the acceleration pressure heads will be eliminated. However, in the portion of the delivery pipe between the cylinder and the point at which the air vessel is fitted to it, the velocity of flow will be fluctuating due to the accelerating effects, corresponding to which small acceleration pressure heads will be developed. Again these acceleration heads may also be considerably reduced by fitting the air vessel to the delivery pipe at a point very close to the cylinder.

As the liquid surface in the air vessel rises and falls some change in the air pressure will take place, which will result in a corresponding variation of the pressure and velocity in the suction and delivery pipes. Such variations of pressure and velocity in these pipes will cause slight variation in the rate of flow of liquid. However, these variations can be considerably reduced by making the air vessels sufficiently large. For a single acting pump it has been observed that an air vessel of volume about six to nine times the volume of the cylinder of the pump is enough. Further the air at the top of the air vessel also gets slowly dissolved in the liquid under pressure. As such the air in the air vessel must be renewed periodically. In the case of larger pumps this is usually done by a special air compressor provided for this purpose. However, in small pumps an automatic air valve called a *snifting valve* is provided which may draw in the necessary quantity of the air.

(a) Rate of Flow of Liquid into and from the Air Vessel. Consider a single acting pump with air vessels fitted to both the suction and delivery pipes. Since a liquid in the suction and delivery pipes beyond the air vessel will move with a constant mean velocity, the mean discharge in the delivery pipe beyond the air vessel or in the suction pipe below the air vessel is given by Eq. 23.32 as

$$(Q_d)_{mean} = \frac{A\omega r}{\pi}$$

Further the instantaneous discharge to or from the cylinder of the pump is given by Eq. 23.31 as

$$Q_d = A\omega r \sin\theta$$

Obviously the difference of the above two discharge will be equal to the rate of flow of liquid into or from the air vessel.

∴ Rate of flow of liquid into the air vessel

$$= \left(A\omega r \sin\theta - \frac{A\omega r}{\pi}\right)$$

$$= A\omega r\left(\sin\theta - \frac{1}{\pi}\right) \quad ...(23.53)$$

If Eq. 23.53 is positive, it means that the liquid is flowing into the air vessel fitted to the delivery pipe. But if this equation is negative, then it means that the liquid is flowing from the air vessel.

However, if we consider the flow into or from the air vessel fitted to the suction pipe then the above condition will be reversed. That is if Eq. 23.53 is positive then the liquid is flowing from the air vessel. But if Eq. 23.53 is negative then the liquid is flowing into the air vessel.

For no flow of liquid into or from the air vessel, Eq. 23.53 is put equal to zero. That is,

$$A\omega r\left(\sin\theta - \frac{1}{\pi}\right) = 0; \quad \text{or } \sin\theta = \frac{1}{\pi} = 0.3183$$

$$\therefore \quad \theta = 18°\,34' \text{ or } 161°\,26'$$

Thus for crank angle θ = 18° 34′ and 161° 26′ there will be no flow into or from the air vessel. Further for crank angle θ = 0° to 18° 34′ and θ = 161° 26′ to 180° the instantaneous discharge is less than the mean discharge and for θ = 18° 34′ to 161° 26′ the instantaneous discharge is more than the mean discharge.

For a 'double acting pump' the mean discharge in the suction pipe below the air vessel or in the delivery pipe beyond the air vessel is given by Eq. 23.37 as

$$(Q_d)_{mean} = \frac{2A\omega r}{\pi}$$

Since in this case also the instantaneous discharge to or from the cylinder is given by Eq. 23.31 as

$$Q_d = A\omega r \sin\theta$$

Rate of flow of liquid into the air vessel

$$= \left(A\omega r \sin\theta - \frac{2A\omega r}{\pi}\right)$$

$$= A\omega r\left(\sin\theta - \frac{2}{\pi}\right) \quad ...(23.54)$$

Again, if Eq. 23.54 is positive it means that the liquid is flowing into the air vessel fitted to delivery pipe. But if this equation is negative it means that the liquid is flowing from the air vessel.

For the air vessel fitted to the suction pipe the above condition will be reversed. Thus, if Eq. 23.54 is positive then the liquid is flowing from the air vessel. But if this equation is negative then the liquid is flowing into the air vessel.

For no flow of liquid into or from the vessel, Eq. 23.54 is equated to zero. That is,

$$A\omega r\left(\sin\theta - \frac{2}{\pi}\right) = 0; \quad \text{or } \sin\theta = \frac{2}{\pi} = 0.6366$$

$$\therefore \quad \theta = 39°\,32' \text{ or } 140°\,28'$$

Thus, in the case of a double acting pump for crank angle $\theta = 39° 32'$ and $140° 28'$ there will be no flow into or from the air vessel. Again, for crank angle $\theta = 0°$ to $39° 32'$ and $\theta = 140° 28'$ to $180°$, the instantaneous discharge is less than the mean discharge and for $\theta = 39° 32'$ to $140° 28'$ the instantaneous discharge is more than the mean discharge.

(b) Pressure Heads in the Cylinder during Delivery Stroke of a Pump with Air Vessel. If l_{dv} is the length of the delivery pipe between the cylinder and the point at which an air vessel is fitted to it as shown in Fig. 23.6 then the acceleration head developed is given by Eq. 23.18 as

$$(H_{ad})_v = \frac{l_{dv}}{g} \times \frac{A}{a_d}(\omega^2 r \cos\theta) \qquad ...(23.55)$$

Similarly the head loss due to friction in this position of the pipe is given by Eq. 23.25 as

$$(h_{fd})_2 = \left(\frac{fl_{dv}}{2gd_d}\right)\left(\frac{A}{a_d}\omega r \sin\theta\right)^2 \qquad ...(23.56)$$

If l_d is the length of the delivery pipe beyond the air vessel and v_d is the constant mean velocity in the delivery pipe beyond the air vessel then the head loss due to friction in this portion of the delivery pipe is given by Darcy-Weisbach equation as

$$(h_{fd})_1 = \frac{fl_d v_d^2}{2gd_d} \qquad ...(23.57)$$

The constant mean velocity v_d is given by Eq. 23.28 for a single acting pump and by Eq. 23.35 for a double acting pump.

The total pressure head developed during the delivery stroke of a reciprocating pump with air vessel may thus by obtained for any crank angle θ which may be expressed as

$$H_{td} = H_d + \frac{v_d^2}{2g} + (H_{ad})_v + (h_{fd})_1 + (h_{fd})_2 \qquad ...(23.58)$$

and $$(H_{td})_{absolute} = H_{td} + H_{atm} \qquad ...(23.59)$$

From Eqs. 23.55 to 23.59, by putting $\theta = 0°$, $90°$ and $180°$, for the beginning, middle and end of the delivery stroke respectively, the corresponding values of H_{td} can be obtained. However, if l_{dv} is very small, $(H_{ad})v$ and $(h_{fd})_2$ can be neglected. Sometimes the velocity head $(v_d^2/2g)$ is also neglected then

$$H_{td} = H_d + (h_{fd})_1 \qquad ...(23.60)$$

Further the total pressure head of the air in the air vessel on the delivery side reckoned above the centre of the cylinder will be approximately equal to the total pressure head in the delivery pipe above the same datum. Thus if h_{dv} represents the pressure head of the air in the air vessel and H_{dv} is the vertical height of the liquid surface in the air vessel above the centre of the cylinder, then

$$(h_{dv} + H_{dv}) = H_{td} + H_{atm} = (H_{td})_{absolute} \qquad ...(23.61)$$

where H_{td} can be obtained from Eq. 23.58 or 23.60.

(c) Pressure Heads in the Cylinder during Suction Stroke of a Pump with Air Vessel. The same analysis as that for the delivery stroke given above in section 23.8 (b) may be applied to compute the various pressure heads developed during the suction stroke. Thus, the corresponding relationships for suction stroke may be written by using subscript s in place of d in Eqs. 23.55 to 23.61. Thus

$$(H_{as})_v = \frac{l_{sv}}{g} \times \frac{A}{a_s}(\omega^2 r \cos\theta) \qquad ...(23.62)$$

$$(h_{fs})_2 = \frac{fl_{sv}}{2gd_s}\left(\frac{A}{a_s}\omega r \sin\theta\right) \qquad ...(23.63)$$

$$(h_{fs})_1 = \frac{fl_s v_s^2}{2gd_s} \qquad ...(23.64)$$

in which v_s is the constant mean velocity in the suction pipe below the point at which the air vessel is connected to it, which for single acting pump is given by

$$v_s = \frac{2ArN}{60a_s} = \frac{ALN}{60a_s} = \frac{A\omega r}{\pi a_s}$$

and for a double acting pump by

$$v_s = \frac{4ArN}{60a_s} = \frac{2ALN}{60a_s} = \frac{2A\omega r}{\pi a_s}$$

The total pressure head developed during the suction stroke of a reciprocating pump with air vessel may thus be obtained for any crank angle θ which may be expressed as

$$H_{ts} = H_s + \frac{v_s^2}{2g} + (H_{as})_v + (h_{fs})_1 + (h_{fs})_2 \qquad ...(23.65)$$

Again if $(H_{as})_v$, $(h_{fs})_2$ and $(v_s^2/2g)$ are negligibly small, then

$$H_{ts} = H_s + (h_{fs})_1 \qquad ...(23.66)$$

Since during the suction stroke the total pressure head developed is below atmospheric pressure or it is vaccum pressure head and hence

$$(H_{ts})_{absolute} = H_{atm} - H_{ts} \qquad ...(23.67)$$

Again if h_{sv} represents the total pressure head of the air in the air vessel on the suction side then as in the case of delivery stroke

$$h_{sv} - H_{sv} = H_{atm} - H_{ts} \qquad ...(23.68)$$

(d) Work Done by a Reciprocating Pump with Air Vessels. For a pump fitted with air vessel on both suction and delivery pipes, the theoretical work done per second can be expressed as

$$\text{Work done} = wQ_{th}\left[(H_s + H_d) + (h_{fs})_1 + (h_{fd})_1 + \left(\frac{v_s^2}{2g} + \frac{v_d^2}{2g}\right) + \frac{2}{3}(h_{fs}) + \frac{2}{3}(h_{fd})_2\right] \qquad ...(23.69)$$

Since the acceleration heads do not affect the total work done. In Eq. 23.69 the values of $(h_{fs})_2$ and $(h_{fd})_2$ correspond to $\theta = 90°$ i.e., these are respectively the maximum values of the head loss due to friction in the portions of the suction and delivery pipes between the cylinder and the air vessels. However, if all the small quantities are neglected then

$$\text{Work done} = wQ_{th}\,[(H_s + H_d) + (h_{fs})_1 + (h_{fd})_1)] \quad \ldots(23.70)$$

Equation 23.70 also gives the theoretical power required to drive the pump fitted with air vessels.

(e) Effect of Air Vessels on the Indicator Diagram. Since by fitting air vessels the pressure heads developed during the suction and delivery strokes of the pump will also be modified. Thus if all the quantities of Eqs 23.58 and 23.65 are taken into account then the modified indicator diagram will be as shown in Fig. 23.6. However, if small quantities are neglected then from Eqs 23.60 and 23.66, the indicator diagram will be simply rectangular as shown in Fig. 23.6, since $(h_{fs})_1$ and $(h_{fd})_1$ are independent of the crank angle θ.

(f) Maximum Speed of a Reciprocating Pump with Air Vessel. As indicated in section 23.7 (c) the maximum permissible speed of a reciprocating pump is governed by the drop of pressure in the cylinder at the beginning of the suction stroke. Since the pressure in the cylinder must not drop below H_{sep}, the maximum speed of the pump may be computed by the expression

$$H_{sep} = H_s + (h_{fs})_1 + \frac{v_s^2}{2g} + (H_{as})_v$$

or

$$H_{sep} = H_s + \frac{f l_s v_s^2}{2gd_s} + \frac{v_s^2}{2g} + \frac{l_{sv}}{g} \times \frac{A}{a_s}(\omega^2 r)$$

or

$$H_{sep} = H_s + \frac{f l_s}{2gd_s}\left(\frac{A\omega r}{\pi a_s}\right)^2 + \frac{1}{2g}\left(\frac{A\omega r}{\pi a_s}\right)^2 + \frac{l_{sv}}{g} \times \frac{A}{a_s}(\omega^2 r)$$

or

$$H_{sep} = H_s + \frac{f l_s}{2gd_s}\left(\frac{2ArN}{60a_s}\right)^2 + \frac{1}{2g}\left(\frac{2ArN}{60a_s}\right)^2 + \frac{l_{sv}}{g} \times \frac{A}{a_s}\left(\frac{2\pi N}{60}\right)^2 r \quad \ldots(23.71)$$

However, if the small quantities are neglected then the limiting speed of the pump may be computed by the following expression:

$$H_{sep} = H_s + \frac{f l_s}{2gd_s}\left(\frac{2ArN}{60a_s}\right)^2 \quad \ldots(23.72)$$

(g) Work Saved against Friction by Fitting Air Vessel. When a pump is equipped with air vessels, the fluctuations in the velocity of flow in suction and delivery pipes are eliminated, which results in reducing the head loss due to friction in these pipes and hence certain amount of energy is saved. The saving in the energy may be obtained by the following analysis which is applicable to both the suction as well as delivery strokes.

For a single acting pump without air vessel on the pipe the work W_1 lost in friction per stroke is given by

$$W_1 = wAL\left(\frac{2}{3}h_f\right)$$

$$= wA\left(\frac{4r}{3}\right)\left(\frac{fl}{2gd}\right)\left(\frac{A}{a}\omega r\right)^2 \quad \ldots(23.73)$$

Now if an air vessel is fitted to the pipe just outside the cylinder, the velocity of flow in the pipe will be constant being equal to the mean velocity and therefore the work W_2 lost in friction per stroke is given by

$$W_2 = wAL(h_f)_1 = wA\,(2r)\left[\frac{flv^2}{2gd}\right] \quad \ldots(23.74)$$

But from the Eq. 23.32

$$v = \frac{A\omega r}{\pi a}$$

Thus

$$W_2 = wA\,(2r)\left[\frac{fl}{2gd}\right]\left[\frac{A}{a}\frac{\omega r}{\pi}\right]^2 \quad \ldots(23.75)$$

The work saved per stroke by fitting an air vessel to the suction or delivery pipe of a single acting pump is obtained by subtracting Eq. 23.75 from Eq. 23.73 as

$$W_1 - W_2 = (2r)wA\left[\frac{fl}{2gd}\right]\left[\frac{A}{a}\omega r\right]^2\left[\frac{2}{3} - \frac{1}{\pi^2}\right] \quad \ldots(23.76)$$

The percentage of the work saved during the stroke

$$= \frac{W_1 - W_2}{W_1} \times 100 = \frac{\left(\frac{2}{3}\right) - \left(\frac{1}{\pi^2}\right)}{\left(\frac{2}{3}\right)} \times 100$$

$$= 84.8\%$$

For a 'double acting pump' without air-vessel the work W_1 lost in friction per stroke is same as i[n] the case of single acting pump as given by Eq. 23.73.

When an air vessel is fitted to the pipe outside the cylinder, then the work W_2 lost in friction pe[r] stroke is also given by Eq. 23.74. But for a double acting pump from Eq. 23.37

$$v = \frac{2A\omega r}{\pi a}$$

$$\therefore \quad W_2 = wA\,(2r)\left[\frac{fl}{2gd}\right]\left[\frac{2A\omega r}{\pi a}\right]^2 \quad \ldots(23.7$$

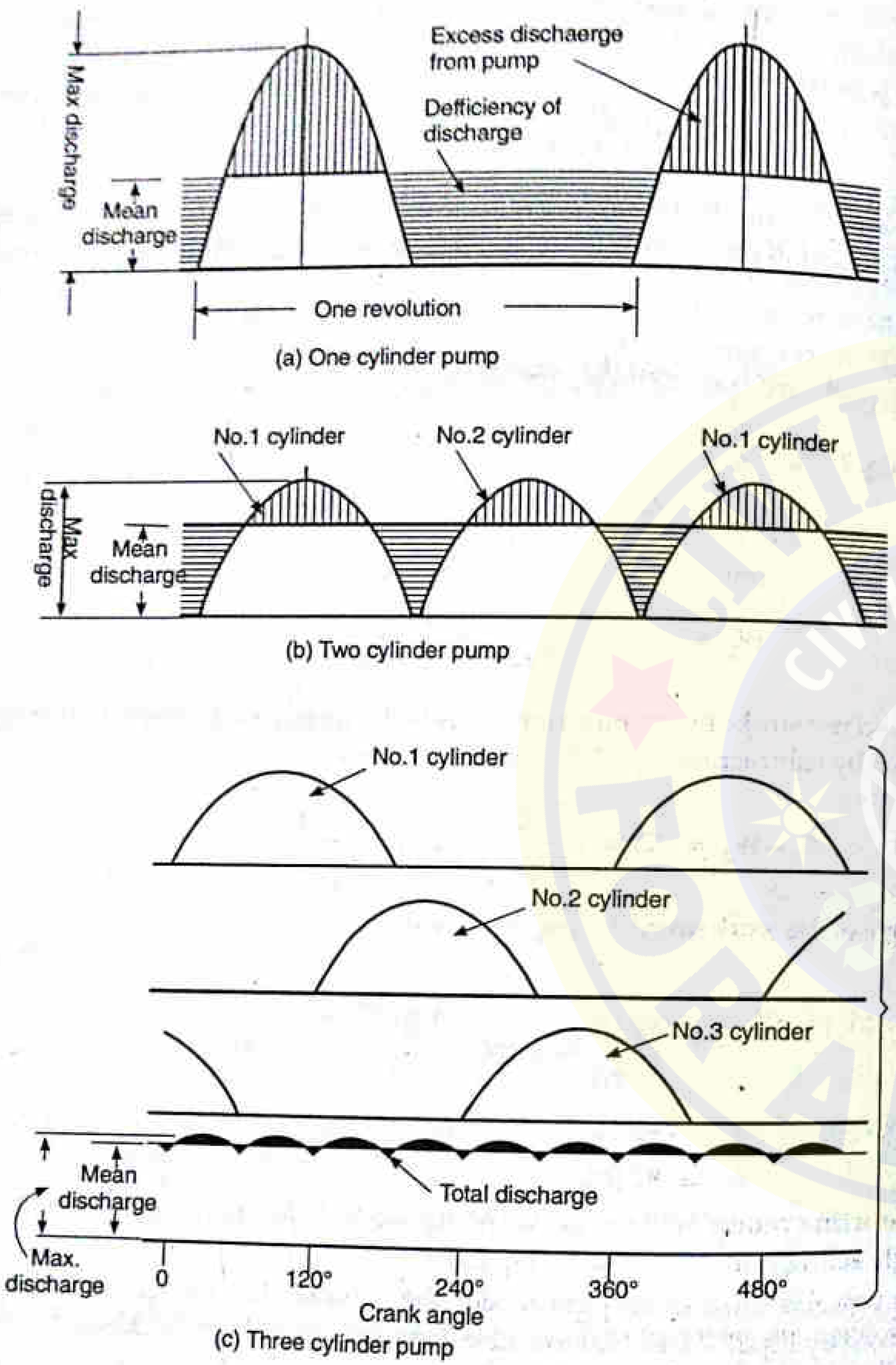


Figure 23.7 *Discharge diagram for one, two, and three cylinder pumps (without air vessels)*

By subtracting Eq. 23.77 from Eq. 23.73 the work saved per stroke by fitting an air vessel to the suction or delivery pipe of a double acting pump is obtained as

$$W_1 - W_2 = (2r)wA\left[\frac{fl}{2gd}\right]\left[\frac{A}{a}\omega r\right]^2\left[\frac{2}{3}-\frac{4}{\pi^2}\right] \quad \text{...(23.78)}$$

Again the percentage of the work saved during any stroke for a double acting pump

$$= \frac{W_1 - W_2}{W_1} \times 100 = \frac{\left(\frac{2}{3}\right)-\left(\frac{4}{\pi^2}\right)}{\left(\frac{2}{3}\right)} \times 100$$

$$= 39.2\%.$$

23.9 MULTI-CYLINDER PUMPS

As discussed earlier, pumps having more than one cylinder are known as 'multicylinder pumps'. Some of these pumps are shown in Fig. 23.2. The main advantage of multicylinder pumps are that these pumps even without air vessels deliver liquid more uniformly as compared with a single cylinder pump. This is made clear by the various discharge diagrams for different types of pumps shown in Fig. 23.7, in which the instantaneous discharge from each individual cylinder is compared with the mean discharge from the pump. It will be observed that by a suitable disposition of the discharge from the individual cylinder, the total discharge of a multicylinder pump may be made nearly equal to the mean discharge with almost negligible fluctuation. On account of much smaller fluctuations of velocity in both the suction and delivery pipes of a multicylinder pump, no air-vessel is required for the same.

23.10 OPERATING CHARACTERISTIC CURVES OF RECIPROCAING PUMPS

The operating characteristic curves indicating the performance of a reciprocating pump are shown in Fig. 23.8. These curves are obtained by plotting discharge, power input and overall efficiency against the head developed by the pump when it is operating at a constant speed. As shown in Fig. 23.8, under

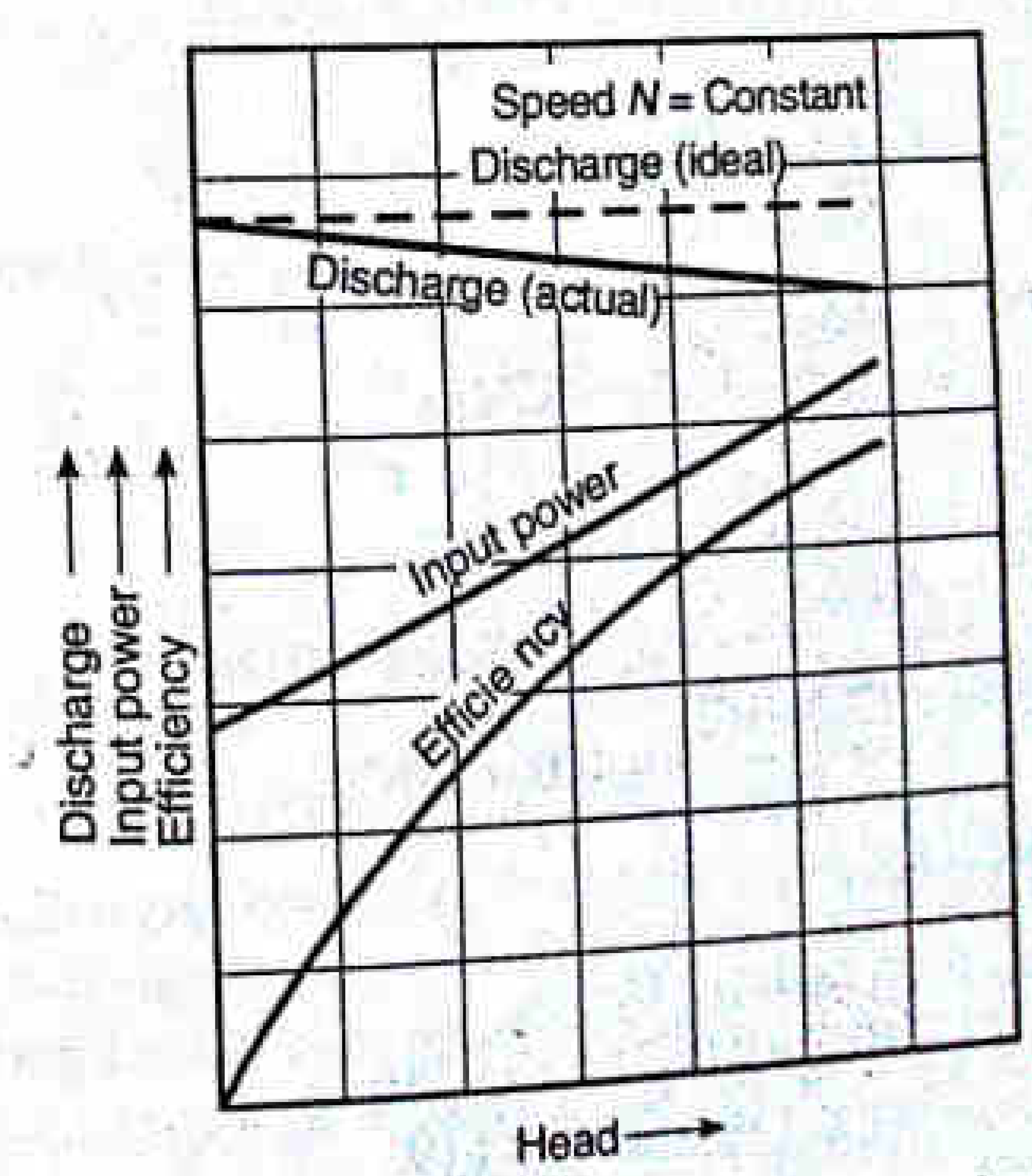


Figure 23.8 *Operating characteristic curves of a reciprocating pump*

ideal conditions the discharge of a reciprocating pump operating at constant speed is independent of the head developed by the pump. However, in actual practice it is observed that the discharge of a reciprocating pump slightly decreases as the head developed by the pump increases. Further the input power for a reciprocating pump increases almost linearly beyond a certain minimum value with the increase in the head developed by the pump. The overall efficiency of a reciprocating pump also increases with the increase in the head developed by the pump as shown in Fig. 23.8.

In addition to the above noted constant speed characteristic curves, sometimes variable speed characteristic curves of a reciprocating pump are also plotted. For this the pump is operated at different speeds and its discharge corresponding to a constant head is plotted against the speed. It is observed that the discharge of a reciprocating pump increases almost linearly with the increase in the speed of the pump, which is in accordance with the expression for the theoretical discharge given by Eqs. 23.1 and 23.8.

ILLUSTRAITVE EXAMPLES

Example 23.1. *A single acting reciprocating pump has a plunger of diameter 250 mm and stroke of 350 mm. If the speed of the pump is 60 r.p.m. and it delivers 16.5 litres per second of water against a suction head of 5 m and a delivery head of 20 m, find the theoretical discharge, coefficient of discharge, the slip, the percentage slip of the pump and the power required to drive the pump.*

Solution

Theoretical discharge of the pump

$$Q_{th} = \frac{ALN}{60}$$

$$= \frac{\pi}{4}\left(\frac{250}{1000}\right)^2 \times \frac{350}{1000} \times \frac{60}{60}$$

$$= 0.0172 \text{ m}^3/\text{s} = 17.2\ l/\text{s}$$

Coefficient of discharge

$$C_d = \frac{Q_a}{Q_{th}} = \frac{16.5}{17.2} = 0.96$$

$$\text{Slip of the pump} = (Q_{th} - Q_a)$$

$$= (17.2 - 16.5) = 0.7\ l/\text{s}$$

$$\text{Percentage slip} = \frac{Q_{th} - Q_a}{Q_h} \times 100$$

$$= \frac{17.2 - 16.5}{17.2} \times 100 = 4.07\%.$$

Power required to drive the pump

$$p = wQ_{th}\,(H_s + H_d)$$

$$= 9810 \times 0.0172 \times (5 + 20)$$

$$= 4218.3 \text{ W} = 4.2183 \text{ kW}$$

Example 23.2. *A single acting reciprocating pump has piston of diameter 150 mm and stroke of length 250 mm. The piston makes 50 double strokes per minute. The suction and delivery heads are 5 m and 15 m respectively. Find (i) discharge capacity of the pump in litres per minute; (ii) force required to work the piston during the suction and delivery strokes if the efficiency of suction and delivery strokes are 60% and 75% respectively; and (iii) power required to operate the pump.*

Solution

(i) Theoretical discharge of the pump

$$Q_{th} = \frac{ALN}{60}$$

$$= \frac{\pi}{4}\left(\frac{150}{1000}\right)^2 \times \frac{250}{1000} \times \frac{50}{60}$$

$$= 0.00368 \text{ m}^3/\text{s}$$

$$= 3.68\ l/\text{s} = 221 \text{ litres/min}$$

(ii) Force required to work the piston during the suction stroke

$$= \left(\frac{wH_sA}{\eta_1}\right)$$

$$= \frac{9810 \times 5 \times (\pi/4)(150/1000)^2}{0.6}$$

$$= 1444.64 \text{ N}$$

Force required to work the piston during the delivery stroke

$$= \left(\frac{wH_dA}{\eta_2}\right)$$

$$= \frac{9810 \times 15 \times (\pi/4)(150/1000)^2}{0.75}$$

$$= 3467.14 \text{ N}$$

(iii) Power required to operate the pump

$$P = wQ_{th}\left(\frac{H_s}{\eta_1} + \frac{H_d}{\eta_2}\right)$$

$$= 9810 \times 0.00368 \times \left(\frac{5}{0.60} + \frac{15}{0.75}\right)$$

$$= 1023 \text{ W} = 1.023 \text{ kW}$$

Note: Equations 23.5 and 23.6 give only theoretical force acting on the piston or plunger during the suction and delivery strokes respectively and hence in order to obtain the actual force acting on the piston or plunger during these strokes the expressions in these equations are divided by the efficiencies of the respective strokes.

Example 23.3. *A double acting reciprocating pump has piston of diameter 250 mm and piston rod of diameter 50 mm which is on one side only. Length of piston stroke is 350 mm and speed of crank moving the* *piston is 60 r.p.m. The suction and delivery heads are 4.5 m and 18 m respectively. Determine the discharge capacity of the pump and the power required to operate the pump.*

Solution

For a double acting pump the theoretical discharge is given by

$$Q_{th} = \frac{(2A - A_p)LN}{60}$$

$$A = \frac{\pi}{4}\left(\frac{250}{1000}\right)^2 = 0.04909 \text{ m}^2$$

$$A_p = \frac{\pi}{4}\left(\frac{50}{1000}\right)^2 = 0.00196 \text{ m}^2$$

$$L = \frac{350}{1000} = 0.35 \text{ m}; N = 60 \text{ r.p.m}$$

Thus by substitution, we get

$$Q_{th} = \frac{(2 \times 0.04909 - 0.00196) \times 0.35 \times 60}{60}$$

$$= 0.0337 \text{ m}^3/\text{s} = 33.7 \; l/\text{s}$$

Power required to drive the pump

$$= wQ_{th}(H_s + H_d)$$

$$= 9810 \times 0.0337 \; (4H5 + 18)$$

$$= 7438 \text{ W} = 7.438 \text{ kW}$$

Example 23.4. *A double acting reciprocating pump having piston area 0.1 m² has a stroke 0.30 m long. The pump is discharging 2.4 m³ of water per minute at 45 r.p.m. through a height of 10 m. Find the slip of the pump and the power required to drive the pump.*

Solution

Neglecting the area of the piston rod, the theoretical discharge of the pump is

$$Q_{th} = \frac{2ALN}{60} = \frac{2 \times 0.1 \times 0.3 \times 45}{60}$$

$$= 0.045 \text{ m}^3/\text{s}$$

Actual discharge $\quad Q_a = \dfrac{2.4}{60} = 0.040 \text{ m}^3/\text{s}$

$\therefore$ Slip of the pump

$$= (Q_{th} - Q_a)$$

$$= (0.045 - 0.040) = 0.005 \text{ m}^3/\text{s}$$

Power required to drive the pump

$$= wQ_{th}(H_s + H_d)$$

$$= 9810 \times 0.045 \times 10$$

$$= 4\,415 \text{ W} = 4.415 \text{ kW}$$

Example 23.5. *A single acting reciprocating pump has a piston diameter 0.15 and stroke length 0.30. The centre of the pump is 5 m above the level of water in sump and 33 m below delivery water level. The length of suction and delivery pipes are 6.5 m and 39 m respectively and both the pipes have the same diameter of 75 mm. If the pump is working at 30 r.p.m. find the pressure head on the piston at the beginning, middle and end of both suction and delivery strokes, and find the power required to drive the pump. Take atmospheric pressure as 10.3 m of water and Darcy's friction factor for both the pipes as 0.04.*

Solution

Suction Stroke:

At the ends of the stroke

$$H_{as} = \frac{l_s}{g} \times \frac{A}{a_s}\omega^2 r$$

$$= \frac{6.5}{9.81} \times \left[\frac{\left(\frac{\pi}{4}\right) \times (0.15)^2}{\left(\frac{\pi}{4}\right) \times (0.075)^2}\right] \times \left(\frac{2\pi \times 30}{60}\right)^2 \times 0.15$$

$$= 3.92 \text{ m of water}$$

At the middle of the stroke

$$h_{fs} = \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r\right)^2$$

$$= \frac{0.04 \times 6.5}{2 \times 9.81 \times 0.075} \times \left[\frac{\left(\frac{\pi}{4}\right) \times (0.15)^2}{\left(\frac{\pi}{4}\right) \times (0.075)^2} \times \left(\frac{2\pi \times 30}{60}\right) \times 0.15\right]^2$$

$$= 0.628 \text{ m of water}$$

$\therefore$ Pressure heads in metres of water during suction stroke will be as follows:

At beginning $\quad = (H_s + H_{as}) = (5 + 3.92) = 8.92$ m (vacuum)

$\quad = (10.3 - 8.92) = 1.38$ m (abs)

At middle $\quad = (H_s + h_{fs}) = (5 + 0.628) = 5.628$ m (vacuum)

$\quad = (10.3 - 5.628) = 4.672$ m (abs)

At end $\quad = (H_s - H_{as}) = (5 - 3.92) = 1.08$ m (vacuum)

$\quad = (10.3 - 1.08) = 9.22$ m (abs)

Delivery Stroke:

At the ends of the strokes

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r$$

$$= \frac{39}{9.81}\left[\frac{\left(\frac{\pi}{4}\right)\times(0.15)^2}{\left(\frac{\pi}{4}\right)\times(0.075)^2}\right]\times\left(\frac{2\pi\times 30}{60}\right)^2\times 0.15$$

$$= 23.54 \text{ m of water}$$

At the middle of the stroke

$$h_{fd} = \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r\right)^2$$

$$= \frac{0.04\times 39}{2\times 9.81\times 0.075}\times\left[\frac{\left(\frac{\pi}{4}\right)\times(0.15)^2}{\left(\frac{\pi}{4}\right)\times(0.075)^2}\times\left(\frac{2\pi\times 30}{60}\right)\times 0.15\right]^2$$

$$= 3.767 \text{ m of water}$$

∴ Pressure heads in metres of water during suction stroke will be as follows:

At beginning $= (H_d + H_{ad}) = (33 + 23.54) = 56.54$ m (gage)

$= (10.3 + 56.54) = 66.84$ m (abs)

At middle $= (H_d + h_{fd}) = (33 + 3.767) = 36.767$ m (gage)

$= (10.3 + 36.767) = 47.067$ m (abs)

At end $= (H_d - H_{ad}) = (33 - 23.54) = 9.46$ m (gage)

$= (10.3 + 9.46) = 19.76$ m (abs)

Theoretical discharge of the pump

$$Q_{th} = \frac{ALN}{60}$$

$$= \frac{\pi}{4}(0.15)^2 \times (0.30) \times \left(\frac{30}{60}\right)$$

$$= 2.65 \times 10^{-3} \text{ m}^3/\text{s}$$

Work done by the pump

$$= wQ\left[H_s + H_d + \frac{2}{3}h_{fs} + \frac{2}{3}h_{fd}\right]$$

$$= (9810 \times 2.65 \times 10^{-3})\left[5 + 33 + \frac{2}{3}(0.628) + \frac{2}{3}(3.767)\right]$$

$$= 1\,064 \text{ N.m/s}$$

Power required to drive the pump

$$= 1064 \text{ W} = 1.064 \text{ kW}.$$

Example 23.6. *For a pump with no air vessel, stroke length = 300 mm, piston diameter = 125 mm, suction pipe diameter = 75 mm, length of suction pipe = 6 m and suction head = 3 m. Atmospheric pressure = 10.3 m of water and separation may be assumed to occur when the absolute pressure ... of water.*

Calculate the maximum speed at which the pump may be run if separation is to be avoided.

Solution

From Eq. 23.43, we have

$$H_{sep} = (H_s + H_{as})$$

$$H_{sep} = (10.3 - 2.5) = 7.8 \text{ m of water}$$

$$H_s = 3.0 \text{ m}$$

At the beginning of the suction stroke, we have

$$H_{as} = \frac{l_s}{g}\times\frac{A}{a_s}\times\omega^2 r$$

$$= \frac{6}{9.81}\times\left[\frac{(\pi/4)\times(0.125)^2}{(\pi/4)\times(0.075)^2}\right]\times\omega^2\times 0.15$$

$$= 0.255\ \omega^2$$

Thus by substitution, we have

$$7.8 = 3.0 + 0.255\ \omega^2$$

or

$$\omega = 4.34 \text{ rad/s}$$

Since

$$\omega = \frac{2\pi N}{60}$$

$$4.34 = \frac{2\pi N}{60}$$

∴

$$N = 41.4 \text{ r.p.m.}$$

Examples 23.7. *A single acting reciprocating pump has a plunger of 80 mm diameter and a stroke of length 150 mm. It takes its supply of water from a sump 3 m below the pump through a pipe 4.5 m long and 30 mm diameter. It delivers water to a tank 12 m above the pump through a pipe 25 mm diameter and 15 m long.*

If separation occurs at 78.48 kN/m² below atmospheric pressure, find the maximum speed at which the pump may be operated without separation, assume the plunger to have simple harmonic motion.

Solution

At the beginning of the suction stroke, we have

$$H_{as} = \frac{l_s}{g}\times\frac{A}{a_s}\times\omega^2 r$$

$$= \frac{4.5}{9.81}\times\left[\frac{(\pi/4)\times(0.08)^2}{(\pi/4)\times(0.03)^2}\right]\times\omega^2\times 0.075$$

$$= 0.245\ \omega^2$$

From Eq. 23.43, we have

$$H_{sep} = H_s + H_{as}$$

$$H_{sep} = \frac{78.48\times 10^3}{9810} = 8 \text{ m of water}$$

$$H_s = 3\text{ m}$$

Thus by substitution, we get

$$8 = 3 + 0.245\,\omega^2$$

or
$$\omega = \left(\frac{5}{0.245}\right)^{1/2} = 4.518\text{ rad/s}$$

Since
$$\omega = \frac{2\pi N}{60}$$

$$4.518 = \frac{2\pi N}{60}$$

$$\therefore \quad N = 43.14\text{ r.p.m.}$$

At the end of the delivery stroke, we have

$$H_{ad} = \frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r$$

$$= \frac{15}{9.81} \times \left[\frac{(\pi/4)\times(0.08)^2}{(\pi/4)\times(0.025)^2}\right] \times \omega^2 \times 0.075$$

$$= 1.174\,\omega^2$$

From the limiting condition, we have

$$H_{ad} - H_d = H_{sep}$$

or
$$1.174\,\omega^2 - 12 = 8$$

or
$$\omega = \left(\frac{20}{1.174}\right)^{1/2} = 4.127\text{ rad/s}$$

Since
$$\omega = \frac{2\pi N}{60}$$

$$4.127 = \frac{2\pi N}{60}$$

$$\therefore \quad N = 39.41\text{ r.p.m.}$$

Thus the maximum speed at which the pump may be operated without separation is the minimum of the two speeds obtained above viz., 39.41 r.p.m.

Example 23.8. *Determine the maximum speed at which a double acting reciprocating pump can be operated under the following conditions: (a) no air vessel on the suction side; (b) a very large air vessel on the suction side close to the pump.*

The suction lift is 4 m, length of suction pipe 6.5 m, diameter of suction pipe 100 mm, diameter of piston 150 mm and length of stroke is 0.45 m. Assume simple harmonic motion, atmospheric pressure head as 10.3 m of water and separation occurs at 2.6 m of water absolute. Take Darcy's $f = 0.024$.

Solution

(a) When no air vessel is fitted, we have from Eq. 23.43

$$H_{sep} = H_s + H_{as}$$

$$H_{sep} = (10.3 - 2.6) = 7.7\text{ m of water}$$

$$H_s = 4\text{ m}$$

$$H_{as} = \frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r$$

$$= \frac{6.5}{9.81} \times \left[\frac{(\pi/4)\times(0.15)^2}{(\pi/4)\times(0.10)^2}\right] \times \omega^2 \times \left(\frac{0.45}{2}\right)$$

$$= 0.335\,\omega^2$$

Thus by substitution, we get

$$7.7 = 4 + 0.335\,\omega^2$$

or
$$\omega = \left(\frac{3.7}{0.335}\right)^{1/2} = 3.323\text{ rad/s}$$

But
$$\omega = \frac{2\pi N}{60}$$

or
$$3.323 = \frac{2\pi N}{60}$$

$$\therefore \quad N = 31.73\text{ r.p.m.}$$

(b) With an air vessel fitted very close to the cylinder, we have from Eq. 23.71

$$H_{sep} = H_s + \frac{v_s^2}{2g} + (h_{fs})_1$$

$$H_{sep} = (10.3 - 2.6) = 7.7\text{ m of water}$$

$$H_s = 4\text{ m}$$

$$Q = \frac{2ALN}{60}$$

$$= 2 \times \frac{\pi}{4} \times (0.15)^2 \times 0.45 \times \left(\frac{N}{60}\right)$$

$$= (2.651 \times 10^{-4} N)$$

$$\therefore \quad v_s = \frac{Q}{a_s}$$

$$= \frac{(2.651\times10^{-4}N)}{(\pi/4)\times(0.10)^2} = (3.375 \times 10^{-2} N)$$

and
$$\frac{v_s^2}{2g} = \frac{(3.375\times10^{-2}N)^2}{2\times9.81} = (0.58 \times 10^{-4} N^2)$$

$$h_{fs1} = \frac{fl_s v_s^2}{2gd_s}$$

$$= \frac{0.024 \times 6.5 \times 0.58 \times 10^{-4} N^2}{0.10} = 0.905 \times 10^{-4} N^2$$

Thus by substitution, we get

$$7.7 = 4.0 + 0.58 \times 10^{-4} N^2 + 0.905 \times 10^{-4} N^2$$

or

$$N^2 = \frac{3.7}{1.485 \times 10^{-4}}$$

$$\therefore \quad N = 158 \text{ r.p.m.}$$

Note: It may be noted that by fitting an air vessel the speed of a pump may be increased considerably without causing the separation to occur.

Example 23.9. *A single acting reciprocating pump of 0.2 m diameter and 0.4 m stroke runs at 30 r.p.m. The suction and delivery heads are 3.6 m and 30 m respectively. The length of suction pipe is 9 m and that of delivery pipe is 60 m and the diameter of each pipe is 100 mm. Large air vessels are provided 2 m away from the pump on the suction side and 6 m away on the delivery side, both measured along the pipeline. Darcy's friction factor for the pipes is 0.032 and H_{atm}=10.3 m of water.*

(a) Determine the expressions for total absolute pressure heads H_{ts} and H_{td} in terms of θ.

(b) Calculate the work done per second and the power required to drive the pump.

(c) If the mean level of water in the vessel on delivery side is 1 m above the centreline of the pump, find the pressure of air in the air vessel in metres of water for $\theta = 0$.

Solution

(a) From Eq. 23.65, we have

$$H_{ts} = H_s + \frac{v_s^2}{2g} + (H_{as})_v + (h_{fs})_1 + (h_{fs})_2$$

$$H_s = 3.6 \text{ m}$$

$$v_s = \frac{ALN}{60a_s}$$

$$= \frac{(\pi/4)(0.2)^2 \times 0.4 \times 30}{60 \times (\pi/4)(0.1)^2} = 0.8 \text{ m/s}$$

$$\frac{v_s^2}{2g} = \frac{(0.8)^2}{2 \times 9.81} = 0.033 \text{ m}$$

$$(H_{as})_v = \frac{l_{sv}}{g} \times \frac{A}{a_s} \times (\omega^2 r \cos\theta)$$

$$= \frac{2}{9.81} \times \frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2} \times \left(\frac{2\pi \times 30}{60}\right)^2 \times 0.2 \cos\theta$$

$$= 1.61 \cos\theta$$

$$(h_{fs})_1 = \frac{fl_s v_s^2}{2gd_s} = \frac{0.032 \times (9-2) \times 0.033}{0.1} = 0.074 \text{ m}$$

$$(h_{fs})_2 = \left(\frac{fl_{sv}}{2gd_s}\right)\left(\frac{A}{a_s}\omega r \sin\theta\right)^2$$

$$= \frac{0.032 \times 2}{2 \times 9.81 \times 0.10}\left[\frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2} \times \left(\frac{2\pi \times 30}{60}\right) \times 0.2 \sin\theta\right]^2$$

$$= 0.206 \sin^2\theta$$

Thus by substitution, we have

$$H_{ts} = 3.6 + 0.033 + 1.61\cos\theta + 0.074 + 0.206\sin^2\theta$$

or

$$H_{ts} = 3.707 + 1.61\cos\theta + 0.206\sin^2\theta$$

From Eq. 23.67, we have

$$(H_{ts})_{\text{absolute}} = H_{atm} - H_{ts}$$

$$= 10.3 - [3.707 + 1.61\cos\theta + 0.206\sin^2\theta]$$

$$= 6.593 - 1.61\cos\theta - 0.206\sin^2\theta$$

Again from Eq. 23.58, we have

$$H_{td} = H_d + \frac{v_d^2}{2g} + (H_{ad})_v + (h_{fd})_1 + (h_{fd})_2$$

$$H_d = 30 \text{ m}; \; v_d = v_s = 0.8 \text{ m/s}$$

$$\frac{v_d^2}{2g} = \frac{(0.8)^2}{2 \times 9.81} = 0.033 \text{ m}$$

$$(H_{ad})_v = \frac{l_{dv}}{g} \times \frac{A}{a_d}(\omega^2 r \cos\theta)$$

$$= \frac{6}{9.81} \times \frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2} \times \left(\frac{2\pi \times 30}{60}\right)^2 \times 0.2\cos\theta$$

$$= 4.83\cos\theta$$

$$(H_{fd})_1 = \frac{fl_d v_d^2}{2gd_d} = \frac{0.032 \times (60-6) \times 0.033}{0.1} = 0.57 \text{ m}$$

$$(H_{fd})_2 = \left(\frac{fl_{dv}}{2gd_d}\right)\left(\frac{A}{a_d}\omega r \sin\theta\right)^2$$

$$= \frac{0.032 \times 6}{2 \times 9.81 \times 0.1}\left[\frac{(\pi/4)(0.2)^2}{(\pi/4)(0.1)^2} \times \left(\frac{2\pi \times 30}{60}\right) \times 0.2\sin\theta\right]^2$$

$$= 0.618\sin^2\theta$$

Thus by substitution, we have

$$H_{td} = 30 + 0.033 + 4.83\cos\theta + 0.57 + 0.618\sin^2\theta$$

or

$$H_{td} = 30.603 + 4.83\cos\theta + 0.618\sin^2\theta$$

From Eq. 23.59, we have

$$(H_{td})_{absolute} = H_{atm} + H_{td}$$
$$= 10.3 + 30.603 + 4.83\cos\theta + 0.618\sin^2\theta$$
$$= 40.903 + 4.83\cos\theta + 0.618\sin^2\theta$$

(b)

$$Q = \frac{ALN}{60}$$
$$= \frac{\pi}{4}(0.2)^2 \times 0.4 \times \left(\frac{30}{60}\right) = 0.006\,28 \text{ m}^3/\text{s}$$

From Eq. 23.69 the work done is given as

Work done $= 9\,810 \times (0.006\,28)\,[(3.6 + 30) + (0.074 + 0.57) + (0.033 + 0.033) + \frac{2}{3}(0.206 + 0.618)]$

$$= 2148 \text{ N.m/s}$$

$\therefore$ Power required to drive the pump

$$= 2148 \text{ W} = 2.148 \text{ kW}.$$

(c) At $\theta = 0$

$$(H_{td})_{absolute} = (40.903 + 4.83) = 45.733 \text{ m}$$

From Eq. 23.61, we have

$$h_{dv} + H_{dv} = H_{td} + H_{atm} = (H_{td})_{absolute}$$
$$\therefore \quad h_{dv} = (H_{td})_{absolute} - H_{dv}$$
$$= (45.733 - 1.0) = 44.733 \text{ m of water}$$

Example 23.10. *A double acting pump has a bore of 0.2 m and stroke of 0.4 m. The suction pipe has a diameter of 0.1 m and is fitted with an air vessel. Find the rate of flow into or from the air vessel when the crank makes angles of 30°, 90° and 120° with inner dead centre. Determine also the crank angles at which there is no flow to or from the air vessel. Take the speed as 120 r.p.m., and assume that the plunger has simple harmonic motion.*

Solution

From Eq. 23.54 the rate of flow of liquid into the air vessel

$$= A\omega r\left[\sin\theta - \frac{2}{\pi}\right]$$

$$A = \frac{\pi}{4} \times (0.2)^2 = 0.0314 \text{ m}^2$$

$$r = \frac{0.4}{2} = 0.2 \text{ m}$$

$$\omega = \frac{2\pi N}{60} = \frac{2\pi \times 120}{60} = 12.566 \text{ rad/s}$$

Thus rate of flow

$$= (0.0314 \times 12.566 \times 0.2)\left[\sin\theta - \frac{2}{\pi}\right]$$
$$= 0.079\left[\sin\theta - \frac{2}{\pi}\right]$$

For $\theta = 30°$

The rate of flow $= 0.079\left[\sin 30° - \frac{2}{\pi}\right]$

$$= -0.010\,8 \text{ m}^3/\text{s}$$

Since this is negative and we are considering the air vessel fitted to the suction pipe, then as indicated earlier the flow is taking place into the air vessel.

