PIE Tech
POLLACHI INSTITUTE OF ENGINEERING AND TECHNOLOGY
(Approved by AICTE and Affiliated to Anna University)
sky is the limit

DEPARTMENT OF MECHANICAL ENGINEERING

Regulation 2021

II Year – IV Semester

CE3391- Fluid Mechanics and Machinery

FLUID MECHANICS & MACHINERY

UNIT–I FLUID PROPERTIES & FLOW CHARACTERISTICS

Units and dimensions - Types of flows - Properties of fluids - mass density, specific weight, specific volume, specific gravity, viscosity, compressibility, vapor pressure – Gas laws - Surface tension and capillarity. Flow characteristics – concept of control volume – Bernoulli's Theorem – Concept of control volume – Application of continuity equation, energy equation, momentum equation and moment of momentum equation.

UNIT - II FLOW THROUGH CIRCULAR CONDUITS

Hydraulic and energy gradient - Laminar flow through circular conduits and circular annuli-Hydraulic and energy gradient-Boundary layer concepts – types of boundary layer thickness – Darcy Weisbach equation –friction factor- Moody diagram- commercial pipes- minor losses – Flow through pipes in series and parallel.

UNIT–III-DIMENSIONAL ANALYSIS

Dimensional analysis – methods of dimensional analysis - Similitude –types of similitude – Dimensionless parameters- Application of dimensionless parameters – Model analysis.

UNIT–IV HYDRAULIC PUMPS

Impact of jets - Euler's equation - Theory of roto-dynamic machines – various efficiencies - velocity triangles - Centrifugal pumps– Multi stage centrifugal pumps - working principle - work done by the impeller - performance curves – Priming – Cavitation - Reciprocating pump- working principle – Air vessels – Indicator diagram - Rotary pumps – Working Principles.

UNIT–V HYDRAULIC TURBINES

Hydraulic turbines – Classification - working principles - Pelton wheel, Kaplan turbine - Francis turbine - velocity triangles - theory of draft tubes – Performance – Specific speed – Unit Quantities - Selection of turbines - governing of turbines - hydraulic coupling - Torque converter.

TEXT BOOK

- K.L. Kumar, Engineering Fluid Mechanics, S. Chand Publishing, 2016.
- Modi P.N. & Seth, S.M. "Hydraulics and Fluid Mechanics", Standard Book House, New Delhi 2013.

REFERENCES

- K. Som, G. Biswas, S Chakraborty, Introduction to Fluid Mechanics and Fluid Machines, Tata McGraw Hill, 2008, 3rd Edition.
- R. Arora, Fluid Mechanics Hydraulics and Hydraulic Machines, Standard Publishers, 2007, 9th Edition.
- P. Kothandaraman & R. Rudramoorthy. Fluid Mechanics and Machinery, New Academia Science, 2011, 3rd Edition.
- Douglas J.F, Solving Problems in Fluid Mechanics Vol I & II, John Wiley & Sons Inc., 1986.
- Victor L. Streeter and E. Benjamin Wylie & Keith W.Bedford. Fluid Mechanics, Mc Graw-Hill 1999, 8th Edition.

UNIT - 1
Fluids Mechanics

What is fluid mechanics?

As its name suggests it is the branch of applied mechanics concerned with the statics and dynamics of fluids - both liquids and gases. The analysis of the behavior of fluids is based on the fundamental laws of mechanics which relate continuity of mass and energy with force and momentum together with the familiar solid mechanics properties.

Objectives of this section

- Define the nature of a fluid.
- Show where fluid mechanics concepts are common with those of solid mechanics and indicate some fundamental areas of difference.
- Introduce viscosity and show what are Newtonian and non-Newtonian fluids
- Define the appropriate physical properties and show how these allow differentiation between solids and fluids as well as between liquids and gases.

Fluids

There are two aspects of fluid mechanics which make it different to solid mechanics:

1. The nature of a fluid is much different to that of a solid
2. In fluids we usually deal with *continuous* streams of fluid without a beginning or end. In solids we only consider individual elements.

We normally recognise three states of matter: solid; liquid and gas. However, liquid and gas are both fluids: in contrast to solids they lack the ability to resist deformation. Because a fluid cannot resist the deformation force, it moves, it *flows* under the action of the force. Its shape will change continuously as long as the force is applied. A solid can resist a deformation force while at rest, this force may cause some displacement but the solid does not continue to move indefinitely.

The deformation is caused by *shearing* forces which act tangentially to a surface. Referring to the figure below, we see the force F acting tangentially on a rectangular (solid lined) element ABDC. This is a shearing force and produces the (dashed lined) rhombus element A'B'DC.

$$\sigma = \frac{\rho_{substance}}{\rho_{H_2O(at 4^\circ c)}}$$

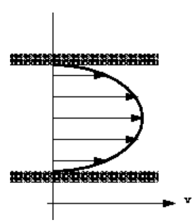
We can then say

A Fluid is a substance which deforms continuously, or flows, when subjected to shearing force, and conversely this definition implies the very important point that

If a fluid is at rest there are no shearing forces acting. All forces must be perpendicular to the planes which they are acting.

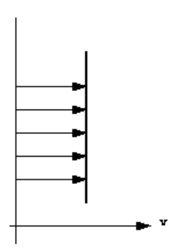
When a fluid is in motion shear stresses are developed if the particles of the fluid move relative to one another. When this happens adjacent particles have different velocities. If fluid velocity is the same at every point then there is no shear stress produced: the particles have zero *relative* velocity.

Consider the flow in a pipe in which water is flowing. At the pipe wall the velocity of the water will be zero. The velocity will increase as we move toward the centre of the pipe. This change in velocity across the direction of flow is known as velocity profile and shown graphically in the figure below:



Velocity profile in a pipe.

Because particles of fluid next to each other are moving with different velocities there **are** shear forces in the moving fluid i.e. shear forces are **normally** present in a moving fluid. On the other hand, if a fluid is a long way from the boundary and all the particles are travelling with the same velocity, the velocity profile would look something like this:

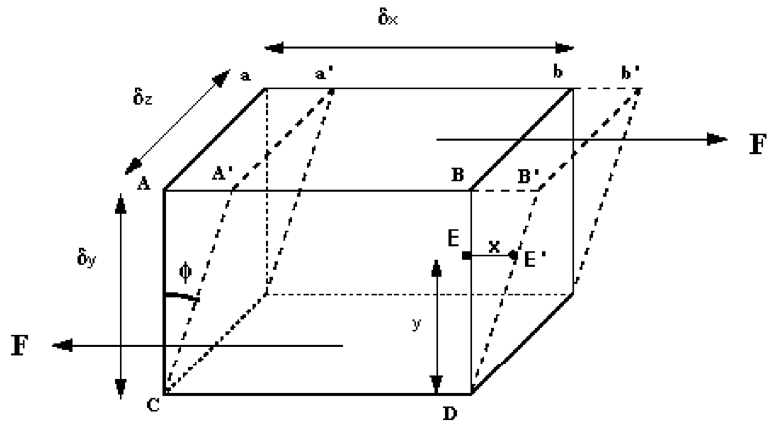


Velocity profile in uniform flow

and there will be no shear forces present as all particles have zero relative velocity. In practice we are concerned with flow past solid boundaries; aeroplanes, cars, pipe walls, river channels etc. and shear forces will be present.

Newton's Law of Viscosity

We can start by considering a 3d rectangular element of fluid, like that in the figure below.



Fluid element under a shear force

The shearing force F acts on the area on the top of the element. This area is given by $A = \delta z \times \delta x$. We

$$\text{shear stress},\ \tau = \frac{F}{A}$$

The deformation which this shear stress causes is measured by the size of the angle ϕ and is know as *shear strain*.

In a solid shear strain, ϕ, is constant for a fixed shear stress τ.

In a fluid ϕ increases for as long as τ is applied - the fluid flows.

It has been found experimentally that the *rate of shear stress* (shear stress per unit time, τ/time) is directly proportional to the shear stress.

If the particle at point E (in the above figure) moves under the shear stress to point E' and it takes time *t* to get there, it has moved the distance *x*. For small deformations we can write

$$\phi = \frac{x}{y}$$

$$\begin{aligned}\text{rate of shear strain} &= \frac{\phi}{t}\\ &= \frac{x}{ty} = \frac{x}{t}\frac{1}{y}\\ &= \frac{u}{y}\end{aligned}$$

where $\frac{x}{t} = u$ is the velocity of the particle at E.

Using the experimental result that shear stress is proportional to rate of shear strain

then

$$\tau = \text{Constant} \times \frac{u}{y}$$

The term $\frac{u}{y}$ is the change in velocity with y, or the velocity gradient, and may be written in the differential form $\frac{du}{dy}$. The constant of proportionality is known as the dynamic viscosity, μ, of the fluid, giving $\tau = \mu \frac{du}{dy}$

Fluids and Solids

In the above we have discussed the differences between the behaviour of solids and fluids under an applied force. Summarising, we have;

- For a **solid** the strain is a function of the applied stress (providing that the elastic limit has not been reached). For a **fluid**, the rate of strain is proportional to the applied stress.

- The strain in a **solid** is independent of the time over which the force is applied and (if the elastic limit is not reached) the deformation disappears when the force is removed. A **fluid** continues to flow for as long as the force is applied and will not recover its original form when the force is removed.

It is usually quite simple to classify substances as either solid or liquid. Some substances, however, (e.g. pitch or glass) appear solid under their own weight. Pitch will, although appearing solid at room temperature, deform and spread out over days - rather than the fraction of a second it would take water.

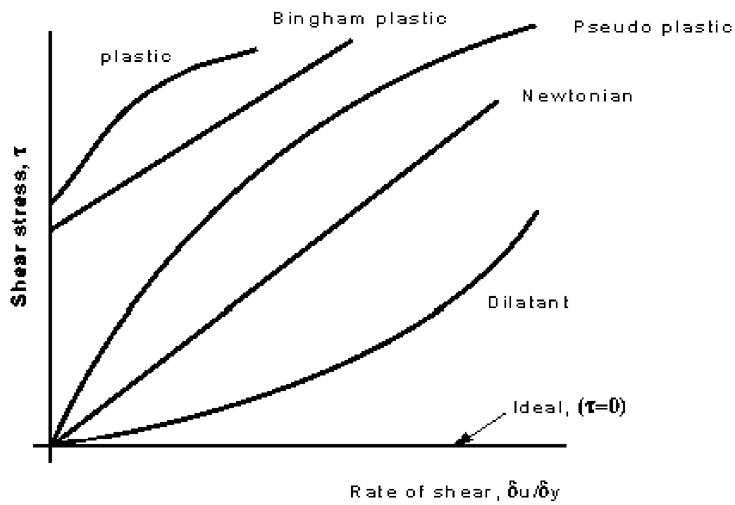
As you will have seen when looking at properties of solids, when the elastic limit is reached they seem to flow. They become plastic. They still do **not** meet the definition of true fluids as they will only flow after a certain minimum shear stress is attained.

Newtonian / Non-Newtonian Fluids

Even among fluids which are accepted as fluids there can be wide differences in behaviour under stress. Fluids obeying Newton's law where the value of μ is constant are known as **Newtonian** fluids. If μ is constant the shear stress is linearly dependent on velocity gradient. This is true for most common fluids.

Fluids in which the value of μ is not constant are known as **non-Newtonian** fluids. There are several categories of these, and they are outlined briefly below.

These categories are based on the relationship between shear stress and the velocity gradient (rate of shear strain) in the fluid. These relationships can be seen in the graph below for several categories



Shear stress vs. Rate of shear strain

Each of these lines can be represented by the equation

$$\tau = A + B\left(\frac{\delta u}{\delta y}\right)^n$$

where A, B and n are constants. For Newtonian fluids A = 0, B = μ and n = 1.

Below are brief description of the physical properties of the several

categories:

- *Plastic:* Shear stress must reach a certain minimum before flow commences.
- *Bingham plastic:* As with the plastic above a minimum shear stress must be achieved. With this classification n = 1. An example is sewage sludge.
- *Pseudo-plastic:* No minimum shear stress necessary and the viscosity decreases with rate of shear, e.g. colloidial substances like clay, milk and cement.
- *Dilatant substances;* Viscosity increases with rate of shear e.g. quicksand.
- *Thixotropic substances:* Viscosity decreases with length of time shear force is applied

e.g. thixotropic jelly paints.

- *Rheopectic substances:* Viscosity increases with length of time shear force is applied
- *Viscoelastic materials:* Similar to Newtonian but if there is a sudden large change in shear they behave like plastic.

There is also one more - which is not real, it does not exist - known as the **ideal fluid**. This is a fluid which is assumed to have no viscosity. This is a useful concept when theoretical solutions are being considered
- it does help achieve some practically useful solutions.

Liquids and Gasses

Although liquids and gasses behave in much the same way and share many similar characteristics, they also possess distinct characteristics of their own. Specifically

- A liquid is difficult to compress and often regarded as being incompressible.A gas is easily to compress and usually treated as such - it changes volume with pressure.A
- given mass of liquid occupies a given volume and will occupy the container it is in and form a free surface (if the container is of a larger volume).A gas has no fixed volume, it changes volume to expand to fill the containing vessel. It will completely fill the vessel so no free surface is formed.

Causes of Viscosity in Fluids

Viscosity in Gasses

The molecules of gasses are only weakly kept in position by molecular cohesion (as they are so far apart). As adjacent layers move by each other there is a continuous exchange of molecules. Molecules ofa slower layer move to faster layers causing a drag, while molecules moving the other way exert an acceleration force. Mathematical considerations of this momentum exchange can lead to Newton law of viscosity.

If temperature of a gas increases the momentum exchange between layers will increase thus increasing viscosity.

Viscosity will also change with pressure - but under normal conditions this change is negligible in gasses.

Viscosity in Liquids

There is some molecular interchange between adjacent layers in liquids - but as the molecules are so much closer than in gasses the cohesive forces hold the molecules in place much more rigidly. This cohesion

plays an important roll in the viscosity of liquids.

Increasing the temperature of a fluid reduces the cohesive forces and increases the molecular interchange. Reducing cohesive forces reduces shear stress, while increasing molecular interchange increases shear stress. Because of this complex interrelation the effect of temperature on viscosity has something of the form:

$$\mu_T = \mu_0 (1 + AT + BT)$$

Where μ_T is the viscosity at temperature TC, and μ_0 is the viscosity at temperature 0C. A and B are constants for a particular fluid.

High pressure can also change the viscosity of a liquid. As pressure increases the relative movement of molecules requires more energy hence viscosity increases.

Properties of Fluids

The properties outlines below are general properties of fluids which are of interest in engineering. The symbol usually used to represent the property is specified together with some typical values in SI units for common fluids. Values under specific conditions (temperature, pressure etc.) can be readily found in many reference books. The dimensions of each unit is also give in the MLT system (see later in the section on dimensional analysis for more details about dimensions.)

Density

The density of a substance is the quantity of matter contained in a unit volume of the substance. It can be expressed in three different ways.

Mass Density

Mass Density, ρ, is defined as the mass of substance per unit volume.

Units: Kilograms per cubic metre, kg/m^3 (or kgm^{-3})

Dimensions: ML^{-3}

Typical values:

Water = 1000 kgm^{-3}, Mercury = 13546 kgm^{-3} Air = 1.23 kgm^{-3}, Paraffin Oil = 800 kgm^{-3}.

$\times 10^{-3}\ Nm^{-2}$

(at pressure =1.013 and Temperature = 288.15 K.)

Specific Weight

Specific Weight ω, (sometimes , and sometimes known as *specific gravity*) is defined as the weight per unit volume. *or*

The force exerted by gravity, g, upon a unit volume of the substance.

The Relationship between g and ω can be determined by Newton's 2nd Law, since

weight per unit volume = mass per unit volume g $\omega = \rho g$

Units: Newton's per cubic metre, N/m^3 (or Nm^{-3})

Dimensions: $ML^{-2}T^{-2}$.

Typical values:

Water =9814 Nm^{-3}, Mercury = 132943 Nm^{-3}, Air =12.07 Nm^{-3}, Paraffin Oil =7851 Nm^{-3}

Relative Density

Relative Density, σ, is defined as the ratio of mass density of a substance to some standard mass density. For solids and liquids this standard mass density is the maximum mass density for water (which

occurs at 4°c) at atmospheric pressure.

$$\sigma = \frac{\rho_{substance}}{\rho_{H_2O(at\,4^\circ c)}}$$

Units: None, since a ratio is a pure number.

Dimensions: 1.

Typical values: Water = 1, Mercury = 13.5, Paraffin Oil =0.8.

Viscosity

Viscosity, μ,is the property of a fluid, due to cohesion and interaction between molecules, which offers resistance to sheer deformation. Different fluids deform at different rates under the same shear stress. Fluid with a high viscosity such as syrup, deforms more slowly than fluid with a low viscosity such as water.

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

All fluids are viscous, "Newtonian Fluids" obey the linear relationship given by Newton's law of viscosity. , which we saw

earlier. where is the shear stress,

Units $N m^{-2}$; $kg\, m^{-1} s^{-2}$

Dimensions $ML^{-1}T^{-2}$.

$\frac{du}{dy}$ is the velocity gradient or rate of shear strain, and has

Units: $radians s^{-1}$,

Dimensions t^{-1}

μ is the "coefficient of dynamic viscosity" - see below.

Coefficient of Dynamic Viscosity

The Coefficient of Dynamic Viscosity, μ, is defined as the shear force, per unit area, (or shear stress τ), required to drag one layer of fluid with unit velocity past another layer a unit distance away.

$$\mu = \tau \bigg/ \frac{du}{dy} = \frac{\text{Force}}{\text{Area}} \bigg/ \frac{\text{Velocity}}{\text{Distance}} = \frac{\text{Force} \times \text{Time}}{\text{Area}} = \frac{\text{Mass}}{\text{Length} \times \text{Area}}$$

Units: Newton seconds per square metre, $N s m^{-2}$ or Kilograms per meter per second, $kg m^{-1} s^{-1}$.

(Although note that μ is often expressed in Poise, P, where 10 P = 1 $kg m^{-1} s^{-1}$.)

Typical values:

Water =1.14 $\times 10^{-3}$ $kg m^{-1} s^{-1}$, Air =1.78 $\times 10^{-5}$ $kg m^{-1} s^{-1}$, Mercury =1.552 $kg m^{-1} s^{-1}$, Paraffin Oil =1.9 $kg m^{-1} s^{-1}$.

Kinematic Viscosity

Kinematic Viscosity, ν, is defined as the ratio of dynamic viscosity to mass density.

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

Units: square metres per second, $m^2 s^{-1}$

(Although note that ν is often expressed in Stokes, St, where 10^4 St = 1 $m^2 s^{-1}$.)

Dimensions: $L^2 T^{-1}$.

Typical values:

Water =1.14 $\times 10^{-6}\ m^2s^{-1}$, Air =1.46 $\times 10^{-5}\ m^2s^{-1}$, Mercury =1.145 $\times 10^{-4}\ m^2s^{-1}$,
Paraffin Oil =2.375 $\times 10^{-3}\ m^2s^{-1}$.

Example - Problems

1. Explain why the viscosity of a liquid decreases while that of a gas increases with a temperature rise.

The following is a table of measurement for a fluid at constant temperature.

Determine the dynamic viscosity of the fluid.

du/dy (s^{-1})	*0.0*	*0.2*	*0.4*	*0.6*	*0.8*
τ(N m^{-2})	0.0	1.0	1.9	3.1	4.0

Using Newton's law of viscocity $\tau = \mu \frac{\partial u}{\partial y}$

where μ is the viscosity. So viscosity is the gradient of a graph of shear stress against vellocity gradient of the above data, or

$$\mu = \frac{\tau}{\frac{\partial u}{\partial y}}$$

Calculate the gradient for each section of the line

du/dy (s^{-1})	*0.0*	*0.2*	*0.4*	*0.6*	*0.8*
(N m^{-2})	0.0	1.0	1.9	3.1	4.0
Gradient τ	-	5.0	4.75	5.17	5.0

Thus the mean gradient = viscosity = 4.98 N s / m^2

2. The density of oil is 850 kg/m^3. Find its relative density and Kinematic viscosity if the dynamic viscosity is 5 × 10^{-3} kg/ms.

ρ_{oil} = 850 kg/m3 ρ_{water} = 1000 kg/m3

γ

μ ×

oil = 850 / 1000 = 0.85

Dynamic viscosity = = 5 10^{-3} kg/ms

Kinematic viscosity = $\nu = \mu / \rho$

$$\nu = \frac{\mu}{\rho} = \frac{5\times10^{-3}}{1000} = 5\times10^{-6}\, m^2 s^{-1}$$

3. The velocity distribution of a viscous liquid (dynamic viscosity $\mu = 0.9$ Ns/m^2) flowing over a fixed plate is given by u = 0.68y - y^2 (u is velocity in m/s and y is the distance from the plate in m).

What are the shear stresses at the plate surface and at y=0.34m?

$$u = 0.68y - y^2$$
$$\frac{\partial u}{\partial y} = 0.68 - 2y$$

At the plate face y = 0m,

$$\frac{\partial u}{\partial y} = 0.68$$

Calculate the shear stress at the plate face

$$\tau = \mu \frac{\partial u}{\partial y} = 0.9\times0.68 = 0.612\, N/m^2$$

At y = 0.34m,

$$\frac{\partial u}{\partial y} = 0.68 - 2\times0.34 = 0.0$$

As the velocity gradient is zero at y=0.34 then the shear stress must also be zero.

4. 5.6m^3 of oil weighs 46 800 N. Find its mass density, ρ, and relative density, γ.

Weight 46 800 = mg

Mass m = 46 800 / 9.81 = 4770.6 kg

Mass density ρ = Mass / volume = 4770.6 / 5.6 = 852 kg/m^3

Relative density $\gamma = \frac{\rho}{\rho_{water}} = \frac{852}{1000} = 0.852$

5. From table of fluid properties the viscosity of water is given as 0.01008 poises.
What is this value in Ns/m^2 and Pa s units?

$\mu = 0.01008$ poise

1 poise = 0.1 Pa s = 0.1 Ns/m^2

$\mu = 0.001008$ Pa s = 0.001008 Ns/m^2

6. In a fluid the velocity measured at a distance of 75mm from the boundary is 1.125m/s. The fluid has absolute viscosity 0.048 Pa s and relative density 0.913. What is the velocity gradient and shear stress at the boundary assuming a linear velocity distribution.

$\mu = 0.048$ Pa s

$\gamma = 0.913$

$$\frac{\partial u}{\partial y} = \frac{1.125}{0.075} = 15 s^{-1}$$

$$\tau = \mu \frac{\partial u}{\partial y}$$

$$= 0.048 \times 15 = 0.720 Pa s$$

System of units

As any quantity can be expressed in whatever way you like it is sometimes easy to become confused as to what exactly or how much is being referred to. This is particularly true in the field of fluid mechanics. Over the years many different ways have been used to express the various quantities involved. Even today different countries use different terminology as well as different units for the same thing - they even use the same name for different things e.g. an American pint is 4/5 of a British pint!

