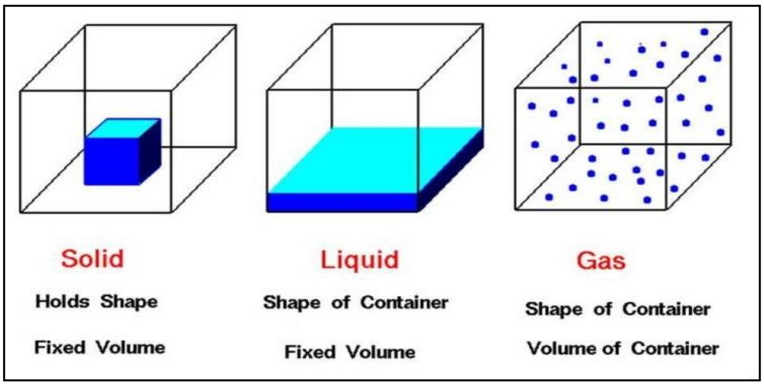
INTRODUCTION

1.1 Hydraulics:
Hydraulics (this word has been derived from a Greek word 'Hudour' which means water)
Fluid Mechanics:
Fluid mechanics may be defined as that branch of Engineering-science which deals with the behavior of fluid under the conditions of rest and motion.
The fluid mechanics may be divided into two parts: Statics and dynamics.
Statics: The study of incompressible fluids under static conditions.
Dynamics: It deals with the relations between velocities, accelerations of fluid with the forces or energy causing them.

Fluid
A fluid is a substance which deforms continuously when subjected to external shearing force.
A fluid has the following characteristics:
1. It has no definite shape of its own, but conforms to the shape of the containing vessel.
2. Even a small amount of shear force exerted on a liquid/fluid will cause a deformation which continues as long as the force continues to be applied.
A fluid may be classified as follows:
a) (i) Liquid (ii) Gas (iii) Vapour.
b) (i) Ideal fluids (ii) Real fluids.

Liquid: It possesses a definite volume
As the contraction of volume of a liquid under compression is extremely small, it is usually ignored and the liquid is assumed to be incompressible.
A liquid will withstand a slight amount of tension due to molecular attraction between the particles which will cause an apparent shear resistance, between two adjacent layers. This phenomenon is known as viscosity.
Gas: It possesses no definite volume and is compressible.
Vapour. It is a gas whose temperature and pressure are such that it is very near to the liquid state (e.g., steam).
Ideal fluids: An ideal fluid is one which has no viscosity and surface tension and is incompressible. In true sense no such fluid exists in nature. However fluids which have low viscosities such as water and air can be treated as ideal fluids under certain conditions. The assumption of ideal fluids helps in simplifying the mathematical analysis.
Real fluids: A real practical fluid is one which has viscosity, surface tension and compressibility in addition to the density. The real fluids are actually available in nature.



1.2 Dimension: Generalization of "unit" telling us what kind of units are involved in a quantitative statement.
The primary quantities of fluid are:

Table 1 Basic Dimensions and Their Units

Quantity	Dimension	SI Units	English Units
Length l	L	meter m	foot ft
Mass m	M	kilogram kg	slug slug
Time t	T	second s	second sec
Temperature T	Θ	kelvin K	Rankine R
Plane angle		radian rad	radian rad

Table 2 Derived Dimensions and Their Units

Quantity	Dimension	SI Units	English Units
Area A	L^2	m^2	ft^2
Volume* $\forall$	L^3	m^3 or L (liter)	ft^3
Velocity V	L/T	m/s	ft/sec
Acceleration a	L/T^2	m/s^2	ft/sec^2
Force F	ML/T^2	$kg \cdot m/s^2$ or N	$slug \cdot ft/sec^2$ or lb
Density ρ	M/L^3	kg/m^3	$slug/ft^3$
Specific weight γ	M/L^2T^2	N/m^3	lb/ft^3
Frequency f	T^{-1}	s^{-1}	sec^{-1}
Pressure p	M/LT^2	N/m^2 or Pa	lb/ft^2
Stress τ	M/LT^2	N/m^2 or Pa	lb/ft^2
Surface tension σ	M/T^2	N/m	lb/ft
Work W	ML^2/T^2	$N \cdot m$ or J	$ft \cdot lb$
Energy E	ML^2/T^2	$N \cdot m$ or J	$ft \cdot lb$
Heat rate $\dot{Q}$	ML^2/T^3	J/s	Btu/sec
Torque T	ML^2/T^2	$N \cdot m$	$ft \cdot lb$
Power $\dot{W}$	ML^2/T^3	J/s or W	$ft \cdot lb/sec$
Mass flux $\dot{m}$	M/T	kg/s	slug/sec
Flow rate Q	L^3/T	m^3/s	ft^3/sec
Specific heat c	$L^2/T^2\Theta$	$J/kg \cdot K$	$Btu/slug \cdot {}^\circ R$
Viscosity μ	M/LT	$N \cdot s/m^2$	$lb \cdot sec/ft^2$
Kinematic viscosity ν	L^2/T	m^2/s	ft^2/sec

*We use the special symbol $\forall$ to denote volume and V to denote velocity.

Table 3 | Multiplier factors for SI units.

Multiplier	Prefix	Abbreviation
10^{12}	tera	T
10^{9}	giga	G
10^{6}	mega	M
10^{3}	kilo	k
10^{2}	hecto	h
10^{-2}	centi	c
10^{-3}	milli	m
10^{-6}	micro	μ
10^{-9}	nano	n
10^{-12}	pico	p
10^{-18}	atto	a

1.3 PHYSICAL PROPERTIES OF FLUIDS

1.3.1 Density

1. Mass density

The density (also known as mass density or specific mass) of a liquid may be defined as the mass per unit volume $\left(\frac{m}{V}\right)$ at a standard temperature and pressure. It is usually denoted by $\boldsymbol{\rho}$ (rho).

Its units are kg/m^3 i.e. $\rho = \left(\frac{m}{V}\right)\ \frac{kg}{m^3}$

2. Weight density

The weight density (also known as specific weight) is defined as the weight per unit volume at the standard temperature and pressure. It is usually denoted by (w).

$$w = \rho . g$$

3. Specific volume

It is defined as volume per unit mass of fluid. It is denoted by v. Mathematically,

$$v = \frac{V}{m} = \frac{1}{\rho} \quad \frac{m^3}{kg}$$

1.3.2 Specific Gravity

Specific gravity is the ratio of the specific weight of the liquid to the specific weight of a standard fluid. It is dimensionless and has no units. It is represented by *S*.

For liquids, the standard fluid is pure water at 4°C.

$$Specific\ gravity = \frac{Specific\ weight\ of\ liquid}{Specific\ weight\ of\ pure\ water} = \frac{w_{liquid}}{w_{water}}$$

Illustrative Example

Calculate the specific weight, specific mass, specific volume and specific gravity of a liquid having a volume of 6 m^3 and weight of 44 kN.

Solution:

Volume of the liquid = 6 m^3

Weight of the liquid = 44 kN

Specific weight, w: $w = \frac{weight\ of\ liquid}{volume\ of\ liquid} = \frac{44}{6} = 7.33 \quad kN/m^3$

Specific mass or mass density, :	$\rho = \frac{w}{g} = \frac{7.33\times1000}{9.81} = 747.5\ kg/m^3$
Specific volume,	$v = \frac{V}{m} = \frac{1}{\rho} = \frac{1}{747.5} = \ 0.00134\, m3/kg$
Specific Gravity, *S*	$S = \frac{w_{liquid}}{w_{water}} = \frac{7.333}{9.81}$

1.3.3 Viscosity

Viscosity may be defined as the property of a fluid which determines its resistance to shearing stresses. It is a measure of the internal fluid friction which causes resistance to flow. Viscosity of fluids is due to cohesion and interaction between particles.

Refer Fig(2). When two layers of fluid, at a distance *dy*, move one over the other at different velocities, say *u* and *u* + *du*, the viscosity together with relative velocity causes a shear stress acting between the fluid layers. The top layer causes a shear stress on the adjacent lower layer while the lower layer causes a shear stress on the

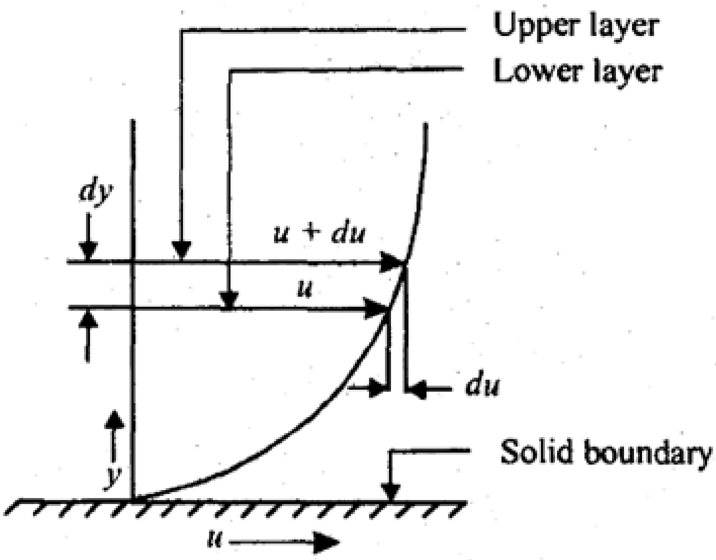


Fig. 2 Velocity variation near a solid boundary.

adjacent top layer. This shear stress is proportional to the rate of change of velocity with respect to *y*. It is denoted by τ (called Tau).

Mathematically

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

Or

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

where, μ = Constant of proportionality and is known as coefficient of dynamic viscosity or only viscosity.

So

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

Units of Viscosity:
In S.I. Units: N.s/m^2

$$\left[\mu = \frac{force/area}{(length/time)\times\frac{1}{length}} = \frac{force/length^2}{\frac{1}{time}} = \frac{force\times time}{length^2}\right] = \frac{N.s}{m^2}$$

The unit of viscosity is also called poise, One poise = $\frac{1}{10}$N.s/m^2

Note. The viscosity of water at 20°C is $\frac{1}{100}$ poise or one centipoise.

Kinematic Viscosity:
Kinematic viscosity is defined as the ratio between the dynamic viscosity and density of fluid
It is denoted by $\boldsymbol{v}$ (called nu). Mathematically:

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

Units of kinematic viscosity:
In SI units: m^2/s
Or stokes, one stokes = 0.0001 m^2/s

Newton's Law of Viscosity
This law states that the shear stress (τ) on a fluid element layer is directly proportional to the rate of shear strain. The constant of proportionality is called the co-efficient of viscosity.

$$\tau = \mu.\frac{du}{dy} \qquad 1$$

Types of fluids
The fluids may be of the following
'Refer to Fig. (3)
1. **Newtonian fluids**. These fluids follow Newton's viscosity equation (i.e. eq. 1). For such fluids μ does not change with rate of deformation. [Examples. Water, kerosene, air].
Newtonian fluids: $\tau = \mu.\frac{du}{dy}$
2. **Non-Newtonian fluids**, fluids which do not follow the linear relationship between shear stress and rate of deformation given by eqn. 1 are termed as non-Newtonian fluids. Such fluids relatively uncommon. [Examples. Solutions or suspensions, mud, blood]. These fluids are generally complex mixtures.
Non-Newtonian fluids: $\tau = \mu.\left(\frac{du}{dy}\right)^n$
3. **Plastic fluids**. its non-Newtonian fluid These substances are represented by straight line intersecting the vertical axis Refer to Fig.(3).
4. **Ideal fluid**. An ideal fluid is one which is incompressible and has 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 (τ = 0).
Ideal fluid: τ = 0

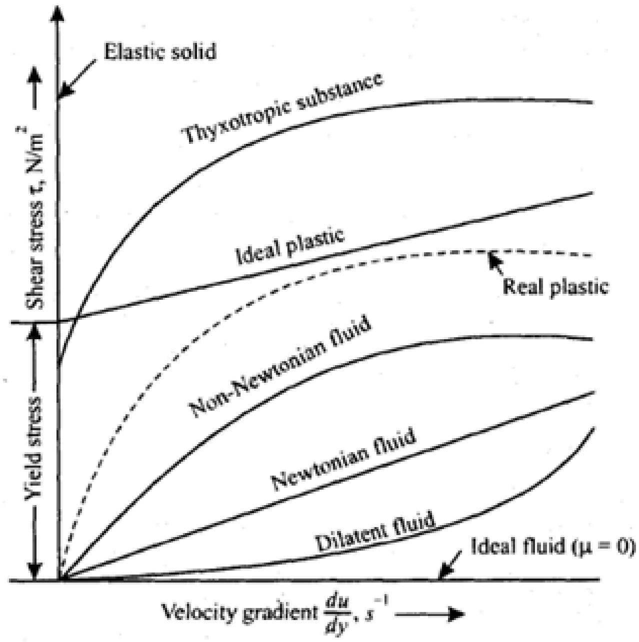


Fig. 3 Variation of shear stess with velocity gradient.

Effect of Temperature on Viscosity

Viscosity is effected by temperature. The viscosity of liquids decreases but that of gases increases with increase in temperature. This is due to the reason that in liquids the shear stress is due to the inter-molecular cohesion which decreases with increase of temperature. In gases the inter-molecular cohesion is negligible and the shear stress is due to exchange of momentum of the molecules, normal to the direction of motion. The molecular activity increases with rise in temperature and so does the viscosity of gas.

Effect of Pressure on Viscosity

The viscosity under ordinary conditions is not appreciably affected by the changes in pressure. However, the viscosity of some oils has been found to increase with increase in pressure.

1.3.4 Surface tension

It results from the attractive forces between molecules. It allows steel to float, droplets to form, and small droplets and bubbles to be spherical. Consider the free-body diagram of a spherical droplet and a bubble, as shown in Fig. (4).

The pressure force inside the droplet balances the force due to surface tension around the circumference:

$$P\pi r^2 = 2\pi r\sigma$$

So

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

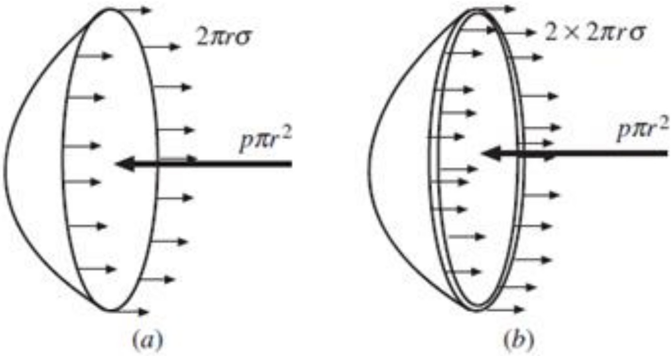


Figure 4 Free-body diagrams of (*a*) a droplet and (*b*) a bubble.

Notice that in a bubble there are two surfaces so that the force balance provides

$$P = \frac{4\sigma}{r}$$

So, if the internal pressure is desired, it is important to know if it is a droplet or a bubble.

A second application where surface tension causes an interesting result is in the rise of a liquid in a capillary tube. The free-body diagram of the water in the tube is shown in Fig. (5). Summing forces on the column of liquid gives

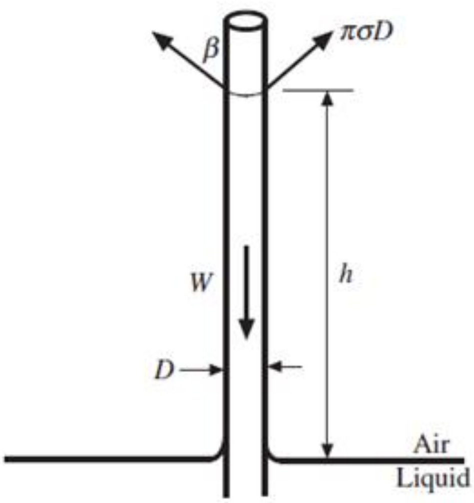


Figure 5 The rise of a liquid in a small tube.

$$\sigma\pi D\ cosB = \rho g\frac{\pi D^2}{4}h$$

where the right-hand side of the equation is the weight *W*. This provides the height the liquid will climb in the tube:

$$h = \frac{4.\sigma.cosB}{w.D}$$

Illustrative Example

A machine creates small 1.0-mm-diameter bubbles of 20°C-water. Estimate the pressure that exists inside the bubbles.

Solution

Bubbles have two surfaces leading to the following estimate of the pressure:

$$p = \frac{4\sigma}{r} = \frac{4 \times 0.0736}{0.0005} = 589 \text{ Pa}$$

where the surface tension was taken from Table

1.3.5 Vapor pressure

Molecules escape and reenter a liquid that is in contact with a gas, such as water in contact with air. The *vapor pressure* is that pressure at which there is equilibrium between the escaping and reentering molecules. If the pressure is below the vapor pressure, the molecules will escape the liquid; it is called *boiling* when water is heated to the temperature at which the vapor pressure equals the atmospheric pressure.

1.4 Useful Information

1-The shear stress [symbol: τ (tau)]

It is the force per unit surface area that resists the sliding of the fluid layers. The common units used of shear stress is ($N/m^2 \equiv Pa$)

2-The pressure [symbol: P]

It is the force per unit cross sectional area normal to the force direction.
The common units used of shear stress is ($N/m^2 \equiv Pa$), (atm) (bar) (Psi) (mmHg). The pressure difference between two points refers to (ΔP).
The pressure could be expressed as liquid height (or head) (h)

3-The energy [symbol: E]

Energy is defined as the capacity of a system to perform work or produce heat.
There are many types of energy such as [Internal energy (U), Kinetic energy (K.E), Potential energy (P.E), Pressure energy (Prs.E), and others.
The common units used for energy is ($J \equiv N.m$), (Btu), (cal).
The energy could be expressed in relative quantity per unit mass or mole (J/kg or mol).

4-The Power [symbol: P]

It is the energy per unit time. The common units used for Power is ($W \equiv J/s$)

5- The flow rate

5.1-Volumetric flow rate [symbol: Q]
It is the volume of fluid transferred per unit time.

$$Q = u\,A$$

where A: is the cross sectional area of flow normal to the flow direction. The common units used for volumetric flow is (m^3/s), (cm^3/s), (ft^3 /s).

5.2-Mass flow rate [symbol: m]
It is the mass of fluid transferred per unit time.

$$\dot{m} = Q\,\rho = u\,A\,\rho$$

The common units used is (kg/s), (g/s), (lb/s).

1.5 Important Laws

1-Law of conservation of mass

“ *The mass can neither be created nor destroyed, and it cannot be created from nothing”*

2-Law of conservation of energy (First law of thermodynamics)

“ *The energy can neither be created nor destroyed, though it can be transformed from one form into another”*

Newton’s Laws of Motion

Newton has formulated three law of motion, which are the basic assumption on which the whole system of dynamics is based.

3-Newton’s first laws of motion

“Everybody continues in its state of rest or of uniform motion in a straight line, unless it is acted upon by some external forces”

4-Newton’s second laws of motion

“The rate of change in momentum is directly proportional to the impressed force and takes place in the same direction in which the force acts”[momentum = mass × velocity]

5-Newton’s third laws of motion

“To every action, there is always an equal and opposite reaction”

CHAPTER TWO

FLUID STATICS

The subject of fluid statics involves fluid problems in which there is no relative motion between fluid particles. If no relative motion exists between particles of a fluid, viscosity can have no effect.

Pressure-Density-Height Relationships.

The fundamental equation of fluid statics is that relating pressure, density, and vertical distance in a fluid. This equation may be derived readily by considering the vertical equilibrium of an element of fluid such as the small cube of Fig. 2.1. Let this cube be differentially small and have dimensions *dx*, *dy*, and *dz*, and assume that the density of the fluid in the cube is uniform. If the pressure upward on the bottom face of this cube is p, the force due to this pressure will be given by $(p\ dx\ dy)$. Assuming an increase of pressure in the positive direction of z, the pressure downward on the top face of the cube will be $\left(p + \frac{dp}{dz}dz\right)$, and the force due to this pressure will be $\left(p + \frac{dp}{dz}dz\right)dx\ dy$. The other vertical force involved is the weight, dW, of the cube, given by

$$dW = w\ dxdy\ dz$$

The vertical equilibrium of the cube will be expressed by

$$\left(p + \frac{dp}{dz}dz\right)dx\ dy + w\ dxdy\ dz - p\ dx\ dy = 0$$

This will be

$$\frac{d}{d}$$

the fundamental equation of fluid statics, which must be integrated for the solution of engineering problems. Such integration may be accomplished by transposing the terms w and dz, resulting in

$$\frac{d|}{w}$$

which may be integrated as follows :

$$\int_{p_2}^{p_1} \frac{dp}{w} = -$$

Gives

$$\int_{p_2}^{p}$$

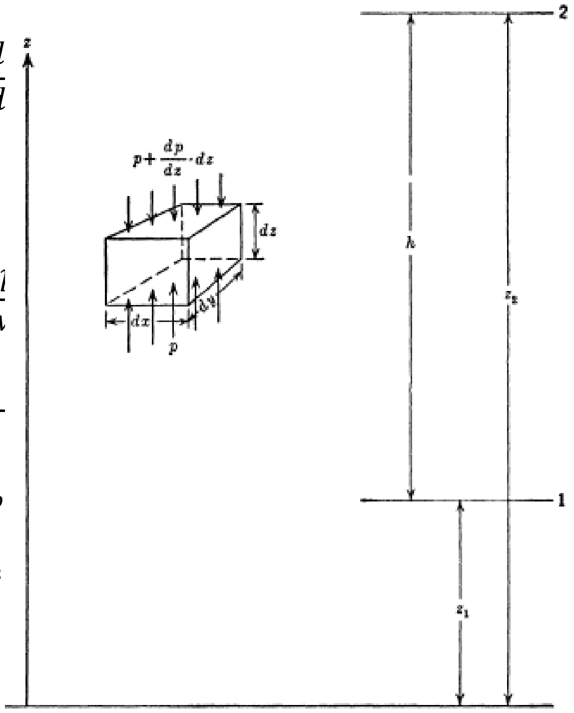


Fig 2.1

in which p_1 is the greater pressure existing at the lower point 1, p_2 the lesser pressure existing at the upper point 2, and h the vertical distance between these points. The integration of the left-hand side of the equation cannot be carried out until $w = f(p)$ is known. For gases this relationship may be obtained from certain laws of thermodynamics. For liquids the specific weight, w, is sensibly constant allowing integration of the equation to

$$\frac{p_1 - p_2}{w} = \Delta h \quad \text{or} \quad p_1 - p_2 = w\Delta h = \rho g \Delta h \qquad (1)$$

permitting ready calculation of the increase in pressure in a liquid as depth is gained. It should be noted that equation 1 embodies certain basic and familiar facts concerning fluids at rest. It shows that, if $h = 0$, the pressure difference is zero and thus pressure is constant over horizontal planes in a fluid.
Equation 1 also indicates the fact that pressure at a point in a liquid of given density is dependent solely upon the height of the liquid above the point, allowing this vertical height, or "head," of liquid to be used as an indication of pressure. Thus pressures maybe quoted in "cm of mercury," "meter of water/' etc.

Pressure in a Fluid

In Figure (2.2) a stationary column of fluid of height (h_2) and cross-sectional area A, where $A=A_o=A_1=A_2$, is shown. The pressure above the fluid is P_o, it could be the pressure of atmosphere above the fluid. The fluid at any point, say h_1, must support all the fluid above it. It can be shown that the forces at any point in a nonmoving or static fluid must be the same in all directions. Also, for a fluid at rest, the pressure or (force / unit area) in the same at all points with the same elevation. For example, at h_1 from the top, the pressure is the same at all points on the cross-sectional area A_1.
The total mass of fluid for h_2, height and ρ density is: $(h_2\, A\, \rho)$ *(kg)*
The pressure is defined as ($P = F/A = h_2\, \rho\, g$) (N/m^2 or Pa)
So the total force of the fluid on area (A) due to the fluid only is: -

$$F = h_2\, A\, \rho\, g\ (N)$$

This is the pressure on A_2 due to the weight of the fluid column above it. However to get the total pressure P_2 on A_2, the pressure P_o on the top of the fluid must be added,

$$i.e. P_2 = h_2\, \rho\, g + P_o\ (N/m^2\ or\ Pa)$$

Thus to calculate P_1:

$$P_1 = h_1 \rho\, g + P_o\ (N/m^2\ or\ Pa)$$

The pressure difference between points 1 and 2 is: -

$$P_2 - P_1 = (h_2\, \rho\, g + P_o) - (h_1\, \rho\, g + P_o)$$

$$\Rightarrow P_2 - P_1 = (h_2 - h_1)\, \rho\, g \qquad \textit{see eq.1}$$

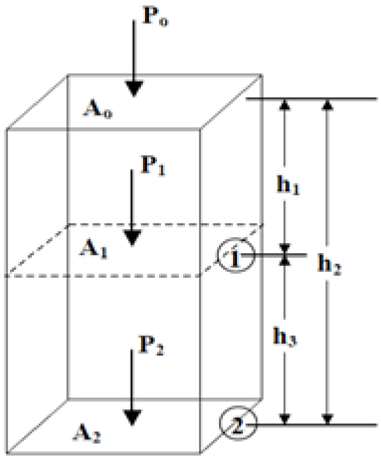


Fig. 2.2

Since it is vertical height of a fluid that determines the pressure in a fluid, the shape of the vessel does not affect the pressure. For example in Figure (2.3) the pressure P_1 at the bottom of all three vessels is the same

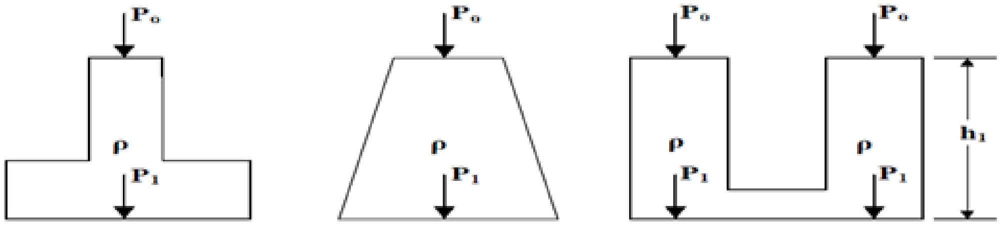


Figure 2.3: Pressure in vessel of various shapes.

and equal to ($h_1\, \rho\, g + P_o$).

Absolute and Relative Pressure

The term pressure is sometimes associated with different terms such as atmospheric, gauge, absolute, and vacuum. The meanings of these terms have to be understood well before solving problems in fluid mechanics.

1-Atmospheric Pressure

It is the pressure exerted by atmospheric air on the earth due to its weight. This pressure is change as the density of air varies according to the altitudes. Greater the height lesser the density. Also it may vary because of the temperature and humidity of air. Hence for all purposes of calculations the pressure exerted by air at sea level is taken as standard and that is equal to: -

1 atm = 1.01325 bar = 101.325 kPa

2-Gauge Pressure or Positive Pressure

It is the pressure recorded by an instrument. This is always above atmospheric. The zero mark of the dial will have been adjusted to atmospheric pressure.

3-Vacuum Pressure or Negative Pressure

This pressure is caused either artificially or by flow conditions. The pressure will be less than the atmospheric pressure.

4-Absolute Pressure

Absolute pressure is the algebraic sum of atmospheric pressure and gauge pressure. Atmospheric pressure is usually considered as the datum line and all other pressures are recorded either above or below it.

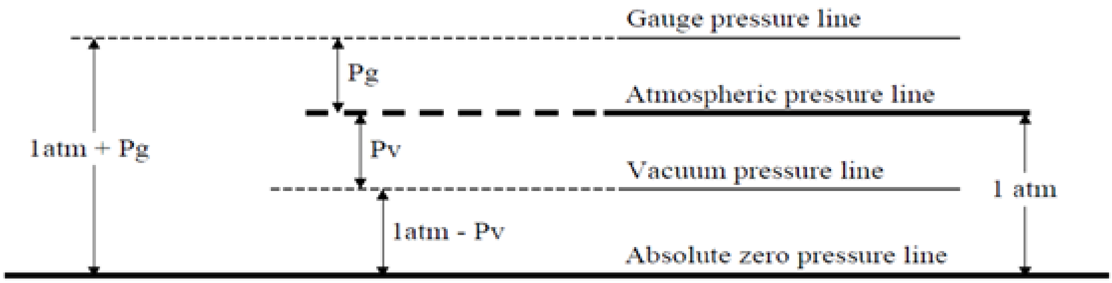


Fig 2.4

Absolute Pressure = Atmospheric Pressure + Gauge Pressure

Absolute Pressure = Atmospheric Pressure – Vacuum Pressure

For example if the vacuum pressure is 0.3 atm

Absolute pressure = 1.0 – 0.3 = 0.7 atm

Note: -

Barometric pressure is the pressure that recorded from the barometer (apparatus used to measure atmospheric pressure).

3.4 Head of Fluid

Pressures are given in many different sets of units, such as N/m^2, or Pa. However a common method of expressing pressures is in terms of head (m, cm, mm) of a particular fluid. This height or head of the given fluid will exert the same pressure as the pressures it represents.

$P = h \rho g.$

Measurement of Fluid Pressure

In chemical and other industrial processing plants it is often to measure and control the pressure in vessel or process and/or the liquid level vessel.

The pressure measuring devices are: -

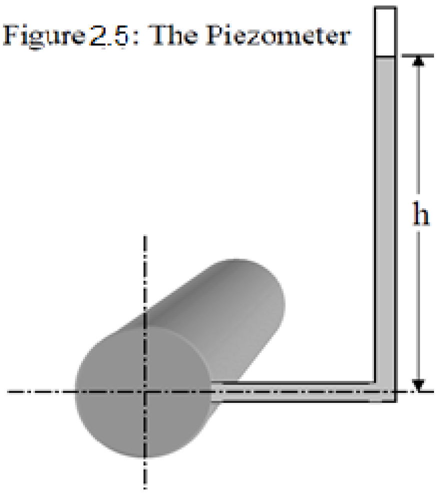
1- Piezometer tube

The piezometer consists a tube open at one end to atmosphere, the other end is capable of being inserted into vessel or pipe of which pressure is to be measured. The height to which liquid rises up in the vertical tube gives the pressure head directly.

i.e. $P = h\,\rho\,g$

Piezometer is used for measuring moderate pressures.

Figure 2.5: The Piezometer



2- Manometers

The manometer is an improved (modified) form of a piezometer. It can be used for measurement of comparatively *high pressures and of both gauge and vacuum pressures.*

Following are the various types of manometers: -

a- Simple manometer

b- The well type manometer

c- Inclined manometer

d- The inverted manometer

e- The two-liquid manometer

A)Simple manometer

It consists of a transparent U-tube containing the fluid A of density (ρ_A) whose pressure is to be measured and an immiscible fluid (B) of higher density (ρ_B). The limbs are connected to the two points between which the pressure difference (P_2 - P_1) is required. If P_2 is greater than P_1, the interface between the two liquids in limb ❶ will be depressed a distance (hm) (say) below that in limb ❷.