For $\theta = 90°$

The rate of flow $= 0.079\left[\sin 90° - \frac{2}{\pi}\right]$

$$= +0.0287 \text{ m}^3/\text{s}$$

Since this is positive and we are considering the air vessel fitted to the suction pipe, then as indicated earlier the flow is taking place from the air vessel.

For $\theta = 120°$

The rate of flow $= 0.079\left[\sin 120° - \frac{2}{\pi}\right]$

$$= +0.0181 \text{ m}^3/\text{s}$$

Since this is positive, the flow is taking place from the air vessel.

For no flow from or into air vessel

$$0.079\left[\sin\theta - \frac{2}{\pi}\right] = 0; \text{ or } \sin\theta = 0.636\,6$$

$$\theta = 39°\,32' \text{ and } 140°\,28'$$

Example 23.11. *A double acting single cylinder reciprocating pump of 20 cm bore and 40 cm stroke runs at 35 r.p.m. The suction and delivery lifts are respectively 1 m and 30 m. The piston rod is 7.5 cm in diameter. The suction pipe is 2.5 m long and the delivery pipe is 40 m long and both of them are 10 cm in diameter. No air vessel is provided either on the suction or delivery pipe. The local losses can be neglected and for both pipes the friction factor can be taken as 0.03. The motion of the piston can be assumed to be simple harmonic.*

Determine the net force due to fluid pressure on the piston when it has moved through a distance of 10 cm from the inner dead centre.

Solution

In this case when the piston moves from the inner dead centre, on one side of the piston there is suction stroke and on the other side of the piston there is delivery stroke.

During suction stroke the absolute pressure head on the piston for any position of the crank i.e., for any crank angle θ is given by Eq. 23.51 as

$$H_{ts} = H_{atm} - H_s - \left(\frac{l_s}{g} \times \frac{A}{a_s} \times \omega^2 r \cos\theta\right) - \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r \sin\theta\right)^2$$

$H_s = 1$ m; $A = \frac{\pi}{4}(0.20)^2$; $a_s = \frac{\pi}{4}(0.10)^2$; $\omega = \frac{2\pi \times 35}{60} = 3.67$ rad/s;

$r = \frac{40}{2} = 20$ cm $= 0.2$ m; $\cos\theta = \frac{20-10}{20} = \frac{1}{2}$, i.e., $\theta = 60°$;

$\sin\theta = \frac{\sqrt{3}}{2}$; $f = 0.03$; $l_s = 2.5$ m; $d_s = 10$ cm $= 0.1$ m

The atmospheric pressure is taken as 10.3 m of water.

Thus by substitution, we get

$$H_{ts} = 10.3 - 1 - \left[\frac{2.5}{9.81} \times \frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times (3.67)^2 \times 0.2 \times \frac{1}{2}\right]$$

$$-\left(\frac{0.03 \times 2.5}{2 \times 9.81 \times 0.10}\right)\left[\frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times 3.67 \times 0.2 \times \frac{\sqrt{3}}{2}\right]^2$$

or

$$H_{ts} = [10.3 - 1 - 1.373 - 0.247] = 7.68 \text{ m}$$

During delivery stroke the absolute pressure head on the piston for any position of the crank, i.e., for any crank angle θ is given by Eq. 23.52 as

$$H_{td} = H_{atm} + H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right) + \frac{fl_d}{2gd_d}\left(\frac{A}{a_d} \times \omega r \sin\theta\right)^2$$

$H_d = 30$ m; $A = \frac{\pi}{4}(0.20)^2$; $a_d = \frac{\pi}{4}(0.10)^2$; $\omega = 3.67$ rad/s;

$r = 0.2$ m; $\cos\theta = \frac{1}{2}$; $\sin\theta = \frac{\sqrt{3}}{2}$; $f = 0.03$; $l_d = 40$ m;

$d_d = 0.1$ m ; and $H_{atm} = 10.3$ m

Thus by substitution, we get

$$H_{td} = 10.3 + 30 + \left[\frac{40}{9.81} \times \frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times (3.67)^2 \times 0.2 \times \frac{1}{2}\right]$$

$$+\left(\frac{0.03 \times 40}{2 \times 9.81 \times 0.10}\right)\left[\frac{(\pi/4)(0.20)^2}{(\pi/4)(0.10)^2} \times 3.67 \times 0.2 \times \frac{\sqrt{3}}{2}\right]^2$$

$$= [10.3 + 30 + 21.968 + 3.954] = 66.222 \text{ m}$$

Total pressure acting on the piston on the side in which suction stroke is taking place is

$$P_1 = wH_{ts} \times A$$

$$= 9810 \times 7.68 \times \frac{\pi}{4} \times (0.20)^2 = 2367 \text{ N}$$

Total pressure acting on the piston on the side in which delivery stroke is taking place is

$$P_2 = wH_{td} \times (A - A_p)$$

$$= 9810 \times 66.222 \times \left[\frac{\pi}{4} \times (0.20)^2 - \frac{\pi}{4} \times (0.075)^2\right]$$

$$= 17539 \text{ N}$$

The net force due to fluid pressure acting on the piston is

$$P = (P_2 - P_1)$$

$$= (17539 - 2367) = 15172 \text{ N}$$

Example 23.12. *A single acting reciprocating pump works against a total static lift of 105 m of which 5 m is suction static lift, and when running at 30 r.p.m, it is required to force 45 litres per second of water along a delivery pipe, 25 cm diameter, 110 m long. There are no air vessels in the pump. Darcy's friction factor for the pipe is 0.024. The slip is 3%. The stroke of the plunger is twice the diameter. What would be the diameter of the cylinder and maximum and minimum pressures in the pipe when (i) one cylinder is used; and (ii) two cylinders are used in parallel.*

Solution

(i) *When one cylinder is used*

$$Q_{th} = \frac{ALN}{60}$$

Let D be the diameter of the cylinder, then

$$A = \frac{\pi D^2}{4}; L = 2D; \text{ and } N = 30 \text{ r.p.m}$$

$$\therefore \quad Q_{th} = \frac{(\pi/4)D^2 \times 2D \times 30}{60} = \frac{\pi D^3}{4}$$

$$Q_a = 45 \text{ l/s} = 0.045 \text{ m}^3\text{/s}$$

$$\text{Percentage slip} = \frac{Q_{th} - Q_a}{Q_{th}} \times 100$$

or

$$3 = \frac{Q_{th} - Q_a}{Q_{th}} \times 100$$

or

$$0.97\,Q_{th} = Q_a$$

or

$$0.97 \times \left(\frac{\pi D^3}{4}\right) = 0.045$$

or

$$D^3 = 0.05907$$

$$\therefore \quad D = 0.39 \text{ m} = 39 \text{ cm}$$

During delivery stroke the pressure head developed in the delivery pipe for any position of the crank, i.e., for any crank angle θ is given by the equation

$$H_{td} = H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right) + \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r \sin\theta\right)^2$$

$$H_d = 105 - 5 = 100 \text{ m}; l_d = 110 \text{ m}; \; A = \frac{\pi}{4} \times (0.39)^2;$$

$$a_d = \frac{\pi}{4} \times (0.25)^2; \; \omega = \frac{2\pi N}{60} = \frac{2 \times \pi \times 30}{60} = \pi \text{ rad/s};$$

$$r = \frac{L}{2} = \frac{2D}{2} = D = 0.39 \text{ m}; \; f = 0.024$$

Thus by substitution, we get

$$H_{td} = 100 + \left[\frac{110}{9.81} \times \frac{(\pi/4) \times (0.39)^2}{(\pi/4) \times (0.25)^2} \times (\pi)^2 \times 0.39 \cos\theta\right]$$

$$+ \left[\frac{0.024 \times 110}{2 \times 9.81 \times 0.25} \times \left\{\frac{(\pi/4) \times (0.39)^2}{(\pi/4) \times (0.25)^2} \times \pi \times 0.39 \sin\theta\right\}^2\right]$$

The maximum pressure will occur at the beginning of the delivery stroke for which θ = 0. Thus, we have

$$(H_{td})_{max} = (100 + 105.04 + 0) \text{ m}$$
$$= 205.04 \text{ m}$$
$$= 205.04 \times 9.81 \text{ kN/m}^2 = 2011.44 \text{ kN/m}^2$$

The minimum pressure will occur at the end of the delivery stroke for which θ = 180°. Thus, we have

$$(H_{td})_{min} = (100 - 105.04 - 0)$$
$$= -5.04 \text{ m}$$
$$= -5.04 \times 9.81 \text{ kN/m}^2 = -49.44 \text{ kN/m}^2$$

(ii) *When two cylinders are used in parallel*

$$Q_{th} = \frac{2ALN}{60}$$

Let D be the diameter of each cylinder, then

$$A = \frac{\pi D^2}{4}; \; L = 2D; \text{ and } N = 30 \text{ r.p.m.}$$

$$\therefore \quad Q_{th} = \frac{2 \times (\pi/4) \times D^2 \times 2D \times 30}{60} = \frac{\pi D^3}{2}$$

$$Q_a = 45 \text{ l/s} = 0.045 \text{ m}^3\text{/s}$$

$$\text{Percentage slip} = \frac{Q_{th} - Q_a}{Q_{th}} \times 100$$

or $$3 = \frac{Q_{th} - Q_a}{Q_{th}} \times 100$$

or $$0.97 \times Q_{th} = Q_a$$

or $$0.97 \times \frac{\pi D^3}{2} = 0.045$$

or $$D^3 = 0.0295$$

$$\therefore \quad D = 0.31 \text{ m} = 31 \text{ cm}$$

During delivery stroke the pressure head developed in the delivery pipe for any position of the crank, i.e., for any crank angle θ is given by the equation

$$H_{td} = H_d + \left(\frac{l_d}{g} \times \frac{A}{a_d} \times \omega^2 r \cos\theta\right) + \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r \sin\theta\right)^2$$

Thus by substitution, we get

$$H_{td} = 100 + \left[\frac{110}{9.81} \times \frac{(\pi/4) \times (0.31)^2}{(\pi/4) \times (0.25)^2} \times (\pi)^2 \times 0.31 \cos\theta\right]$$

$$+ \left[\frac{0.024 \times 110}{2 \times 9.81 \times 0.25} \times \left\{\frac{(\pi/4) \times (0.31)^2}{(\pi/4) \times (0.25)^2} \times \pi \times 0.31 \sin\theta\right\}^2\right]$$

The maximum pressure will occur at the beginning of the delivery stroke for which θ = 0. Thus, we have

$$(H_{td}) = (100 + 52.75 + 0) \text{ m}$$
$$= 152.75 \text{ m}$$
$$= 152.75 \times 9.81 \text{ kN/m}^2 = 1498.48 \text{ kN/m}^2$$

The minimum pressure will occur at the end of the delivery stroke for which θ = 180°. Thus, we have

$$(H_{td})_{min} = (100 - 52.75 - 0) \text{ m}$$
$$= 47.25 \text{ m}$$
$$= 47.25 \times 9.81 \text{ kN/m}^2 = 463.52 \text{ kN/m}^2$$

It may be noted that when one cylinder is used, negative or vacuum pressure is developed at the end of the delivery stroke. On the other hand when two cylinders are used in parallel, no negative or vacuum pressure is developed. In other words by using two cylinders development of negative or vacuum pressure is avoided.

SUMMARY OF MAIN POINTS

1. A reciprocating pump is a positive displacement pump which consists of a piston or plunger which moves to and fro in a close fitting cylinder which is connected to suction and delivery pipes each of which is provided with a non-return or one way value called suction valve and delivery valve respectively.

2. Theoretical discharge of the pump is given as

   (a) For single acting pump

$$Q_{th} = \frac{ALN}{60}$$

   (b) For a double acting pump

$$Q_{th} = \frac{(2A - A_p)LN}{60}$$

where

A = cross-sectional area of the piston;
A_p = area of cross-section of the piston rod;
L = length of the stroke; and
N = number of delivery strokes per minute.

3. Theoretical power required to drive the pump is given by

   (a) For a single acting pump

$$P = \frac{W(ALN)(H_s + H_d)}{60}$$

   (b) For a double acting pump

$$P = \frac{w(2A - A_p)LN(H_s + H_d)}{60}$$

where

H_s = suction head;
H_d = delivery head; and
w = specific weight of the liquid.

4. The difference between the theoretical discharge and the actual discharge is known as the ship of the pump. That is

$$\text{Slip} = (Q_{th} - Q_a)\text{ ; and}$$

$$\text{Percentage slip} = \frac{(Q_{th} - Q_a)}{Q_{th}} \times 100$$

In some cases the actual discharge of the pump may be more than the theoretical discharge, in which case the coefficient discharge C_d will be more than one and the slip will be negative.

5. The pressure head H_a due to acceleration of liquid in the suction and delivery pipes is given by

$$H_{as} = \frac{l_s}{g}\frac{A}{a_s}(\omega^2 r)\cos\theta$$

$$H_{ad} = \frac{l_d}{g}\frac{A}{a_d}(\omega^2 r)\cos\theta$$

6. The loss of head due to friction in suction and delivery pipes is given by

$$h_{f_s} = \frac{fl_s}{2gd_s}\left(\frac{A}{a_s}\omega r\sin\theta\right)$$

$$h_{f_d} = \frac{fl_d}{2gd_d}\left(\frac{A}{a_d}\omega r\sin\theta\right)$$

7. Mean velocity of flow is given by

   (a) For a single acting pump

$$V_d = \frac{A}{a_d}\frac{\omega r}{\pi}$$

   (b) For a double acting pump

$$V_d = 2\frac{A}{\alpha_d}\frac{\omega r}{\pi}$$

8. Indicator diagram is a graph between the pressure head in the cylinder and the distance travelled by the piston from the inner dead centre for one complete revolution of the crank.

9. The work done by the pump is proportional to the area of the indicator diagram. Area of ideal indicater diagram is same as the area of the indicator diagram modified due to acceleration of liquid in the suction and delivery pipes.

10. Work done by the pump per second is given by

   (a) For a single acting pump

$$\text{Work done} = \frac{wALN}{60}\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right]$$

   (b) For a double acting pump

$$\text{Work done} = \frac{\omega(2A - A_p)LN}{60}\left[H_s + H_d + \frac{2}{3}h_{f_s} + \frac{2}{3}h_{f_d}\right]$$

11. An air vessel is used to obtain a continuous supply of liquid at a uniform rate thereby save a considerable amount of work and run the pump at a high speed without separation.

12. Work done by single acting reciprocating pump fitted with air vessels is given by

$$\text{Work done} = \frac{wALN}{60}\left[H_s + H_d + h_{f_s} + h_{f_d}\right]$$

13. Work saved by fitting air vessels in a single acting reciprocating pump is 84.8%, while in a double acting reciprocating pum the work saved is 39.2%.

PROBLEMS

23.1 Why is a reciprocating pump called positive displacement pump?

23.2 Explain the term 'slip' and 'cavitation' with reference to reciprocating pumps.

23.3 Compare discharge curves for single and double acting reciprocating pumps and indicate under what conditions are either type used.

23.4 Explain the working principles of reciprocating pump with sketches.

23.5 Discuss the conditions under which 'cavitation' and 'negative slip' occur and state why air vessels are used in reciprocating pumps.

23.6 Explain with a neat sketch of its main parts, and a diagram of velocity v/s stroke, the working of a three throw pump. Is an air vessel necessary for this type of pump? Explain your answer.

23.7 The work saved by fitting an air vessel on a single acting reciprocating pump is much more than in a double acting reciprocating pump. Why?

23.8 Explain the functions of air vessels in a reciprocating pump.

23.9 Show that the maximum inertia head in a reciprocating pump without air vessel is given by

$$H_a = \frac{l}{g} \times \frac{A}{a}\omega^2 r$$

with usual notation.

23.10 What is an indicator diagram of a reciprocating pump? Sketch the theoretical indicator diagram for a single-acting reciprocating pump not fitted with an air vessel. With the help of the diagram explain clearly the effect of acceleration and friction on both suction and delivery strokes. How is this modified if air vessels are provided on both suction and delivery pipes?

23.11 Working from first principles show that, in a single acting pump the work done against pipe friction, if large air vessels are fitted near to the cylinder on both suction and delivery pipes, compared with that done if there are no vessels is. $(3/2\pi^2)$.

23.12 A single acting reciprocating pump has a plunger diameter 200 mm and stroke length 300 mm. It is to be driven with S.H.M., at 60 r.p.m., and draws water from a sump 3.6 m below the pump centreline. Find the least diameter of suction pipe, which is 4.5 m long in order to prevent separation at this speed. Take atmospheric pressure as 10.3 m of water, and the separation occurs at 2.6 m of water absolute. [Ans. 163 mm]

23.13 A single acting pump has a piston diameter of 0.15 m and a crank radius of 0.30 m. The delivery pipe is 0.10 m in diameter and 30 m long. The water is lifted 20 m above the centre of the pump. Find the maximum speed at which the pump can be run so that no cavitation takes place during the delivery stroke. Take H_{atm} = 10.3 m of water, and separation occurs at 2.6 m of water absolute. [Ans. 49.5 r.p.m.]

23.14 A single acting reciprocating pump has a plunger of diameter 0.3 m and a stroke of length 0.4 m. If the speed of the pump if 60 r.p.m., and coefficient of discharge is 0.97, determine the percentage slip and actual discharge of the pump. [Ans. 3%; 27.43 l/s]

23.15 A single acting reciprocating pump is to raise a liquid of specific weight 11.77 kN/m³ through a vertical height of 11.5 m, from 2.5 m below the pump axis to 9 m above it. The plunger moves with S.H.M., and has a diameter 125 mm and stroke length 225 mm. The suction and delivery pipes are 75 mm diameter, and 3.5 m and 13.5 m long respectively. There is a large air vessel placed on the delivery pipe near the pump axis but there is no air vessel on the suction pipe. If separation takes place at 88.3 kN/m² below the atmospheric pressure, find (a) the maximum speed at which the pump can run without separation taking place; (b) the power required to drive the pump if $f = 0.02$. Neglect slip for the pump.

[*Ans.* 64 r.p.m; 0.406 kW]

23.16 A double acting reciprocating pump has a bore of 150 mm and stroke of 300 mm. The suction pipe has a diameter of 100 mm and is fitted with an air vessel. Find the rate of flow into or from the air vessel when the crank makes angles of 30°, 90° and 120° with inner dead centre. The speed is 120 r.p.m., and the plunger has simple harmonic motion.

[*Ans.* 4.55 *l*/s (into air vessel); 12.10 *l*/s (from air vessel); 7.64 *l*/s (from air vessel)]

23.17 If in the Problems 23.16 the delivery pipe has a diameter of 100 mm and it is also fitted with an air vessel, then find the rate of flow into or from the air vessel when the crank makes angles of 30°, 90° and 120° with the outer dead centre and the speed of the pump is 120 r.p.m. The piston moves with S.H.M.

[*Ans.* 4.55 l/s (from air vessel); 12.10 l/s (into air vessel); 7.64 *l*/s(into air vessel)]

23.18 A single acting reciprocating pump runs at 60 r.p.m. The diameter of the plunger is 0.15 m and crank radius is 0.15 m. The suction pipe is 0.1 m in diameter and 5 m long. Calculate the maximum permissible value of suction lift H_s if the separation takes place at 2.6 m of water absolute.

[*Ans.* 0.91 m]

23.19 For a single acting reciprocating pump stroke length = 0.2 m, piston diameter = 0.1 m, suction head = 5 m, delivery head = 32 m, diameter of suction and delivery pipes = 50 mm, length of suction pipe = 6 m; length of delivery pipe = 35 m. Darcy's friction factor f for both pipes = 0.02. If the pump is working at 30 r.p.m., find the pressure head in metres of water on the piston at the beginning, middle and end of both suction and delivery strokes. Also find the power required to drive the pump. State if separation may take place at any point.

[*Ans.* 7.41 m, 5.19 m, 2.59 m of water (vacuum) (Suction) 46.09 m, 33.13 m, 17.92 m of water (gage) (Delivery); 0.292 kW; No possibility of separation]

23.20 If a single acting pump having diameter of plunger 150 mm and stroke length 225 mm, has for the suction side d_s = 75 mm, l_s = 6 m, H_s = 3 m, and for the delivery side d_d = 75 mm, l_d = 60 m, H_d = 27 m. Find the maximum possible speed at which separation of water could be avoided. Hence find the maximum theoretical discharge at this speed.

[*Ans.* 34 r.p.m.; 2.25 *l*/s]

23.21 A single acting reciprocating pump has a plunger 375 mm diameter and stroke length 600 mm. The delivery pipe is 90 m long and 150 mm diameter. Find the power saved by installing an air vessel near the pump, if the pump runs at 50 r.p.m. Take $f = 0.032$.

[*Ans.* 28.92 kW]

23.22 A single acting reciprocating pump running at 60 r.p.m., has a plunger diameter of 0.25 m and stroke length of 0.5 m. The delivery pipe is 0.1 m diameter and 50 m long. If the motion of the piston is simple harmonic, find the power required to overcome friction of the delivery pipe, when (a) no air vessel is fitted, (b) a large air vessel is fitted at the centreline of the pump.Take $f = 0.04$.

[*Ans.* (a) 15.77 kW; (b) 2.4 kW]

23.23 (a) Derive an expression for pressure head due to acceleration of the piston of a reciprocating pump, assuming motion of the piston to be S.H.M.

(b) Find the power required to drive a double acting pump and the quantity of water delivered per hour under the following conditions: head of water excluding friction = 16 m; diameter of main = 1 m; length of main = 2000 m; diameter of cylinder = 0.75 m; speed of pump = 24 r.p.m.; length of stroke = 1.5 m; positive slip = 4%; mechanical efficiency = 80%. Assume $f = 0.01$.

[*Ans.* 104.62 kW; 1832 m³/hr]

Centrifugal Pumps

Chapter 24

24.1 INTRODUCTION

Centrifugal pumps are classified as rotodynamic type of pumps in which a dynamic pressure is developed which enables the lifting of liquids from a lower to a higher level. The basic principle on which a centrifugal pump works is that when a certain mass of liquid is made to rotate by an external force, it is thrown away from the central axis of rotation and a centrifugal head is impressed which enables it to rise to a higher level. Now if more liquid is constantly made available at the centre of rotation, a continuous supply of liquid at a higher level may be ensured. Since in these pumps the lifting of the liquid is due to centrifugal action, these pumps are called 'centrifugal pumps'. In addition to the centrifugal action, as the liquid passes through the revolving wheel or impeller, its angular momentum changes, which also results in increasing the pressure of the liquid. As such centrifugal pumps behave quite differently from positive displacement pumps. A centrifugal pump does not push the liquid as in the case of a positive displacement pump, but it modifies the hydraulic gradient such that the liquid is lifted to a higher level.

According to the general direction of flow of liquid within the passage of the rotating wheel or impeller the rotodynamic pumps are classified as,

(i) Centrifugal pumps,

(ii) Half axial or screw or mixed flow pumps,

(iii) Axial flow or propeller pumps.

In the impeller of a centrifugal pump the liquid flows in the outward radial direction, while the flow of liquid in a propeller pump impeller is in the axial direction, parallel to the rotating shaft. The mixed flow pump impeller has an intermediate form so that the flow of liquid is in between the radial and axial directions. However, there are no rigid boundaries separating these three types of pumps, and often all the three types of pumps are called centrifugal pumps.

In general all the rotodynamic pumps closely resemble reaction type of hydraulic turbines and they may be regarded as reversed reaction turbines. Thus the action of a centrifugal pump is just the reverse of a radially inward flow reaction turbine. Similarly the axial flow pumps are reverse of propeller or Kaplan turbines and the mixed flow pumps are the reverse of mixed flow type turbines such as Francis turbine. In the present chapter only centrifugal pumps have been described.

24.2 ADVANTAGES OF CENTRIFUGAL PUMPS OVER RECIPROCATING PUMPS

The main advantage of a centrifugal pump is that its discharging capacity is very much greater than that of a reciprocating pump which can handle relatively small quantity of liquid only. A centrifugal pump can be used for lifting highly viscous liquids such as oils, muddy and sewage water, paper pulp, sugar molasses, chemicals etc. But a reciprocating pump can handle only pure water or less viscous liquids free from impurities as otherwise its valves may cause frequent trouble. A centrifugal pump can be operated at very high speeds without any danger of separation and cavitation. As such it can be coupled directly through flanged coupling to electric motor. On the other hand as explained in Chapter 23 the maximum speed of a reciprocating pump is limited from the considerations of separation and cavitation. As such reciprocating pumps can be operated at low speeds only and for that these pumps are mostly belt driven. The maintenance cost of a centrifugal pump is low and only periodical check up is sufficient. But for a reciprocating pump the maintenance cost is high because the parts such as valves etc. may need frequent replacement. However, a reciprocating pump can build up very high pressures as high as 69×10^6 N/m^2 [700 kg(f)/cm^2] or even more and hence these pumps are used for lifting oil from very deep oil wells.

24.3 COMPONENT PARTS OF A CENTRIFUGAL PUMP

Figure 24.1 shows the main component parts of a centrifugal pump which are described below:

1. Impeller. It is a wheel or rotor which is provided with a series of backward curved blades or vanes. It is mounted on a shaft which is coupled to an external source of energy (usually an electric motor) which imparts the required energy to the impeller thereby making it to rotate.

The impellers may be classified as (a) closed or shrouded impeller, (b) semi-open impeller; and (c) open impeller, which are all shown in Fig. 24.1. A 'closed or shrouded impeller' is that whose vanes are provided with metal cover plates or shrouds on both sides. These plates or shrouds are known as crown plate and lower or base plate as shown in Fig. 24.1. The closed impeller provides better guidance for the liquid and is more efficient. However, this type of impeller is most suited when the liquid to be pumped is pure and comparatively free from debris.

If the vanes have only the base plate and no crown plate, then the impeller is known as 'semi-open type impeller'. Such an impeller is suitable even if the liquids are charged with some debris.

An 'open impeller' is that whose vanes have neither the crown plate nor the base plate. Such impellers are useful in the pumping of liquids containing suspended solid matter, such as paper pulp, sewage and water containing sand or grit. These impellers are less liable to clog when handling liquids charged with a large quantity of debris.

2. Casing. It is an airtight chamber which surrounds the impeller. It is similar to the casing of a reaction turbine. The different types of casings that are commonly adopted are described later.

3. Suction Pipe. It is a pipe which is connected at its upper end to the inlet of the pump or to the centre of the impeller which is commonly known as eye. The lower end of the suction pipe dips into liquid in a suction tank or a sump from which the liquid is to be pumped or lifted up.

The lower end of the suction pipe is fitted with a *foot valve and strainer*. The liquid first enters the strainer which is provided in order to keep the debris (such as leaves, wooden pieces and other rubbish) away from the pump. It then passes through the foot valve to enter the suction pipe. A 'foot valve' is a non-return or one-way type of valve which opens only in the upward direction. As such the liquid will pass through the foot valve only upwards and it will not allow the liquid to move downwards back to the sump.

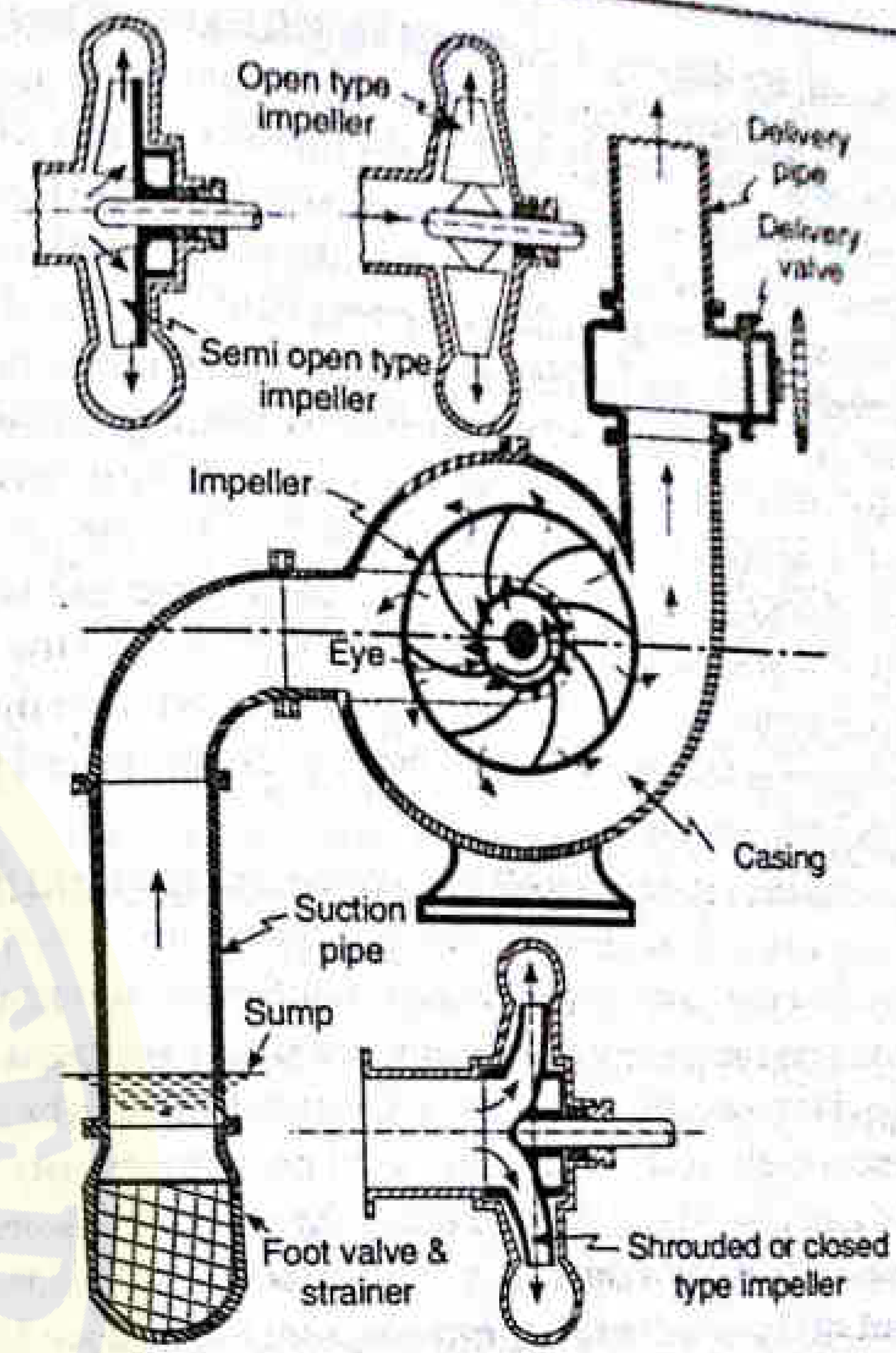


Figure 24.1 *Component parts of a centrifugal pump*

4. Delivery Pipe. It is a pipe which is connected at its lower end to the outlet of the pump and it delivers the liquid to the required height. Just near the outlet of the pump on the delivery pipe a delivery valve is invariably provided. A delivery valve is a regulating valve which is of sluice type and is required to be provided in order to control the flow from the pump into delivery pipe.

24.4 WORKING OF CENTRIFUGAL PUMP

The first-step in the operation of a centrifugal pump is priming. *Priming* is the operation in which the suction pipe, casing of the pump and the portion of the delivery pipe upto the delivery valve are completely filled with the liquid which is to be pumped, so that all the air (or gas or vapour) from this portion of the pump is driven out and no air pocket is left. It has been observed that even the presence of a small air pocket in any of the portion of pump may result in no delivery of liquid from the pump. The necessity of priming a centrifugal pump is due to the fact that the pressure generated in a centrifugal pump impeller is directly proportional to the density of the fluid that is in contact with it. Hence if an impeller is made to rotate in the presence of air, only a negligible pressure would be produced with the result that no liquid will be lifted up by the pump. As such it is essential to properly prime a centrifugal pump before it can be started. The various methods used for priming a centrifugal pump are discussed later.

After the pump is primed, the delivery valve is still kept closed and the electric motor is started to rotate the impeller. The delivery valve is kept closed in order to reduce the starting torque for the motor. The rotation of the impeller in the casing full of liquid produces a forced vortex which imparts a centrifugal head to the liquid and thus results in an increase of pressure throughout the liquid mass. The increase of pressure at any point is proportional to the square of the angular velocity and the distance of the point from the axis of rotation. Thus if the speed of rotation of the impeller of the pump is sufficiently high, the pressure in the liquid surrounding the impeller is considerably increased. Now as long the delivery valve is closed and the impeller is rotating, it just churns the liquid in the casing. When the delivery valve is opened the liquid is made to flow in an outward radial direction thereby leaving the vanes of the impeller at the outer circumference with high velocity and pressure. At the eye of the impeller due to the centrifugal action a partial vacuum is created. This causes the liquid from the sump, which is at atmospheric pressure, to rush through the suction pipe to the eye of the impeller thereby replacing the liquid which is being discharged from the entire circumference of the impeller. The high pressure of the liquid leaving the impeller is utilized in lifting the liquid to the required height through the delivery pipe.

As the liquid flows through the rotating impeller it receives energy from the vanes which results in an increase in both pressure and velocity energy. As such the liquid leaves the impeller with a high absolute velocity. In order that the kinetic energy corresponding to the high velocity of the leaving liquid is not wasted in eddies and efficiency of the pump thereby lowered, it is essential that this high velocity of the leaving liquid is gradually reduced to a lower velocity of the delivery pipe, so that the larger portion of the kinetic energy is converted into useful pressure energy. Usually this is achieved by shaping the casing such that the leaving liquid flows through a passage of gradually expanded area. The gradually increased cross-sectional area of the casing also helps in maintaining uniform velocity of flow throughout, because as the flow proceeds from the tongue T (Fig. 24.2) to the delivery pipe, more and more liquid is added from the impeller. There are different types of casings that are adopted for this purpose and on the basis of the type of casing used, the centrifugal pumps are classified into different types as described in the next section.

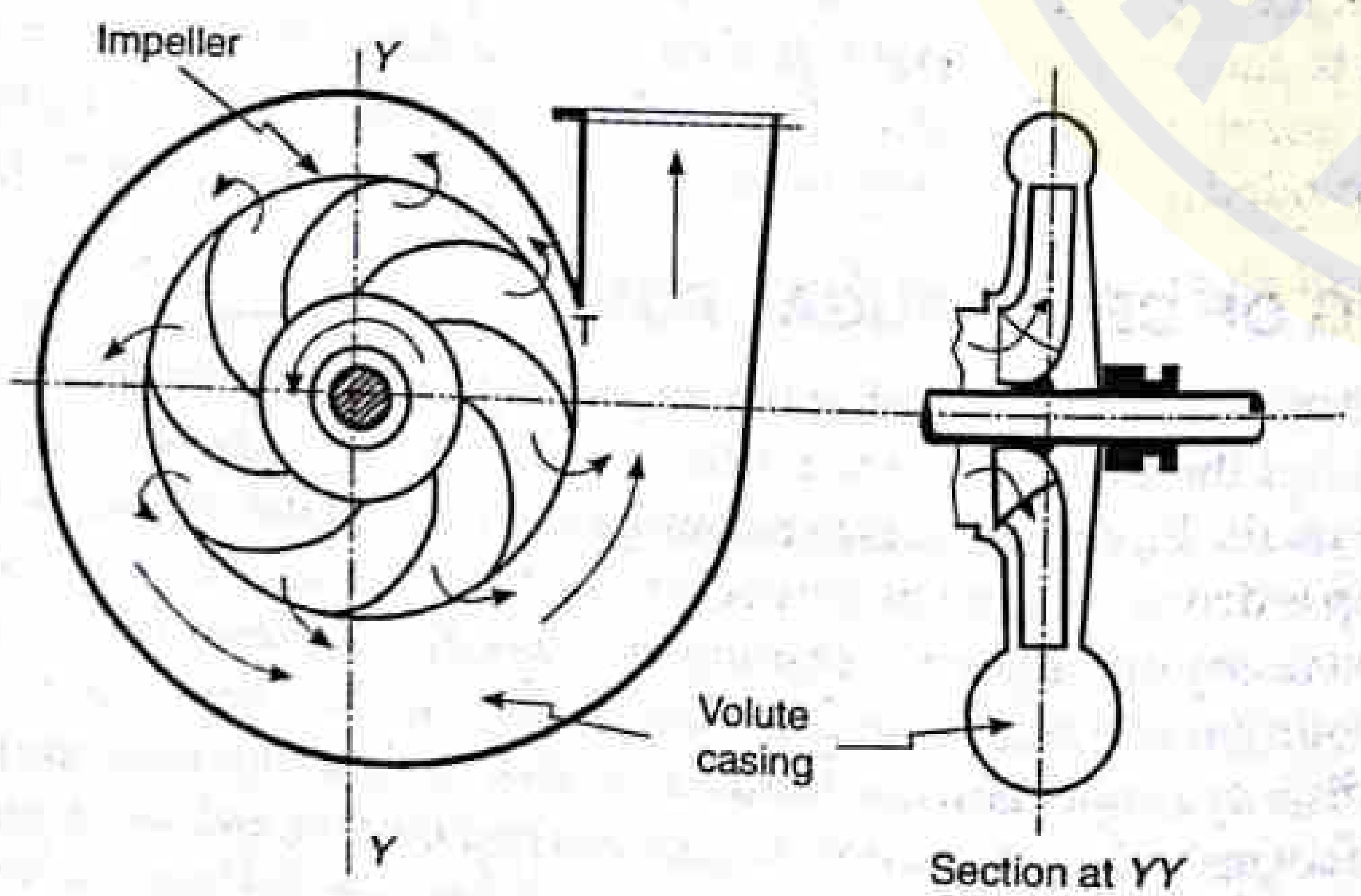


Figure 24.2 *Volute pump*

24.5 TYPES OF CENTRIFUGAL PUMPS

According to the type of casing provided, centrifugal pumps are classified into the following two classes:

(1) Volute pump.
(2) Diffuser or turbine pump.

1. Volute Pump. In a volute pump the impeller is surrounded by a spiral shaped casing which is known as *volute chamber*. As shown in Fig. 24.2 the shape of the casing is such that the sectional area of flow around the periphery of the impeller gradually increases from the tongue T towards the delivery pipe. This increase in the cross-sectional area results in developing a uniform velocity throughout the casing, because as the flow progresses from the tongue T towards the delivery pipe, more and more liquid is added to the stream from the periphery of the impeller. The volute casing may be designed to have the velocity of flow approximately equal to that of the liquid leaving the impeller. If the casing is designed according to this consideration then the loss of energy is considerably reduced, but the conversion of kinetic energy into useful pressure energy will not be possible. On the other hand if the casing is so designed that the casing velocity may be kept down to the value of the velocity in the delivery pipe, then there will be considerable loss of energy due to the difference between the casing velocity and that of the liquid discharged from the impeller. As such a compromise design is often used in which the casing is gradually enlarged so that the velocity is gradually reduced, from the velocity of the liquid leaving the impeller to that in the delivery pipe.

However, in actual practice it has been found that in volute type of casing there is only a slight increase in the efficiency of the pump, because a considerable loss of energy takes place in eddies developed in the casing.

A subsequent improvement over the simple volute casing was made by J. Thomson by providing a circular chamber between the impeller and the volute chamber as shown in Fig. 24.3. The circular chamber is known as *vortex* or *whirlpool chamber* and such a pump is known as *volute pump with vortex chamber*.

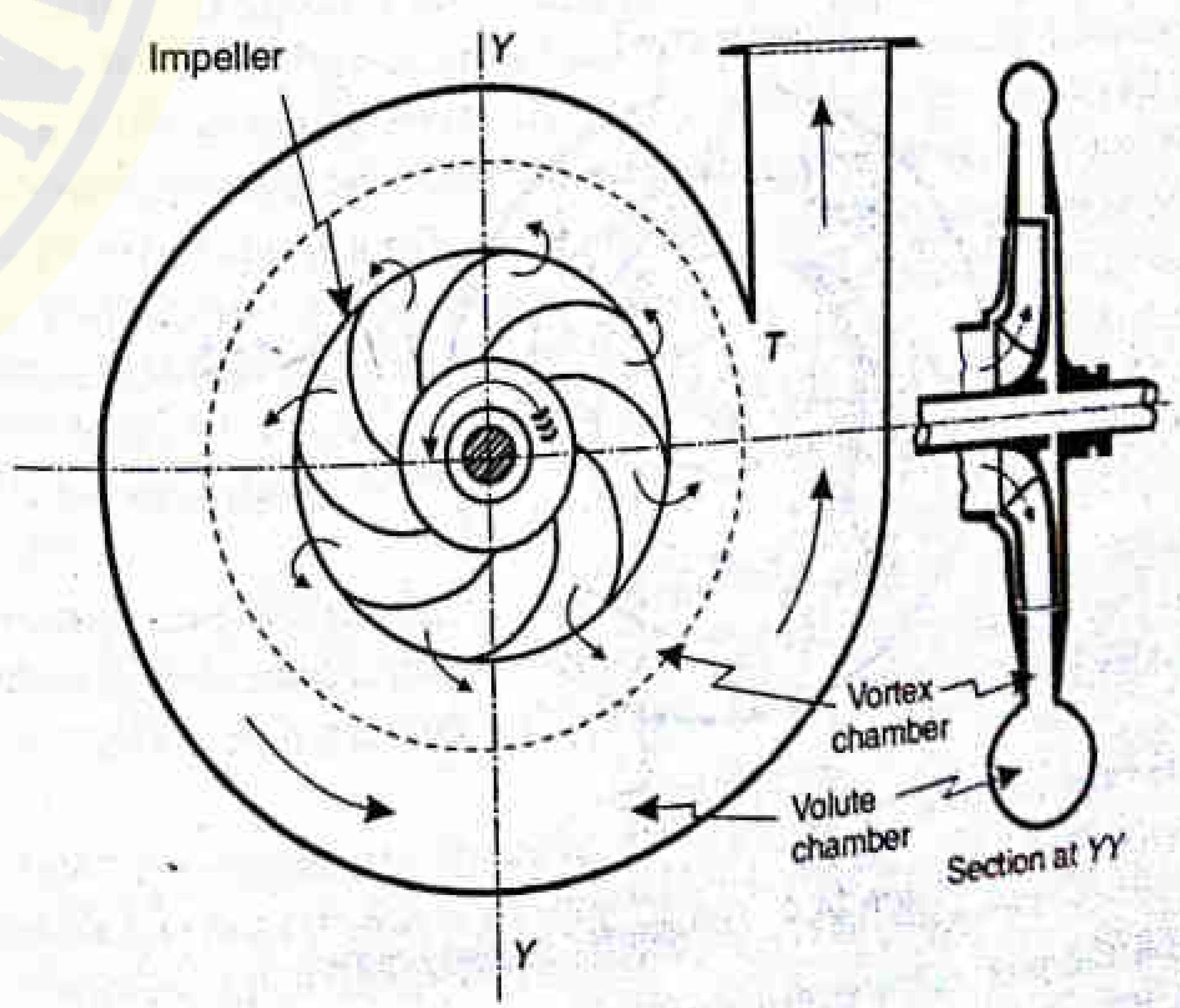


Figure 24.3 *Volute pump with vortex chamber*

The vortex chamber is usually formed as a part of the casing with its side walls parallel or nearly parallel as shown in Fig. 24.3. It acts as a diffuser wherein the conversion of kinetic energy into pressure energy takes place as explained below. The liquid after leaving the impeller enters the vortex chamber with a whirling motion, that is the liquid particles move radially away from the centre following a rotary path while passing through this chamber. Since no work is done on the liquid as it passes through this chamber, its energy remains constant (except for the slight loss by friction). Therefore the torque produced for the liquid does not change and hence a free vortex is formed as the liquid passes through the vortex chamber. Since for a free vortex the velocity of whirl varies inversely as its radial distance from the centre, there is a reduction in velocity of flow of liquid as it passes through the vortex chamber. The reduction in velocity is accompanied by an increase in pressure. As such a vortex chamber serves a dual purpose of reducing the velocity and increasing the efficiency of the pump by converting a large amount of kinetic energy into pressure energy. The liquid after leaving the vortex chamber passes through the volute chamber surrounding it, which further increases the efficiency of the pump.