To avoid any confusion on this course we will always used the SI (metric) system - which you will already be familiar with. It is essential that all quantities be expressed in the same system or the wrong solution will results. Despite this warning you will still find that that this is the most common mistake when you attempt example questions.

The SI System of units

The SI system consists of six **primary** units, from which all quantities may be described. For convenience **secondary** units are used in general practise which are made from combinations of these primary units.

Primary Units

The six **primary** units of the SI system are shown in the table below:

Quantity	SI Unit	Dimension
Length	Metre, m	L
Mass	Kilogram, kg	M
Time	Second, s	T
Temperature	Kelvin, K	Θ
Current	*Ampere, A*	*I*
Luminosity	*Candela*	*Cd*

In fluid mechanics we are generally only interested in the top four units from this table.

Notice how the term 'Dimension' of a unit has been introduced in this table. This is not a property of the individual units, rather it tells what the unit represents. For example a metre is a length which has a dimension L but also, an inch, a mile or a kilometre are all lengths so have dimension of L.

(The above notation uses the MLT system of dimensions, there are other ways of writing dimensions - we will see more about this in the section of the course on dimensional analysis.)

Derived Units

There are many **derived** units all obtained from combination of the above **primary** units. Those most used are shown in the table below:

Quantity	SI Unit		Dimension
velocity	m/s	ms^{-1}	LT^{-1}
acceleration	m/s^2	ms^{-2}	LT^{-2}
force	N $kg\ m/s^2$	$kg\ ms^{-2}$	$M\ LT^{-2}$
energy (or work)	Joule J N m, $kg\ m^2/s^2$	$kg\ m^2s^{-2}$	ML^2T^{-2}
power	Watt W N m/s $kg\ m^2/s^3$	Nms^{-1} $kg\ m^2s^{-3}$	ML^2T^{-3}
pressure (or stress)	Pascal P, N/m^2, $kg/m/s^2$	Nm^{-2} $kg\ m^{-1}s^{-2}$	$ML^{-1}T^{-2}$

density	kg/m^3	kg m^{-3}	ML^{-3}
specific weight	N/m^3 kg/m^2/s^2	kg m^{-2}s^{-2}	ML^{-2}T^{-2}
relative density	a ratio no units		1 no dimension
viscosity	N s/m^2 kg/m s	N sm^{-2} kg m^{-1}s^{-1}	M L^{-1}T^{-1}
surface tension	N/m kg /s^2	Nm^{-1} kg s^{-2}	MT^{-2}

The above units should be used at all times. Values in other units should NOT be used without first converting them into the appropriate SI unit. If you do not know what a particular unit means find out, else your guess will probably be wrong. One very useful tip is to write down the units of any equation you are using. If at the end the units do not match you know you have made a mistake. For example is you have at the end of a calculation, 30 kg/m s = 30 m

You have certainly made a mistake - checking the units can often help find the mistake. More on this subject will be seen later in the section on dimensional analysis and similarity.

Units

A water company wants to check that it will have sufficient water if there is a prolonged drought in the area. The region it covers is 500 square miles and the following consumption figures have been sent in by various different offices. There is sufficient information to calculate the amount of water available, but unfortunately it is in several different units.

Of the total area 100 000 acres is rural land and the rest urban. The density of the urban population is50 per square kilometre. The average toilet cistern is sized 200mm by 15in by 0.3m and on average each person uses this 3 time per day. The density of the rural population is 5 per square mile. Baths are taken twice a week by each person with the average volume of water in the bath being 6 gallons. Local industry uses 1000 m3 per week. Other uses are estimated as 5 gallons per person per day. A US air basein the region has given water use figures of 50 US gallons per person per day.

The average rain fall in 1in per month (28 days). In the urban area all of this goes to the river while in the rural area 10% goes to the river, 85% is lost (to the aquifer) and the rest goes to the one reservoir which supplies the region. This reservoir has an average surface area of 500 acres and is at a depth of 10 fathoms. 10% of this volume can be used in a month.

1. What is the total consumption of water per day in cubic meters?

2. If the reservoir was empty and no water could be taken from the river, would there be enough water if available if rain fall was only 10% of average?

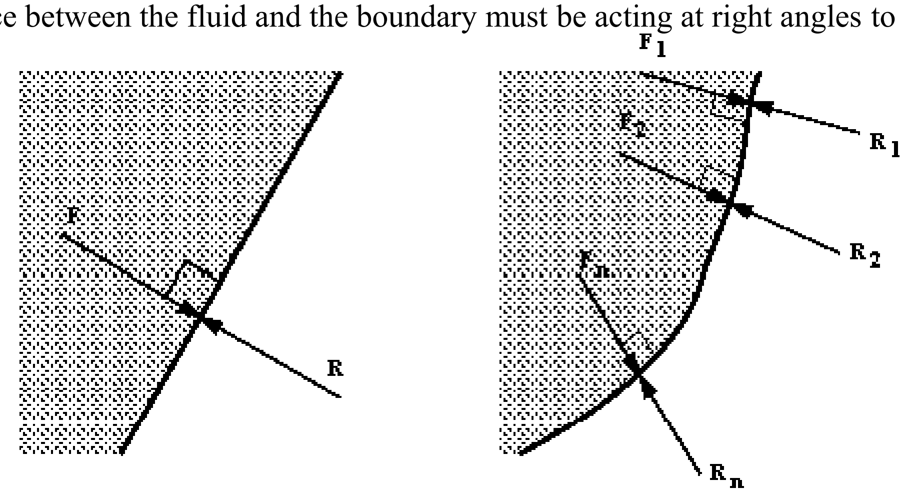
Fluid Statics

This understanding of pressure will then be used to demonstrate methods of pressure measurement that will be useful later with fluid in motion and also to analyse the forces on submerges surface/structures.

Fluids statics

The general rules of statics (as applied in solid mechanics) apply to fluids at rest. From earlier we know that:

- a static fluid can have **no shearing force** acting on it, and that
- any force between the fluid and the boundary must be acting at right angles to the boundary.



Pressure force normal to the boundary Note that this statement is also true for curved surfaces, in this case the force acting at any point is normal to the surface at that point. The statement is also true for any imaginary plane in a static fluid. We use this fact in our analysis by considering elements of fluid bounded by imaginary planes.

We also know that:

- For an element of fluid at rest, the element will be in equilibrium - the sum of the components of forces in any direction will be zero.
- The sum of the moments of forces on the element about any point must also be zero.

It is common to test equilibrium by resolving forces along three mutually perpendicular axes and also by taking moments in three mutually perpendicular planes an to equate these to zero.

Pressure

As mentioned above a fluid will exert a normal force on any boundary it is in contact with. Since these boundaries may be large and the force may differ from place to place it is convenient to work in terms of pressure, p, which is the force per unit area.

If the force exerted on each unit area of a boundary is the same, the pressure is said to be *uniform*.

$$\text{pressure} = \frac{\text{Force}}{\text{Area over which the force is applied}}$$

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

Units:Newton's per square metre, $N m^{-2}$, $kg m^{-1} s^{-2}$.

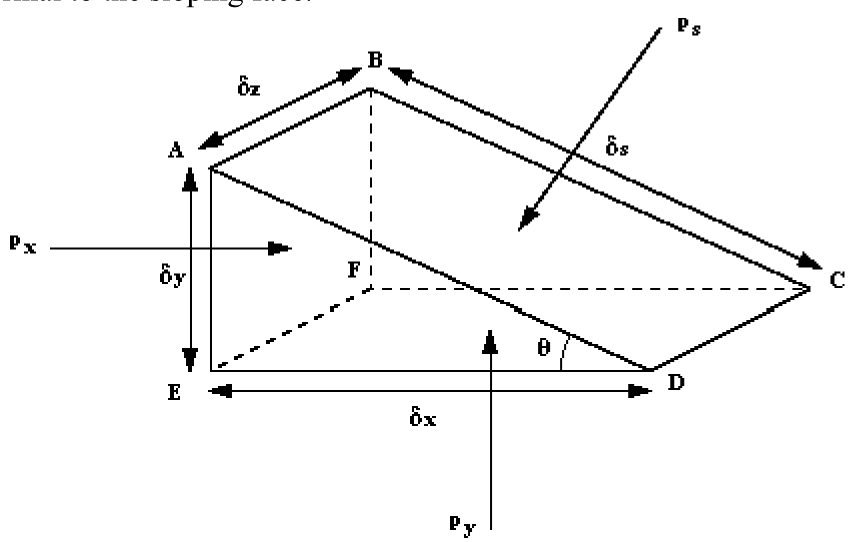
(The same unit is also known as a Pascal, *Pa*, i.e. 1*Pa* = 1 $N m^{-2}$)

(Also frequently used is the alternative SI unit the *bar,* where $1 bar = 10^5 N m^{-2}$)

Dimensions: $ML^{-1}T^{-2}$.

Pascal's Law

By considering a small element of fluid in the form of a triangular prism which contains a point P, we can establish a relationship between the three pressures p_x in the x direction, p_y in the y direction and p_s in the direction normal to the sloping face.



Triangular prismatic element of fluid

The fluid is a rest, so we know there are no shearing forces, and we know that all force are acting at right angles to the surfaces .i.e.

p_s acts perpendicular to surface ABCD,

p_x acts perpendicular to surface ABFE and

p_y acts perpendicular to surface FECD.

And, as the fluid is at rest, in equilibrium, the sum of the forces in any direction is zero.

Summing forces in the x-direction:

Force due to p_x

$$F_{x_x} = p_x \times Area_{ABFE} = p_x \delta x \delta y$$

Component of force in the x-direction due to p_s

$$F_{x_s} = -p_s \times Area_{ABCD} \times \sin\theta$$
$$= -p_s \delta s \delta z \frac{\delta y}{\delta s}$$
$$= -p_s \delta y \delta z$$

Component of force in x-direction due to p_y,

$$F_{x_y} = 0$$

To be at rest (in equilibrium)

$$F_{x_x} + F_{x_s} + F_{x_y} = 0$$
$$p_x \delta x \delta y + \left(-p_s \delta y \delta z\right) = 0$$
$$p_x = p_s$$

Similarly, summing forces in the y-direction. Force due to p_y,

$$F_{y_y} = p_y \times Area_{EFCD} = p_y \delta x \delta z$$

Component of force due to

$$F_{y_s} = -p_s \times Area_{ABCD} \times \cos\theta$$
$$= -p_s \delta s \delta z \frac{\delta x}{\delta s}$$
$$= -p_s \delta x \delta z$$
$$\cos\theta = \delta x / \delta s)$$

Component of force due to p_x; $F_{y_x} = 0$

Force due to gravity,

$$\text{weight} = \text{- specific weight} \times \text{volume of element}$$
$$= -\rho g \times \frac{1}{2} \delta x \delta y \delta z$$

To be at rest (in equilibrium)

$$F_{y_y} + F_{y_s} + F_{y_x} + \text{weight} = 0$$

δx ξ

$$p_y \delta x \delta y + \left(-p_s \delta x \delta z\right) + \left(-\rho g \frac{1}{2} \delta x \delta y \delta z\right) = 0$$

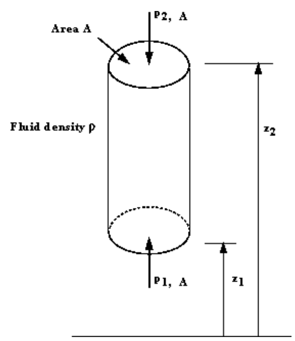
The element is small i.e. , and are small, and so is very small and considered negligible, hence

$$p_x = p_y = p_s \text{thus}$$
$$p_y = p_s$$

Considering the prismatic element again, p_s is the pressure on a plane at any angle θ, the x, y and z directions could be any orientation. The element is so small that it can be considered a point so the derived expression $p_x = p_y = p_s$ indicates that pressure at any point is the same in all directions. (The proof may be extended to include the z axis).

*Pressure at any point is the same in all directions. This is known as **Pascal's Law** and applies to fluids at rest.*

Variation of Pressure Vertically In A Fluid Under Gravity



Vertical elemental cylinder of fluid

In the above figure we can see an element of fluid which is a vertical column of constant cross sectional area, A, surrounded by the same fluid of mass density ρ. The pressure at the bottom of the cylinder is p_1 at level z_1, and at the top is p_2 at level z_2. The fluid is at rest and in equilibrium so all the forces in the vertical direction sum to zero. i.e. we have

$$\begin{aligned} \text{Force due to } p_1 \text{ on A (upward)} &= p_1 A \\ \text{Force due to } p_2 \text{ on A (downward)} &= p_2 A \\ \text{Force due to weight of element (downward)} &= mg \\ &= \text{mass density} \times \text{volume} = \rho g A(z_2 - z_1) \end{aligned}$$

Taking upward as positive, in equilibrium we have

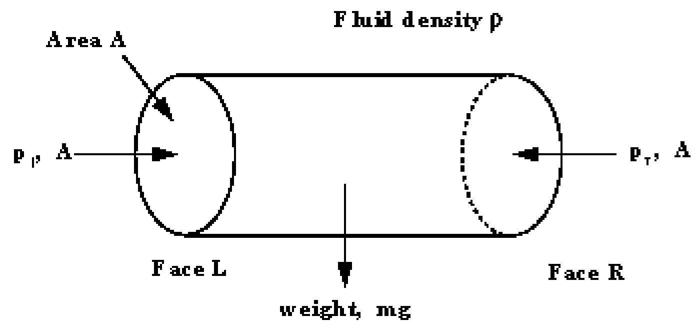
$$p_1 A - p_2 A - \rho g A(z_2 - z_1) = 0$$

$$p_2 - p_1 = -\rho g(z_2 - z_1)$$

Thus in a fluid under gravity, pressure decreases with increase in height $z = (z_2 - z_1)$.

Equality of Pressure At The Same Level In A Static Fluid

Consider the horizontal cylindrical element of fluid in the figure below, with cross-sectional area A, in a fluid of density ρ, pressure p_l at the left hand end and pressure p_r at the right hand end.



Horizontal elemental cylinder of fluid

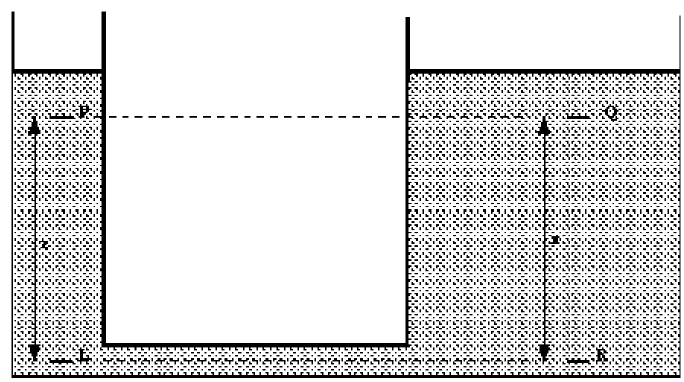
The fluid is at equilibrium so the sum of the forces acting in the x direction is zero.

$$p_l A = p_r A$$

$$p_l = p_r$$

Pressure in the horizontal direction is constant.

This result is the same for any *continuous* fluid. It is still true for two connected tanks which appear not to have any direct connection, for example consider the tank in the figure below.



Two tanks of different cross-section connected by a pipe

We have shown above that $p_l = p_r$ and from the equation for a vertical pressure change we have

$$p_l = p_p + \rho g z$$

and

$$p_r = p_q + \rho g z$$

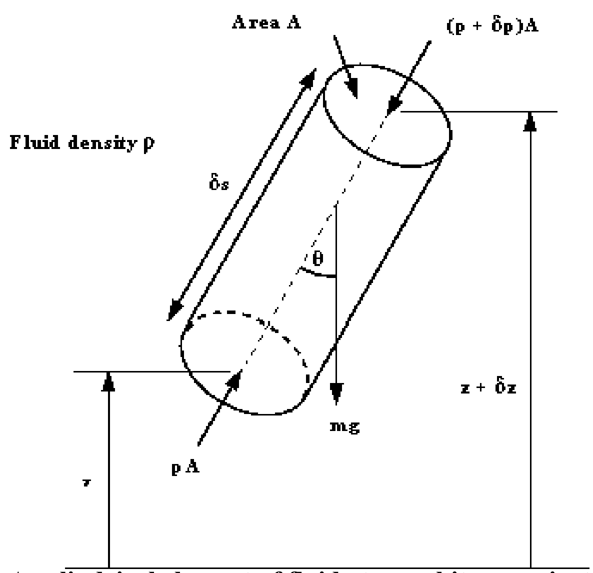
so

$$p_p + \rho g z = p_q + \rho g z$$
$$p_p = p_q$$

This shows that the pressures at the two equal levels, P and Q are the same.

General Equation for Variation of Pressure in a Static Fluid

Here we show how the above observations for vertical and horizontal elements of fluids can be generalised for an element of any orientation.



A cylindrical element of fluid at an arbitrary orientation.

Consider the cylindrical element of fluid in the figure above, inclined at an angle θ to the vertical, length δs, cross-sectional area A in a static fluid of mass density ρ. The pressure at the end with height z is p and at the end of height $z+\delta z$ is $p+\delta p$.

The forces acting on the element are

$$
\begin{aligned}
&pA \quad \text{acting at right-angles to the end of the face at } z \\
&(p+\delta p)A \quad \text{acting at right-angles to the end of the face at } z+\delta z \\
&mg = \text{the weight of the element acting vertically down} \\
&\quad = \text{mass density} \times \text{volume} \times \text{gravity} \\
&\quad = \rho A \delta s g
\end{aligned}
$$

There are also forces from the surrounding fluid acting normal to these sides of the element.

For equilibrium of the element the resultant of forces in any direction is zero.

Resolving the forces in the direction along the central axis gives

$$pA - (p+\delta p)A - \rho g A \delta s \cos\theta = 0$$

$$\delta p = -\rho g \delta s \cos\theta$$

$$\frac{\delta p}{\delta s} = -\rho g \cos\theta$$

Or in the differential form

$$\frac{dp}{ds} = -\rho g \cos\theta$$

If $\theta = 90^\circ$ then s is in the x or y directions, (i.e. horizontal),so

$$\left(\frac{dp}{ds}\right)_{\theta=90^\circ} = \frac{dp}{dx} = \frac{dp}{dy} = 0$$

Confirming that pressure on any horizontal plane is

zero. If $\theta = 0^\circ$ then s is in the z direction (vertical) so

$$\left(\frac{dp}{ds}\right)_{\theta=0^\circ} = \frac{dp}{dz} = -\rho g$$

Confirming the result

$$\frac{p_2 - p_1}{z_2 - z_1} = -\rho g$$

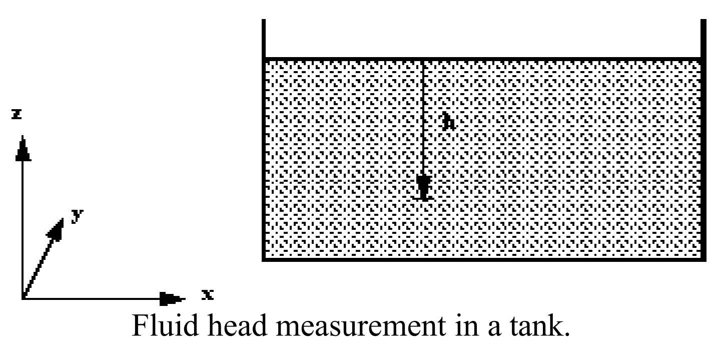
$$p_2 - p_1 = -\rho g(z_2 - z_1)$$

Pressure and Head

In a static fluid of constant density we have the relationship $\frac{dp}{dz} = -\rho g$, as shown above. This can be integrated to give

$$p = -\rho g z + \text{constant}$$

In a liquid with a free surface the pressure at any depth z measured from the free surface so that $z = -h$ (see the figure below)



Fluid head measurement in a tank.

This gives the pressure

$$p = \rho g h + \text{constant}$$

At the surface of fluids we are normally concerned with, the pressure is the atmospheric pressure, $p_{atmospheri}$. So

$$p = \rho g h + p_{atmospheri}$$

As we live constantly under the pressure of the atmosphere, and everything else exists under this pressure, it is convenient (and often done) to take atmospheric pressure as the datum. So we quote pressure as above or below atmospheric.

Pressure quoted in this way is known as gauge pressure i.e.

Gauge pressure is

$$p_{gauge} = \rho g h$$

The lower limit of any pressure is zero - that is the pressure in a perfect vacuum. Pressure measured above this datum is known as absolute pressure i.e.

Absolute pressure is

$$p_{absolute} = \rho g h + p_{atmospheric}$$

$$\text{Absolute pressure} = \text{Gauge prssure} + \text{Atmospheric pressure}$$

As *g* is (approximately) constant, the gauge pressure can be given by stating the vertical height of any fluid of density ρ which is equal to this pressure.

$$p = \rho g h$$

This vertical height is known as **head** of fluid.

Note: If pressure is quoted in *head*, the density of the fluid *must* also be given.

Example:
We can quote a pressure of $500\,K\,N m^{-2}$ in terms of the height of a column of water of $\rho = 1000 kg m^{-3}$. density, Using $p = \rho g h$,

$$h = \frac{p}{\rho g} = \frac{500 \times 10^3}{1000 \times 9.81} = 50.95 m \text{ of water}$$

And in terms of Mercury with density, $\rho = 13.6 \times 10^3 kg m^{-3}$

$$h = \frac{500 \times 10^3}{13.6 \times 10^3 \times 9.81} = 3.75 m \text{ of Mercury}$$

Pressure Measurement

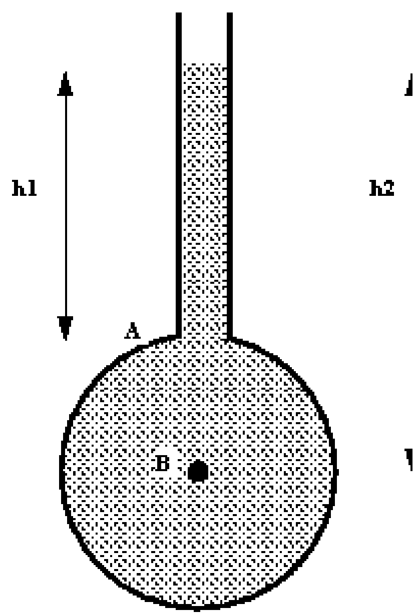
The relationship between pressure and head is used to measure pressure with a manometer (also known as a liquid gauge).

Objective:

- To demonstrate the analysis and use of various types of manometers for pressure measurement.