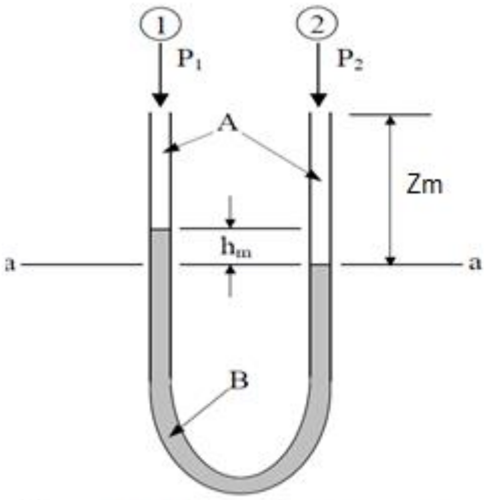


Figure 2.6: The simple manometer

The pressure at the level a-a must be the same in each of the limbs and, therefore:

$$P_2 + Z_m\rho_A\, g = P_1 + (Z_m - h_m)\,\rho_A\, g + h_m\rho_B\, g$$

$$\Rightarrow \Delta p = P_2 - P_1 = h_m\,(\rho_B - \rho_A)\, g$$

If fluid A is a gas, the density ρ_A will normally be small compared with the density of the manometer fluid pm so that:

$$\Delta p = P_2 - P_1 = h_m\rho_B\, g$$

B) The well-type manometer

In order to avoid the inconvenience of having to read two limbs, and in order to measure low pressures, where accuracy is of much importance, the well-type manometer shown in Figure (2.7) can be used. If A_w and A_c are the cross-sectional areas of the well and the column and h_m is the increase in the level of the column and h_w the decrease in the level of the well, then:

$$P_2 = P_1 + (h_m + h_w)\rho g \quad or: \quad \Delta p = P_2 - P_1 = (h_m + hw)\rho g$$

The quantity of liquid expelled from the well is equal to the quantity pushed into the column so that:

$$A_w h_w = A_c h_m \Rightarrow h_w = (A_c/A_w)\, h_m$$

$$\Rightarrow \Delta p = P_2 - P_1 = \rho g h_m (1 + A_c/A_w)$$

If the well is large in comparison to the column then:

$$i.e.\ (A_c/A_w) \rightarrow \approx 0 \Rightarrow \Delta p = P_2 - P_1 = \rho g h_m$$

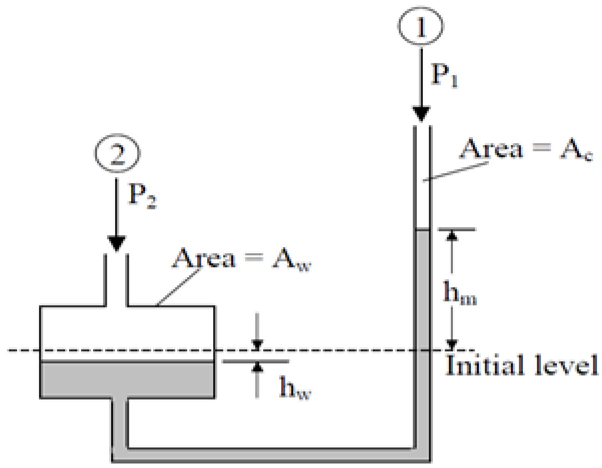


Figure 2.7: The well-type manometer

C) The inclined manometer

Shown in Figure (2.8) enables the sensitivity of the manometers described previously to be increased by measuring the length of the column of liquid. If θ is the angle of inclination of the manometer (typically about 10-20°) and L is the movement of the column of liquid along the limb, then:

$$h_m = L \sin\theta$$

If θ = 10°, the manometer reading L is increased by about 5.7 times compared with the reading h_m which would have been obtained from a simple manometer.

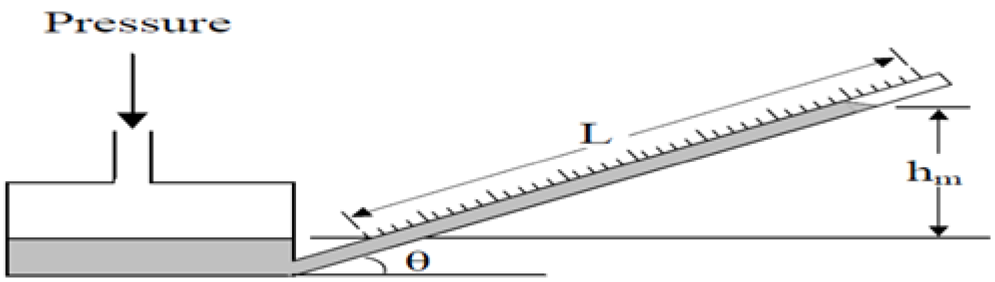


Figure 2.8 : The inclined manometer

3- Mechanical Gauges

Whenever a very high fluid pressure is to be measured, and a very great sensitivity a mechanical gauge is best suited for these purposes. They are also designed to read

vacuum pressure. A mechanical gauge is also used for measurement of pressure in boilers or other pipes, where tube manometer cannot be conveniently used.
There are many types of gauge available in the market. But the principle on which all these gauge work is almost the same. The followings are some of the important types of mechanical gauges: -

1- The Bourdon gauge
2- Diaphragm pressure gauge
3- Dead weight pressure gauge

The Bourdon gauge

The pressure to be measured is applied to a curved tube, oval in cross-section, and the deflection of the end of the tube is communicated through a system of levers to a recording needle. This gauge is widely used for steam and compressed gases, and frequently forms the indicating element on flow controllers. The simple form of the gauge is illustrated in Figures (2.9) shows a Bourdon type gauge with the sensing element in the form of a helix; this instrument has a very much greater sensitivity and is suitable for very high pressures.

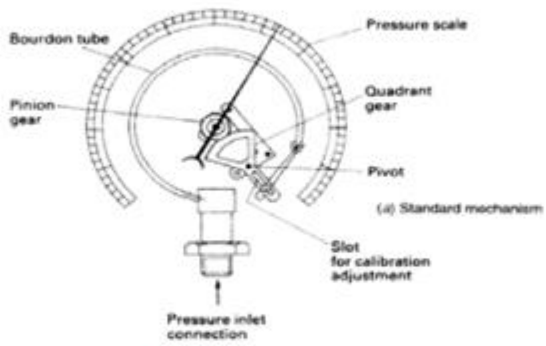


Fig. 2.9

FORCES ON SUBMERGED PLANE SURFACES.

The calculation of the magnitude, direction, and location of the total forces on surfaces submerged in a liquid is essential in the design of dams, bulkheads, gates, tanks, etc. For a submerged, **plane, horizontal** area the calculation of these force properties is simple, because the pressure does not vary over the area;

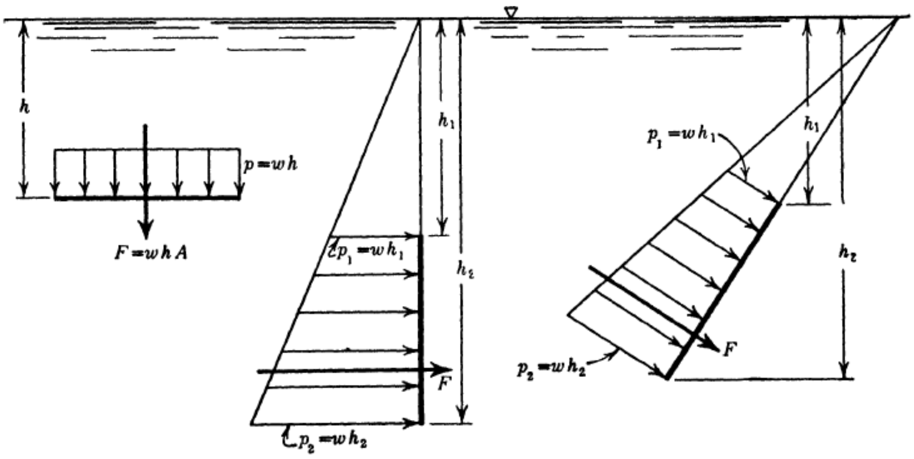


FIG. 2.10

For **non-horizontal planes** the problem is complicated by pressure variation. Pressure in liquids, however, has been shown to vary linearly with depth (eq.1), resulting in the typical pressure diagrams and resultant forces of Fig. 2.10.

Now consider the general case of a plane submerged area AB, such as that of Fig. 2.11, located in any inclined plane X-X.

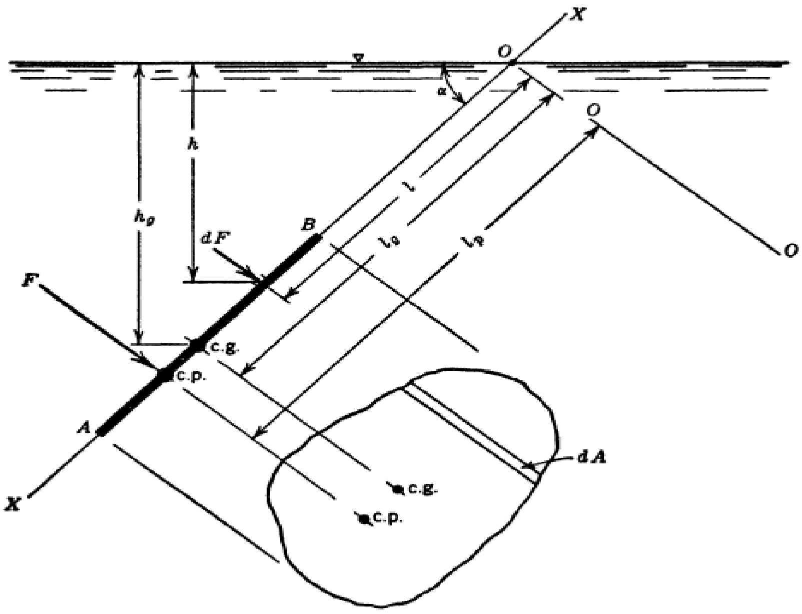


Fig 2.11

Let the center of gravity of this area be located as shown, at a depth h_g and at a distance l_g from the line of intersection, 0-0, of plane X-X and the liquid surface.

Calculating the force, dF, on the small area, dA,

$$dF = pdA = whdA$$

but $h = l\,sin\alpha$, substituting this value for h

$$dF = wl\,sin\,\alpha\,dA$$

And the total force on the area AB will result from the integration of this expression, giving

$$F = w\,sin\,\alpha \int^{A} ldA \qquad (2)$$

but $\int^{A} ldA$ is recognized as the statical moment of the area AB, about the line 0-0 which is also given by the product of the area, A, and the perpendicular distance, l_g, from 0-0 to the center of gravity of the area. Thus

$$\int^{A} ldA = l_g A$$

and substituting this in eq. 2

$$F = wAl_g\,sin\,\alpha$$

but $h_g = l_g\,sin\,\alpha$, giving

$$F = wh_gA \qquad (3)$$

indicating that the magnitude of the resultant force on a submerged plane area may be calculated by *multiplying the area, A, by the pressure at its center of gravity,* wh_g.

The magnitude of the resultant force having been calculated, its direction and location must be considered. Its direction, is normal to the plane, and its point of application may be found if the moment of the force can be calculated and divided by the magnitude of the force.

Referring again to Fig. 2.11, the moment, dM, of the force, dF, about the line 0-0 is given by

$$dM = ldF$$

in which

$$dF = wl \sin\alpha \, dA$$

Therefore, by substitution,

$$dM = wl^2 \sin\alpha \, dA$$

and integrating to obtain the total moment, M,

$$M = w \sin\alpha \int^A l^2 dA$$

in which $\int^A l^2 dA$ is the moment of inertia I of the area A , about the line 0-0, thus

$$I_{o-o} = \int^A l^2 dA$$

So:

$$M = w \sin\alpha \, I_{o-o}$$

Designating the point of intersection of the resultant force and the plane as the "center of pressure" and its distance from 0-0 as *lp*, *lp* will be given by

$$l_p = \frac{M}{F}$$

in which

$$M = w \sin\alpha \, I_{o-o}$$

and

$$F = wAl_g \sin\alpha$$

Substituting these values above gives

$$l_p = \frac{M}{F} = \frac{w \sin\alpha \, I_{o-o}}{wAl_g \sin\alpha} = \frac{I_{o-o}}{l_g A}$$

thus locating the resultant force in respect to the line 0-0, and completing the solution of the general problem.

The above equation may be made more usable by .placing it in terms of the moment of inertia, I_g , about an axis parallel to 0-0 through the center of gravity of the area. Using the equation for transferring moment of inertia of an area from one axis to another,

$$I_{o-o} = I_g + {l_g}^2 A$$

and substituting in the equation for l_p

$$l_p = \frac{I_g + {l_g}^2 A}{l_g A} = \frac{I_g}{l_g A} + l_g$$

which may be written as

$$l_p = \frac{I_g}{l_g A} + l_g$$

$$h_p = l_p \sin\alpha$$

ILLUSTRATIVE PROBLEM

A circular gate 2.4 m in diameter lies in a plane sloping 60 with the horizontal. If water stands above the center of the gate to a depth of 3 m, calculate the magnitude, direction, and location of the total force exerted by water on gate.

Direction: normal to gate

Magnitude:

F =

Location

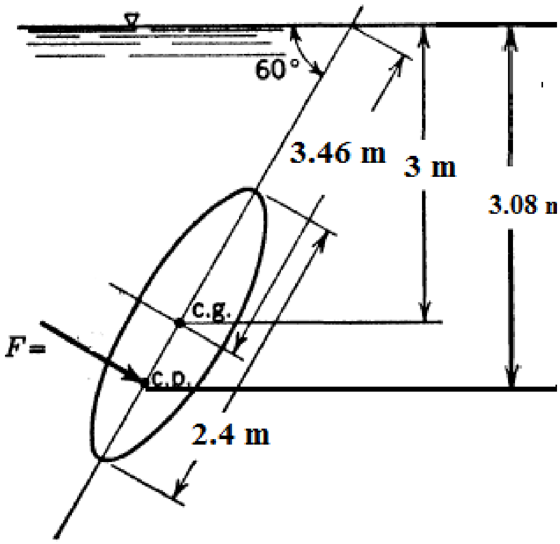


Fig. of Illustrative problem

l_p

h_p =

Therefore force passes through a point (c.p.) located 3.08 m below the free surface

Table 2.1

The centre of gravity (G) and moment of inertia (I) of some important geometrical figures:

S No.	Name of figure	C.G. from the base	Area	I about an axis passing through C.G. and parallel to the base	I about base
1	Triangle Fig. a	$x = \frac{h}{3}$	$\frac{bh}{2}$	$\frac{bh^3}{36}$	$\frac{bh^3}{12}$
2	Rectangle Fig. b	$x = \frac{d}{2}$	bd	$\frac{bd^3}{12}$	$\frac{bd^3}{3}$
3	Circle Fig. c	$x = \frac{d}{2}$	$\frac{\pi d^2}{4}$	$\frac{\pi d^4}{64}$	—
4	Trapezium Fig. d	$x = \left[\frac{2a+b}{a+b}\right]\frac{h}{3}$	$\left(\frac{a+b}{2}\right)h$	$\left(\frac{a^2+4ab+b^2}{36(a+b)}\right) \times h^2$	—

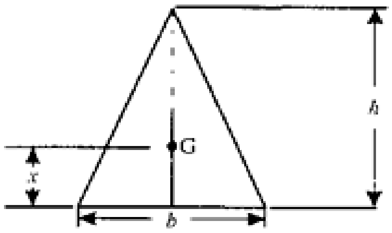


Fig. a

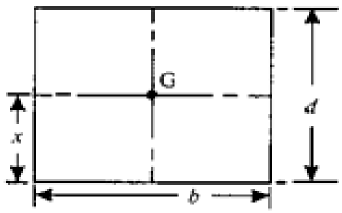


Fig. b

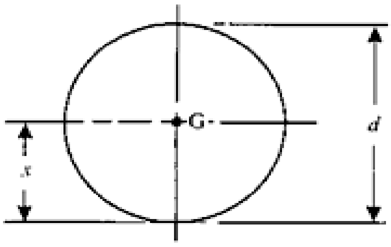


Fig. c

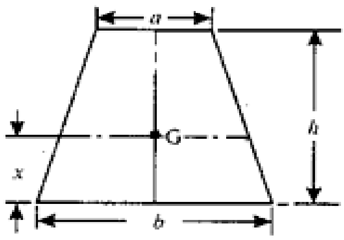


Fig. d

For **vertical plane surfaces**

See fig. 2.12

h

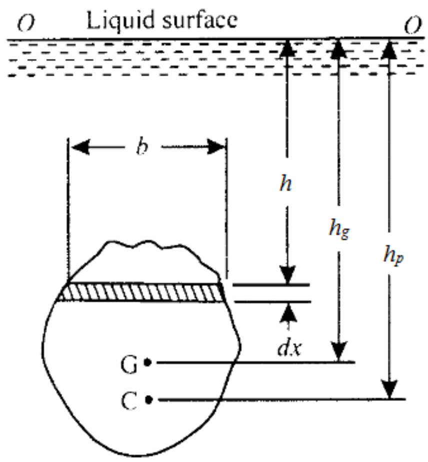


Fig. 2.12 Vertically immersed surface.

Curved surfaces

The total forces on submerged curved areas cannot be calculated by the foregoing methods. These forces may be readily obtained, however, by calculating the horizontal and vertical components of the forces as indicated below.

See the curved area, AB, of Fig. 2.13 the vertical component of force on the area AB is simply the weight of liquid, ABCD, thus

$$F_V = W_{ABCD}$$

and the line of action of this force will pass through the center of gravity of ABCD.

The horizontal component of force may be calculated by the methods of vertical plane surfaces.

Its location: $h_p = \frac{I_g}{h_g\,A} + h_g$

And the resultant force, F, may be obtained by composition of the horizontal and vertical components

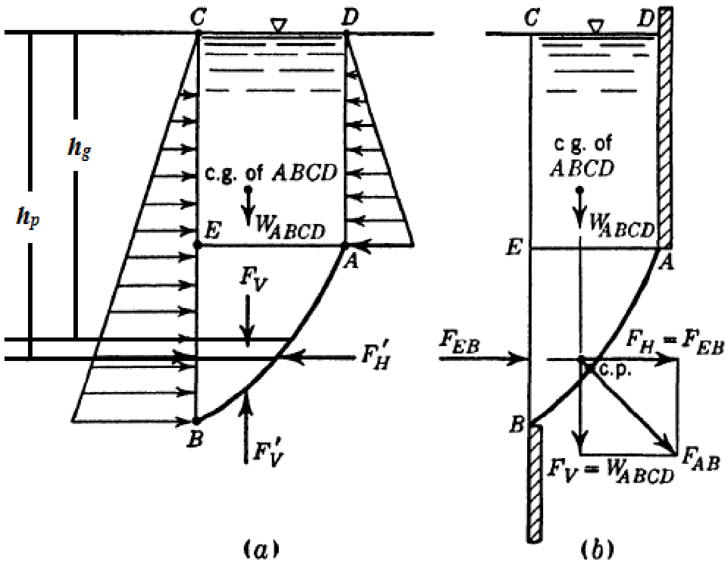


Fig 2.13

Then the angle of direction of the resultant force

$$\theta = \tan^{-1}\frac{F_V}{F_H}$$

Buoyancy

Whenever a body is immersed wholly or partially in a fluid; 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 is due to an upward force opposite to action of gravity is known as buoyancy. The force tending to lift the body under such conditions is known as buoyant force or force of buoyancy or up thrust. The magnitude of the buoyant force can be determined by Archimedes principle which states as follows:

"*The buoyant force acting on a body immersed in a fluid is equal to the weight of the fluid displaced by the body, and it acts upward through the centroid of the displaced volume*."

$$weight\ of\ fluid\ displaced\ by\ the\ body\ =\ buoyant\ force$$

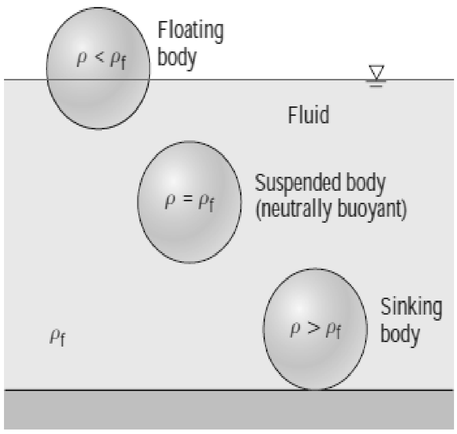
For floating bodies, the weight of the entire body must be equal to the buoyant force, which is the weight of the fluid whose volume is equal to the volume of the submerged portion of the floating body. That is

$$Total\ weight\ of\ the\ body\ immersed\ in\ a\ fluid\ (W) =\ weight\ of\ fluid\ displaced (F_B)$$

$$(W) = (F_B)$$

$$\rho_{body} \times g \times V_{Total\ body} = \rho_f \times g \times V_{immersed\ part\ of\ body}$$

$$\frac{\rho_{body}}{\rho_f} = \frac{V_{immersed\ part\ of\ body}}{V_{Total\ body}}$$



Therefore, the submerged volume fraction of a floating body is equal to the ratio of the average density of the body to the density of the fluid. Note that when the density ratio is equal to or greater than one, the floating body becomes completely submerged.

It follows from these discussions that: a body immersed in a fluid

(1) Remains at rest at any point in the fluid when its density is equal to the density of the fluid

(2) Sinks to the bottom when its density is greater than the density of the fluid

(3) Rises to the surface of the fluid and floats when the density of the body is less than the density of the fluid (see the Fig.).

The buoyant force is proportional to the density of the fluid, and thus we might think that the buoyant force exerted by gases such as air is negligible.

For example, the volume of a person is about 0.1 m^3, and taking the density of air to be 1.2 kg/m^3, the buoyant force exerted by air on the person is

$F_B = \rho_f \times g \times V = (1.2\ kg/m^3)(9.81\ m/s^2)(0.1\, m^3) =\ 1.2\, N$

The weight of an 80-kg person is: $80\ \times\ 9.81\ =\ 788\, N$.

Therefore, ignoring the buoyancy in this case results in a simple error in weight which is negligible.

But the buoyancy effects in gases dominate some important natural phenomena such as the rise of warm air in a cooler environment and the rise of hot-air or helium balloons. A helium balloon, for example, rises as a result of the buoyancy effect until it reaches an altitude where the density of air (which decreases with altitude) equals the density of helium in the balloon ignoring the weight of the balloon's skin.

Centre of Buoyancy

The point of application of the force of buoyancy on the body is known as the centre of buoyancy. It is always the centre of gravity of the volume of fluid displaced.

Problem *Find the volume of the water displaced and position of centre of buoyancy for a wooden block of width 2.5 m and of depth 1.5 m, when it floats horizontally in water. The density of wooden block is 650 kg/m^3 and its length 6.0 m.*

Solution. Given :

Width $= 2.5$ m

Depth $= 1.5$ m

Length $= 6.0$ m

Volume of the block $= 2.5 \times 1.5 \times 6.0 = 22.50 \text{ m}^3$

Density of wood, $\rho = 650 \text{ kg/m}^3$

$\therefore$ Weight of block $= \rho \times g \times \text{Volume}$

$= 650 \times 9.81 \times 22.50 \text{ N} = 143471 \text{ N}$

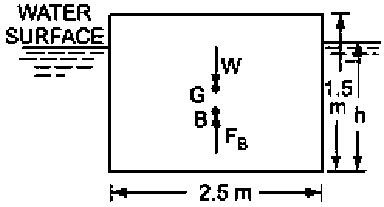


Fig.

For equilibrium the weight of water displaced = Weight of wooden block

$= 143471$ N

$\therefore$ Volume of water displaced

$$= \frac{\text{Weight of water displaced}}{\text{Weight density of water}} = \frac{143471}{1000 \times 9.81} = \mathbf{14.625 \text{ m}^3}. \text{ **Ans.**}$$

($\because$ Weight density of water $= 1000 \times 9.81 \text{ N/m}^3$)

Position of Centre of Buoyancy

Volume of wooden block in water = Volume of water displaced

or $2.5 \times h \times 6.0 = 14.625 \text{ m}^3$, where h is depth of wooden block in water

$$\therefore \quad h = \frac{14.625}{2.5 \times 6.0} = 0.975 \text{ m}$$

$$\therefore \quad \text{Centre of Buoyancy} = \frac{0.975}{2} = \mathbf{0.4875 \text{ m from base.}} \text{ **Ans.**}$$

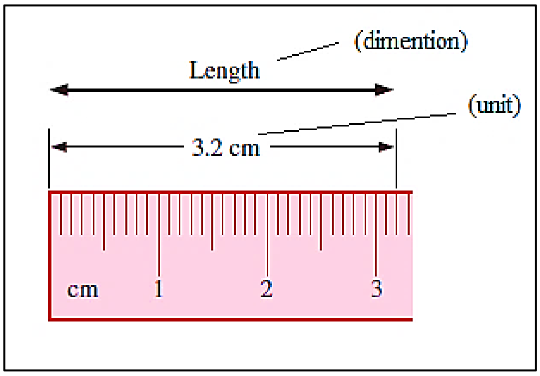
Example 1: A body of dimensions 1.5 m x 1m x2 m. weighs 1962 N in water. Find its weight in air. What will be its specific gravity?

Example 2: A wooden block of specific gravity 0.7 and having a size of $2\,m \times 0.5m \times 0.25\,m$ flouting in water. Determine the volume of concrete of specific weight 25 kN/m^3, that may be placed which will immerse the (i) block completely in water and (ii) block and concrete completely in water.

CHAPTER THREE
DIMENSIONAL ANALYSIS

Dimensions and units

A **dimension** is a measure of a physical quantity (without numerical values), while a **unit** is a way to assign a number to that dimension. For example, length is a dimension that is measured in units such as meters (m), feet (ft), centimeters (cm), kilometers (km), etc. (Fig).



There are seven **primary dimensions** (also called fundamental or basic dimensions): mass, length, time, temperature, electric current, amount of light, and amount of matter.

All nonprimary dimensions can be formed by some combination of the seven primary dimensions. For example, force has the same dimensions as mass times acceleration (by Newton's second law). Thus, in terms of primary dimensions,

$$\textit{Dimensions of force:} \quad \{\text{Force}\} = \left\{\text{Mass}\,\frac{\text{Length}}{\text{Time}^2}\right\} = \{\text{mL/t}^2\}$$

TABLE 1

Primary dimensions and their associated primary SI and English units

Dimension	Symbol*	SI Unit	English Unit
Mass	m	kg (kilogram)	lbm (pound-mass)
Length	L	m (meter)	ft (foot)
Time†	t	s (second)	s (second)
Temperature	T	K (kelvin)	R (rankine)
Electric current	I	A (ampere)	A (ampere)
Amount of light	C	cd (candela)	cd (candela)
Amount of matter	N	mol (mole)	mol (mole)

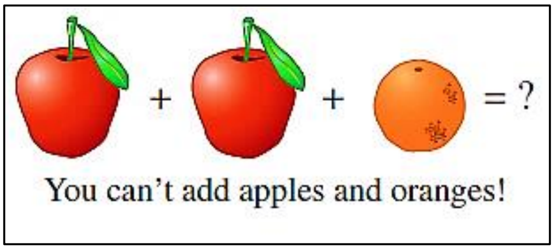
EXAMPLE 1 **Primary Dimensions of Surface Tension**

$$\textit{Dimensions of surface tension:} \quad \{\sigma_s\} = \left\{\frac{\text{Force}}{\text{Length}}\right\} = \left\{\frac{\text{m}\cdot\text{L/t}^2}{\text{L}}\right\} = \{\text{m/t}^2\}$$

Dimensional Homogeneity

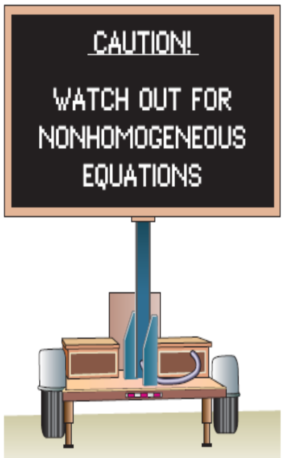
We've all heard the old saying; (you can't add apples and oranges) (Fig). This is actually a simplified expression of a far more global and fundamental mathematical law for equations, the law of dimensional homogeneity, stated as

Every additive term in an equation must have the same dimensions.



EXAMPLE **Dimensional Homogeneity of the Bernoulli Equation**

Probably the most well-known (and most misused) equation in fluid mechanics is the Bernoulli equation (Fig beside), discussed in Chap.4 ،. One standard form of the Bernoulli equation for incompressible irrotational fluid flow is



Bernoulli equation:

$$P + \frac{1}{2}\rho V^2 + \rho g z = C \qquad (1)$$

(*a*) Verify that each additive term in the Bernoulli equation has the same dimensions. (*b*) What are the dimensions of the constant *C*?

SOLUTION We are to verify that the primary dimensions of each additive term in Eq. 1 are the same, and we are to determine the dimensions of constant *C*.

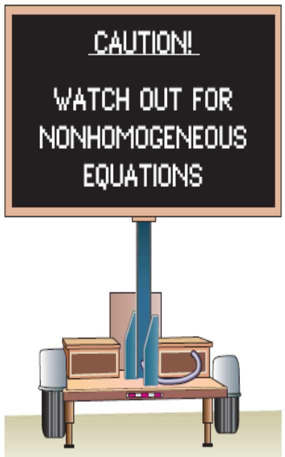
Analysis (*a*) Each term is written in terms of primary dimensions,

$$\{P\} = \{\text{Pressure}\} = \left\{\frac{\text{Force}}{\text{Area}}\right\} = \left\{\text{Mass}\,\frac{\text{Length}}{\text{Time}^2}\,\frac{1}{\text{Length}^2}\right\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

$$\left\{\frac{1}{2}\rho V^2\right\} = \left\{\frac{\text{Mass}}{\text{Volume}}\left(\frac{\text{Length}}{\text{Time}}\right)^2\right\} = \left\{\frac{\text{Mass}\times\text{Length}^2}{\text{Length}^3\times\text{Time}^2}\right\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

$$\{\rho g z\} = \left\{\frac{\text{Mass}}{\text{Volume}}\,\frac{\text{Length}}{\text{Time}^2}\,\text{Length}\right\} = \left\{\frac{\text{Mass}\times\text{Length}^2}{\text{Length}^3\times\text{Time}^2}\right\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

Indeed, **all three additive terms have the same dimensions.**



(*b*) From the law of dimensional homogeneity, the constant must have the same dimensions as the other additive terms in the equation. Thus,

Primary dimensions of the Bernoulli constant:

$$\{C\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$$

Discussion If the dimensions of any of the terms were different from the others, it would indicate that an error was made somewhere in the analysis.

Dimensional analysis and similarity

There are three primary purposes of dimensional analysis:

1. To generate nondimensional parameters that help in the design of experiments (physical and/or numerical) and in the reporting of experimental results
2. To obtain scaling laws so that prototype performance can be predicted from model performance
3. To (sometimes) predict trends in the relationship between parameters

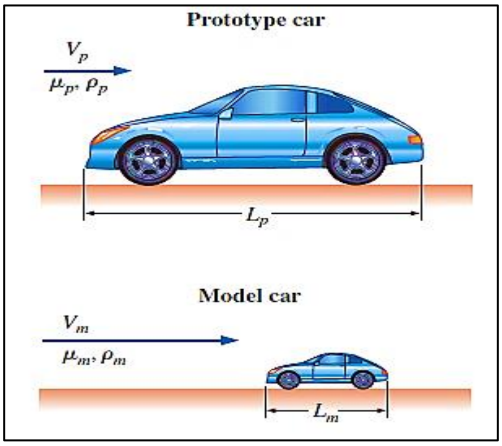
In many cases in real-life engineering, the equations are either not known or too difficult to solve; oftentimes experimentation is the only method of obtaining reliable information.

In most experiments, to save time and money, tests are performed on a geometrically scaled **model,** rather than on the full-scale **prototype.** In such cases, care must be taken to properly scale the results. We introduce here a powerful technique called **dimensional analysis.**

Before discussing the *technique* of dimensional analysis, we first explain the principle of **similarity.** There are three necessary conditions for complete similarity between a model and a prototype. The first condition is

- **Geometric similarity**—the model must be the same shape as the prototype, but may be scaled by some constant scale factor. The second condition is
- **Kinematic similarity**, which means that the velocity at any point in the model flow must be proportional (by a constant scale factor) to the velocity at the corresponding point in the prototype flow, for kinematic similarity the velocity at corresponding points must scale in magnitude and must point in the same relative direction. The third similarity condition is that
- **Dynamic similarity:** Dynamic similarity is achieved when all forces in the model flow scale by a constant factor to corresponding forces in the prototype flow.

(All three similarity conditions must exist for complete similarity to be ensured)



EXAMPLE **Similarity between Model and Prototype Cars**

The aerodynamic drag of a new sports car is to be predicted at a speed of 50.0 mi/h at an air temperature of 25°C. Automotive engineers build a one-fifth scale model of the car to test in a wind tunnel. It is winter and the wind tunnel is located in an unheated building; the temperature of the wind tunnel air is only about 5°C. Determine how fast the engineers should run the wind tunnel in order to achieve similarity between the model and the prototype.

SOLUTION We are to utilize the concept of similarity to determine the speed of the wind tunnel.