2. Diffuser or Turbine Pump. In the diffuser pump, the impeller is surrounded by a series of guide vanes mounted on a ring called diffuser ring as shown in Fig. 24.4. The diffuser ring and the guide vanes are fixed in position. The adjacent guide vanes provide gradually enlarged passages for the flow of liquid. The liquid after leaving the impeller passes through these passages of increasing area, wherein the velocity of flow decreases and the pressure increases. The guide vanes are so designed that the liquid emerging from the impeller enters these passages without shock. This condition may however be achieved by making the tangent to the guide vane at the inlet tip to coincide with the direction of the absolute velocity of liquid leaving the impeller. After passing through the guide vanes the liquid flows into the surrounding casing which may be circular, and concentric with the impeller or it may be volute-shaped like that of volute pump. However, the common practice is to adopt circular casings for these pumps.

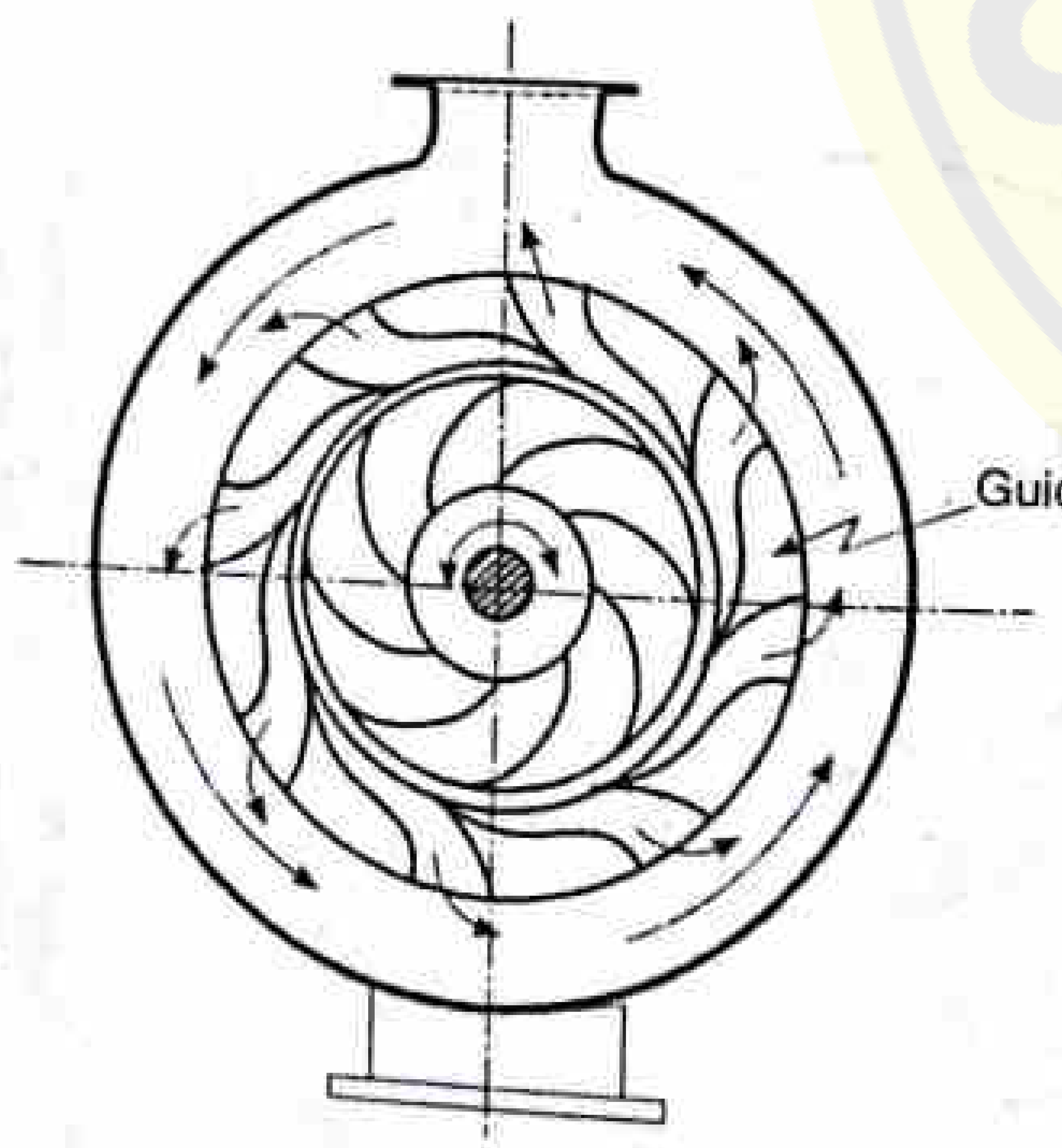


Figure 24.4 *Diffuser (or turbine) pump*

These pumps which are provided with diffuser ring and guide vanes very much resemble a reversed turbine and hence they are also known as *turbine pumps*. It has been found from tests that a well designed diffuser pump is capable of converting as much as 75 percent of the kinetic energy of the liquid discharged from the impeller into pressure energy. However these pumps will work with maximum efficiency only for one rate of discharge at given impeller speed. This is so because the guide vanes will be correctly set or shaped for one rate of discharge only and for other discharges a loss of energy by shock or turbulence will occur at the entrance to the guide vanes, thereby resulting in a low efficiency. Moreover turbine pumps are more costly than the simple volute pumps. As such the arrangement of diffuser ring is usually employed only in multistage pumps.

The centrifugal pumps may also be classified on the basis of certain other factors as indicated below:

(a) Number of impellers per shaft.
(b) Relative direction of flow of liquid through impeller.
(c) Number of entrances to the impeller.
(d) Disposition of shaft, and
(e) Working head.

According to the number of impellers provided the pumps may be classified as single-stage and multi-stage. A *single stage centrifugal pump* has only one impeller mounted on the shaft. A *multi-stage centrifugal pump* has two or more impellers connected in series, which are mounted on the same shaft and are enclosed in the same casing.

On the basis of the direction of flow of the liquid through the impeller the pump may be classified as radial flow pump, mixed flow pump and axial flow pump.

A *radial flow pump* is that in which the liquid flows through the impeller in the radial direction only. Ordinarily all the centrifugal pumps are provided with radial flow impellers. In *mixed flow pumps* the liquid flows through the impeller axially as well as radially, that is there is a combination of radial and axial flows. A mixed flow impeller is just a modification of radial flow type in this respect that the former is capable of discharging a large quantity of liquid. As such mixed flow pumps are generally used where a large quantity of liquid is to be dicharged to low heights. In *axial flow pumps* the flow of liquid through the impeller is in the axial direction only. Axial flow pumps are usually designed to deliver very large quantities of liquid at relatively low heads. However, it is not justified to call axial flow pumps as centrifugal pumps, because there is hardly any centrifugal action in their operation.

Depending on the number of entrances to the impeller the centrifugal pumps may be classified as single suction pump and double suction pump. In a *single suction (or entry) pump* liquid is admitted from a suction pipe on one side of the impeller. In a *double suction (or entry) pump* liquid enters from both sides of the impeller. A double suction pump has an advantage that by this arrangement the axial thrust on the impeller is neutralised. Further it is suitable for pumping large quantities of liquid since it provides a large inlet area.

The centrifugal pumps may be designed with either horizontal or vertical disposition of shafts. Generally the pumps are provided with horizontal shafts. However, for deep wells and mines the pumps with vertical shafts are more suitable because the pumps with vertically disposed shafts occupy less space.

According to the head developed, the centrifugal pumps may be classified as *low head*, *medium head* and *high head* pumps. A *low head* pump is the one which is capable of working against a total head upto 15 m. A *medium head* pump is that which is capable of working against a total head more than 15 m but upto 40 m. A *high head* pump is the one which is capable of working against a total head above 40 m. Generally high head pumps are multi-stage pumps.

24.6 WORK DONE BY THE IMPELLER

The expression for the work done (or the energy supplied) by the impeller of a centrifugal pump on the liquid flowing through it may be derived in the same way as for a turbine. The liquid enters the impeller at its centre and leaves at its outer periphery. Figure 24.5 shows a portion of the impeller of a Centrifugal pump with one vane and the velocity triangles at the inlet and the outlet tips of the vane. For the sake of convenience the same system of notation is employed as that for turbines. Thus, V is absolute velocity of liquid, u is peripheral (or tangential) velocity of the impeller V_r is relative velocity of liquid, V_f is velocity of flow of liquid, and V_w is velocity of whirl of the liquid at the entrance to the impeller. Similarly V_1, u_1 V_{r1}, V_{f1} and V_{w1} represent their counterparts at the exit point of the impeller. Further θ represents the impeller vane angle at the entrance and ϕ represents the impeller vane angle at the outlet. Similarly α is the angle between the direction of the absolute velocity of entering liquid and the peripheral velocity of the impeller at the entrance, and β is the angle between the absolute velocity of leaving liquid and the peripheral velocity of the impeller at the exit point.

At the entrance to the impeller since there are no guide vanes (as in the case of turbines) the direction of the absolute velocity of liquid at this point of the impeller is not directly known. However, for best efficiency of the pump it is commonly assumed that the liquid enters the impeller radially that is the absolute velocity of the liquid at the entrance to the impeller (or at the inlet tip of the impeller vane) is radial in direction. Thus, in this case $\alpha = 90°$ and the velocity of whirl V_w at inlet is equal to zero. Further it is desired that the liquid enters and leaves the vane without shock. This can be ensured if the inlet and outlet tips of the vane are parallel to the direction of the relative velocities at the two tips. As such it is assumed that the relative velocities V_r and V_{r1} are parallel to the tangents to the vane at the inlet and outlet tips respectively as shown in Fig. 24.5.

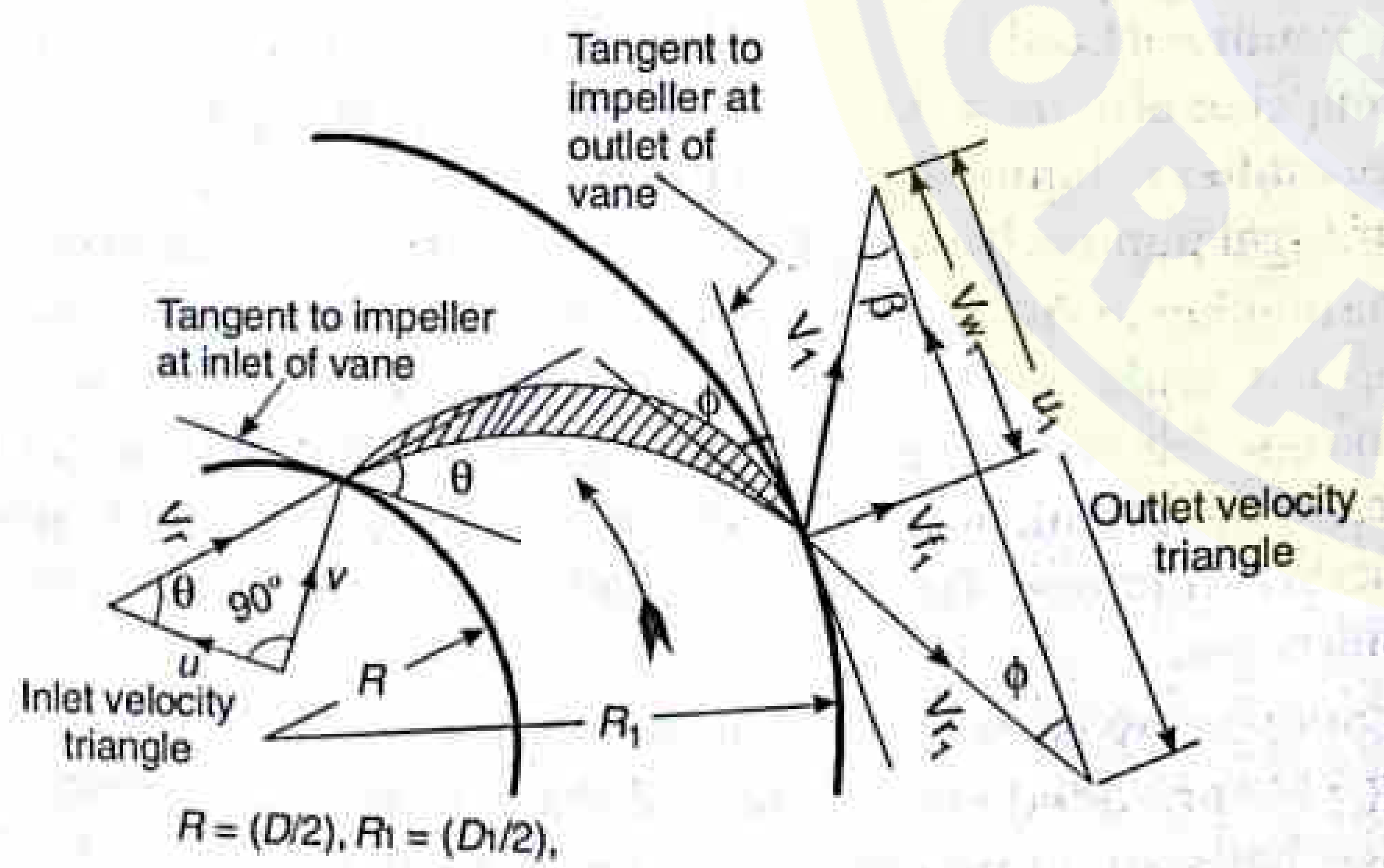


Figure 24.5 *Velocity triangles for an impeller vane*

In the case of a radially inward flow reaction turbine the work done per second by the liquid on the runner = $(W/g)\,(V_w u \pm V_{w1} u_1)$; where W is weight of liquid striking the runner per second. Since a centrifugal pump is just the reverse of a radially inward flow reaction turbine, the same analysis as used for turbines may be applied to pumps. Thus, the work done per second by the impeller on the liquid may be written as

$$\text{Work done} = \frac{W}{g}(V_{w1}u_1 - V_w u) \qquad ...(24.1)$$

where W is the weight of liquid per second that passes through the impeller. Since in this case as stated earlier the liquid enters the impeller radially, $\alpha = 90°$ and hence $V_w = 0$.

Thus Eq. 24.1 becomes

$$\text{Work done} = \frac{W}{g}(V_{w1}u_1) \qquad ...(24.2)$$

and work done per unit weight of liquid

$$= \frac{1}{g}(V_{w1}u_1) \qquad ...(24.3)$$

Thus Eq. 24.3 represents the head imparted by the impeller to the liquid.

Further from the outlet velocity triangle of Fig. 24.5, $V_{w1} = (u_1 - V_{f1}\cot\phi)$. As in the case of turbines u_1 and V_{f1} can be expressed in terms of the speed ratio K_u and the flow ratio ψ as follows:

$$u_1 = K_u\left(\sqrt{2gH_m}\right) \text{ and } V_{f1} = \psi\left(\sqrt{2gH_m}\right)$$

where H_m is the total (or manometric) head as defined later. The usual range of values for the speed ratio and the flow ratio of impellers are given in Table 24.1.

TABLE 24.1

	Low N_s Impeller		High N_s Impeller
Speed ratio K_u	0.95	to	1.25
Flow ratio ψ	0.10	to	0.25

Moreover, Eq. 24.1 can be transformed in the following form:

Work done per kg per second

$$= \left(\frac{V_1^2 - V^2}{2g} + \frac{u_1^2 - u^2}{2g} + \frac{V_r^2 - V_{r1}^2}{2g}\right) \qquad ...(24.4)$$

Equation 24.4 shows that work done on the liquid consists of three parts. The first part represents the change in kinetic energy of the liquid, the second part represents effect of centrifugal head and the last part indicates the change in static pressure energy of the liquid if the losses in the impeller and the effect of difference in elevations of the inlet and the outlet points of the impeller are neglected. Usually Eq. 24.4 is known as the *fundamental equation* of centrifugal pump.

24.7 HEAD OF PUMP

The head of a centrifugal pump may be expressed in the following two ways:

(a) Static head,

(b) Manometric head (or total head or gross head or effective head).

(a) Static Head. As shown in Fig. 24.6 the static head is the vertical distance between the liquid surfaces in the sump and the tank to which the liquid is delivered by the pump. Thus if h_s is the vertical height of the centre line of the pump shaft above the liquid surface in the sump from which the liquid is being raised; and h_d is the vertical height of the liquid surface in the tank to which the liquid is delivered above the centre line of the pump shaft, then the static head (or lift) H_s may be expressed as

$$H_s = h_s + h_d \quad ...(24.5)$$

The term h_s is known as static suction lift, and h_d is known as static delivery lift. Thus static head (or lift) is the net total vertical height through which the liquid is lifted by the pump.

(b) Manometric Head. It is the total head that must be produced by the pump to satisfy the external requirements. If there are no energy losses in the impeller and the casing of the pump, then the manometric head H_m will be equal to the energy given to the liquid by the impeller, which for radial entry to the impeller $= (V_{w1}u_1/g)$ N-m/N. But if the losses occur in the pump then

$$H_m = \frac{V_{w1}u_1}{g} - \text{(losses of head in the pump)} \quad ...(24.6)$$

The various losses of head that may occur in the pump have been described in the next section.

The manometric head may also be expressed in other forms which are as indicated below:

Applying Bernoulli's equation between the points, O at the liquid surface in the sump and 1 in the suction pipe just at the inlet to the pump (i.e., at the centre line of the pump), the following expression is obtained if the liquid surface in the sump is taken as datum.

$$O = \frac{p_s}{w} + \frac{V_s^2}{2g} + h_s + h_{fs}$$

or

$$\frac{p_s}{w} = -\left(\frac{V_s^2}{2g} + h_s + h_{fs}\right) \quad ...(24.7)$$

where p_s is the pressure at point 1; V_s is the velocity of flow in the suction pipe; h_s is the suction lift and h_{fs} is the head loss in the suction pipe which includes the head loss due to friction and the inlet and the other minor losses. It may however be pointed out that if the pump is situated below the level of the liquid surface in the sump, h_s will be negative. Equation 24.7 indicates that at the inlet to the pump there is always a suction or vacuum pressure developed which will be recorded by the vacuum gage provided at this point as shown in Fig. 24.6. The head expressed by Eq. 24.7 is called the *suction head* of the pump.

Also, applying Bernoulli's equation between points 1 and 2, which is just at the outlet of the impeller and is assumed to be at the same level as point 1, then since the impeller imparts a head equal to $(V_{w1}u_1/g)$ to the liquid the following expression is obtained.

$$\frac{p_s}{w} + \frac{V_s^2}{2g} + \frac{V_{w1}u_1}{g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_{Li} \quad ...(24.8)$$

where p_2 is the pressure and V_1 is the absolute velocity of the liquid leaving the impeller and h_{Li} is the loss of head in the impeller.

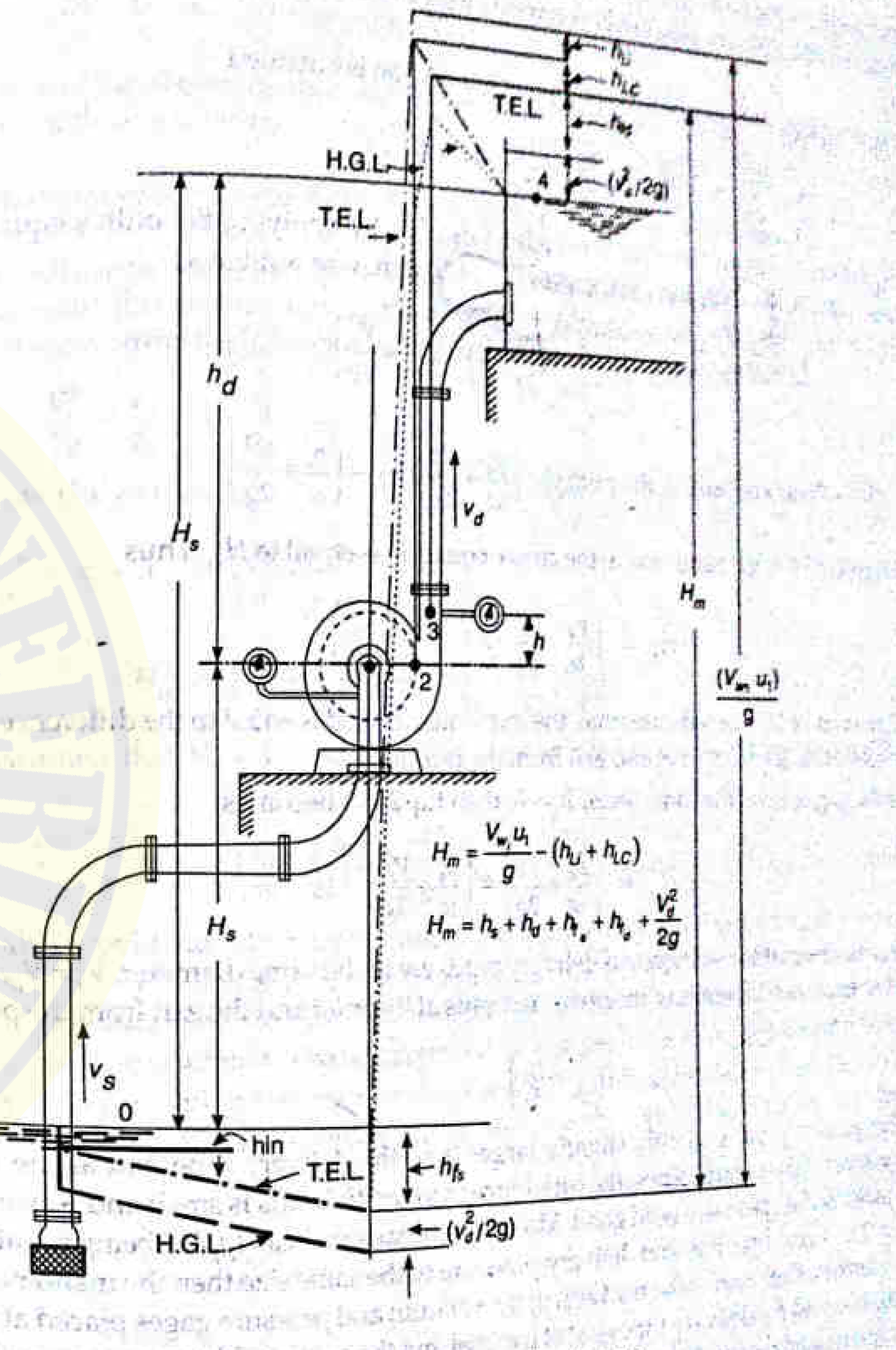


Figure 24.6 *Head on a centrifugal pump*

Further applying Bernoulli's equation between points 2 and 3, which is [as shown in Fig. 24.6] lying in the delivery pipe just at the exit from the pump, the following expression is obtained.

$$\frac{p_2}{w} + \frac{V_1^2}{2g} = \frac{p_d}{w} + \frac{V_d^2}{2g} + h + h_{Lc} \quad ...(24.9)$$

where p_d is the pressure in the delivery pipe at point 3; V_d is the velocity of flow in the delivery pipe; h

is the height of the point 3 above the centre line of the pump and h_{lc} is the loss of head in the casing. The pressure in the delivery pipe can be recorded by the pressure gage provided at point 3, as shown in Fig. 24.6.

Adding Eqs. 24.8 and 24.9 the following expression is obtained

$$\frac{p_s}{w}+\frac{V_s^2}{2g}+\frac{V_{w1}u_1}{g} = \frac{p_d}{w}+\frac{V_d^2}{2g}+h+h_{li}+h_{lc} \quad ...(24.10)$$

Equation 24.10 can however be obtained directly by applying Benoulli's equation between the points 1 and 3. Rearranging the terms of Eq. 24.10 it may be written as

$$\frac{V_{w1}u_1}{g}-(h_{li}+h_{lc}) = \left(\frac{p_d}{w}+\frac{V_d^2}{2g}+h\right)-\left(\frac{p_s}{w}+\frac{V_s^2}{2g}\right)$$

or $\frac{V_{w1}u_1}{g}$ – (losses of head in the pump)= $\left(\frac{p_d}{w}+\frac{V_d^2}{2g}+h\right)-\left(\frac{p_s}{w}+\frac{V_s^2}{2g}\right)$

From Eq. 24.6, left hand side of the above equation is equal to H_m. Thus

$$H_m = \left(\frac{p_d}{w}+\frac{V_d^2}{2g}+h\right)-\left(\frac{p_s}{w}+\frac{V_s^2}{2g}\right) \quad ...(24.11)$$

Equation 24.11 thus indicates that 'the menometric head is equal to the difference between the total energies of the liquid at inlet and exit from the pump'.

If the gages are at the same level, $h = 0$, then Eq. 24.11 becomes

$$H_m = \left(\frac{p_d}{w}+\frac{V_d^2}{2g}\right)-\left(\frac{p_s}{w}+\frac{V_s^2}{2g}\right) \quad ...(24.12)$$

Furthermore if the suction and delivery pipes are of the same diameter, $V_s = V_d$, then manometric head is equal to difference of the pressure heads at the inlet and the exit from the pump, that is

$$H_m = \left(\frac{p_d}{w}-\frac{p_s}{w}\right) \quad ...(24.13)$$

The suction pipe is usually slightly larger than the delivery pipe, but as the velocities are not excessive in either of the pipes, the difference in velocity heads is small and is usually neglected, in that case also Eq. 24.13 will hold good. Moreover the suction head (p_s/w) being negative, it is indicated by Eq. 24.13 that if suction and delivery pipes are of the same size then the manometric head is equal to the arithmetical sum of the readings of the vacuum and pressure gages placed at the same level on the suction and the delivery pipes just at the inlet and the exit from the pump respectively. However, if the vacuum and the pressure gages are not placed at the same level then the manometric head is equal to the arithmetic sum of the readings of the vacuum and pressure gages plus the vertical difference in the levels of the vacuum and pressure gages. i.e.,

$$H_m = \frac{p_d}{w}-\frac{p_s}{w}+x \quad ...(24.13a)$$

where x is the vertical difference in the levels of the vacuum and pressure gages.

Equation 24.9 may also be written as

$$\frac{p_d}{w}-\frac{p_2}{w} = \frac{V_1^2}{2g}-\frac{V_d^2}{2g}-h-h_{lc}$$

The left hand side of the above equation represents the increase in the pressure head between the point 2 at the outlet of the impeller and the point 3 at the outlet of the pump. This increase in pressure head is due to partial conversion of velocity head of liquid into pressure head and is equal to $\left(\frac{V_1^2}{2g}-\frac{V_d^2}{2g}\right)$ where V_1 is the velocity with which the liquid leaves the impeller and V_d is the velocity at point 3 in the delivery pipe. Generally this gain in pressure head is expressed as $k(V_1^2/2g)$, where k is a coefficient the value of which varies from 0.4 for volute casing to 0.7 for a turbine pump. Thus

$$\frac{V_1^2}{2g}-\frac{V_d^2}{2g} = k\frac{V_1^2}{2g} \quad ...(24.14)$$

Equation 24.8 may be written as

$$\frac{V_{w1}u_1}{g}-h_{li}-h_{lc} = \left(\frac{p_2}{w}-\frac{p_s}{w}\right)+\frac{V_1^2}{2g}-\frac{V_s^2}{2g}-h_{lc}$$

or

$$H_m = \left(\frac{p_2}{w}-\frac{p_s}{w}\right)+\left(\frac{V_1^2}{2g}-\frac{V_d^2}{2g}\right)+\frac{V_d^2}{2g}-\frac{V_s^2}{2g}-h_{lc}$$

Now if it is assumed that $V_d = V_s$ and Eq. 24.14 is introduced in the above expression then it becomes

$$H_m = \left(\frac{p_2}{w}-\frac{p_s}{w}\right)+k\frac{V_1^2}{2g}-h_{lc} \quad ...(24.15)$$

The Eq. 24.15 also provides another expression for the manometric head according to which

H_m = the actual pressure rise in the impeller + the pressure rise in the volute chamber or the guide blade ring (i.e., between the points 2 and 3) – the loss of head in the casing.

Finally the manometric head may also be expressed by another expression as indicated below.

Applying Bernoulli's equation between points 3 and 4 (point 4, as shown in Fig. 24.6, is at the liquid surface in the delivery tank), the following expression is obtained if the datum is considered at point 3,

$$\frac{p_d}{w}+\frac{V_d^2}{2g} = h_d+h_{fd}+\frac{V_d^2}{2g} \quad ...(24.16)$$

or

$$\frac{p_d}{w} = h_d+h_{fd}$$

where h_d is the delivery lift and h_{fd} is the head loss in the delivery pipe which includes the head loss due to friction and the other minor losses.

The head $\left(h_d+h_{fd}+\frac{V_d^2}{2g}\right)$ is called the *delivery head* of the pump.

Now introducing Eqs 24.7 and 24.16 in Eq. 24.12 it becomes

$$H_m = h_s + h_d + h_{fs} + h_{fd} + \frac{V_d^2}{2g} \quad ...(24.17)$$

Equation 24.17 represents another expression for the manometric head. It is observed from Eq. 24.17 that all the manometric head is not used to lift the liquid against the static head, since some of it is used to overcome the various losses in the pipes. Hence

H_m = static head + friction and minor head loss in suction and delivery pipes + the velocity head in the delivery pipe.

If the velocity head in the delivery pipe $(V_d^2/2g)$ is relatively small it may be neglected and then Eq. 24.17 becomes

$$H_m = h_s + h_d + h_{fs} + h_{fd} \quad ...(24.17a)$$

24.8 LOSSES AND EFFICIENCIES

(A) **Losses.** The various losses occurring during the operation of a centrifugal pump may be classified as follows:

(1) Hydraulic losses.
(2) Mechanical losses.
(3) Leakage losses.

(1) Hydraulic Losses. The *hydraulic losses* that may occur in a centrifugal pump installation may be grouped as

(a) Hydraulic losses in the pump.
(b) Other hydraulic losses.

The hydraulic losses that may occur within the pump consist of the following:
(i) Shock or eddy losses at the entrance to and the exit from the impeller.
(ii) Friction losses in the impeller.
(iii) Friction and eddy losses in the guide vanes (or diffuser) and casing.

It can be seen from Fig. 24.5 that for the given values of the blade angles θ and φ, and the speed of rotation, there can be only one rate of discharge that will ensure tangential entry to the impeller and tangential exit from the impeller. But often the pump is required to operate under varying conditions which results in the variation in the rate of discharge. As such at the entrance and the exit of the impeller the shock losses generally occur. Furthermore at the exit from the impeller there occurs a loss of energy due to the more or less abrupt change in the direction of the velocity of liquid as it enters the casing.

The other hydraulic losses consist of the following:
(i) Friction and other minor losses in the suction pipe.
(ii) Friction and other minor losses in the delivery pipe.

(2) Mechanical Losses. The *mechanical losses* occur in the centrifugal pump on account of the following:
(i) Disc friction between the impeller and the liquid which fills the clearance spaces between the impeller and the casing.
(ii) Mechanical friction of the main bearings and glands.

(3) Leakage Losses. In centrifugal pumps as ordinarily built, it is not possible to provide a completely water tight seal between the delivery and suction spaces. As such there is always a certain amount of liquid which slips or leaks from the high pressure to the low pressure points in the pump and it never

passes through the delivery pipe. The liquid which escapes or leaks from a high pressure zone to a low pressure zone carries with it energy which is subsequently wasted in eddies. This loss of energy due to leakage of liquid represents the *leakage loss.*

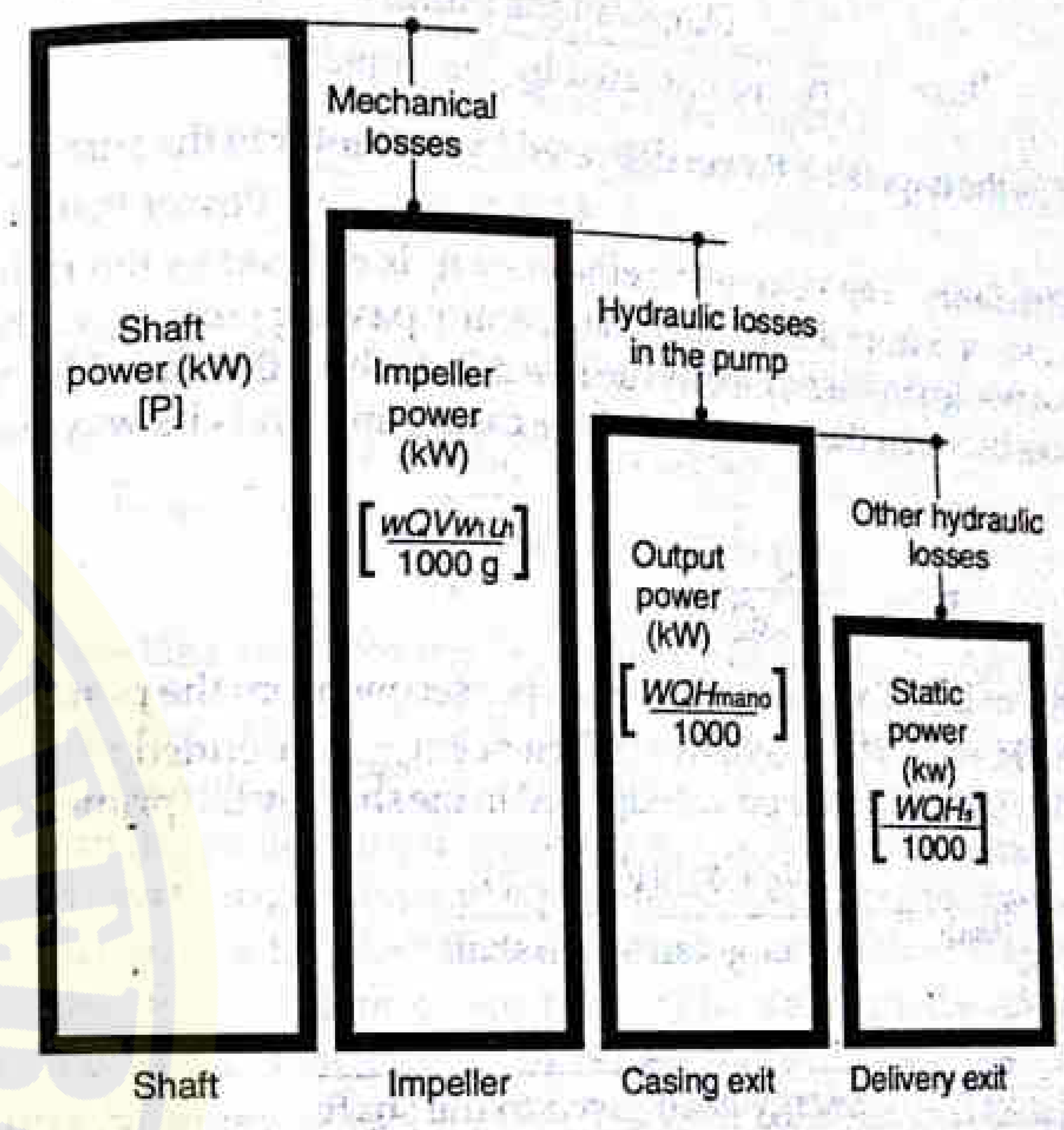


Figure 24.7 *Head losses in centrifugal pump*

(B) **Efficiencies.** The efficiency of a centrifugal pump is expressed in the following form:
(i) Manometric efficiency,
(ii) Volumetric efficiency,
(iii) Mechanical efficiency,
(iv) Overall efficiency.

(i) **Manometric Efficiency.** The manometric efficiency η_{mano} is defined as the ratio of the manometric head developed by the pump to the head imparted by the impeller to the liquid. Thus

$$\eta_{mano} = \frac{H_m}{(V_{w1}u_1/g)} = \frac{gH_m}{V_{w1}u_1} \quad ...(24.18)$$

or

$$\eta_{mano} = \frac{H_m}{H_m + \text{Losses in the pump}}$$

If Q is the volume of liquid actually delivered per second by the pump and w is the specific weight of the liquid then

$$\eta_{mano} = \frac{wQH_m}{wQ(V_{w1}u_1/g)} \quad ...(24.19)$$

or

$$\eta_{mano} = \frac{\text{Power actually delivered by the pump}}{\text{Power imparted by the impeller}}$$

or

$$\eta_{mano} = \frac{\text{Output of the pump}}{\text{Power imparted by the impeller}} \quad ...(24.20)$$

[Power imparted by the impeller = Power delivered by the motor to the shaft (or shaft power) – Power lost in mechanical losses]

(ii) Volumetric Efficiency. The volumetric efficiency η_v is defined as the ratio of the quantity of liquid discharged per second from the pump to the quantity passing per second through the impeller. As stated earlier these two quantities differ by the rate ΔQ at which the liquid from the impeller leaks through the clearances between the impeller and the casing and finds its way back to the eye of the impeller. Thus

$$\eta_v = \frac{Q}{(Q+\Delta Q)} \quad ...(24.21)$$

where Q is the quantity of liquid actually discharged per second from the pump.

(iii) Mechanical Efficiency. The mechanical efficiency η_{mech} is defined as the ratio of the power actually delivered by the impeller to the power supplied to the shaft by the prime mover or motor. Thus

$$\eta_{mech} = \frac{w(q+\Delta Q)(V_{w_1}u_1/g)}{\text{Power given to the shaft}} \quad ...(24.22)$$

or

$$\eta_{mech} = \frac{(V_{w1}u_1/g)}{\text{Energy head given to the shaft}}$$

or

$$\eta_{mech} = \frac{(V_{w1}u_1/g)}{(V_{w1}u_1/g)+(\text{mechanical head losses in bearing})} \quad ...(24.22a)$$

(iv) Overall Efficiency. The overall efficiency η_o of the pump is defined as the ratio of the power output from the pump to the power input from the prime mover driving the pump. Thus

$$\eta_o = \frac{wQH_m}{\text{Power given to the shaft}} \quad ...(24.23)$$

The overall efficiency is also equal to the product of all the three efficiencies described above. That is

$$\eta_o = (\eta_{mano}) \times (\eta_v) \times (\eta_{mech})$$

$$= \frac{H_m}{(V_{w1}u_1/g)} \times \frac{Q}{(Q+\Delta Q)} \times \frac{w(Q+\Delta Q)(V_{w1}u_1/g)}{\text{Power given to the shaft}}$$

$$= \frac{wQh_m}{\text{Power given to the shaft}}$$

which is same as Eq. 24.23.

(C) Effect of Vane Angle ϕ on Manometric Efficiency. If the loss of head in the pump is neglected then the manometric head may be expressed as

$$H_m = \frac{V_{w1}u_1}{g} - \frac{V_1^2}{2g} \quad ...(24.24)$$

From the outlet velocity triangle shown in Fig. 24.5,

$$V_1^2 = (V_{w1}^2 + V_{f1}^2) \text{ and } V_{w1} = (u_1 - V_{f1}\cot\phi).$$

Then

$$H_m = \frac{u_1^2 - V_{f1}^2\operatorname{cosec}^2\phi}{2g} \quad ...(24.25)$$

Under the ideal condition assumed above the manometric efficiency of the pump will become

$$\eta_{mano} = \frac{gH_m}{V_{w1}u_1} = \frac{(u_1^2 - V_{f_1}^2\operatorname{cosec}^2\phi)}{2u_1(u_1 - V_{f_1}\cot\phi)} \quad ...(24.26)$$

Now assuming a constant value for the flow ratio $\psi\left[= V_{f1}/\sqrt{2gH_m}\right] = 0.25$ and computing the value of u_1 in terms of H_m from Eq. 24.25 for different values of ϕ, it will be observed that as the value of ϕ varies from 90° to 20°, the value of η_{mano} increases from 0.47 to 0.73. It is obvious from equation 24.26 that a further decrease in the value of ϕ will result in the increase in the efficiency, but it is impracticable to have the values of ϕ less than 20°, since it will result in the long and narrow passages causing high frictional losses. As such in practice the value of ϕ is not reduced below 20°, but some other devices in the form of guide vanes etc., are adopted which will convert a part of the velocity energy into useful pressure energy, thereby increasing the efficiency of the pump.

(D) Effect of Finite Number of Vanes of the Impeller on Head and Efficiency. Eq. 24.3 or 24.4 represents the head imparted by the impeller to the liquid only if there are infinite number of vanes (or blades) in the impeller and it is usually known as *Euler's head*. Similarly the velocity triangles shown in Fig. 24.5 will be developed only if there are infinite number of vanes in the impeller and these velocity triangles are also known as *Euler's velocity triangles*. However, in practice the impeller has only a finite number of vanes. Therefore although the vanes are designed in accordance with the consideration of Euler's velocity triangles, but the actual velocity triangles developed in the case of impellers with finite number of vanes are different from the Euler's velocity triangles. On account of secondary or circulatory flow effects developed in the impeller the actual velocity of whirl V_{w1} developed in the case of impeller with finite number of vanes is always less than that corresponding to the Euler's velocity triangles. Consequently the actual head H_i imparted by the pump impeller (with finite number of vanes) to the liquid is always less than the Euler's head H_e (given by Eq. 24.3 or 24.4). The difference between the actual head H_i imparted by the impeller with finite number of vanes to the liquid and the Euler's head H_e is not considered as loss of head. Further the ratio (H_i/H_e) is usually termed as *vane efficiency* or *vane effectiveness* ε (Greek 'epsilon') i.e.,

$$\varepsilon = \frac{H_i}{H_e}$$

Experiments made on impellers of identical size, speed and vane angle but with different number of vanes have indicated that as the number of vanes is increased the value of ε increases and approaches unity. In other words, it means that the actual head imparted by the impeller (with finite number of vanes) to the liquid, approaches the Euler's head as the number of vanes is increased. In addition, the number of vanes, the value of ε also depends on the shape of the vane and the outlet vane angle. In general for radial flow pumps the value of ε varies from 0.6 to 0.8 as the number of vanes is increased from 4 to 12. However, for impellers with vanes more than 24 the value of ε may be taken as unity.

A portion of the head H_i actually imparted by the impeller (with finite number of vanes) to the liquid will be lost in the pump and hence the net manometric head H_m available from the pump may be expressed as

$$H_m = H_i - \text{losses of head in the pump.}$$

Further the ratio of the manometric head H_m available from a pump and the head H_i actually imparted by the impeller to the liquid is termed as *hydraulic efficiency* η_h of the pump i.e.,

$$\eta_h = \frac{H_i - \text{losses of head in the pump}}{H_i}$$

or

$$\eta_h = \frac{H_m}{H_i}$$

However, the manometric efficiency η_{mano} of a pump has been defined earlier as (Eq. 24.18)

$$\eta_{mano} = \frac{H_m}{(V_{w1}u_1/g)} = \frac{H_m}{H_e}$$

Thus combining the above noted expressions the manometric efficiency becomes

$$\eta_{mano} = \frac{H_m}{H_i} \times \frac{H_i}{H_e} = (\eta_h \times \varepsilon)$$

Again for $\varepsilon = 1; H_i = H_e$ and $\eta_{mano} = \eta_h$

As stated earlier since the actual shape of the velocity triangles for impellers with finite numbers of vanes being not known, the impeller vanes are designed in accordance with the Euler's velocity triangles. As such in the various illustrative examples the value of ε has been assumed as 1 and accordingly H_i has been considered to be same as H_e.

24.9 MINIMUM STARTING SPEED

When the pump is started, there will be no flow of water until the pressure difference in the impeller is large enough to overcome the gross or manometric head. If the impeller is rotating, but there is no flow, then the water is rotating in a forced vortex. Therefore a centrifugal head or pressure head caused by the centrifugal force on the rotating water will be $(u_1^2 - u^2)/2g$. The flow will commence only if

$$[(u_1^2 - u^2)/2g] \geq H_m$$

or

$$\left[\frac{(\pi D_1 N/60)^2}{2g} - \frac{(\pi DN/60)^2}{2g}\right] \geq H_m$$

or

$$\left[\left(\frac{\pi N}{60}\right)^2 (D_1^2 - D^2)\right] \geq (2gH_m)$$

For determining the minimum speed required for the pump to commence the flow, the above expression may be written as

$$\left[\left(\frac{\pi N}{60}\right)^2 (D_1^2 - D^2)\right] = (2gH_m) \quad \text{...(24.27)}$$

from which the required value of N may be computed.

24.10 LOSS OF HEAD DUE TO REDUCED OR INCREASED FLOW

The efficiency of a pump will be maximum only when it is running and discharging at its designed speed. But if the discharge is either reduced or increased, then there will be a loss of head due to shock at the entrance to the impeller, which will result in lowering the efficiency of the pump.

As shown in Fig. 24.8, let *abd* be the inlet velocity triangle for the pump when running under normal conditions. The vanes at inlet tip will be parallel to *ab*. Now if the radial flow through the pump is reduced (or increased) from *bd* to *cd*, while the speed of rotation remains the same, the velocity triangle will be represented by *acd*, *ac* being the new relative velocity. But the angle of the vane at inlet will be the same as before, the relative velocity, therefore, will no longer be parallel to the vane and the shock will occur.