The Piezometer

The simplest manometer is a tube, open at the top, which is attached to the top of a vessel containing liquid at a pressure (higher than atmospheric) to be measured. An example can be seen in the figure below. This simple device is known as a *Piezometer tube*. As the tube is open to the atmosphere the pressure measured is relative to atmospheric so is **gauge pressure**.



A simple piezometer tube manometer

pressure at A = pressure due to column of liquid above A

$$p_A = \rho g h_1$$

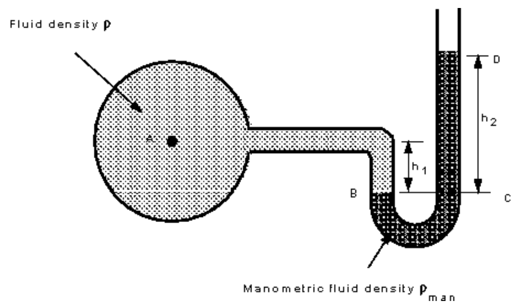
pressure at B = pressure due to column of liquid above B

$$p_B = \rho g h_2$$

This method can only be used for liquids (i.e. **not** for gases) and only when the liquid height is convenient to measure. It must not be too small or too large and pressure changes must be detectable.

The "U"-Tube Manometer

Using a "U"-Tube enables the pressure of both liquids and gases to be measured with the sameinstrument. The "U" is connected as in the figure below and filled with a fluid called the *manometric fluid*. The fluid whose pressure is being measured should have a mass density less than that of the manometric fluid and the two fluids should not be able to mix readily - that is, they must be immiscible.



A "U"-Tube manometer

Pressure in a continuous static fluid is the same at any horizontal level so,

$$\text{pressure at B} = \text{pressure at C}$$
$$p_B = p_C$$

For the **left hand** arm

$$\text{pressure at B} = \text{pressure at A} + \text{pressure due to height } h_1 \text{ of fluid being measured}$$
$$p_B = p_A + \rho g h_1$$

For the **right hand** arm

$$\text{pressure at C} = \text{pressure at D} + \text{pressure due to height } h_2 \text{ of manometric fluid}$$
$$p_C = p_{Amospheric} + \rho_{man} g h_2$$

As we are measuring *gauge pressure* we can subtract $p_{Atmospheri}$ giving

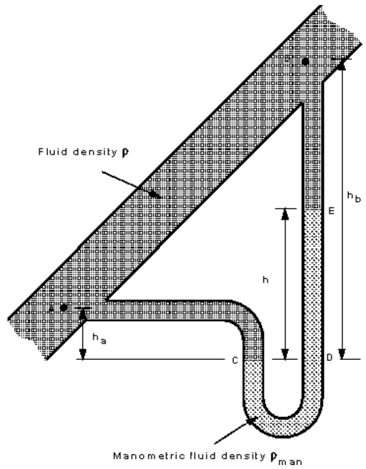
$$p_B = p_C$$
$$p_A = \rho_{man} g h_2 - \rho g h_1$$

If the fluid being measured is a gas, the density will probably be very low in comparison to the density of the manometric fluid i.e. $\rho_{man} >> \rho$. In this case the term $\rho g h_1$ can be neglected, and the gauge pressure give by

$$p_A = \rho_{man} g h_2$$

Measurement of Pressure Difference

If the "U"-tube manometer is connected to a pressurised vessel at two points the *pressure difference* between these two points can be measured.



Pressure difference measurement by the "U"-Tube manometer

If the manometer is arranged as in the figure above, then

$$\text{pressure at C} = \text{pressure at C}$$
$$p_C = p_D$$
$$p_C = p_A + \rho g h_a$$
$$p_D = p_B + \rho g(h_b - h) + \rho_{man} g h$$
$$p_A + \rho g h_a = p_B + \rho g(h_b - h) + \rho_{man} g h$$

Giving the pressure difference

$$p_A - p_B = \rho g(h_b - h_a) + (\rho_{man} - \rho) g h$$

Again, if the fluid whose pressure difference is being measured is a gas and $\rho_{man} >> \rho$, then the terms involving ρ can be neglected, so

$p_A - p_B = \rho_{man} g h$

The Momentum Equation and Its Applications

We have all seen moving fluids exerting forces. The lift force on an aircraft is exerted by the air moving over the wing. A jet of water from a hose exerts a force on whatever it hits. In fluid mechanics the analysis of motion is performed in the same way as in solid mechanics - by use of Newton's laws of motion. Account is also taken for the special properties of fluids when in motion.

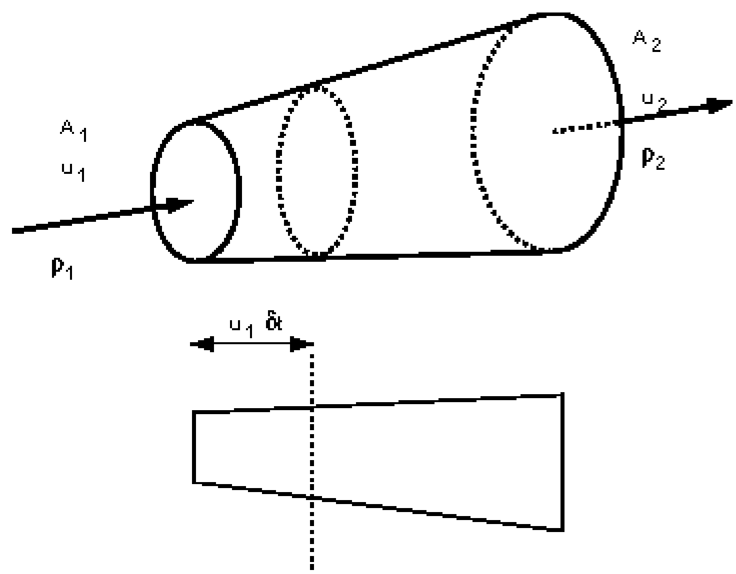
The momentum equation is a statement of Newton's Second Law and relates the sum of the forces acting on an element of fluid to its acceleration or rate of change of momentum. You will probably recognise the equation $F = ma$ which is used in the analysis of solid mechanics to relate applied force to acceleration. In fluid mechanics it is not clear what mass of moving fluid we should use so we use a different form of the equation.

Newton's 2nd Law can be written:

The Rate of change of momentum of a body is equal to the resultant force acting on the body, and takes place in the direction of the force.

To determine the rate of change of momentum for a fluid we will consider a streamtube as we did for the Bernoulli equation,

We start by assuming that we have *steady* flow which is *non-uniform* flowing in a stream tube.



A streamtube in three and two-dimensions

In time δt a volume of the fluid moves from the inlet a distance $u\delta t$, so the volume entering the streamtube in the time δt is

$$\text{volume entering the stream tube} = \text{area} \times \text{distance} = A_1 u_1 \delta t$$

this has mass,

$$\text{mass entering stream tube} = \text{volume} \times \text{density} = \rho_1 A_1 u_1 \delta t$$

and momentum

$$\text{momentum of fluid entering stream tube} = \text{mass} \times \text{velocity} = \rho_1 A_1 u_1 \delta t \, u_1$$

Similarly, at the exit, we can obtain an expression for the momentum leaving the steamtube:

$$\text{momentum of fluid leaving stream tube} = \rho_2 A_2 u_2 \delta t \, u_2$$

We can now calculate the force exerted by the fluid using Newton's 2nd Law. The force is equal to the rate of change of momentum. So

$$\text{Force} = \text{rate of change of momentum}$$

$$F = \frac{(\rho_2 A_2 u_2 \delta t\, u_2 - \rho_1 A_1 u_1 \delta t\, u_1)}{\delta t}$$

We know from continuity that $Q = A_1 u_1 = A_2 u_2$ and if we have a fluid of constant density, i.e. $\rho_1 = \rho_2 = \rho$, then we can write

$$F = Q\rho(u_2 - u_1)$$

For an alternative derivation of the same expression, as we know from conservation of mass in a stream tube that

$$\text{mass into face 1} = \text{mass out of face 2}$$

we can write

$$\text{rate of change of mass} = \dot{m} = \frac{dm}{dt} = \rho_1 A_1 u_1 = \rho_2 A_2 u_2$$

The rate at which momentum leaves face 1 is

$$\rho_2 A_2 u_2 u_2 = \dot{m} u_2$$

The rate at which momentum enters face 2 is

$$\rho_1 A_1 u_1 u_1 = \dot{m} u_1$$

Thus the rate at which momentum changes across the stream tube is

$$\rho_2 A_2 u_2 u_2 - \rho_1 A_1 u_1 u_1 = \dot{m} u_2 - \dot{m} u_1$$

i.e.

$$\text{Force} = \text{rate of change of momentum}$$

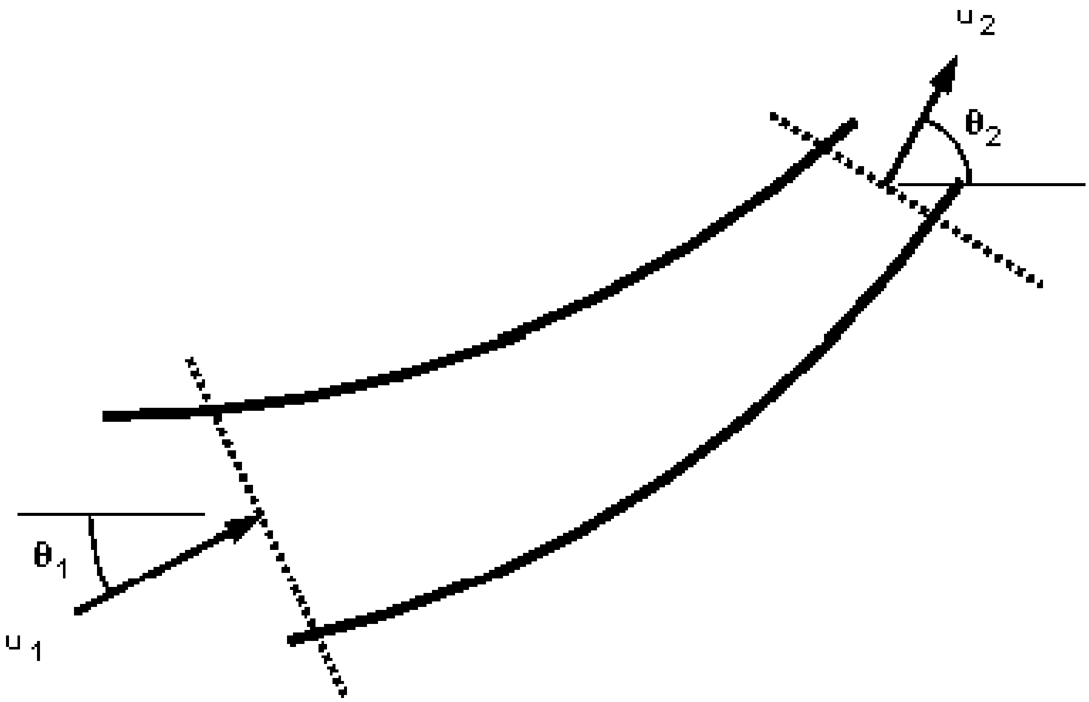
$$F = \dot{m}(u_2 - u_1)$$

$$F = Q\rho(u_2 - u_1)$$

This force is acting in the direction of the flow of the fluid.

This analysis assumed that the inlet and outlet velocities were in the same direction - i.e. a one dimensional system. What happens when this is not the case?

Consider the two dimensional system in the figure below:



Two dimensional flow in a streamtube

At the inlet the velocity vector, u_1 makes an angle, θ_1, with the x-axis, while at the outlet u_2 make an angle θ_2. In this case we consider the forces by resolving in the directions of the co-ordinate axes.

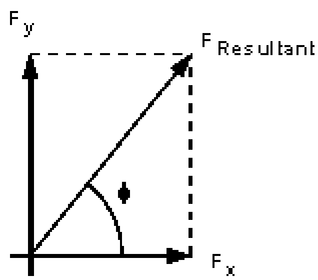
The force in the x-direction

$$
\begin{aligned}
F_x &= \text{Rate of change of momentum in x - direction} \\
&= \text{Rate of change of mass} \times \text{change in velocity in x - direction} \\
&= \dot{m}(u_2 \cos\theta_2 - u_1 \cos\theta_1) \\
&= \dot{m}(u_{2x} - u_{1x}) \\
&= \rho Q(u_2 \cos\theta_2 - u_1 \cos\theta_1) \\
&= \rho Q(u_{2x} - u_{1x})
\end{aligned}
$$

And the force in the y-direction

$$F_y = \dot{m}(u_2 \sin\theta_2 - u_1 \sin\theta_1)$$
$$= \dot{m}(u_{2y} - u_{1y})$$
$$= \rho Q(u_2 \sin\theta_2 - u_1 \sin\theta_1)$$
$$= \rho Q(u_{2y} - u_{1y})$$

We then find the **resultant force** by combining these vectorially:



$$F_{resultant} = \sqrt{F_x^2 + F_y^2}$$

And the angle which this force acts at is given by

$$\phi = \tan^{-1}\left(\frac{F_y}{F_x}\right)$$

For a three-dimensional (x, y, z) system we then have an extra force to calculate and resolve in the z-direction. This is considered in exactly the same way.

In summary we can say:

The total force **exerted on** the fluid = rate of change of momentum through the control volume

$$F = \dot{m}(u_{out} - u_{in})$$
$$= \rho Q(u_{out} - u_{in})$$

Remember that we are working with vectors so F is in the direction of the velocity. This force is made up of three components:

F_R = Force exerted on the fluid by any solid body touching the control

F_B = volume Force exerted on the fluid body (e.g. gravity)

F_P = Force exerted on the fluid by fluid pressure outside the control volume

So we say that the total force, F_T, is given by the sum of these forces:

$$F_T = F_R + F_B + F_P$$

The force exerted **by** the fluid **on** the solid body touching the control volume is opposite to F_R. So the reaction force, R, is given by

$$R = -F_R$$

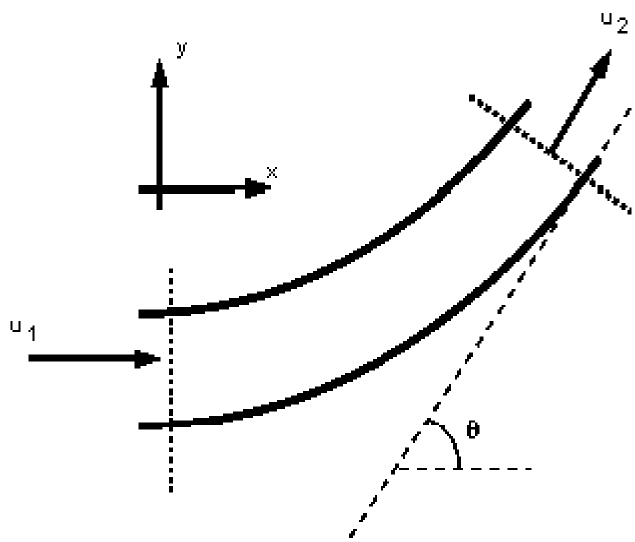
Application of the Momentum Equation

In this section we will consider the following examples:

1. Force due to the flow of fluid round a pipe bend.
2. Force on a nozzle at the outlet of a pipe.
3. Impact of a jet on a plane surface.

1. The force due the flow around a pipe bends

Consider a pipe bend with a constant cross section lying in the horizontal plane and turning through an angle of θ°.



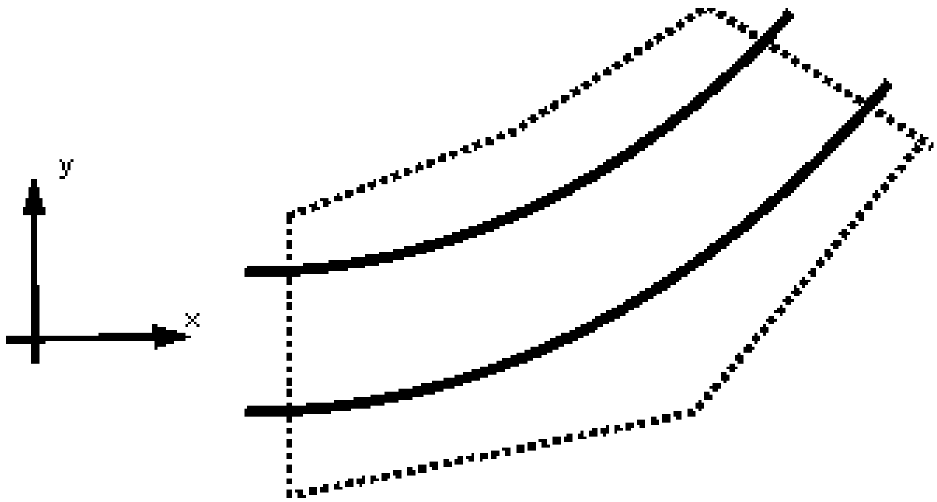
Flow round a pipe bend of constant cross-section

Why do we want to know the forces here? Because the fluid changes direction, a force (very large in the case of water supply pipes,) will act in the bend. If the bend is not fixed it will move and eventually break at the joints. We need to know how much force a support (thrust block) must withstand.

Step in Analysis:

- Draw a control volume
- Decide on co-ordinate axis system
- Calculate the **total** force

- Calculate the **pressure** force
- Calculate the **body** force
- Calculate the **resultant** force



- Control Volume

The control volume is draw in the above figure, with faces at the inlet and outlet of the bend and encompassing the pipe walls.

- Co-ordinate axis system

It is convenient to choose the co-ordinate axis so that one is pointing in the direction of the inlet velocity. In the above figure the x-axis points in the direction of the inlet velocity.

- Calculate the **total** force

In the x-direction:

$$F_{T_x} = \rho Q\left(u_{2_x} - u_{1_x}\right)$$
$$u_{1_x} = u_1$$
$$u_{2_x} = u_2 \cos\theta$$
$$F_{T_x} = \rho Q\left(u_2 \cos\theta - u_1\right)$$

In the y-direction:

$$F_{T_y} = \rho Q\left(u_{2_y} - u_{1_y}\right)$$
$$u_{1_y} = u_1 \sin 0 = 0$$
$$u_{2_y} = u_2 \sin\theta$$
$$F_{T_y} = \rho Q u_2 \sin\theta$$

- Calculate the **pressure** force

$$F_P = \text{pressure force at 1} - \text{pressure force at 2}$$
$$F_{P_x} = p_1 A_1 \cos 0 - p_2 A_2 \cos\theta = p_1 A_1 - p_2 A_2 \cos\theta$$
$$F_{P_y} = p_1 A_1 \sin 0 - p_2 A_2 \sin\theta = -p_2 A_2 \sin\theta$$

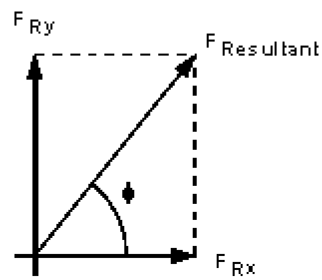
- Calculate the **body** force

There are no body forces in the x or y directions. The only body force is that exerted by gravity (which acts into the paper in this example - a direction we do not need to consider).

- Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{T_y} = F_{R_y} + F_{P_y} + F_{B_y}$$
$$F_{R_x} = F_{T_x} - F_{P_x} - 0 = \rho Q(u_2 \cos\theta - u_1) - p_1 A_1 + p_2 A_2 \cos\theta$$
$$F_{R_y} = F_{T_y} - F_{P_y} - 0 = \rho Q u_2 \sin\theta + p_2 A_2 \sin\theta$$

And the resultant force **on the fluid** is given by



$$F_R = \sqrt{F_{R_x}^2 - F_{R_y}^2}$$

And the direction of application is

$$\phi = \tan^{-1}\left(\frac{F_{R_y}}{F_{R_x}}\right)$$

the force **on the bend** is the same magnitude but in the opposite direction $R = -F_R$

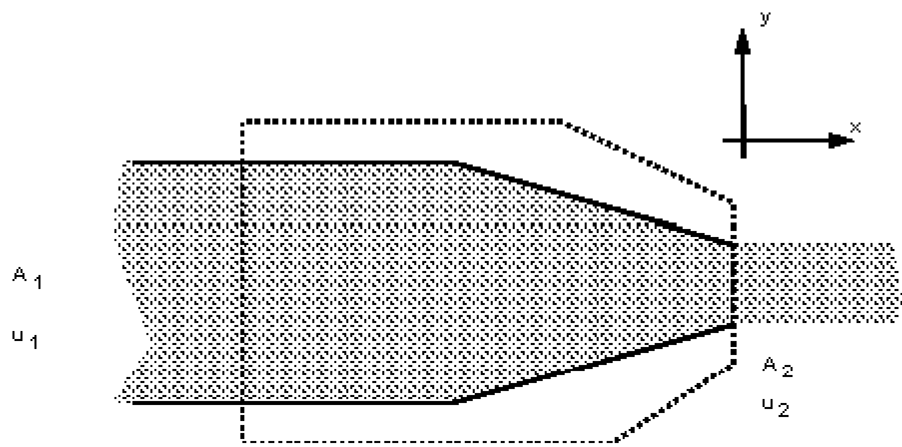
2. Force on a pipe nozzle

Force on the nozzle at the outlet of a pipe. Because the fluid is contracted at the nozzle forces are induced in the nozzle. Anything holding the nozzle (e.g. a fireman) must be strong enough to withstand these forces.

The analysis takes the same procedure as above:

- Draw a control volume
- Decide on co-ordinate axis system
- Calculate the **total** force
- Calculate the **pressure** force
- Calculate the **body** force
- Calculate the **resultant** force

- Control volume and Co-ordinate axis are shown in the figure below.



Notice how this is a one dimensional system which greatly simplifies matters.

- Calculate the **total** force

$$F_T = F_{T_x} = \rho Q(u_2 - u_1)$$

By continuity, $Q = A_1 u_1 = A_2 u_2$, so

$$F_{T_x} = \rho Q^2 \left(\frac{1}{A_2} - \frac{1}{A_1} \right)$$

- Calculate the **pressure** force

$$F_P = F_{P_x} = \text{pressure force at 1 - pressure force at 2}$$

We use the Bernoulli equation to calculate the pressure

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

Is friction losses are neglected, $h_f = 0$

the nozzle is horizontal, $z_1 = z_2$

and the pressure outside is atmospheric, $p_2 = 0$;

and with continuity gives

$$p_1=\frac{\rho Q^2}{2}\left(\frac{1}{A_2^2}-\frac{1}{A_1^2}\right)$$

- Calculate the **body** force

The only body force is the weight due to gravity in the y-direction - but we need not consider this as the only forces we are considering are in the x-direction.