Properties For air at atmospheric pressure and at $T = 25°C$, $\rho = 1.184\ kg/m^3$ and $\mu = 1.849 \times 10^{-5}$ kg/m·s. Similarly, at $T = 5°C$, $\rho = 1.269\ kg/m^3$ and $\mu = 1.754 \times 10^{-5}$ kg/m·s.

$$\Pi_{2,m} = \mathrm{Re}_m = \frac{\rho_m V_m L_m}{\mu_m} = \Pi_{2,p} = \mathrm{Re}_p = \frac{\rho_p V_p L_p}{\mu_p}$$

which we solve for the unknown wind tunnel speed for the model tests, V_m,

$$V_m = V_p\left(\frac{\mu_m}{\mu_p}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{L_p}{L_m}\right)$$

$$= (50.0\ \mathrm{mi/h})\left(\frac{1.754 \times 10^{-5}\ \mathrm{kg/m{\cdot}s}}{1.849 \times 10^{-5}\ \mathrm{kg/m{\cdot}s}}\right)\left(\frac{1.184\ \mathrm{kg/m^3}}{1.269\ \mathrm{kg/m^3}}\right)(5) = \mathbf{221\ mi/h}$$

Discussion This speed is quite high (about 100 m/s), and the wind tunnel may not be able to run at that speed. Furthermore, the incompressible approximation may come into question at this high speed (we discuss this in more detail in next example

Suppose, for example, that the engineers in the above example use a water tunnel instead of a wind tunnel to test their one-fifth scale model. Using the properties of water at room temperature (20°C is assumed), the water tunnel speed required to achieve similarity is easily calculated as

$$V_m = V_p\left(\frac{\mu_m}{\mu_p}\right)\left(\frac{\rho_p}{\rho_m}\right)\left(\frac{L_p}{L_m}\right)$$

$$= (50.0\ \mathrm{mi/h})\left(\frac{1.002 \times 10^{-3}\ \mathrm{kg/m{\cdot}s)}}{1.849 \times 10^{-5}\ \mathrm{kg/m{\cdot}s}}\right)\left(\frac{1.184\ \mathrm{kg/m^3}}{998.0\ \mathrm{kg/m^3}}\right)(5) = 16.1\ \mathrm{mi/h}$$

As can be seen, one advantage of a water tunnel is that the required water tunnel speed is much lower than that required for a wind tunnel using the same size model.

The method of repeating variables and the Buckingham pi theorem

We have seen the usefulness and power of dimensional analysis. Now we are ready to learn how to *generate* the nondimensional parameters, i.e., the Π's. There are several methods that have been developed for this purpose, but the most popular (and simplest) method is the method of repeating variables, popularized by Edgar Buckingham (1867–1940).

We can think of this method as a step-by-step procedure for obtaining nondimensional parameters.

These steps are explained in figure (a) and in detail as we work through an example, as the best way to learn is by example and practice.

The Method of Repeating Variables

Step 1: List the parameters in the problem and count their total number n.

Step 2: List the primary dimensions of each of the n parameters.

Step 3: Set the *reduction j* as the number of primary dimensions. Calculate k, the expected number of Π's,
$$k = n - j$$

Step 4: Choose *j repeating parameters*.

Step 5: Construct the k Π's, and manipulate as necessary.

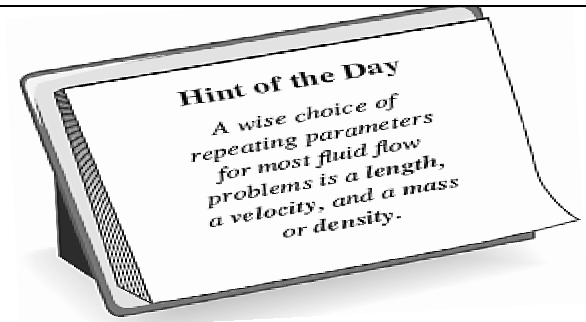
Step 6: Write the final functional relationship and check your algebra.

Figure (**a**)

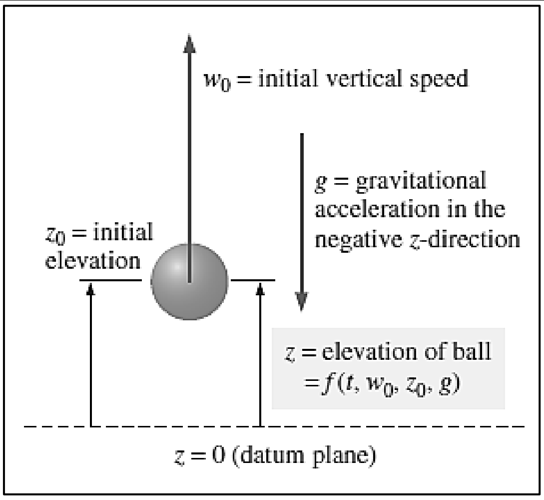
Figure (b)

Guidelines for choosing *repeating parameters* in step 4 of the method of repeating variables*

Guideline	Comments and Application to Present Problem
1. Never pick the *dependent* variable. Otherwise, it may appear in all the Π's, which is undesirable.	In the present problem we cannot choose z, but we must choose from among the remaining four parameters. Therefore, we must choose two of the following parameters: t, w_0, z_0, and g.
2. The chosen repeating parameters must not *by themselves* be able to form a dimensionless group. Otherwise, it would be impossible to generate the rest of the Π's.	In the present problem, any two of the independent parameters would be valid according to this guideline. For illustrative purposes, however, suppose we have to pick three instead of two repeating parameters. We could not, for example, choose t, w_0, and z_0, because these can form a Π all by themselves (tw_0/z_0).
3. The chosen repeating parameters must represent *all* the primary dimensions in the problem.	Suppose for example that there were *three* primary dimensions (m, L, and t) and *two* repeating parameters were to be chosen. You could not choose, say, a length and a time, since primary dimension mass would not be represented in the dimensions of the repeating parameters. An appropriate choice would be a density and a time, which together represent all three primary dimensions in the problem.



For a simple first example, consider a ball falling in a vacuum. Let us pretend that we do not know much physics concerning falling objects. Elevation z of the ball must be a function of time t, initial vertical speed w_0, initial elevation z_0, and gravitational constant g (Fig). The beauty of dimensional analysis is that the only other thing we need to know is the primary dimensions of each of these quantities. As we go through each step of the method of repeating



variables, we explain some of the subtleties of the technique in more detail using the falling ball as an example.

Step 1
There are five parameters (variables and constants) in this problem; n = 5. They are listed in functional form, with the dependent variable listed as a function of the independent parameters:

$$List\ of\ relevant\ parameters: z = f(t, w_0, z_0, g) \qquad n = 5$$

Step 2
The primary dimensions of each parameter are listed here. We recommend writing each dimension with exponents since this helps later.

$$z = \{L^1\} \qquad t = \{t^1\} \qquad w_0 = \{L^1 t^{-1}\} \qquad z_0 = \{L^1\} \qquad g = \{L^1 t^{-2}\}$$

Step 3
As a first guess, j is set equal to 2, the number of primary dimensions represented in the problem (L and t).
Reduction: j = 2
If this value of j is correct, the number of Π's predicted by the Buckingham Pi theorem is
Number of expected Π's: $k = (n - j) = (5 - 2) = 3$

Step 4
We need to choose two repeating parameters since j = 2. (Several guidelines about choosing repeating parameters are listed in figure (b).
The wisest choice of two repeating parameters is w_0 and z_0.
Repeating parameters: w_0 and z_0

Step 5
Now we combine these repeating parameters into products with each of the remaining parameters, one at a time, to create the Π's. The first Π is always the *dependent* P and is formed with the dependent variable z.

Dependent Π: $\Pi_1 = z w_0^{a_1} z_0^{b_1}$ 1

Where a_1 and b_1 are constant exponents that need to be determined. We apply the primary dimensions of step 2 into Eq. 1 and force the Π to be dimensionless by setting the exponent of each primary dimension to zero:

$$\Pi_1 = \{L^0 t^0\} = \left\{z w_0^{a_1} z_0^{b_1}\right\} = L^1 (L^1 t^{-1})^{a_1} (L^1)^{b_1}$$

We equate the exponents of each primary dimension independently to solve for exponents a_1 and b_1

$Time$: $\{t^0\} = \{t^{-a_1}\}$ $-a_1 = 0$ $a_1 = 0$

$Length$: $\{L^0\} = \{L^1 L^{a_1} L^{b_1}\} \rightarrow 0 = 1 + a_1 + b_1 \rightarrow b_1 = -1 - a_1 \rightarrow b_1 = -1$

Equation 1 becomes: $\Pi_1 = \frac{z}{z_0}$

In similar fashion we create the first independent Π (Π_2) by combining the repeating parameters with independent variable t.

$$\Pi_2 = t w_0^{a_2} z_0^{b_2}$$

Dimensions of Π_2: $\{\Pi_2\} = \{L^0 t^0\} = \{t w_0^{a_2} z_0^{b_2}\} = \{t(L^1 t^{-1})^{a_2} L^{b_2}\}$

Equating exponents,

Time: $\{t^0\} = \{t^1 t^{-a_2}\}$ $\quad 0 = 1 - a_2 \quad a_2 = 1$

Length: $\{L^0\} = \{L^{a_2} L^{b_2}\}$ $\quad 0 = a_2 + b_2 \quad b_2 = -a_2 \quad b_2 = -1$

$$\Pi_2 = \frac{t w_0}{z_0}$$

Finally we create the second independent Π (Π_3) by combining the repeating parameters with g and *forcing* the Π to be dimensionless

$$\Pi_3 = g w_0^{a_3} z_0^{b_3}$$

Dimensions of Π_3: $\{\Pi_3\} = \{L^0 t^0\} = \{g w_0^{a_3} z_0^{b_3}\} = \{L^1 t^{-2} (L^1 t^{-1})^{a_3} L^{b_3}\}$

Equating exponents,

Time: $\{t^0\} = \{t^{-2} t^{-a_3}\}$ $\quad 0 = -2 - a_3 \quad a_3 = -2$

Length: $\{L^0\} = \{L^1 L^{a_3} L^{b_3}\}$ $\quad 0 = 1 + a_3 + b_3 \quad b_3 = -1 - a_3 \quad b_3 = 1$

$$\Pi_3 = \frac{g z_0}{w_0^2}$$

Step 6

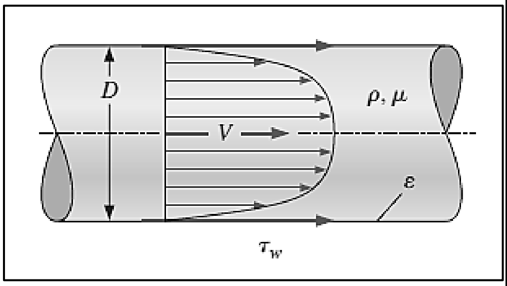
We are finally ready to write the functional relationship between the nondimensional parameters.

$$\Pi_1 = f\,(\Pi_2, \Pi_3) \rightarrow \frac{z}{z_0} = f\left(\frac{t w_0}{z_0}, \frac{g z_0}{w_0^2}\right)$$

(The method of repeating variables cannot predict the exact mathematical form of the equation)

Example: friction in a pipe

Consider flow of an incompressible fluid of density ρ and viscosity μ through a long, horizontal section of round pipe of diameter D. The velocity profile is sketched in the Fig.; V is the average speed across the pipe cross section, which by conservation of mass remains constant down the pipe. For a very long pipe, the flow eventually becomes hydrodynamically fully developed, which means that the velocity profile also remains uniform down the pipe.



Because of frictional forces between the fluid and the pipe wall, there exists a shear stress τ_w on the inside pipe wall as sketched. The shear stress is also constant down the pipe in the fully developed region. We assume some constant average roughness height ε along the inside wall of the pipe. Develop a nondimensional relationship between shear stress τ_w and the other parameters in the problem.

Step 1 There are six variables and constants in this problem; $n = 6$. They are listed in functional form, with the dependent variable listed as a function of the independent variables and constants:

List of relevant parameters: $\tau_w = f(V, \varepsilon, \rho, \mu, D) \qquad n = 6$

Step 2 The primary dimensions of each parameter are listed. Note that shear stress is a force per unit area, and thus has the same dimensions as pressure.

τ_w	V	ε	ρ	μ	D
$\{m^1L^{-1}t^{-2}\}$	$\{L^1t^{-1}\}$	$\{L^1\}$	$\{m^1L^{-3}\}$	$\{m^1L^{-1}t^{-1}\}$	$\{L^1\}$

Step 3 As a first guess, j is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

Reduction: $j = 3$

If this value of j is correct, the expected number of Π's is $k = n - j = 6 - 3 = 3$.

Step 4 We choose three repeating parameters since $j = 3$. Following the guidelines of Fig b , we cannot pick the dependent variable τ_w. We cannot choose both ε and D since their dimensions are identical, and it would not be desirable to have μ or ε appear in all the Π's. The best choice of repeating parameters is thus V, D, and ρ.

Repeating parameters: V, D, and ρ

Step 5 The dependent Π is generated:

$$\Pi_1 = \tau_w V^{a_1} D^{b_1} \rho^{c_1} \quad \rightarrow \quad \{\Pi_1\} = \{(m^1L^{-1}t^{-2})(L^1t^{-1})^{a_1}(L^1)^{b_1}(m^1L^{-3})^{c_1}\}$$

from which $a_1 = -2$, $b_1 = 0$, and $c_1 = -1$, and thus the dependent Π is

$$\Pi_1 = \frac{\tau_w}{\rho V^2}$$

Π_1: $$\Pi_1 = \frac{\tau_w}{\rho V^2} = \text{friction factor} = f$$

Similarly, the two independent Π's are generated, the details of which are left for you to do on your own:

$$\Pi_2 = \mu V^{a_2} D^{b_2} \rho^{c_2} \quad \rightarrow \quad \Pi_2 = \frac{\rho V D}{\mu} = \text{Reynolds number} = \text{Re}$$

$$\Pi_3 = \varepsilon V^{a_3} D^{b_3} \rho^{c_3} \quad \rightarrow \quad \Pi_3 = \frac{\varepsilon}{D} = \text{Roughness ratio}$$

Step 6 We write the final functional relationship as

$$f = \frac{8\tau_w}{\rho V^2} = f\left(Re, \frac{\varepsilon}{D}\right)$$

CHAPTER FOUR
FLUID DYNAMIC

Methods of Describing Fluid Motion

The fluid motion is described by two methods. They are (i) Lagrangian Method, and (ii) Eulerian Method. In the Lagrangian method, a single fluid particle is followed during its motion and its velocity, acceleration, density, etc., are described. In case of Eulerian method, the velocity, acceleration, pressure, density etc., are described at a point in flow field. The Eulerian method is commonly used in fluid mechanics.

Types of Fluid Flow

Fluids may be classified as follows:

1. Steady and unsteady flows
2. Uniform and non-uniform flows
3. One, two and three dimensional flows
4. Rotational and irrotational flows
5. Laminar and turbulent flows
6. Compressible and incompressible flows.

1- Steady and Unsteady Flows

Steady flow: The type of flow in which the fluid characteristics like velocity, pressure, density, etc. at a point do not change with time is called steady flow.

Example flow of liquid at a constant flow rate and has a velocity equation in the form $u=ax^2 +bx + c$ which is independent of time t).

Unsteady flow: It is that type of flow in which the velocity, pressure or density at a point change with time.

The flow in a pipe whose valve is being opened or closed gradually (velocity equation is in the form $u = ax^2 + bxt$).

2- Uniform and Non-uniform Flows

Uniform flow: The type of flow, in which the velocity at any given time does not change with respect to space, is called uniform flow.

Example: flow through a straight pipe of constant diameter

Non-uniform flow: It is that type of flow in which the velocity at any given time changes with respect to space.

Example: Flow around a uniform diameter pipe-bend or a canal bend.

3- One, Two and Three Dimensional Flows

One dimensional flow: It is that type of flow in which the flow parameter such as velocity is function of time and one space co-ordinate only

Example: Flow in a pipe.

Two dimensional flow: The flow in which the velocity is a function of time and two rectangular space coordinates is called two dimensional flow.

Examples: (i) Flow between parallel plates of infinite extent,

(ii) Flow in the main stream of a wide river.

Three dimensional flow: It is that type of flow in which the velocity is a function of time and three perpendicular directions.

Examples: Flow in a prismatic open channel in which the width and the water depth are of the same order of magnitude.

4- Rotational and Irrotational Flows

Rotational flow: A flow is said to be rotational if the fluid particles while moving in the direction of flow rotate about their mass centers.
Example: Flow near the solid boundaries is rotational.
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 centers. Flow outside the boundary layer is generally considered irrotational.
Example: Flow above a drain hole of a stationary tank or a wash basin.

Note: If the flow is irrotational as well as steady, it is known as Potential flow.

5- Laminar and Turbulent Flows

Laminar and turbulent flows are characterized on the basis of Reynolds number
For Reynolds number (Re) < 2300 ... flow in pipes is laminar,
For Reynolds number (Re) > 4000 ... flow in pipes is turbulent
For Re between 2000 and 4000 ... flow in pipes may be laminar or turbulent.

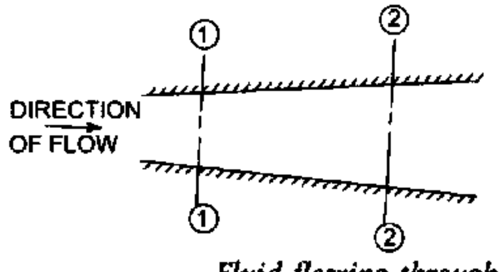
6- Compressible and Incompressible Flows

Compressible flow: It is that type of flow in which the density (ρ) of the fluid changes from point to point (or in other words density is not constant for this flow).
Example: Flow of gases through orifices, nozzles, gas turbines, etc.

Incompressible flow: It is that type of flow in which density is constant for the fluid flow; liquids are generally considered flowing incompressibly.

Continuity Equation

The equation based on the principle of conservation of mass is called continuity equation. Thus for a fluid flowing through the pipe at all the cross-section, the quantity of fluid per second is constant.
Consider two cross-sections of a pipe as shown in Fig.
Let u_1= Average velocity at cross-section l-l
ρ_1 = Density at section 1-1
A_1 = Area of pipe at section l-l



Fluid flowing through a pipe.

And u_2, ρ_2, A_2 are corresponding values at section, 2-2.
Then rate of flow at section l-l = $u_1\rho_1A_1$
Rate of flow at section 2-2 = $u_2\rho_2A_2$
According to law of conservation of mass
Rate of flow at section l-l = Rate of flow at section 2-2
Or $u_1\rho_1A_1 = u_2\rho_2A_2$
The above equation is applicable to the compressible as well as incompressible fluids and is called Continuity Equation. If the fluid is incompressible, then $\rho_1 = \rho_2$ and continuity equation reduces to

$$u_1A_1 = u_2A_2$$
$$Q_1 = Q_2$$

Where Q is the discharge (the volume of fluid flowing across the section per second)

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

Solution. Given :

At section 1, $D_1 = 10 \text{ cm} = 0.1 \text{ m}$

$$A_1 = \frac{\pi}{4}(D_1^2) = \frac{\pi}{4}(.1)^2 = 0.007854 \text{ m}^2$$

$V_1 = 5$ m/s.

At section 2, $D_2 = 15 \text{ cm} = 0.15 \text{ m}$

$$A_2 = \frac{\pi}{4}(.15)^2 = 0.01767 \text{ m}^2$$

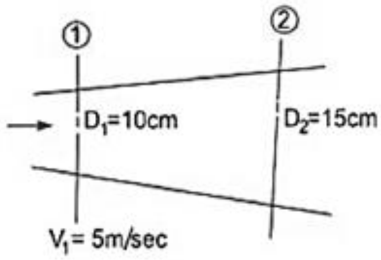


Fig.

(*i*) Discharge through pipe

or $$Q = A_1 \times V_1$$

$$= 0.007854 \times 5 = \mathbf{0.03927\ m^3/s.\ Ans.}$$

we have $A_1V_1 = A_2V_2$

(*ii*) $\therefore$ $$V_2 = \frac{A_1V_1}{A_2} = \frac{0.007854}{0.01767} \times 5.0 = \mathbf{2.22\ m/s.\ Ans.}$$

EQUATIONS OF MOTION

According to Newton's second law of motion, the net force F_x acting on a fluid element in the direction of x is equal to mass of the fluid element multiplied by the acceleration a_x in the x-direction. Thus mathematically:

$$F_x = m \times a_x$$

In the fluid flow, the following forces are present:

1. F_g, gravity force.
2. F_p, the pressure force.
3. F_v, force due to viscosity.
4. F_t, force due to turbulence,
5. F_c, force due to compressibility.

The net force:

$$F_x = (F_g)_x + (F_p)_x + (F_v)_x + (F_t)_x + (F_c)_x$$

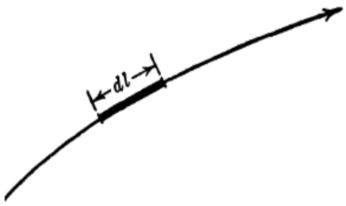
- If the force due to compressibility, F_c is negligible, the resulting net force
$$F_x = (F_g)_x + (F_p)_x + (F_v)_x + (F_t)_x$$
and equation of motions are called **Reynolds's equations** of motion.
- For flow, where (F_t) is negligible, the resulting equations of motion are known as **Navier-Stokes Equation**: $F_x = (F_g)_x + (F_p)_x + (F_v)_x$
- If the flow is assumed to be ideal, viscous force (F_v) is zero and equation of motions are known as **Euler's equation** of motion. $F_x = (F_g)_x + (F_p)_x$

Velocity and acceleration of fluid flow particle

The velocity of a particle moving along a streamline in a fluid flow (see fig.) may be expressed by

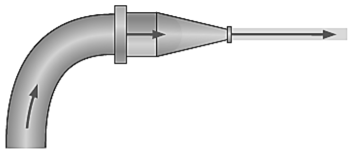
$$u = \frac{dl}{dt}$$

In which (see Fig) *dl* is the distance covered by the particle in time *dt*.



If the velocity changes, an acceleration, a, exists

One may be think that acceleration is zero in steady flow since acceleration is the rate of change of velocity with time, and in steady flow there is no change with time. Well, a garden hose nozzle tells us that this understanding is not correct. Even in steady flow and thus constant mass flow rate, water accelerates through the nozzle (see the Fig). Steady simply means no change with time at a specified location, but the value of a quantity may change from one location to another. In the_case of a nozzle, the velocity of water remains constant at a specified point, but it changes from the inlet to the exit (water accelerates along the nozzle)._Mathematically, this can be expressed as follows: We take the velocity *u* of a fluid particle to be a function of *l* and *t*.



During steady flow, a fluid may not accelerate in time at a fixed point, but it may accelerate in space.

Taking the total differential of $u = f(l, t)$

$$du = \frac{\partial u}{\partial l} dl + \frac{\partial u}{\partial t} dt$$

And dividing both sides by dt:

$$\frac{du}{dt} = \frac{\partial u}{\partial l}\frac{dl}{dt} + \frac{\partial u}{\partial t}$$

$$a = \frac{\partial u}{\partial l}\frac{dl}{dt} + \frac{\partial u}{\partial t}$$

In which the first term is called "convective" acceleration, and the second "local" acceleration. Obviously, local acceleration is a term peculiar to unsteady flow and vanishes from the above equation when it is applied to steady flow.

In steady flow $\frac{\partial u}{\partial t} = 0$, the acceleration becomes

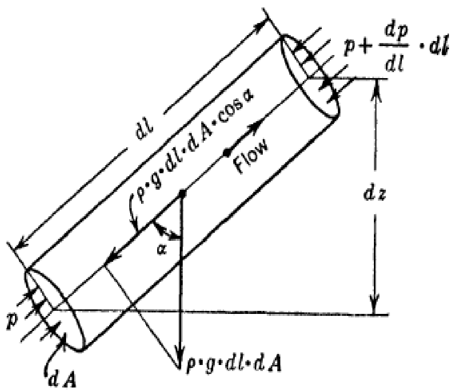
$$a_s = \frac{\partial u}{\partial l}\frac{dl}{dt} = u\frac{\partial u}{\partial l}$$

Where $u = \frac{\partial l}{\partial t}$ if we are following a fluid particle as it moves along a streamline, Therefore, acceleration in steady flow is due to the change of velocity with position.

Euler's Equation

By applying Newton's law to the motion of fluid masses, Leonhard Euler (1750) laid the groundwork for the study of the dynamics of ideal fluids.

Consider a differentially small section of streamtube having the dimensions shown in Fig. The forces tending to accelerate the fluid mass contained therein are:



(1) The component of weight in the direction of motion, and

(2) The forces on the ends of the element in the direction of motion due to pressure. Assuming that motion is in an upward direction and that pressure and velocity increase in this direction, the force dF_W due to the weight of the element is given by

$$dF_W = -\rho g\ dl\ dA\ \cos\alpha = -\rho g\ dl\ dA\ \frac{dz}{dl}$$

The forcedF_p, in the direction of motion, due to the pressure on the ends of the element, is

$$dF_p = p\ dA - \left(p + \frac{\partial p}{\partial l}dl\right)dA = -\frac{\partial p}{\partial l}dl\ dA$$

The mass dM of fluid being accelerated is

$$dM\ =\ \rho\ dl\ dA$$

And the total acceleration (a) is

$$a = u\frac{\partial u}{\partial l} + \frac{\partial u}{\partial t}$$

And for study flow

$$a = u\frac{du}{dl}$$

Substituting the above values in the Newtonian equation,

$$dF_W + dF_p = dM\ a$$

there results

$$-\left(\rho g\ dl\ dA\ \frac{dz}{dl}\right) - \left(\frac{\partial p}{\partial l}dl\ dA\right) = (\rho\ dl\ dA)\ \frac{udu}{dl}$$

Dividing by $\rho\ dl\ dA$ gives:

$$g\frac{dz}{dl} + \frac{dp}{\rho dl} + \frac{udu}{dl}\ =\ 0$$

$$\frac{dp}{\rho} + udu + gdz\ =\ 0 \qquad 2$$

the fundamental equation of steady fluid motion. By dividing this equation by g an alternate form of the equation is obtained

$$\frac{dp}{w} + \frac{udu}{g} + dz\ =\ 0 \qquad 3$$

Bernoulli's Equation

Euler's equation may be integrated along the streamtube with the following result

$$\int\frac{dp}{\rho} + \int udu + \int gdz\ =\ Constant$$

and if the fluid is a liquid, or a gas flowing with negligible change of density, the integrations may be carried out giving

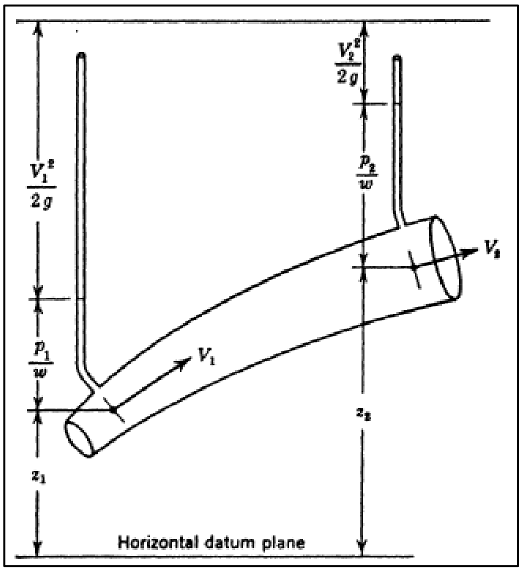
$$\frac{p}{\rho} + \frac{u^2}{2} + gz\ =\ Constant$$

or, multiplying by ρ:

$$p + \rho\frac{u^2}{2} + \rho gz\ =\ Constant \qquad 4$$

or, dividing by w:

$$\frac{p}{w} + \frac{u^2}{2g} + z\ =\ Constant \qquad 5$$



Now from the Bernoulli equation it becomes evident that the sum of three terms involving pressure, velocity, and vertical elevation will also be a constant at every point along the streamtube.

Writing equation 5 between two points on the typical streamtube of Fig. 3.3

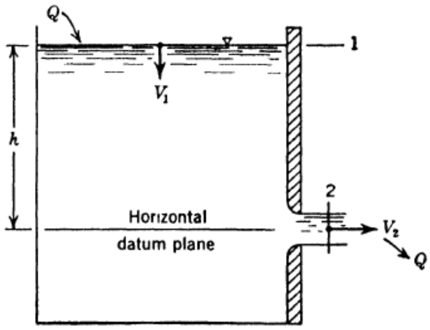
$$\frac{p_1}{w} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{u_2^2}{2g} + z_2$$

The Bernoulli terms in equation 5 thus are seen to have the dimensions of meter, or "meter of the fluid flowing," since w, the specific weight of the flowing fluid, appears in one of the terms. The Bernoulli terms in equation 4 will have the dimensions of pressure (Newton per square meter) and are designated respectively as pressure or (static pressure), velocity pressure and potential pressure.

Bernoulli's equation gives further aid in the interpretation of streamline pictures, equations 4 and 5 indicating that, when velocity increases, the sum of pressure and potential head must decrease. So where velocity is high pressure is low.

Application of Bernoulli's equation

a)$u = \sqrt{2gh}$



The above equation may be derived from Bernoulli's equation by considering steady flow through the reservoir and orifice of Fig. Taking section 1 at the free reservoir surface, section 2 in the jet immediately outside of the orifice, and the datum plane at the center of the orifice, Bernoulli's equation may be written as:

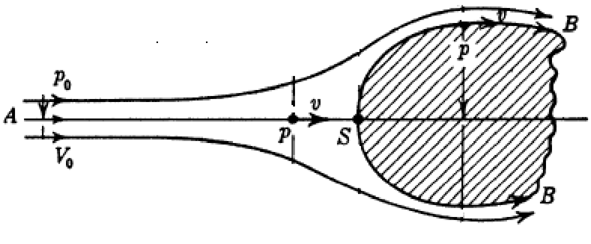
$$\frac{p_1}{w} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{u_2^2}{2g} + z_2$$

But, since the tank is very large compared to the orifice, u_1 will be very small and when squared usually becomes negligible. The pressure on the reservoir surface, p_1, is atmospheric and may be taken as zero. Atmospheric pressure surrounds the free jet, and thus the pressure in the jet at section 2 will be zero. Obviously, $z_1 = h$ and $z_2 = 0$; therefore, the Bernoulli equation becomes:

$$0 + 0 + h = 0 + \frac{u_2^2}{2g} + 0$$

Giving

$$u_2 = \sqrt{2gh}$$



b) Another useful special application of the Bernoulli principle is to the streamtube which approaches and remains adjacent to a solid body placed in a flowing fluid (Fig). Let this streamtube have an infinitesimal cross section and be represented by the streamline AB. Because of the interference of the body, the fluid particles moving on the streamline AB will decelerate as they approach the body and will temporarily come to rest at the point S, called the stagnation point; they then will move around the contour of the body with a variation in velocity approximately as shown on the figure. From Bernoulli's equation 4 the pressure variation with these velocity changes will be about as shown, and the pressure at the stagnation point, the stagnation pressure, p_s , may be calculated from:

$$p_s + \rho\frac{u_s^2}{2} = p_o + \rho\frac{u_o^2}{2}$$

p_o and u_o being respectively the pressure and velocity in the undisturbed fluid ahead of the solid body. In this equation u_s = 0; Therefore:

$$p_s = p_o + \rho\frac{u_o^2}{2}$$

Illustrative problem

A submarine moves through salt water at a depth of 50 *m* and at a speed of 22 *m/s*. Calculate the pressure on the nose of the submarine.

$$p_s = p_o + \rho\frac{u_o^2}{2}$$

$$p_s = 50 \times 9810 + \frac{1}{2} \times 1000 \times (22)^2$$

$$p_s = 501500\ N/m^2$$

Bernoulli's Equation For Real Fluid

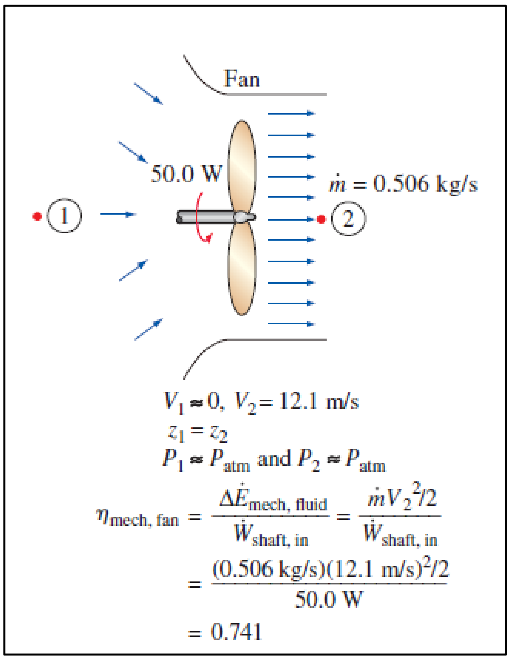
The Bernoulli's equation was derived on the assumption that fluid is inviscid (non-viscous) and therefore frictionless. But all the real fluids are viscous and hence offer resistance to flow. Thus there are always some losses in fluid flows and hence in the application of Bernoulli's equation, these losses have to be taken into consideration. Thus the Bernoulli's equation for real fluids between points 1 and 2 is given as:

$$\frac{p_1}{w} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{u_2^2}{2g} + z_2 + h_L$$

Where h_L is loss of energy between points 1 and 2.