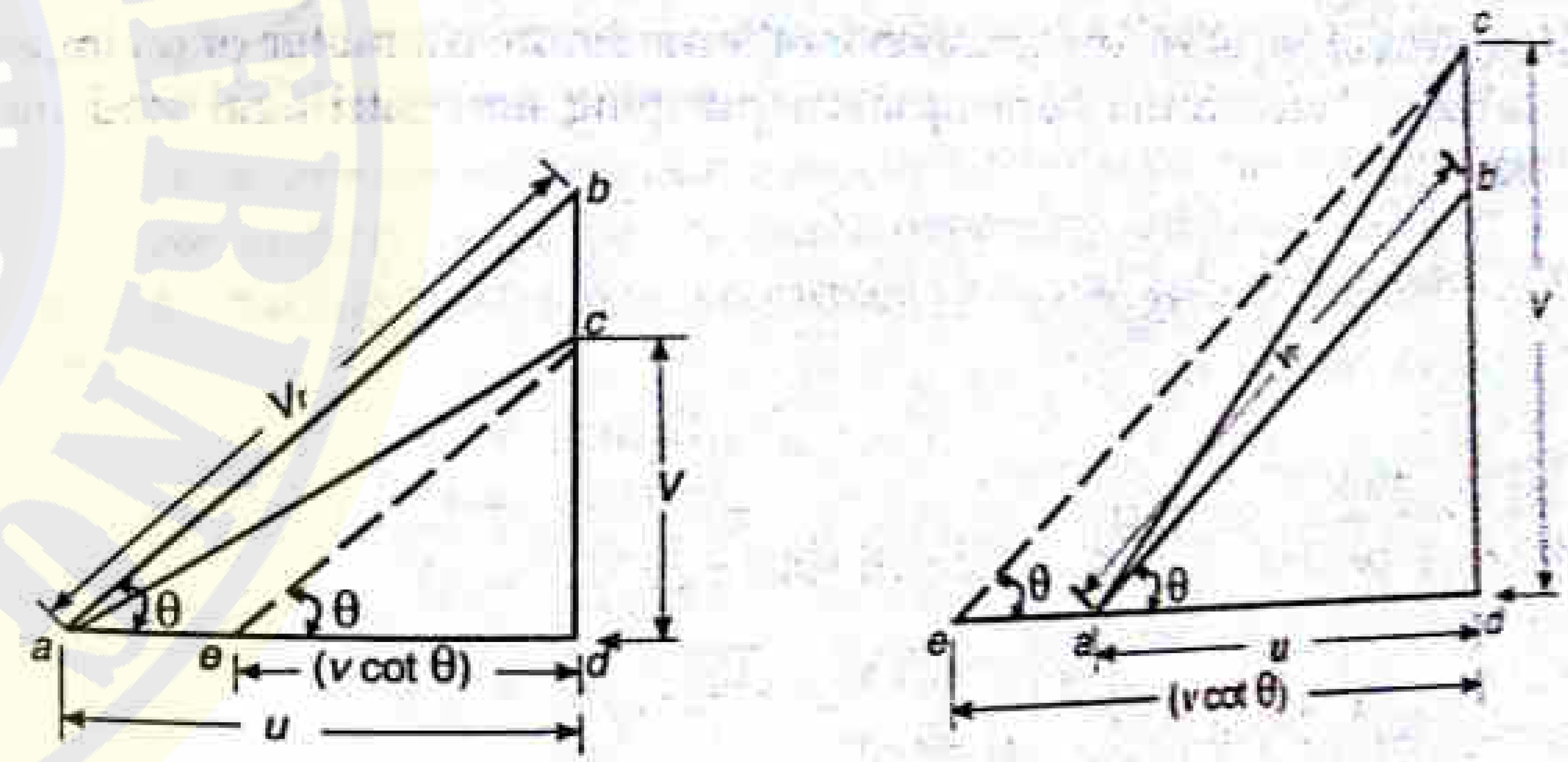


Figure 24.8 *Inlet velocity triangle with reduced and increased flow*

Now as the new velocity of flow *cd* is fixed and also the water must pass along the vane, it follows that the velocity diagram will be triangle *ecd*, *ec* being parallel to *ab*. Therefore a tangential change of velocity *ae* will suddenly take place resulting in the shock causing the head loss. The loss of head due to sudden change in velocity

$$= \left[\frac{(\text{change in velocity})^2}{2g}\right]$$

∴ Loss of head at the entrance

$$= \left[\frac{(ae)^2}{2g}\right] = \left[\frac{(u - V\cot\theta)^2}{2g}\right]$$

24.11 DIAMETERS OF IMPELLER AND PIPES

(a) Outside the Diameter of Impeller. If the outside diameter of the impeller is D_1 and the speed of the pump shaft is N r.p.m., then

$$u_1 = (\pi D_1 N/60)$$

Further if H_m is the total head, then

$$u_1 = K_u\left(\sqrt{2gH_m}\right)$$

where K_u is the speed ratio for the pump.

$$\therefore \quad D_1 = \frac{60k_u\sqrt{2gH_m}}{\pi N} = \frac{(84.60)K_u(\sqrt{H_m})}{N} \quad ...(24.28)$$

The value of K_u usually ranges from 0.95 to 1.25 as shown in Table 24.1. Equation 24.28 can, however, be used to determine the head which a pump can develop if D_1 and N are known. This will serve at a check for an existing pump.

(b) Inlet Diameter of Impeller. The inlet diameter D is $\frac{2}{3}$ to $\frac{1}{3}$ of D_1 depending upon the specific speed N_s or total head H_m. However in most of the cases, $D = (0.5)\ D_1$.

(c) Least Diameter of Impeller. An expression for the minimum diameter of an impeller can be derived on the basis of the fact that for the pump to start pumping, the centrifugal head must be equal to the total head H_m. Thus

$$\left[\frac{(u_1^2 - u^2)}{2g}\right] = H_m$$

or

$$\left[\left(\frac{\pi D_1 N}{60}\right)^2 - \left(\frac{\pi DN}{60}\right)^2\right] = 2gH_m$$

or

$$(D_1^2 - D^2) = \left(\frac{60}{\pi N}\right)^2 \times (2gHm)$$

Taking $D = 0.5\,D_1$; and solving for D_1

$$D_1 = \frac{2\times 60\times(\sqrt{2gH_m})}{(\sqrt{3})\pi N} = \frac{(97.68)(\sqrt{H_m})}{N} \quad ...(24.29)$$

A comparison of Eqs 24.28 and 24.29 indicates that for speed ratio K_u equal to $(2\sqrt{3})$ both the equations become same.

(d) Diameter of Suction Pipe. If d_s is the diameter of the suction pipe and V_s is the velocity of flow in suction pipe, then the amount of water to be pumped is given by

$$Q = \left[\frac{(\pi d_s^2)V_s}{4}\right]$$

or

$$d_s = \left[\sqrt{\frac{(4Q)}{(\pi V_s)}}\right] \quad ...(24.30)$$

Usually V_s is 1.5 to 3 m/s.

(e) Diameter of Delivery Pipe. If d_d is the diameter of the delivery pipe and V_d is velocity of flow in delivery pipe, then

$$Q =$$

or

$$d_d = \left[\sqrt{\frac{(4Q)}{(\pi V_d)}}\right] \quad ...(24.31)$$

Usually V_d is 1.5 to 3.5 m/s. The value of V_d is generally equal to or or slightly higher than that of V_s.

24.12 SPECIFIC SPEED

In order to compare the performance of different pumps, it is necessary to have some term which will be common to all the centrifugal pumps. The term ordinarily used for this purpose is the 'specific speed'. This is a type characteristic and can be used to predict the behaviour of one pump based on tests of similar, but different sized pumps.

The specific speed of a centrifugal pump may be defined as *the speed in revolutions per minute of a geometrically similar pump of such a size that under corresponding conditions it would deliver 1 litre of liquid per second against a head of 1 metre.* It is represented by N_s, and it may be expressed by a relation as derived below:

Discharge $\quad Q = (k\pi B_1 D_1)\,V_{f1}$

or $\quad Q \propto B1D1\,V_{f1}$

or $\quad Q \propto D_1^2\,V_{f1}$; since $B_1 \propto D_1$

Also $\quad V_{f1} = \psi\left(\sqrt{2gH_m}\right)$; or $V_{f1} \propto \left(\sqrt{H_m}\right)$

Thus $\quad Q \propto D_1^2\left(\sqrt{H_m}\right) \quad ...(24.32)$

or $\quad \dfrac{q}{d_1^2(\sqrt{H_m})} = \text{constant}$

Further $\quad u_1 = (\pi D_1 N/60)$

or $\quad D_1 \propto (u_1/N)$

Also $\quad u_1 = K_u\left(\sqrt{2gH_m}\right)\ u_1 \propto \left(\sqrt{H_m}\right)$

Thus $\quad D_1 \propto \left(\sqrt{H_m}/N\right) \quad ...(24.33)$

or $\quad \dfrac{\sqrt{H_m}}{D_1 N} = \text{constant}$

Substituting the value of D_1 from Eq. 24.33 in Eq. 24.32 it becomes

$$Q \propto (H_m^{3/2}/N^2)$$

or $$N^2 \propto (H_m^{3/2}/Q)$$

or $$N \propto \left(H_m^{3/4}\sqrt{Q}\right) \quad ...(24.34)$$

or $$N = C\left(H_m^{3/4}\sqrt{Q}\right); \text{ where } C \text{ is a constant}$$

or $$\frac{N\sqrt{Q}}{H_m^{3/4}} = C \quad ...(24.35)$$

Now according to the definition of the specific speed, putting $Q = 1$ litre/second and $H_m = 1$ m, then $C = N = N_s$. Thus

$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}} \quad ...(24.36)$$

Equation 24.36 represents the expression for the specific speed of a centrifugal pump, which is based on the unit discharge. It is the most commonly adopted expression for the specific speed of the pumps. The value of N_s usually varies from about 300 to 15,000 for a single impeller. Further the value of N_s is same is SI and metric system of units and it is equal to 0.67 times N_s in F.P.S system, where Q is in gallons per minute and H_m is in feet.

The values of Q and H_m to be substituted in Eq. 24.36 are those corresponding to the maximum efficiency of the pump at its normal working speed.

For a multi-stage pump the value of H_m to be used in Eq. 24.36 is obtained by dividing the total head developed by the number of stages. Similarly for a double suction pump half the actual discharge delivered by the pump is taken as Q.

Sometimes another definition of the specific speed of a centrifugal pump is used, which is based on unit power, though it is not very common. According to this basis the specific speed is defined as 'the speed in revolutions per minute of a geometrically similar pump of such a size that under corresponding conditions it would absorb 1 kW power when working against a head of 1 metre'. If the power required to drive the pump is P, then

$$P = [(wQH_m)/(\eta_0)]$$

Assuming η_0 to be constant and substituting for Q from Eq. (24.34), we have $P \propto (H_m^{5/2}/N^2)$; or $P = \text{constant}\,(H_m^{5/2}/N^2)$

Hence $$N = [C_1 H_m^{5/4}/P^{1/2}) \text{ where } C_1 \text{ is a constant.} \quad ...(24.37)$$

Now according to the definition of the specific speed, putting $H_m = 1$ and $P = 1$, then $C_1 = N = N_s$

Thus

$$N_s = \frac{N\sqrt{P}}{H_m^{5/4}} \quad ...(24.38)$$

Equation 24.38 is, however, not very common with the pumps. As such if nothing is mentioned, then the specific speed of a pump generally refers to that obtained on the basis of the unit discharge criterion, as represented by Eq. 24.36.

Centrifugal pumps may also be classified on the basis of the specific speed. Table 24.2 gives the different types of centrifugal pumps with the corresponding ranges of specific speeds. The values of N_s given in this table are as given by Eq. 24.36.

TABLE 24.2

S. No.	Type of pump	Specific speed N_s
1	Slow speed radial flow	300 to 900
2	Medium speed radial flow	900 to 1500
3	High speed radial flow	1500 to 2400
4	Mixed flow or screw type	2400 to 5000
5	Axial flow or propeller type	3400 to 15000

24.13 MODEL TESTING OF PUMPS

The large-sized pumps are usually manufactured only after testing their small scale models, because from the test results of the models it would be possible to predict the performance of the prototype in advance and also it would be possible to make necessary alterations in the design of the prototype pump. The performance of the prototype pump will be correctly predicted by its model tests only if the model bears a complete similitude with its prototype. The necessary conditions for the complete similarity to exist between the model of a pump and its prototype may be obtained as indicated below.

The variables involved in this case are discharge Q, manometric or total head H, speed of rotation of impeller N, impeller diameter D, power P, mass density and viscosity μ of the fluid pumped. Again as in the case of turbines considering gH as the variable in place of H, these variables may be grouped into the dimensionless parametres viz., discharge number $\left(\frac{Q}{ND^3}\right)$, head number $\left(\frac{gH}{N^2D^2}\right)$, power number $\left(\frac{P}{\rho gHND^3}\right)$ and Reynolds number $\left(\frac{\mu}{\rho ND^2}\right)$. By dividing the square root of discharge number by head number raised to power (3/4) the following dimensionless parameter n_s is obtained

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

The parameter n_s is known as *dimensionless specific speed* or *shape number* of the pump.

For complete similarity to exist between the model and the prototype pumps the above noted parameters must have the same value for the model and the prototype pumps. However, as described in Chapter 22 (section 22.6) since the flow in the prototype pump is turbulent, if the flow in the model pump is also turbulent then even if the Reynolds numbers are not equal for the model and the prototype pumps, the similarity between them can be ensured. Thus for complete similarity to exist between the model and the prototype pumps the necessary conditions are as given below: ...(24.39)

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p \quad ...(24.40)$$

$$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p$$

$$\left(\frac{P}{\rho g Q H}\right)_m = \left(\frac{P}{\rho g Q H}\right)_p$$

or
$$\left(\frac{P}{\rho g H N D^3}\right)_m = \left(\frac{P}{\rho g H N D^3}\right)_p$$

or
$$\left(\frac{P}{\rho N^3 D^5}\right)_m = \left(\frac{P}{\rho N^3 D^5}\right)_p \quad ...(24.41)$$

$$\left[\frac{N\sqrt{Q}}{(gH)^{3/4}}\right]_m = \left[\frac{N\sqrt{Q}}{(gH)^{3/4}}\right]_p \quad ...(24.42)$$

Since in most of the cases $g_m = g_p$ and $\rho_m = \rho_p$ the above conditions may be simplified.

The above noted conditions for establishing the similarity between the model and the prototype pumps are based on the assumption that the efficiency of the model is equal to that of the prototype. However, the efficiencies of the model and the prototype pumps are not equal. This is so because on account of the losses in the model and the prototype pumps being disproportionate, the model efficiency is usually lower than that of the prototype. As such in order to correctly predict the performance of the prototype pump on the basis of the model test results the above noted expressions for the similarity conditions needs to be modified and the modified expressions are as indicated below.

$$\left(\frac{gH}{\eta_{mano} N^2 D^2}\right)_m = \left(\frac{gH}{\eta_{mano} N^2 D^2}\right)_p \quad ...(24.40\ a)$$

$$\left(\frac{P\eta_0}{\rho g Q H}\right)_m = \left(\frac{P\eta_0}{\rho g Q H}\right)_p$$

or
$$\left(\frac{P\eta_0}{\rho g H N D^3}\right)_m = \left(\frac{P\eta_0}{\rho g H N D^3}\right)_p$$

or
$$\left(\frac{P\eta_0}{\rho N^3 D^5}\right)_m = \left(\frac{P\eta_0}{\rho N^3 D^5}\right)_p \quad ...(24.41\ a)$$

$$\left[\frac{N\sqrt{Q}\ \ \eta_{mano}^{3/4}}{gH^{3/4}}\right]_m = \left[\frac{N\sqrt{Q}\ \ \eta_{mano}^{3/4}}{gH^{3/4}}\right]_p \quad ...(24.42\ a)$$

Further the efficiency of the prototype pump being not equal to that of the model pump, by applying suitable correction factor, the efficiency of the prototype pump can be predicated from the efficiency obtained for its model. Based on the test efficiencies of single stage centrifugal pumps, G.F. Wislicenus has suggested the following expression relating the efficiencies of the model and the prototype pumps

$$\frac{0.95-\eta_m}{0.95-\eta_p} = \left[\frac{0.658+\log_{10} Q_p}{0.658+\log_{10} Q_m}\right]^2 \quad ...(24.43)$$

in which Q is the discharge in litres per minute.

24.14 PUMP IN SERIES—MULTI-STAGE PUMPS

The head produced by a centrifugal pump depends on the rim speed of the impeller. To increase the rim speed, either the rotative speed or the diameter of the impeller or both must be increased. Increasing either of these has the effect of increasing the stress in the impeller material. For this reason it is usually not possible to produce very high head with one impeller. Normally a pump with a single impeller can be used to deliver the required discharge against a maximum head of about 100 m. But if the liquid is required to be delivered against a still larger head then it can be done by using two or more *pumps in series*. The pumps in series may be used in boiler feeding system to satisfy high head demand, in multi-stations systems along pipelines such as long oil pipelines, for boosting the liquid like pumping installations interposed in the water mains, etc. The higher heads may also be produced by using multi-stage pumps.

A multi-stage pump consists of two or more identical impellers mounted on the same shaft, and enclosed in the same casing. All the impellers are connected in series, so that liquid discharged with increased pressure from one impeller passes through the connecting passages to the inlet of the next impeller and so on, till the discharge from the last impeller passes into the delivery pipe. The impellers are surrounded by guide vanes which are generally provided within the connecting passages, and are meant for the recuperation of the velocity energy of the liquid leaving the impeller into pressure energy. According to the number of impellers fitted in the casing a multi-stage pump is designated as two stage, three-stage, etc. Figure 24.9 shows a three-stage centrifugal pump.

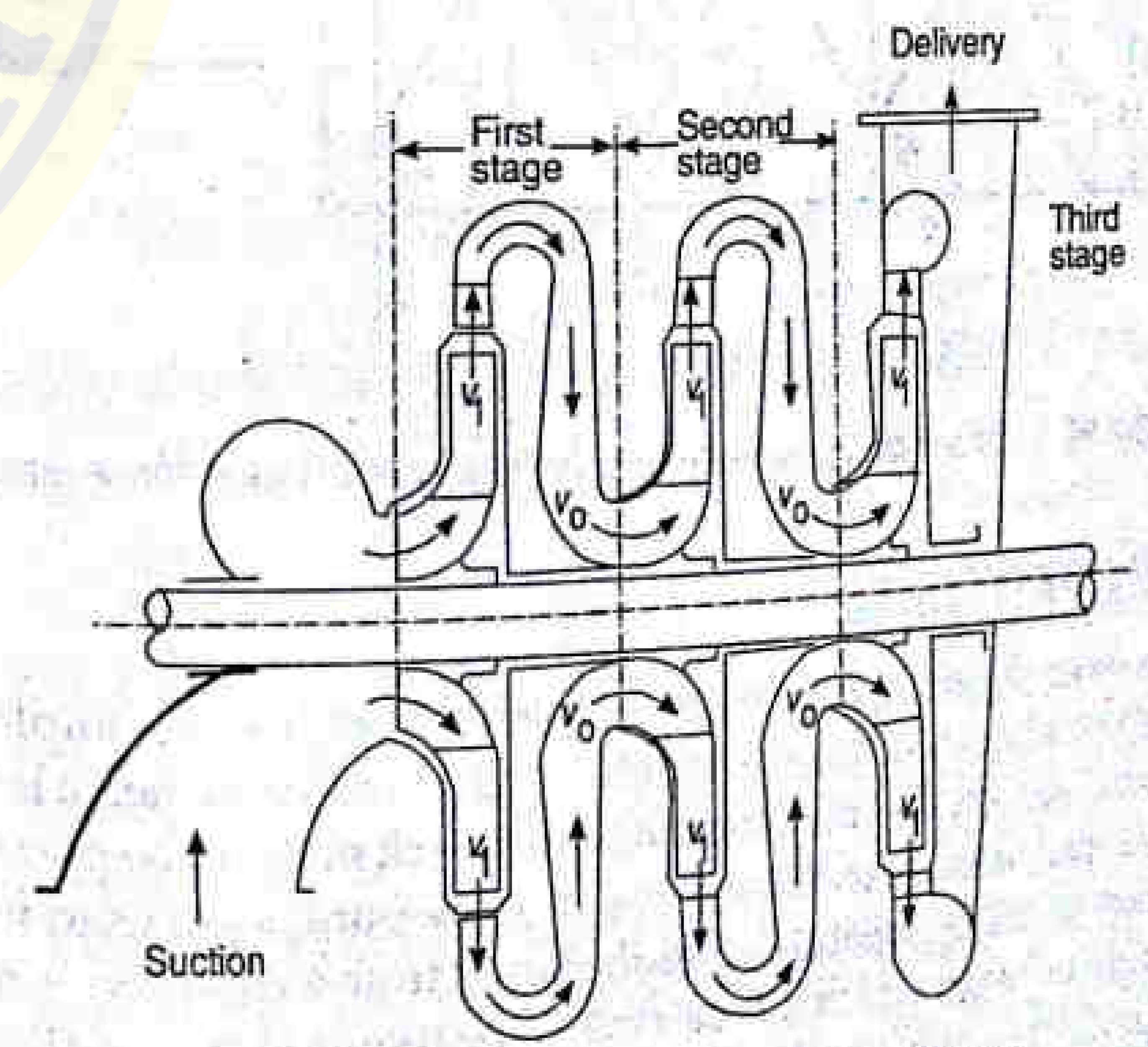


Figure 24.9 *Three-stage centrifugal pump*

Figure 24.10 indicates the variations of the velocity and the pressure head of the liquid as it passes through the pump. As the liquid passes through each impeller the absolute velocity of the liquid increases to V_1 and in each connecting passage the absolute velocity decreases again to V_0; but the pressure head continuously increases.

If H_1 and H_2 are the pressure heads gained by the liquid in each impeller and the surrounding guide vanes respectively, then the pressure head impressed on the liquid at each stage is $H_m = (H_1 + H_2)$. Now, if there are n impellers then since at each stage the pressure head will be raised by the same amount, the total head H developed by this multi-stage pump will be

$$H = n(H_m) = n(H_1 + H_2) \qquad ...(24.44)$$

Since the same liquid flows through each impeller, the discharge of a multi-stage pump is same as the discharge passing through each impeller of the series.

The number of stages to be adopted depends on the limitations of speed of the driving motor and also upon the discharge and the total head. If these factors are known, the head per stage can be fixed on the basis of the fact that specific speed N_s per impeller should not be less than about 700, with a provisional maximum limit of head per stage of 160 m.

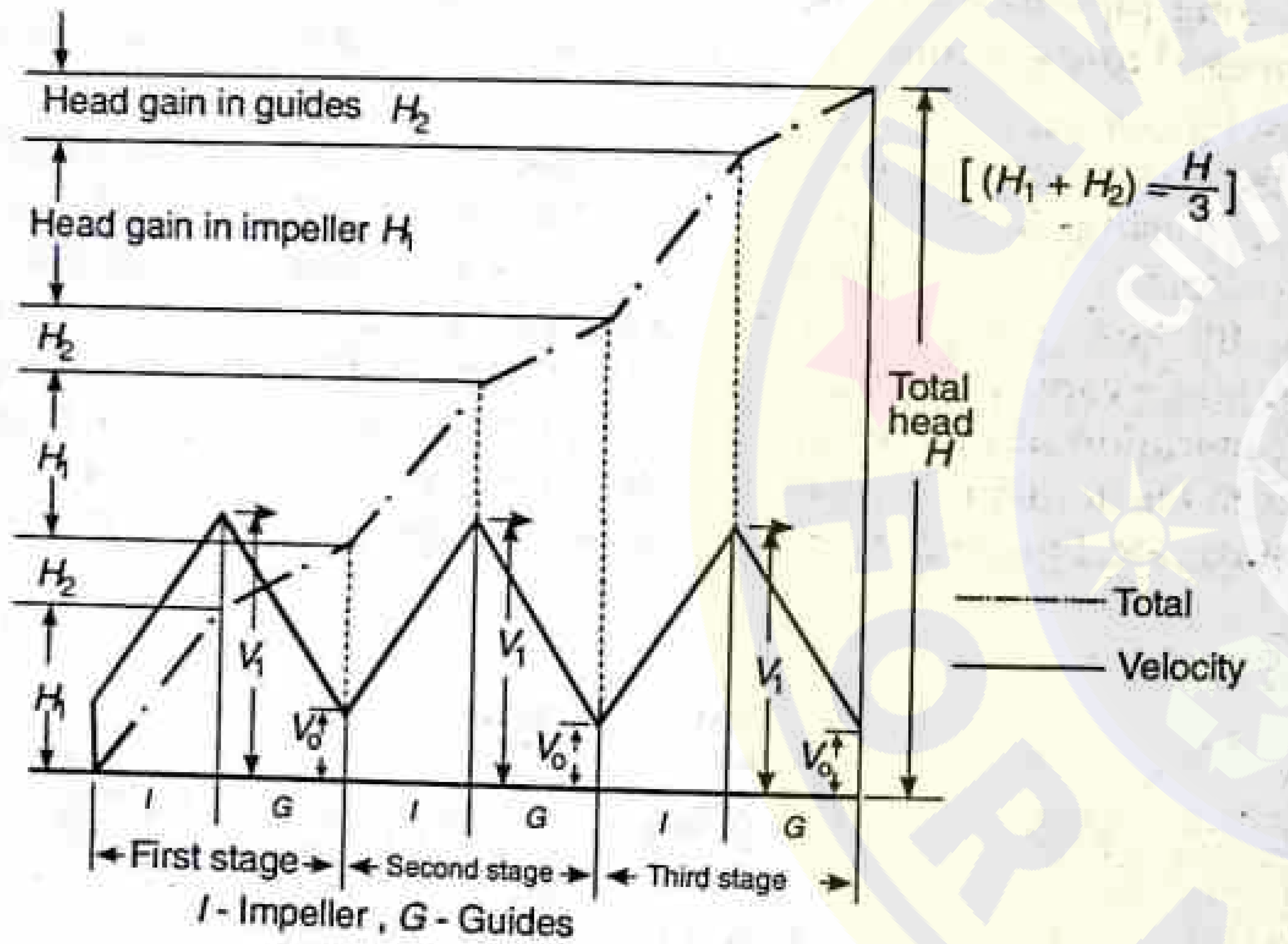


Figure 24.10 *Variations of velocity and pressure head in a three-stage pump*

24.15 PUMPS IN PARALLEL

The multi-stage pumps or the pumps in series as described earlier are employed for delivering a relatively small quantity of liquid against very high heads. However, when a large quantity of liquid is required to be pumped against a relatively small head, then it may not be possible for a single pump to deliver the required discharge. In such cases two or more pumps are used which are so arranged that each of these pumps working separately lifts the liquid from a common sump and delivers it to a common collecting pipe through which it is carried to the required height (Fig. 24.11). Since in this case each of the pumps delivers the liquid against the same head, the arrangement is known as *pumps in parallel*. If $Q_1, Q_2, Q_3 .... Q_n$ are the discharging capacities of n pumps arranged in parallel then the total discharged delivered by these pumps will be

$$Q_t = (Q_1 + Q_2 + Q_3 .... Q_n) \qquad ...(24.45)$$

If the discharging capacity of each of the n pumps is same, equal to Q then the total discharge delivered by these pumps will be

$$Q_t = nQ \qquad ...(24.46)$$

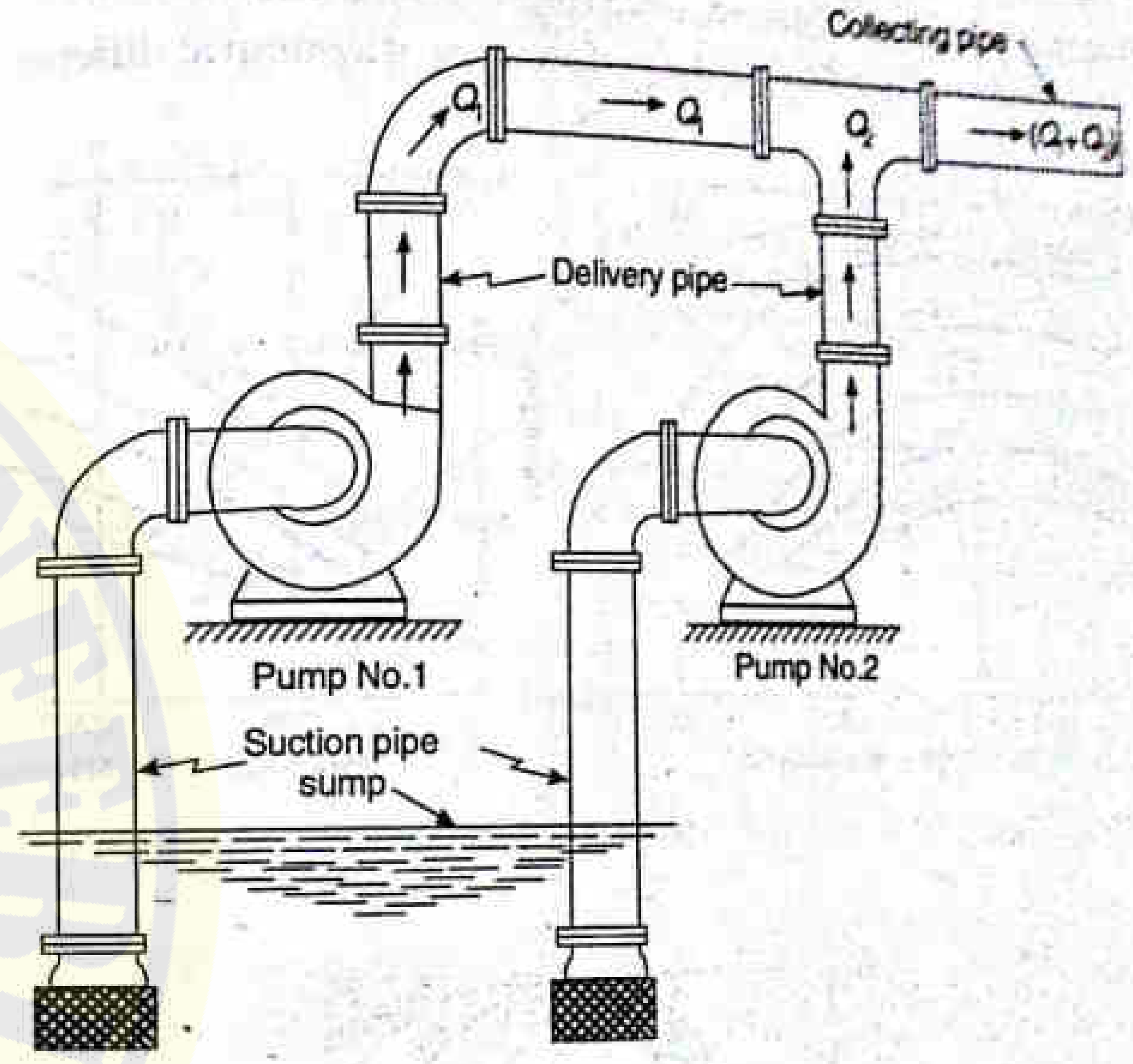


Figure 24.11 *Two centrifugal pumps arranged in parallel*

24.16 PERFORMANCE OF PUMPS—CHARACTERISTIC CURVES

A pump is usually designed for one speed, flow rate and head, but in actual practice the operation may be at some other condition of head or flow rate, and for the changed conditions the behaviour of the pump may be quite different. As indicated earlier, if the flow through the pump is less than the designed quantity, the value of the velocity of flow of liquid through the impeller will be changed, and as a result, the values of V_w and V_{w1} will be altered, thereby changing the head developed by the pump, and at the same time the losses will increase so that the efficiency of the pump will be lowered. Therefore, in order to predict the behaviour and performance of a pump under varying conditions, tests are performed, and the results of the tests are plotted. The curves thus obtained are known as the characteristic curves of the pump. The following three types of characteristic curves are usually prepared for the centrifugal pumps:

(a) Main and operating characteristics.
(b) Constant efficiency or Muschel curves.
(c) Constant head and constant discharge curves.

(a) Main and Operating Characteristics In order to obtain the main characteristic curves of a pump it is operated at different speeds. For each speed the rate of flow Q is varied by means of a delivery valve and for the different values of Q the corresponding values of manometric head H_m, shaft power P, and overall efficiency η_0, are measured or calculated. The same operation is repeated for different speeds of the pump. Then $H_m\ v/s\ Q$; $P\ v/s\ Q$ and $\eta_0\ v/s\ Q$ curves for different speeds are plotted, so that three sets of curves, as shown in Fig. 24.12 are obtained, which represent the *main characteristics* of a pump. The main characteristics are useful in indicating the performance of a pump at different speeds.

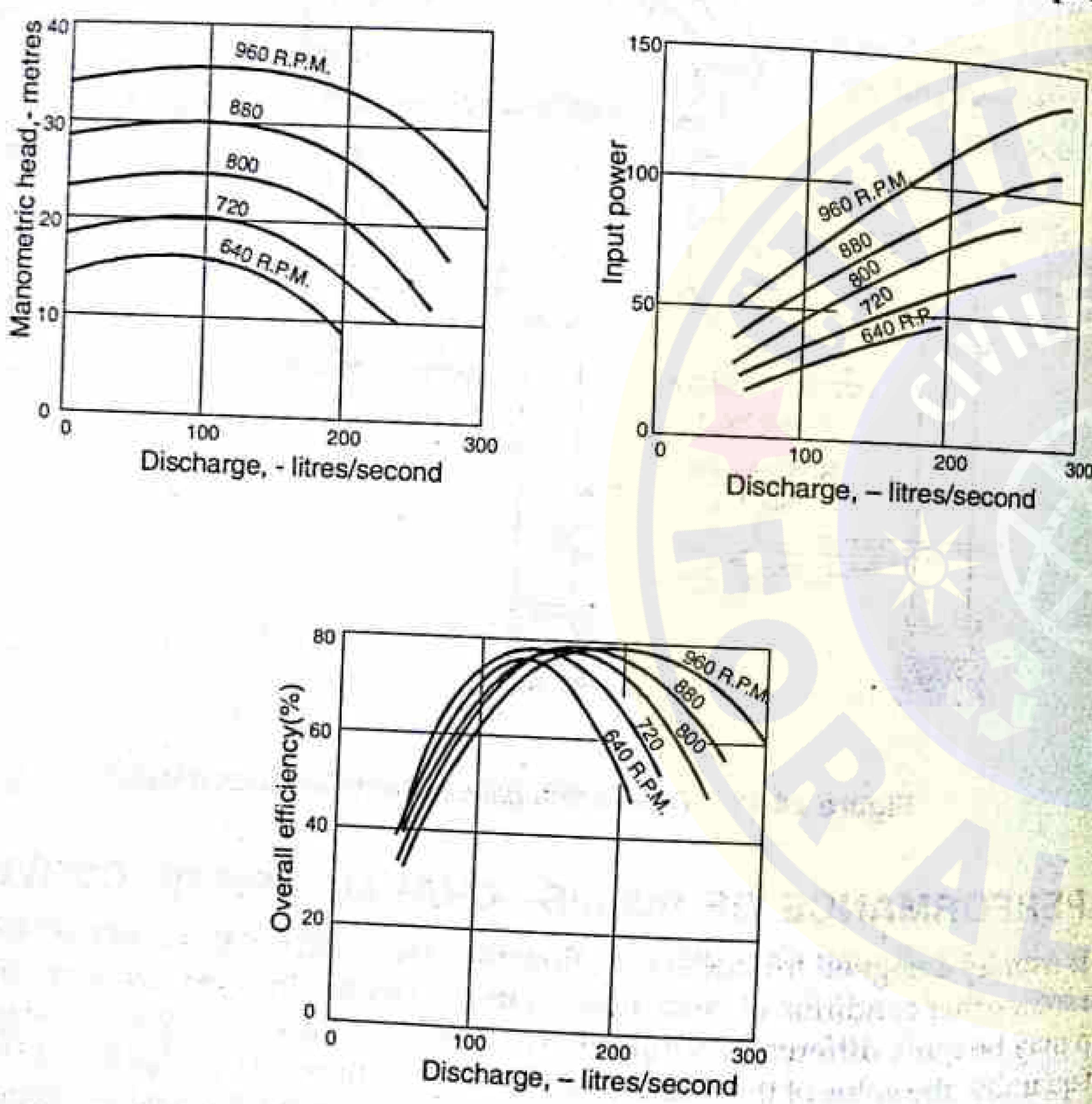


Figure 24.12 *Main characteristics of a centrifugal pump*

During the operation a pump is normally required to run at a constant speed, which is its designed speed (same as the speed of the driving motor). As such that particular set of main characteristics which corresponds to the designed speed is mostly used in the operations of a pump and is, therefore, known as the *operating characteristics*. A typical set of such characteristics of a pump is shown in Fig. 24.13.

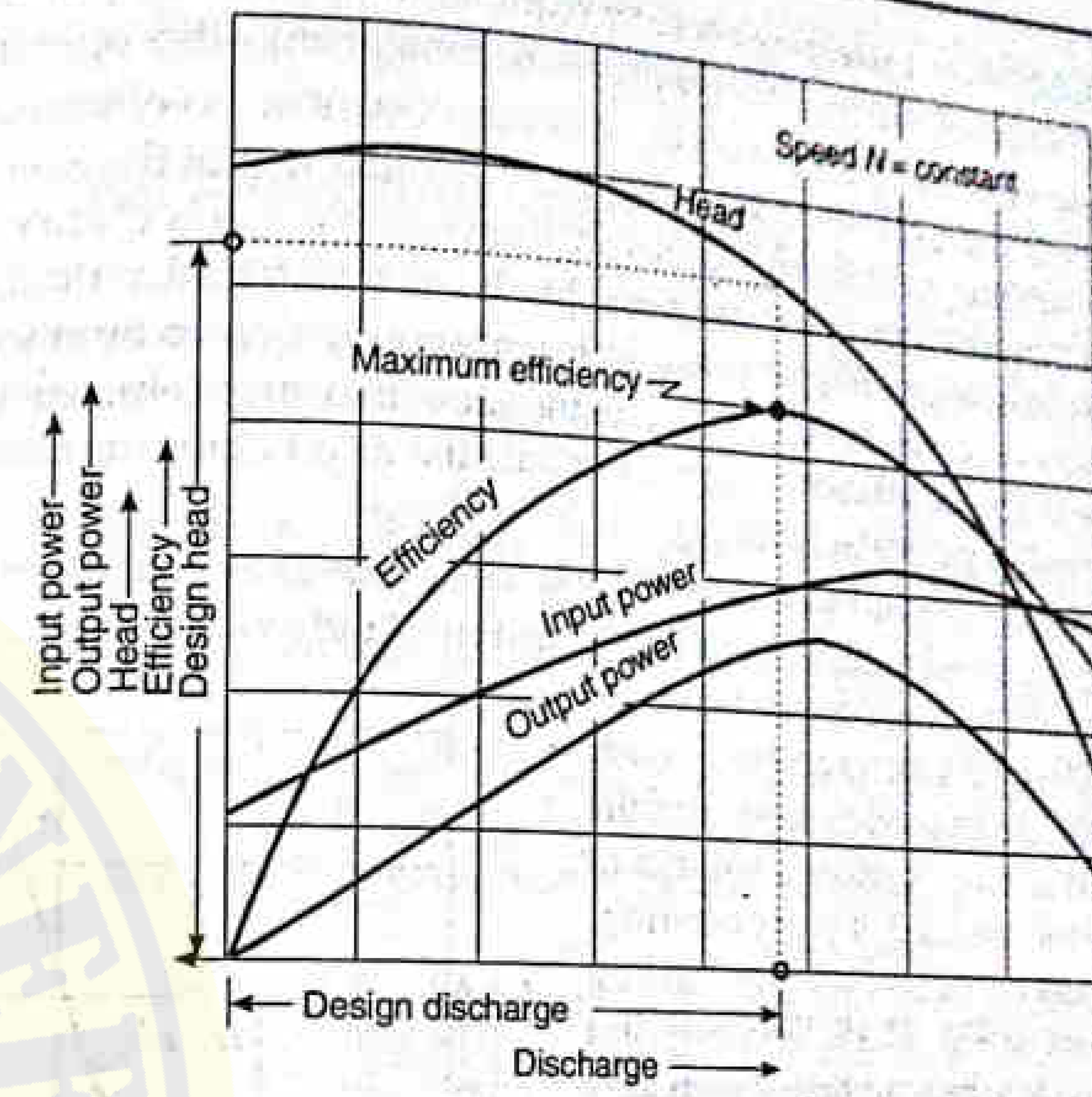


Figure 24.13 *Operating characteristic curves of a centrifugal pump*

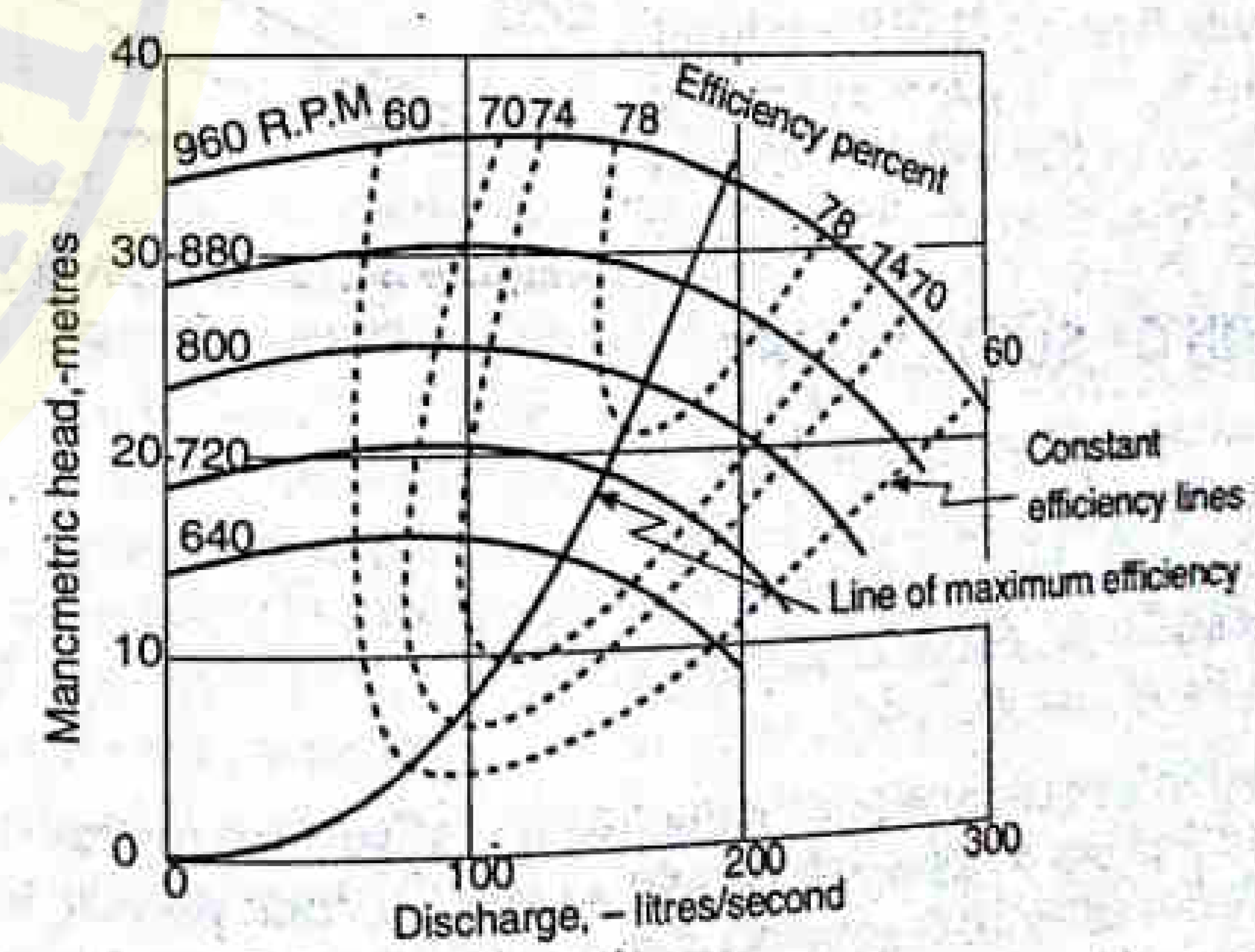


Figure 24.14 *Constant efficiency curves of a centrifugal pump*

The values of the head and the discharge corresponding to the maximum efficiency are known as the normal (or designed) head and the normal (or designed) discharge of a pump. The head corresponding to zero or no discharge is known as the *shut-off head* of the pump. From these characteristics it is possible to determine whether the pump will handle the necessary quantity of liquid against the desired head and what will happen if the head is increased or decreased. The

P v/s Q curve will show what size motor will be required to operate the pump at the required conditions and whether or not the motor will be overloaded under any other operating conditions.

(b) Constant Efficiency Curves As shown in Fig. 24.14 the constant or iso-efficiency curves may be obtained from H_m v/s Q and η_0 v/s Q curves of Fig. 24.12. In order to plot the iso-efficiency curves horizontal lines representing constant efficiencies are drawn on the $\eta_0 v/s$ Q curves. The points, at which these lines cut the efficiency curves at various speeds, are transferred to the corresponding H_m v/s Q curves. The points corresponding to the same efficiency are then joined by smooth curves which represent the iso-efficiency curves. From these curves the line of maximum efficiency as shown in Fig. 24.14 may be obtained. The iso-efficiency curves facilitate the direct determination of the range of operation of a pump with a particular efficiency.

(c) Constant Head and Constant Discharge Curves These curves are useful in determining the performance of a variable speed pump for which the speed constantly varies. In such cases if the head H_m is maintained constant then as the speed N varies the rate of flow Q will vary. As such a plot of Q v/s N can be prepared which can be used to determine the speeds required to discharge varying amounts of liquid at a constant pressure head. It is obvious from Eq. 24.39 that $Q \sim N$ and hence Q v/s N plot will be a straight line as shown in Fig. 24.15. Similarly if a constant rate of flow Q is to be maintained then as N varies, H_m will vary. Thus a plot of H_m v/s N can be prepared which can be used to determine the speeds required to discharge a certain quantity of liquid at different pressure heads. From Eq. 24.40 it is evident that $H_m \sim N^2$ and hence H_m v/s N plot is a parabolic curve as shown in Fig. 24.15. Similarly it is obvious from Eq. 24.41 that $P \sim N^3$, as shown in Fig. 24.15.