- Calculate the **resultant** force

$$F_{T_x}=F_{R_x}+F_{P_x}+F_{B_x}$$
$$F_{R_x}=F_{T_x}-F_{P_y}-0$$
$$F_{T_x}=\rho Q^2\left(\frac{1}{A_2}-\frac{1}{A_1}\right)-\frac{\rho Q^2}{2}\left(\frac{1}{A_2^2}-\frac{1}{A_1^2}\right)$$

So the fireman must be able to resist the force of

$$R=-F_{T_x}$$

Uniform Flow, Steady Flow

It is possible - and useful - to classify the type of flow which is being examined into small number of groups. If we look at a fluid flowing under normal circumstances - a river for example - the conditions at one point will vary from those at another point (e.g. different velocity) we have non-uniform flow. If the conditions at one point vary as time passes then we have unsteady flow.

Under some circumstances the flow will not be as changeable as this. He following terms describes the states which are used to classify fluid flow:

- *Uniform flow:* If the flow velocity is the same magnitude and direction at every point in the fluid it is said to be *uniform*.
- *Non-uniform:* If at a given instant, the velocity is **not** the same at every point the flow is *non-uniform*. (In practice, by this definition, every fluid that flows near a solid boundary will be non-uniform - as the fluid at the boundary must take the speed of the boundary, usually zero. However if the size and shape of the of the cross-section of the stream of fluid is constant the flow is considered *uniform*.)
- *Steady:* A steady flow is one in which the conditions (velocity, pressure and cross-section) may differ from point to point but DO NOT change with time.
- *Unsteady:* If at any point in the fluid, the conditions change with time, the flow is described as *unsteady*. (In practise there is always slight variations in velocity and pressure, but if the average values are constant, the flow is considered *steady*.

Combining the above we can classify any flow in to one of four types:

1. *Steady uniform flow*. Conditions do not change with position in the stream or with time. An example is the flow of water in a pipe of constant diameter at constant velocity.
2. *Steady non-uniform flow.* Conditions change from point to point in the stream but do not change with time. An example is flow in a tapering pipe with constant velocity at the inlet - velocity will change as you move along the length of the pipe toward the exit.
3. *Unsteady uniform flow*. At a given instant in time the conditions at every point are the same, but will change with time. An example is a pipe of constant diameter connected to a pump pumping at a constant rate which is then switched off.

4. *Unsteady non-uniform flow.* Every condition of the flow may change from point to point and with time at every point. For example waves in a channel.

Compressible or Incompressible

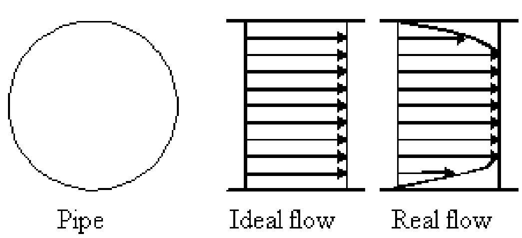
All fluids are compressible - even water - their density will change as pressure changes. Under steady conditions, and provided that the changes in pressure are small, it is usually possible to simplify analysis of the flow by assuming it is incompressible and has constant density. As you will appreciate, liquids are quite difficult to compress - so under most steady conditions they are treated as incompressible. In some unsteady conditions very high pressure differences can occur and it is necessary to take these into account

- even for liquids. Gasses, on the contrary, are very easily compressed, it is essential in most cases to treat these as compressible, taking changes in pressure into account.

Three-dimensional flow

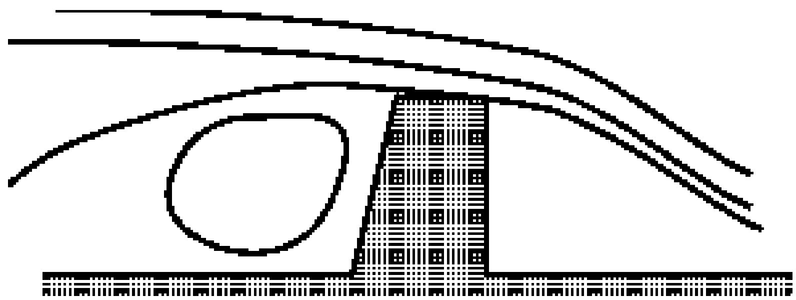
Although in general all fluids flow three-dimensionally, with pressures and velocities and other flow properties varying in all directions, in many cases the greatest changes only occur in two directions or even only in one. In these cases changes in the other direction can be effectively ignored making analysis much more simple.

Flow is *one dimensional* if the flow parameters (such as velocity, pressure, depth etc.) at a given instant in time only vary in the direction of flow and not across the cross-section. The flow may be unsteady, in this case the parameter vary in time but still not across the cross-section. An example of one-dimensional flow is the flow in a pipe. Note that since flow must be zero at the pipe wall - yet non-zero in the centre - there is a difference of parameters across the cross-section. Should this be treated as two-dimensional flow? Possibly - but it is only necessary if very high accuracy is required. A correction factor is then usually applied.



One dimensional flow in a pipe.

Flow is *two-dimensional* if it can be assumed that the flow parameters vary in the direction of flow and in one direction at right angles to this direction. Streamlines in two-dimensional flow are curved lines on a plane and are the same on all parallel planes. An example is flow over a weir foe which typical streamlines can be seen in the figure below. Over the majority of the length of the weir the flow is the same - only at the two ends does it change slightly. Here correction factors may be applied.

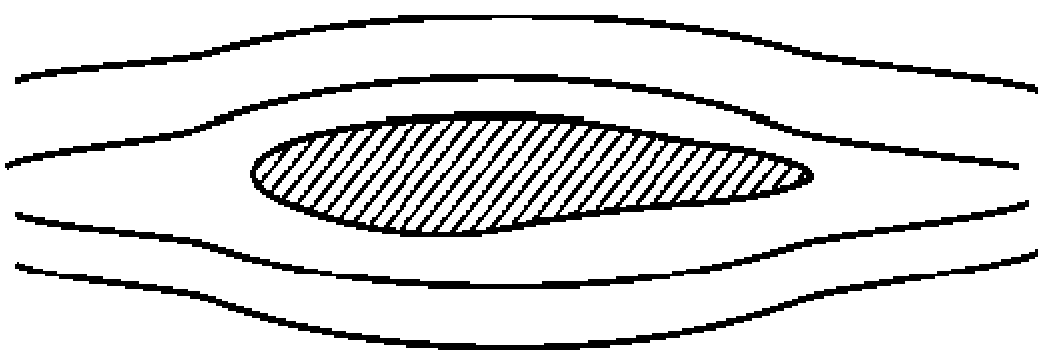


Two-dimensional flow over a weir.

In this course we will **only** be considering steady, incompressible one and two-dimensional flow.

Streamlines and stream tubes

In analysing fluid flow it is useful to visualise the flow pattern. This can be done by drawing lines joining points of equal velocity - velocity contours. These lines are know as *streamlines*. Here is a simple example of the streamlines around a cross-section of an aircraft wing shaped body:



Streamlines around a wing shaped body

When fluid is flowing past a solid boundary, e.g. the surface of an aerofoil or the wall of a pipe, fluid obviously does not flow into or out of the surface. So very close to a boundary wall the flow direction must be parallel to the boundary.

Close to a solid boundary streamlines are parallel to that boundary

At all points the direction of the streamline is the direction of the fluid velocity: this is how they are defined. Close to the wall the velocity is parallel to the wall so the streamline is also parallel to the wall. It is also important to recognize that the position of streamlines can change with time - this is the case in unsteady flow. In steady flow, the position of streamlines does not change.

Some things to know about streamlines

- Because the fluid is moving in the same direction as the streamlines, fluid can not cross a streamline.
- Streamlines cannot cross each other. If they were to cross this would indicate two different velocities at the same point. This is not physically possible.
- The above point implies that any particles of fluid starting on one streamline will stay on that same streamline throughout the fluid.

Continuity and Conservation of Matter

1. Mass flow rate

If we want to measure the rate at which water is flowing along a pipe. A very simple way of doing this is to catch all the water coming out of the pipe in a bucket over a fixed time period. Measuring the weight of the water in the bucket and dividing this by the time taken to collect this water gives a rate of accumulation of mass. This is know as the *mass flow rate.*

For example an empty bucket weighs 2.0kg. After 7 seconds of collecting water the bucket weighs 8.0kg, then:

$$\begin{aligned} \text{mass flow rate} = \dot{m} &= \frac{\text{mass of fluid in bucket}}{\text{time taken to collect the fluid}} \\ &= \frac{8.0 - 2.0}{7} \\ &= 0.857 kg/s \quad (kg s^{-1}) \end{aligned}$$

Performing a similar calculation, if we know the mass flow is 1.7kg/s, how long will it take to fill a container with 8kg of fluid?

$$\begin{aligned} \text{time} &= \frac{\text{mass}}{\text{mass flow rate}} \\ &= \frac{8}{1.7} \\ &= 4.7 s \end{aligned}$$

2. Volume flow rate - Discharge.

More commonly we need to know the volume flow rate - this is more commonly know as *discharge.* (It is also commonly, but inaccurately, simply called flow rate). The symbol normally used for discharge is *Q*. The discharge is the volume of fluid flowing per unit time. Multiplying this by the density of the fluid gives us the mass flow rate. Consequently, if the density of the fluid in the above example is 850 $kg m^3$ then:

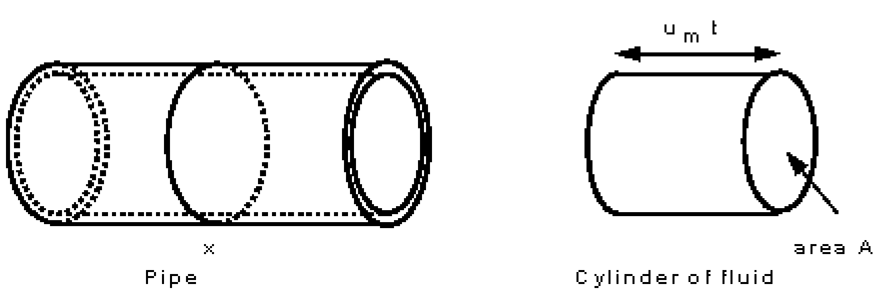
$$\text{discharge}, \quad Q = \frac{\text{volume of fluid}}{\text{time}} = \frac{\text{mass of fluid}}{\text{density} \times \text{time}} = \frac{\text{mass flow rate}}{\text{density}} = \frac{0.857}{850} = 0.001008\, m^3/s \quad (m^3 s^{-1}) = 1.008 \times 10^{-3} m^3/s = 1.008\, l/s$$

An important aside about units should be made here:

As has already been stressed, we must always use a consistent set of units when applying values to equations. It would make sense therefore to always quote the values in this consistent set. This set of units will be the SI units. Unfortunately, and this is the case above, these actual practical values are very small or very large (0.001008m^3/s is very small). These numbers are difficult to imagine physically. In these cases it is useful to use *derived units*, and in the case above the useful derived unit is the litre. (*1 litre = 1.0* $10^{-3}m^3$). So the solution becomes $1.008\, l/s$. It is far easier to imagine 1 litre than *1.0* $10^{-3}m^3$. Units must always be checked, and converted if necessary to a consistent set before using in an equation.

3. Discharge and mean velocity

If we know the size of a pipe, and we know the discharge, we can deduce the mean velocity



Discharge in a pipe

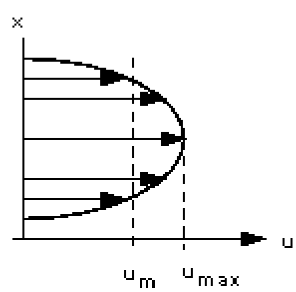
If the area of cross section of the pipe at point X is A, and the mean velocity here is u_m. During a time t, a cylinder of fluid will pass point X with a volume u_m t. The volume per unit time (the discharge) willthus be

$$Q = \frac{\text{volume}}{\text{time}} = \frac{A \times u_m \times t}{t}$$
$$Q = Au_m$$

So if the cross-section area, A, is $1.2 \times 10^{-3} m^2$ and the discharge, Q is $24 l/s$, then the mean velocity, u_m, of the fluid is

$$\begin{aligned} u_m &= \frac{Q}{A} \\ &= \frac{2.4 \times 10^{-3}}{1.2 \times 10^{-3}} \\ &= 2.0 m/s \end{aligned}$$

Note how carefully we have called this the *mean* velocity. This is because the velocity in the pipe is not constant across the cross section. Crossing the centreline of the pipe, the velocity is zero at the walls increasing to a maximum at the centre then decreasing symmetrically to the other wall. This variation across the section is known as the velocity profile or distribution. A typical one is shown in the figure below.



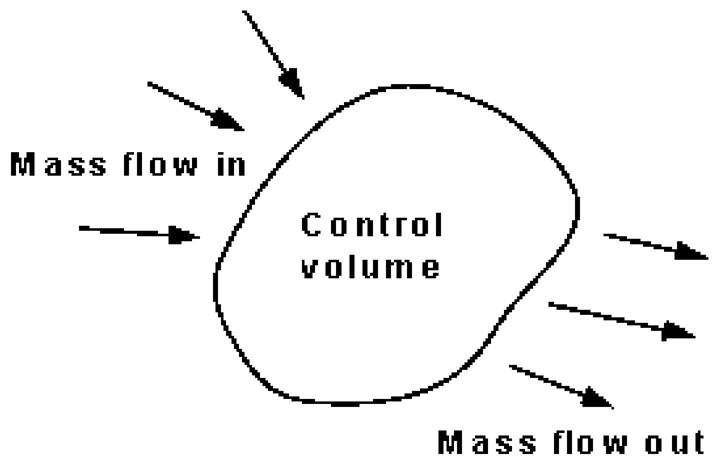
A typical velocity profile across a pipe

This idea, that mean velocity multiplied by the area gives the discharge, applies to all situations - not just pipe flow.

4. Continuity

Matter cannot be created or destroyed - (it is simply changed in to a different form of matter). This principle is know as the *conservation of mass* and we use it in the analysis of flowing fluids.

The principle is applied to fixed volumes, known as control volumes (or surfaces), like that in the figure below:



An arbitrarily shaped control volume.

For any control volume the principle of *conservation of mass* says

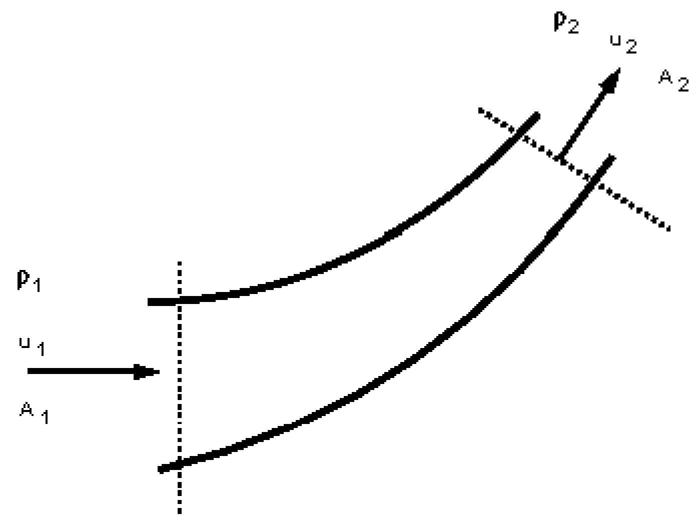
Mass entering per unit time = Mass leaving per unit time + Increase of mass in the control volume per unit time

For **steady** flow there is no increase in the mass within the control volume, so

For steady flow

Mass entering per unit time = Mass leaving per unit time

This can be applied to a streamtube such as that shown below. No fluid flows across the boundary made by the streamlines so mass only enters and leaves through the two ends of this streamtube section.



A streamtube

We can then write

$$\text{mass entering per unit time at end 1} = \text{mass leaving per unit time at end 2}$$

$$\rho_1 \delta A_1 u_1 = \rho_2 \delta A_2 u_2$$

Or for steady flow,

$$\rho_1 \delta A_1 u_1 = \rho_2 \delta A_2 u_2 = \text{Constant} = \dot{m}$$

This is the equation of continuity.

The flow of fluid through a real pipe (or any other vessel) will vary due to the presence of a wall - in this case we can use the *mean* velocity and write

$$\rho_1 A_1 u_{m1} = \rho_2 A_2 u_{m2} = \text{Constant} = \dot{m}$$

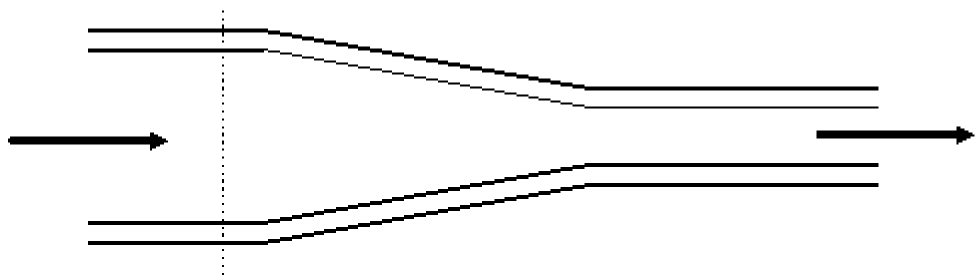
When the fluid can be considered incompressible, i.e. the density does not change, (dropping the *m* subscript) $\rho_1 = \rho_2 = \rho$ so

$$A_1 u_1 = A_2 u_2 = Q$$

This is the form of the continuity equation most often used. This equation is a very powerful tool in fluid mechanics and will be used **repeatedly** throughout the rest of this course.

Some example applications

We can apply the principle of continuity to pipes with cross sections which change along their length. Consider the diagram below of a pipe with a contraction:



A liquid is flowing from left to right and the pipe is narrowing in the same direction. By the continuity principle, the *mass flow rate* must be the same at each section - the mass going into the pipe is equal to the mass going out of the pipe. So we can write:

$$A_1 u_1 \rho_1 = A_2 u_2 \rho_2$$

(with the sub-scripts 1 and 2 indicating the values at the two sections)

As we are considering a liquid, usually water, which is *not* very compressible, the density changes very little so we can say $\rho_1 = \rho_2 = \rho$. This also says that the *volume flow rate* is constant or that

$$\text{Discharge at section 1} = \text{Discharge at section 2}$$
$$Q_1 = Q_2$$
$$A_1 u_1 = A_2 u_2$$

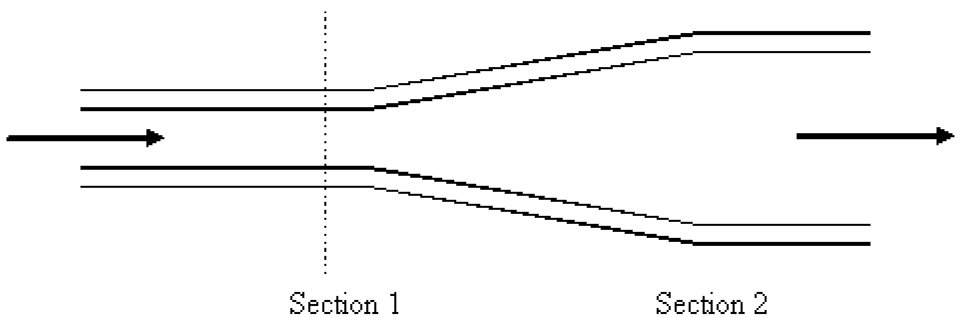
For example if the area $A_1 = 10 \times 10^{-3} m^2$ and $A_2 = 3 \times 10^{-3} m^2$ and the upstream mean velocity, $u_1 = 2.1 m/s$, then the downstream mean velocity can be calculated by

$$u_2 = \frac{A_1 u_1}{A_2}$$
$$= 7.0 m/s$$

Notice how the downstream velocity only changes from the upstream by the ratio of the two areas of the pipe. As the area of the circular pipe is a function of the diameter we can reduce the calculation further,

$$u_2 = \frac{A_1}{A_2}u_1 = \frac{\pi d_1^2/4}{\pi d_2^2/4}u_1 = \frac{d_1^2}{d_2^2}u_1$$
$$= \left(\frac{d_1}{d_2}\right)^2 u_1$$

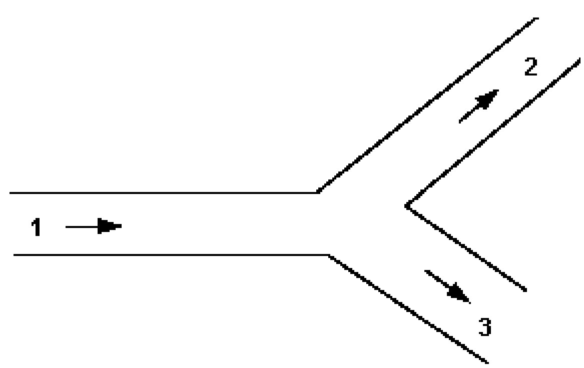
Now try this on a *diffuser,* a pipe which expands or diverges as in the figure below,



If the diameter at section 1 is $d_1 = 30mm$ and at section 2 $d_2 = 40mm$ and the mean velocity at section 2 is $u_2 = 3.0m/s$. The velocity entering the diffuser is given by,

$$u_1 = \left(\frac{40}{30}\right)^2 3.0$$
$$= 5.3m/s$$

Another example of the use of the continuity principle is to determine the velocities in pipes coming from a junction.



Total mass flow into the junction = Total mass flow out of the junction

$$\rho Q_1 = \rho Q_2 + \rho Q_3$$

When the flow is incompressible (e.g. if it is water) $\rho_1 = \rho_2 = \rho$

$$Q_1 = Q_2 + Q_3$$
$$A_1 u_1 = A_2 u_2 + A_3 u_3$$

If pipe 1 diameter = 50mm, mean velocity 2m/s, pipe 2 diameter 40mm takes 30% of total discharge and pipe 3 diameter 60mm. What are the values of discharge and mean velocity in each pipe?

$$Q_1 = A_1 u_1 = \left(\frac{\pi d^2}{4}\right) u$$
$$= 0.00392 m^3 / s$$

$$Q_2 = 0.3 Q_1 = 0.001178 m^3 / s$$
$$Q_1 = Q_2 + Q_3$$
$$Q_3 = Q_1 - 0.3 Q_1 = 0.7 Q_1$$
$$= 0.00275 m^3 / s$$

$$Q_2 = A_2 u_2$$
$$u_2 = 0.936 m / s$$

$$Q_3 = A_3 u_3$$
$$u_3 = 0.972 m / s$$

The Bernoulli equation

1. Work and energy

We know that if we drop a ball it accelerates downward with an acceleration $g = 9.81 m/s^2$ (neglecting the frictional resistance due to air). We can calculate the speed of the ball after falling a distance *h* by the formula $v^2 = u^2 + 2as$ (*a* = *g* and *s* = *h*). The equation could be applied to a falling droplet of water as the same laws of motion apply

A more general approach to obtaining the parameters of motion (of both solids and fluids) is to apply the principle of ***conservation of energy***. When friction is negligible the

Sum of kinetic energy and gravitational potential energy is constant.