Mechanical energy and efficiency

In fluid systems, we are usually interested in increasing the pressure, velocity, and/or elevation of a fluid. This is done by supplying mechanical energy to the fluid by a pump, a fan, or a compressor (we will refer to all of them as pumps). Or we are interested in the reverse process of extracting mechanical energy from a fluid by a turbine and producing mechanical power in the form of a rotating shaft that can drive a generator or any other rotary device. The degree of perfection of the conversion process between the mechanical work supplied or extracted and the mechanical energy of the fluid is expressed by the pump efficiency and turbine efficiency, defined as



$$\eta_{pump} = \frac{Mechanical\ energy\ increase\ of\ the\ fluid}{Mechanical\ energy\ input}$$

$$= \frac{\Delta \dot{E}_{mech,fluid}}{\dot{W}_{shaft,in}}$$

Where $\Delta \dot{E}_{mech,fluid} = \dot{E}_{mech,out} - \dot{E}_{mech,in}$ is the rate of increase in the mechanical energy of the fluid, which is equivalent to the useful pumping power $W_{pump,u}$ supplied to the fluid, and.

$$\eta_{turbine} = \frac{Mechanical\ energy\ output}{Mechanical\ energy\ decrease\ of\ the\ fluid} = \frac{W_{shaft,out}}{\Delta \dot{E}_{mech,fluid}}$$

$\Delta \dot{E}_{mech,fluid} = \dot{E}_{mech,in} - \dot{E}_{mech,out}$ is the rate of decrease in the mechanical energy of the fluid, which is equivalent to the mechanical power extracted from the fluid by the turbine $W_{turbine,e}$. A pump or turbine efficiency of 100 percent indicates perfect conversion between the shaft work and the mechanical energy of the fluid, and this value can be approached (but never attained) as the frictional effects are minimized.

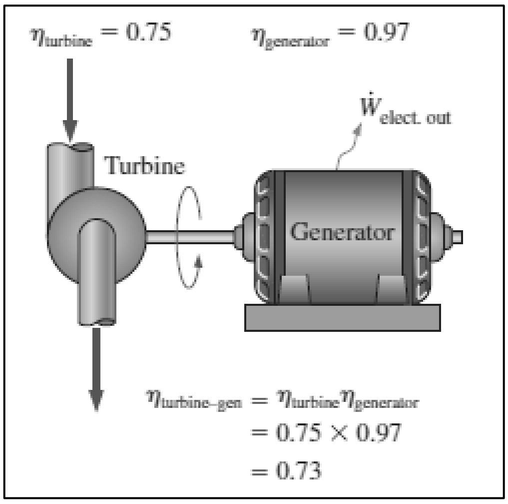
The mechanical efficiency should not be confused with the motor efficiency and the generator efficiency, which are defined as:

Motor: $$\eta_{motor} = \frac{\text{Mechanical power output}}{\text{Electric power input}} = \frac{\dot{W}_{shaft,\ out}}{\dot{W}_{elect,\ in}}$$

and

Generator: $$\eta_{generator} = \frac{\text{Electric power output}}{\text{Mechanical power input}} = \frac{\dot{W}_{elect,\ out}}{\dot{W}_{shaft,\ in}}$$

A pump is usually packaged together with its motor, and a turbine with its generator. Therefore, we are usually interested in the combined or overall efficiency of pump–motor and turbine–generator combinations (see the fig).



INTERNAL FLOW

Fluid flow is classified as <u>external</u> or <u>internal</u>, depending on whether the fluid is forced to flow over a surface or in a conduit. Internal and external flows exhibit very different characteristics

Liquid or gas flow through pipes or ducts is commonly used in heating and cooling applications and fluid distribution networks. The fluid in such applications is usually forced to flow by a fan or pump through a flow section. We pay particular attention to friction, which is directly related to the pressure drop and head loss during flow through pipes and ducts. The pressure drop is then used to determine the pumping power requirement. A typical piping system involves pipes of different diameters connected to each other by various fittings or elbows to route the fluid, valves to control the flow rate, and pumps to pressurize the fluid.

Laminar and Turbulent Flows

Reynolds experiment

The type of flow is determined from the Reynolds number i.e. $\frac{\rho u d}{\mu}$. This was demonstrated by Reynold in 1883. His apparatus is shown in Fig.

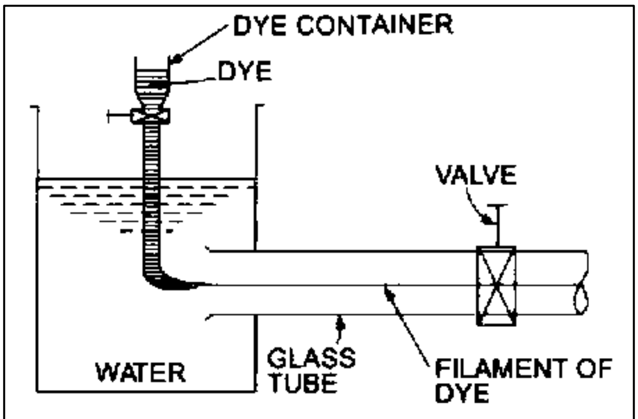


Fig. *Reynold apparatus.*

The apparatus consists of:

1. A tank containing water at constant head.
2. A small tank containing some dye.
3. A glass tube having a bell-mouthed entrance at one end and a regulating value at other ends.

The water from the tank was allowed to flow through the glass tube. The velocity of flow was varied by the regulating valve. A liquid dye having same specific weight as water was introduced into the glass tube as shown in Fig.
The following observations were made by Reynold:

- When the velocity of flow was low. The dye filament in the glass tube was in the form of a straight line. This straight line of dye filament was parallel to the glass tube, which was the case of laminar flow as shown in Fig. (a).
- With the increase of velocity of flow, the dye-filament was no longer a straight-line but it became a wavy one as shown in Fig. (b). this shows that flow is no longer laminar.
- With further increase of velocity of flow, the wavy dye-filament broke-up and finally diffused in water as shown in Fig. (c). this means that the fluid particles of the dye at this higher velocity are moving in random fashion, which shows the case of turbulent flow. Thus in case of turbulent flow the mixing of dye-filament and water is intense and flow is irregular, random and disorderly.

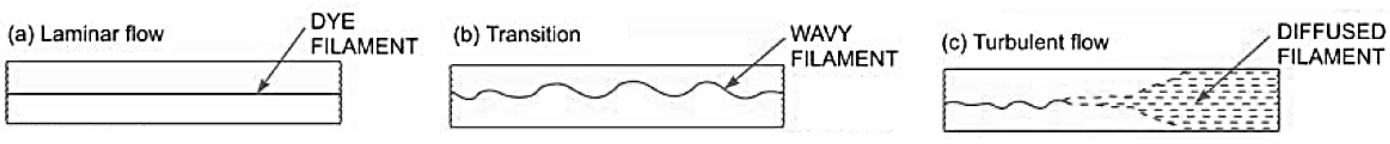


Fig. *Different stages of filament.*

Laminar and turbulent flows are characterized on the basis of Reynolds number
For Reynolds number (Re) < 2300 ... flow in pipes is laminar,
For Reynolds number (Re) > 4000 ... flow in pipes is turbulent
For Re between 2000 and 4000 ... flow in pipes may be laminar or turbulent.

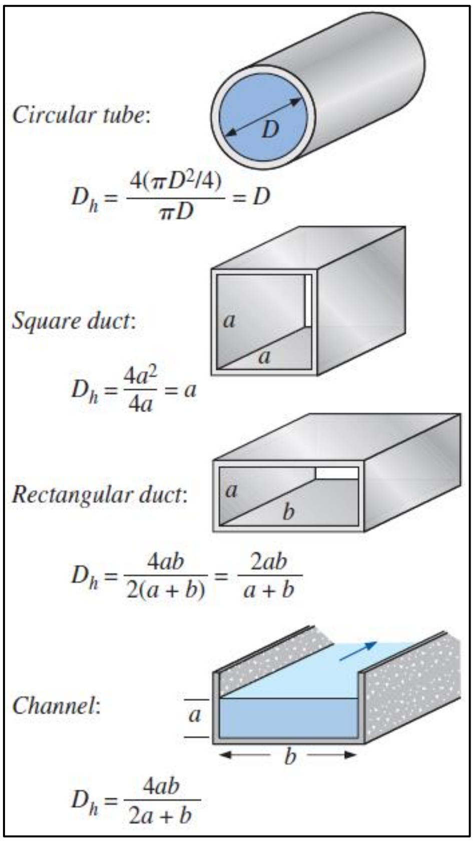
Reynolds number
Reynolds number is a dimensionless number, defined as:
Re = (Inertia forces/Viscous forces). Or,

$$Re = \frac{\rho u d}{\mu}$$

Where
u = free stream velocity, m/s
d = hydraulic diameter
When the Reynolds number is low, i.e. when the flow is laminar, inertia forces are small compared to viscous forces and the velocity fluctuations are 'damped out' by the viscosity effects and the layers of fluid flow systematically, parallel to each other. When the Reynolds number is large, i.e. when the flow is turbulent, inertia forces are large compared to the viscous forces and the flow becomes chaotic
For flow through noncircular pipes, the Reynolds number is based on the hydraulic diameter D_h defined as (the Fig.)

$$Hydraulic\ diameter: D_h = \frac{4\,A_c}{P}$$

Where A_c is the cross-sectional area of the pipe and p is its wetted perimeter.

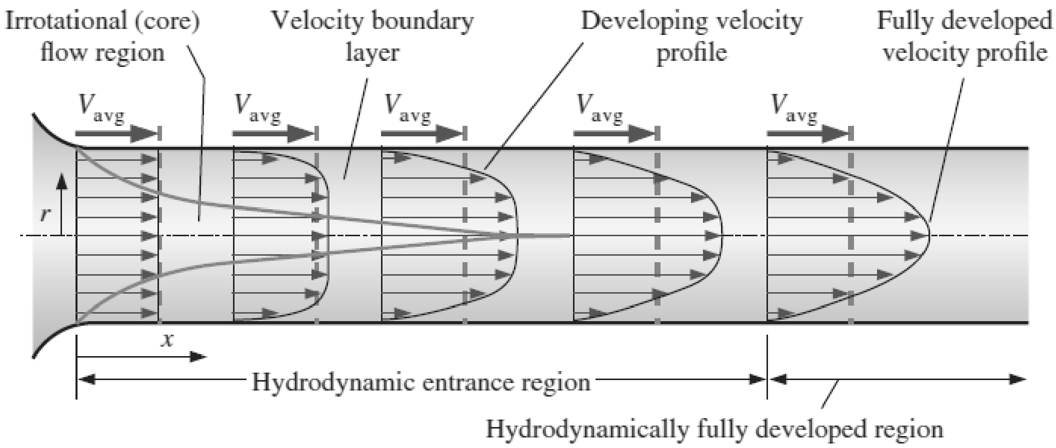


The entrance region

Consider a fluid entering a circular pipe at a uniform velocity. Because of the no-slip condition, the fluid particles in the layer in contact with the wall of the pipe come to a complete stop. This layer also causes the fluid particles in the adjacent layers to slow down gradually as a result of friction. To make up for this velocity reduction, the velocity of the fluid at the midsection of the pipe has to increase to keep the mass flow rate through the pipe constant. As a result, a velocity gradient develops along the pipe.

The region of the flow in which the effects of the viscous shearing forces caused by fluid viscosity are felt is called the velocity boundary layer or just the boundary layer. The hypothetical boundary surface divides the flow in a pipe into two regions: the boundary layer region, in which the viscous effects and the velocity changes are significant, and the irrotational (core) flow region, in which the frictional effects are negligible and the velocity remains essentially constant in the radial direction.

The thickness of this boundary layer increases in the flow direction until the boundary layer reaches the pipe center and thus fills the entire pipe, as shown in Fig., and the velocity becomes fully developed.



The region from the pipe inlet to the point at which the velocity profile is fully developed is called the hydrodynamic entrance region, and the length of this region is called the hydrodynamic entry length.

- Flow in the entrance region is called hydrodynamically developing flow since this is the region where the velocity profile develops.
- The region beyond the entrance region in which the velocity profile is fully developed and remains unchanged is called the hydrodynamically fully developed region.

However, the pipes used in practice are usually several times the length of the entrance region, and thus the flow through the pipes is often assumed to be fully developed for the entire length of the pipe. This simplistic approach gives reasonable results for long pipes but sometimes poor results for short ones.

Laminar Flow

Flow of Viscous Fluid in Circular Pipes-Hagen Poiseuille Law

Hagen-Poiseuille theory is based on the following assumptions:

1- The fluid follows Newton's law of viscosity.
2- There is no slip of fluid particles at the boundary (i.e. the fluid particles adjacent to the pipe will have zero velocity).

The fig. shows a horizontal circular pipe of radius R, having laminar flow of fluid through it. Consider a small concentric cylinder (fluid element) of radius r and length

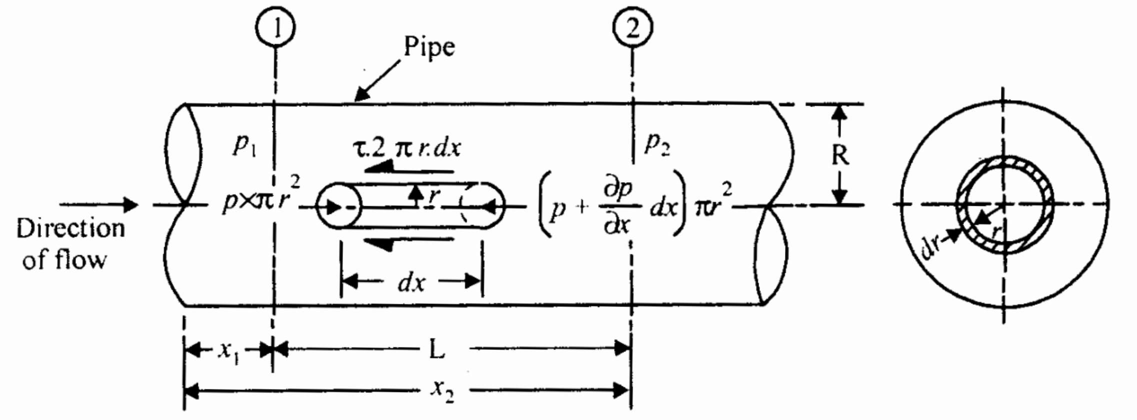


Fig. Viscous/laminar flow through a circular pipe.

dx as a free body.

If τ : is the shear stress, the shear force F is given by

$$F = \tau \times 2\pi r \times dx$$

Let P be the intensity of pressure at left end and the intensity of pressure at the right end be

$$\left(P + \frac{\partial P}{\partial x} dx\right)$$

Thus the forces acting on the fluid element are:

1- The shear force, $\tau \times 2\pi r \times dx$ on the surface of fluid element.
2- The pressure force, $P \times \pi r^2$ on the left end.
3- The pressure force, $\left(P + \frac{\partial P}{\partial x} dx\right) \times \pi r^2$ on the right end.

For steady flow, the net force on the cylinder must be zero.

$$\left[P \times \pi r^2 - \left(P + \frac{\partial P}{\partial x} dx\right) \times \pi r^2\right] - \tau \times 2\pi r \times dx = 0$$

$$-\frac{\partial P}{\partial x} dx \times \pi r^2 - \tau \times 2\pi r \times dx = 0$$

$$\tau = -\frac{\partial P}{\partial x} \times \frac{r}{2} \qquad (1)$$

- Eqn. (1) shows that flow will occur only if pressure gradient exists in the direction of flow.

- The negative sign shows that pressure decreases in the direction of flow.

- Eqn. (1) indicates that the shear stress varies linearly across the section (see the next Fig.). Its value is zero at the centre of pipe (r = 0) and maximum at the pipe wall

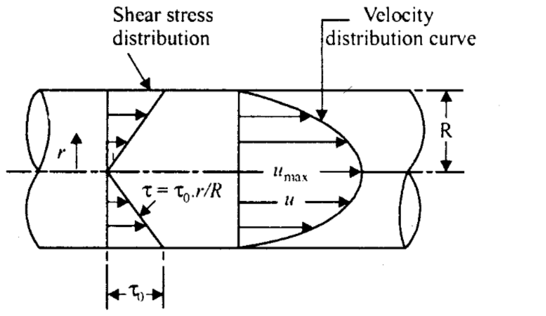


Fig. Shear stress and velocity distribution across a section.

(i) Velocity Distribution. To obtain the velocity distribution across a section, the value of shear stress $\tau = \mu \frac{du}{dy}$ is substituted in equation (1).

But in the relation $\tau = \mu \frac{du}{dy}$, y is measured from the pipe wall. Hence

$$y = R - r \quad \text{and} \quad dy = -dr$$

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

Substituting this value in (1), we get

$$-\mu \frac{du}{dr} = -\frac{\partial p}{\partial x}\frac{r}{2} \quad \text{or} \quad du = \frac{1}{2\mu}\left(\frac{\partial p}{\partial x}\right) r \cdot dr$$

Integrating this above equation w.r.t. 'r', we get

$$u = \frac{1}{4\mu}\frac{\partial p}{\partial x} r^2 + C$$

where C is the constant of integration and its value is obtained from the boundary condition that at $r = R$, $u = 0$.

$$\therefore \qquad 0 = \frac{1}{4\mu}\frac{\partial p}{\partial x} R^2 + C$$

$$\therefore \qquad C = -\frac{1}{4\mu}\frac{\partial p}{\partial x} R^2$$

Substituting this value of C in equation we get

$$u = \frac{1}{4\mu}\frac{\partial p}{\partial x} r^2 - \frac{1}{4\mu}\frac{\partial p}{\partial x} R^2$$

$$= -\frac{1}{4\mu}\frac{\partial p}{\partial x}[R^2 - r^2]$$

In the above equation, values of $\frac{\partial P}{\partial x}$ and R are constant, which means the velocity, u varies with the square of r. The equation is equation of parabola. This shows that the velocity distribution across the section of a pipe is parabolic. This velocity distribution is shown in the above Fig.

Ratio of maximum velocity to average velocity

The velocity is maximum, when r = 0 in the equation. Thus maximum velocity, U_{max} is obtained as

$$U_{max} = -\frac{1}{4\mu}\frac{\partial P}{\partial x}R^2$$

Let the average velocity, $\bar{u}$ to be

$$\bar{u} = \frac{U_{max}}{2}$$

So

$$\bar{u} = -\frac{1}{8\mu}\frac{\partial P}{\partial x}R^2$$

Drop of Pressure for a given Length (L) of a pipe

From the above equation we have

$$\bar{u} = \frac{1}{8\mu}\left(\frac{-\partial p}{\partial x}\right)R^2 \quad \text{or} \quad \left(\frac{-\partial p}{\partial x}\right) = \frac{8\mu\bar{u}}{R^2}$$

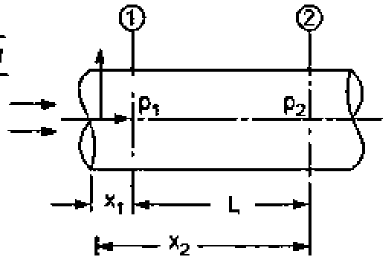


Fig.

Integrating the above equation w.r.t. x, we get

$$-\int_2^1 dp = \int_2^1 \frac{8\mu\bar{u}}{R^2}\,dx$$

$$\therefore \qquad -[p_1 - p_2] = \frac{8\mu\bar{u}}{R^2}[x_1 - x_2] \text{ or } (p_1 - p_2) = \frac{8\mu\bar{u}}{R^2}[x_2 - x_1]$$

$$= \frac{8\mu\bar{u}}{R^2}L \qquad \{\because \; x_2 - x_1 = L \text{ from Fig.}\}$$

$$= \frac{8\mu\bar{u}L}{(D/2)^2} \qquad \left\{\because \; R = \frac{D}{2}\right\}$$

or

$$(p_1 - p_2) = \frac{32\mu\bar{u}L}{D^2}, \quad \text{where } p_1 - p_2 \text{ is the drop of pressure.}$$

$$\therefore \quad \text{Loss of pressure head} = \frac{p_1 - p_2}{\rho g}$$

$$\therefore \qquad \frac{p_1 - p_2}{\rho g} = h_f = \frac{32\mu\bar{u}L}{\rho g D^2}$$

Equation is called **Hagen Poiseuille Formula.**

Illustrative Example

A capillary tube is 30 mm long and 1 mm bore. The head required to produce a flow rate of 8 mm3/s is 30 mm. The fluid density is 800 kg/m3.

Calculate the dynamic and kinematic viscosity of the oil.

SOLUTION

Rearranging Poiseuille's equation we get

$$\mu = \frac{h_f \rho g D^2}{32 L u_m}$$

$$A = \frac{\pi d^2}{4} = \frac{\pi \times 0.001^2}{4} = 7.85 \times 10^{-7} m^2$$

$$u_m = \frac{Q}{A} = \frac{8 \times 10^{-9}}{7.85 \times 10^{-7}} = 0.01019 \frac{m}{s}$$

$$\mu = \frac{0.03 \text{ x } 800 \text{ x } 9.81 \text{ x } 0.001^2}{32 \text{ x } 0.03 \text{ x } 0.01018} = 0.0241 \text{ N s/m or } 24.1 \text{ cP}$$

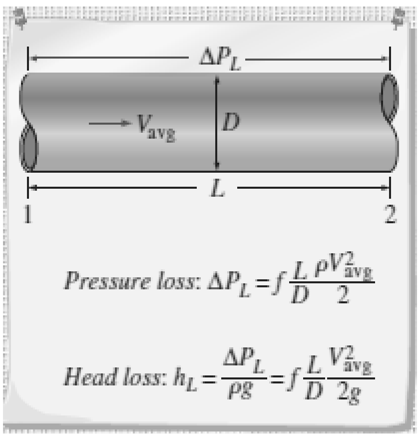
$$\nu = \frac{\mu}{\rho} = \frac{0.0241}{800} = 30.11 \text{ x } 10^{-6} \text{ m}^2/\text{s}$$

Pressure Drop and Head Loss

A quantity of interest in the analysis of pipe flow is the pressure drop ΔP since it is directly related to the power requirements of the fan or pump to maintain flow.
From (Hagen Poiseuille) equation we have

$$\Delta P = P_1 - P_2 = \frac{32\mu L \bar{u}}{D^2} \qquad (a)$$

Note from Eq. (a) that the pressure drop is proportional to the viscosity μ of the fluid, and ΔP would be zero if there were no friction. Therefore, the drop of pressure from P_1 to P_2 in this case is due entirely to viscous effects.
In practice, it is convenient to express the pressure loss for all types of fully developed internal flows (laminar or turbulent flows, circular or noncircular pipes, smooth or rough surfaces, horizontal or inclined pipes) as shown in (Fig.)



$$\Delta P_L = f\frac{L}{D}\frac{\rho \bar{u}^2}{2} \qquad (b)$$

Setting Eqs. (a) and (b) equal to each other and solving for f gives the friction factor for fully developed laminar flow in a circular pipe,

$$f = \frac{64\mu}{\rho D\bar{u}} = \frac{64}{Re} \qquad (c)\ for\ more\ information\ see\ the\ appendex\ of\ ch.4$$

This equation shows that in laminar flow, the friction factor is a function of the Reynolds number only and is independent of the roughness of the pipe surface (assuming, of course, that the roughness is not extreme).
In the analysis of piping systems, pressure losses are commonly expressed in terms of the equivalent fluid column height, called the **head loss** h_L. Noting from fluid statics that $\Delta P = \rho g h$ and thus a pressure difference of ΔP corresponds to a fluid height of $h_L = {\Delta P}/{\rho g}$, the pipe head loss is obtained by:

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{\bar{u}^2}{2g} \qquad (d)$$

The head loss h_L represents the additional height that the fluid needs to be raised by a pump in order to overcome the frictional losses in the pipe. Equations (b) and (d) are valid for both laminar and turbulent flow in both circular and noncircular pipes, but Eq. (c) is valid only for fully developed laminar flow in circular pipes.
Once the pressure loss (or head loss) is known, the required pumping power to overcome the pressure loss is determined from

$$\dot{W}_{pump,L} = Q\Delta P_L = Q\rho g h_L = \dot{m} g h_L$$

Where Q is the volume flow rate and ṁ is the mass flow rate.

Turbulent Flow in Pipes

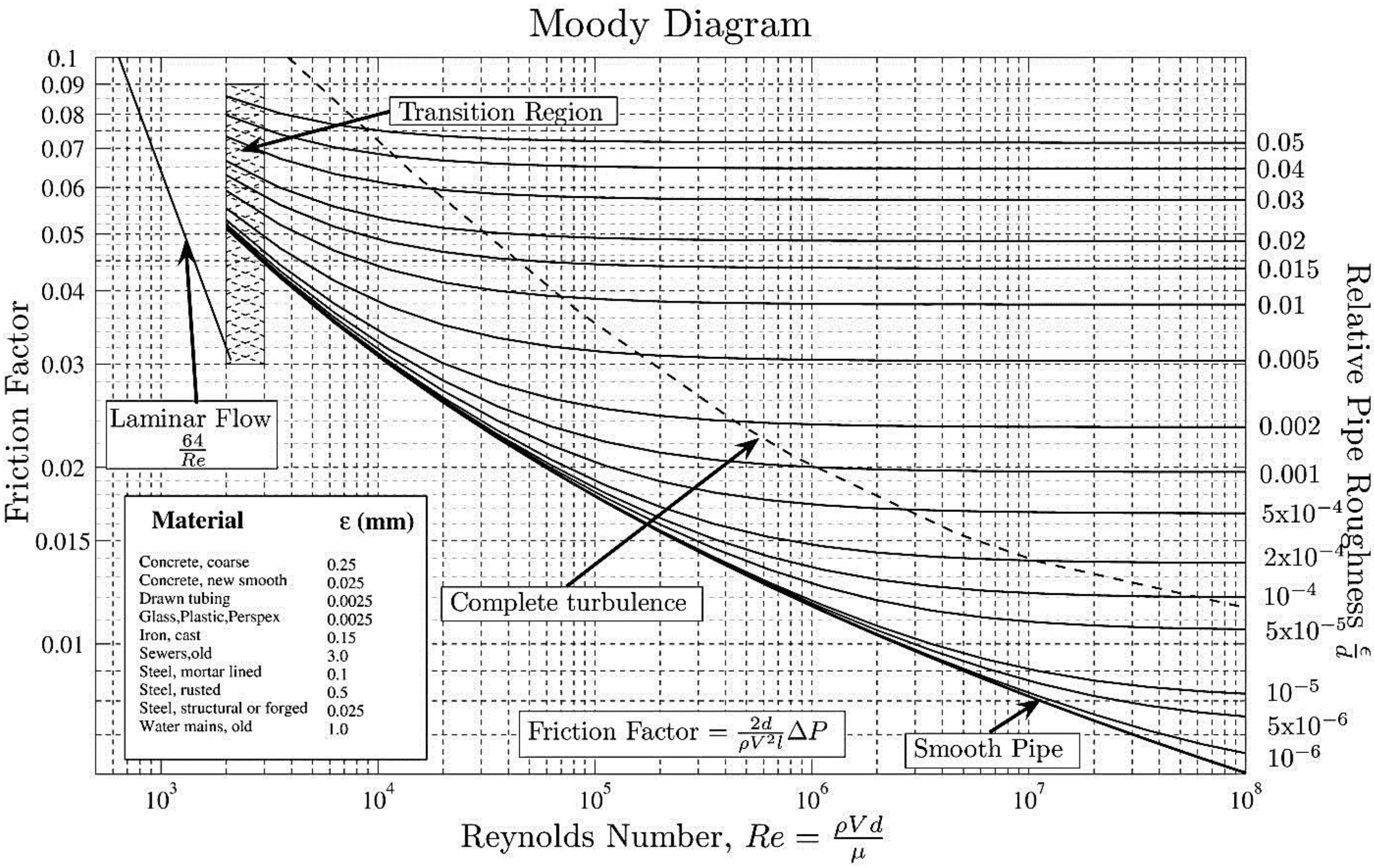
Most flows encountered in engineering practice are turbulent, and thus it is important to understand how turbulence affects wall shear stress. However, turbulent flow is a complex mechanism dominated by fluctuations, and despite tremendous amounts of work done in this area by researchers, turbulent flow still is not fully understood. Therefore, we must rely on experiments and the empirical or semi-empirical correlations developed for various situations.
Turbulent flow is characterized by disorderly and rapid fluctuations of swirling regions of fluid, called eddy, throughout the flow. These fluctuations provide an additional mechanism for momentum and energy transfer.
In laminar flow, fluid particles flow in an orderly manner along pathlines, and momentum and energy are transferred across streamlines by molecular diffusion.
In turbulent flow, the swirling eddies transport mass, momentum, and energy to other regions of flow much more rapidly than molecular diffusion, greatly enhancing mass, momentum, and heat transfer

Friction Factor Calculation for Turbulent Flow
The moody chart and the Colebrook equation

The friction factor in fully developed turbulent pipe flow depends on the Reynolds number and the relative roughness ε/D, which is the ratio of the mean height of roughness of the pipe to the pipe diameter. The functional form of this dependence cannot be obtained from a theoretical analysis, and all available results are obtained from experiments using artificially roughened surfaces.
Most such experiments were conducted by Prandtl’s student J. Nikuradse in 1933, followed by the works of others. The friction factor was calculated from measurements of the flow rate and the pressure drop.

The experimental results are presented in tabular and graphical. A formula that gives an approximate answer for any surface roughness is that given by Haaland.

$$\frac{1}{\sqrt{C_f}} = -1.8\, log_{10}\left[\frac{6.9}{R_e} + \left(\frac{\varepsilon}{3.71}\right)^{1.11}\right]$$



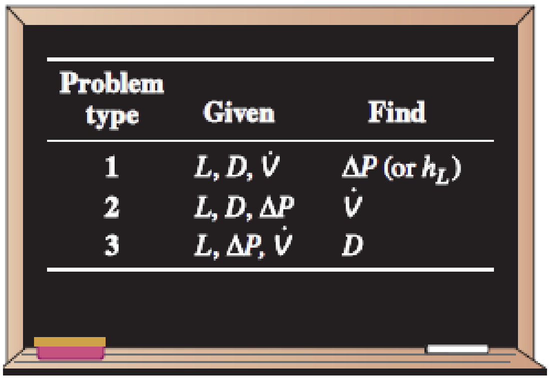
We make the following observations from the Moody chart:

- For laminar flow, the friction factor decreases with increasing Reynolds number, and it is independent of surface roughness.
- The friction factor is a minimum for a smooth pipe (but still not zero because of the no-slip condition) and increases with roughness.
- For smooth pipes, use the bottom curve on the diagram, (surface roughness$\cong$ 0).
- At very large Reynolds numbers the friction factor curves corresponding to specified relative roughness curves are nearly horizontal, and thus the friction factors are independent of the Reynolds number. The flow in that region is called fully rough flow

Types of fluid flow problems

In the design and analysis of piping systems that involve the use of the Moody chart, we usually encounter three types of problems (the fluid and the roughness of the pipe are assumed to be specified in all cases) (Fig):

1. Determining the **pressure drop** (or head loss) when the pipe length and diameter are given for a specified flow rate (or velocity)
2. Determining the **flow rate** when the pipe length and diameter are given for a specified pressure drop (or head loss)
3. Determining the **pipe diameter** when the pipe length and flow rate are given for a specified pressure drop (or head loss)

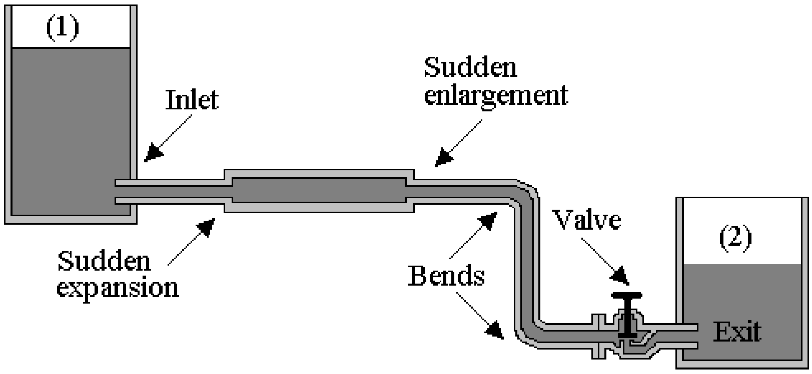


Problem type	Given	Find
1	$L, D, \dot{V}$	ΔP (or h_L)
2	$L, D, \Delta P$	$\dot{V}$
3	$L, \Delta P, \dot{V}$	D

Minor Losses

The fluid in a typical piping system passes through various fittings, valves, bends, elbows, tees, inlets, exits, expansions, and contractions in addition to the straight sections of piping. These components interrupt the smooth flow of the fluid and cause additional losses because of the flow separation and mixing they induce.

- In a system with long pipes, these losses are minor compared to the head loss in the straight sections (the major losses) and are called minor losses.
- But, in some cases the minor losses may be greater than the major losses. This is the case, for example, in systems with several turns and valves in a short distance.