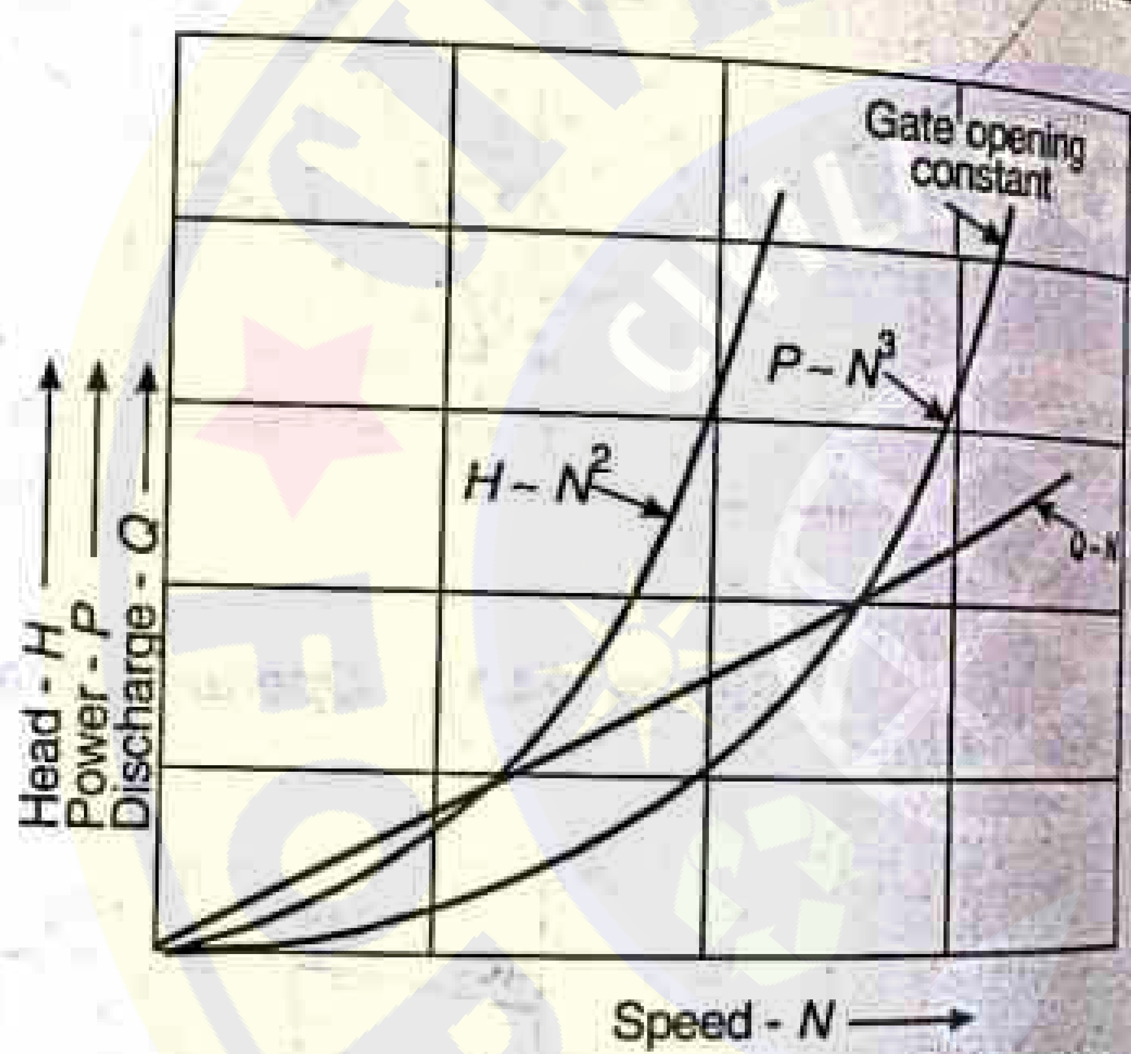


Figure 24.15 *Q v/s N; H_m v/s N; and P v/s N curves of a centrifugal pump*

24.17 LIMITATION OF SUCTION LIFT

From Eq. 24.7 the absolute pressure head at the inlet to the pump may be expressed as

$$\frac{p_a}{w} + \frac{p_s}{w} = \frac{p_a}{w} - \left[\frac{V_s^2}{2g} + h_s + h_{fs}\right] \qquad \ldots(24.47)$$

where p_a is the atmospheric pressure acting on the free liquid surface in the sump, p_s is the gage pressure at the inlet to the pump, V_s is the velocity of flow in the suction pipe, h_s is the suction lift and h_{fs} is the head lost in the foot valve, strainer and the suction pipe. It is not possible to create at the pump inlet, an absolute pressure lower than the vapour pressure of the liquid. Thus if p_v is the vapour pressure of the liquid in absolute units, then in the limiting case $(p_a + p_s) = p_v$, and hence from Eq. 24.47 the limiting value of the suction lift h_s is obtained as

$$h_s = \left(\frac{p_a - p_v}{w}\right) - \frac{V_s^2}{2g} - h_{fs} \qquad \ldots(24.48)$$

The suction lift should in no case be more than that given by Eq. 24.48, because a greater h_s may result in a rapid vaporization of the liquid due to the reduction of the pressure, which may ultimately lead to cavitation.

24.18 NET POSITIVE SUCTION HEAD (NPSH)

The *net positive suction head* (or NPSH) is defined as the absolute pressure head at the inlet to the pump, minus the vapour pressure head (in absolute units) corresponding to the temperature of the liquid pumped, plus the velocity head at this point. Thus

$$\text{NPSH} = \left(\frac{p_a}{w} + \frac{p_s}{w}\right) - \frac{p_v}{w} + \frac{V_s^2}{2g} \qquad \ldots(24.49)$$

Introducing the value of (p_s/w) from Eq. 24.47 in the above expression it becomes

$$\text{NPSH} = \frac{p_a}{w} - \frac{p_v}{w} - h_s - h_{fs} \qquad \ldots(24.50)$$

The right hand side of Eq. 24.50 also represents the *total suction head* H_{sv}, i.e.,

$$H_{sv} = \frac{p_a}{w} - \frac{p_v}{w} - h_s - h_{fs} \qquad \ldots(24.51)$$

and hence

$$\text{NPSH} = H_{sv} \qquad \ldots(24.51a)$$

In other words, NPSH may also be defined as the head required to make the liquid to flow through the suction pipe to the impeller.

The term NPSH is very commonly used in the pump industry. For any pump installation a distinction is usually made between the required NPSH and the available NPSH. The required NPSH for a pump varies with the pump design, the speed of the pump and its capacity. The required NPSH for a pump may be determined experimentally as indicated below and its magnitude must be given by the manufacturer of the pump. When the pump is installed the available NPSH can be determined from the above noted expression. Both the available and the required NPSH vary with the discharge in such a manner that with the increase in the discharge the available NPSH decreases but the required NPSH increases. In order to have cavitation free operation of a centrifugal pump the available NPSH should be more than the required NPSH. It may also be mentioned that the NPSH requirements for a given pump as determined experimentally, by definition are tied to the size of the suction pipe. Therefore, the available NPSH of a given installation should be based on the same size of the suction pipe.

For determining the required NPSH, the pump is operated at the designed speed with different suction lifts and for each setting of the pump, the head v/s discharge and efficiency v/s discharge characteristics are obtained. In each case it will be observed that with the decrease in the head there is only discharge increases upto a certain value of the head. With further reduction in the head there is an slight increase in the discharge till a certain value of discharge is reached, when further reduction in head causes no increase in the discharge. In other words, at a certain value of discharge as well as abrupt reduction in the head developed by the pump. Further corresponding to this discharge abrupt reduction in the efficiency of the pump also occurs. The abrupt reduction in the head as well as the efficiency of the pump indicates the inception of the cavitation. As such the value of the discharge at which there is an abrupt reduction in the head as well as the efficiency of the pump is the maximum discharge which the pump is able to provide without cavitation. For the different settings of the pump the values of such maximum discharges are determined and corresponding to each of these discharges the value of NPSH is also computed by Eq. 24.49, which thus represents the required NPSH for each setting of the pump. Usually the values of the required NPSH are plotted against the corresponding values of the discharge to obtain a curve of required NPSH for the pump. Further if a pump is required to be operated at different speeds then by adopting the same procedure, for each of the different speeds a curve for required NPSH may be obtained.

24.19 CAVITATION IN CENTRIFUGAL PUMPS

If the pressure at the suction side of the pump drops below the vapour pressure of the liquid then the cavitation may occur. The cavitation in a pump can be noted by a sudden drop in efficiency and head.

24.19.1 Thoma's Cavitation Factor

As in the case of turbines, for pumps also, Thoma's cavitation factor is used to indicated whether cavitation will occur. For pumps, Thoma's cavitation factor is defined as

$$\sigma = \frac{[(p_a - p_v)/w](h_s + h_{fs})}{H_m} \quad ...(24.52)$$

By comparing Eq. 24.52 with 24.50 and 24.51, cavitation factor may also be expressed as

$$\sigma = \frac{\text{NPSH}}{H_m} = \frac{H_{sv}}{H_m} \quad ...(24.53)$$

The cavitation will occur if the value of σ is less than the critical value, s_c at which the cavitation just begins. The critical value σ_c is that at which a drop in the efficiency or head or some other property begins. The value of σ_c for any specified operating conditions depends upon the design of the particular pump, and in any important installation it should be determined experimentally from the model tests. However, σ_c depends on the specific speed of a pump and for the pumps of normal design the following empirical relation may be used to determine σ_c.

$$\sigma_c = 0.103\,(N_s/1000)^{4/3} \quad ...(24.54)$$

where N_s is the specific speed of the pump defined by Eq. 24.36.

Equation 24.52 indicates that the value of σ can be increased by reducing the suction lift h_s. In some cases, especially when hot liquids are to be handled, the pump may have to be installed either at the liquid surface or even below the liquid surface in the sump. In the first case, $h_s = 0$, but in the second case, $h_s < 0$, which indicates that there is positive pressure at the inlet of the pump. A higher value of σ also ensures that there is sufficient NPSH available.

24.10.2 Suction Specific Speed

In addition to Thoma's cavitation factor, suction specific speed is another cavitation parameter, which is commonly used in the case of pumps to indicate whether the cavitation will occur. The *suction specific speed S* may be defined as the speed in r.p.m. of a geometrically similar pump which under corresponding conditions would deliver 1 litre of liquid per second when the total suction head is 1 m (in absolute units). According to this definition the expression for suction specific speed may be obtained by replacing the manometric head H_m in Eq. 24.36 for the specific speed by the total suction head H_{sv}. Thus

$$S = \frac{N\sqrt{Q}}{H_{sv}^{3/4}} \quad ...(24.55)$$

Further by combining Eqs 24.36, 24.53 and 24.55, the following relationship between σ, N_s and S may be obtained

$$\sigma = \left(\frac{N_s}{S}\right)^{4/3} \quad ...(24.56)$$

For most of the centrifugal and propeller pumps for cavitation free flow the limiting value of the suction specific speed have been found to lie between 4700 and 6700 when S is expressed in [r.p.m./(litres/second)$^{1/2}$/m$^{3/4}$] units.

By having the same value of the Thoma's cavitation factor σ or the suction specific speed S for the model and the prototype pumps the similarity in respect of cavitation can be established. However for the pumps the suction specific speed is a more pertinent cavitation parameter for ensuring similarity in the model and the prototype pumps in respect of cavitation. This is so because although both σ and S depend on the inlet flow conditions, σ also depends on the specific speed N_s of the pump. As such a constant value of σ can be obtained only if in addition to the inlet flow conditions, the N_s is also held constant and only then it will ensure similar cavitation conditions. On the other hand suction specific speed is independent of the specific speed of the pump and therefore a constant value of S alone will ensure similar cavitation conditions in the model and the prototype pumps. For instance, if the specific speed of pump is changed by changing the outside diameter of the impeller, leaving the inlet flow conditions unchanged, it can be assumed that the cavitation conditions remain the same. The suction specific speed S will not change since the inlet flow conditions remain the same, but σ will change since its value depends on the head H and hence on the outer diameter of the impeller. As such even if the value of σ is not same, as long as the value of S is same for the model and prototype pumps, similar cavitation conditions can be ensured.

24.20 COMPUTATION OF THE TOTAL HEAD OF PUMPING—SYSTEM HEAD CURVES

The total head of pumping consists of the following components.

(i) Static head H_s, which is the vertical distance between the liquid surfaces in the sump and the tank to which the liquid is delivered by the pump.

(ii) Velocity head $(V^2/2g)$ at the point of discharge.

(iii) Head loss H_L, which includes the head loss due to friction in suction and delivery pipes as well as head loss in valves, bends and all such appurtenances both on the suction and the delivery sides.

Thus the total head H against which the pump has to operate is given by

$$H = H_s + H_L + \frac{V^2}{2g} \quad ...(24.57)$$

With the pump running, if the discharge of the pump is more than the inflow, the level of the liquid in the sump would keep falling. By this the suction head component in the total head would keep increasing. Reverse will be the case when the inflow is more than the discharge by the pump.

By varying the size of the suction and delivery pipes in a pumping system the velocity of flow would vary which in turn would cause a change in the velocity head which varies as the square of the velocity of flow. This also causes a variation in the frictional losses because these also vary as the square of the velocity of flow or flow rate.

Thus the total head of pumping varies with the flow rate or discharge of the pump. At the stage of planning it is therefore necessary to compute the total head of pumping over a range of flow rates or discharges, for different variations in the levels of liquid in the sump and for different options of piping sizes and layouts. The values of the total heads of pumping so computed are plotted against the flow rates or discharges. A plot of total head of pumping versus flow rate of discharge is known as *system head curve* for the pumping system. Figure 24.16 shows a typical system head curve for a pumping system with a given static head and a given piping size and layout.

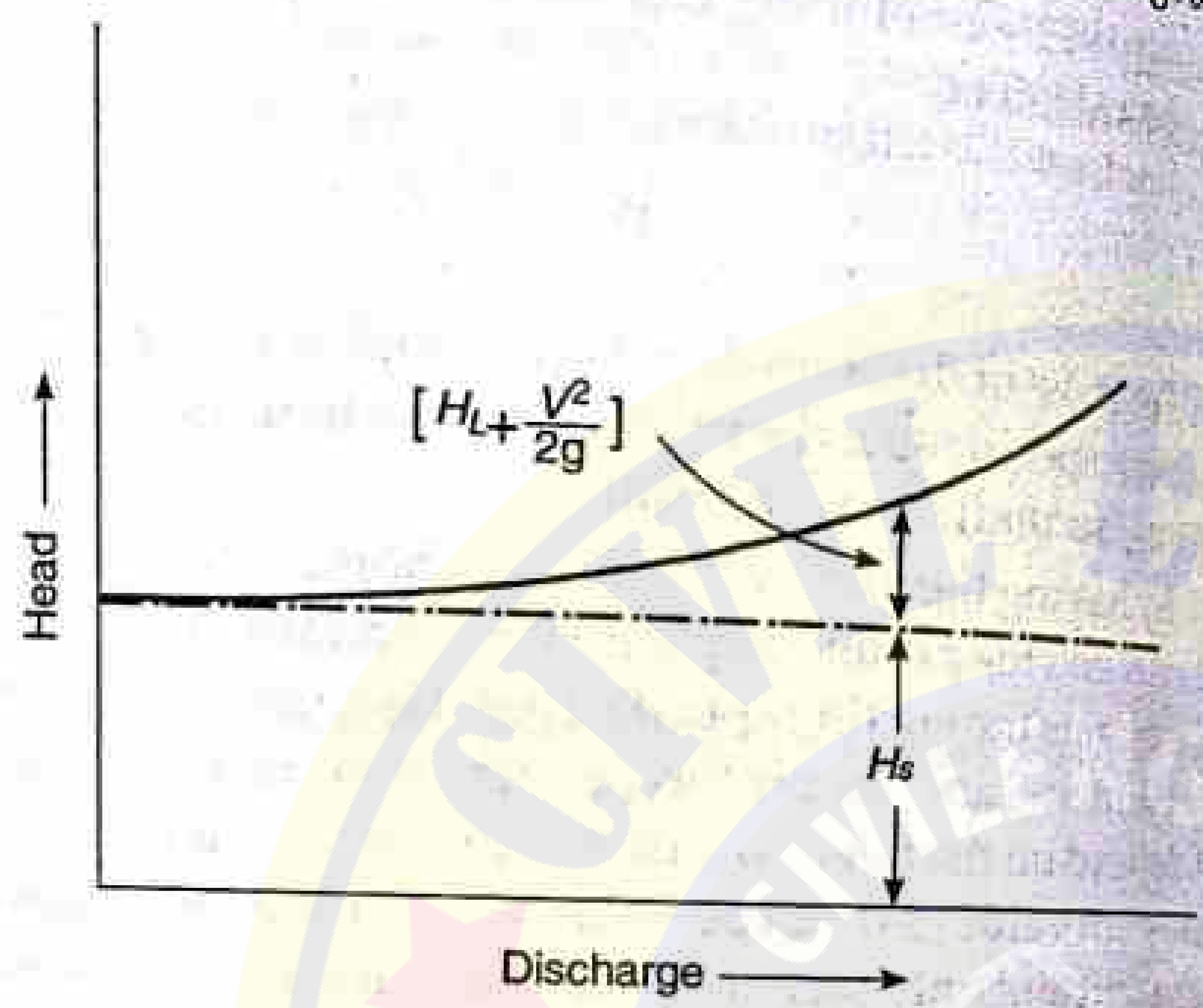


Figure 24.16 *System head curve for a pumping system.*

With an increase only in the static head the new system head curve will be a curve shifted parallel upwards as shown in Fig. 24.17. Further for a smaller size of piping the parabolic portion in the system head curve will be steeper as shown in Fig. 24.18.

From the system head curves for a pumping system with variations in static levels and different piping sizes and layouts, one can know what the total head would be for the most average operating conditions, which then can be specified as the total head of pumping.

24.20.1 Operating Point or Operating Range of a Centrifugal Pump

The *operating point* of a centrifugal pump is the point of intersection of the system head curve with the head-discharge (H v/s Q) characteristic curve of the pump. Thus for obtaining the operating point the

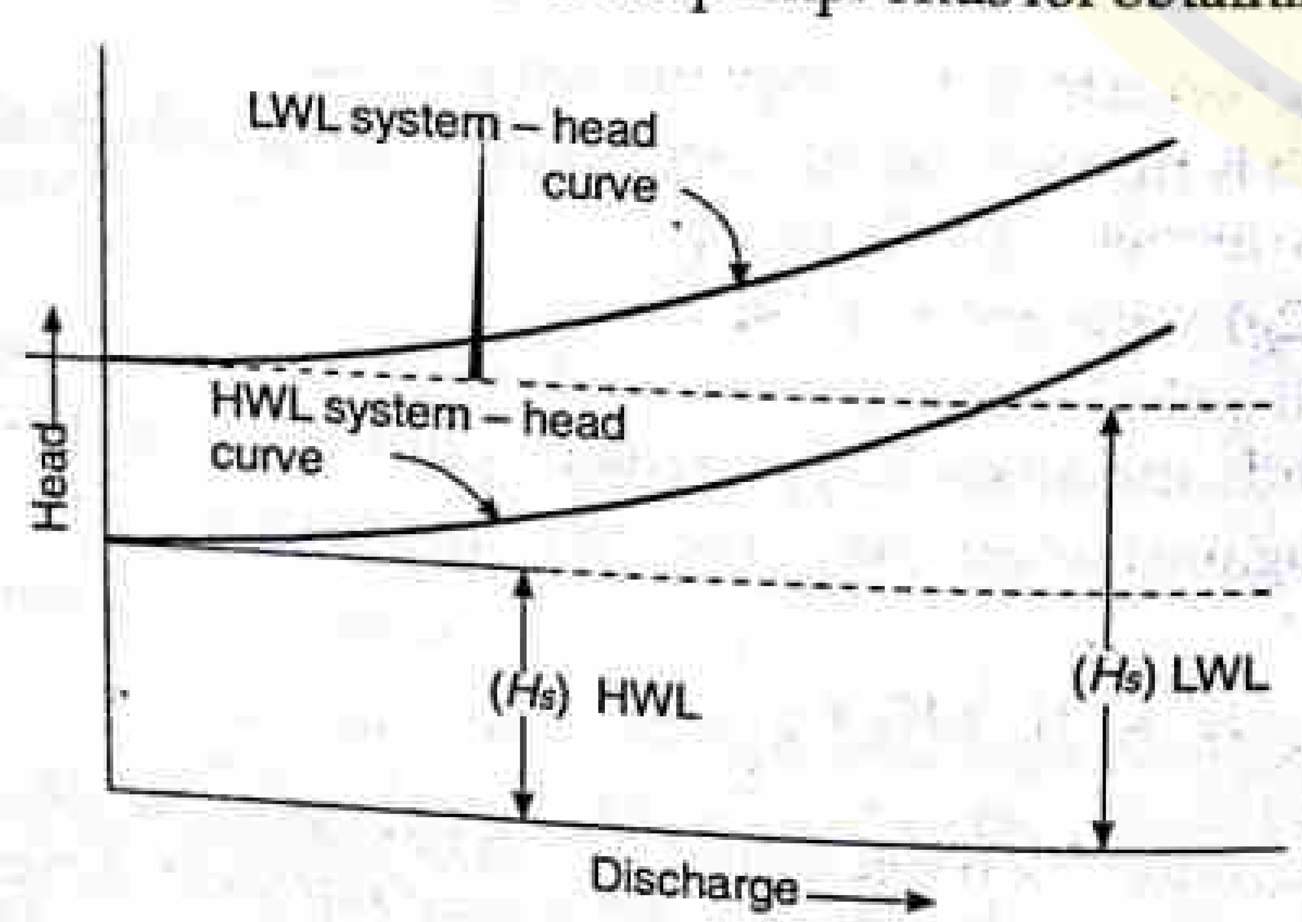


Figure 24.17 *System head curves for low water level (LWL) and high water level (HWL) in the sump*

system head curve and the head-discharge characteristic curve of the pump are plotted on the same sheet as shown in Fig. 24.19. The head-discharge characteristic curve of a centrifugal pump is a drooping parabola with the pump discharge being less when the head is more. When the pump is put into a system it meets the head as demanded by the system, which is as per the system head curve. At the point of intersection *P* of the system head curve with the head discharge curve of the pump, since the head demanded by the system for a certain flow rate or discharge is equal to the operating head of the pump at the same flow rate or discharge, point *P* represents the operating point of the pump.

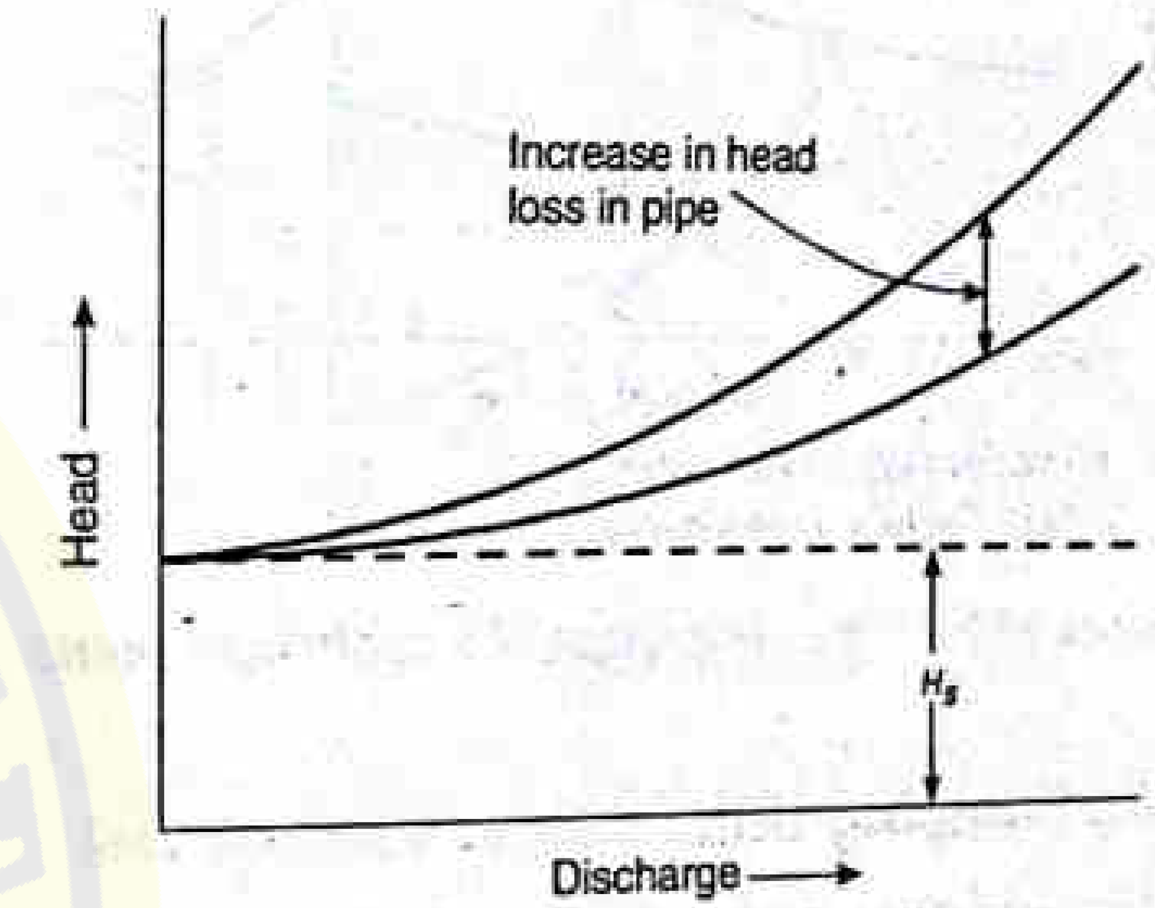


Figure 24.18 *System head curves with change in pipe sizes*

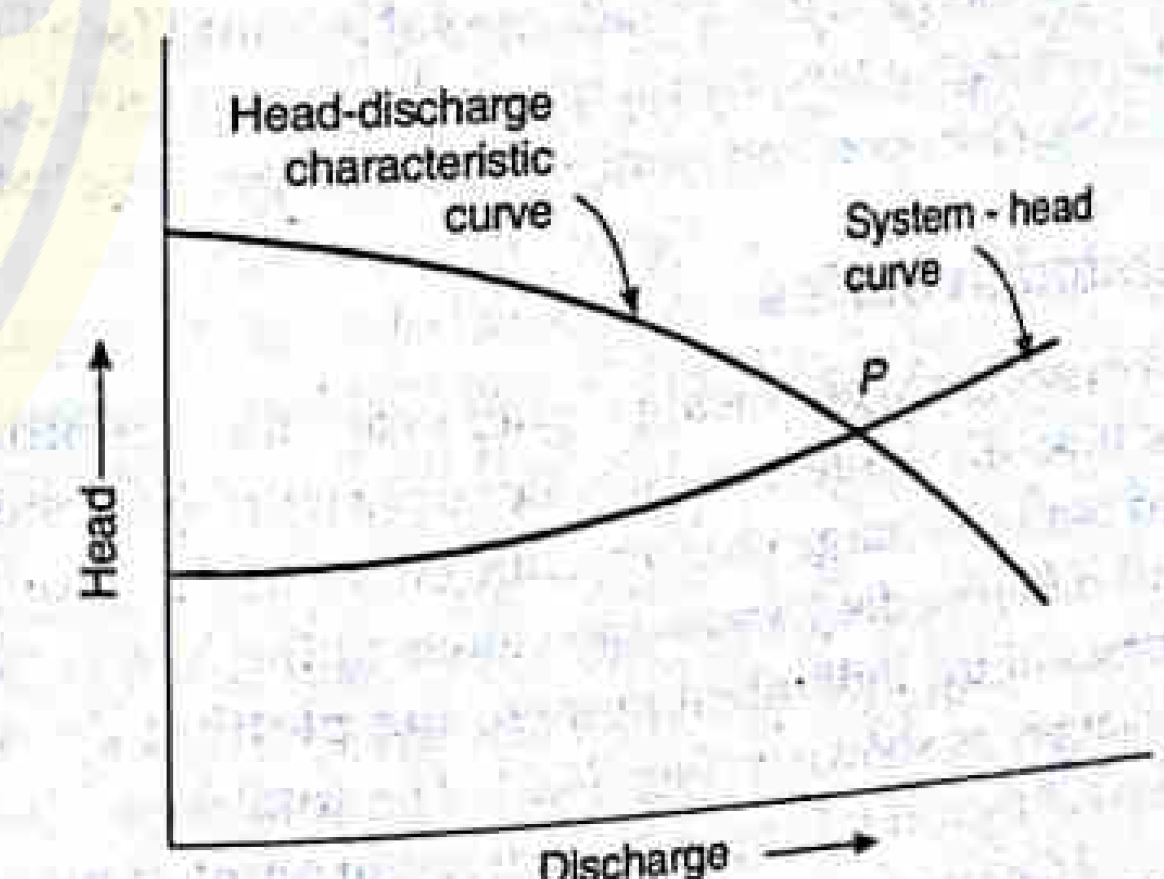
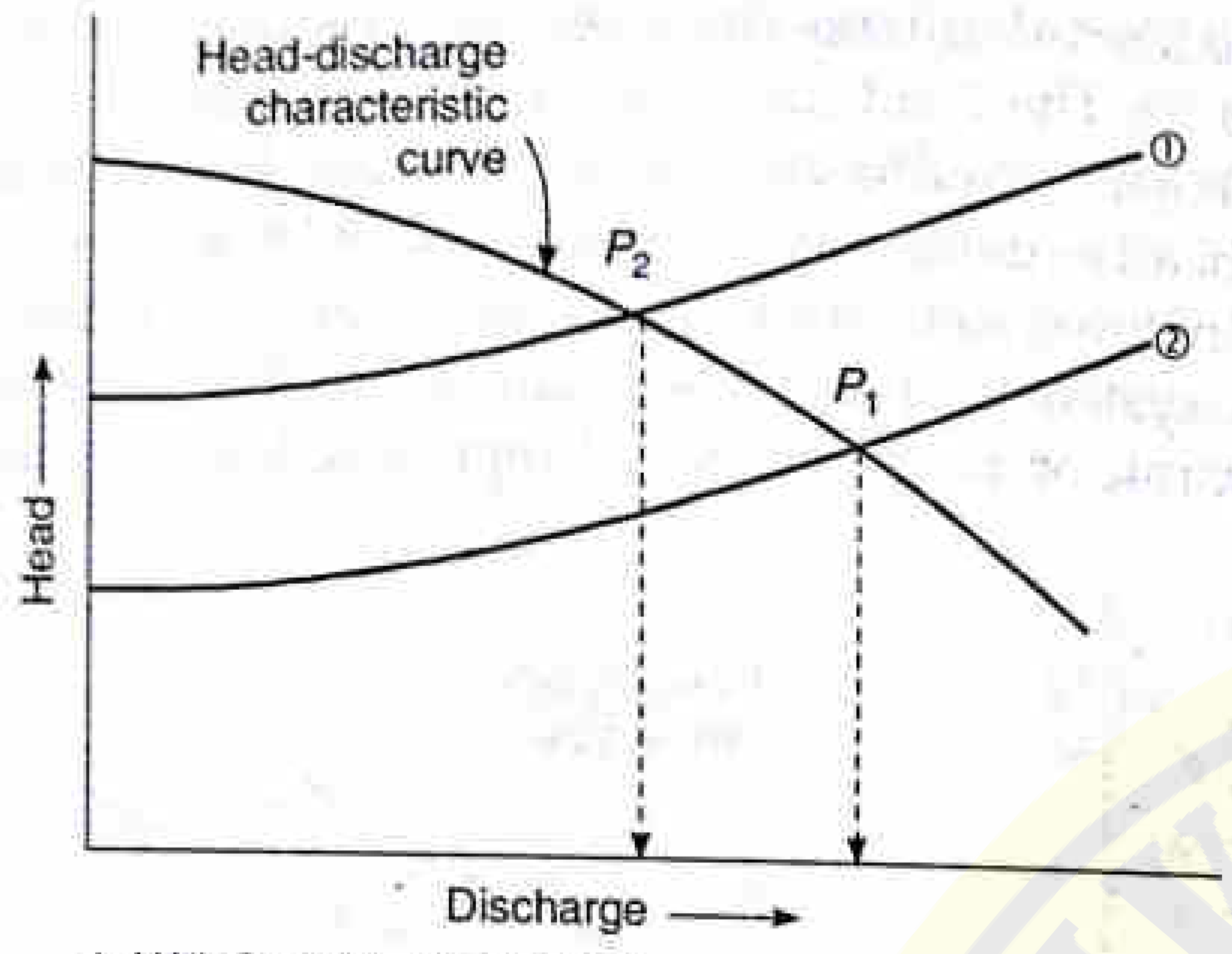


Figure 24.19 *Operating point of a centrifugal pump*

As indicated earlier the system head curve of a pumping system will change due to any change made in the system. If the system head curve changes the operating point will shift. The most significant change that may occur in a pumping system is due to variation in the level of liquid in the sump. There will be different system head curves corresponding to low water level (LWL) and high water level (HWL) in the sump as shown in Fig. 24.20. Thus if the level of liquid in the sump depletes during pumping from HWL to LWL, the operating point of the pump would vary from a low-head and high-discharge point P_1 to high-head and low-discharge point P_2 as shown in Fig. 24.20. The range between points P_1 and P_2 is known as the *operating range* of the pump due to fluctuations in the level of liquid in the sump.



1. LWL System – head curve
2. HWL System – head curve

Figure 24.20 *Operating range of a centrifugal pump.*

24.20.2 Selection of a Pumping Unit

With a given or known system head curve it is required to select a pump to deliver the anticipated flow at the required height. For this the system head curve and the head-discharge characteristic curve of the pump are plotted on the same sheet and the operating point is found (see Fig. 24.19). The operating point P gives the head and discharge or flow rate at which the pump will be operating. As such the pump selected should be such that the operating point is as close as possible to the best efficiency point.

24.20.3 Pumps Operated in Series

When pumps are operated in series for producing higher head, the operating point is the point of intersection of the combined head-discharge characteristic curve with the system head curve as shown in Fig. 24.21. The combined head-discharge characteristic curve of the pumps operated in series is obtained by reading against different discharges, the values of the heads obtained from the head-discharge characteristic curves of the individual pumps, and plotting the sum of the head values against the respective discharges as shown in Fig. 24.21. The total head is thus equal to the head added by each pump corresponding to discharge at the operating point. It may, however, be noted that since the point of intersection of the system head curve with the combined head-discharge characteristic curve is at a discharge larger than that at the point of intersection of system head curve with the head-discharge characteristic curve of a single pump, the head developed by each of the pumps when operating in series at the point of operation gets reduced. Thus as shown in Fig. 24.21 when only one pump is operating the head at the operating point P_1 is H_1, and when two identical pumps are operated in series the point of operation gets shifted upwards to P_2 corresponding to which the total head developed is H_2, but $H_2 < 2H_1$. It means that to double the head it is not adequate to commission two identical pumps in series operation. For this one must study what combination of pumps with different head-discharge characteristic curves can give such a combined head-discharge characteristic curve which will have its intersection with the system head curve at a point having total head as desired.

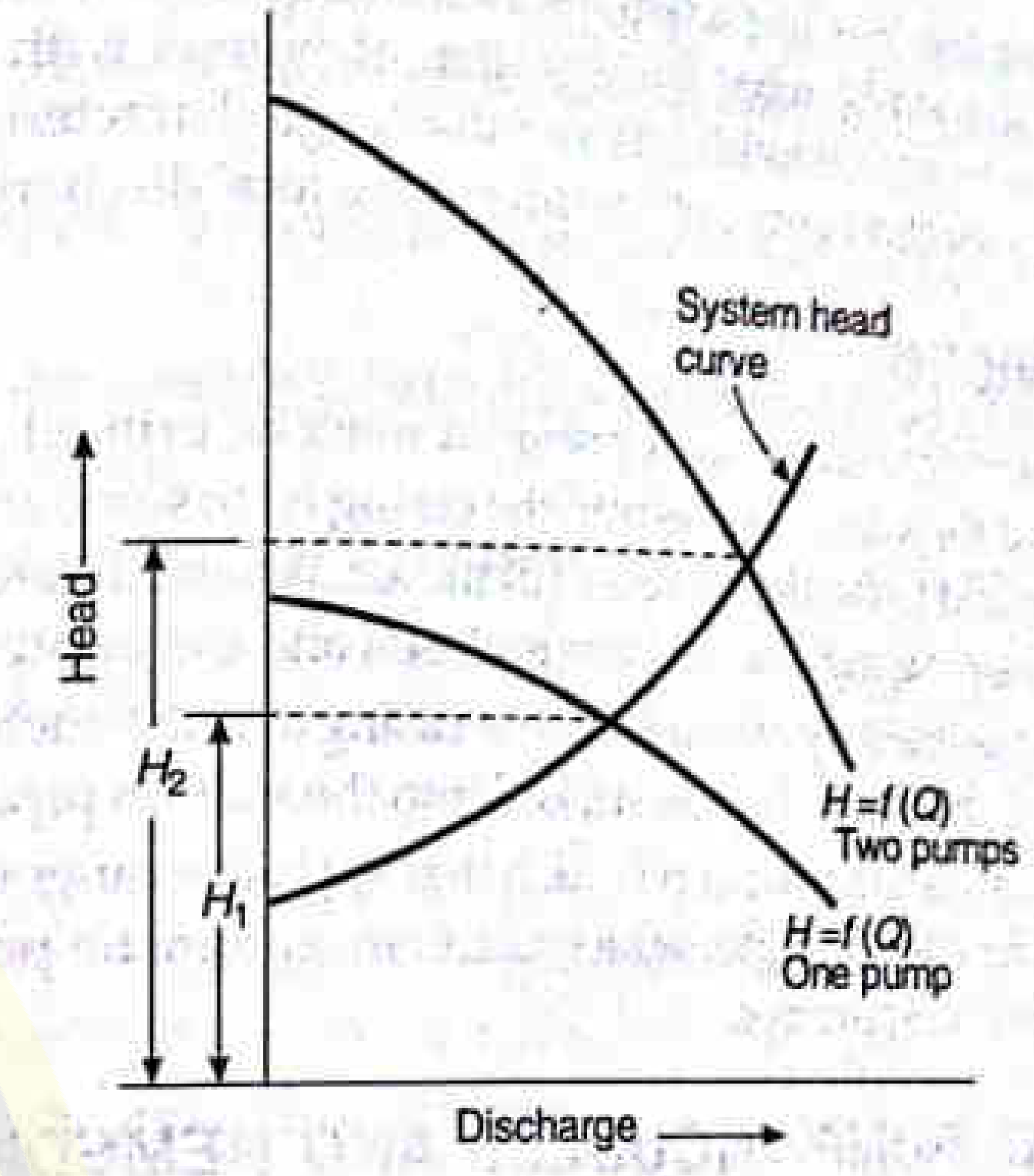


Figure 24.21 *Head-discharge characteristic curves for pumps operated in series*

24.20.4 Pumps Operated in Parallel

When pumps are operated in parallel for deliverying relatively large quantity of liquid against relatively small heads, the operating point is again the point of intersection of the combined head-discharge characteristic curve with the system head curve as shown in Fig. 24.22. The combined head-discharge characteristic curve of the pumps operated in parallel is obtained by reading against different heads, the values of the discharges obtained from the head-discharge characteristic curves of the individual pumps, and plotting the sum of the discharge values against the respective heads, as shown in Fig. 24.22. The total discharge is thus equal to the discharge added by each pump corresponding to head at the operating point. It may, however, be noted that since the point of intersection of the system head curve with the combined head-discharge characteristic curve is at a head higher than that at the point of intersection of system head curve with the head-discharge characteristic curve of a single pump, the discharge of each of the pumps when operating in parallel at the point of operation gets reduced. Thus as shown in Fig. 24.22 when only one pump is operating the discharge at the operating point P_1 is Q_1, and when two identical pumps are operated in parallel the point of operation gets shifted upwards to P_2 corresponding to which the total discharge is Q_2, but $Q_2 < 2Q_1$. It means that to double the

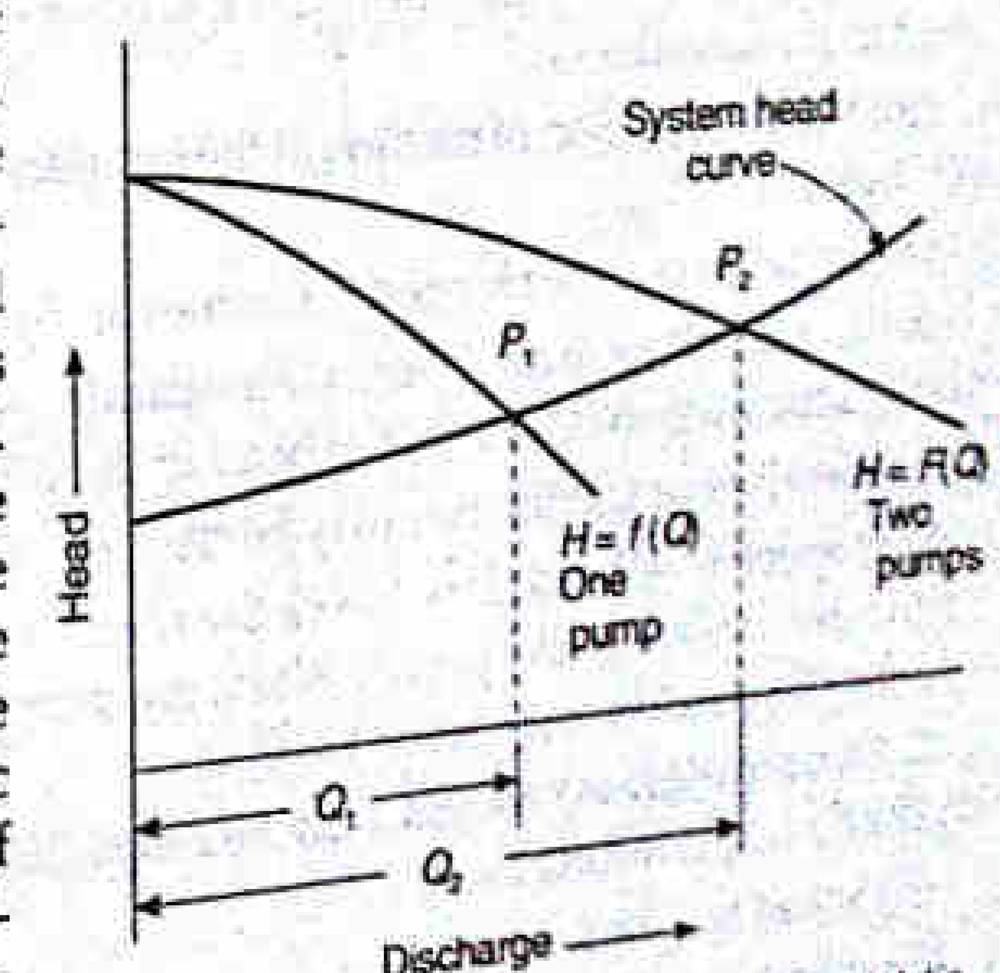


Figure 24.22 *Head-discharge characteristic curves for pumps operated in parallel*

discharge capacity of pumping, it is not adequate to commission two identical pumps in parallel operation. For this one must study what combination of pumps with different head-discharge characteristic curves can give such a combined head-discharge characteristic curve which will have its intersection with the system head curve at a point having total discharge as desired.

24.21 PRIMING DEVICES

As stated earlier, before starting a centrifugal pump, it must be primed. Small pumps are usually primed by pouring the liquid through a funnel into the casing from some external source. The air vent provided in the casing is opened to facilitate the exit of the air. When all the air has been removed from the suction pipe and the pump casing, the air vent is closed and the pump is primed.

Large pumps are usually primed by evacuating the casing and the suction pipe with the aid of an air pump or a stream ejector, the liquid is thus sucked into the suction pipe from the sump.

In some pumps, their internal construction is such that special arrangements containing a supply of liquid are provided in the suction pipe, which facilitate automatic priming of the pump. Such pumps are known as 'self priming pumps'.

24.22 CENTRIFUGAL PUMP-TROUBLES AND REMEDIES

Some of the troubles commonly experienced during the operation of the centrifugal pumps and the remedial measures to be taken for the same are as listed below:

1. *Pump fails to start pumping*. (i) Pump is probably not properly primed. Reprime the pump, opening the air vent until a steady and unbroken stream of liquid is obtained.

(ii) Total static head is probably much higher than that for which the pump is designed. Check the same with accurate vacuum and pressure gages, or may be determined by actual measurement the difference in elevation between pump and suction liquid level, and also between pump and point of discharge. Add to this static head the loss of head due to friction in pipes and that in the other fittings used in the installation.

(iii) Wrong direction of rotation of the impeller. Arrow on the pump case shows the proper direction of rotation.

(iv) Impeller may be clogged. Examine carefully for solids or foreign matter lodged in the impeller.

(v) Suction lift too high. Check with vacuum gage or by actual measurement where possible, adding to the static suction lift the loss of head in the pipe and fittings.

(vi) Strainer or suction line may be clogged. This would cause an excessive suction lift.

(vii) Speed may be too low. Check the speed with a teacheometer and compare it with the one given on the name plate of the pump. When the pump is being driven by electric motor, check up to see whether the voltage may not be too low.

2. *Pump working but not upto capacity and pressure*. (i) Air may be leaking into pump through suction line or through stuffing boxes.

(ii) Speed may be too low.

(iii) Discharge head may be higher than anticipated.

(iv) Suction lift may be too high.

(v) Foot valve or end of suction pipe may not have sufficient submergence, or it may be partly clogged or entirely too small.

(vi) Impeller may be partly clogged or too small in diameter.

(vii) Rotation may be in the wrong direction.

(viii) Wearing rings may be worn, impeller may be damaged, shaft may be loose, stuffing box packing may be defective.

(ix) If hot or volatile liquids are being pumped, there may not be sufficient positive head on the suction.

3. *Pump starts and then stops pumping*. (i) Improperly primed or leaky suction line.

(ii) Air pockets in suction line.

(iii) Air may be entering suction pipe because of liquid being delivered in suction tank or sump too near the pump suction line.

(iv) Suction lift may be too high.

4. *Pumps take too much power*:

(i) Speed may be too high.

(ii) Head may be too low and pump delivers too much liquid.

(iii) Liquid may have too high a specific gravity.

(iv) Pump may be operating in wrong direction.

(v) Shaft may be bent, impeller may be rubbing on casing, stuffing boxes may be too light, wearing rings may be worn.

ILLUSTRATIVE EXAMPLES

Example 24.1. *A centrifugal pump has an impeller 0.5 m outer diameter and when running at 600 r.p.m. discharges water at the rate of 8000 litres/minute against a head of 8.5 m. The water enters the impeller without whirl and shock. The inner diameter is 0.25 m, and the vanes are set back at outlet at an angle of 45° and the area of flow which is constant from inlet to outlet of the impeller is 0.06 m². Determine (a) the manometric efficiency of the pump, (b) the vane angle at inlet, and (c) the least speed at which the pump commences to work.*

Solution

$$V_f = V_{f_1}$$

$$= \frac{8000 \times 10^{-3}}{60 \times 0.06} = 2.22 \text{ m/s}$$

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 0.25 \times 600}{60} = 7.85 \text{ m/s}$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$= \frac{\pi \times 0.50 \times 600}{60} = 15.71 \text{ m/s}$$

From the inlet velocity triangle of Fig. 24.5

$$\tan\theta = \left(\frac{V_f}{u}\right)$$

$$= \left(\frac{2.22}{7.85}\right) = 0.2828$$

$\therefore$ $\theta = 15° 47'$.

From the outlet velocity triangle

$$V_{w1} = \left[u_1 - \left(\frac{V_{f_1}}{\tan 45°}\right)\right]$$

$$= (15.71 - 2.22)$$

$$= 13.49 \text{ m/s}$$

From Eq. 24.18

$$\eta_{mano} = \frac{gH_m}{v_{w_1}u_1}$$

$$= \frac{9.81 \times 8.5}{13.49 \times 15.71} = 0.3935 \text{ or } 39.35\%$$

For the least speed, from Eq. 24.27

$$\left[\left(\frac{\pi N}{60}\right)^2 (D_1^2 - D^2)\right] = 2gH_m$$

or $$\left(\frac{\pi N}{60}\right)^2 [(0.50)^2 - (0.25)^2] = (2 \times 9.81 \times 8.5)$$

$\therefore$ $N = 570$ r.p.m.