Kinetic energy $= \frac{1}{2}mv^2$

Gravitational potential energy $= mgh$

m is the mass, v is the velocity and h is the height above the datum

To apply this to a falling droplet we have an initial velocity of zero, and it falls through a height of h.

Initial kinetic energy $= 0$

Initial potential energy $= mgh$

Final kinetic energy $= \frac{1}{2}mv^2$

Final potential energy $= 0$

We know that

kinetic energy + potential energy = constant, so

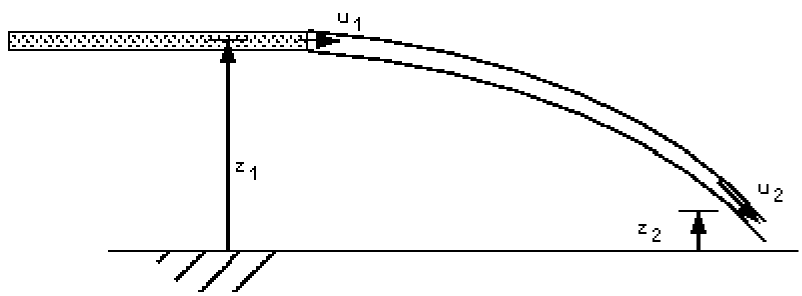
Initial kinetic energy + Initial potential energy = Final kinetic energy + Final potential energy

$$mgh = \frac{1}{2}mv^2$$

so

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

Although this is applied to a drop of liquid, a similar method can be applied to a **continuous jet** of liquid.



The Trajectory of a jet of water

We can consider the situation as in the figure above - a continuous jet of water coming from a pipe with velocity u. One particle of the liquid with mass m travels with the jet and falls from height z_1 to z_2. The velocity also changes from u_1 to u_2. The jet is travelling in air where the pressure is everywhere atmospheric so there is no force due to pressure acting on the fluid. The only force which is acting is thatdue to gravity. The sum of the kinetic and potential energies remains constant (as we neglect energy losses due to friction) so

$$mgz_1 + \frac{1}{2}mu_1^2 = mgz_2 + \frac{1}{2}mu_2^2$$

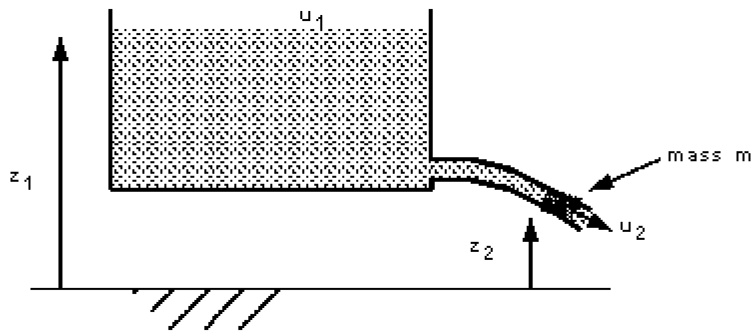
As m is constant this becomes

$$\frac{1}{2}u_1^2 + gz_1 = \frac{1}{2}u_2^2 + gz_2$$

This will give a reasonably accurate result as long as the weight of the jet is large compared to the frictional forces. It is only applicable while the jet is whole - before it breaks up into droplets.

2. Flow from a reservoir

We can use a very similar application of the energy conservation concept to determine the velocity of flow along a pipe from a reservoir. Consider the 'idealized reservoir' in the figure below.



An idealized reservoir

The level of the water in the reservoir is z_1. Considering the energy situation - there is no movement of water so kinetic energy is zero but the gravitational potential energy is mgz_1.

If a pipe is attached at the bottom water flows along this pipe out of the tank to a level z_2. A mass m has flowed from the top of the reservoir to the nozzle and it has gained a velocity u_2. The kinetic energy is now $\frac{1}{2}mu_2^2$ and the potential energy mgz_2. Summarizing

Initial kinetic energy $= 0$

Initial potential energy $= mgz_1$

Final kinetic energy $= \frac{1}{2}mu_2^2$

Final potential energy $= mgz_2$

We know that

kinetic energy + potential energy =

constant so

$$mgz_1 = \frac{1}{2}mu_2^2 + mgz_2$$
$$mg(z_1 + z_2) = \frac{1}{2}mu_2^2$$

so

$$u_2 = \sqrt{2g(z_1 - z_2)}$$

We now have a expression for the velocity of the water as it flows from of a pipe nozzle at a height $(z_1 - z_2)$ below the surface of the reservoir. (Neglecting friction losses in the pipe and the nozzle).

Now apply this to this example: A reservoir of water has the surface at 310m above the outlet nozzle of a pipe with diameter 15mm. What is the a) velocity, b) the discharge out of the nozzle and c) mass flow rate. (Neglect all friction in the nozzle and the pipe).

$$\begin{aligned} u_2 &= \sqrt{2g(z_1 - z_2)} \\ &= \sqrt{2 \times g \times 310} \\ &= 78.0\, m/s \end{aligned}$$

Volume flow rate is equal to the area of the nozzle multiplied by the velocity

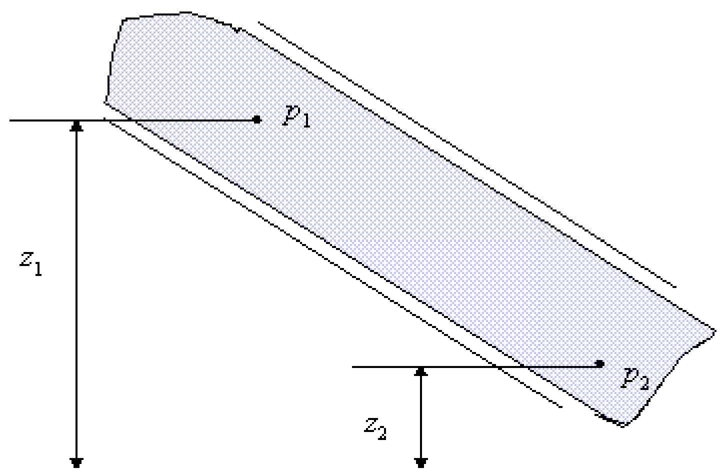
$$\begin{aligned} Q &= Au \\ &= (\pi \times \tfrac{0.015^2}{4}) \times 78.0 \\ &= 0.01378\, m^3/s \end{aligned}$$

The density of water is $1000 kg/m^3$ so the mass flow rate is

$$\begin{aligned} \text{mass flow rate} &= \text{density} \times \text{volume flow rate} \\ &= \rho Q \\ &= 1000 \times 0.01378 \\ &= 13.78\, kg/s \end{aligned}$$

In the above examples the resultant pressure force was always zero as the pressure surrounding the fluid was the everywhere the same - atmospheric. If the pressures had been different there would have been an extra force acting and we would have to take into account the work done by this force when calculating the final velocity.

We have already seen in the hydrostatics section an example of pressure difference where the velocities are zero.



The pipe is filled with stationary fluid of density ρ has pressures p_1 and p_2 at levels z_1 and z_2 respectively. What is the pressure difference in terms of these levels?

$$p_2 - p_1 = \rho g(z_1 - z_2)$$

or

$$\frac{p_1}{\rho} + gz_1 = \frac{p_2}{\rho} + gz_2$$

This applies when the pressure varies but the fluid is stationary.

Compare this to the equation derived for a moving fluid but constant pressure:

$$\frac{1}{2}u_1^2 + gz_1 = \frac{1}{2}u_2^2 + gz_2$$

You can see that these are similar form. What would happen if both pressure and velocity varies?

3. Bernoulli's Equation

Bernoulli's equation is one of the most important/useful equations in fluid mechanics. It may be written,

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

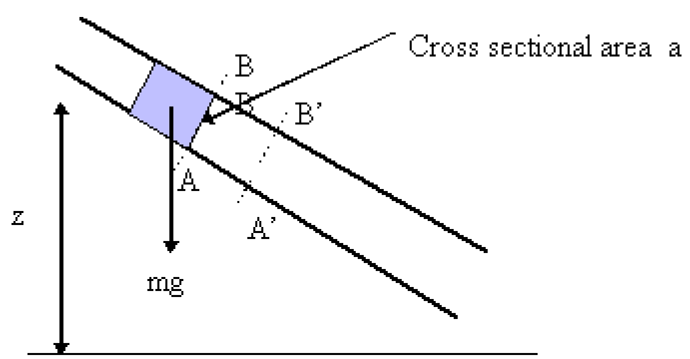
We see that from applying equal pressure or zero velocities we get the two equations from the section above. They are both just special cases of Bernoulli's equation.

Bernoulli's equation has some restrictions in its applicability, they are:

- *Flow is steady;*
- *Density is constant (which also means the fluid is incompressible);*
- *Friction losses are negligible.*
- *The equation relates the states at two points along a single streamline, (not conditions on two different streamlines).*

All these conditions are impossible to satisfy at any instant in time! Fortunately for many real situations where the conditions are *approximately* satisfied, the equation gives very good results.

The derivation of Bernoulli's Equation:



An element of fluid, as that in the figure above, has potential energy due to its height z above a datum and kinetic energy due to its velocity u. If the element has weight mg then

potential energy = mgz potential energy per unit weight = z

kinetic energy = $\frac{1}{2}mu^2$ kinetic energy per unit weight = $\frac{u^2}{2g}$

At any cross-section the pressure generates a force, the fluid will flow, moving the cross-section, so work will be done. If the pressure at cross section AB is *p* and the area of the cross-section is *a* then

force on AB = pa

when the mass *mg* of fluid has passed AB, cross-section AB will have moved to A'B'

volume passing AB = $\frac{mg}{\rho g} = \frac{m}{\rho}$ therefore

distance AA' = $\frac{m}{\rho a}$

work done = force distance AA' = $pa \times \frac{m}{\rho a} = \frac{pm}{\rho}$

work done per unit weight = $\frac{p}{\rho g}$

This term is know as the pressure energy of the flowing stream.

Summing all of these energy terms gives

$$\begin{matrix}\text{Pressure} \\ \text{energy per} \\ \text{unit weight}\end{matrix} + \begin{matrix}\text{Kinetic} \\ \text{energy per} \\ \text{unit weight}\end{matrix} + \begin{matrix}\text{Potential} \\ \text{energy per} \\ \text{unit weight}\end{matrix} = \begin{matrix}\text{Total} \\ \text{energy per} \\ \text{unit weight}\end{matrix}$$

or

$$\frac{p}{\rho g} + \frac{u^2}{2g} + z = H$$

As all of these elements of the equation have units of length, they are often referred to as the following:

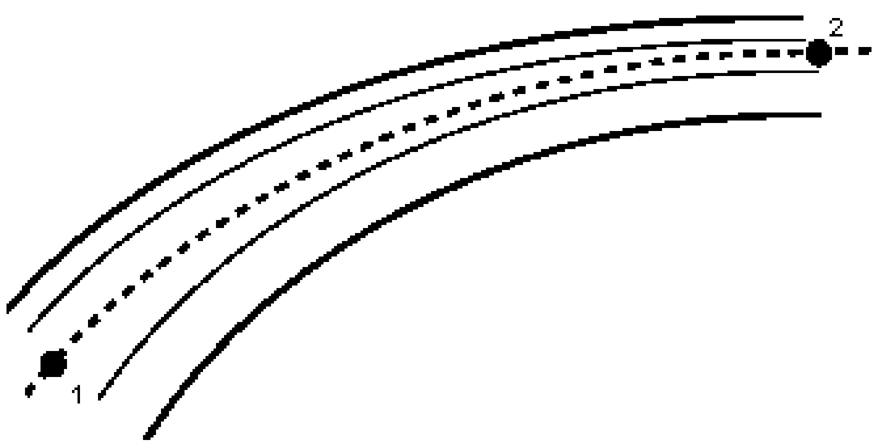
Pressure head = $\frac{p}{\rho g}$ velocity head = $\frac{u^2}{2g}$ potential head = Z

Total head = H

By the principle of conservation of energy the total *energy* in the system does not change, Thus the total *head* does not change. So the Bernoulli equation can be written

$$\frac{p}{\rho g}+\frac{u^2}{2g}+z = H = \text{Constant}$$

As stated above, the Bernoulli equation applies to conditions along a streamline. We can apply it between two points, 1 and 2, on the streamline in the figure below



Two points joined by a streamline

Total energy per unit weight at 1 = total energy per unit weight at 2

Total head at 1 = total head at 2

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

This equation assumes no energy losses (e.g. from friction) or energy gains (e.g. from a pump) along the streamline. It can be expanded to include these simply, by adding the appropriate energy terms:

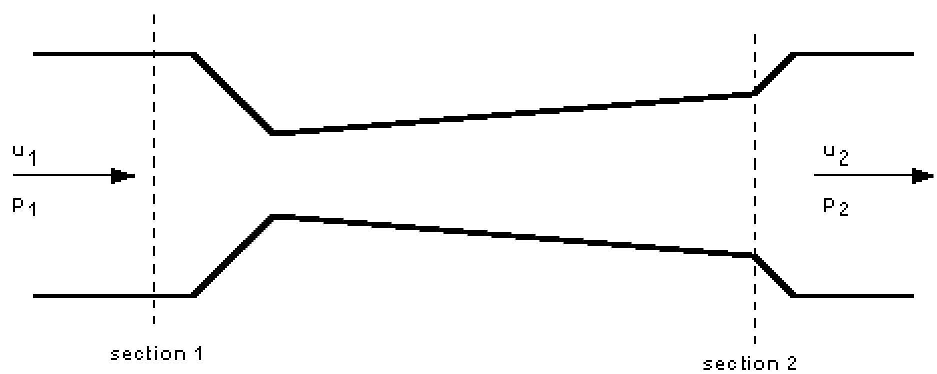
Total energy per unit weight at 1	=	Total energy per unit weight at 2	+	Loss per unit weight	+	Work done per unit weight	−	Energy supplied per unit weight

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

4. An example of the use of the Bernoulli equation.

When the Bernoulli equation is combined with the continuity equation the two can be used to find velocities and pressures at points in the flow connected by a streamline.

Here is an example of using the Bernoulli equation to determine pressure and velocity at within a contracting and expanding pipe.



A contracting expanding pipe

A fluid of constant density = 960 kg/m^3 is flowing steadily through the above tube. The diameters at the sections are $d_1 = 100mm$ and $d_2 = 80mm$. The gauge pressure at 1 is $p_1 = 200kN/m^2$ and the velocity here is $u_1 = 5m/s$. We want to know the gauge pressure at section 2.

We shall of course use the Bernoulli equation to do this and we apply it along a streamline joining section 1 with section 2.

The tube is horizontal, with $z_1 = z_2$ so Bernoulli gives us the following equation for pressure at section 2:

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

But we do not know the value of u_2. We can calculate this from the continuity equation: Discharge intothe tube is equal to the discharge out i.e.

$$A_1 u_1 = A_2 u_2$$

$$u_2 = \frac{A_1 u_1}{A_2}$$

$$u_2 = \left(\frac{d_1}{d_2}\right)^2 u_1$$

$$= 7.8125 m/s$$

So we can now calculate the pressure at section 2

$$p_2 = 200000 - 17296.87$$

$$= 182703 \ N/m^2$$

$$= 182.7 \ kN/m^2$$

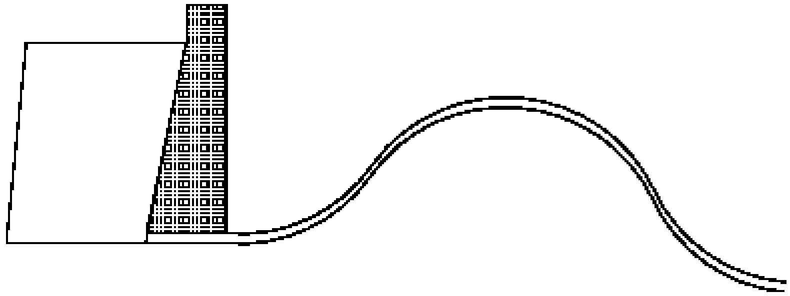
Notice how the velocity has increased while the pressure has decreased. The phenomenon - that pressure decreases as velocity increases - sometimes comes in very useful in engineering. (It is on this principle that carburettor in many car engines work - pressure reduces in a contraction allowing a small amount of fuel to enter).

Here we have used both the Bernoulli equation and the Continuity principle together to solve the problem. Use of this combination is very common. We will be seeing this again frequently throughout the rest of the course.

5. Pressure Head, Velocity Head, Potential Head and Total Head.

By looking again at the example of the reservoir with which feeds a pipe we will see how these different *heads* relate to each other.

Consider the reservoir below feeding a pipe which changes diameter and rises (in reality it may have to pass over a hill) before falling to its final level.



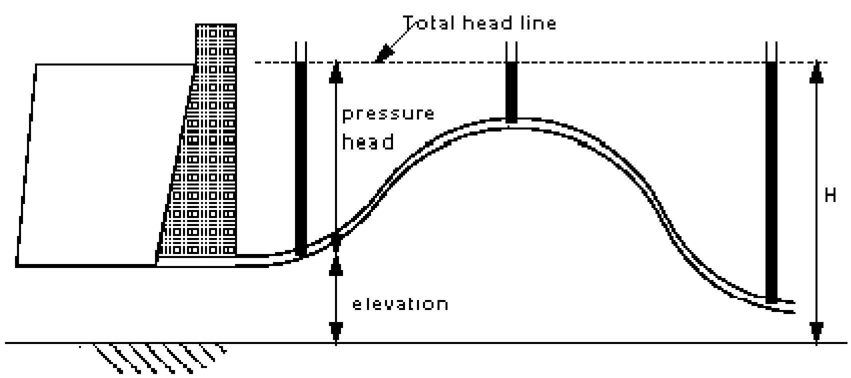
Reservoir feeding a pipe

To analyses the flow in the pipe we apply the Bernoulli equation along a streamline from point 1 on the surface of the reservoir to point 2 at the outlet nozzle of the pipe. And we know that the *total energy per unit weight* or the *total head* does not change - it is **constant** - along a streamline. But what is this value of this constant? We have the Bernoulli equation

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

We can calculate the total head, *H*, at the reservoir, $p_1 = 0$ as this is atmospheric and atmospheric gauge pressure is zero, the surface is moving very slowly compared to that in the pipe so $u_1 = 0$, so all we are left with is $total\,head = H = z_1$ the elevation of the reservoir.

A useful method of analysing the flow is to show the pressures graphically on the same diagram as the pipe and reservoir. In the figure above the *total head* line is shown. If we attached piezometers at points along the pipe, what would be their levels when the pipe nozzle was closed? (Piezometers, as you will remember, are simply open ended vertical tubes filled with the same liquid whose pressure they are measuring).



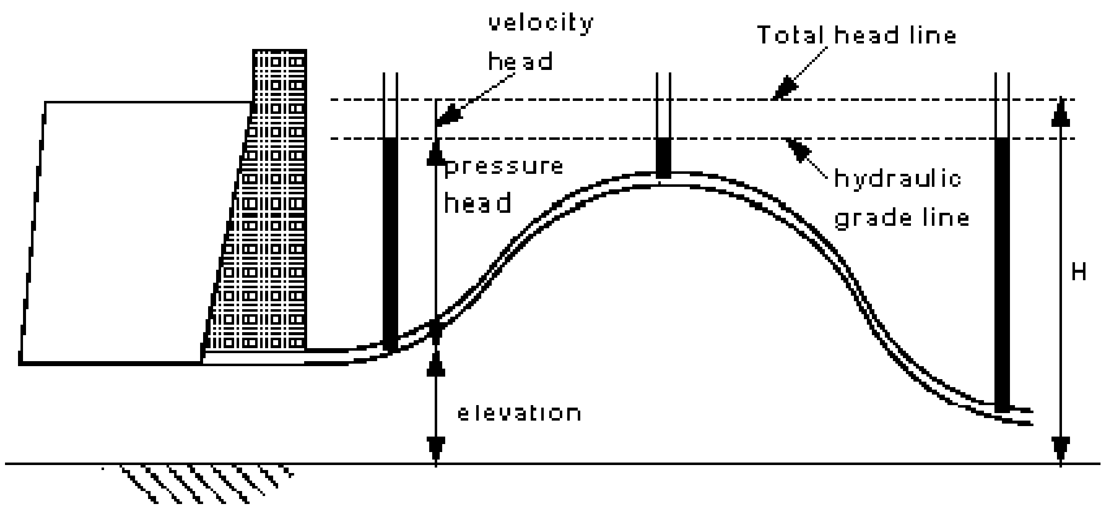
Piezometer levels with zero velocity

As you can see in the above figure, with zero velocity all of the levels in the piezometers are equal and the same as the total head line. At each point on the line, when *u = 0*

$$\frac{p}{\rho g}+z=H$$

The level in the piezometer is the *pressure head* and its value is given by $\frac{p}{\rho g}$

What would happen to the levels in the piezometers (pressure heads) if the if water was flowing with velocity = *u*? We know from earlier examples that as velocity increases so pressure falls …

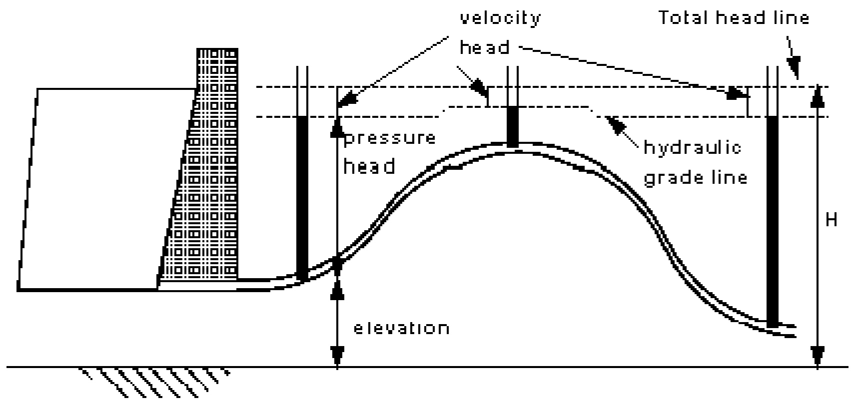


Piezometer levels when fluid is flowing

$$\frac{p}{\rho g}+\frac{u^2}{2g}+z=H$$

We see in this figure that the levels have reduced by an amount equal to the velocity head, $\frac{u^2}{2g}$. Now asthe pipe is of constant diameter we know that the velocity is constant along the pipe so the velocity headis constant and represented graphically by the horizontal line shown. (this line is known as the *hydraulic grade line*).

What would happen if the pipe were not of constant diameter? Look at the figure below where the pipe from the example above is replaced be a pipe of three sections with the middle section of larger diameter



Piezometer levels and velocity heads with fluid flowing in varying diameter pipes

The velocity head at each point is now different. This is because the velocity is different at each point. By considering continuity we know that the velocity is different because the diameter of the pipe is different. Which pipe has the greatest diameter?