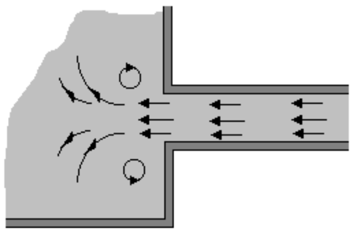


The head loss introduced by a Flow through valves and fittings is very complex, and a theoretical analysis is generally not posable. Therefore, minor losses are determined experimentally, usually by the manufacturers of the components.

Minor losses are usually expressed in terms of the loss coefficient K_L (also called the resistance coefficient), defined as following:

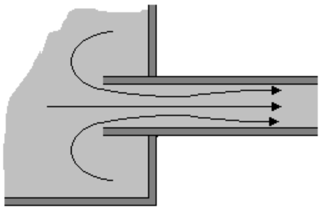
1. Exit from a pipe into a tank.

The liquid emerges from the pipe and collides with stationary liquid causing it to swirl about before finally coming to rest. All the kinetic energy is dissipated by friction. It follows that all the kinetic head is lost so k = 1.0



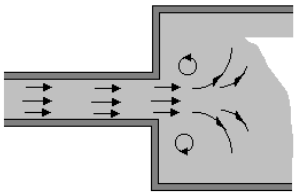
2. Entry to a pipe from a tank

The value of k varies from 0.78 to 0.04 depending on the shape of the inlet. A good rounded inlet has a low value but the case shown is the worst.



3. Sudden enlargement

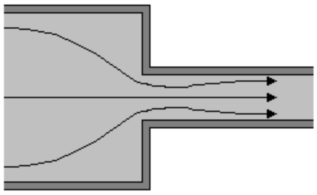
This is similar to a pipe discharging into a tank but this time it does not collide with static fluid but with slower moving fluid in the large pipe. The resulting loss coefficient is given by the following expression.



$$k = \left\{1 - \left(\frac{d_1}{d_2}\right)^2\right\}^2$$

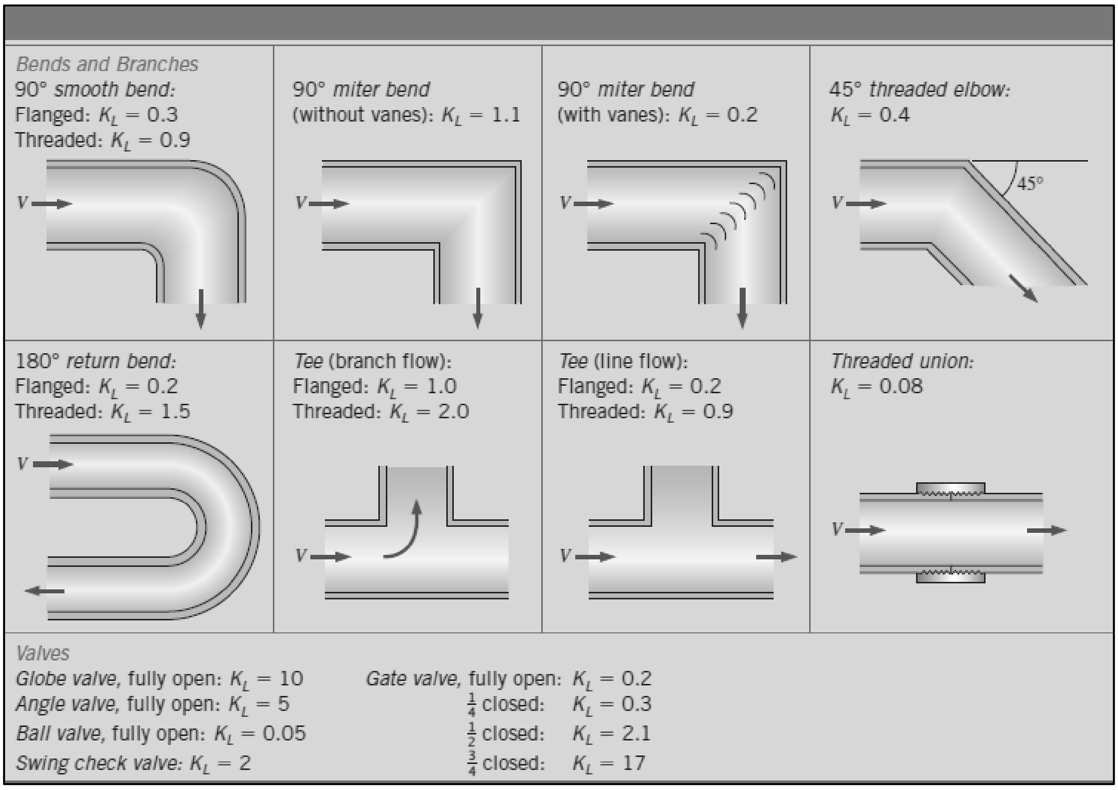
4. Sudden contraction

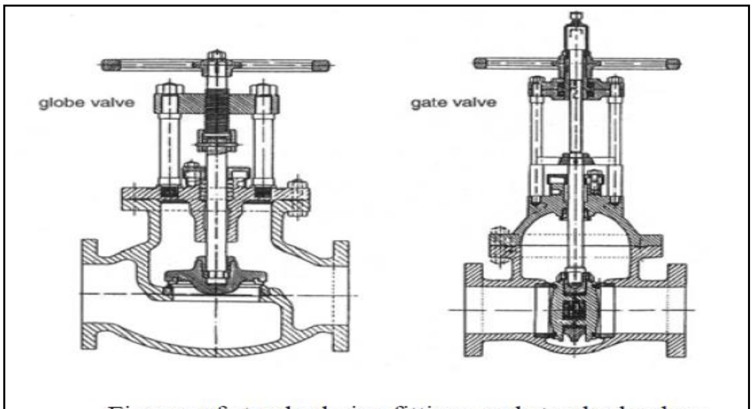
This is similar to the entry to a pipe from a tank. The best case gives k = 0 and the worst case is for a sharp corner which gives k = 0.5.



5. Bends and fittings

The k value for bends depends upon the radius of the bend and the diameter of the pipe. The k value for bends and the other cases is on various data sheets. For fittings, the manufacturer usually gives the k value. Often instead of a k value, the loss is expressed as an equivalent length of straight pipe that.



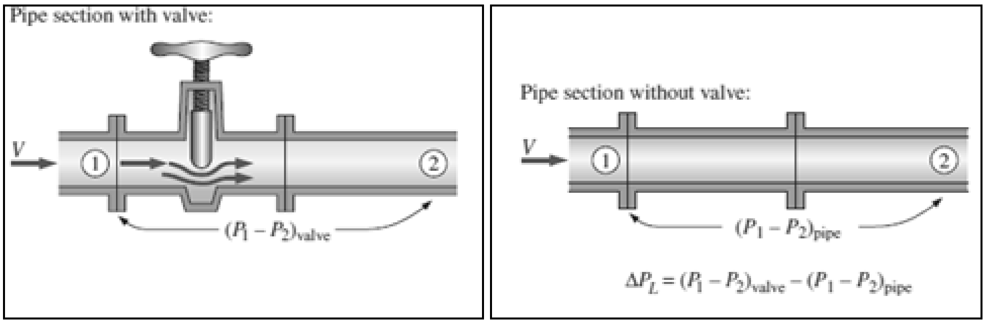


Figures of standard pipe fittings and standard valves

From the above points for the values of loss coefficient K_L Minor losses are calculated as following:

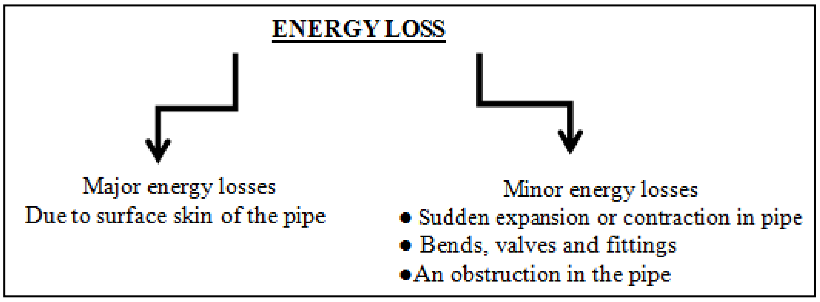
$$h_L = K_L \frac{u^2}{2g}$$

Where h_L the additional head loss is in the piping system caused by insertion of the component, and is defined as $h_L = \frac{\Delta P_L}{\rho g}$. For example, imagine replacing the valve in the next Fig.



Total Friction Losses

The frictional losses from the friction in the straight pipe (skin friction), enlargement losses, contraction losses, and losses in fittings and valves are all incorporated in h_L term in Bernoulli's equation, so that.



$$h_L = f\frac{l}{d}\frac{{u_1}^2}{2g} + K_e\frac{{u_1}^2}{2g} + K_c\frac{{u_2}^2}{2g} + K_f\frac{u^2}{2g}$$

$$h_L = f\frac{l}{d}\frac{\left(\frac{Q}{A_1}\right)^2}{2g} + K_e\frac{\left(\frac{Q}{A_1}\right)^2}{2g} + K_c\frac{\left(\frac{Q}{A_2}\right)^2}{2g} + K_f\frac{\left(\frac{Q}{A}\right)^2}{2g}$$

$$h_L = \left(\frac{8fL}{\pi^2 g d_1^5} + \frac{8K_e}{\pi^2 g d_1^4} + \frac{8K_c}{\pi^2 g d_2^4} + \frac{8K_f}{\pi^2 g d^4}\right) \times Q^2$$

If all the velocity u, u_1, and u_2 are the same, then this equation becomes, for this special case;

$$h_L = \left[f\frac{l}{d} + K_e + K_c + K_f\right]\frac{u^2}{2g}$$

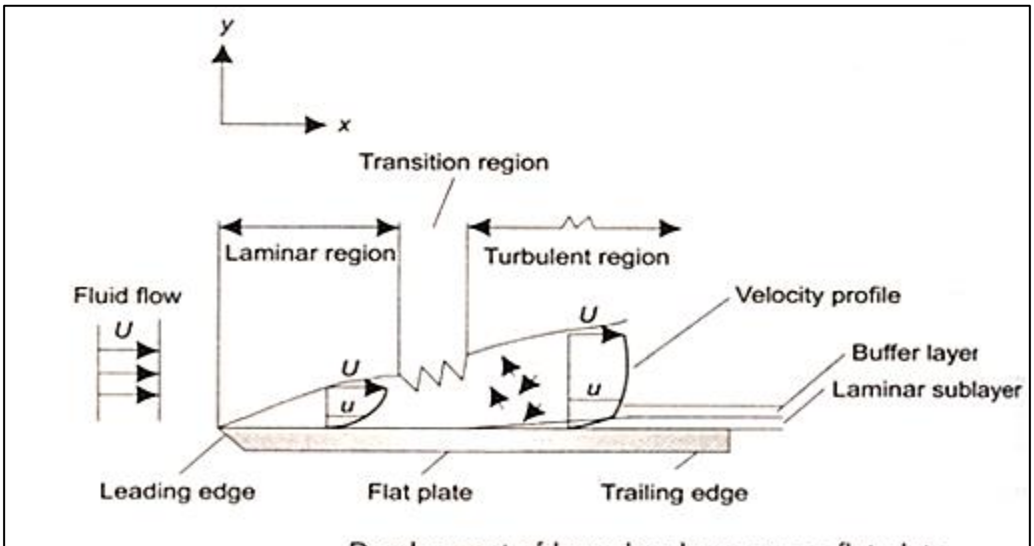
EXTERNAL FLOW

Parallel flow over flat plate

Velocity boundary layer

Let us first study the development of boundary layer for a flow over a flat plate. Flow over a flat plate is important from a practical point of view, since flow over turbine blades and aerofoil sections of air plane wings can be approximated as flow over a flat plate. See Fig.

Consider a thin, flat plate. The leading edge and the trailing edge of the plate are shown in the Fig. Let a fluid approach the flat plate at a free stream velocity of U. The fluid layer immediately in contact with the plate surface adheres to the surface and remains stationary, and in fluid mechanics, this phenomenon is known as 'no slip condition. Then, the fluid layer next to this stationary layer has its velocity retarded because of the viscosity effects i.e. due to the frictional force or 'drag' exerted between the stationary and the moving layers. This effect continues with subsequent layers up to some distance in the y-direction till the velocity equals the free stream velocity U. This region of fluid layer in which the viscosity effects are predominant is known as the 'velocity (or hydrodynamic) boundary layer', or simply the 'boundary layer'.



Development of boundary layer over a flat plate

Note the following points in connection with the boundary layer:

1. The boundary layer divides the flow field into two regions: one, 'the boundary layer region' where the viscosity effects are predominant and the velocity gradients are very steep, and, second, inviscid region' where the frictional

effects are negligible and the velocity remains essentially constant at the free stream value.

2. Since the fluid layers in the boundary layer travel at different velocities, the faster layer exerts a drag force (or frictional force) on the slower layer below it; the drag force per unit area is known as shear stress (τ)'. Shear stress is proportional to the velocity gradient at the surface. This is the reason why in fluid mechanics, the velocity profile has to be found out to determine the frictional force exerted by a fluid on the surface. Shear stress is given by:

$$\tau_s = \mu.\left(\frac{dU}{dy}\right)_{y=0} \qquad N/m^2$$

   Where μ is 'dynamic viscosity' of the fluid; its unit is kg/(ms). Viscosity is a measure of resistance to flow.

3. Use of the above Eq. to determine the surface shear stress is not very convenient, since it requires a mathematical expression for the velocity profile; so, in practice, surface shear stress is determined in terms of the free stream velocity from the following relation:

$$\tau_s = C_{fa}\frac{\rho U^2}{2} \qquad N/m^2$$

   where C_f is a 'friction coefficient' or 'drag coefficient', ρ is the density of the fluid. C_{fa} is determined experimentally in most cases. Then the drag force over the entire plate surface is determined from:

$$F_D = C_{fa}.A.\frac{\rho U^2}{2} \qquad N$$

   where A = surface area, m^2.

4. Starting from the leading edge of the plate, for some distance along the length of the plate, the flow in the boundary layer is 'laminar' i.e. the layers of fluid are parallel to each other and the flow proceeds in a systematic, orderly manner. However, after some distance, disturbances appear in the flow and beyond this 'transition region', flow becomes completely chaotic and there is complete mixing of 'chunks' of fluid moving in a random manner i.e. the flow becomes 'turbulent'.

5. Transition from laminar to turbulent flow depends primarily on the free stream velocity, fluid properties, surface temperature and surface roughness, and is characterized by 'Reynolds number'. Reynolds number is a dimensionless number, defined as:

$$Re = \frac{U.x}{v}$$

   Where
   U = free stream velocity, m/s
   x = characteristic length i.e. for a flat plate it is the length along the plate in the flow direction, from the leading edge, and
   v = kinematic viscosity of fluid = μ/ρ , m^2/s, where ρ is the density of fluid.
   For a flat plate, in general, for practical purposes, the 'critical Reynolds number, at which the flow changes from laminar to turbulent is taken as ($5 * 10^5$). It should be understood clearly that this is not a fixed value but depends on many parameters including the surface roughness.

6. Turbulent region of boundary layer is preceded by transition region as shown in Fig.

7. Turbulent boundary layer itself is made of three layers: a very thin layer called laminar sub-layer', then, a ''buffer layer' and, finally, the 'turbulent layer',
8. Thickness of the boundary layer, δ, increases along the flow direction; δ is related to the Reynolds number as follows: in the laminar flow region:

$$\delta_{lam} = \frac{5.x}{(Re_x)^{0.5}}$$

$$Cf_{average} = \frac{1.33}{(Re_L)^{0.5}}$$

and for turbulent flow region:

$$\delta_{turb} = \frac{0.376.x}{(Re_x)^{0.2}}$$

$$Cf_{average} = \frac{0.074}{(Re_L)^{0.5}}$$

where Re_x is the Reynolds number at position x from the leading edge.

DISCHARGE FROM A TANK THROUGH AN ORIFICE

Co-efficient of Discharge (Cd)

It is defined as the ratio of the actual discharge from an orifice to the theoretical discharge from the orifice. It is denoted by Cd. If Q is actual discharge and Q_{th} is the theoretical discharge then mathematically Cd, is given as

$$Cd = \frac{Q}{Q_{th}} = \frac{Actual\ velocity\ x\ Actual\ area}{Theoretical\ velocity\ x\ Theoretical\ area}$$

The value of Cd, varies from 0.61 to 0.65. For general purpose the value of Cd is taken as 0.62.

Experimental determination of co-efficient of discharge (Cd).

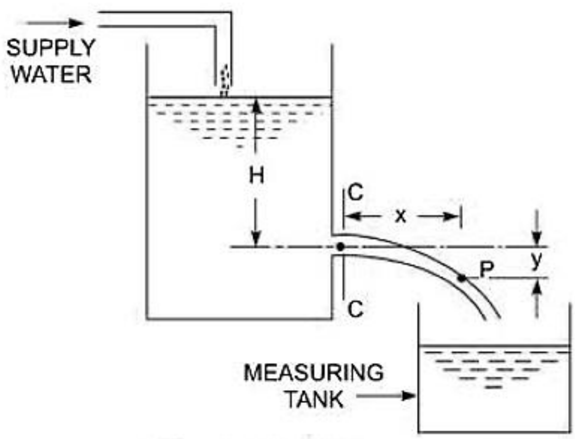
The water is allowed to flow through an orifice fitted to a tank under a constant head. H as shown in Fig. The water is collected in a measuring tank for a known time, t. The height Of water in the measuring tank is noted down. Then actual discharge through orifice:

$$Q = \frac{Area\ of\ measuring\ tank\ \times Height\ of\ water\ in\ measuring\ tank}{Time(t)}$$

And

$$theoretical\ discharge = area\ of\ orifice \times \sqrt{2gH}$$

$$Cd = \frac{Q}{a \times \sqrt{2gH}}$$



Time Of Emptying A Tank Through An Orifice At Its Bottom

Consider a tank containing some liquid up to a height of H_1 Let an orifice is fitted at the bottom of the tank. It is required to find the time for the liquid surface to fall from the height H_1 to a height H_2.

Let A = Area of the tank

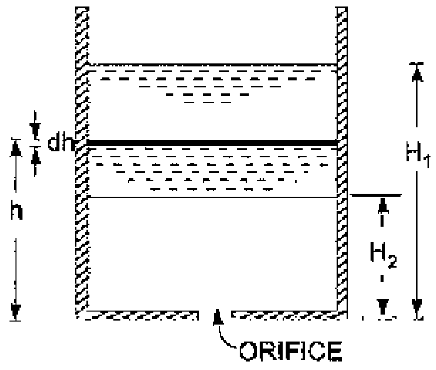
a = Area of the orifice

H_1 = Initial height of the liquid

H_2 = Final height of the liquid

T= Time in seconds for the liquid to fall from H_1 to H_2

Let at any lime, the height of liquid from orifice is h and let the liquid surface fall by a small height dh in time dT. Then



Volume of liquid leaving the tank in time, $= A \times dh$

Also the theoretical velocity through orifice, $u = \sqrt{2gh}$

∴ **Discharge through orifice/sec,**

$$dQ = C_d \times \text{Area of orifice} \times \text{Theoretical velocity} = C_d \,.\, a \,.\, \sqrt{2gh}$$

∴ **Discharge through orifice in time interval**

$$= C_d \,.\, a \,.\, \sqrt{2gh}\,.\, dT$$

As the volume of liquid leaving the tank is equal to the volume of liquid flowing through orifice in time dT, we have

$$A(-dh) = C_d \,.\, a \,.\, \sqrt{2gh}\,.\, dT$$

– ve sign is inserted because with the increase of time, head on orifice decreases.

∴ $$-A dh = C_d \,.\, a \,.\, \sqrt{2gh}\,.\, dT \text{ or } dT = \frac{-A\,dh}{C_d \,.\, a \,.\, \sqrt{2gh}} = \frac{-A(h)^{-1/2}}{C_d \,.\, a \,.\, \sqrt{2g}} dh$$

By integrating the above equation between the limits H_1 and H_2, the total time, T is obtained as

$$\int_0^T dT = \int_{H_1}^{H_2} \frac{-Ah^{-1/2}\,dh}{C_d \,.\, a \,.\, \sqrt{2g}} = \frac{-A}{C_d \,.\, a \,.\, \sqrt{2g}} \int_{H_1}^{H_2} h^{-1/2}\,dh$$

or $$T = \frac{-A}{C_d \,.\, a \,.\, \sqrt{2g}} \left[\frac{h^{-\frac{1}{2}+1}}{-\frac{1}{2}+1}\right]_{H_1}^{H_2} = \frac{-A}{C_d \,.\, a \,.\, \sqrt{2g}} \left[\frac{\sqrt{h}}{\frac{1}{2}}\right]_{H_1}^{H_2}$$

$$= \frac{-2A}{C_d \,.\, a \,.\, \sqrt{2g}}\left[\sqrt{H_2} - \sqrt{H_1}\right] = \frac{2A\left[\sqrt{H_1} - \sqrt{H_2}\right]}{C_d \,.\, a \,.\, \sqrt{2g}}$$

For emptying the tank completely, $H_2 = 0$ and hence

$$T = \frac{2A\sqrt{H_1}}{C_d \,.\, a \,.\, \sqrt{2g}}.$$

VORTEX FLOW

Vortex flow is defined as the flow of a fluid along a curved path or the flow of a rotating mass of fluid is known a 'Vortex Flow". The vortex flow is of two types namely :

1. Forced vortex flow.
2. Free vortex flow.

- **Forced Vortex Flow**. Forced vortex flow is defined as that type of vortex flow, in which some external torque is required to rotate the fluid mass. The fluid mass in this type of flow, rotates at constant angular velocity, w. The tangential velocity of any fluid particle is given by

$$v = w \times r \qquad (1)$$

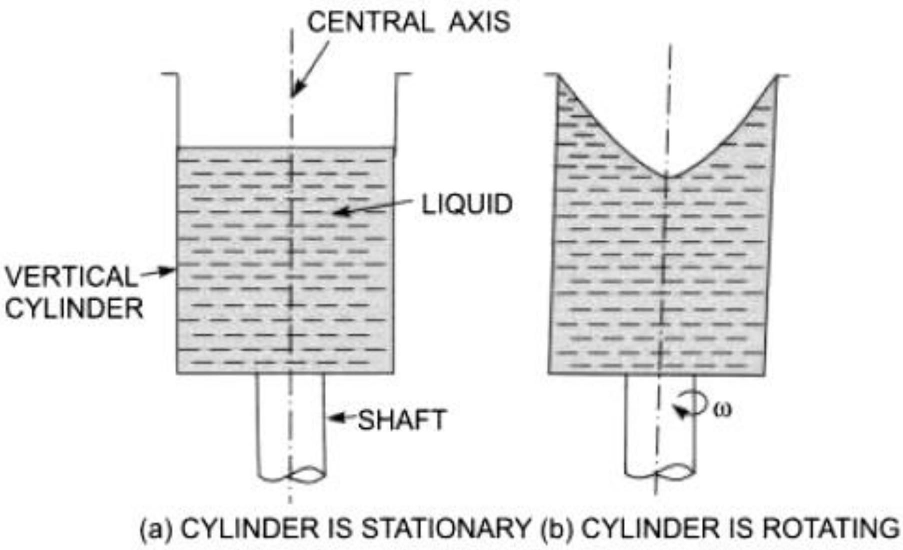


Fig. *Forced vortex flow.*

Hence angular velocity w is given by

$$w = \frac{v}{r} = constant.$$

Examples of forced vortex are:

1. A vertical cylinder containing liquid which is rotated about its central axis with a constant angular velocity to. As shown in the Fig.

2. Flow of liquid inside the impeller of a centrifugal pump.

- **Free Vortex Flow**. When no external torque is required to rotate the fluid mass, that type of flow is called free vortex flow.

Example for the free vortex flow is the flow of liquid through a hole provided at the bottom of a container.

Equation of Motion for Vortex Flow

Consider a fluid element ABCD (shown shaded in the Fig). rotating at a uniform velocity in a horizontal plane about an axis perpendicular to the plane of paper and passing through 0.

Let

- r= Radius of the element from 0.
- $\Delta\theta$ = Angle subtended by the element at 0.
- dr = Radial thickness of the element.
- dA = Area of cross-section of element.

The forces acting on the element are:

1. Pressure force, $P\ dA$, on the face AB.
2. Pressure force, $\left(P\ dA + \frac{\partial}{\partial r} P\ dA\ dr\right)$ on the face CD.
3. Centrifugal force, $\frac{m\,v^2}{r}$ acting in the direction away from the centre, 0.

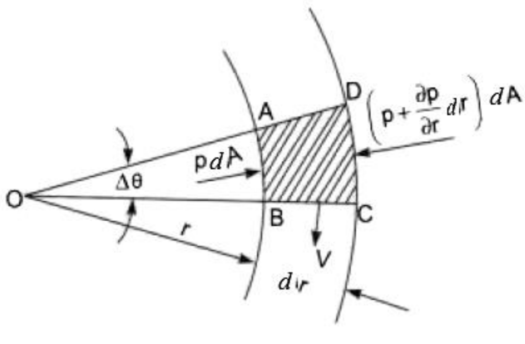


Fig.

Now, the mass of the element = density x Volume

$$\rho\ dA\ dr$$

So the centrifugal force = $\rho\ dA\ dr \frac{v^2}{r}$

Equating the forces in the radial direction, we get

$$P\ dA - \left(P\ dA + \frac{\partial}{\partial r} P\ dA\ dr\right) + \rho\ dA\ dr \frac{v^2}{r} = 0$$

$$\frac{\partial}{\partial r} P\ dA\ dr = \rho\ dA\ dr \frac{v^2}{r}$$

$$\frac{\partial P}{\partial r} = \rho \frac{v^2}{r} \qquad (a)$$

The expression $\frac{dP}{\partial r}$ is called pressure gradient in the radial direction. As $\frac{dP}{\partial r}$ is positive, hence pressure increases with the increase of radius r.

But the pressure varies with the vertical plane which given by the hydrostatic law:

$$\frac{\partial p}{\partial z} = -\rho g \qquad (b)$$

The pressure, p varies with respect to r and z or p is a function of r and z and hence total derivative of p is

$$dP = \frac{\partial P}{\partial r} dr + \frac{\partial p}{\partial z} dz$$

Substituting the values of $\frac{\partial P}{\partial r}$ from equation (a) and $\frac{\partial p}{\partial z}$ from equation (b), we get

$$dP = \rho \frac{v^2}{r} dr - \rho g dz \qquad (2)$$

Eq. (2) gives the variation of pressure of a rotating fluid in any plane

Equation for forced vortex flow

For the forced vortex flow, from equation (1).we have

$v = w \times r$

Where w = Angular velocity = Constant.

Substituting the value of v in equation (2), we get

$$dP = \rho \frac{w^2 r^2}{r} dr - \rho g dz$$

Consider two points 1 and 2 in the fluid having forced vortex flow as shown in the Fig.

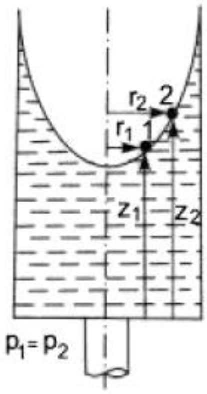


Fig.

Integrating the above equation for points 1 and 2, we get

$$\int_1^2 dP = \int_1^2 \rho w^2 r\, dr - \int_1^2 \rho g dz$$

$$P_2 - P_1 = \frac{\rho w^2}{2}[r_2^2 - r_1^2] - \rho g[z_2 - z_1]$$

$$P_2 - P_1 = \frac{\rho}{2}[w^2 r_2^2 - w^2 r_1^2] - \rho g[z_2 - z_1]$$

$$P_2 - P_1 = \frac{\rho}{2}[v_2^2 - v_1^2] - \rho g[z_2 - z_1] \qquad as\ | v_2 = w r_2\ and\ v_1 = w r_1 |$$

If the points 1 and 2 lie on the free surface of the liquid, then $P_2 = P_1$ and hence above equation becomes:

$$0 = \frac{\rho}{2}[v_2^2 - v_1^2] - \rho g[z_2 - z_1]$$

$$\rho g[z_2 - z_1] = \frac{\rho}{2}[v_2^2 - v_1^2]$$

$$[z_2 - z_1] = \frac{1}{2g}[v_2^2 - v_1^2]$$

Let point 1 lies on the axis of rotation, then $v_1 = w r_1 = w \times 0 = 0$. The above equation becomes as:

$$[z_2 - z_1] = \frac{v_2^2}{2g}$$

Let $z_2 - z_1 = Z$

So the Eq. becomes:

$$Z = \frac{v_2^2}{2g} = \frac{w^2 r_2^2}{2g} \qquad (3)$$

Thus Z varies with the square of r. Hence equation (3) is an equation of parabola. This means the free surface of the liquid is a parabolic.

Problem *Prove that in case of forced vortex, the rise of liquid level at the ends is equal to the fall of liquid level at the axis of rotation.*

Solution. Let R = radius of the cylinder.

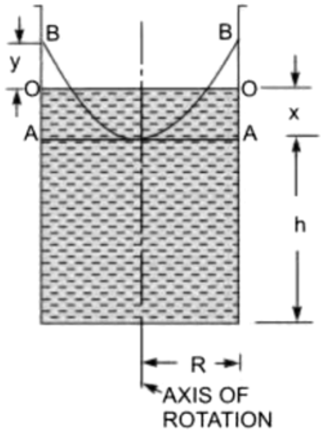
O-O = Initial level of liquid in cylinder when the cylinder is not rotating.

$\therefore$ Initial height of liquid $= (h + x)$

$\therefore$ Volume of liquid in cylinder $= \pi R^2 \times$ Height of liquid

$$= \pi R^2 \times (h + x) \qquad \ldots(i)$$

Let the cylinder is rotated at constant angular velocity ω. The liquid will rise at the ends and will fall at the centre.



Fig

Let y = Rise of liquid at the ends from O-O

x = Fall of liquid at the centre from O-O.

Then volume of liquid

= [Volume of cylinder upto level B-B]

− [Volume of paraboloid]

= [$\pi R^2 \times$ Height of liquid upto level B-B]

$$- \left[\frac{\pi R^2}{2} \times \text{Height of paraboloid}\right]$$

$$= \pi R^2 \times (h + x + y) - \frac{\pi R^2}{2} \times (x + y)$$

$$= \pi R^2 \times h + \pi R^2 (x + y) - \frac{\pi R^2}{2} \times (x + y)$$

$$= \pi R^2 \times h + \frac{\pi R^2}{2} (x + y) \qquad \ldots(ii)$$

Equating (i) and (ii), we get

$$\pi R^2 (h + x) = \pi R^2 \times h + \frac{\pi R^2}{2} (x + y)$$

or
$$\pi R^2 h + \pi R^2 x = \pi R^2 \times h + \frac{\pi R^2}{2} x + \frac{\pi R^2}{2} y$$

or
$$\pi R^2 x - \frac{\pi R^2}{2} x = \frac{\pi R^2}{2} y \quad \text{or} \quad \frac{\pi R^2}{2} x = \frac{\pi R^2}{2} y \quad \text{or} \quad x = y$$

or Fall of liquid at centre = Rise of liquid at the ends.

Problem *An open circular tank of 20 cm diameter and 100 cm long contains water upto a height of 60 cm. The tank is rotated about its vertical axis at 300 r.p.m., find the depth of parabola formed at the free surface of water.*

Solution. Given :

Diameter of cylinder = 20 cm

$\therefore$ Radius, $R = \frac{20}{2} = 10$ cm

Height of liquid, $H = 60$ cm

Speed, $N = 300$ r.p.m.

Angular velocity, $\omega = \frac{2\pi N}{60} = \frac{2 \times \pi \times 300}{60} = 31.41$ rad/sec.

Let the depth of parabola $= Z$

$$Z = \frac{\omega^2 r_2^2}{2g}, \text{ where } r_2 = R$$

$$= \frac{\omega^2 R^2}{2g} = \frac{(31.41)^2 \times (10)^2}{2 \times 981} = \textbf{50.28 cm. Ans.}$$

Closed Cylindrical Vessels

If a cylindrical vessel is closed at the top, which contains some liquid, the shape of parabolic formed due to rotation of the vessel will be as shown in the Fig. for different speed of rotations.