Example 24.2. *A centrifugal pump has to discharge 225 litres of water per second against a head of 25 m when the impeller rotates at a speed of 1 500 r.p.m. Determine (a) the impeller diameter, and (b) the vane angle at the outlet edge of the impeller. Assume that $h_{mano} = 0.75$; the loss of head in pump in metres due to fluid resistance is $0.03\,V_1^2$, where V_1 m/s is the absolute velocity of water leaving the impeller, the area of the impeller outlet surface is $(1.2\,D_1^2)$ m², where D_1 is the impeller diameter in m, and water enters the impeller without whirl.*

Solution

$\eta_{mano} = (gH/V_{w1}u_1)$; or $0.75 = [25 \times g)/V_{w1}u_1]$

$\therefore$ $[(V_{w1}u_1)/g] = 33.33$ m ...(i)

From Eq. 24.6, the manometric head is $H_m = (V_{w1}u_1/g)$ – losses of head in the pump.

$\therefore$ Losses in the pump

$$= (33.33 - 25) = 8.33 \text{ m}$$

Thus $0.03\,V_1^2 = 8.33$

$\therefore$ $V_1 = 16.66$ m/s

$$V_{f1} = \frac{Q}{A} = \frac{225 \times 10^{-3}}{1.2 \times D_1^2} = \frac{0.188}{D_1^2} \text{ m/s}$$

$$u_1 = [(\pi D_1 N)/60] = 78.54\,D_1 \text{ m/s}$$

Substituting the above value of u_1 in (i)

$$V_{w1} = \frac{33.33 \times 9.81}{78.54 D_1} = \frac{4.16}{D_1} \text{ m/s}$$

Now from the outlet velocity triangle of Fig. 24.5

$$V_{f1} = \left(V_1^2 - V_{w_1}^2\right)^{1/2}$$

or $(0.188/D_1^2) = [(16.66)^2 - (4.16/D1)^2]^{1/2}$

or $278 D_1^4 - 17.31\,D_1^2 - 0.035 = 0$

$\therefore$ $D_1 = 0.253\,4$ m or 253.4 mm.

Thus $V_{f1} = 2.93$ m/s; $u_1 = 19.90$ m/s

and $V_{w1} = 16.42$ m/s

$\tan\phi = [V_{f1}(u_1 - V_{w1})] = [2.93/(19.90 - 16.42)]$

$= 0.8420$

$\therefore$ $\phi = 40°6'$.

Example 24.3. *Find the power required to drive a centrifugal pump which delivers 40 litres of water per second to a height of 20 m through a 150 mm diameter and 100 m long pipeline. The overall efficiency of pump is 70% and Darcy's $f = 0.06$ for the pipeline. Assume inlet losses in suction pipe equal to 0.33 m.*

Solution

$Q = 40$ l/s $= 0.04$ m³/s ; $H_s = (h_s + h_d) = 20$ m;

$d_s = d_d = 150$ mm $= 0.15$ m ; $h_{in} = 0.33$ m ; $\eta_0 = 70\%$

$\therefore$ $$V_s = V_d = \frac{0.04}{(\pi/4)(0.15)^2} = 2.264 \text{ m/s}$$

and $$\frac{V_d^2}{2g} = 0.26 \text{ m}$$

$$(h_{fs} + h_{fd}) = \frac{fLV^2}{2gd}$$

or $$(h_{fs} + h_{fd}) = \frac{0.06 \times 100 \times (2.264)^2}{2 \times 9.81 \times 0.15} = 10.45 \text{ m}$$

The manometric head H_m is given by

$\therefore$ $$H_m = h_s + h_d + h_{fs} + h_{fd} + h_{in} + \frac{V_d^2}{2g}$$

$$H_m = (20 + 10.45 + 0.33 + 0.26)$$

$$= 31.04 \text{ m}$$

Power required to drive the pump is given by

$$P = \frac{wQH_m}{\eta_0}$$

$$\therefore \quad P = \frac{9810 \times 0.04 \times 31.04}{0.70}$$

$$= 17400 \text{ W} = 17.4 \text{ kW}$$

Example 24.4. *Find the rise in pressure in the impeller of a centrifugal pump through which water is flowing at the rate of 15 litres per second. The internal and external diameters of the impeller are 200 mm and 400 mm respectively. The widths of the impeller at inlet and outlet are 16 mm and 8 mm respectively. The pump is running at 1200 r.p.m. The water enters the impeller radially at inlet and impeller vane angle at outlet is 30°. Loss of head through the impeller may be assumed as 1.15 m*

Solution

$$Q = 15 \text{ l/s} = 0.015 \text{ m}^3/\text{s}$$
$$D = 200 \text{ mm} = 0.2 \text{ m}; D_1 = 400 \text{ mm} = 0.4 \text{ m}$$
$$B = 16 \text{ mm} = 0.016 \text{ m}; B_1 = 8 \text{ mm} = 0.008 \text{ m}$$
$$N = 1200 \text{ r.p.m}; \phi = 30°; h_{li} = 1.15 \text{ m}$$

$$u = \frac{\pi DN}{60}$$

$$= \frac{\pi \times 0.2 \times 1200}{60} = 12.57 \text{ m/s.}$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$= \frac{\pi \times 0.4 \times 1200}{60} = 25.13 \text{ m/s}$$

$$V_f = \frac{0.015}{(\pi \times 0.2 \times 0.016)} = 1.49 \text{ m/s}$$

$$V_{f1} = \frac{0.015}{(\pi \times 0.4 \times 0.008)} = 1.49 \text{ m/s}$$

i.e., $V_f = V_{f1}$

As indicated in Example 24.6, the pressure rise in the impeller is given by

$$\frac{p_1 - p}{w} = \left[\frac{1}{2g}\left(V_f^2 + u_1^2 - V_{f1}^2 \text{cosec}^2\phi\right)\right] - h_{li}$$

or

$$\frac{p_1 - p}{w} = \frac{1}{2 \times 9.81}\left[(1.49)^2 + (25.13)^2 - (1.49)^2(\text{cosec } 30°)^2\right] - 1.15$$

$$= 30.7 \text{ m}$$

Example 24.5. *A centrifugal pump has the following characteristics: outer diameter of impeller = 800 mm; width of impeller vanes at outlet = 100 mm; angle of impeller vanes at outlet = 40°. The impeller runs at 550 r.p.m and delivers 0.98 cubic metres of water per second under an effective head of 35 m. A 500 kW motor is used to drive the pump. Determine the manometric, mechanical and overall efficiencies of the pump. Assume water enters the impeller vanes radially at inlet.*

Solution

$$Q = 0.98 \text{ m}^3/\text{s}; D_1 = 800 \text{ mm} = 0.8 \text{ m}; N = 550 \text{ r.p.m};$$
$$B_1 = 100 \text{ mm} = 0.1 \text{ m}; \phi = 40°; H_m = 35 \text{ m}$$
$$P = 500 \text{ kW}$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$= \frac{\pi \times 0.8 \times 550}{60} = 23.04 \text{ m/s}$$

$$V_{f1} = \frac{Q}{\pi D_1 B_1}$$

$$= \frac{0.98}{\pi \times 0.8 \times 0.1} = 3.90 \text{ m/s}$$

$$V_{w1} = (u_1 - V_{f1} \cot \phi)$$
$$= (23.04 - 3.90 \cot 40°)$$
$$= 18.39 \text{ m/s}$$

Manometric efficiency is given by

$$\eta_{mano} = \frac{gH_m}{V_{w_1} u_1}$$

$$= \frac{9.81 \times 35}{18.39 \times 23.04} = 0.81 \text{ or } 81\%$$

Overall efficiency is given by

$$\eta_0 = \frac{wQH_m}{P}$$

$$= \frac{9810 \times 0.98 \times 35}{500 \times 10^3} = 0.67 \text{ or } 67\%$$

Mechanical efficiency is given by

$$\eta_{mech} = \frac{\eta_0}{\eta_{mano}}$$

$$= \frac{0.67}{0.81} = 0.83 \text{ or } 83\%.$$

Example 24.6. *Show that the pressure rise in the impeller of a centrifugal pump is given by $[V_f^2 + u_1^2 - V_{f1}^2 \operatorname{cosec}^2 \phi/2g]$ provided the frictional and other losses in the impeller are neglected.*

Solution

Applying Bernoulli's equation between inlet and outlet tips of the impeller

$$\frac{p}{w} + \frac{V^2}{2g} = \frac{p_1}{w} + \frac{V_1^2}{2g} - \frac{V_{w_1} u_1}{g}$$

For radial entry $V = V_f$, thus

$$\frac{p_1 - p}{w} = \frac{v_f^2}{2g} + \frac{V_{w_1} u_1}{g} - \frac{V_1^2}{2g}$$

From the outlet velocity triangle

$$V_{w1} = (u_1 - V_{f1} \cot \phi)$$

$$V_1^2 = V_{f_1}^2 + V_{w_1}^2 = \left[V_{f_1}^2 + (u_1 - \cot \phi)^2\right]$$

nce $(1 + \cot^2 \phi) = \operatorname{cosec}^2 \phi$

$$V_1^2 = \operatorname{cosec}^2\phi + u_1^2 - 2u_1 V_{f1} \cot \phi$$

Hence

$$\frac{p_1 - p}{w} = \frac{v_f^2}{2g} + \frac{u_1(u_1 - V_{f_1} \cot \phi)}{g} - \frac{V_{f_1}^2 \operatorname{cosec}^2\phi + u_1^2 - 2u_1 V_{f_1} \cot\phi}{2g}$$

$$\therefore \quad \frac{p_1 - p}{w} = \left[\frac{V_f^2 + u_1^2 - V_{f_1}^2 \operatorname{cosec}^2\phi}{2g}\right]$$

Example 24.7. *Prove that, in general, for a centrifugal pump running at speed N and giving a discharge Q, the manometric head is expressible in the form, $H_m = [AN^2 + BNQ + CQ^2]$ where A, B and C are constants.*

Solution

If the loss of head in the pump is neglected, the manometric head is given by the pressure rise through the impeller, together with a certain percentage of the kinetic head at the exit of the impeller which is recovered in the volute chamber or the diffuser ring.

As derived in Example 24.6.

$$\text{Pressure rise} = \frac{1}{2g}[V_f^2 + u_1^2 - \operatorname{cosec}^2 \phi]$$

Let the percentage of kinetic head recovered in diffuser ring $= (k V_1^2)$ m. Then it can be shown that

$$kV_1^2 = \frac{c}{2g}\left[V_{f_1}^2 + u_1^2 - 2u_1 V_{f_1} \cot\phi + V_{f_1}^2 \cot^2 \phi\right]$$

where c is a constant

$$\therefore \quad H_m = \frac{1}{2g}\left[u_1^2(1+c) - 2cu_1 V_{f_1} \cot\phi + V_{f_1}^2 \times (c + c\cot^2\phi - \operatorname{cosec}^2\phi) + V_f^2\right]$$

Assuming $V_f = V_{f1}$,

$$H_m = \frac{1}{2g}\left[Au_1^2 + Bu_1 V_{f_1} + CV_{f_1}^2\right]$$

Since $u_1 = (\pi D_1 N/60). \therefore u_1 \sim N$

and $V_{f1} = (Q/A_1); \quad \therefore V_{f_1}^2 \sim Q^2$

$$\therefore \quad H_m = \frac{1}{2g}\left[AN^2 + BNQ + CQ^2\right]$$

Example 24.8. *A centrifugal pump has an impeller 0.29 m diameter running at 960 r.p.m. with an effective outlet vane angle of 28°. The velocity of flow (assumed uniform throughout the system) is 2 m/s. The static suction lift is 2.8 m. The energy losses in metres of water are: in suction pipe, 0.61 m; in impeller, 0.49 m; in volute casing, 0.88 m. From these particulars calculate the readings of vacuum or pressure gages placed (i) at inlet to the pump, (ii) at impeller outlet (in clearance space between impeller and volute), (iii) at pump outlet or delivery flange, 0.2 m above the centerline of the pump.*

Solution

From Eq. 24.7 the pressure head at the inlet to the pump is given by

$$\frac{p_s}{w} = -\left[\frac{V_s^2}{2g} + h_s + h_{f_s}\right]$$

or

$$\frac{p_s}{w} = -\left[\left(\frac{2\times 2}{(2\times 9.81)}\right) + 2.8 + 0.61\right]$$

$$= -3.614 \text{ m}$$

From Eq. 24.8

$$\frac{p_s}{w} + \frac{v_s^2}{2g} + \frac{v_{w1} u_1}{g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_{li}$$

Now

$$u_1 = [(\pi D_1 N)/60] = [(\pi \times 0.29 \times 960)/60]$$
$$= 14.58 \text{ m/s}$$
$$V_{w1} = (u_1 - V_{f1} \cot \phi) = (14.58 - 2 \cot 28°)$$
$$= 10.82 \text{ m/s}$$
$$V_1 = \left(V_{w_1}^2 + V_{f1}^2\right)^{1/2} = [(10.82)^2 + (2)^2]^{1/2}$$
$$= 11.00 \text{ m/s}$$

Thus by substitution in the above eqution

$$-3.614 + 0.204 + \left(\frac{10.82 \times 14.58}{9.81}\right) = \frac{p_2}{w} + \left(\frac{11\times 11}{2\times 9.81}\right) + 0.49$$

$\therefore$ Pressure head at impeller outlet is

$$(p_2/w) = 6.01 \text{ m}$$

From Eq. 24.9

$$\frac{p_2}{w}+\frac{V_1^2}{2g} = \frac{p_d}{w}+\frac{V_d^2}{2g}+h+h_{ic}$$

or
$$6.01+\left(\frac{11\times 11}{2\times 9.81}\right) = \frac{p_d}{w}+\left(\frac{2\times 2}{2\times 9.81}\right)+0.20+0.88$$

or
$$\frac{p_d}{w} = 10.89 \text{ m}$$

i.e., the pressure head at pump outlet is 10.89 m.

Example 24.9. *A centrifugal pump lifts water against a static head of 40 m, of which 4 m is suction lift. The suction and delivery pipes are both 150 mm diameter; the head loss in the suction pipe is 2.3 m and in the delivery pipe, 7.4 m. The impeller is 420 mm diameter and 25 mm wide at the mouth; it revolves at 1200 r.p.m. and its effective vane angle at exit is 35°. If $\eta_{mano} = 82\%$ and $\eta_0 = 72\%$, determine the discharge delivered by the pump and power required to drive the pump. Also find the pressure head indicated at the suction and delivery branches of the pump.*

Solution

If
$$V_p = \text{Velocity in pipes,}$$

then discharge
$$Q = [\pi(0.15)^2/4]\,V_p = 0.0177\,V_p$$

Also
$$Q = \pi D_1 B_1 \times V_{f1} \text{ (neglecting the vane thickness)}$$

or
$$Q = (\pi \times 0.42 \times 0.025)\,V_{f1} = 0.0177\,V_p$$

or
$$V_{f1} = 0.537\,V_p \approx 0.54\,V_p$$

$$u_1 = (\pi D_1 N/60) = (\pi \times 0.42 \times 1200/60) = 26.39 \text{ m/s}$$

From Eq. 24.17, since $V_d = V_p$

$$H_m = (h_s+h_d)+h_{fs}+h_{fd}+\frac{V_p^2}{2g}$$

or
$$H_m = 40+2.3+7.4+\frac{V_p^2}{2g} = 49.7+\frac{V_p^2}{2g}$$

But
$$\eta_{mano} = \frac{gH_m}{V_{w_1}u_1};\ \text{or } 0.82 = \frac{9.81\times H_m}{u_1(u_1-V_{f_1}\cot\phi)}$$

or
$$H_m = \frac{(0.82\times 26.39)(26.39-0.54\,V_p\cot 35°)}{9.81}$$

Equating the two values of H_m, we have

$$49.7+\left(\frac{V_p^2}{2\times 9.81}\right) = \frac{(0.82\times 26.39)(26.39-0.54V_p\times 1.428)}{9.81}$$

Solving this equation, we get

$$V_p = 4.42 \text{ m/s}$$

Hence discharge

$$Q = (0.0177\times 4.42) = 0.078 \text{ m}^3/\text{s}$$

$$H_m = \left(49.7+\frac{(4.42)^2}{2\times 9.81}\right) = 50.70 \text{ m}$$

Shaft power
$$= \frac{wQH_m}{\eta_0} = \frac{9810\times 0.078\times 50.70}{0.72} = 53\,881 \text{ W} = 53.881 \text{ kW}$$

Since velocity head

$$= \frac{(4.42)^2}{2\times 9.81} = 0.996 \text{ m} \approx 1 \text{ m}$$

Thus at pump suction the pressure head is given by Eq. 24.7 as

$$\frac{p_s}{w} = -\left(\frac{V_s^2}{2g}+h_s+h_{f_s}\right) = -[1.0+4+2.3] = -7.3 \text{ m}$$

Similarly at pump delivery the pressure head is given by Eq. 24.13 as

$$\frac{p_d}{w} = \left(h_m+\frac{p_s}{w}\right) = (50.7-7.3) = 43.4 \text{ m.}$$

Example 24.10. *Water is pumped from a low level reservoir through a main pipe pipeline of 0.45 m diameter and 1400 m length. The pump is located at the low level reservoir. At a point along the main line at a distance of 450 m from the high level reservoir, a branch line of 0.3 m diameter and 360 m length takes off to discharge 180 l/s in the atmosphere.*

Level of water surface in high level reservoir	= + 30.00 m
Level of water in the open end of the 0.3 m diameter branch line	= 25.50 m
Level of water in the low level reservoir	= 18.00 m
Darcy's frictional coefficient for both pipes	= 0.032

Determine the flow rate into the high level reservoir and the theoretical power of the pump assuming the delivery valve of the pump to be at + 20.00 m.

Solution

Let Q be the total discharge delivered by the pump; Q_1 be the discharge through the branch pipe delivered in the atmosphere; and Q_2 be the discharge delivered to the high level reservoir.

Thus by continuity equation, we have

$$Q = Q_1+Q_2$$

But $\quad Q_1 = 180\ l/s = 0.18\ m^3/s$

$Q = 0.18 + Q_2$

From Eqs. 24.7, 24.8 and 24.9, we have with usual notation

$$O = \frac{p_s}{w} + \frac{V_s^2}{2g} + h_s + h_{fs} \quad ...(i)$$

$$\frac{p_s}{w} + \frac{V_s^2}{2g} + \frac{V_{w_1}u_1}{g} = \frac{p_2}{w} + \frac{V_1^2}{2g} + h_{ii} \quad ...(ii)$$

$$\frac{p_2}{w} + \frac{V_1^2}{2g} = \frac{p_d}{w} + (20 - 18 - h_s) + h_{ic} + \frac{V_d^2}{2g} \quad ...(iii)$$

Adding Eqs. (ii), (iii) and (iv) we get ...(iv)

$$\frac{V_{w_1}u_1}{g} - h_{ii} - h_{ic} = \frac{p_d}{w} + \frac{V_d^2}{2g} + 2 + h_{fs}$$

or

$$H_m = \frac{p_d}{w} + \frac{V_d^2}{2g} + 2 + h_{fs} \quad ...(v)$$

Since the pump is located close to the low level reservoir, h_{fs} may be neglected. Thus from Eq. (v) we have

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = H_m - 2 \quad ...(vi)$$

Let h_f be the head loss due to friction in the main pipeline between the delivery point of the pump and the point where the branch line takes off; h_{f1} be the head loss due to friction in the branch line; and h_{f2} be the head loss due to friction in the main pipeline between the point where the branch line takes off and the high level reservoir.

Thus applying Bernoulli's equation between the delivery point of the pump and the high level reservoir, neglecting the minor losses we get

$$\frac{p_d}{w} + \frac{V_d^2}{2g} + 20 = 30 + h_f + h_{f2}$$

or

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = 10 + h_f + h_{f2} \quad ...(vii)$$

Similarly applying Bernoulli's equation between the delivery point of the pump and the open end of the branch line, neglecting the minor losses, we get

$$\frac{p_d}{w} + \frac{V_d^2}{2g} + 20 = 25.5 + h_f + h_{f1}$$

or

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = 5.5 + hf + h_{f1} \quad ...(viii)$$

From Eqs (vii) and (viii), we have

$$h_{f_1} - h_{f_2} = 4.5 \quad ...(ix)$$

The velocity of flow in the branch line is

$$V_1 = \frac{0.18}{(\pi/4)(0.3)^2} = 2.546\ m/s$$

$\therefore$

$$h_{f_1} = \frac{0.032 \times 360 \times (2.546)^2}{2 \times 9.81 \times 0.3} = 12.687\ m$$

Thus from Eq. (ix), we get

$$h_{f_2} = (12.687 - 4.5) = 8.187\ m$$

If V_2 is the velocity of flow in the main pipeline between the branch line and the high level reservoir, then

$$h_{f_2} = \frac{0.032 \times 450 \times V_2^2}{2 \times 9.81 \times 0.45}$$

or

$$8.187 = \frac{0.032 \times 450 \times V_2^2}{2 \times 9.81 \times 0.45}$$

$\therefore$

$$V_2 = 2.240\ m/s$$

Thus

$$Q_2 = \frac{\pi}{4} \times (0.45)^2 \times 2.240 = 0.356\ m^3/s$$

$\therefore$

$$Q = (0.18 + 0.356) = 0.536\ m^3/s$$

If V is the velocity of flow in the main pipeline between the delivery point of the pump and the point where the branch line takes off, then

$$V = \frac{0.536}{(\pi/4)(0.45)^2} = 3.370\ m/s$$

and

$$h_f = \frac{0.032 \times (1400 - 450) \times (3.370)^2}{2 \times 9.81 \times 0.45} = 39.104\ m$$

Thus from Eq. (vii), we have

$$\frac{p_d}{w} + \frac{V_d^2}{2g} = (10 + 39.104 + 8.187) = 57.291\ m$$

Thus from Eq. (vi), we have

$$H_m = (57.291 + 2) = 59.291\ m$$

Theoretical power of the pump is given by

in SI units

$$P = wQH_m$$

$$P = \frac{9810 \times 0.536 \times 59.291}{1000} = 311.762\ kW$$

In Metric units

$$P = \frac{1000 \times 0.536 \times 59.291}{75} = 423.733 \text{ h.p.}$$

Example 24.11. *A centrifugal pump running at 1400 r.p.m. has the characteristics as given below:*

Discharge Q Litres / second	12.5	18.7	25.0	31.3	37.5	43.8	50.0
Head H_m meters	28.3	27.4	26.4	25.0	23.4	20.7	18.0
η_0 (present)	65	70	73	74	72	69	63

Draw the operating characteristics of the pump and determine its specific speed. If the pump is driven by a motor, determine the power of the motor when operating at maximum efficiency.

Solution

The curves of head and efficiency are plotted on a base of discharge shown in the Fig. Ex. 24.11

From the plot, maximum $\eta_0 = 74.1\%$ and the corresponding $H_m = 25.3$ m and $Q = 30.4$ l/s.

$$\therefore \quad N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}$$

$$= \frac{1400 \times (30.4)^{1/2}}{(25.3)^{3/4}}$$

$$= 684.3$$

$$\eta_0 = \frac{(wQH_m)}{P}$$

$$\therefore \text{ Power} \quad P = \frac{(wQH_m)}{\eta_0}$$

$$= \frac{9810 \times (30.4 \times 10^{-3}) \times 25.3}{0.741}$$

$$= 10\,182 \text{ W} = 10.182 \text{ kW}$$

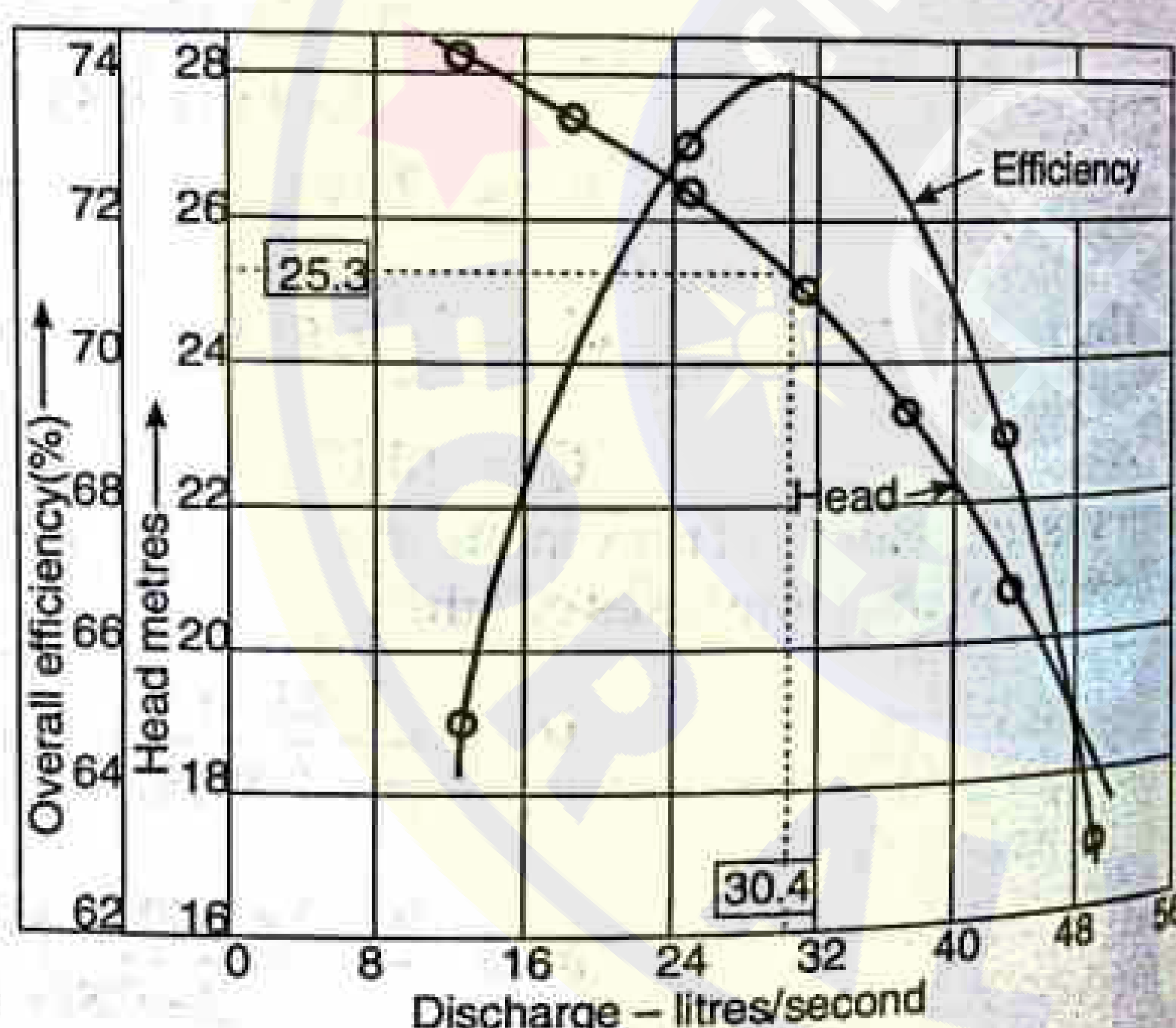


Figure Ex. 24.11

Example 24.12. *Determine the number of the impellers required for a multistage pump to lift 4 200 litres/minute against a total head of 185 m, at a speed of 750 r.p.m. The specific speed is not to exceed 700.*

Solution

The limiting N_s for each impeller = 700

Thus
$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}; \text{ or } 700 = \frac{750(4200/60)^{1/2}}{H_m^{3/4}}$$

$\therefore$ Head per stage $= H_m = 18.62$ m

$$\therefore \text{ Number of stages} = \frac{185}{18.62} = 10.$$

Example 24.13. *Water is to be pumped out of a deep well under a total head of 95 m. A number of identical pumps of design speed 1000 r.p.m. and specific speed 900 r.p.m., with a rated capacity of 150 litres/second are available. How many pumps will be needed and how should they be connected?*

Solution

If H_m is the manometric head developed by each pump then since

$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}; \; 900 = \frac{1000(150)^{1/2}}{H_m^{3/4}}$$

$$H_m = 32.5 \text{ m}$$

$\therefore$ Number of pumps needed

$$= \frac{95}{32.5} = 3$$

Since the total head required to be developed is more than the head developed by each pump, the pumps should be connected in series.

Example 24.14. *Two centrifugal pumps have the head and discharge characterstics as follows:*

Discharge Q litres/second	0	4	8	12	16	20	24
PUMP 1 Head H_{m_1} (metres)	50.0	51.8	50.8	48.0	42.5	32.5	18.3
PUMP2 Head H_{m_2} (metres)	46.7	45.9	44.2	40.3	34.3	26.0	17.0

Both pumps are installed together and are required to pump water through a pipe 150 mm diameter having f = 0.02. Calculate the heads under which pumps are working and discharge in litres per second pumped by them if:

(a) the pumps are connected in series : static lift is 65 m and the suction and the delivery pipes are 800 m long.
(b) the pumps are connected in parallel ; static lift is 15 m and the suction and the delivery pipes are 360 m long.

Solution

(a) When the pumps are connected in series then neglecting the velocity head in the delivery pipe, the total head required to be developed by both the pumps is

$$H_m = (h_s + h_d) + (h_{fs} + h_{fd})$$

or
$$H_m = 65 + \frac{0.02 \times 800 \times (Q \times 10^{-3})^2}{[\pi(0.15)^2/4]^2 \times 2 \times 9.81 \times 0.15} \quad ...(i)$$

or
$$H_m = 65 + \frac{Q^2}{57.44}$$

where Q is the discharge in litres/second. The computed values of H_m for different values of Q in respect of Eq. (i) are as given below:

Q	0	4	8	12	16	20	24
H_m	65.0	65.3	66.1	67.5	69.5	72.0	75.0

Curves of H_{m_1}, H_{m_2} and $(H_{m_1}+H_{m_2})$ are plotted on a base of Q as shown in the Fig. Ex. 24.14 (a). On the same graph H_m v/s Q curve in respect of the above noted values is also plotted. It may be seen that the curves of H_m and $(H_{m_1}+H_{m_2})$ intersect at a point corresponding to which $Q = 17.6$ litres/second; $H_{m_1} = 39.0$ m and $H_{m_2} = 31.5$ m.

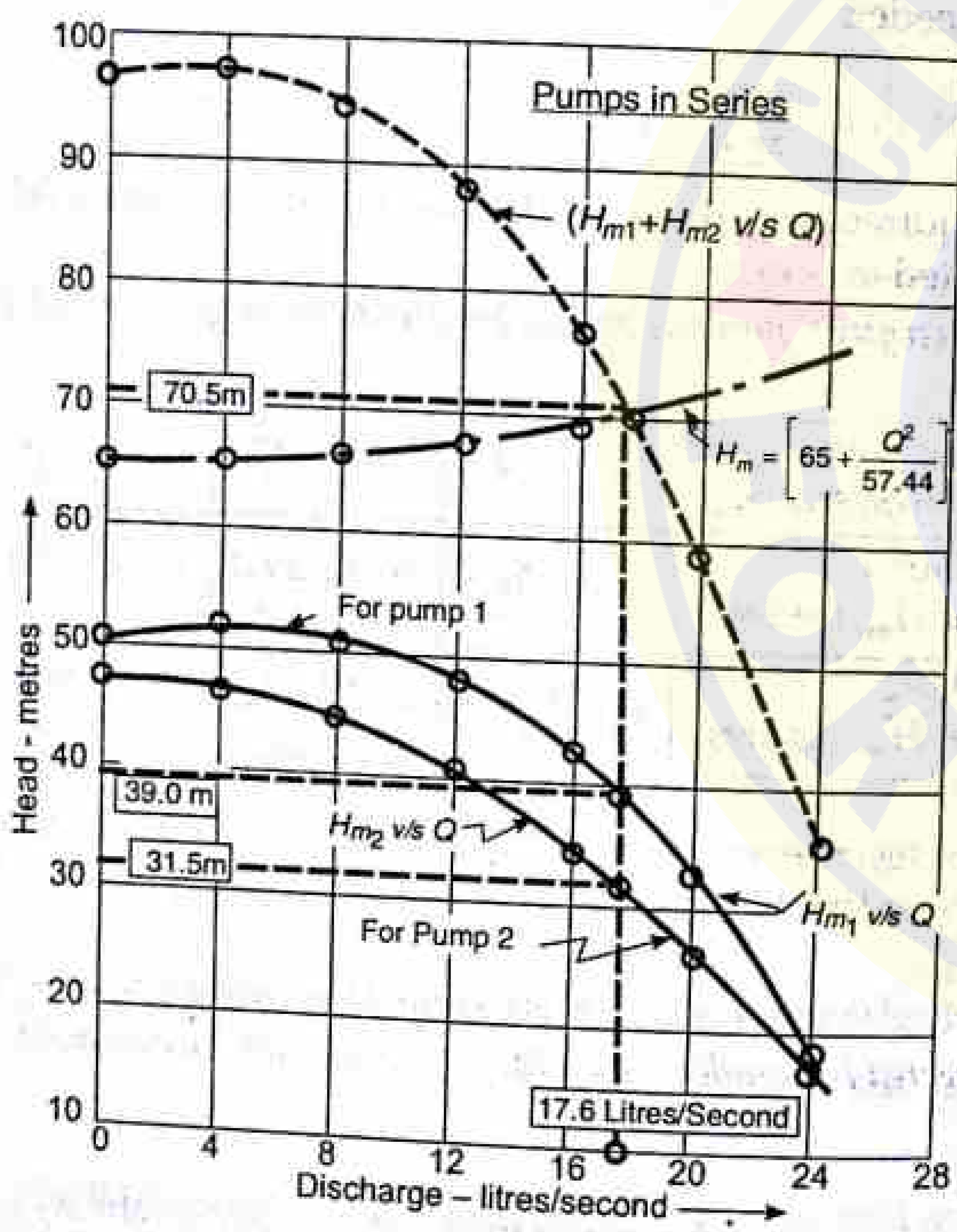


Figure Ex. 24.14 (a)

(b) Let Q_1 and Q_2 be the discharges in litres/second, flowing through the pumps 1 and 2 respectively which are connected in *parallel*. Then the total discharge flowing through the suction and delivery pipes $= (Q_1 + Q_2)$.

The total head required to be developed is

$$H_m = (h_s + h_d) + h_{f_s} + h_{f_d}$$

$$H_m = 15 + \frac{0.02 \times 360 \times [(Q_1+Q_2) \times 10^{-3}]^2}{[\pi(0.15)^2/4]^2 \times 2 \times 9.81 \times 0.15}$$

or

$$H_m = 15 + \frac{(Q_1+Q_2)^2}{127.64}$$

or ...(ii)

Curves of H_{m_1} and are H_{m_2} plotted on the base of Q as shown in Fig. 24.14 (b). From these curves for the same values of H_m, the values of Q_1, Q_2, and (Q_1+Q_2) are obtained which are as given below. Curve of (Q_1+Q_2) v/s H_m is also plotted on the same graph.

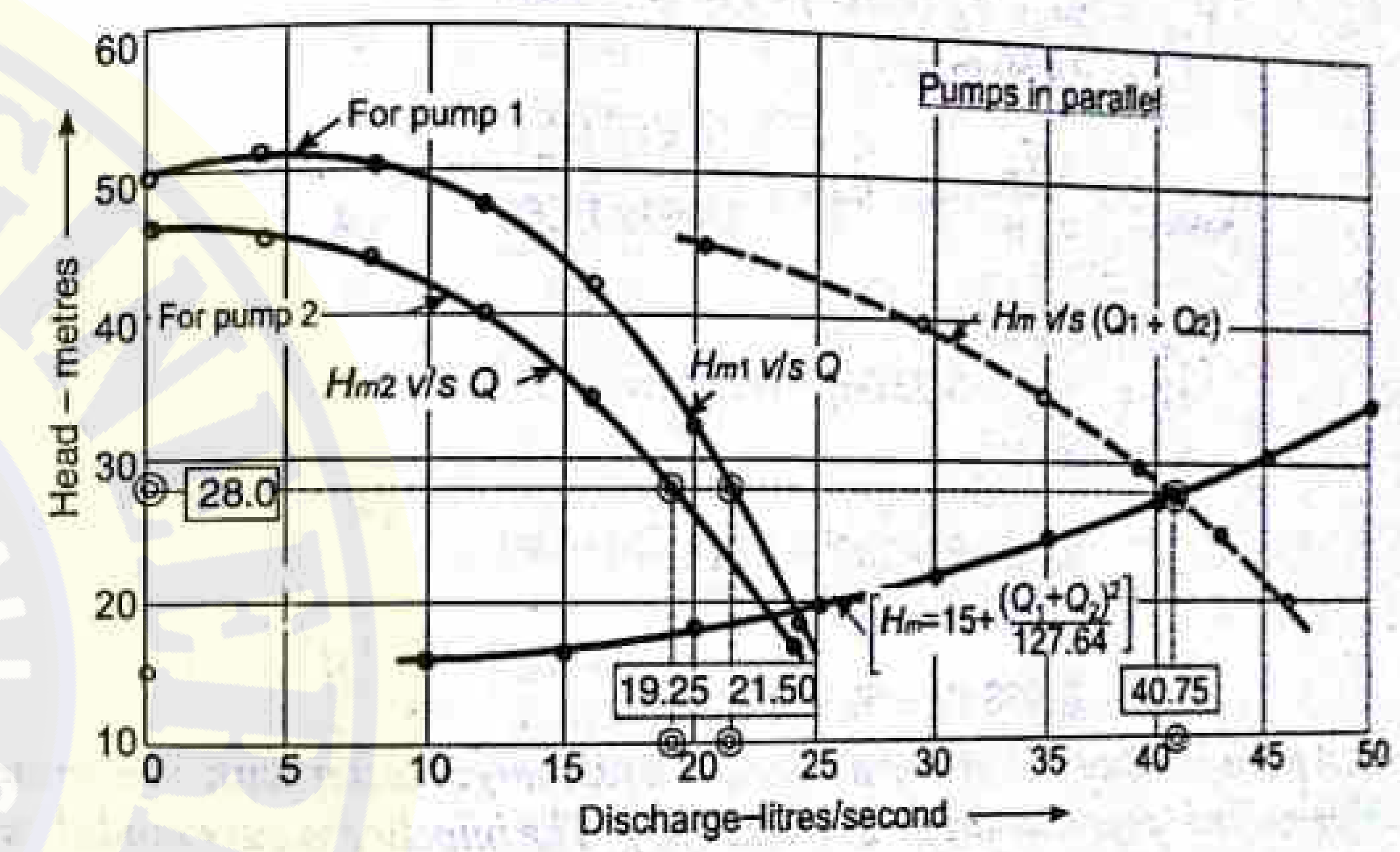


Figure Ex 24.14 (b)

H_m	20	25	30	35	40	45
Q_1	23.4	22.3	20.8	19.2	17.2	14.5
Q_2	22.7	20.5	18.2	15.6	12.2	6.0
(Q_1+Q_2)	46.1	42.8	39.0	34.8	29.4	20.5

Similarly for different values of (Q_1+Q_2) the corresponding value of H_m in respect of Eq. (ii) are computed as given below; and these are also plotted on the same graph.

(Q_1+Q_2)	10	15	20	25	30	35	40	45	50
H_m	15.8	16.8	18.1	19.9	22.1	24.6	27.5	30.9	34.6

It may be seen that the two (Q_1+Q_2) v/s H_m curves intersect at a point corresponding to which $H_m = 28.0$ and $(Q_1+Q_2) = 40.75$ litres/second. For $H_m = 28$ m, from the graph, $Q_1 = 21.50$ litres/second and $Q_2 = 19.25$ litres/second.

Example 24.15. *A three-stage centrifugal pump has impellers 375 mm diameter and 18 mm wide at outlet. The outlet vane angle is 45° and the vanes occupy 8% of the outlet area. The manometric efficiency is 84% and the overall efficiency is 75%. What head the pump will generate when running at 900 r.p.m., discharging 60 litres/second? What is the input power?*

Solution

$$V_{f_1} = \frac{60\times10^{-3}}{\pi\times0.375\times0.018\times0.92} = 3.08\ \text{m/s}$$

$$u_1 = [(\pi D_1 N)/60]$$

$$= \frac{\pi\times0.375\times900}{60} = 17.67\ \text{m/s}$$

Now from the outlet velocity triangle of Fig. 24.5,

$$V_{w1} = (u_1 - V_{f1}\cot 45°) = (17.67 - 3.08)$$

$$= 14.59\ \text{m/s}$$

$$\eta_{mano} = \frac{gH_m}{V_{w_1}u_1};\ \text{or}\ 0.84 = \frac{9.81\times H_m}{14.59\times17.66}$$

Thus $H_m = 22.06$ m

∴ Total head $H = 3H_m = 3(22.06) = 66.18$ m

$$\eta_0 = (wQH)/P$$

or

$$P = \frac{9810\times60\times10^{-3}\times66.18}{0.75}$$

$$= 51938\ \text{W} = 51.938\ \text{kW}$$

Example 24.16. *A pump impeller is 375 mm diameter and it discharges water with component velocities of 2 and 12 m/s in the radial and tangential directions respectively. The impeller is surrounded by a concentric cylindrical chamber with parallel sides the outer diameter being 450 mm. If the flow in this chamber is a free spiral vortex, find the component velocities of water on leaving and increase in pressure if there is no loss.*

Solution

For free cylindrical vortex from Eq. 7.57

or

$$12\times(187.5) = V_2\times(225)$$

or

$$V_2 = \frac{12\times187.5}{225} = 10\ \text{m/s}$$

For radial flow from Eq. 7.65

$$V_1R_1 = V_2R_2$$

$$2\times(187.5) = V_2\times(225)$$

or

$$V_2 = \frac{2\times187.5}{225} = 1.67\ \text{m/s}$$

Hence the component velocities of water on leaving are 1.67 m/s and 10 m/s in the radial and tangential directions respectively.

For spiral vortex motion the increase in pressure is given by Eq. 7.71 as

$$\frac{p_2}{w} - \frac{p_1}{w} = \left[\frac{(12)^2-(10)^2}{2\times9.81} + \frac{(2)^2-(1.67)^2}{2\times9.81}\right] = 2.304\ \text{m}$$

Example 24.17. *Pressures on the suction side and the discharge side of a pump are read by the pressure gage as (i) vacuum of 250 mm of mercury and (ii) 137.34 kN/m² (1.4 kg/cm²) respectively.*

The gage on the suction side is placed 600 mm below the centreline of the pump where as the gage on the discharge side is placed 170 mm above the centreline. If the diameters of the pipes on suction and discharge sides are 150 mm and 100 mm respectively and the quantity of water to be pumped is 30 litres per second, calculate the power required to drive the pump assuming its efficiency as 60%.

Solution

$$\frac{p_s}{w} = -\frac{250}{1000}\times13.6$$

$$= -3.4\ \text{m of water}$$

$$\frac{p_d}{w} = \frac{137.34\times10^3}{9810}$$

$$= 14\ \text{m of water}$$

$$V_s = \frac{30\times10^{-3}}{(\pi/4)(0.15)^2} = 1.698\ \text{m/s}$$

$$\therefore \frac{V_s^2}{2g} = \frac{(1.698)^2}{2\times9.81} = 0.147\ \text{m}$$

$$V_d = \frac{30\times10^{-3}}{(\pi/4)(0.10)^2} = 3.820\ \text{m/s}$$

$$\therefore \frac{V_d^2}{2g} = \frac{(3.820)^2}{2\times9.81} = 0.744\ \text{m}$$

The manometric head is given as

$$H_m = \left(\frac{p_d}{w}+\frac{V_d^2}{2g}+h\right) - \left(\frac{p_s}{w}+\frac{V_s^2}{2g}-h_1\right)$$

where h is the height of the pressure gage on the discharge side above the centreline of the pump; and h_1 is the depth of the vacuum gage on the suction side below the centreline of the pump.

$$h = (170\times10^{-3}) = 0.17\ \text{m; and}\ h_1 = (600\times10^{-3}) = 0.60\ \text{m}$$

Thus by substituting the values in the above equation, we get

$$H_m = (14 + 0.744 + 0.17) - (-3.4 + 0.147 - 0.60)$$

$$= 18.767\ \text{m}$$

The power required to drive the pump is given by

$$P = \frac{wQH_m}{\eta_0}$$

$$= \frac{9810 \times 30 \times 10^{-3} \times 18.767}{0.60}$$

$$= 9205 \text{ W} = 9.205 \text{ kW}$$

Example 24.18. *At what height from water surface a centrifugal pump may be installed in the following case to avoid cavitation: atmospheric pressure 101 kPa; vapour pressure 2.34 kPa; inlet and other losses in suction pipe 1.55 m; effective head of pump 52.5 m; and cavitation parameter $\sigma = 0.118$.*

Solution

Cavitation factor is given by

$$\sigma = \frac{[(p_a - p_v)/w] - (h_s + h_{f_s})}{H_m}$$

$$p_a = 101 \text{ kPa} = 101000 \text{ N/m}^2$$

$$p_v = 2.34 \text{ kPa} = 2340 \text{ N/m}^2$$

$$H_m = 52.5 \text{ m}; h_{f_s} = 1.55 \text{ m}; \sigma = 0.118$$

Thus by substitution, we get

$$0.118 \times 52.5 = \frac{(101000 - 2340)}{9810} - h_s - 1.55$$

$$\therefore \quad h_s = 2.312 \text{ m}$$

i.e., the pump may be installed at a height of 2.312 m above the water surface in the supply tank.

Example 24.19. *A pump operates at a maximum efficiency of 82% and delivers 2.25 m³/s under a head of 18 m while running at 3600 r.p.m speed. Compute the specific speed of the pump. Also determine the discharge, head and power input to pump at a shaft speed of 2400 r.p.m. Cite the assumption made, if any.*

Solution

$$Q = 2.25 \text{ m}^3/\text{s}; H = 18 \text{ m}; \eta_0 = 82\% = 0.82;$$

$$N = 3600 \text{ r.p.m}$$

$$\text{Power } P = \frac{wQH}{\eta_0}$$

$$= \frac{9810 \times 2.25 \times 18}{0.82}$$

$$= 484518 \text{ W} = 484.518 \text{ kW}$$

The specific speed is given by

$$N_s = \frac{N\sqrt{Q}}{H^{3/4}}$$

$$Q = 2.25 \text{ m}^3/\text{s} = 2.25 \times 10^3 \text{ l/s}$$

$$\therefore \quad N_s = \frac{3600\sqrt{2.25 \times 10^3}}{(18)^{3/4}} = 19541 \text{ [r.p.m; l/s; m]}$$

However, if discharge Q is taken in m³/s,

$$N_s = \frac{3600\sqrt{2.25}}{(18)^{3/4}} = 618 \text{ [r.p.m, m}^3\text{/s, m]}$$

Since $Q \propto N$; $H \propto N^2$; and $P \propto N^3$, we have

$$\frac{Q}{N} = \frac{Q_1}{N_1}$$

$$\therefore \quad Q_1 = \left(\frac{N_1}{N}\right) Q$$

$$= \frac{2400}{3600} \times 2.25 = 1.5 \text{ m}^3/\text{s}.$$

$$\frac{H}{N^2} = \frac{H_1}{N_1^2}$$

$$\therefore \quad H_1 = \left(\frac{N_1^2}{N^2}\right) H$$

$$= \frac{(2400)^2}{(3600)^2} \times 18 = 8 \text{ m}$$

$$\frac{P}{N^3} = \frac{P_1}{N_1^3}$$

$$\therefore \quad P_1 = \left(\frac{N_1^3}{N^3}\right) P$$

$$= \frac{(2400)^3}{(3600)^3} \times 484.518 = 143.56 \text{ kW}$$

The above expressions used for determining discharge, head and power at other speed are based on the assumption that the efficiency remains constant at all the speeds.