Pipe 2, because the velocity, and hence the velocity head, is the smallest.

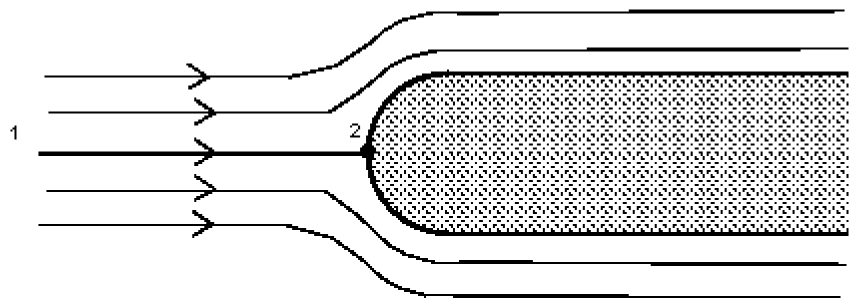
This graphical representation has the advantage that we can see at a glance the pressures in the system. For example, where along the whole line is the lowest pressure head? It is where the hydraulic grade line is nearest to the pipe elevation i.e. at the highest point of the pipe.

Applications of the Bernoulli Equation

The Bernoulli equation can be applied to a great many situations not just the pipe flow we have been considering up to now. In the following sections we will see some examples of its application to flow measurement from tanks, within pipes as well as in open channels.

1. Pitot Tube

If a stream of uniform velocity flows into a blunt body, the stream lines take a pattern similar to this:



Streamlines around a blunt body

Note how some move to the left and some to the right. But one, in the centre, goes to the tip of the blunt body and stops. It stops because at this point the velocity is zero - the fluid does not move at this one point. This point is known as the *stagnation point*.

From the Bernoulli equation we can calculate the pressure at this point. Apply Bernoulli along the central streamline from a point upstream where the velocity is u_1 and the pressure p_1 to the stagnation point of the blunt body where the velocity is zero, $u_2 = 0$. Also $z_1 = z_2$.

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

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

$$p_2 = p_1 + \frac{1}{2}\rho u_1^2$$

This increase in pressure which bring the fluid to rest is called the *dynamic pressure*.

Dynamic pressure = $\frac{1}{2}\rho u_1^2$ or Converting this to head (using $h = \frac{p}{\rho g}$)

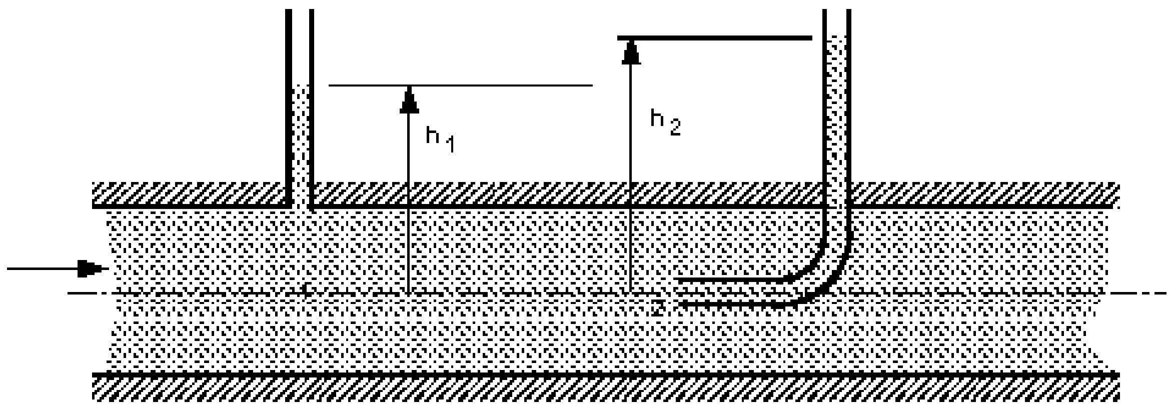
Dynamic head = $\frac{1}{2g}u_1^2$

The total pressure is known as the *stagnation pressure* (or *total pressure*)

Stagnation pressure = $p_1 + \frac{1}{2}\rho u_1^2$ or in terms of head

Stagnation head = $\frac{p_1}{\rho g} + \frac{1}{2g}u_1^2$

The blunt body stopping the fluid does not have to be a solid. I could be a static column of fluid. Two piezometers, one as normal and one as a Pitot tube within the pipe can be used in an arrangement shown below to measure velocity of flow.



A Piezometer and a Pitot tube

Using the above theory, we have the equation for p_2 ,

$$p_2 = p_1 + \frac{1}{2}\rho u_1^2$$

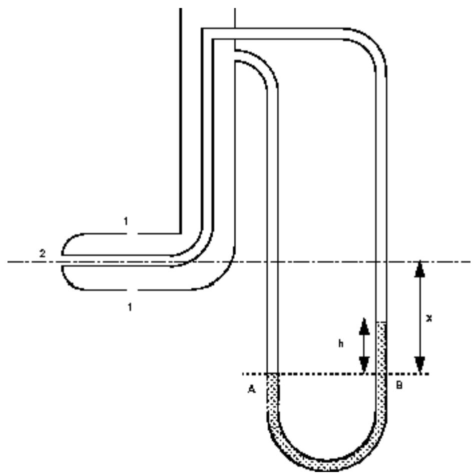
$$\rho g h_2 = \rho g h_1 + \frac{1}{2}\rho u_1^2$$

$$u = \sqrt{2g(h_2 - h_1)}$$

We now have an expression for velocity obtained from two pressure measurements and the application of the Bernoulli equation.

2. Pitot Static Tube

The necessity of two piezometers and thus two readings make this arrangement a little awkward. Connecting the piezometers to a manometer would simplify things but there are still two tubes. The *Pitot static* tube combines the tubes and they can then be easily connected to a manometer. A Pitot statictube is shown below. The holes on the side of the tube connect to one side of a manometer and register the *static head,* (h_1), while the central hole is connected to the other side of the manometer to register, asbefore, the *stagnation head* (h_2).



A Pitot-static tube

Consider the pressures on the level of the centre line of the Pitot tube and using the theory of the manometer,

$$p_A = p_2 + \rho g X$$
$$p_B = p_1 + \rho g(X - h) + \rho_{man} g h$$
$$p_A = p_B$$
$$p_2 + \rho g X = p_1 + \rho g(X - h) + \rho_{man} g h$$

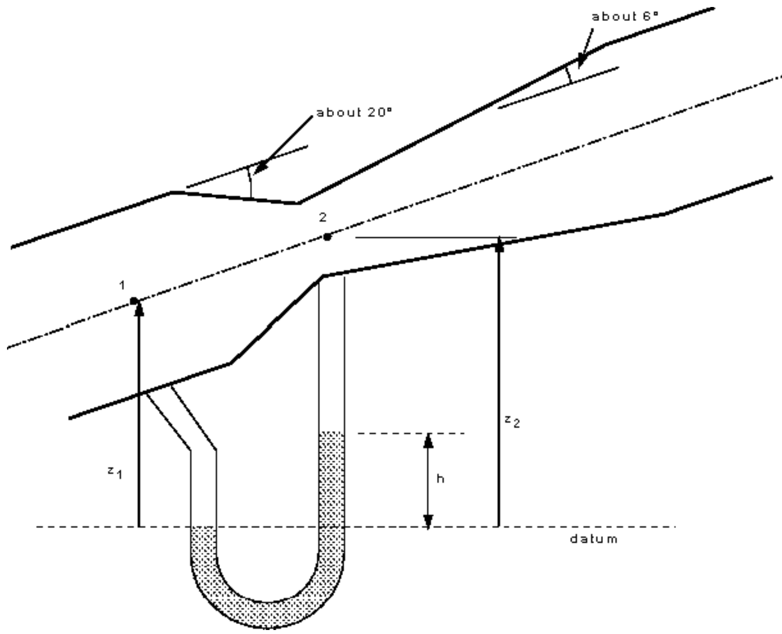
We know that $p_2 = p_{static} = p_1 + \frac{1}{2}\rho u_1^2$, substituting this in to the above gives

$$p_1 + hg(\rho_{man} - \rho) = p_1 + \frac{\rho u_1^2}{2}$$
$$u_1 = \sqrt{\frac{2gh(\rho_m - \rho)}{\rho}}$$

The Pitot/Pitot-static tubes give velocities at points in the flow. It does not give the overall discharge of the stream, which is often what is wanted. It also has the drawback that it is liable to block easily, particularly if there is significant debris in the flow.

3. Venturi Meter

The Venturi meter is a device for measuring discharge in a pipe. It consists of a rapidly converging section which increases the velocity of flow and hence reduces the pressure. It then returns to the original dimensions of the pipe by a gently diverging 'diffuser' section. By measuring the pressure differences the discharge can be calculated. This is a particularly accurate method of flow measurement as energy loss are very small.



A Venturi meter

Applying Bernoulli along the streamline from point 1 to point 2 in the narrow *throat* of the Venturi meter we have

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

By the using the continuity equation we can eliminate the velocity u_2,

$$Q = u_1 A_1 = u_2 A_2$$

$$u_2 = \frac{u_1 A_1}{A_2}$$

Substituting this into and rearranging the Bernoulli equation we get

$$\frac{p_1 - p_2}{\rho g} + z_1 - z_2 = \frac{u_1^2}{2g}\left[\left(\frac{A_1}{A_2}\right)^2 - 1\right]$$

$$u_1 = \sqrt{\frac{2g\left[\frac{p_1 - p_2}{\rho g} + z_1 - z_2\right]}{\left(\frac{A_1}{A_2}\right)^2 - 1}}$$

To get the theoretical discharge this is multiplied by the area. To get the actual discharge taking in to account the losses due to friction, we include a coefficient of discharge

$$Q_{ideal} = u_1 A_1$$
$$Q_{actual} = C_d Q_{ideal} = C_d u_1 A_1$$
$$Q_{actual} = C_d A_1 A_2 \sqrt{\frac{2g\left[\frac{p_1 - p_2}{\rho g} + z_1 - z_2\right]}{A_1^2 - A_2^2}}$$

This can also be expressed in terms of the manometer readings

$$p_1 + \rho g z_1 = p_2 + \rho_{man} g h + \rho g (z_2 - h)$$
$$\frac{p_1 - p_2}{\rho g} + z_1 - z_2 = h\left(\frac{\rho_{man}}{\rho} - 1\right)$$

Thus the discharge can be expressed in terms of the manometer reading::

$$Q_{actual} = C_d A_1 A_2 \sqrt{\frac{2gh\left(\frac{\rho_{man}}{\rho} - 1\right)}{A_1^2 - A_2^2}}$$

Notice how this expression does not include any terms for the elevation or orientation (z_1 or z_2) of the Venturimeter. This means that the meter can be at any convenient angle to function.

The purpose of the diffuser in a Venturi meter is to assure gradual and steady deceleration after the throat. This is designed to ensure that the pressure rises again to something near to the original value before the Venturi meter. The angle of the diffuser is usually between 6 and 8 degrees. Wider than this and the flow might separate from the walls resulting in increased friction and energy and pressure loss.

If the angle is less than this the meter becomes very long and pressure losses again become significant. The efficiency of the diffuser of increasing pressure back to the original is rarely greater than 80%.

UNIT II

Real fluids

The flow of real fluids exhibits viscous effect that is they tend to "stick" to solid surfaces and have stresses within their body. You might remember from earlier in the course Newtons law of viscosity:

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

This tells us that the shear stress, τ,in a fluid is proportional to the velocity gradient - the rate of change of velocity across the fluid path. For a "Newtonian" fluid we can write:

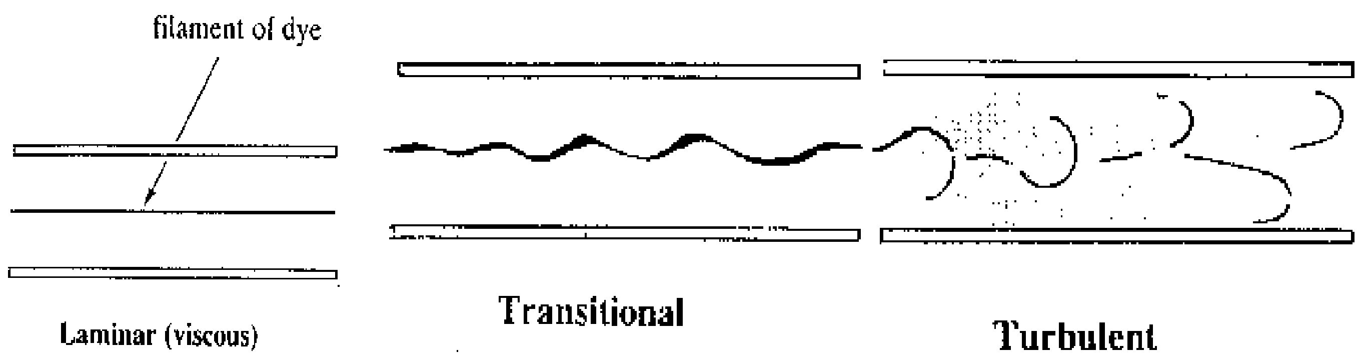
$$\tau = \mu \frac{du}{dy}$$

where the constant of proportionality, μ is known as the coefficient of viscosity (or simply viscosity). We saw that for some fluids - sometimes known as exotic fluids - the value μ of changes with stress orvelocity gradient. We shall only deal with Newtonian fluids.

In his lecture we shall look at how the forces due to momentum changes on the fluid and viscous forces compare and what changes take place.

Laminar and turbulent flow

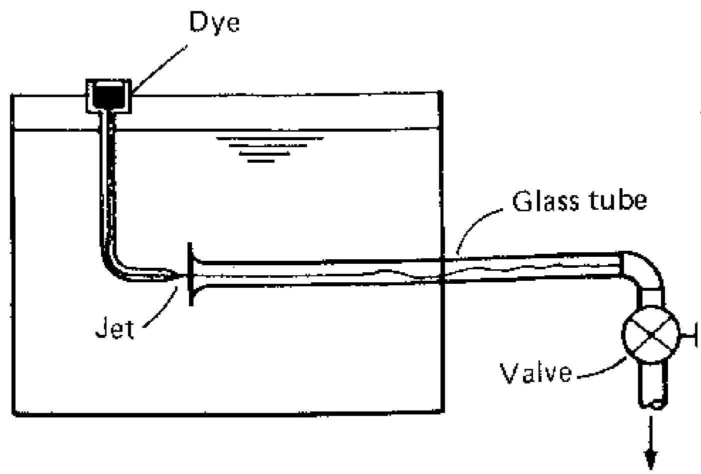
If we were to take a pipe of free flowing water and inject a dye into the middle of the stream, what would we expect to happen?



Actually both would happen - but for different flow rates. The top occurs when the fluid is flowing fast and the lower when it is flowing slowly.The top situation is known as **turbulent** flow and the lower as **laminar** flow. In laminar flow the motion of the particles of fluid is very orderly with all particles moving in straight lines parallel to the pipe walls.

But what is fast or slow? And at what speed does the flow pattern change? And why might we want to know this?

The phenomenon was first investigated in the 1880s by Osbourne Reynolds in an experiment which has become a classic in fluid mechanics.



He used a tank arranged as above with a pipe taking water from the centre into which he injected a dye through a needle. After many experiments he saw that this expression

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

where ρ = density, u = mean velocity, d = diameter and μ = viscosity

would help predict the change in flow type. If the value is less than about 2000 then flow is laminar, if greater than 4000 then turbulent and in between these then in the transition zone.

This value is known as the Reynolds number, Re:

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

Laminar flow: Re < 2000

Transitional flow: 2000 < Re < 4000

Turbulent flow: Re > 4000

What are the units of this Reynolds number? We can fill in the equation with SI units:

$$\rho = kg/m^3, \quad u = m/s, \quad d = m$$
$$\mu = Ns/m^2 = kg/ms$$

$$\mathrm{Re} = \frac{\rho u d}{\mu} = \frac{kg}{m^3}\frac{m}{s}\frac{m}{1}\frac{m}{kg} = 1$$

i.e. it has **no units**. A quantity that has no units is known as a **non-dimensional** (or dimensionless) quantity. Thus the Reynolds number, Re, is a non-dimensional number.

We can go through an example to discover at what velocity the flow in a pipe stops being laminar.

If the pipe and the fluid have the following properties:

water density ρ= 1000 kg/m^3

pipe diameter d = 0.5m

(dynamic) viscosity, μ= 0.55x10^3 Ns/m^2

We want to know the maximum velocity when the Re is 2000.

$$\mathrm{Re} = \frac{\rho u d}{\mu} = 2000$$
$$u = \frac{2000\mu}{\rho d} = \frac{2000 \times 0.55 \times 10^{-3}}{1000 \times 0.5}$$
$$u = 0.0022\, m/s$$

If this were a pipe in a house central heating system, where the pipe diameter is typically 0.015m, the limiting velocity for laminar flow would be, 0.0733 m/s.Both of these are very slow. In practice it very rarely occurs in a piped water system - the velocities of flow are much greater. Laminar flow does occur in situations with fluids of greater viscosity - e.g. in bearing with oil as the lubricant.At small values of Re above 2000 the flow exhibits small instabilities. At values of about 4000 we can say that the flow is truly turbulent. Over the past 100 years since this experiment, numerous more experiments have shown this phenomenon of limits of Re for many different Newtonian fluids - including gasses.

What does this abstract number mean?

We can say that the number has a physical meaning, by doing so it helps to understand some of the reasons for the changes from laminar to turbulent flow.

$$Re = \frac{\rho u d}{\mu}$$
$$= \frac{\text{inertial f orces}}{\text{viscous fo rces}}$$

It can be interpreted that when the inertial forces dominate over the viscous forces (when the fluid is flowing faster and Re is larger) then the flow is turbulent. When the viscous forces are dominant (slow flow, low Re) they are sufficient enough to keep all the fluid particles in line, then the flow is laminar.

In summary:

Laminar flow

- Re < 2000
- 'low' velocity
- Dye does not mix with water
- Fluid particles move in straight lines
- Simple mathematical analysis possible
- Rare in practice in water systems.

Transitional flow

- 2000 > Re < 4000
- 'medium' velocity
- Dye stream wavers in water - mixes slightly.

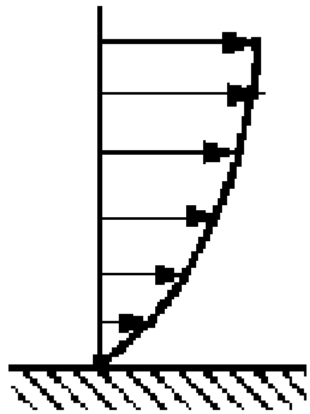
Turbulent flow

- Re > 4000
- 'high' velocity
- Dye mixes rapidly and completely
- Particle paths completely irregular
- Average motion is in the direction of the flow
- Cannot be seen by the naked eye
- Changes/fluctuations are very difficult to detect. Must use laser.
- Mathematical analysis very difficult - so experimental measures are used
- Most common type of flow.

Pressure loss due to friction in a pipeline.

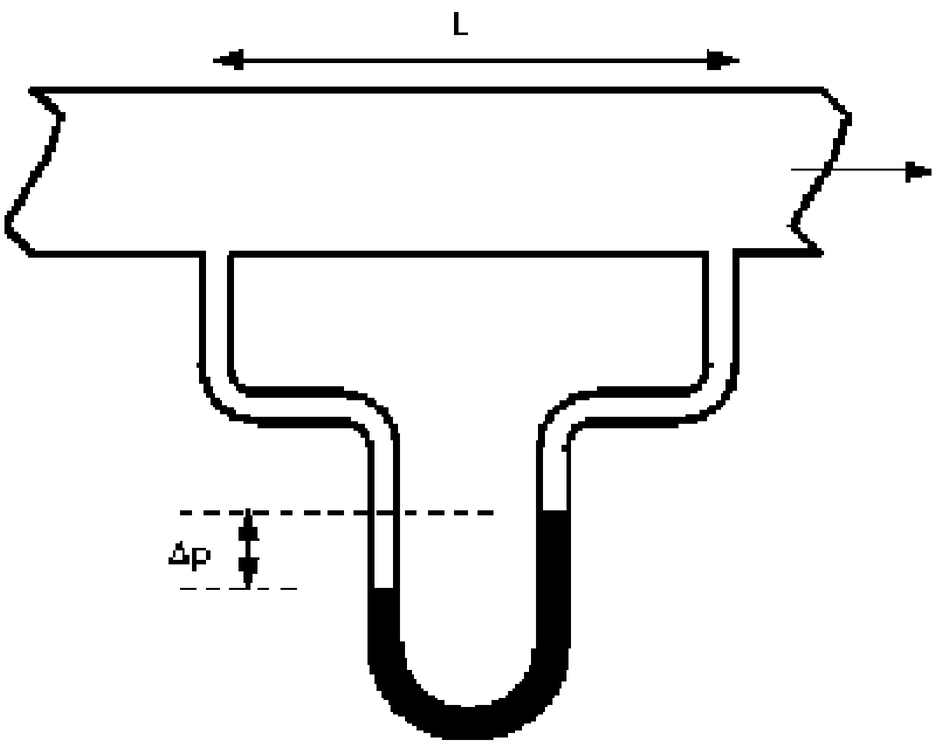
Up to this point on the course we have considered ideal fluids where there have been no losses due to friction or any other factors. In reality, because fluids are viscous, energy is lost by flowing fluids due to friction which must be taken into account. The effect of the friction shows itself as a pressure (or head) loss.

In a pipe with a real fluid flowing, at the wall there is a shearing stress retarding the flow, as shown below.



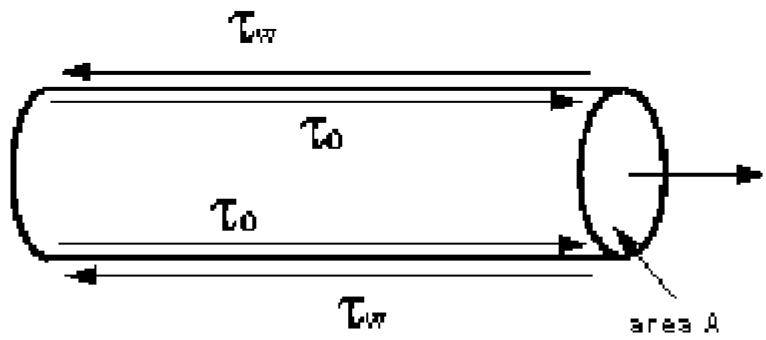
If a manometer is attached as the pressure (head) difference due to the energy lost by the fluid overcoming the shear stress can be easily seen.

The pressure at 1 (upstream) is higher than the pressure at 2.



We can do some analysis to express this loss in pressure in terms of the forces acting on the fluid.