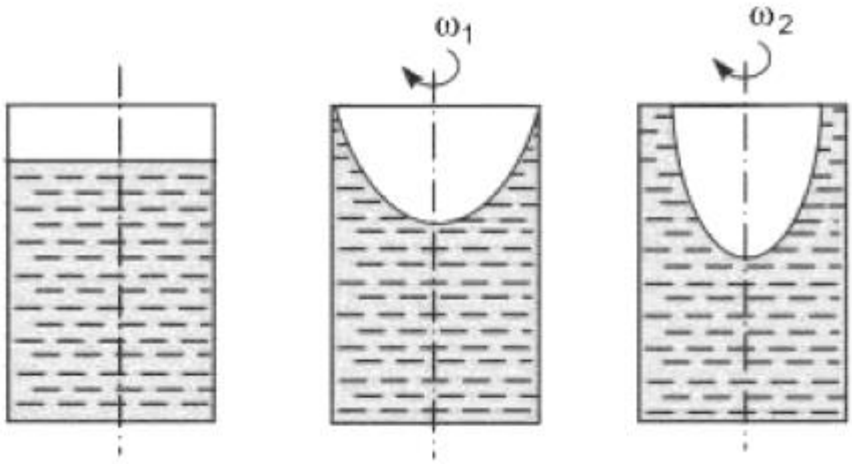
The Fig.(a) shows the initial stage of the cylinder, when it is not rotated. Fig. (b) shows the shape of the parabolic formed when the speed of rotation is w. If the speed is increased further say w_2, the shape of paraboloid formed will be as shown in Fig. (c). In this case the radius of the parabola at the top of the vessel is unknown. Also the height of the parabolic formed corresponding to angular speed w_2 is unknown. Thus to solve the two unknown, we should have two equations. One equation is

$$Z = \frac{w^2 r_2^2}{2g}$$

The second equation is obtained from the fact that for closed vessel, volume of air before rotation is equal to the volume of air after rotation.

Volume of air before rotation = Volume of closed vessel - Volume of liquid in vessel

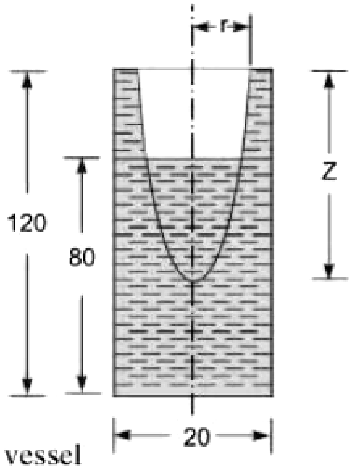
Volume of air after rotation = Volume of parabolic formed $= \frac{\pi r^2}{2} Z$



(a) (b) (c)

Problem *A vessel, cylindrical in shape and closed at the top and bottom, contains water upto a height of 80 cm. The diameter of the vessel is 20 cm and length of vessel is 120 cm. The vessel is rotated at a speed of 400 r.p.m. about its vertical axis. Find the height of paraboloid formed.*

Solution. Given :



Initial height of water $= 80$ cm

Diameter of vessel $= 20$ cm

$\therefore$ Radius, $R = 10$ cm

Length of vessel $= 120$ cm

Speed, $N = 400$ r.p.m.

$$\therefore \quad \omega = \frac{2\pi N}{60} = \frac{2\pi \times 400}{60} = 41.88 \text{ rad/s}$$

When the vessel is rotated, let Z

$= $ Height of paraboloid formed

$r = $ Radius of paraboloid at the top of the vessel

This is the case of closed vessel.

$\therefore$ Volume of air before rotation = Volume of air after rotation

$$\text{or} \quad \frac{\pi}{4} D^2 \times L - \frac{\pi}{4} D^2 \times 80 = \pi r^2 \times \frac{Z}{2}$$

where Z = Height of paraboloid, r = Radius of parabola.

$$\text{or} \quad \frac{\pi}{4} D^2 \times 120 - \frac{\pi}{4} D^2 \times 80 = \pi r^2 \times \frac{Z}{2}$$

$$\text{or} \quad \frac{\pi}{4} \times D^2 \times (120 - 80) = \frac{\pi}{4} D^2 \times 40 = \pi r^2 \times \frac{Z}{2}$$

$$\text{or} \quad \frac{\pi}{4} \times 20^2 \times 40 = 4000 \times \pi = \pi r^2 \times \frac{Z}{2}$$

$$\therefore \quad r^2 \times Z = \frac{4000 \times \pi \times 2}{\pi} = 8000 \qquad \ldots(i)$$

Using relation $$Z = \frac{\omega^2 r^2}{2g}, \text{ we get } Z = \frac{41.88^2 \times r^2}{2g} = \frac{41.88^2 \times r^2}{2 \times 981} = 0.894\, r^2$$

$$\therefore \quad r^2 = \frac{z}{0.894}$$

Substituting this value of r^2 in (i), we get

$$\frac{Z}{0.894} \times Z = 8000$$

$$\therefore \quad Z^2 = 8000 \times 0.894 = 7152$$

$$\therefore \quad Z = \sqrt{7152} = \mathbf{84.56 \text{ cm. Ans.}}$$

Equation for free vortex flow

In this case of flow Bernoulli’s equation is applicable.

APPENDIX

Friction Coefficient for Laminar Flow

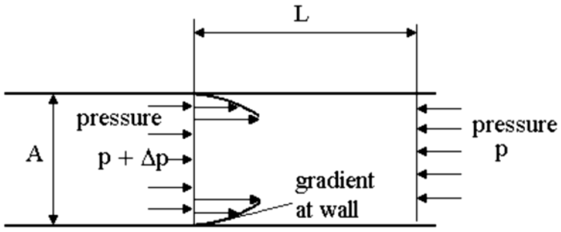
The friction coefficient is a convenient idea that can be used to calculate the pressure drop in a pipe. It is defined as follows.

$$Cf = \frac{\text{Wall Shear Stress}}{\text{Dynamic Pressure}}$$

- Dynamic pressure
  Consider a fluid flowing with mean velocity um. If the kinetic energy of the fluid is converted into flow or fluid energy, the pressure would increase. The pressure rise due to this conversion is called the dynamic pressure.

$$p \ = \ ½\, \rho \, \bar{u}^2$$

- Wall shear stress τ
  The wall shear stress is the shear stress in the layer of fluid next to the wall of the pipe.



The shear stress in the layer next to the wall is wall $\tau = \mu \frac{du}{dy}$

The shear force resisting flow is $F_s = \tau\pi\text{LD}$

The resulting pressure drop produces a force of

$$F_p = [-P + (P + \Delta P)] \times \frac{\pi D^2}{4}$$

$$F_p = \frac{\Delta p \; \pi D^2}{4}$$

Equating forces(F_s, F_p) gives

$$\tau\pi\text{LD} = \frac{\Delta p \; \pi D^2}{4}$$

$$\tau = \frac{\Delta p \; D}{4L}$$

$$Cf = \frac{\text{Wall Shear Stress}}{\text{Dynamic Pressure}} = \frac{D\Delta p}{4L \frac{\rho \bar{u}^2}{2}}$$

$$4 \; Cf = \frac{D\Delta p}{L \frac{\rho \bar{u}^2}{2}}$$

Where Cf is the Fanning friction coefficient, named after the American engineer John Fanning (1837–1911), which is defined as $Cf = f/4$

Where f is called the Darcy–friction factor, named after the Frenchman Henry Darcy (1803–1858)

So

$$f = \frac{D\Delta p}{L \frac{\rho \bar{u}^2}{2}}$$

$$\Delta p = f \frac{L}{D} \frac{\rho \bar{u}^2}{2}$$

From Poiseuille's equation $\Delta P = \frac{32\mu L \bar{u}}{D^2}$

$$f \frac{L}{D} \frac{\rho \bar{u}^2}{2} = \frac{32\mu L \bar{u}}{D^2}$$

$$f = \frac{64\mu}{\rho \bar{u}^2 D} = \frac{64}{Re}$$

CHAPTER 5
FLOW RATE AND VELOCITY MEASUREMENT

A major application area of fluid mechanics is the determination of the flow rate of fluids, and numerous devices have been developed over the years for the purpose of flow metering. Flowmeters range widely in their level of size, cost, accuracy, versatility, capacity, pressure drop, and the operating principle. We give an overview of the meters commonly used to measure the flow rate of liquids and gases flowing through pipes or ducts. We limit our consideration to incompressible flow.

- Some flowmeters measure the flow rate directly by discharging and recharging a measuring chamber of known volume continuously and keeping track of the number of discharges per unit time. But most flowmeters measure the flow rate indirectly they measure the average velocity u or a quantity that is related to average velocity such as pressure, and determine the volume flow rate Q from, $Q = u \times A_c$ Where Ac is the cross-sectional area of flow
  Therefore, measuring the flow rate is usually done by measuring flow velocity, and many flowmeters are simply velocimeters used for the purpose of metering flow.

- The velocity in a pipe varies from zero at the wall to a maximum at the center, and it is important to keep this in mind when taking velocity measurements. For laminar flow, for example, the average velocity is half the centerline velocity. But this is not the case in turbulent flow, and it may be necessary to take the average or an integral of several local velocity measurements to determine the average velocity.

The flow rate of water through a garden hose, for example, can be measured simply by collecting the water in a bucket of known volume and dividing the amount collected by the collection time (Fig).
A crude way of estimating the flow velocity of a river is to drop a float on the river and measure the drift time between two specified locations.
In this chapter we discuss devices that are commonly used to measure velocity and flow rate.

Pitot-tube
It is a device used for measuring the velocity at any point in a pipe or a channel. It is based on the principle that if the velocity of flow at a point becomes zero, the pressure there is increased due to the conversion of the kinetic energy into pressure energy. The pitot-tube consists of both a stagnation pressure tap and several circumferential static pressure taps and it measures both stagnation and static pressures (Fig)

Let:
p_1 = intensity of pressure at point (1)
u_1 = velocity of flow at (1) which is zero
p_2 = pressure at point (2)
u_2 = velocity at point (2)
Applying Bernoulli's equation at points (1) and (2), we get

$$\frac{p_1}{\rho g} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{u_2^2}{2g} + z_2$$

But z_1= z_2 as points (1) and (2) are on the same line and u_1 = 0.
$\frac{p_1}{\rho g}$ = pressure head at (1) = h_1
$\frac{p_2}{\rho g}$= pressure head at (2) = h_2
Substituting these values, we get

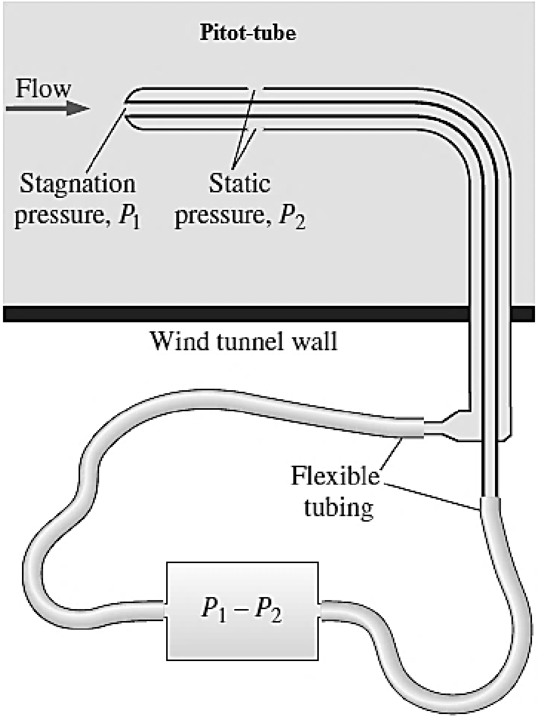
$$h_1 = h_2 + \frac{u_2^2}{2g}$$

$$u_2 = \sqrt{2g\Delta h}$$

This is theoretical velocity. Actual velocity is given by

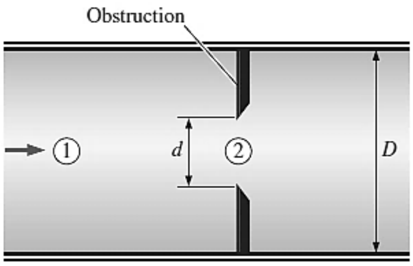
$$(u_2)_{act} = C_d\sqrt{2g\Delta h}$$

Where C_d is coefficient of pitot-tube



- It is used to measure velocity in both liquids and gases.

Obstruction Flowmeters: Orifice, Venturi, and Nozzle Meters
Consider incompressible steady flow of a fluid in a horizontal pipe of diameter *D* that is constricted to a flow area of diameter *d*, as shown in Fig.
The mass balance and the Bernoulli equations between a location before the constriction (point 1) and the location where constriction occurs (point 2) are written as



Mass balance: $Q = A_1u_1 = A_2u_2 \rightarrow$

$$u_1 = (A_2/A_1)u_1 = (d/D)^2\, u_2$$

Bernoulli equation ($z_1 = z_2$): $\frac{p_1}{\rho g} + \frac{u_1^2}{2g} = \frac{p_2}{\rho g} + \frac{u_2^2}{2g}$

Combining the above Eqs. and solving for velocity u_2 gives

$$u_2^2 = \left(\frac{p_1 - p_2}{\rho g} + \frac{u_1^2}{2g}\right) \times 2g$$

$$u_2^2 = \left(\frac{2(p_1 - p_2)}{\rho} + ((d/D)^2\, u_2)^2\right)$$

$$u_2^2 = \left(\frac{2(p_1 - p_2)}{\rho} + (\beta^4\, u_2)^2\right)$$

Obstruction (with no loss):

$$u_2 = \sqrt{\frac{2(P_1 - P_2)}{\rho(1 - \beta^4)}}$$

Where $\beta = d/D$ is the diameter ratio. Once u_2 is known, the flow rate can be determined from u_2

$$Q = A_2 u_2 = (\pi d^2/4) u_2.$$

Noting that the pressure drop between two points along the flow is measured easily by a differential pressure manometer, it appears that a simple flow rate measurement device can be built by obstructing the flow. Flowmeters based on this principle are called **obstruction flowmeters** and are widely used to measure flow rates of gases and liquids.

- The velocity in Eq. above is obtained by assuming no loss, and thus it is the maximum velocity that can occur at the constriction site. In reality, some pressure losses due to frictional effects are inevitable, and thus the actual velocity is less. Also, the fluid stream continues to contract past the obstruction, and the vena contracta area (see the fig below) is less than the flow area of the obstruction. Both losses can be accounted for by incorporating a correction factor called the **discharge coefficient** *Cd* whose value (which is less than 1) is determined experimentally. Then the flow rate for obstruction flowmeters is expressed as

  Obstruction flowmeters: $Q = A_2 \times Cd \times \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$

  Where A_2 the cross-sectional area of the throat or orifice

  The value of *Cd* depends on both β and the Reynolds number. Charts and correlations for *Cd* are available for various types of obstruction meters.

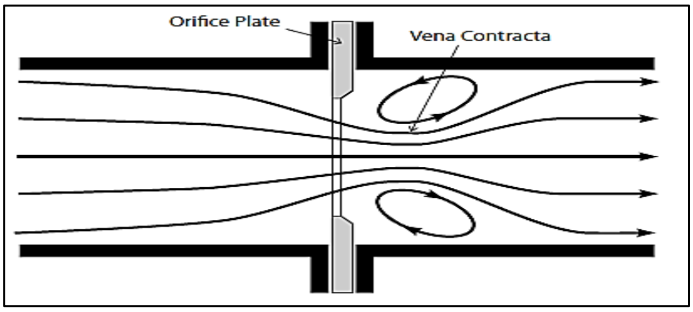
  Numerous types of obstruction meters are available; those most widely used are orifice meters, flow nozzles, and Venturi meters (Fig, next page).

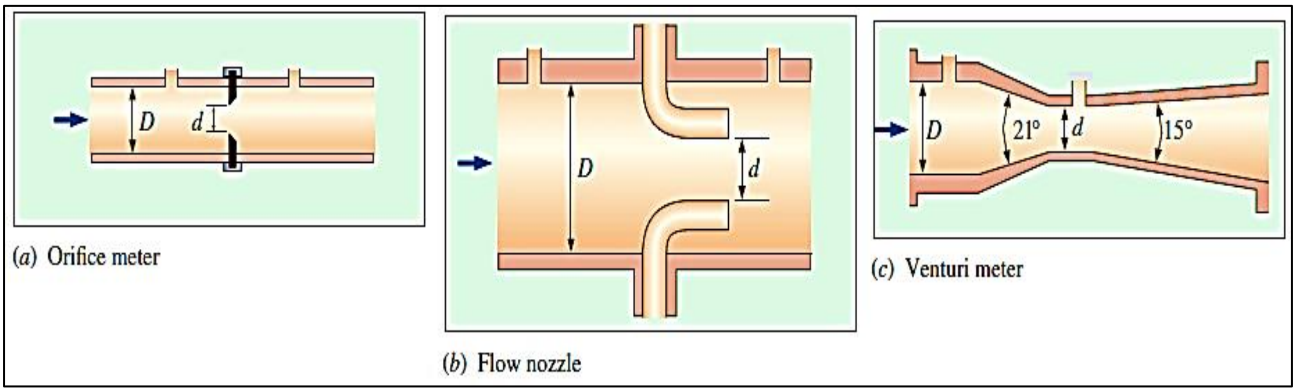
  Orifice meters: $C_d = 0.5959 + 0.0312\beta^{2.1} - 0.184\beta^8 + \frac{91.71\beta^{2.5}}{\mathrm{Re}^{0.75}}$

  Nozzle meters: $C_d = 0.9975 - \frac{6.53\beta^{0.5}}{\mathrm{Re}^{0.5}}$

  The Reynolds number depends on the flow velocity. Therefore, the solution is iterative in nature therefore the value of *Cd* can be taken to be 0.96 for flow nozzles and 0.61 for orifices.

  Owing to its streamlined design, the discharge coefficients of Venturi meters are very high, ranging between 0.95 and 0.99 in the absence of specific data, we can take *Cd* = 0.98 for Venturi meters.





(a) Orifice meter

(b) Flow nozzle

(c) Venturi meter

a. The orifice meter has the simplest design and it occupies minimal space as it consists of a plate with a hole in the middle, but the sudden change in the flow area in orifice meters causes considerable swirl and thus significant head loss or permanent pressure loss.
b. In nozzle meters, the plate is replaced by a nozzle, and thus the flow in the nozzle is streamlined. As a result, the vena contracta is practically eliminated and the head loss is smaller. However, flow nozzle meters are more expensive than orifice meters.
c. The Venturi meter, is the most accurate flowmeter in this group, but it is also the most expensive. Its gradual contraction and expansion prevent flow separation and swirling, and it suffers only frictional losses on the inner wall surfaces. Venturi meters cause very low head losses, and thus, they should be preferred for applications that cannot allow large pressure drops.

Venturimeter

A Venturimeter is a device used for measuring the rate of a flow of a fluid flowing through a pipe. It consists of three parts:

(i) A short converging part, (ii) Throat, and (iii) Diverging part. It is based on the Principle of Bernoulli's equation.

Consider a Venturimeter fitted in a horizontal pipe through which a fluid is flowing (say water), as shown in the fig.

Let

d_1 = diameter at inlet or at section (1)

p_1 = pressure at section (1)

u_1 = velocity of fluid at section (1)

a_1= area at section (1) $= \frac{\pi}{4} d_1^2$

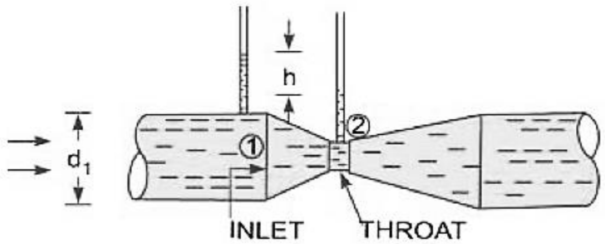


Fig. *Venturimeter.*

And d_2, P_2, u_2 and a_2 are corresponding values at section (2). Applying Bernoulli's equation at sections (1) and (2), we get

$$\frac{p_1}{w} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{w} + \frac{u_2^2}{2g} + z_2$$

As the pipe is horizontal, $z_1 = z_2$

$$\frac{p_1}{\rho g} + \frac{u_1^2}{2g} = \frac{p_2}{\rho g} + \frac{u_2^2}{2g}$$

Or

$$\frac{p_1 - p_2}{\rho g} = \frac{u_2^2}{2g} - \frac{u_1^2}{2g}$$

But $\frac{p_1 - p_2}{\rho g}$ is the difference of pressure heads at sections 1 and 2 and it is equal to Δh or

$$\frac{p_1 - p_2}{\rho g} = \Delta h$$

Substituting this in the above equation, we get

$$\Delta h = \frac{u_2^2}{2g} - \frac{u_1^2}{2g} \qquad (a)$$

Now applying continuity equation at sections 1 and 2

$$a_1 u_1 = a_2 u_2 \quad or \quad u_1 = \frac{a_2 u_2}{a_1}$$

Substituting this value of u_1 in equation (a)

$$\Delta h = \frac{u_2^2}{2g} - \frac{\left(\frac{a_2 u_2}{a_1}\right)^2}{2g} = \frac{u_2^2}{2g}\left[1 - \frac{a_2^2}{a_1^2}\right] = \frac{u_2^2}{2g}\left[\frac{a_1^2 - a_2^2}{a_1^2}\right]$$

$$u_2^2 = 2g\Delta h \frac{a_1^2}{a_1^2 - a_2^2}$$

$$u_2 = \sqrt{2g\Delta h \frac{a_1^2}{a_1^2 - a_2^2}} = \frac{a_1}{\sqrt{a_1^2 - a_2^2}}\sqrt{2g\Delta h}$$

$$Q = a_2 u_2$$

$$Q = \frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}}\sqrt{2g\Delta h}$$

The above Equation gives the discharge under ideal conditions and is called, theoretical discharge. Actual discharge will be less than theoretical discharge.

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

Where C_d = Co-efficient of venturimeter

Value of (Δh) given by differential U-tube manometer

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

S_h = Sp. gravity of the heavier liquid

S_o = Sp. gravity of the liquid flowing through pipe

x = Difference of the heavier liquid column in U-tube

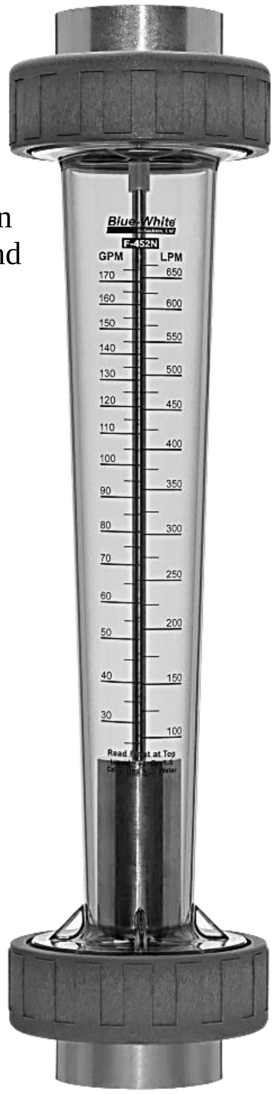
Then

$$\Delta h = x\left(\frac{S_h}{S_o} - 1\right)$$

Variable-Area Flowmeters (Rotameters)

A simple, reliable, inexpensive, and easy-to-install flowmeter with reasonably low pressure drop and no electrical connections that gives a direct reading of flow rate for a wide range of liquids and gases is the variable-area flowmeter, also called a rotameter or floatmeter. A variable-area flowmeter consists of a vertical tapered conical transparent tube made of glass or plastic with a float inside that is free to move, as shown in Fig.

We know from experience that high winds knock down trees, break power lines, and blow away hats or umbrellas. This is because the drag force increases with flow velocity. The weight and the buoyancy force acting on the float are constant, but the drag force changes with flow velocity. Also, the velocity along the tapered tube decreases in the flow direction because of the increase in the cross-sectional area. There is a certain velocity that generates enough drag to balance the float weight and the buoyancy force, and the location at which this velocity occurs around the float is the location where the float settles. The degree of tapering of the tube can be made such that the vertical rise changes linearly with flow rate, and thus the tube can be calibrated linearly for flow rates.



Design equation

$$Q = C_d\, A_{ann.} \left[2gV_{fl} (\rho_{fl} - \rho_f) / A_f \rho_f \right]^{1/2}$$

where,

Q = Volume flow rate,

C_d = Co-efficient of discharge,

$A_{ann.}$ = Annular area between float and tube,

V_{fl} = Volume of float,

ρ_{fl} = Density of float material,

ρ_f = Density of fluid, and

A_f = Maximum cross-sectional area of the fluid.

Paddlewheel flowmeter

- To measure liquid flow
- *Working*

  A sensor detects the passage of each of the paddlewheel blades and transmits a signal. A microprocessor then converts this rotational speed information to flow rate.

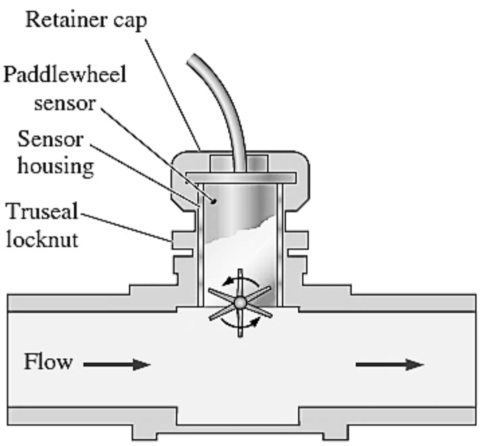
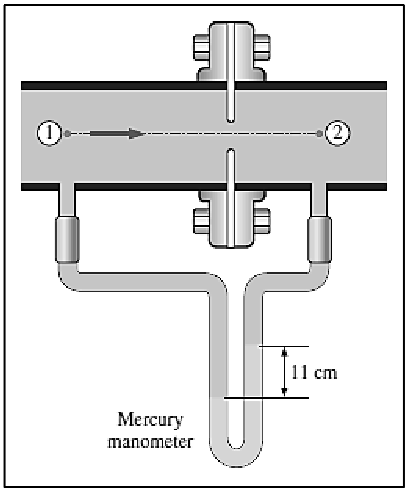


FIGURE Paddlewheel flowmeter

Example 1: A pilot-static tube is used to measure the velocity of water in a pipe. The stagnation pressure head is 6 m and static pressure head is 5 m. calculate the velocity of flow assuming the coefficient of tube equal to 0.98.

Example 2: A Pitot tube is placed at a center of a 30 cm I.D. pipe line has one orifice pointing upstream and other perpendicular to it. The mean velocity in the pipe is 0.84 of the center velocity. Find the discharge through the pipe if the fluid flow through the pipe is water and the pressure difference between orifices is 6 cm H_2O. Take Cp = 0.98.

Example 3: The flow rate of methanol at 20°C (ρ = 788.4 kg/m3 and $\mu = 5.857 \times 10^{-4}\ kg/m \cdot s$) through a 4-cm-diameter pipe is to be measured with a 3-cm diameter orifice meter equipped with a mercury manometer across the orifice plate, as shown in Fig. If the differential height of the manometer is 11 cm, determine the flow rate of methanol through the pipe and the average flow velocity. Take the discharge coefficient of the orifice meter Cd = 0.61.



Example 4: An oil of sp. gr. 0.8 is flowing through a venturimeter having inlet diameter 20 cm and throat diameter 10 cm. The oil-mercury differential manometer shows a reading of 25 cm. Calculate the discharge of oil through the horizontal venturimeter. Take C_d = 0.98.

Example 5: A horizontal venturimeter with inlet diameter 20 cm and throat diameter 10 cm is used to measure the flow of oil of sp. gr. 0.8. The discharge of oil through venturimeter is 60 Liters/s. Find the reading of the oil-mercury differential manometer. Take C_d = 0.98.

CHAPTER SIX
PUMPS

Introduction

Pumps are devices for supplying energy or head to a flowing liquid in order to overcome head losses due to friction or to raise liquid to a higher level. The energy required by the pump will depend on the height through which the fluid is raised, the pressure required at delivery point, the length and diameter of the pipe, the rate of flow, together with the physical properties of the fluid, particularly its viscosity and density.

The pumping of liquids such as sulphuric acid or petroleum products from bulk store to process buildings, or the pumping of fluids to the reaction units and through heat exchangers, are examples for the use of pumps in the process industries.

Fundamental parameters are used to analyze the performance of a pump:

1- The mass flow rate of fluid through the pump, for incompressible flow, it is more common to use volume flow rate rather than mass flow rate.
2- The performance of a pump is characterized by its net head H, defined as the change in Bernoulli head between the inlet and outlet of the pump

$$\frac{p_{in}}{\rho g} + \frac{u_{in}^2}{2g} + z_{in} + H_{pump} = \frac{p_{out}}{\rho g} + \frac{u_{out}^2}{2g} + z_{out}$$

$$H_{pump} = \left(\frac{p}{\rho g} + \frac{u^2}{2g} + z\right)_{out} - \left(\frac{p}{\rho g} + \frac{u^2}{2g} + z\right)_{in} \qquad 1$$

The dimension of net head is length, and it is often listed as an equivalent column height of water, even for a pump that is not pumping water.

By dimensional reasoning, we must multiply the net head of Eq. 1 by mass flow rate and gravitational acceleration to obtain dimensions of power (W). Thus,

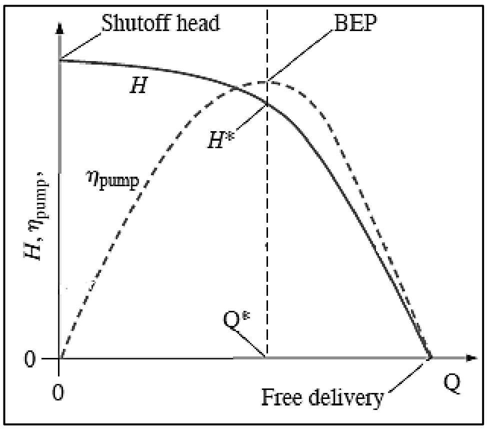
$$\dot{W}_{pump} = \dot{m}gH = Q\rho gH$$

We define pump efficiency h_{pump} as the ratio of useful power to supplied power;

$$\eta_{pump} = \frac{Mech.\,energy\ increase\ of\ the\ fluid(\dot{W}_{pump})}{\dot{W}_{supplied}} = \frac{\rho gHQ}{\dot{W}_{supplied}} \qquad 2$$

Pump Performance Curves and Matching a Pump to a Piping System

The maximum volume flow rate through a pump occurs when it's net head is zero, H = 0; this flow rate is called the pump's **free delivery**. The free delivery condition is achieved when there is no flow restriction at the pump inlet or outlet, in other words when there is no load on the pump. At this operating point, Q is large, but H is zero; the pump's efficiency is zero because the pump is doing no useful work, as is clear from Eq. 2. At the other extreme, the **shutoff head** is the net head that occurs when the volume flow rate is zero, Q = 0, and is achieved when the outlet port of the pump is blocked off. Under these conditions, H is large but Q is zero; the pump's efficiency (Eq. 2) is again zero, because the pump is doing no useful work. Between these two extremes, from shutoff to free delivery, the pump's net head is occurring. The pump's efficiency reaches its maximum value somewhere between the shutoff condition and the free delivery condition; this operating point of maximum efficiency is appropriately called the best efficiency point (BEP), and is notated by an asterisk (H*, Q*). Curves of H and η_{pump}, as functions of Q are called pump performance curves.

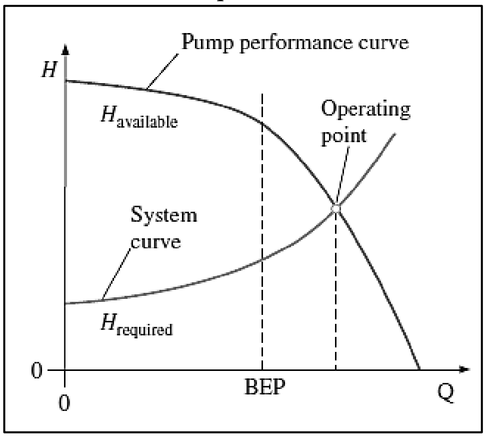


It is important to realize that for steady conditions, a pump can operate only along its performance curve. Thus, the operating point of a piping system is determined by matching system requirements (required net head) to pump performance (available net head). In a typical application, $H_{required}$ and $H_{available}$ match at one unique value of flow rate, this is the operating point or duty point of the system.

The volume flow rate of a piping system is established where $H_{required} = H_{available}$.

For a given piping system with its major and minor losses, elevation changes, etc., the required net head increases with volume flow rate.

On the other hand, the available net head of most pumps decreases with flow rate, as in Fig.



Hence, the system curve and the pump performance curve intersect as sketched in the Fig. above, and this establishes the operating point. If we are lucky, the operating point is at or near the best efficiency point of the pump.