Example 24.20. *A centrifugal pump operates against a manometric head of 30 m with a manometric efficiency of 75%. The pressure rise through the impeller is 65% of the total head developed by the pump. The radial velocity of flow which is constant is 3 m/s. The outer diameter of the impeller is 400 mm and the width at outlet is 15 mm. The blades at inlet are curved backwards at 60° to the wheel tangent. Calculate (i) the discharge in litres per minute, (ii) speed, (iii) blade angle at outlet, and (iv) diameter of impeller at inlet.*

Solution

(i) The discharge Q delivered by the pump is given by

$$Q = k\pi B_1 D_1 V_{f_1}$$

Given

$$B_1 = 15 \text{ mm} = 0.015 \text{ m}$$

$$D_1 = 400 \text{ mm} = 0.400 \text{ m}$$

$$V_{f_1} = 3 \text{ m/s}$$

Assuming $k = 1$, we get

$$Q = 1 \times \pi \times 0.015 \times 0.400 \times 3$$

or $$Q = 0.0565 \text{ m}^3\text{/s} = 3390 \text{ litres/minute}$$

(ii) Manometric efficiency is given by

$$\eta_{mano} = \frac{gH_m}{V_{w_1} u_1}$$

or $$\frac{V_{w_1} u_1}{g} = \frac{H_m}{\eta_{mano}}$$

$$H_m = 30 \text{ m}; \quad \eta_{mano} = 75\% = 0.75$$

$\therefore$ $$\frac{V_{w_1} u_1}{g} = \frac{30}{0.75} = 40 \text{ m}$$

i.e., total head developed by the pump = 40 m

The pressure rise through the impeller $\left(\frac{p_1 - p}{w}\right)$ is 65% of the total head developed by the pump.

Thus

$$\left(\frac{p_1 - p}{w}\right) = (0.65 \times 40) = 26 \text{ m}$$

Applying Bernoulli's equation between the inlet and outlet tips of the impeller and neglecting the head loss in the impeller, we have

$$\frac{p}{w} + \frac{V^2}{2g} = \frac{p_1}{w} + \frac{V_1^2}{2g} - \frac{V_{w_1} u_1}{g}$$

or $$\left(\frac{p_1 - p}{w}\right) = \frac{V^2}{2g} + \frac{V_{w_1} u_1}{g} - \frac{V_1^2}{2g}$$

$$V = V_f = 3 \text{ m/s}$$

Thus by substitution, we get

$$26 = \frac{(3)^2}{2 \times 9.81} + 40 - \frac{V_1^2}{2g}$$

or $$V_1 = 16.84 \text{ m/s}$$

But $$V_1 = \sqrt{V_{w_1}^2 + V_{f_1}^2}$$

$\therefore$ $$V_{w1} = \sqrt{V_1^2 - V_{f_1}^2}$$

$$V_{f_1} = 3 \text{ m/s}$$

Thus by substitution, we get

$$V_{w1} = \sqrt{(16.84)^2 - (3)^2} = 16.57 \text{ m/s}$$

By substituting in Eq. (i), we get

$$\frac{16.57 \times u_1}{9.81} = 40$$

or $$u_1 = \frac{40 \times 9.81}{16.57} = 23.68 \text{ m/s}$$

But $$u_1 = \frac{\pi D_1 N}{60}$$

or $$23.68 = \frac{\pi \times 0.400 \times N}{60}$$

or $$N = \frac{23.68 \times 60}{\pi \times 0.400} = 1131 \text{ r.p.m.}$$

i.e., speed of the pump = 1131 r.p.m.

(iii) From outlet velocity triangle, we have

$$\tan \phi = \frac{V_{f_1}}{u_1 - V_{w_1}}$$

or $$\tan \phi = \frac{3}{23.68 - 16.57} = 0.4219$$

$\therefore$ $$\phi = 22°52'$$

i.e., blade angle at outlet = 22°52′

(iv) From inlet velocity triangle

$$\tan \phi = \frac{V_f}{u}$$

or $$u = V_f \cot \theta$$

$$\theta = 60°$$

$\therefore$ $$u = 3 \cot 60° = 1.732 \text{ m/s}$$

Also $$u = \frac{\pi DN}{60}$$

or $$1.732 = \frac{\pi \times D \times 1131}{60}$$

or $$D = \frac{1.732 \times 60}{\pi \times 1131}$$

or $$D = 0.029 \text{ m} = 29 \text{ mm}$$

i.e., the diameter of the impeller at inlet = 29 mm

Example 24.21. *During an experiment a centrifugal pump operating at 1150 r.p.m. developed a head of 25 m and a discharge of 3800 litres/minute at the best efficiency regime. At this regime the power drawn by the pump is 28 h.p. The mechanical losses due to bearings and packing is 2% of the pump output and the disc friction loss is 1.6 h.p. The leakage is estimated to be 160 litres/minute.*

The impeller is of 38 cm diameter and is 2 cm wide at the exit. The blade angle at outlet is 22°. There are seven blades and their thickness is constant throughout and equal to 5 mm. Assuming that the fluid approaches the impeller with prerotation, determine the overall efficiency, head slip and actual absolute flow angle at impeller exit.

Solution

The overall efficiency of the pump is given by Eq. 24.23 as

$$\eta_0 = \frac{wQH_m}{\text{Power given to the shaft}}$$

$$w = 9810\ \text{N/m}^3;\ Q = \frac{(3800-160)}{1000\times 60} = 0.061\ \text{m}^3/\text{s};$$

$H_m = 25$ m; and Power given to the shaft = 28 × 746 W

Thus by substitution, we get

$$\eta_0 = \frac{9810\times 0.061\times 25}{28\times 746} = 0.716 \text{ or } 71.6\%$$

$$\begin{aligned}\text{Pump output} &= wQH_m\\ &= 9810\times 0.061\times 25\ \text{W}\\ &= 14960.25\ \text{W}\end{aligned}$$

∴ Mechanical losses due to bearings and packing

$$14960.25\times\frac{2}{100} = 299.205\ \text{W}$$

$$\begin{aligned}\text{Disc friction loss} &= 1.6\ \text{h.p.}\\ &= 1.6\times 746\ \text{W} = 1193.6\ \text{W}\end{aligned}$$

∴ Total mechanical losses

$$\begin{aligned}&= (1193.6 + 299.205)\ \text{W}\\ &= 1492.805\ \text{W}\end{aligned}$$

Thus the power imparted by the impeller

$$\begin{aligned}&= [(28\times 746) - 1492.805]\ \text{W}\\ &= 19395.195\ \text{W}\end{aligned}$$

Again power imparted by the impeller

$$\begin{aligned}&= w(Q+\Delta Q)\times\frac{V_{w_1}u_1}{g}\\ &= 9810\times\frac{3800}{60\times 1000}\times\frac{V_{w_1}u_1}{g}\\ &= 63.33\ V_{w_1}u_1\end{aligned}$$

$$\therefore\quad 63.33\ V_{w_1}u_1 = 19395.195$$

or

$$V_{w_1}u_1 = \frac{19395.195}{63.33} = 306.26$$

$$u_1 = \frac{\pi D_1 N}{60}$$

$$D_1 = 0.38\ \text{m; and } N = 1150\ \text{r.p.m}$$

Thus

$$u_1 = \frac{\pi\times 0.38\times 1150}{60} = 22.88\ \text{m/s}$$

And

$$V_{w_1} = \frac{306.26}{22.88} = 13.39\ \text{m/s}$$

The discharge delivered by the pump is given by

$$Q = \pi(D_1 - zt)B_1V_{f_1}$$

$$z = 7;\ t = 0.005\ \text{m; and } B_1 = 0.02\ \text{m}$$

Thus

$$\frac{3800}{1000\times 60} = \pi[0.38-(0.005\times 7)]\times 0.02\times V_{f_1}$$

$$\therefore\quad V_{f_1} = \frac{3800}{1000\times 60\times\pi\times 0.345\times 0.02} = 2.92\ \text{m/s}$$

From outlet velocity triangle, we have

$$\begin{aligned}\tan\beta &= \frac{V_{f_1}}{V_{w_1}}\\ &= \frac{2.92}{13.39} = 0.2181\end{aligned}$$

$$\therefore\quad \beta = 12°18'$$

Thus actual absolute flow angle at impeller exit is 12°18′

Again from outlet velocity triangle, we have

$$\begin{aligned}\tan\phi &= \frac{V_{f_1}}{u_1 - V_{w_1}}\\ &= \frac{2.92}{(22.88-13.39)} = 0.3077\end{aligned}$$

$$\therefore\quad \phi = 17°6'$$

However, the actual blade angle provided at outlet is 22°, corresponding to which if the value of V_{w_1} is V'_{w_1} it may be obtained as

$$\tan 22° = \frac{2.92}{22.88 - V'_{w_1}}$$

or

$$V'_{w_1} = 15.65\ \text{m/s}$$

The head corresponding to the value of V'_{w_1}

$$= \frac{V'_{w_1} u_1}{g}$$

$$= \frac{15.65 \times 22.88}{9.81} = 36.50 \text{ m}$$

But the actual head developed is

$$\frac{V_{w_1} u_1}{g} = \frac{306.26}{9.81} = 31.22 \text{ m}$$

$\therefore$ Head slip $= (36.50 - 31.22)$

$= 5.28$ m

Example 24.22. *Two identical pumps are connected to a pipe system as shown in Fig. Ex. 24.22 to pump water into the high level reservoir. The head discharge characteristic of the pumps is given by the equation*

$$H = 40 + 0.14Q - 0.0012Q^2$$

where Q is the pump discharge in litres per second and H is the head developed by the pump in m.

The pipe AB is 100 m long and is of diameter 30 cm. The pipe BC is 45 cm in diameter and 200 m long. For both pipes the friction factor can be taken as 0.03. 'V' is a regulating valve on the pipe line BC. The static lifts against which the pumps are working are 20 m and 30 m.

For a given position of the valve 'V' the pump operating against a static lift of 30 m develops a discharge of 140 litres per second. Determine the discharge of the second pump and the loss coefficient of the valve, neglecting all other local losses.

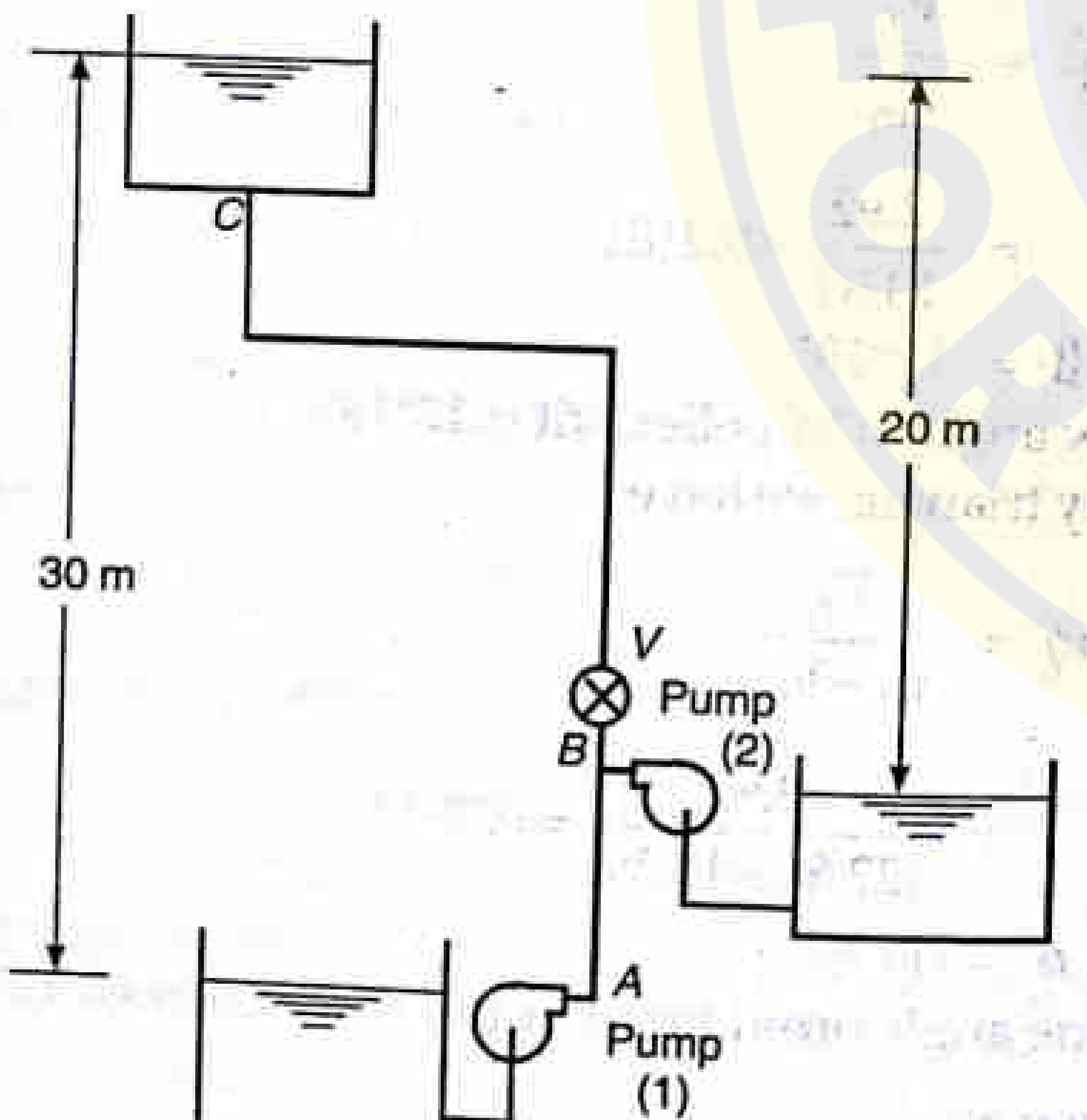


Figure Ex. 24.22

Solution

Discharge of pump No. 11 =140 l/s

$\therefore$ Head developed by pump No. 1 is

$$H_1 = 40 + 0.14 \times 140 - 0.0012 \times (140)^2 = 36.08 \text{ m}$$

When only Pump No.1 is operating, then

$$V_{AB} = \frac{140}{1000 \times (\pi/4) \times (0.30)^2} = 1.98 \text{ m/s}$$

$$V_{BC} = \frac{140}{1000 \times (\pi/4) \times (0.45)^2} = 0.88 \text{ m/s}$$

$$h_{f_{AB}} = \frac{0.03 \times 100 \times (1.98)^2}{2 \times 9.81 \times 0.30} = 1.998 \text{ m}$$

$$h_{f_{BC}} = \frac{0.03 \times 200 \times (0.88)^2}{2 \times 9.81 \times 0.45} = 0.526 \text{ m}$$

Let K be the loss coefficient of the valve, then head loss in the valve $= K\frac{V_{BC}^2}{2g}$

Thus we have

$$H_1 = H_{s_1} + h_{f_{AB}} + K\frac{V_{BC}^2}{2g} + h_{f_{BC}} + \frac{V_{BC}^2}{2g}$$

or

$$36.08 = 30 + 1.998 + K\frac{(0.88)^2}{2 \times 9.81} + 0.526 + \frac{(0.88)^2}{2 \times 9.81}$$

$\therefore$ $K = K = 89.11 \approx 89$

Let the discharge of Pump No. 2 be Q l/s.

Then head developed by Pump No. 2 is

$$H_2 = 40 + 0.14\,Q - 0.0012\,Q^2$$

When both the pumps are operating the sum of the heads developed by the two pumps is

$$H_1 + H_2 = [36.08 + 40 + 0.14\,Q - 0.0012\,Q^2]$$

Discharge through pipe AB = 140 l/s

$\therefore$

$$V_{AB} = \frac{140}{1000 \times (\pi/4) \times (0.30)^2} = 1.98 \text{ m/s}$$

$$h_{f_{AB}} = \frac{0.03 \times 100 \times (1.98)^2}{2 \times 9.81 \times 0.30} = 1.998 \text{ m}$$

Discharge through pipe BC = $(Q + 140)$ l/s

$\therefore$

$$V_{BC} = \frac{(Q+140)}{1000 \times (\pi/4) \times (0.45)^2} = \frac{(Q+140)}{159.04} \text{ m/s}$$

$$h_{f_{BC}} = \frac{0.03\times 200\times(Q+140)^2}{2\times 9.81\times 0.45\times(159.04)^2} = \frac{(Q+140)^2}{37219.71}\text{ m}$$

Head loss in the valve

$$= K\frac{V_{BC}^2}{2g}$$

$$= \frac{89\times(Q+140)^2}{2\times 9.81\times(159.04)^2} = \frac{(Q+140)^2}{5575.99}\text{ m}$$

Thus we have

$$H_1 + H_2 = H_{s_1} + H_{s_2} + h_{f_{AB}} + K\frac{V_{BC}^2}{2g} + h_{f_{BC}} + \frac{V_{BC}^2}{2g}$$

or

$$[36.08 + 40 + 0.14Q\ \ 0.0012\,Q^2] = 30 + 20 + 1.998$$

$$+\frac{(Q+140)^2}{5575.99} + \frac{(Q+140)^2}{37219.71} + \frac{(Q+140)^2}{2\times 9.81\times(159.04)^2}$$

or $\quad Q^2 - 58.36\,Q - 14286 = 0$

$\therefore \quad Q = 152\text{ l/s}$

Discharge of Pump No. 2 is 152 l/s and loss coefficient of the valve is 89.

Example 24.23. *In a pumping station, 18000 cum water is to be raised per day from an intake well to a sedimentation tank under a static head of 21 m. Lengths of suction pipe and rising main are 40 m and 150 m respectively. Diameters of pipes is 50 cm. There are two shifts of working of pumps each of 8 hours. Take coefficient of friction as 0.01 and combined efficiency of motor and pump as 80%. Recommend the number of units of pumps each having B.H.P of 30.*

Solution

Total quantity of water to be raised per day= 18,000 m^3.

Since water is to by raised by pumps working in two shifts each of 8 hours, the entire quantity of water is to be raised in (2×8) = 16 hours.

$\therefore$ Discharge $\quad Q = \dfrac{18{,}000}{16\times 3600} = 0.3125\text{ m}^{3/5}$

Diameters of suction pipe and rising main are

$d_s = d_d = 50\text{ cm} = 0.5\text{ m}$

$\therefore \quad$ Area $A = \dfrac{\pi}{4}\times(0.5)^2 = 0.19634\text{ m}^2$

Velocity $V = \dfrac{0.3125}{0.19634} = 1.5916\text{ m/s}$

Static head $H_s = 21\text{ m}$

Total length of pipe $= (40 + 150) = 190\text{ m}$

Coefficient of friction $f = 0.01$

Head loss due to friction

$$h_f = \frac{f\,LV^2}{2gd}$$

Thus by substitution, we get

$$h_f = \frac{0.01\times 190\times(1.5916)^2}{2\times 9.81\times 0.50}$$

$$= 0.4906\text{ m.}$$

$\therefore$ Total head $H = H_s + h_f$

$$= (21.0 + 0.4906) = 21.4906\text{ m}$$

$$\text{B.H.P} = \frac{WQH}{75}$$

$$= \frac{1000\times 0.3125\times 21.4906}{75}$$

$$= 89.54\text{ h.p}$$

Efficiency of motor and pump = 80% = 0.80

$\therefore$ Actual B.H.P required

$$= \frac{89.54}{0.80}\ 111.93\text{ h.p.}$$

B.H.P. of each pump = 30 h.p.

$\therefore$Number of pumps required

$$= \frac{111.93}{30} = 3.734 = 4$$

SUMMARY OF MAIN POINTS

1. Centrifugal pumps are classified as rotodynamic type of pumps in which dynamic pressure is developed which enables the lifting of liquids from a lower level to a higher level.
2. A centrifugal pump consists of a wheel or rotor called impeller which is provided with a series of backward curved blades or vanes, an airtight casing which surrounds the impeller, a suction pipe fitted with a foot valve and strainer at its lower end and a delivery pipe provided with a delivery valve.
3. The head imparted by the impeller to the liquid is given as

$$= \frac{1}{g}(V_{w_1}u_1)$$

where

V_{w_1} = velocity of whirl at outlet and

u_1 = tangential velocity of impeller at outlet.

4. The head of a centrifugal pump may be expressed as

(i) Static head

(ii) Manometric head

(i) The static head H_s is the vertical distance bewtween the liquid surfaces in the sump from which the liquid is being raised and the tank to which the liquid is delivered by the pump. Thus

$$H_s = h_s + h_d$$

in which h_s is the vertical height of the centre line of the pump shaft above the liquid surface in the sump, and h_d is the vertical height of the liquid surface in the tank to which the liquid is delivered above the centre line of the pump shaft.

The term h_s is known as static suction lift, and h_d is known as static delivery lift.

(ii) The manometric head H_m is the total head that must be produced by the pump to satisfy the external requirements. Thus

$$H_m = \frac{V_{w_1}u_1}{g} - \text{(losses of head in the pump)}$$

5. The manometric head is given by the following expressions.

(i) $$H_m = \left(\frac{p_d}{w} + \frac{V_d^2}{2g} + h\right) - \left(\frac{p_s}{w} + \frac{V_s^2}{2g}\right)$$

(ii) $$H_m = h_s + h_d + h_{f_s} + h_{f_d} + \frac{V_d^2}{2g}$$

where

p_d = pressure at the exit from the pump;
p_s = pressure at inlet to the pump;
V_d = velocity of flow in the delivery pipe;
V_s = velocity of flow in the suction pipe;
h = height of pressure gauge at the exit of the pump above the pressure gauge at the inlet to the pump;
h_{f_s} = head loss in the suction pipe; and
h_{f_d} = head loss in the delivery pipe.

6. The efficiencies of a centrifugal pump are

(i) Manometric efficiency

$$\eta_{mano} = \frac{gH_m}{V_{w_1}u_1}$$

or $$\eta_{mano} = \frac{H_m}{H_m + \text{losses in the pump}}$$

(ii) Volumetric efficiency

$$\eta_v = \frac{Q}{Q + \Delta Q}$$

(iii) Mechanical efficiency

$$\eta_{mech} = \frac{(V_{w_1}u_1/g)}{(V_{w_1}u_1/g) + \begin{bmatrix}\text{mechanical head}\\ \text{losses in bearing}\end{bmatrix}}$$

(iv) Overall efficiency

$$\eta_o = \frac{WQH_m}{\text{Power given to the shaft}}$$

$$\eta_o = (\eta_{mano}) \times (\eta_v) \times (\eta_{mech})$$

7. The minimum speed for starting a centrifugal pump is given by

$$N = \frac{60\sqrt{2gH_m}}{\pi\sqrt{D_1^2 - D^2}}$$

8. The specific speed N_s of a centrifugal pump may be defined as the speed in r.p.m. of a geometrically similar pump of such a size that under corresponding conditions it would deliver 1 litre of liquid per second against a head of 1 metre. The specific speed is given by

$$N_s = \frac{N\sqrt{Q}}{H_m^{3/4}}$$

9. For complete similarity to exis between the model and the prototype pumps the necessary conditions are as follows:

$$\left(\frac{Q}{ND^3}\right)_m = \left(\frac{Q}{ND^3}\right)_p$$

$$\left(\frac{gH}{N^2D^2}\right)_m = \left(\frac{gH}{N^2D^2}\right)_p$$

$$\left(\frac{P}{\rho gQH}\right)_m = \left(\frac{P}{\rho gQH}\right)_p$$

$$\left[\frac{NQ}{(gH)^{3/4}}\right]_m = \left[\frac{NQ}{(gH)^{3/4}}\right]_p$$

10. The efficiency of the prototype pump and that of the model pump are related as

$$\frac{0.95 - \eta_m}{0.95 - \eta_p} = \left[\frac{0.658 + \log_{10} Q_p}{0.658 + \log_{10} Q_m}\right]^2$$

11. To produce a high head the pumps are connected *in series*. The higher heads may also be produced by using multi-stage pumps. A *multistage pump* consists of two or more indentical impellers mounted on the same shaft and enclosed in the same casing.

12. The pumps are connected in *parallel* when a large quantity of liquid is required to be pumped against a relatively small head.

13. In order to predict the behaviour and performance of pump under varying conditions the following three types of characteristics curves are prepared for the centrifugal pumps.
(i) Main and operating characteristics.
(ii) Constant efficiency or Muschel curves
(iii) Constant head and constant discharge curves

14. In centrifugal pumps cavitation may occur at the suction side of the pump if the pressure drops below the vapour pressure of the liquid.

As in the case of turbines for pumps also Thoma's cavitation factor σ is used to indicate whether cavitation will occur. In order to avoid the cavitation the value of σ should not be less than the critical value σ_c given by

$$\sigma_c = 0.103\,(N_s/1000)^{4/3}$$

where

N_s = specific speed of the pump.

15. Before starting a centrifugal pump it should be primed. Priming is the operation in which the suction pipe, casing of the pump and the portion of the delivery pipe upto the delivery valve are completely filled with the liquid which is to be pumped, so that all the air is driven out from this portion of the pump.

PROBLEMS

24.1 Explain with neat sketches the working of a single-stage centrifugal pump.

24.2 Under what headings the centrifugal pumps are classified?

24.3 State the difference between a closed, semi-closed and open impeller.

24.4 Discuss the various methods adopted to increase the efficiency of a centrifugal pump by altering the shape of the casing of chamber surrounding the impeller.

24.5 Why is the efficiency of a volute casing of a centrifugal pump as an energy conversion device low? How is a whirlpool or vortex chamber superior in performance?

24.6 If a centrifugal pump does not deliver any water when started, what may be the probable causes and how can they be remedied?

24.7 Define static and manometric head of a centrifugal pump. State the different types of head losses which may occur in a pump installation.

24.8 What are the different efficiencies of a centrifugal pump?

24.9 What is meant by 'Priming' of a pump? What are the different priming arrangements employed for small and big pumping units?

24.10 Why can the suction lift of a pump not exceed a certain limit?

24.11 A centrifugal pump of the radial type delivers 5 000 litres per minute against a total head of 38 m, when running at a speed of 1450 r.p.m. If the outer diameter of the impeller is 300 mm and its width at the outer periphery is 13 mm, find the vane angle at exit. Assume manometric efficiency as 80%. [*Ans.* 71°21′]

24.12 The impeller of a centrifugal pump has 1.2 m outside diameter. It is used to lift 1800 litres of water per second against a head of 6 m. Its vanes make an angle of 150 with the direction of motion at outlet and runs at 200 r.p.m. If the radial velocity of flow at outlet is 2.5 m/s, find the manometric efficiency. Also find the lowest speed to start the pump, if the diameter of the impeller at inlet is equal to half the diameter at exit. [*Ans.* 57%; 199.4 r.p.m.]

24.13 A centrifugal pump is required to deliver 280 litres of water per second against a head of 16 m. If the vanes of the impeller are radial at outlet and the velocity of flow is constant equal to 2 m/s, find the proportions of the pump. Assume η_{mano} = 80% and the ratio of breadth to diameter at outlet as 0.1.
[*Ans.* β = 8.13°; α = 15.94°; D_1 = 0.67 m; B_1 = 0.067 m]

24.14 A centrifugal pump is required to discharge 600 litres of water per second against a head of 12 m, when running at a speed of 750 r.p.m.

The manometric efficiency is to be 80%, the loss of head in the pump being assumed to be $0.025\,V_1^2$ of water, where V_1 is the absolute velocity of water leaving the impeller. Water enters the impeller without whirl and the velocity of flow is 3 m/s. Determine (a) the impeller diameter and the outlet area; and (b) the vane angle at the outlet edge of the impeller.
[*Ans.* (a) 0.356 m, 0.2 m²; (b) 41°15′]

24.15 A centrifugal pump running at 1 450 r.p.m. has the characteristics as given below:

Discharge (Liters/sec.)	*Heat (meters)*	*Efficiency (%)*
11.3	25.8	65
16.9	25.0	70
22.6	24.1	73
28.3	23.1	74
34.0	21.4	72
39.6	18.9	69
45.2	15.8	62

Draw the operating characteristics of the pump and determine its specific speed.

The pump lifts water against a static head of 12 m through a long pipeline in which the loss of head in metres, due to friction is given by the expression, $h_f = 0.012\,Q^2$, where Q is the discharge in litres/sec. The minor losses in the pipe may be neglected. Determine the power required to drive the pump.
[*Ans.* 730 r.p.m.; 8.81 kW]

24.16 The axis of a centrifugal pump is 2.5 m above water level in the sump and the static lift from the pump centre is 33.5 m. The friction losses in the suction and the delivery pipes are 1 m and 7.5 m respectively. The suction and delivery pipes are each 75 mm diameter. The impeller is 300 mm diameter and 18 mm wide at outlet and its speed is 1 700 r.p.m. The water at inlet has radial flow and the vane angle at outlet is 32° to the tangent to the periphery. Compute the discharge of the pump and the power required to drive the pump. Assume $\eta_{mano} = 77\%$ and $\eta_0 = 72\%$. [*Ans.* 0.038 m³/s; 25.3 kW]

24.17 A four-stage centrifugal pump has impellers 380 mm diameter and 19 mm wide at outlet. The outlet vane angle is 45° and the vanes occupy 8% of the outlet area. The manometric efficiency is 84% and the overall efficiency 72%. Determine the head generated by the pump when running at 900 r.p.m., and discharging 59 litres/second. Also determine the power required to drive the pump. [*Ans.* 92.48 m; 74.34 kW]

24.18 A two-stage centrifugal pump is required for a fire engine for a duty of 3 660 litres per minute at a head of 75 m. If the overall efficiency of the pump is 75% and the specific speed per stage is about 1 300, find (a) the running speed in r.p.m. and (b) the power of the driving engine. If the actual manometric head developed is 65% of the theoretical head, there is no slip, the outlet vane angle is 30°, and the radial flow velocity at exit is 0.15 times the tip speed at exit, find the diameter of the impellers.
[*Ans.* (a) 2 522 r.p.m; (b) 59.84 kW, 0.21 m]

24.19 A multi-stage centrifugal pump is to be designed to deliver 700 litres of water per second against a manometric head of 60 m. There are to be four equal impellers keyed to the same shaft, which has speed of 350 r.p.m. The vanes are curved back so that the direction of the relative velocity at exit makes an angle of 120° with the direction of the corresponding peripheral velocity and the impeller is surrounded by guides. Assume that the water enters radially and the velocity of flow through the impeller is 0.27 of the peripheral velocity and that the losses in the pump amount to one-third of the velocity head at discharge from the impeller. Find: (a) the outer dimensions of the impeller; (b) the manometric efficiency; (c) the angle of the guides.
[*Ans.* (a) $D_1 = 0.78$ m, $B_1 = 74$ mm; (b) 84.5%; (c) 17°44′]

24.20 Two centrifugal pumps operating in parallel are employed for lifting water against a static head of 18 m, the delivery pipe being 0.5 m in diameter and 15 000 m long having $f = 0.032$. Determine the head at the pipe inlet and the discharge; given the following data for the two pumps:

Q Litres/second	0	15	30	45	60
PUMP 1 H_{m_1} meters	45.0	43.5	40.0	32.7	23.3
PUMPS 2 H_{m_2} metres	40.7	39.0	33.7	27.3	20.7

24.21 Explain (i) double suction impeller, (ii) shut-off head, (iii) manometric efficiency, (iv) net positive suction head as applied to a suction pump.

24.22 Assuming that the radial component of flow of fluid through a centrifugal pump remains constant and that the fluid enters radially, prove that the ratio of pressure head H_p to velocity head H_v created by an impeller neglecting the losses is given by an equation

$$\frac{H_p}{H_v} = \left(\frac{u_1 - V_{f_1}\cot\beta}{u_1 + V_{f_1}\cot\beta}\right)$$

where u_1 is rim speed of impeller at outlet, V_{f_1} is velocity of flow and β is the exit angle of impeller blades. Hence show that ideal efficiency of the impeller is given by

$$\eta_i = \frac{u_1 - V_{f_1}\cot\beta}{2u_1}$$

24.23 Show that the pressure rise in the impeller of a centrifugal pump, when friction and other losses in the impeller are neglected, is given by

$$(p_2 - p_1)/w = \left(u_1^2 - V_{f_1}^2\right)/2g$$

provided flow is radial at inlet, velocity of flow is constant throughout and outlet angle of impeller vane = 45°. u_1 and V_{f_1} are peripheral and flow velocities at outlet end of impeller

187. (a)	188. (b)	189. (b)	190. (c)	191. (c)	191. (c)
192. (b)	193. (c)	194. (b)	195. (c)	196. (b)	197. (a)
198. (d)	199. (a)	200. (b)	201. (d)	202. (c)	203. (c)
204. (a)	205. (a)	206. (a)	207. (a)	208. (a)	209. (d)
210. (d)	211. (b)	212. (a)	213. (c)	214. (b)	215. (b)
216. (c)	217. (c)	218. (c)	219. (b)	220. (b)	221. (d)
222. (c)	223. (c)	224. (b)	225. (c)	226. (a)	227. (b)
228. (b)	229. (c)	230. (d)	231. (a)	232. (c)	233. (c)
234. (b)	235. (a)	236. (b)	237. (c)	238. (a)	239. (a)
240. (a)	241. (a)	242. (b)	243. (d)	244. (c)	245. (c)
246. (c)	247. (b)	248. (a)	249. (b)	250. (d)	251. (a)
252. (a)	253. (b)	254. (d)	255. (b)	256. (a)	257. (c)
258. (d)	259. (c)	260. (a)	261. (a)	262. (b)	263. (c)

Appendix I

Main Relations of Fluid Mechanics in Vector Notation

Any vector may be decomposed into three components along mutually perpendicular axes, say x, y, z axes. The component is a scalar quantity. The vector may be expressed in terms of its scalar components by use of unit vectors i, i, k parallel to the x, y, z axes respectively. A unit vector is a vector having unit magnitude. Thus if a is a vector having scalar components a_x, a_y, a_z along the three axes then

$$a = ia_x + ja_y + ka_z$$

The unit vectors combine as follows:

$$i.i = j.j = k.k = 1; i.j = j.k = k.i = 0$$

$$i \times j = k; j' k = i; k \times i = j = -i \times k \text{ etc.}$$

The scalar (or dot) product of two vectors a and b is

$$a.b = (ia_x + ja_y + ka_z).(ib_x + jb_y + kb_z)$$

$$= a_xb_x + a_yb_y + a_zb_z$$

The vector (or cross) product of two vectors a and b is

$$a \times b = (ia_x + ja_y + ka_z) \times (ib_x + jb_y + kb_z)$$

$$= i(a_yb_z - a_zb_y) + j(a_zb_x - a_xb_z) + k(a_xb_y - a_yb_x)$$

The above expression can be conveniently written in determinant form as

$$a \times b = \begin{vmatrix} i & j & k \\ a_x & a_y & a_z \\ b_x & b_y & b_z \end{vmatrix}$$

The various quantities as well as relations commonly used in Fluid Mechanics may be expressed in vector form as indicated below.

Velocity:

If velocity vector V has components u, v and w along three axes x, y, and z in the Cartesian system then

$$V = iu + jv + kw$$

where $$u = u(x, y, z, t)$$

$$v = v(x, y, z, t)$$
$$w = w(x, y, z, t)$$

The magnitude of velocity V is given by

$$|V| = \sqrt{u^2 + v^2 + w^2}$$

In vector analysis, vector differential operator ∇ *'del'* is defined as follows:

$$\nabla = i\frac{\partial}{\partial x} + j\frac{\partial}{\partial y} + k\frac{\partial}{\partial z}$$

The for example for any function ϕ

$$\nabla \phi = i\frac{\partial \phi}{\partial x} + j\frac{\partial \phi}{\partial y} + k\frac{\partial \phi}{\partial z}$$

Further the *gradient* of any function is defined as

$$\text{grad } \phi = i\frac{\partial \phi}{\partial x} + j\frac{\partial \phi}{\partial y} + k\frac{\partial \phi}{\partial z}$$

Thus by comparison, we have

$$\nabla \phi = \text{grad } \phi$$

The velocity vector V may by expressed in terms of the velocity potential as

$$V = -\text{grad } \phi = -\nabla \phi$$

Acceleration:

The acceleration vector a may be represented as

$$a = \frac{DV}{Dt} = \frac{\partial V}{\partial t} + V\,\nabla V$$

where

$$a = ia_x + ja_y + ka_z$$

The three components of the acceleration vector are

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

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

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

Continuity Equation:

The continuity equation in the most general form may be expressed in vector notation as

$$\frac{D\rho}{Dt} + \rho\nabla.\ V = 0$$

which is equivalent to its following form in Cartesian system

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

or

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

The scalar (or dot) product of any quantity with ∇ is interpreted as *divergence* or in short form *'div'*.

Thus in the above expression

$$\nabla.V = div\ V$$

Further it can be readily shown that

$$\nabla.V = \text{div } V = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$$

Equations of Motion:

The equations of motion can be written in vector form as

$$\frac{DV}{Dt} = B - \frac{1}{\rho}\nabla p\ ; \quad \text{or} \quad \frac{DV}{Dt} = B - \frac{1}{\rho}\text{ grad } p$$

where B is body force having components X, Y and Z so that

$$B = iX + jY + kZ$$

Further body forces are assumed to be derived from a potential Ω such that

$$B = -\text{grad } \Omega$$

or

$$X = -\frac{\partial \Omega}{\partial x}, Y = -\frac{\partial \Omega}{\partial y}, Z = -\frac{\partial \Omega}{\partial z}$$

Circulation and vorticity:

In expressing these terms in vector form the operator *curl* is introduced, which is interpreted as the vector (or cross) product of ∇ with any quantity. Thus

$$\text{curl } V = \nabla \times V$$

Circulation Γ may be expressed in vector form as

$$\Gamma = \iint_A \text{curl } V \cdot dA$$

The vorticity ζ is defined as curl of velocity vector V, i.e.,

$$\zeta = \text{curl } V = \nabla \times V$$

Further it can be shown that

$$\nabla \times V = i\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) + j\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) + k\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)$$

The quantities in paranthesis are vorticity components which are twice the value of rotation components

$$\omega_x, \omega_y, \omega_z$$

$$\therefore \quad \nabla \times V = i\,2\omega_x + j\,2\omega_y + k\,2\omega_z$$

The rotation of a fluid at a point may thus be defined as

$$w = \frac{1}{2}\text{ curl } V = \frac{1}{2}\nabla \times V$$

Irrotational flow:

The condition for the flow to be irrotational or the existence of velocity potential may thus be expressed in the vector form as

$$\text{curl } V = \text{curl } (-\text{grad } \phi) = -\nabla \times \nabla \phi = 0$$

which is equivalent to the condition in Cartesian system as

$$\omega_x = \omega_y = \omega_z = 0$$

Further any function ϕ that satisfies the Laplace equation is a possible irrotational fluid-flow case. The Laplace equation can be obtained in vector form as

$$\nabla . V = -\nabla . \nabla\phi = -\nabla^2 \phi = 0$$

and it is expressed as

$$\nabla^2\phi = 0$$

which is equivalent to its form in Cartesian system as

$$\frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

Appendix II

Typical Computer Programmes

In solving the problems of flow through channels, often it is required to determine the critical depth of flow and normal depth of flow for trapezoidal, rectangular and triangular channels. For determining critical depth of flow in trapezoidal channels trial error solution is required to be used. Similarly for determining normal depth of flow in trapezoidal and rectangular channels trial error solution is required to be used. As such computer programmes have been developed to facilitate the computation of critical and normal depths of flow for trapezoidal, rectangular and triangular channel sections. These programmes are in FORTRAN IV compiler language and are based on Newton-Raphson method. Both these programmes are given below.

```
C       THIS PROGRAMME IS FOR DETERMINING CRITICAL DEPTH
C       FOR TRAPEZOIDAL, RECTANGULAR AND TRIANGULAR
C       CHANNEL SECTIONS
C       B IS CHANNEL BED WIDTH IN METER
C       Z IS THE CHANNEL SIDE SLOPE (IV : ZH)
C       Q IS DISCHARGE IN CUMEC
        WRITE (*,15)
    15  FORMAT (1H0, 'FEED B, Z AND Q')
        READ (*,*) B,Z,Q
        IF (Z) 41,42,41
    42  Y=((Q*Q)/(B*B*9.81))**0.333
        GO TO 12
    41  IF (B) 44,45,44
    45  Y=((2.*Q*Q)/(Z*Z*9.81))**0.2
        GO TO 12
    44  Y=SQRT(Q/B)
    11  B1=B=Z*Y
        B2=B+2.*Z*Y
        F=Q*Q/9.81-(B1*Y)**3/B2
        P=3.*Y*Y*B1**3+3.*Z*Y**3*B1*B1
        DF=-(B2*P-2.*Z*Y**3*B1**3)/B2**2
```



```
        DELY=-F/DF
        Y=Y+DELY
        IF(ABS (DELY/Y).LT.0.0001) GO TO 12
        GO TO 11
    12  WRITE(*,20) B,Z,Q,Y
    20  FORMAT (1H0, 4E 12.5)
        STOP
        END
C       THIS PROGRAMME IS FOR DETERMINING NORMAL DEPTH
C       FOR TRAPEZOIDAL, RECTANGULAR AND TRIANGULAR
C       CHANNEL SECTIONS
C       B IS CHANNEL BED WIDTH IN METER
C       Z IS CHANNEL SIDE SLOPE (IV : ZH)
C       Q IS DISCHARGE IN CUMEC
C       RN IS MANNING'S N
C       S IS CHANNEL BED SLOPE
        WRITE(*,15)
    15  FORMAT (1H0, 'FEED B,Z,Q, RN AND S')
        READ (*,*) B,Z,Q,RN,S
        AMAX=100.
        IF(B)44,45,44
    45  Y=SQRT(0.3*Q/Z)
        GO TO 11
    44  Y=SQRT(Q/B)
    11  B1=B+Z*Y
        B2=B+2.*Z*Y
        A=B1*Y
        WP=B+2.*Y*SQRT(1.+Z*Z)
        R=A/WP
        F=RN*Q/A-R**0.667*S**0.5
        D1=WP*B2-2.*Y*B1*SQRT(1.+Z*Z)
        D2=-0.667*S**0.5
        D3=B1**0.333*Y**0.333*WP**1.667
        D4=RN*Q*B2/(B*Y+Z*Y*Y)**2
        DF=D2*D1/D3-D4
        DELY=-F/DF
        Y=Y+DELY
        WRITE (*,*) DELY
        IF (ABS(DELY).GT.AMAX)GO TO 13
        AMAX=ABS(DELY)
        IF(ABS (DELY/Y).LT.0.00001) GO TO 12
        GO TO 11
    13  AMAX=100.
        WRITE (*,16)
    16  FORMAT (1H0, 'FEED TRIAL VALUE OF DEPTH')
        READ (*,*)Y
        GO TO 11
    12  WRITE (*,20) B,Z,Q,RN,S,Y,DELY
    20  FORMAT (1H0, 6E11.4)
        STOP
        END
```

Appendix III

Comparative Study of Flow of Incompreessible and Compressible fluids

Particulars (1)	Incompressible Flow (2)	Compressible Flow (3)
1. Continuity Equation		
(a) Integral Form-One Dimensional Steady Flow	$AV = \text{Constant}$	$\rho AV = \text{Constant}$
(b) Differential Form		
(i) One Dimensional Steady Flow	$\frac{\rho A}{A}+\frac{dV}{V}=0$	$\frac{d\rho}{\rho}+\frac{dA}{A}+\frac{dV}{V}=0$
Unsteady Flow	$\frac{\partial A}{\partial t}+\frac{\partial(AV)}{\partial s}=0$	$\frac{\partial(\rho A)}{\partial t}+\frac{\partial(\rho AV)}{\partial s}=0$
(ii) Two Dimensional Steady Flow	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$	$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0$
Unsteady Flow	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$	$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0$
(iii) Three Dimensional Steady Flow	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$	$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$

Contd.

Contd.

Particulars (1)	Incompressible Flow (2)	Compressible Flow (3)
Unsteady Flow	$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$	$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$
2. Momentum Equation	$\sum F_x=\rho AV[(V_x)_2-(V_x)_1]$	$\sum F_x=[(\rho AVV_x)_2-(\rho AVV_x)_1]$
3. Energy Equation		
(a) In terms of energy per unit weight of fluid	$\frac{p}{w}+z+\frac{V^2}{2g}=$ constant	$E_n+\frac{V^2}{2g}+z=$ constant; in which $E_n=\frac{p}{w}+I=$ Enthalpy per unit weight of fluid $I=$ Internal heat energy of fluid to be expressed in terms of equivalent mechanical energy per unit weight of fluid.
(b) In terms of energy per unit mass of fluid	$\frac{p}{\rho}+gz+\frac{V^2}{2}=$ constant	$E_n+\frac{V^2}{2}+gz=$ constant; in which $E_n=\frac{p}{\rho}+I=$ Enthalpy per unit mass of fluid or specific enthalpy $I=$ Internal heat energy of fluid to be expressed in terms of equivalent mechanical energy per unit mass of fluid. The necessary conversion factors for these may be seen in Appendix IV.
4. Dimensionless Stagnation pressure or Pressure coefficient	$\frac{p_2-p_1}{\frac{\rho_1V_1^2}{2}}=1$	$\frac{p_2-p_1}{\frac{\rho_1V_1^2}{2}}=1+\frac{1}{4}(M_a)_1^2+\frac{2-k}{24}(M_a)_1^4+\ldots$
5. Flow through a pipe of Varying Cross-section	$\frac{dA}{A}<0;\frac{dV}{V}>0$ (Convergent Nozzle) $\frac{dA}{A}>0;\frac{dV}{V}<0$ (Diverge diffuser)	(a) For Subsonic Flow(i.e., $M_a<1$) $\frac{dA}{A}<0;\frac{dV}{V}>0$ (Convergent nozzle) $\frac{dA}{A}>0;\frac{dV}{V}<0$ (Divergent Diffuser)

Contd.