Consider a cylindrical element of incompressible fluid flowing in the pipe, as shown



The pressure at the upstream end is *p*, and at the downstream end the pressure has fallen by The driving force due to pressure (F = Pressure x Area) can then be written Δp to (p-Δp).

driving force = Pressure force at 1 - pressure force at 2

$$pA - (p - \Delta p)A = \Delta p\, A = \Delta p \frac{\pi d^2}{4}$$

The retarding force is that due to the shear stress by the walls

$$= \text{shear stress} \times \text{area over which it acts}$$
$$= \tau_w \times \text{area of pipe wall}$$
$$= \tau_w \pi d L$$

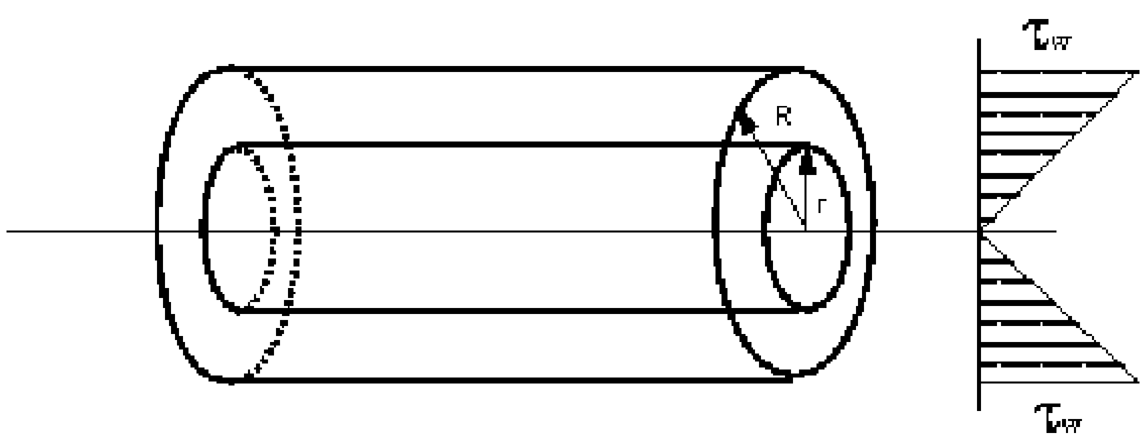
As the flow is in equilibrium,

driving force = retarding force

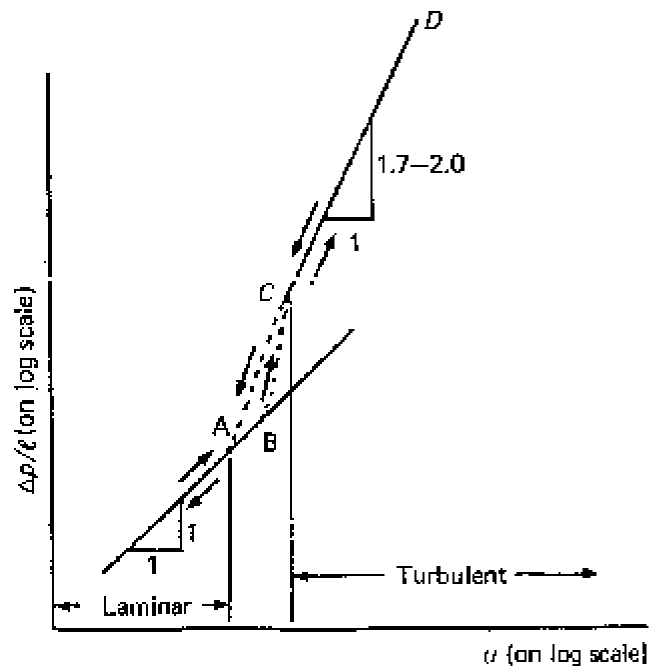
$$\Delta p \frac{\pi d^2}{4} = \tau_w \pi d L$$

$$\Delta p = \frac{\tau_w 4 L}{d}$$

Giving an expression for pressure loss in a pipe in terms of the pipe diameter and the shear stress at the wall on the pipe.



The shear stress will vary with velocity of flow and hence with Re. Many experiments have been done with various fluids measuring the pressure loss at various Reynolds numbers. These results plotted to show a graph of the relationship between pressure loss and Re look similar to the figure below:



This graph shows that the relationship between pressure loss and Re can be expressed as

laminar $\Delta p \propto u$

turbulent $\Delta p \propto u^{1.7 \text{ (or } 2.0)}$

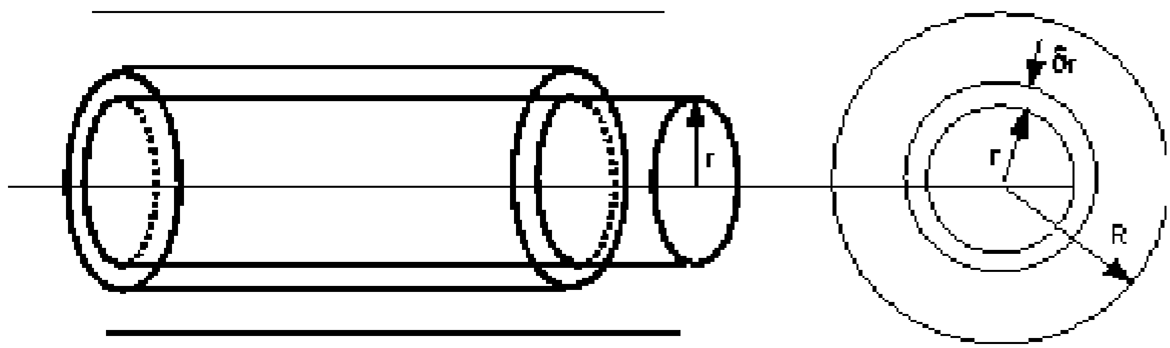
As these are empirical relationships, they help in determining the pressure loss but not in finding the magnitude of the shear stress at the wall τ_w on a particular fluid. If we knew τ_w we could then use it to give a general equation to predict the pressure loss.

Pressure loss during laminar flow in a pipe

In general the shear stress τ_w is almost impossible to measure. But for laminar flow it is possible to calculate a theoretical value for a given velocity, fluid and pipe dimension.

In laminar flow the paths of individual particles of fluid do not cross, so the flow may be considered as a series of concentric cylinders sliding over each other - rather like the cylinders of a collapsible pocket telescope.

As before, consider a cylinder of fluid, length L, radius **r**, flowing steadily in the centre of a pipe.



We are in equilibrium, so the shearing forces on the cylinder equal the pressure forces.

$$\tau\, 2\pi r\, L = \Delta p\, A = \Delta p \pi r^2$$

$$\tau = \frac{\Delta p}{L}\frac{r}{2}$$

By Newtons law of viscosity we have $\tau = \mu \frac{du}{dy}$, where y is the distance from the wall. As we are measuring from the pipe centre then we change the sign and replace y with r distance from the centre, giving

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

Which can be combined with the equation above to give

$$\frac{\Delta p}{L}\frac{r}{2} = -\mu \frac{du}{dr}$$

$$\frac{du}{dr} = -\frac{\Delta p}{L}\frac{r}{2\mu}$$

In an integral form this gives an expression for velocity,

$$u = -\frac{\Delta p}{L}\frac{1}{2\mu}\int r\,dr$$

Integrating gives the value of velocity at a point distance r from the centre

$$u_r = -\frac{\Delta p}{L}\frac{r^2}{4\mu} + C$$

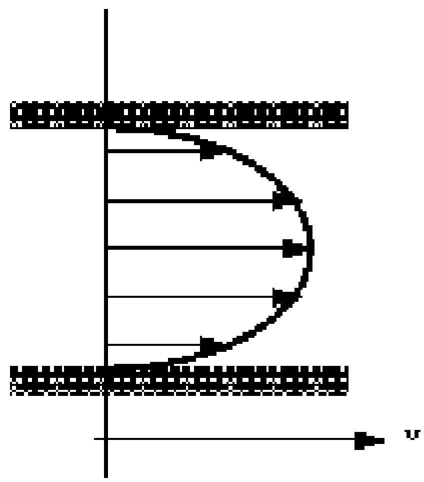
At $r = 0$, (the centre of the pipe), $u = u_{max}$, at $r = R$ (the pipe wall) $u = 0$, giving

$$C = \frac{\Delta p}{L}\frac{R^2}{4\mu}$$

so, an expression for velocity at a point r from the pipe centre when the flow is laminar is

$$u_r = \frac{\Delta p}{L}\frac{1}{4\mu}\left(R^2 - r^2\right)$$

Note how this is a parabolic profile (of the form $y = ax^2 + b$) so the velocity profile in the pipe looks similar to the figure below



What is the discharge in the pipe?

$$Q = u_m A$$

$$u_m = \int_0^R u_r \, dr$$

$$= \frac{\Delta p}{L}\frac{1}{4\mu}\int_0^R \left(R^2 - r^2\right) dr$$

$$= \frac{\Delta p}{L}\frac{R^2}{8\mu} = \frac{\Delta p \, d^2}{32\mu L}$$

So the discharge can be written

$$Q = \frac{\Delta p \, d^2}{32\mu L}\frac{\pi d^2}{4}$$

$$= \frac{\Delta p}{L}\frac{\pi d^2}{128\mu}$$

This is the Hagen-Poiseuille equation for laminar flow in a pipe. It expresses the discharge Q in terms of the pressure gradient ($\frac{\partial p}{\partial x} = \frac{\Delta p}{L}$), diameter of the pipe and the viscosity of the fluid.

We are interested in the pressure loss (head loss) and want to relate this to the velocity of the flow.

Writing pressure loss in terms of head loss h_f, i.e. $p = g\rho h_f$

$$u = \frac{\rho g h_f d^2}{32 \mu L}$$

$$h_f = \frac{32 \mu L u}{\rho g d^2}$$

This shows that pressure loss is directly proportional to the velocity when flow is laminar.

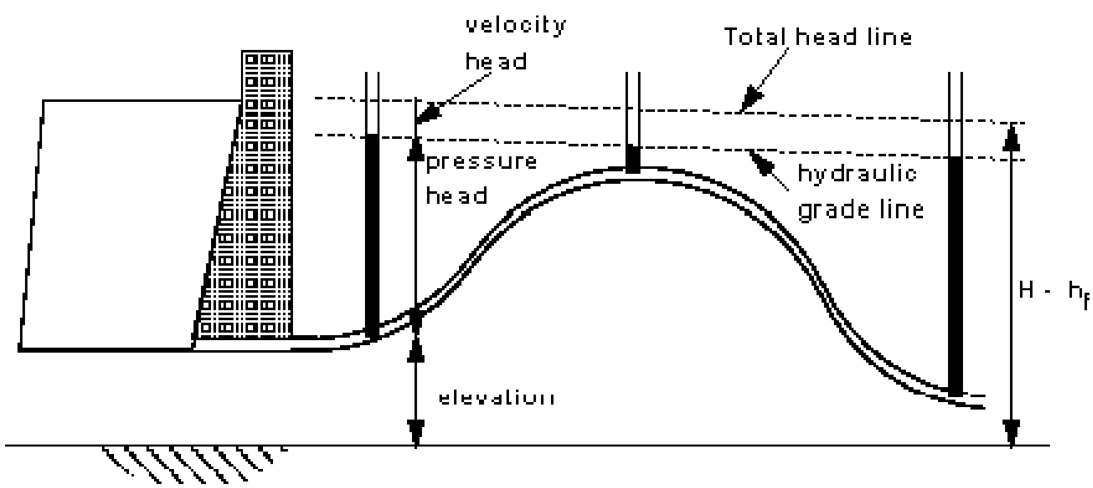
It has been validated many time by experiment.

It justifies two assumptions:

1. fluid does not slip past a solid boundary
2. Newtons hypothesis.

Losses due to friction

In a real pipe line there are energy losses due to friction - these must be taken into account as they can be very significant. How would the pressure and hydraulic grade lines change with friction? Going back to the constant diameter pipe, we would have a pressure situation like this shown below



Hydraulic Grade line and Total head lines for a constant diameter pipe with friction

How can the total head be changing? We have said that the total head - or total energy per unit weight - is constant. We are considering energy conservation, so if we allow for an amount of energy to be lost due to friction the total head will change. We have seen the equation for this before. But here it is again with the energy loss due to friction written as a *head* and given the symbol h_f. This is often know as the *head loss due to friction*.

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

UNIT III

Dimensional Analysis

In engineering the application of fluid mechanics in designs make much of the use of empirical results from a lot of experiments. This data is often difficult to present in a readable form. Even from graphs it may be difficult to interpret. Dimensional analysis provides a strategy for choosing relevant data and how it should be presented.

This is a useful technique in all experimentally based areas of engineering. If it is possible to identify the factors involved in a physical situation, dimensional analysis can form a relationship between them.

The resulting expressions may not at first sight appear rigorous but these qualitative results converted to quantitative forms can be used to obtain any unknown factors from experimental analysis.

Dimensions and units

Any physical situation can be described by certain familiar properties e.g. length, velocity, area, volume, acceleration etc. These are all known as dimensions.

Of course dimensions are of no use without a magnitude being attached. We must know more than that something has a length. It must also have a standardised unit - such as a meter, a foot, a yard etc.

Dimensions are properties which can be measured. Units are the standard elements we use to quantify these dimensions.

In dimensional analysis we are only concerned with the nature of the dimension i.e. its quality not its quantity. The following common abbreviation are used:

length = L

mass = M

time = T

force = F

temperature = Θ

In this module we are only concerned with L, M, T and F (not Θ).We can represent all the physical properties we are interested in with L, T and one of M or F (F can be represented by a combination of LTM). These notes will always use the LTM combination.

The following table (taken from earlier in the course) lists dimensions of some common physical quantities:

Quantity	SI Unit	.	Dimension
velocity	m/s	ms^{-1}	LT^{-1}
acceleration	m/s^2	ms^{-2}	LT^{-2}
force	N kg m/s^2	kg ms^{-2}	M LT^{-2}
energy (or work)	Joule JN m, kg m^2/s^2	kg m^2s^{-2}	ML^2T^{-2}
power	Watt W N m/s kg m^2/s^3	Nms^{-1} kg m^2s^{-3}	ML^2T^{-3}
pressure (or stress)	Pascal P,N/m^2, kg/m/s^2	Nm^{-2} kg $m^{-1}s^{-2}$	$ML^{-1}T^{-2}$
density	kg/m^3	kg m^{-3}	ML^{-3}
specific weight	N/m^3 kg/m^2/s^2	kg $m^{-2}s^{-2}$	$ML^{-2}T^{-2}$
relative density	a ratio no units	.	1 no dimension
viscosity	N s/m^2kg/m s	N sm^{-2} kg m^- $^1s^{-1}$	M $L^{-1}T^{-1}$
surface tension	N/mkg /s^2	Nm^{-1} kg s^{-2}	MT^{-2}

Dimensional Homogeneity

Any equation describing a physical situation will only be true if both sides have the same dimensions. That is it must be **dimensionally homogenous**.

For example the equation which gives for over a rectangular weir (derived earlier in this module) is,

$$Q = \frac{2}{3} B \sqrt{2g} H^{3/2}$$

The SI units of the left hand side are $m^3 s^{-1}$. The units of the right hand side must be the same. Writing the equation with only the SI units gives

$$\begin{aligned} m^3 s^{-1} &= m \left(m s^{-2} \right)^{1/2} m^{3/2} \\ &= m^3 s^{-1} \end{aligned}$$

i.e. the units are consistent.

To be more strict, it is the dimensions which must be consistent (any set of units can be used and simply converted using a constant). Writing the equation again in terms of dimensions,

$$\begin{aligned} L^3 T^{-1} &= L \left(L T^{-2} \right)^{1/2} L^{3/2} \\ &= L^3 T^{-1} \end{aligned}$$

Notice how the powers of the individual dimensions are equal, (for L they are both 3, for T both -1).

This property of dimensional homogeneity can be useful for:

1. Checking units of equations;
2. Converting between two sets of units;
3. Defining dimensionless relationships (see below).

Results of dimensional analysis

The result of performing dimensional analysis on a physical problem is a single equation. This equation relates all of the physical factors involved to one another. This is probably best seen in an example. If we want to find the force on a propeller blade we must first decide what might influence this force.

It would be reasonable to assume that the force, F, depends on the following physical properties:

d - diameter,

u - forward velocity of the propeller (velocity of the plane),

ρ - fluid density, N -revolutions per second, μ - fluid viscosity

Before we do any analysis we can write this equation:

$F = \phi(d, u, \rho, N, \mu)$

or

$0 = \phi(F, d, u, \rho, N, \mu)$

where ϕ and ϕ_1 are unknown functions.

These can be expanded into an infinite series which can itself be reduced to

$F = K d^m u^p \rho^q N^r \mu^s$

where K is some constant and m, p, q, r, s are unknown constant powers.

From dimensional analysis we

1. obtain these powers
2. form the variables into several dimensionless groups

The value of K or the functions ϕ and ϕ_1 must be determined from experiment. The knowledge of the dimensionless groups often helps in deciding what experimental measurements should be taken.

Buckingham's π theorems

Although there are other methods of performing dimensional analysis, (notably the *indicial* method) the method based on the Buckingham π theorems gives a good generalised strategy for obtaining a solution. This will be outlined below.

There are two theorems accredited to Buckingham, and know as his π theorems.

1st π theorem:

A relationship between **m** variables (physical properties such as velocity, density etc.) can be expressed as a relationship between **m-n** *non-dimensional* groups of variables (called π groups), where **n** is the number of fundamental dimensions (such as mass, length and time) required to express the variables.

So if a physical problem can be expressed:

$\phi(Q_1, Q_2, Q_3, \ldots\ldots\ldots, Q_m) = 0$

then, according to the above theorem, this can also be expressed

$\phi(\pi_1, \pi_2, \pi_3, \ldots\ldots\ldots, Q_{m-n}) = 0$

In fluids, we can normally take n = 3 (corresponding to M, L, T).

2nd π theorem:

Each π group is a function of **n** *governing* or *repeating variables* plus one of the remaining variables.

Choice of repeating variables

Repeating variables are those which we think will appear in all or most of the π groups, and are a influence in the problem. Before commencing analysis of a problem one must choose the repeating variables. There is considerable freedom allowed in the choice.

Some rules which should be followed are

1. From the 2nd theorem there can be **n** (= 3) repeating variables.
2. When combined, these repeating variables variable must contain all of dimensions (M, L, T) (That is not to say that each must contain M,L and T).
3. A combination of the repeating variables must not form a dimensionless group.
4. The repeating variables do not have to appear in all π groups.
5. The repeating variables should be chosen to be measurable in an experimental investigation. They should be of major interest to the designer. For example, pipe diameter (dimension L) is more useful and measurable than roughness height (also dimension L).

In fluids it is usually possible to take ρ,**u** and **d** as the threee repeating variables.

This freedom of choice results in there being many different π groups which can be formed - and all are valid. There is not really a wrong choice.

An example

Taking the example discussed above of force *F* induced on a propeller blade, we have the equation

$0 = \phi(F, d, u, \rho, N, \mu)$

n = 3 and m = 6

There are m - n = 3 π groups, so

$\phi(\pi_1, \pi_2, \pi_3) = 0$

The choice of ρ, u, d as the repeating variables satisfies the criteria above. They are measurable, good design parameters and, in combination, contain all the dimension M,L and T. We can now form the three groups according to the 2nd theorem,

$$\pi_1 = \rho^{a_1} u^{b_1} d^{c_1} F \quad \pi_2 = \rho^{a_2} u^{b_2} d^{c_2} N \quad \pi_3 = \rho^{a_3} u^{b_3} d^{c_3} \mu$$

As the π groups are all dimensionless i.e. they have dimensions $M^0L^0T^0$ we can use the principle of dimensional homogeneity to equate the dimensions for each π group.

For the first π group, $\pi_1 = \rho^{a_1} u^{b_1} d^{c_1} F$

In terms of SI units $1 = \left(kgm^{-3}\right)^{a_1} \left(ms^{-1}\right)^{b_1} (m)^{c_1} kgms^{-2}$

And in terms of dimensions

$$M^0 L^0 T^0 = \left(M L^{-3}\right)^{a_1} \left(L T^{-1}\right)^{b_1} (L)^{c_1} M L T^{-2}$$

For each dimension (M, L or T) the powers must be equal on both sides of the equation,

so for M: $0 = a_1 + 1$

$a_1 = -1$

for L: $0 = -3a_1 + b_1 + c_1 + 1$

$0 = 4 + b_1 + c_1$

for T: $0 = -b_1 - 2$

$b_1 = -2$

$c_1 = -4 - b_1 = -2$

Giving π_1 as

$$\pi_1 = \rho^{-1} u^{-2} d^{-2} F$$

$$\pi_1 = \frac{F}{\rho u^2 d^2}$$

And a similar procedure is followed for the other π groups. Group $\pi_2 = \rho^{a_2} u^{b_2} d^{c_2} N$

$$M^0 L^0 T^0 = \left(M L^{-3}\right)^{a_1} \left(L T^{-1}\right)^{b_1} (L)^{c_1} T^{-1}$$

For each dimension (M, L or T) the powers must be equal on both sides of the equation,

so for M: $0 = a_2$

for L: $0 = -3a_2 + b_2 + c_2$

$0 = b_2 + c_2$

for T: $0 = -b_2 - 1$

$b_2 = -1$

$c_2 = 1$

Giving π_2 as

$$\pi_2 = \rho^0 u^{-1} d^1 N$$

$$\pi_2 = \frac{Nd}{u}$$

And for the third, $\pi_3 = \rho^{a_3} u^{b_3} d^{c_3} \mu$

$$M^0 L^0 T^0 = \left(M L^{-3}\right)^{a_3} \left(L T^{-1}\right)^{b_3} \left(L\right)^{c_3} M L^{-1} T^{-1}$$

For each dimension (M, L or T) the powers must be equal on both sides of the equation,

so for M: $0 = a_3 + 1$

$a_3 = -1$

for L: $0 = -3a_3 + b_3 + c_3 - 1$

$b_3 + c_3 = -2$

for T: $0 = -b_3 - 1$

$b_3 = -1$

$c_3 = -1$

Giving π_3 as

$$\pi_3 = \rho^{-1} u^{-1} d^{-1} \mu$$

$$\pi_3 = \frac{\mu}{\rho u d}$$

Thus the problem may be described by the following function of the three non-dimensional π groups,

$$\phi(\pi_1, \pi_2, \pi_3) = 0$$

$$\phi\left(\frac{F}{\rho u^2 d^2}, \frac{Nd}{u}, \frac{\mu}{\rho u d}\right) = 0$$

This may also be written:

$$\frac{F}{\rho u^2 d^2} = \phi\left(\frac{Nd}{u}, \frac{\mu}{\rho u d}\right)$$

Wrong choice of physical properties

If, when defining the problem, extra - unimportant - variables are introduced then extra π groups will be formed. They will play very little role influencing the physical behaviour of the problem concerned and should be identified during experimental work. If an important / influential variable was missed then a π group would be missing. Experimental analysis based on these results may miss significant behavioural changes. It is therefore, very important that the initial choice of variables is carried out with great care.