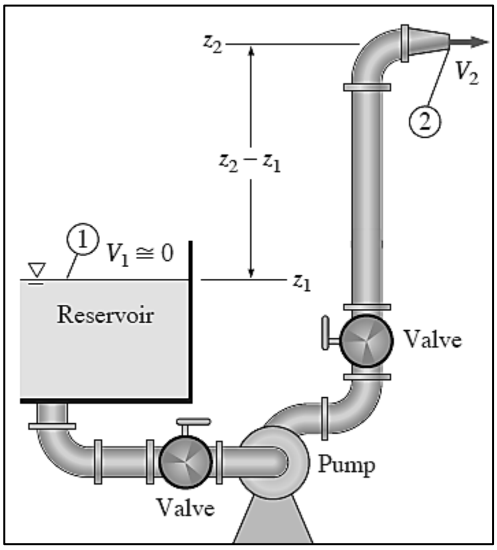
In most cases, however, as illustrated in Fig, the pump does not run at its optimum efficiency. If efficiency is of major concern, the pump should be carefully selected (or a new pump should be designed) such that the operating point is as close to the best efficiency point as possible.

For a general piping system with elevation change, major and minor losses, and fluid acceleration, we begin by solving the energy equation for the required net head $H_{required}$

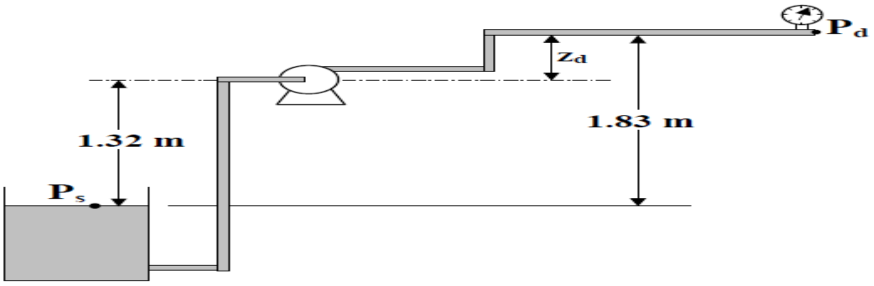
$$H_{required} = \frac{p_2 - p_1}{\rho g} + \frac{u_2^2 - u_1^2}{2g} + (z_2 - z_1) + h_{L,total}$$

*When the fluid is a gas, such as in ventilation and air pollution control problems, the elevation term is almost always negligible.

The most common situation is that an engineer selects a pump that is somewhat higher than actually required. The volume flow rate through the piping system is then a bit larger than needed, and a valve is installed in the line so that the flow rate can be decreased as necessary.



Example 1: A petroleum product is pumped at a rate of 2.525 x 10^{-3} m^3/s from a reservoir under atmospheric pressure to 1.83 m height. If the pump 1.32 m height from the reservoir, the discharge line diameter is 4 cm and the pressure drop along its length 3.45 kPa. The gauge pressure reading at the end of the discharge line 345 kPa. The pressure drop along suction line is 3.45 kPa calculate:-
(i) The required net head of the system **(ii)** The required power of the system **(iii)** The NPSH
Take that: the density of this petroleum product ρ=879 kg/m^3, the dynamic viscosity μ=6.47 x 10^{-4} Pa.s, and the vapor pressure P*v*= 24.15 kPa.



Solution:-

$(\boldsymbol{i})$ *The required net head of the system*

$$\Delta h = \Delta Z + \frac{\Delta P}{\rho g} + \frac{\Delta u^2}{2g} + h_{L\,total}$$

The total elevation=

$$\Delta Z = 1.83\ m$$

The pressure head=

$$\frac{\Delta P}{\rho g} = \frac{345 \times 10^3}{879 \times 9.81} = 40\ m$$

The velocity head=

$$u_1 = 0$$

$$u_2 = \frac{Q}{A} = \frac{2.525 \times 10^{-3}}{\frac{\pi}{4}(0.04)^2} = 2\ \frac{m}{s}$$

So

$$\frac{\Delta u^2}{2g} = \frac{2^2 - 0}{2 \times 9.81} = 0.2\ m$$

The losses head= $h_{L(suction\ side)} + h_{L(discharge\ side)}$

$$= \frac{3.45 \times 10^3 + 3.45 \times 10^3}{879 \times 9.81} = 0.8\ m$$

$The\ required\ net\ head\ of\ the\ system \rightarrow \Delta h = 1.83 + 40 + 0.2 + 0.8 = 42.83\ m$

$(\boldsymbol{ii})\ power\ required = \dot{m} \times g \times h_{requied} = \rho \times Q \times g \times h_{requied}$

$$= 879 \times 2.525 \times 10^{-3} \times 9.81 \times 42.83 = 932.54\ W$$

$(\boldsymbol{iii})see\ page\ (\ 5)$

Example 2: A centrifugal pump used to take water from reservoir to another through 800 m length and 0.15 m id pipe. If the difference in two tank is 8 m, calculate the flow rate of the water and the power required, assume *f =0.016.* If the available pump characteristic is

Q (m^3/h)	0	23	46	69	92	115
Δh (m)	17	16	13.5	10.5	6.6	2.0
η	0	0.495	0.61	0.63	0.53	0.1

Solution:-

$(\boldsymbol{i})\ The\ required\ net\ head\ of\ the\ system$

$$\Delta h = \Delta Z + \frac{\Delta P}{\rho g} + \frac{\Delta u^2}{2g} + h_{L\ total}$$

$$h_{L\ total} = f\frac{L}{d}\frac{u^2}{2g}$$

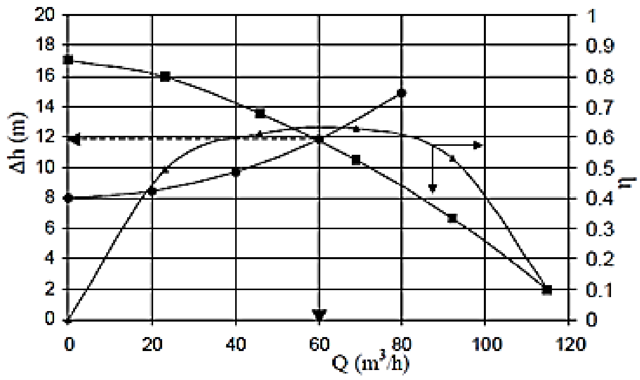
$$u = \frac{Q}{A} = \frac{Q}{\frac{\pi}{4}0.15^2} = 56.6 \times Q \xrightarrow{(if\ Q\ in\ m^3/h)} \frac{56.6 \times Q}{3600}$$

$$h_{L\ total} = 0.016 \times \frac{800}{0.15} \times \frac{\left(\frac{56.6Q}{3600}\right)^2}{2 \times 9.81} = 1.075 \times 10^{-3} \times Q^2$$

$$\Delta h = 8 + 1.075 \times 10^{-3} \times Q^2$$

So the system curve

Q (m^3/h)	0	20	40	60	80
Δh (m)	8	8.43	9.72	11.87	14.88



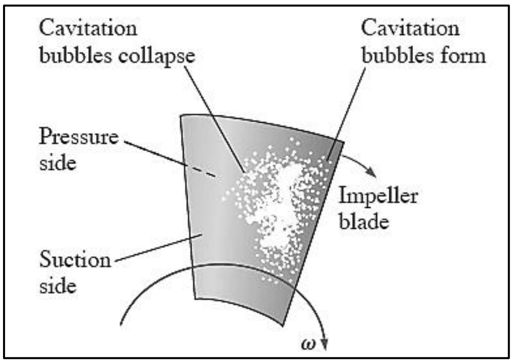
From Figure
Q = 60 m^3/h
Δh = 11.8 m
η = 0.64

Power required for pump = $\frac{Q\Delta h\ \rho\ g}{\eta}$ = (60)(1 h/3600 s)(11.8)(1000)(9.81)/0.64

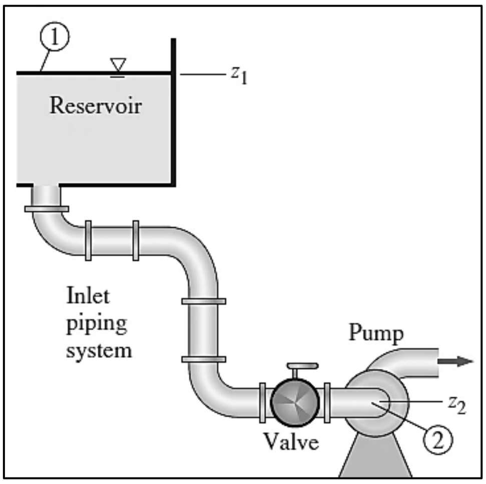
= 3.014 kW

Pump Cavitation and Net Positive Suction Head

When pumping liquids, it is possible for the local pressure inside the pump to fall below the vapor pressure of the liquid, P_v. (P_v is also called the saturation pressure P_{sat}). When $P < P_v$, vapor-filled bubbles called cavitation bubbles appear. In other words, the liquid boils locally, typically on the suction side of the rotating impeller blades where the pressure is lowest. After the cavitation bubbles are formed, they are transported through the pump to regions where the pressure is higher, causing rapid collapse of the bubbles. It is this collapse of the bubbles that is undesirable, since it causes noise, vibration, reduced efficiency, and most importantly, damage to the impeller blades. Repeated bubble collapse near a blade surface leads to pitting or erosion of the blade and cause blade failure.



To avoid cavitation, we must ensure that the local pressure everywhere inside the pump stays above the vapor pressure. Since pressure is most easily measured (or estimated) at the inlet of the pump see the Fig, cavitation criteria are typically specified at the pump inlet. It is useful to employ a flow parameter called net positive suction head (NPSH), defined as the difference between the pump's inlet stagnation pressure head and the vapor pressure head,



$$NPSH = (\text{stagnation pressure})_{pump\ inlet} - \frac{p_v}{\rho g}$$

$$NPSH = \left(\frac{p}{\rho g} + \frac{u^2}{2g}\right)_{pump\ inlet} - \frac{p_v}{\rho g}$$

Pump manufacturers test their pumps for cavitation in a pump test facility, the pump manufacturer then publishes a performance parameter called the required net positive suction head ($NPSH_{required}$), defined as the minimum NPSH necessary to avoid cavitation in the pump. The measured value of $NPSH_{required}$ varies with volume flow rate.

In order to ensure that a pump does not cavitate, the actual or NPSH must be greater than $NPSH_{required}$. It is important to note that the value of NPSH varies not only with flow rate, but also with liquid temperature, since P_v is a function of temperature. NPSH also depends on the type of liquid being pumped, since there is a unique P_v versus T curve for each liquid.

$\boldsymbol{Example\ 1/(iii)}$:

$\Longrightarrow$

Applying Bernoulli equation between (suction point) and the (pump inlet point) we get:

$$\frac{p_{suction\ P.}}{\rho g} + \frac{u^2_{suction\ P.}}{2g} + z_{suction\ P.} = \frac{p_{pump\ inlet}}{\rho g} + \frac{u^2_{pump\ inlet}}{2g} + z_{pump\ inlet} + h_L$$

$$(stagnation\ pressure)_{pump\ inlet} = \left(\frac{p_{pump\ inlet}}{\rho g} + \frac{u^2_{pump\ inlet}}{2g}\right)$$

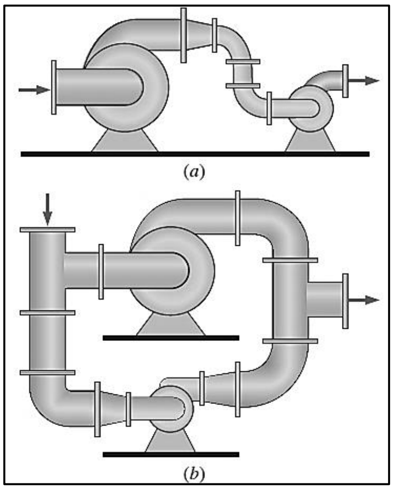
$$= \frac{p_{suction\ P.}}{\rho g} + \frac{u^2_{suction\ P.}}{2g} + \left(z_{suction\ P.} - z_{pump\ inlet}\right) - h_L$$

$$NPSH = (stagnation\ pressure)_{pump\ inlet} - \frac{p_v}{\rho g}$$

$$NPSH = \left(\frac{p_{suction\ P.}}{\rho g} + \frac{u^2_{suction\ P.}}{2g} + \left(z_{suction\ P.} - z_{pump\ inlet}\right) - h_L\right) - \frac{p_v}{\rho g}$$

$$NPSH = \left(\frac{101325}{879 \times 9.81} + 0 + (0 - 1.32) - 0.4\right) - \frac{24150}{879 \times 9.81} = 7.32\ m$$

Pumps in Series and Parallel



When faced with the need to increase volume flow rate or pressure rise by a small amount, you might consider adding an additional smaller pump in series or in parallel with the original pump. While series or parallel arrangement is acceptable for some applications, arranging dissimilar pumps in series or in parallel may lead to problems, especially if one pump is much larger than the other (Fig).

A better course of action is to increase the original pump's speed and/or input power (larger electric motor), replace the impeller with a larger one, or replace the entire pump with a larger one.

Arranging dissimilar pumps in series Fig (a) may create problems because the volume flow rate through each pump must be the same, but the overall pressure rise is equal to the pressure rise of one pump plus that of the other. If the pumps have widely different performance curves, the smaller pump may be forced to operate beyond its free delivery flow rate, whereupon it acts like a head loss, reducing the total volume flow rate.

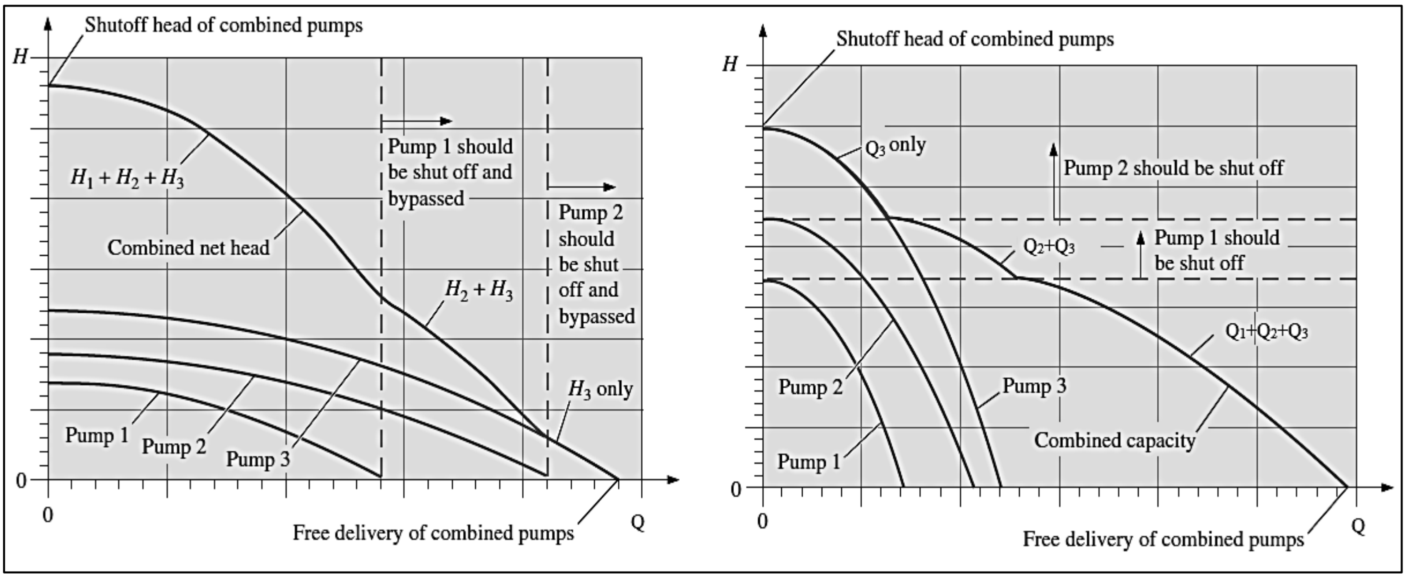
When operated in series, the combined net head is simply the sum of the net heads of each pump (at a given volume flow rate):

$$\boldsymbol{Combined\ net\ head\ for\ n\ pumps\ in\ series}: \qquad H_{combined} = \sum_{i=1}^{n} H_i$$

Arranging dissimilar pumps in parallel Fig (b) may create problems because the overall pressure rise must be the same, but the net volume flow rate is the sum of that through each branch. If the pumps are not sized properly, the smaller pump may not be able to handle the large head imposed on it, and the flow in its branch could actually be reversed; this would inadvertently reduce the overall pressure rise. In either case, the power supplied to the smaller pump would be wasted.

When two or more identical (or similar) pumps are operated in parallel, their individual volume flow rates (rather than net heads) are summed

$$\boldsymbol{Combined\ capacity\ for\ n\ pumps\ in\ parallel}: \qquad Q_{combined} = \sum_{i=1}^{n} Q_i$$



Types of Pumps

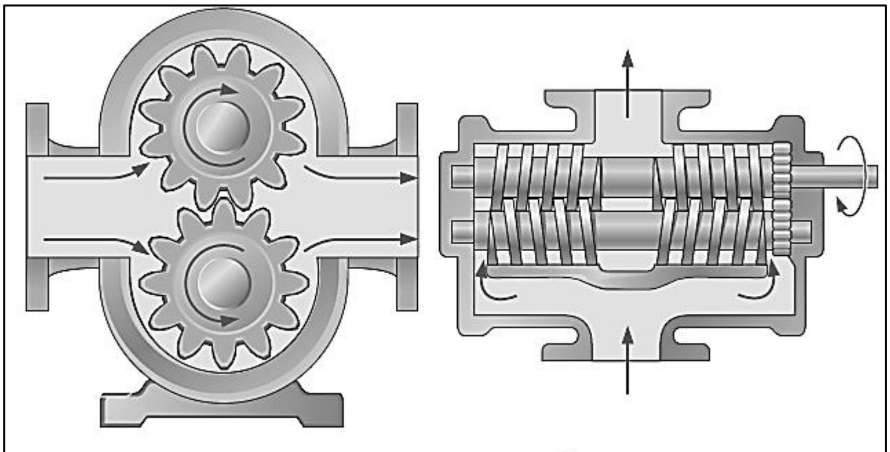
Pumps can be classified into: -

1- Positive displacement pumps. 2- Dynamic Pumps

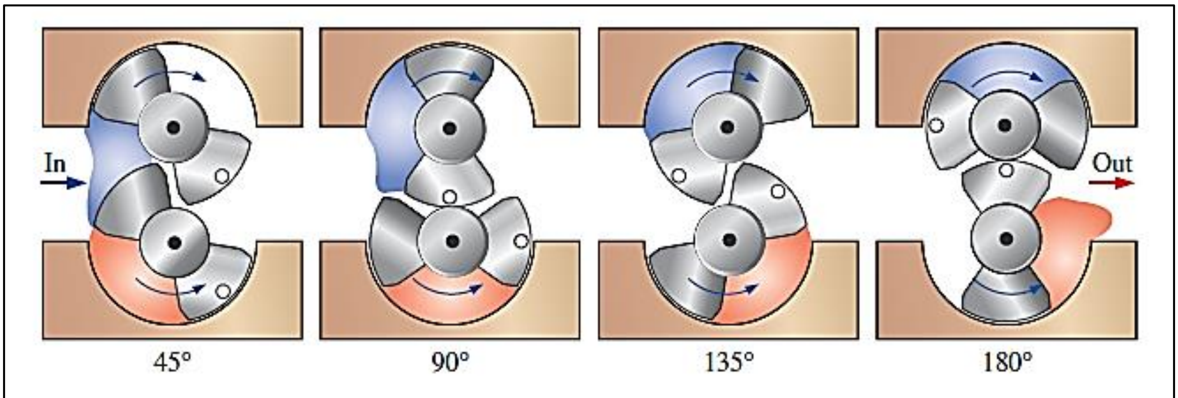
1-Positive-Displacement Pumps

People have designed numerous positive-displacement pumps throughout the centuries, like:

1- The piston Pump
2- The Gear Pump
3- The Cam Pump
4- The Screw pumps
5- Rotary pump



Gear pump **Screw pump**



Two-lobe rotary pump

In each design, fluid is sucked into an expanding volume and then pushed along as that volume contracts, but the mechanism that causes this change in volume differs greatly among the various designs.
These pumps are ideal for high-pressure applications like pumping viscous liquids or thick slurries, and for applications where precise amounts of liquid are to be dispensed or metered, as in medical applications.
To illustrate the operation of a positive-displacement pump, the sketch four phases for half cycle of a simple rotary pump with two lobes (see the Fig. above).
Gaps exist between the rotors and the housing and between the lobes of the rotors themselves, as illustrated in Fig of rotary pump.
Fluid can leak through these gaps, reducing the pump’s efficiency. High viscosity fluids cannot penetrate the gaps as easily; hence the net head (and efficiency) of a rotary pump generally increases with fluid viscosity. This is one reason why rotary pumps (and other types of positive-displacement pumps) are a good choice for pumping highly viscous fluids and slurries. They are used, for example, as automobile engine oil pumps and in the foods industry to pump heavy liquids like syrup, tomato paste, and chocolate, and slurries like soups.

Positive-displacement pumps have many advantages over dynamic pumps.

For example:

1- A positive-displacement pump is better in handling shear sensitive liquids since the induced shear is much less than that of a dynamic pumps operating at similar pressure and flow rate. Blood is a shear sensitive liquid, and this is one reason why positive-displacement pumps are used for artificial hearts.
2- A well-sealed positive-displacement pump can create a significant vacuum pressure at its inlet, even when dry, and is thus able to lift a liquid from several meters below the pump. We refer to this kind of pump as a self-priming pump
3- The rotor(s) of a positive- displacement pump run at lower speeds than the rotor (impeller) of a dynamic pump at similar loads, extending the useful lifetime of seals, etc.

There are some disadvantages of positive-displacement pumps

1- Their volume flow rate cannot be changed unless the rotation rate is changed. (Since most AC electric motors are designed to operate at one or more fixed rotational speeds.)
2- They create very high pressure at the outlet side, and if the outlet becomes blocked, ruptures may occur or electric motors may overheat. Overpressure protection (e.g., a pressure-relief valve) is often required for this reason.
3- Because of their design, positive-displacement pumps sometimes deliver a pulsating flow, which may be unacceptable for some applications.

2-Dynamic Pumps

There are two main types of dynamic pumps that involve rotating blades called impeller blades or rotor blades, which impart momentum to the fluid. They are classified by the manner in which flow exits the pump: centrifugal flow and axial flow. In a **centrifugal-flow pump**, fluid enters axially (in the same direction as the axis of the rotating shaft) in the center of the pump, but is discharged radially (or tangentially) along the outer radius of the pump casing. For this reason centrifugal pumps are also called radial-flow pumps. In an **axial-flow pump**, fluid enters and leaves axially, typically along the outer portion of the pump. Here we will focus on the centrifugal pumps as they most common type

Centrifugal Pumps

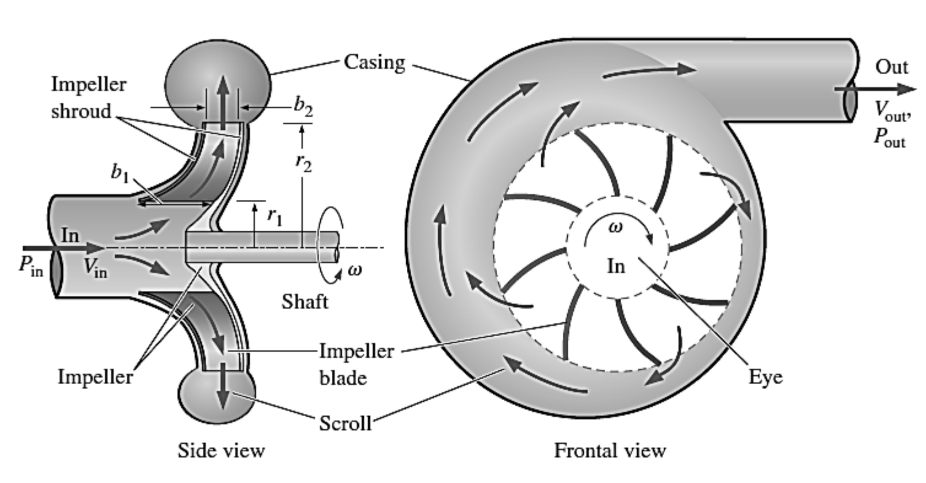
Axial Pump

Centrifugal Pumps

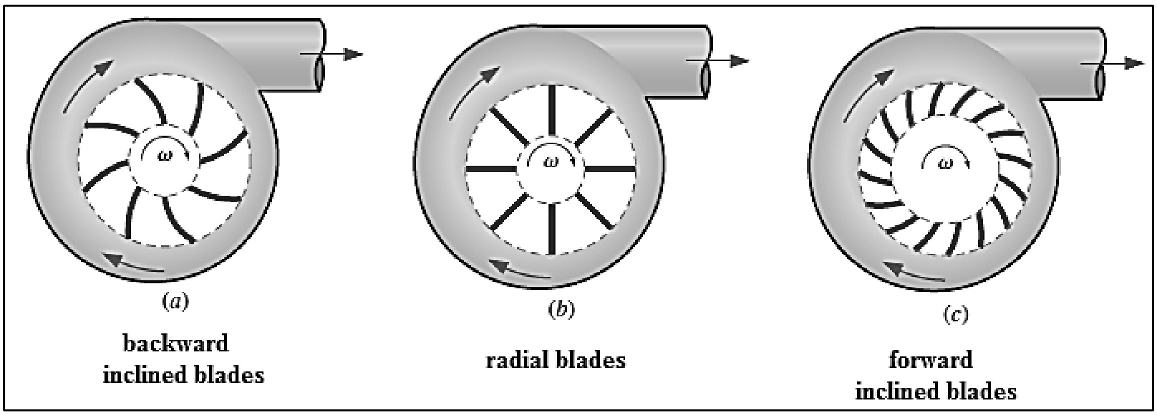
Centrifugal pumps and blowers can be easily identified by their snail-shaped casing, called the scroll (Fig). They are found all around your home—in clothes washers and dryers, vacuum cleaners, leaf blowers, furnaces, etc. They are used in cars—the water pump in the engine, the air blower in the heater/air conditioner unit, etc. Centrifugal pumps are used in industry as well; they are used in building ventilation systems, washing operations, cooling ponds and cooling towers, and in numerous other industrial operations in which fluids are pumped.

A schematic diagram of a centrifugal pump is shown in Fig. Note that a shroud often surrounds the impeller blades. Fluid enters axially through the hollow middle portion of the pump (the eye), after which it reach the rotating blades. It acquires tangential and radial velocity by momentum transfer with the impeller blades, by so-called centrifugal forces.

The flow leaves the impeller after gaining both speed and pressure as it is flung radially outward into the scroll (also called the volute). As sketched in Fig, there by further increasing the fluid's pressure, and to combine and direct the flow from all the blade passages toward a common outlet. As mentioned previously, if the flow is steady in the mean, if the fluid is incompressible, and if the inlet and outlet diameters are the same, the average flow speed at the outlet is identical to that at the inlet. Thus, it is not necessarily the speed, but the pressure that increases from inlet to outlet through a centrifugal pump.



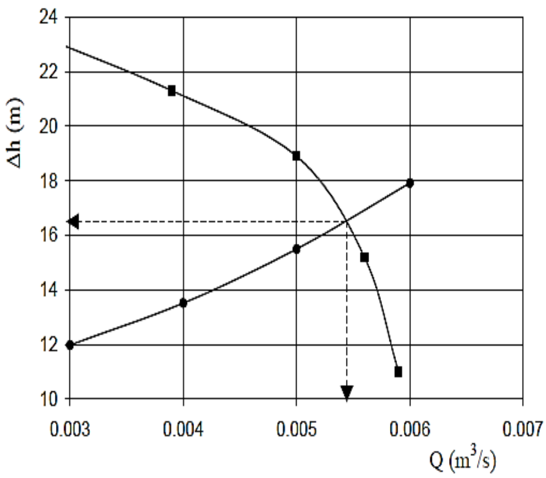
There are three types of centrifugal pump, based on impeller blade geometry, as sketched in Fig: **a) backward-inclined blades b) radial blades c) forward-inclined blades.** Centrifugal pumps with backward-inclined blades (Fig. a) are the most common. These yield the highest efficiency of the three because fluid flows into and out of the blade passages with the least amount of turning.



Exercise1: It is required to pump cooling water from storage pond to a condenser in a process plant situated 10 m above the level of the pond. (200 m of 74.2 mm id) pipe in between and the pump has the characteristics given below. The head loss in the condenser is equivalent to 16 velocity heads based on the flow in the 74.2 mm pipe. If the friction factor = 0.024, estimate the rate of flow and the power to be supplied to the pump assuming $\eta = 0.5$

Q (m^3/s)	0.0028	0.0039	0.005	0.0056	0.0059
Δh (m)	23.2	21.3	18.9	15.2	11.0

Hint:



CHAPTER SEVEN
COMPRESSIBLE FLOW

Introduction

We have limited our consideration so far to flows for which density variations and thus compressibility effects are negligible. In this chapter, we lift this limitation and consider flows that involve significant changes in density. Such flows are called *compressible flows,* and they are frequently encountered in devices that involve the flow of gases at very high speeds. Compressible flow combines fluid dynamics and thermodynamics in that both are necessary to the development of the required theoretical background.

Stagnation properties

Enthalpy of a fluid defined per unit mass as $h = u + Pv$. Whenever the kinetic and potential energies of the fluid are negligible, as is often the case, the enthalpy represents the total energy of a fluid. For high-speed flows, the potential energy of the fluid is still negligible, but the kinetic energy is not. In such cases, it is convenient to combine the enthalpy and the kinetic energy of the fluid into a single term called stagnation (or total) enthalpy h_o, defined per unit mass as

$$h_o = h + \frac{V^2}{2} \qquad 1$$

Where h is the static enthalpy

Notice that the two enthalpies are identical when the kinetic energy of the fluid is negligible.

Consider the steady flow of a fluid through a duct such as a nozzle, diffuser, or some other flow passage where the flow takes place adiabatically and with no shaft or electrical work. Assuming the fluid experiences little or no change in its elevation and its potential energy, the energy balance relation ($E_{in} = E_{out}$) for this single-stream steady-flow device reduces to

$$h_1 + \frac{{V_1}^2}{2} = h_2 + \frac{{V_2}^2}{2} \qquad 2$$

$$h_{o1} = h_{o2}$$

Any increase in fluid velocity in these flow devices creates an equivalent decrease in the static enthalpy of the fluid.

If the fluid were brought to a complete stop, then the velocity at state 2 would be zero and Eq. 2 would become

$$h_1 + \frac{{V_1}^2}{2} = h_2 = h_{o2}$$

During a stagnation process, the kinetic energy of a fluid is converted to enthalpy which results in an increase in the fluid temperature and pressure. The properties of a fluid at the stagnation state are called stagnation properties (stagnation temperature, stagnation pressure, stagnation density, etc.). The stagnation state and the stagnation properties are indicated by the subscript o.

When the fluid is approximated as an ideal gas with constant specific heats, its enthalpy can be replaced by c_pT and Eq. 1 is expressed as

$$c_pT_o = c_pT + \frac{V^2}{2}$$

$$T_o = T + \frac{V^2}{2c_p}$$

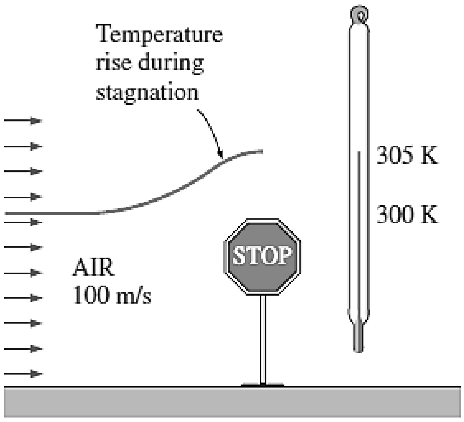
Here, T_o is called the stagnation (or total) temperature, and it represents the temperature of an ideal gas attains when it is brought to rest adiabatically.
The term $V^2/2c_p$ corresponds to the temperature rise during such a process and is called the dynamic temperature.

For example

the dynamic temperature of air flowing at 100 m/s is

$$\frac{(100\, m/s)^2}{(2\, \times\, 1005\, J/kg\cdot K)} = \, 5\, K$$

Therefore, when air at 300 K and 100 m/s is brought to rest adiabatically its temperature rises to the stagnation value of 305 K (Fig).