Contd.

Particulars (1)	Incompressible Flow (2)	Compressible Flow (3)
	$\frac{dA}{A}=0; \frac{dA}{V}=0$ *(Pipe of Uniform Cross-section)*	*(b) For Supersonic Flow (i.e., $M_a>1$)* $\frac{dA}{A}>0; \frac{dV}{V}>0$ *(Divergent Nozzle)* $\frac{dA}{A}<0; \frac{dV}{V}<0$ *(Convergent Diffuser)* *(c) For Sonic Flow (i.e., $M_a = 1$)* $dA = 0$, *i.e.*, the sonic flow will occur at a section of minimum cross-sectional area.
6. Other Phenomena of Similar Nature	Change of supercritical flow to subcritical flow in open channels is through the formation of a hydraulic jump. It is accompanied by an increase in depth and decrease in the velocity of flow.	Change of supersonic flow to subsonic flow is through the formation of shock waves. It is accompanied by an increase in the pressure and density of fluid and a decrease in the velocity.

Appendix IV

Some Important Conversion Factors

FOR ENGLISH (FPS), METRIC (MKS) AND SI UNITS

1. LENGTH			
1 cm	= 0.394 in;	1 in	= 2.54 cm
1 m	= 3.28 ft;	1 ft	= 0.305 m
1 km	= 0.621 mile	1 mile	= 1.609 km
2. AREA			
1 cm^2	= 0.155 in^2;	1 in^2	= 6.452 cm^2
1 m^2	= 10.764 ft^2;	1 ft^2	= 0.093 m^2
1 hectare	= 100 arc = 104 m^2 = 2.471 acres		
1 acre	= 43 560 ft^2 = 4 840 sq. yd. = 0.4047 hectare		
3. VOLUME			
1 cm^3	= 0.061 in^3;	1 in^3	= 16.387 cm^3
1 m^3	= 35.315 ft^3;	1 ft^3	= 0.0283 m^3
1 litre	= 0.0353 ft^3;	1 ft^3	= 28.3 litres
1 litre	= 0.220 imp. gal;	1 imp. gal	= 4.546 litres
1 litre	= 0.264 US gal;	1 US gal	= 3.785 litres
4. MASS			
1 msl	= 0.672 slug;	1 slug	= 1.488 msl
1 kg(m)	= 2.205 lb(m)	1 lb(m)	= 0.453 6 kg(m)
1 kg(m)	= 0.068 5 slug	1 slug	= 14.59 kg(m)
1 slug	= 32.2 lb(m);	1 lb(m)	= 0.031 08 slug
1 msl	= 9.81 kg(m);	1 kg(m)	= 0.1020 msl
5. FORCE			
1 kg(f)	= 2.205 lb(f);	1 lb(f)	= 0.4536 kg(f)

1 N	= 0.1020 kg(f);	1 kg(f)	= 9.81 N
1 N	= 10^5 dynes;	1 lb(f)	= 32.2 pdl
1 N	= 0.2248 lb(f);	1 lb(f)	= 4.448 N
1 tonne	= 1000 kg = 0.984 ton		

6. MASS DENSITY

1 msl/m³	= 0.019 slug/ft³;	1 slug/ft³	= 52.56 msl/m³
1 gm/cc	= 1 kg(m)/litre = 1000 kg(m)/m³		
1 gm/cc	= 102 msl/m³ = 1.94 slug/ft³		
1 gm/cc	= 62.4 lb(m)/ft³;	1 lb(m)/ft³	= 0.01602 gm/cc
1 slug/ft³	= 0.515 4 gm/cc		

7. WEIGHT DENSITY

1 kg(f)/m³	= 0.0624 lb(f)/ft³;	1 lb(f)/ft³	= 16.03 kg(f)/m³
1 kg(f)/m³	= 9.81 N/m³;	1 N/m³	= 0.1020 kg(f)/m³

8. PRESSURE, STRESS

1 kg(f)/cm²	= 14.22 lb(f)/in²;	1 lb(f)/in²	= 0.0703 kg(f)/cm²
1 kg(f)/m²	= 0.205 lb(f)/ft²;	1 lb(f)/ft²	= 4.882 kg(f)/m²
1 N/m²	= 0.1020 kg(f)/m²;	1 kg(f)/m²	= 9.81 N/m²
1 N/m²	= 0.0209 lb(f)/ft²;	1 lb(f)/ft²	= 47.88 N/m²

9. VISCOSITY

1 kg(f)-s/m²	= 0.205 lb(f)-s/ft² = 9.81 N.s/m²		
1 lb(f)-s/ft²	= 4.882 kg(f)-s/m² = 47.88 N.s/m²		
1 poise	= 0.002 09 lb(f)-s/ft²		
1 lb-s/ft²	= 478.8 poise	1 poise	= 0.1 N.s/m²
1 N.s/m²	= 0.1020 kg(f)-s/m²		

10. KINEMATIC VISCOSITY

1 m²/s	= 10.764 ft²/s;	1 ft²/s	= 0.092 9 m²/s
1 stoke	= 0.001 0764 ft²/s;	1 ft²/s	= 929 stokes

11. SURFACE TENSION

1 kg(f)/m	= 0.672 lb(f)/ft;	1 lb(f)/ft	= 1.488 kg(f)/m
1 N/m	= 0.06858 lb(f)/ft;	1 lb(f)/ft	= 14.583 N/m
1 N/m	= 0.1020 kg(f)/m;	1 kg(f)/m	= 9.81 N/m

12. WORK, ENERGY, QUANTITY OF HEAT

1 kg(f)-m	= 7.233 ft-lb(f);	1 ft-lb(f)	= 0.138 2 kg(f)-m
1 J	= 0.737 6 ft-lb(f);	1 ft-lb(f)	= 1.356 J
1 J	= 0.1020 kg(f)-m;	1 kg(f)-m	= 9.81 J
1 kWh	= 3.60 × 10⁶ J = 0.367 × 10⁶ kg(f)-m		
1 Btu	= 778 ft-lb(f) = 1055 J = 0.293 Wh		
1 CHU	= 1400 ft-lb(f) = 1897 J = 0.527 Wh		
1 kcal	= 426.7 kg(f)-m;	1 kg(f)-m	= 2.344 cal
1 J	= N.m = 10⁷ ergs		

13. POWER

1 hp	= 550 ft-lb(f)/sec = 746 W = 76.04 kg(f)-m/sec		
1 metric hp	= 75 kg(f)-m/sec = 0.986 hp = 735.5 W		
1 W	= 1 J/s = 1 N.m/s = 10⁷ ergs/sec		
1 kW	= 1.341 hp = 737.6 ft-lb(f)/sec		
1 kW	= 1897 CHU/hr = 3417 Btu/hr		
1 metric hp	= 632.2 kcal/hr;	1 kW	= 860 kcal/hr

14. TEMPERATURE

T°C (abs)	= TK = t°C + 273.15
T°F (abs)	= T°R* = t°F + 460
1 K	= (5/9)° R*

15. GAS CONSTANT

1 ft-lb(f)/lb(m) deg C abs $= \left(\frac{5}{9}\right)$ ft-lb(f)/lb (m) deg F abs

1 ft-lb(f)/lb(m) deg C abs = 2.99 joule/kg (m) deg C abs

1 kg(f)-m/kg(m) deg C abs = 9.81 joule/kg(m) deg C abs

1 joule/kg(m) deg C abs = 1 m²/sec² deg C abs
= 0.1020 m/deg C abs

16. ANGLE

1 rad	= 57.296°	1°	= 0.017 453 rad

* deg. Rankine

Appendix V

Source, Sink and Doublet

In the study of ideal-fluid flow certain mathematical terms are considered which are defined below.

Three Dimensional Source

A three dimensional source is defined as a point from which fluid is imagined to be issued radially at a uniform rate in all directions. It is characterised by its 'strength' m which is the rate of flow of fluid passing through any surface enclosing the source. The radial velocity V_r at any point at any point at a distance r from the source is thus equal to the strength m divided by the surface area of the sphere of radius r and centre at the source, or

$$V_r = \frac{m}{4\pi r^2}$$

Further if ϕ is the velocity potential due to the source, then since $V_r = -(\partial\phi/\partial r)$ and $V_\theta = 0$

$$-\frac{\partial\phi}{\partial r} = \frac{m}{4\pi r^2}$$

which on integration gives

$$\phi = \frac{m}{4\pi r}$$

Again if ψ is the stream function then in spherical polar coordinates

$$V_r = -\frac{1}{r^2 \sin\theta}\frac{\partial\psi}{\partial\theta} = \frac{m}{4\pi r^2}$$

which on integration gives

$$\psi = \frac{m}{4\pi}\cos\theta$$

The flow pattern resulting from a source may be represented by equipotential lines and streamlines which are given by ϕ = constant and ψ = constant respectively.

Three Dimensional Sink

A three dimensional sink is defined as a point towards which fluid is imagined to flow radially at a

uniform rate from all directions. Thus a sink may be considered as a negative source and hence the strength of sink is $-m$. The radial velocity V_r at a distance r from the sink, velocity potential ϕ and stream function ψ for the sink are therefore given by the following expressions.

$$V_r = -\frac{m}{4\pi r^2}; \phi = -\frac{m}{4\pi r}; \psi = -\frac{m}{4\pi}\cos\theta$$

Two Dimensional Source

A two dimensional source is defined as a line from which fluid is imagined to be issued at right angles to the line from every point on it at a uniform rate in all directions. Thus a two dimensional source is also known as *line source* and it is to be distinguished from a three dimensional source which is a *point source*. The 'strength' m of a two dimensional source is the volume of fluid issued per unit time per unit length of the source. Thus the radial velocity V_r at any point at a distance r from the source at right angles to it is equal to the strength m divided by the surface area of a cylinder of radius r and unit length with its axis coinciding with the source, or

$$V_r = \frac{m}{2\pi r}$$

Further if ϕ is the velocity potential due to the source, then since $V_r = -(\partial\phi/\partial r)$ and $V_\theta = 0$

$$-\frac{\partial\phi}{\partial r} = \frac{m}{2\pi r}$$

which on integration gives

$$\phi = -\frac{m}{2\pi}\log r$$

Again if ψ is the stream function then in cylindrical polar coordinates

$$V_r = -\frac{1}{r}\frac{\partial\psi}{\partial\theta} = \frac{m}{2\pi r}$$

which on integration gives

$$\psi = -\frac{m}{2\pi}\theta$$

In this case also the flow pattern may be represented by the lines of constant ϕ (equipotential lines) and constant ψ (streamlines).

Two Dimensional Sink

A two dimensional sink is defined as a line towards which fluid is imagined to flow at right angles to the line at every point on it at a uniform rate in all directions. Thus a two dimensional sink may be considered as a negative two dimensional source, and the expressions for V_r, ϕ and ψ for a two dimensional sink may be obtained by introducing $-m$ in place of m in the corresponding expressions derived for a two dimensional source.

It is evident from the definition of source and sink that a source implies the creation of fluid and a sink implies the annihilation of fluid. Further it may be stated that through source and sink are entirely imaginary, these are quite useful in obtaining flow patterns in several cases of ideal-fluid flow.

Doublet

A doublet is defined as a combination of a source of strength m and a sink of strength $-m$, placed a small distance δs apart which are allowed to approach each other (i.e., $\delta s \to 0$) in such a manner that the product of their strength and the distance between them (i.e., *mds*) remains a finite constant equal to *m*, called the strength of the doublet. The line joining source and sink is known as the axis of the doublet and is considered positive in the direction from sink to source. A doublet may also be classified as three dimensional doublet and two dimensional doublet depending upon the source and sink being three dimensional and two dimensional respectively. Further by using the principle of superposition the expressions for the velocity potential and the stream function for three dimensional and two dimensional doublets may be obtained as noted below.

Three dimensional doublet

$$\phi = \frac{\mu}{r^2}\cos\theta; \quad \psi = -\frac{\mu}{r}\sin^2\theta$$

Two dimensional doublet

$$\phi = \frac{\mu}{r}\cos\theta; \quad \psi = -\frac{\mu}{r}\sin\theta$$

The flow pattern resulting from a doublet may also be represented by lines of constant ϕ and constant ψ.

Appendix VI

Cavitation

When a liquid flows into a region where its pressure is reduced to vapour pressure, it starts vaporizing (or boiling), and vapour pockets or bubbles are formed in the liquid. These vapour bubbles are carried along with the flowing liquid unit a region of higher pressure is reached, where they suddenly collapse as the vapour condenses to liquid again. Moreover, these bubbles may also collapse even at the point where they are formed, because of the momentary local increase in the pressure at this point. When a vapour bubble collapses a cavity is formed and the surrounding liquid rushed in to fill it. This process of formation of vapour bubbles and their collapsing is called *cavitation*. In addition to this, liquids generally contain some dissolved air. This dissolved air is released as the pressure is reduced and thus air pockets or bubbles are also formed in the liquid. The air pockets are often termed as air lock due to which *air cavitation* occurs.

When the vapour bubble collapse the liquid rushing from all directions collides at the centre of the cavity, thus giving rise to very high local pressure (up to 7×10^8 N/m^2 or 7030 kg(f)/cm^2). Any solid surface in the vicinity of the cavities is also subjected to these intense pressures, because, even if the cavities are not actually at the solid surface, the pressures are propagated from the cavities by pressure waves similar to those encountered in water hammer. The formation and collapse of the large number of bubbles on a boundary surface subject that surface to intense local stressing. The boundary surface is thus alternately stressed and relieved of the stresses, thereby ultimately damaged by fatigue, resulting in serious erosion of the surface which is known as *pitting*. The phenomenon of cavitation is always accompanied by considerable noise and vibrations. Moreover, when the cavitation occurs the flow is considerably disturbed, the flow passage is damaged and the efficiency is lowered. The air cavitation is less damaging then vapour cavitation to surfaces but it also reduces the efficiency of the system considerably.

In any system of flowing liquid, the *cavitation parameter* σ is useful in characterizing the susceptibility of the system to cavitate. It is defined by

$$\sigma = \frac{p - p_v}{(\rho V^2/2)}$$

in which p is the absolute pressure at the point under consideration in the flowing system, p_v is the vapour pressure of the liquid, ρ is mass density of the liquid and V is the reference velocity of flow of liquid. The cavitation parameter is a form of pressure coefficient and it is a dimensionless number. In two geometrically similar systems if the valve of σ is same, they would be equally likely to cavitate or they would have the same degree of cavitation. Further when $\sigma = 0$, the pressure is reduced to vapour pressure of the liquid and the vaporization (or boiling) of liquid would take place.

Tests have shown that chemically pure liquids can sustain high tensile stresses (of the order of several hundreds of N/m^2 or $kg(f)/cm^2$), which is in contradiction of the concept of the formation of the cavities when pressure is reduced to vapour pressure. On the other hand in the case of commercial or practical liquids when vapour pressure is reached there is spontaneous vaporization (or boiling) and hence the cavitation occurs. As such it has been generally accepted that the vapour bubbles form and grow around some nuclei which must be present in the liquid. The nature of the nuclei is not thoroughly understood, but they may be microscopic dust particles or other contaminants, which are widely dispersed through practical liquids.

The complete phenomenon of cavitation viz., the formation of the vapour bubbles on nuclei, their growth, their movement into a region of higher pressure and finally their collapse, may take place in a few thousandths of a second. Further the surface tension of the vapour bubbles appears to be an important property accounting for the high pressure pulses resulting from the collapse of a vapour bubble.

The best protection against cavitation (if practicable) is to adopt such a hydraulic design of the system which would avoid the development of low pressure. For this the absolute pressure head in terms of water should not be allowed to fall below about 2.13 m (i.e., about 2×10^4 N/m^2 or 0.213 $kg(f)/cm^2$). However, it may not be always possible to develop completely cavitation free regions in various flow systems such as turbines, centrifugal pumps etc. In such cases use of special cavitation resistant materials or coatings may be made. Some of the cavitation resistant materials are aluminium-bronze (83% copper, 10–12% aluminium, 4-6% iron) stainless steel (with different percentages of chromium, nickel and iron) etc.

As stated in previous chapters, turbines and centrifugal pumps are subject to cavitation. In addition to these, ship propellers and many hydraulic structures are also subject to cavitation. The curved vanes of turbine runners and pump impellers are particularly susceptible to cavitation on their convex sides and may have localized areas where cavitation causes pitting or failure. The location of turbines and pumps considerably affect the cavitation in these hydraulic machines. A special cavitation factor called *Thoma's cavitation factor* (or number of index) represented by σ is considered for turbines as well as centrifugal pumps which has been indicated in the respective chapters. In addition to this another parameter called *suction specific speed* is also used to represent the cavitation characteristics of turbines and centrifugal pumps which has also been indicated in the respective chapters.

Appendix VII

Flow in Curved Channels

In the alignment of the channels, curves or bends are often required to be provided. The flow around a curved path is quite complex because the streamlines of the flows are not only curvilinear but also interwoven resulting in spiral currents and cross waves. Moreover, the centrifugal force acting on the flow around a bend produces a unique feature known as *superelevation*, that is, a rise in the water surface at the outer bank with an accompanying lowering at the inner bank. Also the velocity distribution in the channel in the bends is very irregular and the coefficients α and β are usually far greater than unity. Furthermore the characteristics of flow in curved channels are also affected by the nature of the flow viz., subcritical and supercritical. In general subcritical flow shows smooth water surface and slight superelevation, whereas supercritical flow exhibits cross-wave disturbance patterns on the surface and thus exaggerates the superelevation.

By assuming a theoretical free-vortex motion for the flow along curved channel, it is possible to obtain the expressions for the velocity distribution and the water surface profile at the section of maximum surface depression. However, this assumption holds good only for the subcritical flow.

For the free-vortex motion the following expression can be written:

$$v = \frac{C}{r} \quad ...(i)$$

where v is the filamental velocity in the curve at a radial distance r from the centre of curvature, and C is the circulation constant in a free vortex motion.

If E be the specific energy at any section and y the depth of flow at a distance r from the centre of curvature, then

$$y = E - \frac{v^2}{2g} \quad ...(ii)$$

The mean velocity of flow is

$$V = \frac{\int_{r_i}^{r_0} (C/r)\,dr}{(r_0 - r_i)} = \frac{C}{r_0 - r_i}\log_e\left(\frac{r_0}{r_i}\right)$$

where r_0 and r_i are the radii of curvature of the outer and inner banks respectively. Also the mean depth of flow is

$$y_m = \frac{\int_{r_i}^{r_0} y\,dr}{(r_0 - r_i)} = \frac{\int_{r_i}^{r_0} \left(E - \frac{C^2}{2gr^2}\right) dr}{r_0 - r_i}$$

or

$$y_m = E - \frac{C^2}{2gr_0 r_i} \quad \text{...(iv)}$$

Now the discharge is

$$Q = Vy_m(r_0 - r_i) = C\left(E - \frac{C^2}{2gr_0 r_i}\right)\log_e\left(\frac{r_0}{r_i}\right) \quad \text{...(v)}$$

If Q, r_0, r_i and E are given, the constant C can be determined from Eq. (v). The velocity and dpeth at any radius r are then obtained by Eqs. (i) and (ii). The superelevation Δh of the water surface can thus be obtained as

$$\Delta y = (y_0 - y_i) = \left(\frac{v^2}{2g}\right)_i - \left(\frac{v^2}{2g}\right)_0 = \frac{C^2}{2gr_0^2 r_i^2}(r_0^2 - r_i^2) \quad \text{...(vi)}$$

As in the case of highway and railroad curves, the bed of curved channel may also be provided with a certain cross slope, so that it would produce the desired cross slope in the water surface, which in turn would provide the necessary centripetal force. The expression defining the transverse bed profile may however be obtained as indicated below.

If H_0 represents the total energy referred to the bed level at a section just upstream of the bend, y_0 and v_0 are depth and velocity of flow respectively at this section, y and v are respectively the depth and velocity of flow at any section of the bend as defined earlier, and z is the height of the bed at any section of the bend above the bed level of the channel at the upstream section, then assuming no loss of energy, one obtains

$$z + y + \frac{v^2}{2g} = H_0 = y_0 + \frac{v_0^2}{2g}$$

But

$$y = \frac{q}{v} = \frac{qr}{C}; \quad \text{and } C = v_0 r_0$$

where q is the discharge per unit width. Thus by substitution the following expression may be obtained which defines the transverse bed profile at the channel bend

$$z + \frac{qr}{v_0 r_0} + \frac{v_0^2 r_0^2}{2gr^2} = H_0 \quad \text{..(vii)}$$

Appendix VIII

Control Valves for Pipes

Valves are required to be introduced in the pipelines for one or all the following purposes:

(i) To stop the flow of liquid altogether.

(ii) To regulate the rate of flow of the liquid.

(iii) To regulate the head or pressure in the pipelines.

According to the motion of the valve element, the control valves may be classified in the following three categories:

(a) Valve element moves transversely across the water-way.

(b) Valve element rotates about an axis transverse to the water-way.

(c) Valve element slides axially or longitudinally.

(a) With transverse motion

In this category of valves, the following two types of valves may be considered.

(i) *Full-way, or sluice or gate valve.* This is a very commonly used valve, in which the moving element has the form of a disc or circular gate, wedge-shaped in cross-section, having a nut which engage with the thread of the operating spindle as shown in the accompanying figure. In the fully-open position, the gate is withdrawn clear of the water-way and hence this valve is designated as full-way valve.

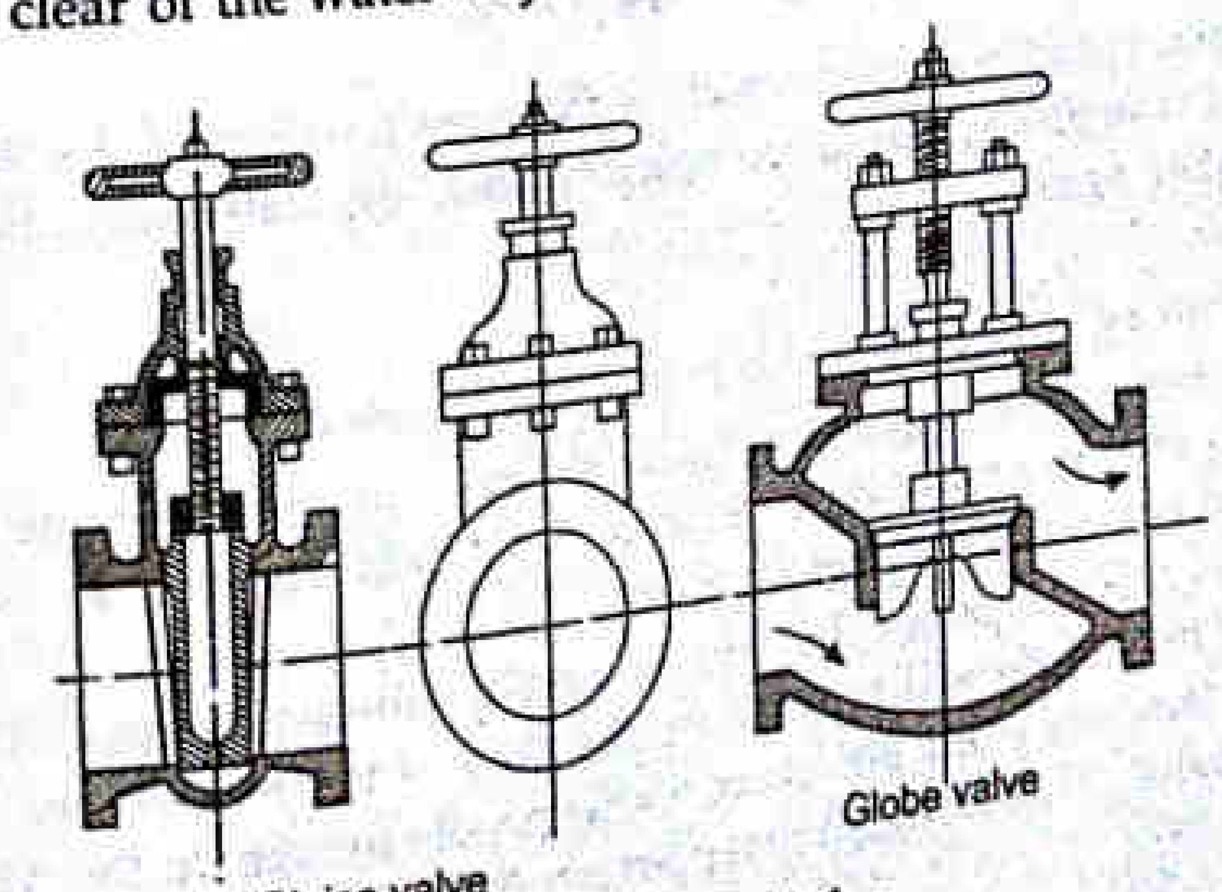


Figure App.VII–1

(ii) *Globe valve or stop valve.* As shown in the App.VIII – 2 in this case the valve element is wing-guided mitre or disc, which in the closed position of the valve is forced on its circular seat by the screwed spindle. When this spindle releases the vale element, the pressure of the incoming liquid is sufficient to lift it, thus opening the valve.

(b) With rotary motion

The following two types of valves may be considered in this category.

(i) *Butterfly valve.* In this valve the cylindrical valve body forms virtually a continuation of the pipe itself, and it houses a moving element of elliptical shape. Rotation of the vale spindle brings the valve element either into the closed position, or into the fully open position in which it is parallel with the pipe axis as shown in App. VIII–2

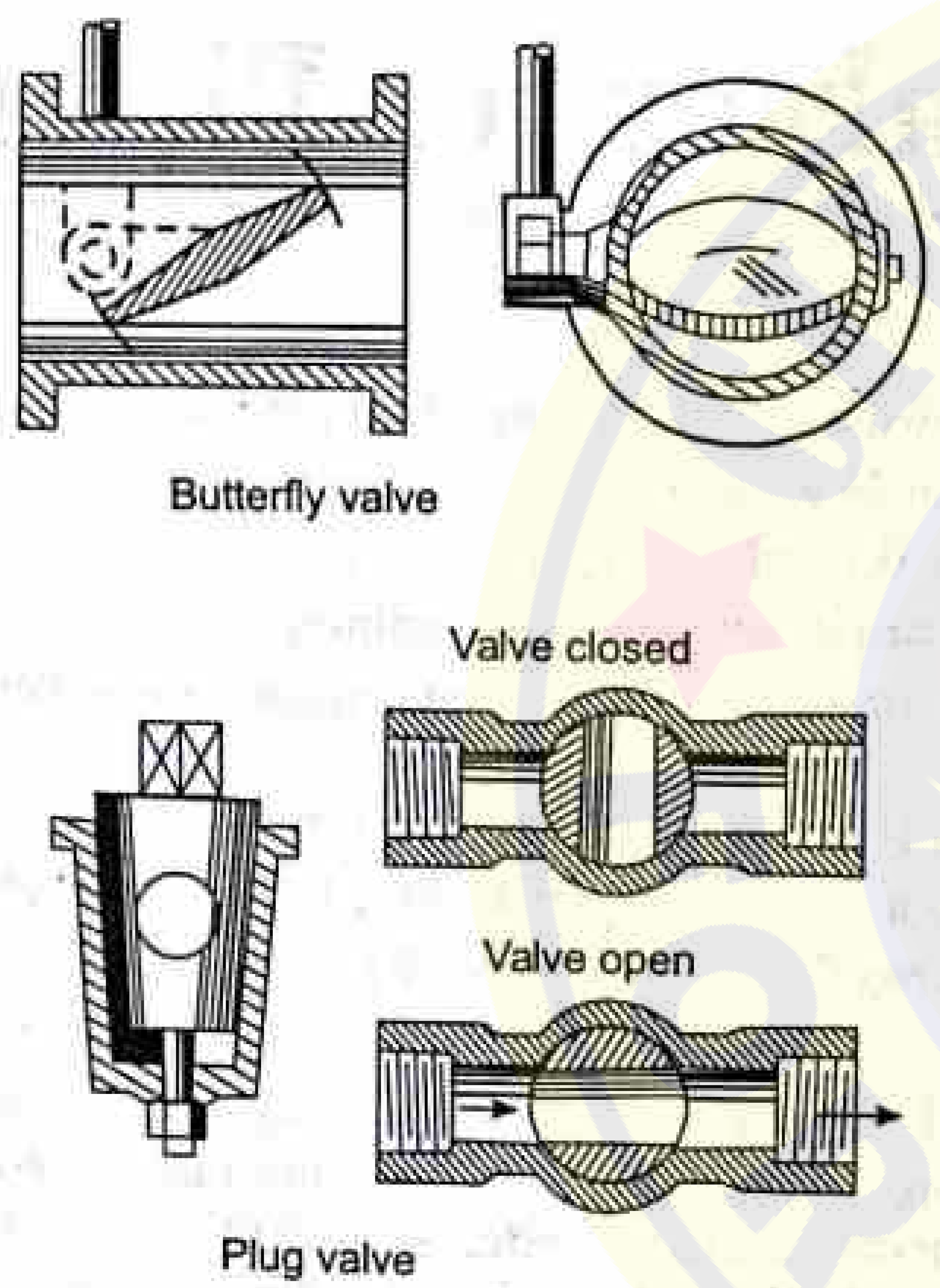


Figure App. VIII–2

(ii) *Plug valve.* A conical plug has a port which in the open position of the valve is in line with the pipe. After being turned through 90°, the plug completely closes the water-way, as shown in the sectional view in the accompanying figure.

(c) With axial motion

In this category the following two types of valves may be included:

(i) *Sleeve valve.* As shown in the figure a cylindrical sleeve surrounding the body of the valve can be moved longitudinally so as to open or close the water-way. Such valves are usually fixed at the outlet end of the pipe, where it serves as an energy disperser.

(ii) *Needle valve.* It is another type of valve with an axially moving element. The operating piston is formed integrally with the needle which regulates the area of the water-way. High pressure water admitted to one side or the other of the piston causes the needle to take up the desired position. Figure App. VIII–3 shows the valve (a) in the fully open position, (b) in the fully closed position, and (c) in the partly closed position.

The needle valve resists cavitation damage quite successfully, because as shown in the figure in the case of needle valve the zone of intense eddying lies near the axis of the water-way, far away from the pipe walls. The sluice valve is however more susceptible to cavitation damage, particularly when used continuously in partly open position. However, in spite of this drawback the sluice valve is the most

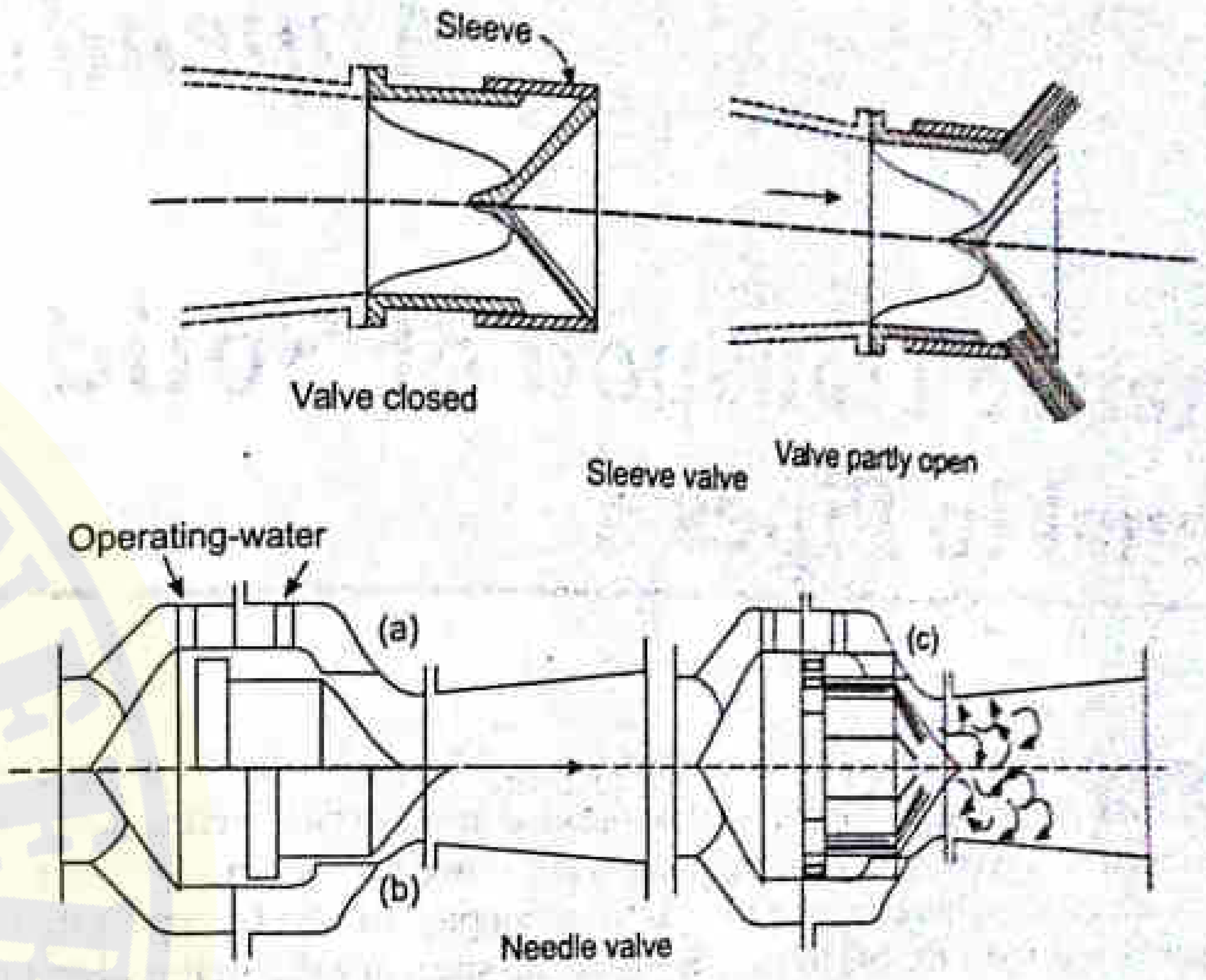


Figure App.VIII –3

commonly used control valve, being suitable for the pipes of very small as well as very large diameters. The globe valve is most widely used in the domestic water tap. However, the globe valves are also used for the pipelines of very large diameters. The butterfly valves are generally used for regulating the flow of water in penstocks supplying water to the turbines installed in hydropower stations. Similarly the needle valves are also used for regulating the flow of liquid through large diameter pipes such as penstocks.

Appendix IX

Hydraulic Transport of Solid Material in Pipes

The solid material may be transported in pipes through a liquid medium which in most of the cases is water. For many years the hydraulic transport of solid material in pipes has been successfully used in a few specialized fields such as dredging, land reclamation etc. However, this mode of transport of solid material is becoming more and more popular and it is being adopted for the transportation of coal, ash, natural sand, gravel, metal ores, crushed rock etc. Since the transport of solid material in pipes involves the flow of liquid-solid mixture, it is evident that the flow characteristics for this case would be different from those for the flow of pure water or any other liquid (free from solid material) in pipes. Moreover, the behaviour of the flow of a given liquid-solid mixture would be quite different in the horizontal and vertical pipes. As such in order to study the characteristics of flow of liquid-solid mixtures in pipes, experimental studies have been carried out by several investigators. The results of some of these investigations are briefly indicated below.

It has been observed that when liquid carries in suspension solid grains or particles not larger than about 0.15 mm, then the mixture can be regarded as a simple liquid having the same viscosity as that of liquid, but of a density increased by the presence of the suspended solids. The specific gravity S of such a liquid-solid mixture may be expressed as

$$S = S_l + C_v(S_s - S_l) \qquad \text{...(i)}$$

where S_l is the specific gravity of the pure liquid, C_v is the concentration of solids by volume (i.e., unit volume of mixture contains a volume C_v of solids) and S_s is the true specific gravity in air of the dry solid particles. For water since $S_l = 1$ the above expression becomes

$$S = 1 + C_v(S_s - 1) \qquad \text{...(ii)}$$

In a horizontal pipe carrying a mixture of water and sand grains in suspension, the concentration of solids is more near the bottom of the pipe than near the top of the pipe. Further it has been observed that when the sand grains are relatively small having a mean diameter of only 0.18 mm, then the velocity distribution curve in a vertical plane is symmetrical about the pipe axis and it is same as that for pure water. This implies that for the flow of mixture of water and sand gains of diameter less than about 0.18 mm in pipes, the various turbulent flow formulae for the flow of pure water in pipes may be applied, no matter whether the pipe is horizontal or vertical or inclined.

When the size of the solid material transported along with the liquid increases, then in a horizontal pipe the solids tend more and more to settle to the bottom of the pipe. The velocity distribution curve becomes unsymmetrical, because the velocity near the bottom of the pipe is considerably reduced. Further in this case the frictional resistance increases with the increase in the concentration of the solids. Figure App. IX–2 shows he results of a study conducted for the flow of liquid-solid mixture consisting of water and lumps of coal of about 12.5 mm diameter in a pipe of 75 mm diameter. It may be noted from the figure that as the concentration of solids is increased from 5% to 20% the head loss is considerably increased. Further when the mixture is flowing at a mean velocity of about 2.4 m/s (or 8 ft/s) or more the suspended lumps of coal do not affect the friction loss very much as it is only a little greater than that for clear water. As the mean velocity diminishes, the head loss also diminishes, but at one stage the head loss again rises as the mean velocity falls. Finally at low flow rates the head loss of each concentration of mixture tends to become independent of the mean velocity of flow. If corresponding to these constant values of the head loss, the values of friction factor f are computed then it is found that these values of f may be about 20 times higher than the equivalent values of f for clear water. The increase in the head loss at low velocities of flow may be due to the fact that at low velocities much of the solid material remains deposited at the bottom of the pipe, on account of which a portion of the pipe cross-section gets virtually blocked. As such the true area available for the flow is much less than the nominal area ($\pi d^2/4$) and the actual mean velocity is higher than the nominal velocity $[Q/(\pi d^2.4)]$.

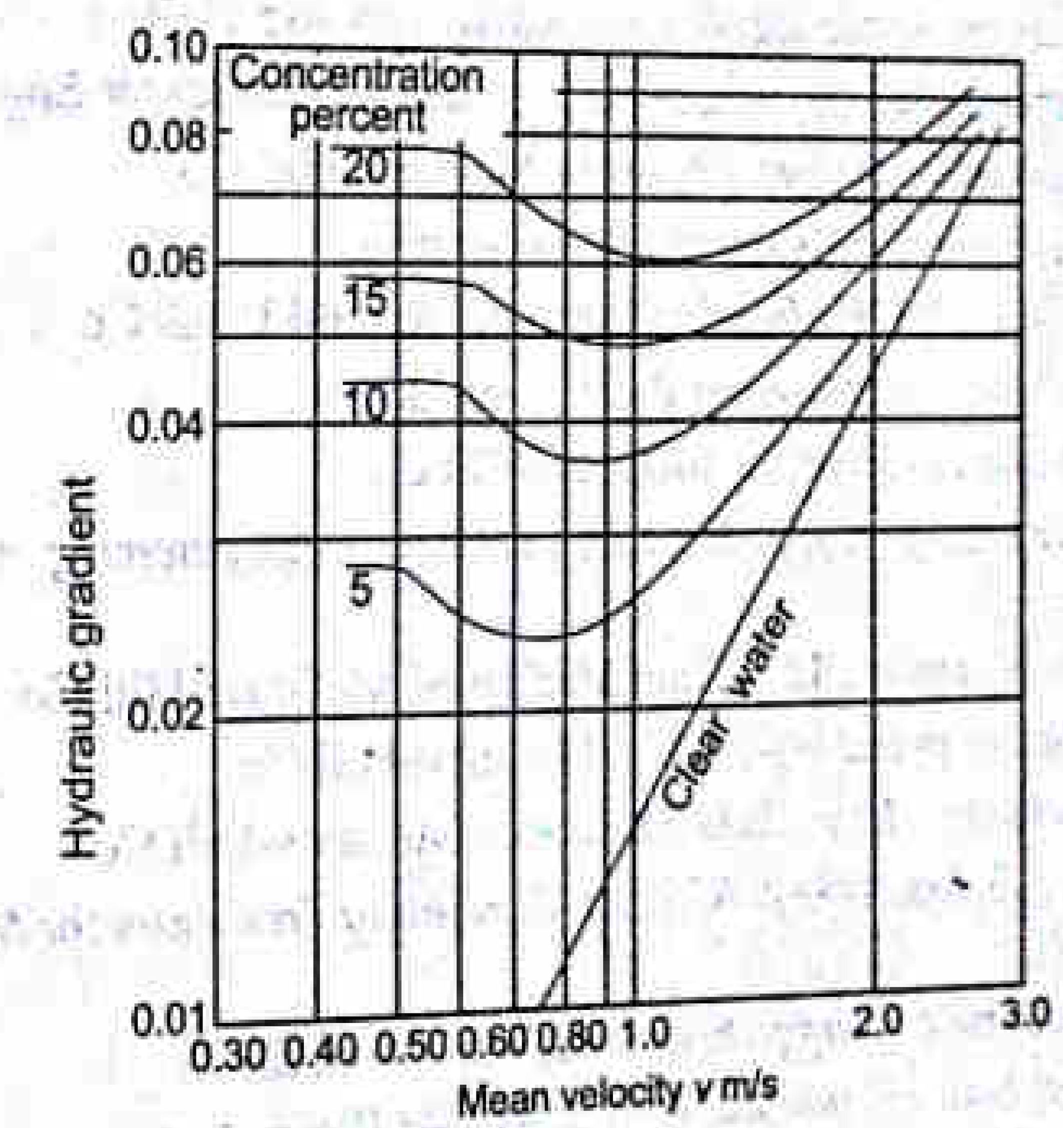


Figure App. IX. 1 *Relation between concentration of solids, mean velocity and hydraulic gradient*

When a liquid-solid mixture flows through a vertical pipe, the conditions are relatively simple as compared to those in a horizontal pipe. In a vertical pipe there is no question of differential velocities or differential settlement. However, in this case the solid particles tend to slip backwards through the water and therefore the mean rate at which the solids rise is less then the mean upward water velocity. This tendency of solids would result in higher friction loss.

For hydraulic transport of solids special centrifugal pumps are utilised whose impellers are capable of passing the largest pieces likely to occur in the mixture. Sometimes the solids and water in the desired proportions are fed to a hopper connected to the suction branch of the pump. The pump then delivers the mixture into the pipeline at a pressure sufficient to overcome the frictional resistance and any static head. When the solids are to be recovered, screening or other separating plant is used at the delivery end of the pipeline.

Bibliography

1. Addison, H, *A Treastise on Applied Hydraulics*, Chapman and Hall.
2. Albertson, M.L., Barton, J.R., and Simons, D.B. , *Fluid Mechanics for Engineers*, Prentice-Hall.
3. Bakhmeteff, B.A., *Hydraulics of Open Channels*, McGraw-Hill Co.
4. Barna, P.S., *Fluid Mechanics for Engineers*, Butterworths.
5. Beitler, S.R. and Lindahl, E.J, *Hydraulic Machinery*, Ronald Press Co.
6. Brown, J.G., *Hydro-Electric Engineering Practice*, Blackie.
7. Chow, V.T., *Open Channel Hydraulics*, McGraw-Hill Co.
8. Daugherty, R.L., and Franzini, J.B., *Fluid Mechanics with Engineering Applications*, McGraw-Hill Co.
9. Daugherty, R.L., and Ingersoll, J.C., *Fluid Mechanics*, McGraw-Hill Co.
10. Davis, C.V., *Handbook of Applied Hydraulics*, McGraw-Hill Co.
11. Dodge, R.A., and Thompson, M.J., *Fluid Mechanics*, McGraw-Hill Co.
12. Duncan, W.J., Thom, A.S, and Young A.D., *An Elementary Treatise on the Mechanics of Fluids*, Arnold Press.
13. Elevatorski, E.A., *Hydraulic Energy Dissipators*, McGraw-Hill Co.
14. Francis, J.R.D., *A Text Book of Fluid Mechanics*, Arnold Press.
15. Giles, R.V., *Theory and Problems of Fluid Mechanics and Hydraulics*, Schaum.
16. Jagdish Lal, *Hydraulic Machines*, Metropolitan Book Co., Delhi. .
17. Jagdish Lal, *Hydraulic Machines*, Metropolitan Book Co., Delhi.
18. Hughes, W.F., and Brighton, J.A., *Theory and Problems of Fluid Dynamics*, Schaum.
19. Kaufmann, W., *Fluid Mechanics*, McGraw-Hill Co.
20. Kovalev, N.N., *Hydroturbines-Design and Construction*, Israel Program for Scientific Translations, Jerusalem.
21. Langhaar, H.L, *Dimensional Analysis and Theory of Models*, John Wiley.
22. Lewitt, E.H., *Hydraulic and the Mechanics of Fluids*, Pitman.
23. Li, W.H., and Lam, S.H., *Principles of Fluid Mechanics*, Addision-Wisley.
24. Massey, B.S., *Mechanics of Fluid*, Van Nostrard Co.
25. Nechleba, M., *Hydraulic Turbines, Their Design and Equipment*, Artia-Prague.

26. Pao, R.H.F., *Fluid Mechanics*, John Wiley.
27. Paradise, R.S., *Problems in Hydraulics*, Blackie.
28. Powell, R.W., *An Elementary Text in Hydraulic and Fluid Mechanics*, Mac Millan.
29. Rouse, H., *Elementary Mechanics of Fluids*, John Wiley.
30. Rouse, H., *Fluid Mechanics for Hydraulic Engineers*, Dover.
31. Rouse, H., *Engineering Hydraulics*, John Wiley.
32. Russell, G.E., *Hydraulics*, Oxford
33. Schlichting, H., *Boundary Layer Theory*, McGraw-Hill Co.
34. Shames, I.H., *Mechanics of Fluids*, McGraw-Hill Co.
35. S mith, D.R.L., and Houghton, J., *Fluid Mechanics*, B.I.P. Bombay.
36. Stepanoff. A.J., *Centrifugal and Axial Flow Pumps*, John Wiley.
37. Streetr, V.L., *Fluid Mechanics*, McGraw-Hill Co.
38. Troskolanski, A.T., *Hydrometry*, Pergaman Press.
39. Vennard, J.K., *Elementary Fluid Mechanics*, John Wiley.
40. Weber, N.B., *Fluid Mechanics for Civil Engineers*, Span.
41. Wislicenus, G.F., *Fluid Mechanics of Turbomachinery*, Dover.
42. Woodward, S.M., and Posey, C.J., *Hydraulics of Steady Flow in Open Channels*, John Wiley

Author Index

Subject Index

D

E

F

G

H

I

T

U

V

W