Note that for low-speed flows, the stagnation and static temperatures are practically the same.
For ideal gases with constant specific heats, P_o is related to the static pressure of the fluid by:

$$\frac{P_o}{P} = \left(\frac{T_o}{T}\right)^{k/(k-1)}$$

The ratio of the stagnation density to static density is expressed as:

$$\frac{\rho_o}{\rho} = \left(\frac{T_o}{T}\right)^{1/(k-1)}$$

Example: air at a speed of 250 m/s and pressure of 54.05 kPa and its temperature is 255.7 K. determine: the stagnation pressure
Sol:

$$T_o = T + \frac{V^2}{2c_p}$$

$$T_o = 255.7 + \frac{250^2}{2 \times 1005} = 286.8\, K$$

$$\frac{P_o}{P} = \left(\frac{T_o}{T}\right)^{k/(k-1)}$$

$$\frac{P_o}{54.05} = \left(\frac{286.8}{255.7}\right)^{1.4/(1.4-1)}$$

$$P_o = 80.77\, kPa$$

Speed of sound and Mach number

An important parameter in the study of compressible flow is the speed of sound c, (or the sonic speed), defined as the speed at which an infinitesimally small pressure wave travels through a medium. The pressure wave may be caused by a small disturbance, which creates a slight rise in local pressure.

Which is related to other fluid properties as:

$$c = \sqrt{\frac{dP}{d\rho}}$$

$$c = \sqrt{kRT}$$

Where k is the specific heat ratio of the gas $k = \frac{c_p}{c_v}$, and R is the specific gas constant $R = c_p - c_v$.

A second important parameter in the analysis of compressible fluid flow is the Mach number Ma. It is the ratio of the actual speed of the fluid (or an object in still fluid) to the speed of sound in the same fluid at the same state:

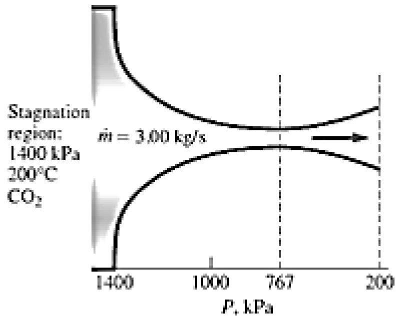
$$Ma = \frac{V}{c}$$

Note that the Mach number depends on the speed of sound, which depends on the state of the fluid.

Fluid flow regimes are often described in terms of the flow Mach number.

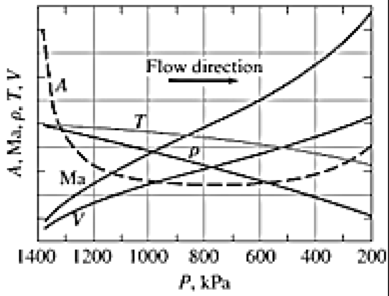
The flow is called sonic when Ma = 1, subsonic when Ma < 1, supersonic when Ma > 1 and hypersonic when Ma >> 1

EXAMPLE **Gas Flow through a Converging–Diverging Duct**



Carbon dioxide flows steadily through a varying cross-sectional area duct such as a nozzle shown in Fig. at a mass flow rate of 3.00 kg/s. The carbon dioxide enters the duct at a pressure of 1400 kPa and 200°C with a low velocity, and it expands in the nozzle to an exit pressure of 200 kPa. The duct is designed so that the flow can be approximated as isentropic. Determine the density, velocity, flow area, and Mach number at each location along the duct that corresponds to an overall pressure drop of 200 kPa.

SOLUTION Carbon dioxide enters a varying cross-sectional area duct at specified conditions. The flow properties are to be determined along the duct.



Assumptions 1 Carbon dioxide is an ideal gas with constant specific heats at room temperature. 2 Flow through the duct is steady, one-dimensional, and isentropic.

Properties For simplicity we use $c_p = 0.846$ kJ/kg·K and $k = 1.289$ throughout the calculations, which are the constant-pressure specific heat and specific heat ratio values of carbon dioxide at room temperature. The gas constant of carbon dioxide is $R = 0.1889$ kJ/kg·K.

Analysis We note that the inlet temperature is nearly equal to the stagnation temperature since the inlet velocity is small. The flow is isentropic, and thus the stagnation temperature and pressure throughout the duct remain constant. Therefore,

$$T_0 \cong T_1 = 200°\text{C} = 473\text{ K}$$

and

$$P_0 \cong P_1 = 1400\text{ kPa}$$

To illustrate the solution procedure, we calculate the desired properties at the location where the pressure is 1200 kPa, the first location that corresponds to a pressure drop of 200 kPa.

$$T = T_0\left(\frac{P}{P_0}\right)^{(k-1)/k} = (473\text{ K})\left(\frac{1200\text{ kPa}}{1400\text{ kPa}}\right)^{(1.289-1)/1.289} = 457\text{ K}$$

$$\begin{aligned} V &= \sqrt{2c_p(T_0 - T)} \\ &= \sqrt{2(0.846\text{ kJ/kg·K})(473\text{ K} - 457\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} \\ &= 164.5\text{ m/s} \end{aligned}$$

From the ideal-gas relation,

$$\rho = \frac{P}{RT} = \frac{1200\text{ kPa}}{(0.1889\text{ kPa·m}^3/\text{kg·K})(457\text{ K})} = 13.9\text{ kg/m}^3$$

From the mass flow rate relation,

$$A = \frac{\dot{m}}{\rho V} = \frac{3.00\text{ kg/s}}{(13.9\text{ kg/m}^3)(161.5\text{ m/s})} = 13.1 \times 10^{-4}\text{ m}^2 = 13.1\text{ cm}^2$$

From Eqs.

$$c = \sqrt{kRT} = \sqrt{(1.289)(0.1889\text{ kJ/kg·K})(457\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 333.6\text{ m/s}$$

$$\text{Ma} = \frac{V}{c} = \frac{161.5\text{ m/s}}{333.6\text{ m/s}} = 0.493$$

The results for the other pressure steps are summarized in Table –1 and are plotted in Fig.

Discussion Note that as the pressure decreases, the temperature and speed of sound decrease while the fluid velocity and Mach number increase in the flow direction. The density decreases slowly at first and rapidly later as the fluid velocity increases.

TABLE –1

Variation of fluid properties in flow direction in the duct described in Example 12–2 for $\dot{m}$ = 3 kg/s = constant

P, kPa	*T*, K	*V*, m/s	*ρ*, kg/m^3	*c*, m/s	*A*, cm^2	Ma
1400	473	0	15.7	339.4	∞	0
1200	457	164.5	13.9	333.6	13.1	0.493
1000	439	240.7	12.1	326.9	10.3	0.736
800	417	306.6	10.1	318.8	9.64	0.962
767*	413	317.2	9.82	317.2	9.63	1.000
600	391	371.4	8.12	308.7	10.0	1.203
400	357	441.9	5.93	295.0	11.5	1.498
200	306	530.9	3.46	272.9	16.3	1.946

We note from the above Example that the flow area decreases with decreasing pressure down to a critical-pressure value where the Mach number is unity, and then it begins to increase with further reductions in pressure. The Mach number is unity at the location of smallest flow area, called the throat.
Note that the velocity of the fluid keeps increasing after passing the throat although the flow area increases rapidly in that region. This increase in velocity past the throat is due to the rapid decrease in the fluid density.
The flow area of the duct considered in this example first decreases and then increases. Such ducts are called converging–diverging nozzles.
These nozzles are used to accelerate gases to supersonic speeds

Variation of Fluid Velocity with Flow Area

It is clear from above Example that the couplings among the velocity, density, and flow areas for isentropic duct flow are rather complex.
We begin our investigation by seeking relationships among the pressure, temperature, density, velocity, flow area, and Mach number for one-dimensional isentropic flow. Consider the mass balance for a steady-flow process:

$$\dot{m} = \rho A V = constant$$

Differentiating and dividing the resultant equation by the mass flow rate, we obtain

$$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dV}{V} = 0 \qquad a$$

From the energy balance

$$h_1 + \frac{{V_1}^2}{2} = h_2 + \frac{{V_2}^2}{2}$$

So

$$h + \frac{V^2}{2} = constant$$

Differentiate

$$dh + VdV = 0 \qquad b$$

Solving b for $\frac{dV}{V}$:

$$\frac{dV}{V} = -\frac{dP}{\rho V^2}$$

Substitute b in a we get

$$\frac{d\rho}{\rho} + \frac{dA}{A} - \frac{dP}{\rho V^2} = 0$$

$$\frac{dA}{A} = \frac{dP}{\rho V^2} - \frac{d\rho}{\rho}$$

$$\frac{dA}{A} = \frac{dP}{\rho}\left(\frac{1}{V^2} - \frac{d\rho}{dP}\right) \qquad c$$

Where: $\frac{d\rho}{dP} = \frac{1}{c^2}$

$$\frac{dA}{A} = \frac{dP}{\rho V^2}(1 - Ma^2) \qquad d$$

This is an important relation for isentropic flow in ducts since it describes the variation of pressure with flow area. We note that A, ρ, and V are positive quantities.
For subsonic flow (Ma< 1), the term $(1 - Ma^2)$ is positive; and thus dA and dP must have the same sign. That is, the pressure of the fluid must increase as the flow area of the duct increases and must decrease as the flow area of the duct decreases. Thus, at

subsonic velocities, the pressure decreases in converging ducts (subsonic nozzles). And increases in diverging ducts (subsonic diffusers)
In supersonic flow (Ma > 1), the term $(1 - Ma^2)$ is negative, and thus dA and dP must have opposite signs. That is, the pressure of the fluid must decrease as the flow area of the duct increases. Thus, at supersonic velocities, the pressure decreases in diverging ducts (supersonic nozzles). And increases in converging ducts (supersonic diffusers)
To accelerate a fluid, we must use a converging nozzle at subsonic velocities and a diverging nozzle at supersonic velocities.
The highest velocity we can achieve by a converging nozzle is the sonic velocity, which occurs at the exit of the nozzle.
(If we extend the converging nozzle by further decreasing the flow area, in hopes of accelerating the fluid to supersonic velocities, we are disappointed).
Based on Eq. d, which is an expression of the conservation of mass and energy principles, we must add a diverging section to a converging nozzle to accelerate a fluid to supersonic velocities. The result is a converging– diverging nozzle. Where the Mach number increases as the flow area of the nozzle decreases, and then reaches the value of unity at the nozzle throat. The fluid continues to accelerate as it passes through a supersonic (diverging) section.
Noting that $\dot{m} = \rho A V$ for steady flow, we see that the large decrease in density makes acceleration in the diverging section possible.

Property relations for isentropic flow of ideal gases
Next we develop relations between the static properties and stagnation properties of an ideal gas in terms of the specific heat ratio k and the Mach number Ma. We assume the flow is isentropic and the gas has constant specific heats.
We have:

$$T_o = T + \frac{V^2}{2c_p}$$

$$\frac{T_o}{T} = 1 + \frac{V^2}{2c_pT}$$

Noting that $cp = kR/(k-1)$, $c = \sqrt{kRT}$, and $Ma = V/c$

$$\frac{V^2}{2c_pT} = \frac{V^2}{2[kR/(k-1)]T} = \left(\frac{(k-1)}{2}\right)\left(\frac{V^2}{c^2}\right) = \left(\frac{(k-1)}{2}\right)Ma^2$$

Substitution yields

$$\frac{T_o}{T} = 1 + \left(\frac{(k-1)}{2}\right)Ma^2$$

Which is the desired relation between T_o and T. The ratio of the stagnation to static pressure is obtained by:

$$\frac{P_o}{P} = \left(1 + \left(\frac{(k-1)}{2}\right)Ma^2\right)^{k/(k-1)}$$

The ratio of the stagnation to static density is expressed as:

$$\frac{\rho_o}{\rho} = \left(1 + \left(\frac{(k-1)}{2}\right)Ma^2\right)^{1/(k-1)}$$

$$R = c_p - c_v$$

$$\frac{R}{c_p} = 1 - \frac{c_v}{c_p}$$

Where: $\frac{c_v}{c_p} = \frac{1}{k}$

$$\frac{R}{c_p} = 1 - \frac{1}{k}$$

$$\frac{R}{c_p} = \frac{k-1}{k}$$

$$c_p = R/\frac{k-1}{k}$$

$$c_p = \frac{Rk}{k-1}$$

Numerical values of $\frac{T_o}{T}$, $\frac{P_o}{P}$ and $\frac{\rho_o}{\rho}$ are listed versus the Mach number in Table 2 for k = 1.4, which are very useful for practical compressible flow calculations involving air.

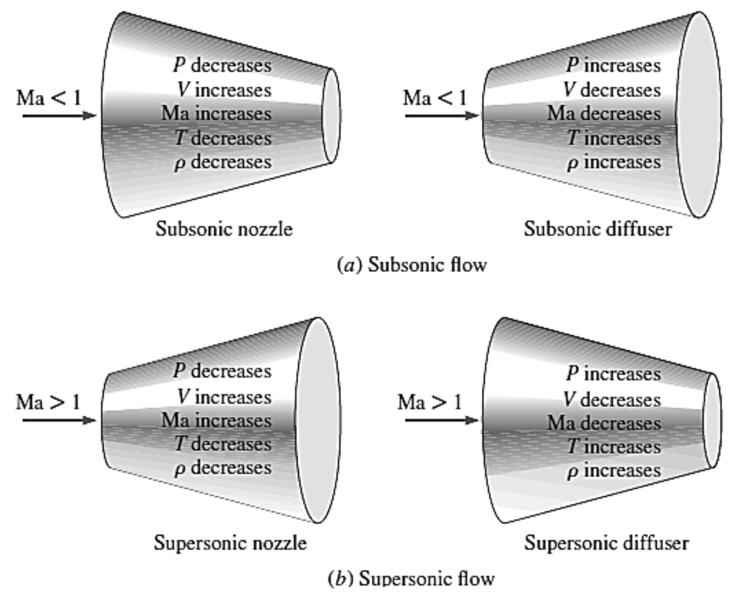
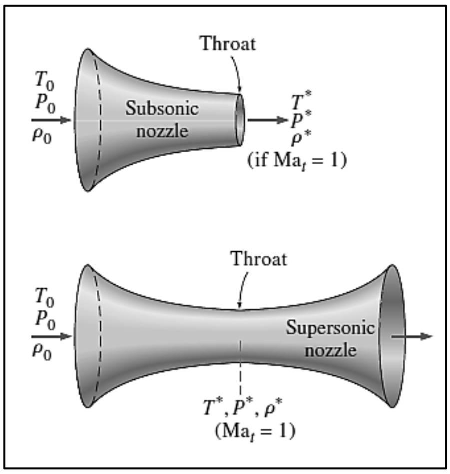


FIGURE
Variation of flow properties in subsonic and supersonic nozzles and diffusers.

The properties of a fluid at a location where the Mach number is unity (the throat) are called critical properties and the ratios in Eqs. above through are called critical ratios when Ma = 1 (Fig). It is standard practice in the analysis of compressible flow to let the superscript (*) represent the critical values. Setting Ma = 1 in Eqs. We get:



$$\frac{T^*}{T_o} = \left(\frac{2}{(k+\ 1)}\right)$$

$$\frac{P^*}{P_o} = \left(\frac{2}{(k+\ 1)}\right)^{k/(k-1)}$$

$$\frac{\rho^*}{\rho_o} = \left(\frac{2}{(k+\ 1)}\right)^{1/(k-1)}$$

These ratios are evaluated for various values of k and are listed in Table 3.

TABLE 3

The critical-pressure, critical-temperature, and critical-density ratios for isentropic flow of some ideal gases

	Superheated steam, $k = 1.3$	Hot products of combustion, $k = 1.33$	Air, $k = 1.4$	Monatomic gases, $k = 1.667$
$\frac{P^*}{P_0}$	0.5457	0.5404	0.5283	0.4871
$\frac{T^*}{T_0}$	0.8696	0.8584	0.8333	0.7499
$\frac{\rho^*}{\rho_0}$	0.6276	0.6295	0.6340	0.6495

EXAMPLE **Critical Temperature and Pressure in Gas Flow**

Calculate the critical pressure and temperature of carbon dioxide for the flow conditions described in Example 1

SOLUTION For the flow discussed in Example 1 , the critical pressure and temperature are to be calculated.

Assumptions **1** The flow is steady, adiabatic, and one-dimensional. **2** Carbon dioxide is an ideal gas with constant specific heats.

Properties The specific heat ratio of carbon dioxide at room temperature is $k = 1.289$.

Analysis The ratios of critical to stagnation temperature and pressure are determined to be

$$\frac{T^*}{T_0} = \frac{2}{k+1} = \frac{2}{1.289+1} = 0.8737$$

$$\frac{P^*}{P_0} = \left(\frac{2}{k+1}\right)^{k/(k-1)} = \left(\frac{2}{1.289+1}\right)^{1.289/(1.289-1)} = 0.5477$$

Noting that the stagnation temperature and pressure are, from Example 1 $T_0 = 473$ K and $P_0 = 1400$ kPa, we see that the critical temperature and pressure in this case are

$$T^* = 0.8737T_0 = (0.8737)(473\text{ K}) = 413\text{ K}$$

$$P^* = 0.5477P_0 = (0.5477)(1400\text{ kPa}) = 767\text{ kPa}$$

Discussion Note that these values agree with those listed in the 5th row of Table :–1, as expected. Also, property values other than these at the throat would indicate that the flow is not critical, and the Mach number is not unity.

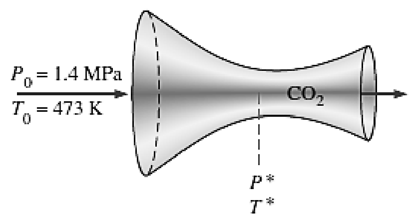


FIGURE
Schematic for Example

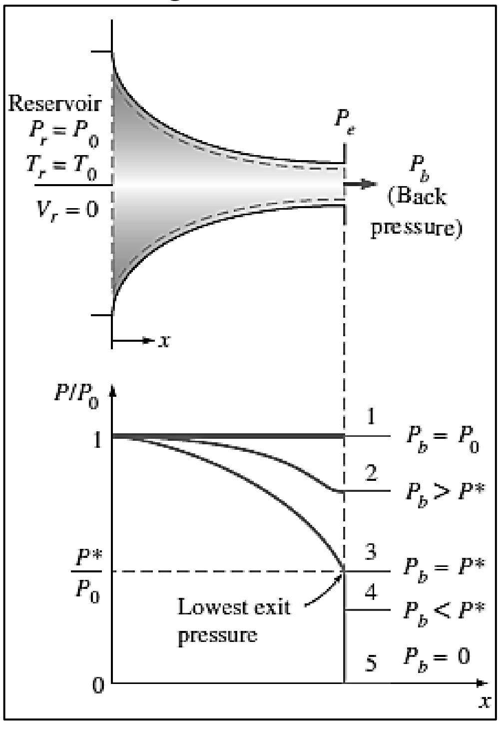
Effects of back pressure for nozzles flow

1- Converging Nozzles

Consider the subsonic flow through a converging nozzle as shown in Fig. The nozzle inlet is attached to a reservoir at pressure P_r and temperature T_r. The reservoir is sufficiently large so that the nozzle inlet velocity is negligible.

Since the fluid velocity in the reservoir is zero and the flow through the nozzle is approximated as isentropic, the stagnation pressure and stagnation temperature of the fluid at any cross section through the nozzle are equal to the reservoir pressure and temperature, respectively.

Now we begin to reduce the back pressure and observe the resulting effects on the pressure distribution along the length of the nozzle, as shown in Fig.



1- If the back pressure P_b is equal to P_1, which is equal to P_r, there is no flow and the pressure distribution is uniform along the nozzle.
2- When the back pressure is reduced to P_2, the exit plane pressure P_e also drops to P_2. This causes the pressure along the nozzle to decrease in the flow direction.
3- When the back pressure is reduced to P_3 (= P^*, which is the pressure required to increase the fluid velocity to the speed of sound at the exit plane or throat), the mass flow reaches a maximum value and the flow is said to be **choked.** Further reduction of the back pressure to level P_4 or

below does not result in additional changes in the pressure distribution, or anything else along the nozzle length.
Under steady-flow conditions, the mass flow rate through the nozzle is constant and is expressed as

$$\dot{m} = \rho A V = \left(\frac{P}{RT}\right) A(\mathrm{Ma}\sqrt{kRT}) = PA\mathrm{Ma}\sqrt{\frac{k}{RT}}$$

$$\dot{m} = \frac{A\mathrm{Ma}P_0\sqrt{k/(RT_0)}}{[1 + (k - 1)\mathrm{Ma}^2/2]^{(k+1)/[2(k-1)]}}$$

The mass flow rate through a nozzle is a maximum when Ma = 1 at the throat. Denoting this area by A* we obtain an expression for the maximum mass flow rate by substituting Ma=1

$$\dot{m}_{\max} = A^* P_0 \sqrt{\frac{k}{RT_0}} \left(\frac{2}{k + 1}\right)^{(k+1)/[2(k-1)]}$$

For all back pressures lower than the critical pressure P*, the pressure at the exit plane of the converging nozzle P_e is equal to P*, the Mach number at the exit plane is unity, and the mass flow rate is the maximum (or choked) flow rate. Because the velocity of the flow is sonic at the throat for the maximum flow rate, a back pressure lower than the critical pressure cannot be sensed in the nozzle upstream flow and does not affect the flow rate.

TABLE 2

One-dimensional isentropic compressible flow functions for an ideal gas with $k = 1.4$

Ma	Ma*	A/A^*	P/P_0	ρ/ρ_0	T/T_0
0	0	∞	1.0000	1.0000	1.0000
0.1	0.1094	5.8218	0.9930	0.9950	0.9980
0.2	0.2182	2.9635	0.9725	0.9803	0.9921
0.3	0.3257	2.0351	0.9395	0.9564	0.9823
0.4	0.4313	1.5901	0.8956	0.9243	0.9690
0.5	0.5345	1.3398	0.8430	0.8852	0.9524
0.6	0.6348	1.1882	0.7840	0.8405	0.9328
0.7	0.7318	1.0944	0.7209	0.7916	0.9107
0.8	0.8251	1.0382	0.6560	0.7400	0.8865
0.9	0.9146	1.0089	0.5913	0.6870	0.8606
1.0	1.0000	1.0000	0.5283	0.6339	0.8333
1.2	1.1583	1.0304	0.4124	0.5311	0.7764
1.4	1.2999	1.1149	0.3142	0.4374	0.7184
1.6	1.4254	1.2502	0.2353	0.3557	0.6614
1.8	1.5360	1.4390	0.1740	0.2868	0.6068
2.0	1.6330	1.6875	0.1278	0.2300	0.5556
2.2	1.7179	2.0050	0.0935	0.1841	0.5081
2.4	1.7922	2.4031	0.0684	0.1472	0.4647
2.6	1.8571	2.8960	0.0501	0.1179	0.4252
2.8	1.9140	3.5001	0.0368	0.0946	0.3894
3.0	1.9640	4.2346	0.0272	0.0760	0.3571
5.0	2.2361	25.000	0.0019	0.0113	0.1667
∝	2.2495	∝	0	0	0

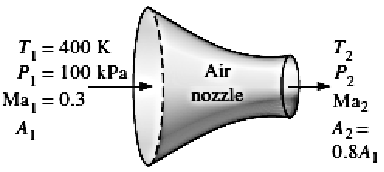


FIGURE
Schematic for Example

EXAMPLE **Effect of Back Pressure on Mass Flow Rate**

Air at 1 MPa and 600°C enters a converging nozzle, shown in Fig. with a velocity of 150 m/s. Determine the mass flow rate through the nozzle for a nozzle throat area of 50 cm² when the back pressure is (*a*) 0.7 MPa and (*b*) 0.4 MPa.

SOLUTION Air enters a converging nozzle. The mass flow rate of air through the nozzle is to be determined for different back pressures.
Assumptions 1 Air is an ideal gas with constant specific heats at room temperature. 2 Flow through the nozzle is steady, one-dimensional, and isentropic.
Properties The constant pressure specific heat and the specific heat ratio of air are $c_p = 1.005$ kJ/kg·K and k = 1.4.
Analysis We use the subscripts *i* and *t* to represent the properties at the nozzle inlet and the throat, respectively. The stagnation temperature and pressure at the nozzle inlet are determined

$$T_{0i} = T_i + \frac{V_i^2}{2c_p} = 873\text{ K} + \frac{(150\text{ m/s})^2}{2(1.005\text{ kJ/kg·K})}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 884\text{ K}$$

$$P_{0i} = P_i\left(\frac{T_{0i}}{T_i}\right)^{k/(k-1)} = (1\text{ MPa})\left(\frac{884\text{ K}}{873\text{ K}}\right)^{1.4/(1.4-1)} = 1.045\text{ MPa}$$

These stagnation temperature and pressure values remain constant throughout the nozzle since the flow is assumed to be isentropic. That is,

$$T_0 = T_{0i} = 884\text{ K} \quad \text{and} \quad P_0 = P_{0i} = 1.045\text{ MPa}$$

The critical-pressure ratio is determined from Table 3. be $P^*/P_0 = 0.5283$.

(*a*) The back pressure ratio for this case is

$$\frac{P_b}{P_0} = \frac{0.7\text{ MPa}}{1.045\text{ MPa}} = 0.670$$

which is greater than the critical-pressure ratio, 0.5283. Thus the exit plane pressure (or throat pressure P_t) is equal to the back pressure in this case. That is, $P_t = P_b = 0.7$ MPa, and $P_t/P_0 = 0.670$. Therefore, the flow is not choked. From Table A–13 at $P_t/P_0 = 0.670$, we read $Ma_t = 0.778$ and $T_t/T_0 = 0.892$.

The mass flow rate through the nozzle can be calculated from
But it can also be determined in a step-by-step manner as follows:

$$T_t = 0.892T_0 = 0.892(884\text{ K}) = 788.5\text{ K}$$

$$\rho_t = \frac{P_t}{RT_t} = \frac{700\text{ kPa}}{(0.287\text{ kPa·m}^3/\text{kg·K})(788.5\text{ K})} = 3.093\text{ kg/m}^3$$

$$V_t = \text{Ma}_t c_t = \text{Ma}_t\sqrt{kRT_t}$$

$$= (0.778)\sqrt{(1.4)(0.287\text{ kJ/kg·K})(788.5\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)}$$

$$= 437.9\text{ m/s}$$

Thus,

$$\dot{m} = \rho_t A_t V_t = (3.093\text{ kg/m}^3)(50 \times 10^{-4}\text{ m}^2)(437.9\text{ m/s}) = 6.77\text{ kg/s}$$

(*b*) The back pressure ratio for this case is

$$\frac{P_b}{P_0} = \frac{0.4\text{ MPa}}{1.045\text{ MPa}} = 0.383$$

which is less than the critical-pressure ratio, 0.5283. Therefore, sonic conditions exist at the exit plane (throat) of the nozzle, and Ma = 1. The flow is choked in this case, and the mass flow rate through the nozzle is calculated

$$\dot{m} = A^*P_0\sqrt{\frac{k}{RT_0}}\left(\frac{2}{k+1}\right)^{(k+1)/[2(k-1)]}$$

$$= (50 \times 10^{-4}\text{ m}^2)(1045\text{ kPa})\sqrt{\frac{1.4}{(0.287\text{ kJ/kg·K})(884\text{ K})}}\left(\frac{2}{1.4+1}\right)^{2.4/0.8}$$

$$= 7.10\text{ kg/s}$$

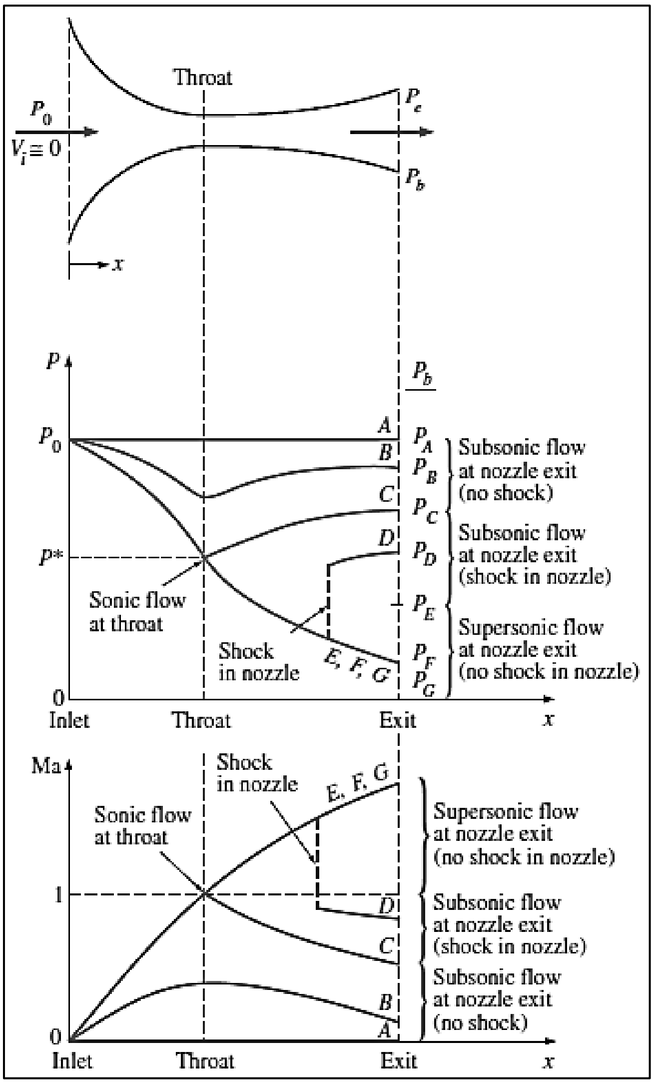
since $\text{kPa·m}^2\sqrt{\text{kJ/kg}} = \sqrt{1000}\text{ kg/s}$.
Discussion This is the maximum mass flow rate through the nozzle for the specified inlet conditions and nozzle throat area.

2- Converging–Diverging Nozzles

When we think of nozzles, we ordinarily think of flow passages whose cross-sectional area decreases in the flow direction. However, the highest velocity to which a fluid can be accelerated in a converging nozzle is limited to the sonic velocity (Ma = 1), which occurs at the exit plane (throat) of the nozzle. Accelerating a fluid to supersonic velocities (Ma > 1) can be accomplished only by attaching a diverging flow section to the subsonic nozzle at the throat. The resulting combined flow section is a converging–diverging nozzle

Forcing a fluid through a converging–diverging nozzle is no guarantee that the fluid will be accelerated to a supersonic velocity. In fact, the fluid may find itself decelerating in the diverging section instead of accelerating if the back pressure is not in the right range. The state of the nozzle flow is determined by the overall pressure ratio P_b / P_o. Therefore, for given inlet conditions, the flow through a converging–diverging nozzle is governed by the back pressure P_b, as will be explained.

Consider the converging–diverging nozzle shown in Fig. A fluid enters the nozzle with a low velocity at stagnation pressure P_o.



1- When $P_b = P_o$ (case A), there is no flow through the nozzle. This is expected since the flow in a nozzle is driven by the pressure difference between the nozzle inlet and the exit.

2- When $P_o > P_b > P_C$, the flow remains subsonic throughout the nozzle, and the mass flow is less than that for choked flow. The fluid velocity increases in the first (converging) section and reaches a maximum at the throat (but Ma < 1). However, most of the gain in velocity is lost in the second (diverging) section of the nozzle, which acts as a diffuser. The pressure decreases in the converging section, reaches a minimum at the throat, and increases at the expense of velocity in the diverging section.

3- When $P_b = P_C$, the throat pressure becomes P* and the fluid achieves sonic velocity at the throat.

4- When $P_C > P_b > P_E$, the fluid that achieved a sonic velocity at the throat continues accelerating to supersonic velocities in the diverging section as the pressure decreases. This acceleration comes to a sudden stop, however, as a normal shock develops at a section between the throat and the exit plane, which causes a sudden drop in velocity to subsonic levels and a sudden increase in pressure. The fluid then continues to decelerate further in the remaining part of the converging–diverging nozzle.

Flow through the shock is highly irreversible, and thus it cannot be approximated as isentropic.

The normal shock moves downstream away from the throat as P_b is decreased, and it approaches the nozzle exit plane as P_b approaches P_E.
When $P_b = P_E$, the normal shock forms at the exit plane of the nozzle.
The flow is supersonic through the entire diverging section in this case, and it can be approximated as isentropic. However, the fluid velocity drops to subsonic levels just before leaving the nozzle as it crosses the normal shock.

5- When $P_E > P_b$, the flow in the diverging section is supersonic, and the fluid expands to P_F at the nozzle exit with no normal shock forming within the nozzle.

Example1: Air in an automobile tire is maintained at a pressure of 220 kPa (gage) in an environment where the atmospheric pressure is 94 kPa. The air in the tire is at the ambient temperature of 25°C. A 4-mm-diameter leak develops in the tire as a result of an accident (Fig). Approximating the flow as isentropic, determine the initial mass flow rate of air through the leak.

Example2: Air enters a converging–diverging nozzle, shown in Fig, at 1.0 MPa and 800 K with negligible velocity. The flow is steady, one-dimensional, and isentropic with k = 1.4. For an exit Mach number of Ma = 2 and a throat area of 20 cm^2, determine (a) the throat conditions, (b) the exit plane conditions, including the exit area, and (c) the mass flow rate through the nozzle.