Manipulation of the π groups

Once identified manipulation of the π groups is permitted. These manipulations do not change the number of groups involved, but may change their appearance drastically.

Taking the defining equation as: $\phi(\pi_1, \pi_2, \pi_3 \ldots\ldots\ \pi_{m-n}) = 0$

Then the following manipulations are permitted:

1. Any number of groups can be combined by multiplication or division to form a new group which replaces one of the existing. E.g. π_1 and π_2 may be combined to form $\pi_{1a} = \pi_1 / \pi_2$ so the defining equation becomes $\phi(\pi_{1a}, \pi_2, \pi_3 \ldots\ldots\ \pi_{m-n}) = 0$
2. The reciprocal of any dimensionless group is valid. So $\phi(\pi_1, 1/\pi_2, \pi_3 \ldots\ldots\ 1/\pi_{m-n}) = 0$ is valid.
3. Any dimensionless group may be raised to any power. So $\phi((\pi_1)^2, (\pi_2)^{1/2}, (\pi_3)3 \ldots\ldots\ \pi_{m-n}) = (0$ is valid.
4. Any dimensionless group may be multiplied by a constant.
5. Any group may be expressed as a function of the other groups, e.g. $\pi_2 = \phi(\pi_1, \pi_3 \ldots\ldots\ \pi_{m-n})$

In general the defining equation could look like

$\phi(\pi_1, 1/\pi_2, (\pi_3)^i \ldots\ldots\ 0.5\,\pi_{m-n}) = 0$

Common π groups

During dimensional analysis several groups will appear again and again for different problems. These often have names. You will recognise the Reynolds number $\rho ud/\mu$. Some common non-dimensional numbers (groups) are listed below.

Reynolds number $\mathrm{Re} = \dfrac{\rho ud}{\mu}$ inertial, viscous force ratio

Euler number $\mathrm{En} = \dfrac{p}{\rho u^2}$ pressure, inertial force ratio

Froude number $\mathrm{Fn} = \dfrac{u^2}{gd}$ inertial, gravitational force ratio

Weber number $\mathrm{We} = \dfrac{\rho ud}{\sigma}$ inertial, surface tension force

ratio $\mathrm{Mn} = \dfrac{u}{c}$

Mach number	Local velocity, local velocity of sound ratio

Examples

The discharge Q through an orifice is a function of the diameter d, the pressure difference p, the density ρ, and the viscosity μ, show that $Q = \frac{d^2 p^{1/2}}{\rho^{1/2}} \phi\left(\frac{d\rho^{1/2} p^{1/2}}{\mu}\right)$, where ϕ is some unknown function.

Write out the dimensions of the

ρ variables ML^{-3} u: LT^{-1}

d: L μ $ML^{-1}T^{-1}$

p: *(force/area)* $ML^{-1}T^{-2}$

We are told from the question that there are 5 variables involved in the problem: d, p, ρ, μ and Q. Choose the three recurring (governing) variables; Q, d, ρ

From Buckingham's π theorem we have m-n = 5 - 3 = 2 non-dimensional groups.

$$\phi(Q, d, \rho, \mu, p) = 0$$

$$\phi(\pi_1, \pi_2) = 0$$

$$\pi_1 = Q^{a_1} d^{b_1} \rho^{c_1} \mu$$

$$\pi_2 = Q^{a_2} d^{b_2} \rho^{c_2} p$$

For the first group, π_1

$$M^0 L^0 T^0 = (L^3 T^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} ML^{-1}T^{-1}$$

M] $0 = c_1 + 1$

$c_1 = -1$

L] $0 = 3a_1 + b_1 - 3c_1 - 1$

$-2 = 3a_1 + b_1$

T] $0 = -a_1 -$

1

$a_1 = -1$

$b_1 = 1$

$$\pi_1 = Q^{-1} d^1 \rho^{-1} \mu$$

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

And the second group π_2:

(note p is a pressure (force/area) with dimensions $ML^{-1}T^{-2}$)

$$M^0 L^0 T^0 = (L^3 T^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} MT^{-2}L^{-1}$$

M] $0 = c_2 + 1$

$c_2 = -1$

L] $0 = 3a_2 + b_2 - 3c_2 - 1$

$-2 = 3a_2 + b_2$

T] $0 = -a_2 - 2$

$a_2 = -2$

$b_2 = 4$

$$\pi_2 = Q^{-2} d^4 \rho^{-1} p$$
$$= \frac{d^4 p}{\rho Q^2}$$

So the physical situation is described by this function of non-dimensional numbers,

$$\phi(\pi_1, \pi_2) = \phi\left(\frac{d\mu}{Q\rho}, \frac{d^4 p}{\rho Q^2}\right) = 0$$

$$or$$

$$\frac{d\mu}{Q\rho} = \phi_1\left(\frac{d^4 p}{\rho Q^2}\right)$$

The question wants us to show : $Q = \frac{d^2 p^{1/2}}{\rho^{1/2}} \phi\left(\frac{d\rho^{1/2} p^{1/2}}{\mu}\right)$

Take the reciprocal of square root of π_2 $\frac{1}{\sqrt{\pi_2}} = \frac{\rho^{1/2} Q}{d^2 p^{1/2}} = \pi_{2a}$,

Convert π_1 by multiplying by this new group, π_{2a}

$$\pi_{1a} = \pi_1 \pi_{2a} = \frac{d\mu}{Q\rho} \frac{\rho^{1/2} Q}{d^2 p^{1/2}} = \frac{\mu}{d\rho^{1/2} p^{1/2}}$$

then we can say

$$\phi(1/\pi_{1a}, \pi_{2a}) = \phi\left(\frac{d\rho^{1/2} p^{1/2}}{\mu}, \frac{d^2 p^{1/2}}{Q\rho^{1/2}}\right) = 0$$

$$or$$

$$Q = \frac{d^2 p^{1/2}}{\rho^{1/2}} \phi\left(\frac{d\rho^{1/2} p^{1/2}}{\mu}\right)$$

Similarity and Models

Hydraulic models may be either true or distorted models. True models reproduce features of the prototype but at a scale - that is they are *geometrically* similar.

Geometric similarity

Geometric similarity exists between model and prototype if the ratio of all corresponding dimensions in the model and prototype are equal.

$$\frac{L_{\text{model}}}{L_{\text{prototype}}} = \frac{L_{\text{m}}}{L_{\text{p}}} = \lambda_L$$

where λ_L is the scale factor for length.

For area

$$\frac{A_{\text{model}}}{A_{\text{prototype}}} = \frac{L_{\text{m}}^2}{L_{\text{p}}^2} = \lambda_L^2$$

All corresponding angles are the same.

Kinematic similarity

Kinematic similarity is the similarity of time as well as geometry. It exists between model and prototype

1. If the paths of moving particles are geometrically similar
2. If the rations of the velocities of particles are similar

Some useful ratios are:

Velocity $$\frac{V_{\text{m}}}{V_{\text{p}}} = \frac{\text{L}_{\text{m}} / T_m}{\text{L}_{\text{p}} / T_p} = \frac{\lambda_L}{\lambda_T} = \lambda_u$$

Acceleration $$\frac{a_{\text{m}}}{a_{\text{p}}} = \frac{L_m / T_m^2}{L_p / T_p^2} = \frac{\lambda_L}{\lambda_T^2} = \lambda_a$$

Discharge $$\frac{Q_{\text{m}}}{Q_{\text{p}}} = \frac{L_m^3 / T_m}{L_p^3 / T_p} = \frac{\lambda_L^3}{\lambda_T} = \lambda_Q$$

This has the consequence that streamline patterns are the same.

Dynamic similarity

Dynamic similarity exists between geometrically and kinematically similar systems if the ratios of all forces in the model and prototype are the same.

Force ratio $\frac{F_m}{F_p} = \frac{M_m a_m}{M_p a_p} = \frac{\rho_m L_m^3}{\rho_p L_p^3} \times \frac{\lambda_L}{\lambda_T^2} = \lambda_\rho \lambda_L^2 \left(\frac{\lambda_L}{\lambda_T}\right)^2 = \lambda_\rho \lambda_L^2 \lambda_u^2$

This occurs when the controlling dimensionless group on the right hand side of the defining equation is the same for model and prototype.

Models

When a hydraulic structure is build it undergoes some analysis in the design stage. Often the structures are too complex for simple mathematical analysis and a hydraulic model is build. Usually the model is less than full size but it may be greater. The real structure is known as the prototype. The model is usually built to an exact geometric scale of the prototype but in some cases - notably river model - this isnot possible. Measurements can be taken from the model and a suitable scaling law applied to predictthe values in the prototype.

- Distorted model
- Undistorted model

To illustrate how these scaling laws can be obtained we will use the relationship for resistance of a body moving through a fluid.

The resistance, *R*, is dependent on the following physical properties:

ρML^{-3} $u: LT^{-1}$ l:(length) L $\mu: ML^{-1}T^{-1}$

So the defining equation is $\phi(R, \rho, u, l, \mu) = 0$

Thus, m = 5, n = 3 so there are 5-3 = 2 π groups

$\pi_1 = \rho^{a_1} u^{b_1} l^{c_1} R$ $\pi_2 = \rho^{a_2} u^{b_2} d^{c_2} \mu$

For the π_1 group $M^0 L^0 T^0 = \left(M L^{-3}\right)^{a_1} \left(L T^{-1}\right)^{b_1} (L)^{c_1} M L T^{-2}$

Leading to π_1 as

$$\pi_1 = \frac{R}{\rho u^2 l^2}$$

For the π_2 group $M^0 L^0 T^0 = \left(M L^{-3}\right)^{a_3} \left(L T^{-1}\right)^{b_3} (L)^{c_3} M L^{-1} T^{-1}$

Leading to π_1 as

$$\pi_2 = \frac{\mu}{\rho u l}$$

Notice how 1/ π_2 is the Reynolds number. We can call this π_{2a}

So the defining equation for resistance to motion is

$\phi(\pi_1, \pi_{2a}) = 0$

We can write

$$\frac{R}{\rho u^2 l^2} = \phi\left(\frac{\rho u l}{\mu}\right)$$

$$R = \rho u^2 l^2 \phi\left(\frac{\rho u l}{\mu}\right)$$

This equation applies whatever the size of the body i.e. it is applicable to a to the prototype and a geometrically similar model. Thus for the model

$$\frac{R_m}{\rho_m u_m^2 l_m^2} = \phi\left(\frac{\rho_m u_m l_m}{\mu_m}\right)$$

and for the prototype

$$\frac{R_p}{\rho_p u_p^2 l_p^2} = \phi\left(\frac{\rho_p u_p l_p}{\mu_p}\right)$$

Dividing these two equations gives

$$\frac{R_m / \rho_m u_m^2 l_m^2}{R_p / \rho_p u_p^2 l_p^2} = \frac{\phi\left(\rho_m u_m l_m / \mu_m\right)}{\phi\left(\rho_p u_p l_p / \mu_p\right)}$$

At this point we can go no further unless we make some assumptions. One common assumption is to assume that the Reynolds number is the same for both the model and prototype i.e.

$$\rho_m u_m l_m / \mu_m = \rho_p u_p l_p / \mu_p$$

This assumption then allows the equation following to be written

$$\frac{R_m}{R_p} = \frac{\rho_m u_m^2 l_m^2}{\rho_p u_p^2 l_p^2}$$

Which gives this scaling law for resistance force:

$$\lambda_R = \lambda_\rho \lambda_u^2 \lambda_L^2$$

That the Reynolds numbers were the same was an essential assumption for this analysis. The consequence of this should be explained.

$$\mathrm{Re}_m = \mathrm{Re}_p$$

$$\frac{\rho_m u_m l_m}{\mu_m} = \frac{\rho_p u_p l_p}{\mu_p}$$

$$\frac{u_m}{u_p} = \frac{\rho_p}{\rho_m}\frac{\mu_m}{\mu_p}\frac{l_p}{l_m}$$

$$\lambda_u = \frac{\lambda_\mu}{\lambda_\rho \lambda_l}$$

Substituting this into the scaling law for resistance gives

$$\lambda_R = \lambda_\rho \left(\frac{\lambda_\mu}{\lambda_\rho}\right)^2$$

So the force on the prototype can be predicted from measurement of the force on the model. But only if the fluid in the model is moving with same Reynolds number as it would in the prototype. That is to say the R_p can be predicted by

$$R_p = \frac{\rho_p u_p^2 l_p^2}{\rho_m u_m^2 l_m^2} R_m$$

provided that $u_p = \frac{\rho_m}{\rho_p}\frac{\mu_p}{\mu_m}\frac{l_m}{l_p} u_m$

In this case the model and prototype are **dynamically similar**.

Formally this occurs when the controlling dimensionless group on the right hand side of the defining equation is the same for model and prototype. In this case the controlling dimensionless group is the Reynolds number.

Dynamically similar model examples

Example 1

An underwater missile, diameter 2m and length 10m is tested in a water tunnel to determine the forces acting on the real prototype. A 1/20th scale model is to be used. If the maximum allowable speed of the prototype missile is 10 m/s, what should be the speed of the water in the tunnel to achieve dynamic similarity?

For dynamic similarity the Reynolds number of the model and prototype must be equal:

$$\mathrm{Re}_m = \mathrm{Re}_p$$

$$\left(\frac{\rho u d}{\mu}\right)_m = \left(\frac{\rho u d}{\mu}\right)_p$$

So the model velocity should be

$$u_m = u_p \frac{\rho_p}{\rho_m}\frac{d_p}{d_m}\frac{\mu_m}{\mu_p}$$

As both the model and prototype are in water then, $\mu_m = \mu_p$ and = so

ρ_m ρ_p

$$u_m = u_p \frac{d_p}{d_m} = 10\frac{1}{1/20} = 200 m/s$$

Note that this is a **very** high velocity. This is one reason why model tests are not always done at exactly equal Reynolds numbers. Some relaxation of the equivalence requirement is often acceptable when the Reynolds number is high. Using a wind tunnel may have been possible in this example. If this were the case then the appropriate values of the and ratios need to be used in the above equation. ρ μ

Geometric distortion in river models

When river and estuary models are to be built, considerable problems must be addressed. It is very difficult to choose a suitable scale for the model and to keep geometric similarity. A model which has a suitable depth of flow will often be far to big - take up too much floor space. Reducing the size and retaining geometric similarity can give tiny depth where viscous force come into play. These result in the following problems:

1. accurate depths and depth changes become very difficult to measure;
2. the bed roughness of the channel becomes impracticably small;
3. laminar flow may result - (turbulent flow is normal in river hydraulics.)

The solution often adopted to overcome these problems is to abandon strict geometric similarity by having different scales in the horizontal and the vertical. Typical scales are 1/100 in the vertical and between 1/200 and 1/500 in the horizontal. Good overall flow patterns and discharge characteristics can be produced by this technique, however local detail of flow is not well modelled.

In these model the Froude number (u^2/d) is used as the dominant non-dimensional number. Equivalence in Froude numbers can be achieved between model and prototype even for distorted models. To address the roughness problem artificially high surface roughness of wire mesh or small blocks is usually used.

1. A stationary sphere in water moving at a velocity of 1.6m/s experiences a drag of 4N. Another sphere of twice the diameter is placed in a wind tunnel. Find the velocity of the air and the drag which will give dynamically similar conditions. The ratio of kinematic viscosities of air and water is 13, and the density of air 1.28 kg/m^3.

[10.4m/s 0.865N]

Draw up the table of values you have for each variable:

variable	Water	air
u	1.6m/s	u_{air}
Drag	4N	D_{air}
ν	ν	13 ν

ρ	1000 kg/m^3	1.28 kg/m^3
d	D	2d

Kinematic viscosity is dynamic viscosity over density = $\nu = \mu/\rho$.

The Reynolds number = $\mathrm{Re} = \frac{\rho u d}{\mu} = \frac{ud}{\nu}$

Choose the three recurring (governing) variables; u, d, ρ.

From Buckinghams π theorem we have m-n = 5 - 3 = 2 non-dimensional groups.

$$\phi(u, d, \rho, D, \nu) = 0$$
$$\phi(\pi_1, \pi_2) = 0$$
$$\pi_1 = u^{a_1} d^{b_1} \rho^{c_1} D$$
$$\pi_2 = u^{a_2} d^{b_2} \rho^{c_2} \nu$$

As each π group is dimensionless then considering the dimensions, for the first group, π_1:

(note D is a force with dimensions MLT^{-2})

$$M^0 L^0 T^0 = (LT^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} MLT^{-2}$$

M] $0 = c_1 + 1$

$c_1 = -1$

L] $0 = a_1 + b_1 - 3c_1 + 1$

$-4 = a_1 + b_1$

T] $0 = -a_1 - 2$

$a_1 = -2$

$b_1 = -2$

$$\pi_1 = u^{-2} d^{-2} \rho^{-1} D$$
$$= \frac{D}{\rho u^2 d^2}$$

And the second group π_2 :

$$M^0 L^0 T^0 = \left(LT^{-1}\right)^{a_2} (L)^{b_2} \left(ML^{-3}\right)^{c_2} L^2 T^{-1}$$

M] 0 = c_2

L] 0 = $a_2 + b_2 - 3c_2 + 2$

-2 = $a_2 + b_2$

T] 0 = $-a_2$ -

1

a_2 = -1

b_2 = -1

$$\pi_2 = u^{-1} d^{-1} \rho^0 \nu$$
$$= \frac{\nu}{ud}$$

So the physical situation is described by this function of nondimensional numbers,

$$\phi(\pi_1, \pi_2) = \phi\left(\frac{D}{\rho u^2 d^2}, \frac{\nu}{ud}\right) = 0$$

For dynamic similarity these non-dimensional numbers are the same for the both the sphere in water and in the wind tunnel i.e.

$$\pi_{1air} = \pi_{1water}$$
$$\pi_{2air} = \pi_{2water}$$

For π_1

$$\left(\frac{D}{\rho u^2 d^2}\right)_{air} = \left(\frac{D}{\rho u^2 d^2}\right)_{water}$$
$$\frac{D_{air}}{1.28 \times 10.4^2 \times (2d)^2} = \frac{4}{1000 \times 1.6^2 \times d^2}$$
$$D_{air} = 0.865 N$$

For π_2

$$\left(\frac{\nu}{ud}\right)_{air} = \left(\frac{\nu}{ud}\right)_{water}$$
$$\frac{13\nu}{u_{air} \times 2d} = \frac{\nu}{1.6 \times d}$$
$$u_{air} = 10.4 m/s$$

2. Explain briefly the use of the Reynolds number in the interpretation of tests on the flow of liquid in pipes.Water flows through a 2cm diameter pipe at 1.6m/s. Calculate the Reynolds number and find also the velocity required to give the same Reynolds number when the pipe is transporting air. Obtain the ratio of pressure drops in the same length of pipe for both cases. For the water the kinematic viscosity was 1.3110^{-6} m^2/s and the density was 1000 kg/m^3.
For air those quantities were 15.110^{-6} m^2/s and 1.19kg/m^3.
[24427, 18.4m/s, 0.157]

Draw up the table of values you have for each variable:

variable	Water	air
u	1.6m/s	u_{air}
p	p_{water}	p_{air}
ρ	1000 kg/m^3	1.19kg/m^3
ν	$1.3110^{-6}\ m^2/s$	$15.110^{-6}\ m^2/s$
ρ	1000 kg/m^3	1.28 kg/m^3
d	0.02m	0.02m

Kinematic viscosity is dynamic viscosity over density = $\nu = \mu/\rho$.

The Reynolds number = $$Re = \frac{\rho u d}{\mu} = \frac{ud}{\nu}$$

Reynolds number when carrying water:

$$Re_{water} = \frac{ud}{\nu} = \frac{1.6 \times 0.02}{1.31 \times 10^{-6}} = 24427$$

To calculate Re_{air} we know,

$$Re_{water} = Re_{air}$$
$$24427 = \frac{u_{air} 0.02}{15 \times 10^{-6}}$$
$$u_{air} = 18.44 m/s$$

To obtain the ratio of pressure drops we must obtain an expression for the pressure drop in terms of governing variables.

Choose the three recurring (governing) variables; u, d, ρ.

From Buckinghams π theorem we have m-n = 5 - 3 = 2 non-dimensional groups.

$$\phi(u, d, \rho, \nu, p) = 0$$
$$\phi(\pi_1, \pi_2) = 0$$
$$\pi_1 = u^{a_1} d^{b_1} \rho^{c_1} \nu$$
$$\pi_2 = u^{a_2} d^{b_2} \rho^{c_2} p$$

As each π group is dimensionless then considering the dimensions, for the first group, π_1

$$M^0 L^0 T^0 = (LT^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} L^2 T^{-1}$$

M] $0 = c_1$

L] $0 = a_1 + b_1 - 3c_1 + 2$

$-2 = a_1 + b_1$

T] $0 = -a_1 -$

1

$a_1 = -1$

$b_1 = -1$

$$\pi_1 = u^{-1} d^{-1} \rho^0 \nu$$
$$= \frac{\nu}{ud}$$

And the second group π_2:

(note p is a pressure (force/area) with dimensions $ML^{-1}T^{-2}$)

$$M^0 L^0 T^0 = (LT^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} MT^{-2} L^{-1}$$

M] $0 = c_2 + 1$

$c_2 = -1$

L] $0 = a_2 + b_2 - 3c_2 - 1$

$-2 = a_2 + b_2$

T] $0 = -a_2 - 2$

$a_2 = -2$

$b_2 = 0$

$$\pi_2 = u^{-2}\rho^{-1}p$$
$$= \frac{p}{\rho u^2}$$

So the physical situation is described by this function of nondimensional numbers,

$$\phi(\pi_1, \pi_2) = \phi\left(\frac{\nu}{ud}, \frac{p}{\rho u^2}\right) = 0$$

For <u>dynamic similarity</u> these non-dimensional numbers are the same for the both water and air in the pipe.

$$\pi_{1air} = \pi_{1water}$$
$$\pi_{2air} = \pi_{2water}$$

We are interested in the relationship involving the pressure i.e. π_2

$$\left(\frac{p}{\rho u^2}\right)_{air} = \left(\frac{p}{\rho u^2}\right)_{water}$$

$$\frac{p_{water}}{p_{air}} = \frac{\rho_{water}u^2_{water}}{\rho_{air}u^2_{air}}$$
$$= \frac{1000 \times 1.6^2}{1.19 \times 18.44^2} = \frac{1}{0.158} = 6.327$$

3. A cylinder 0.16m in diameter is to be mounted in a stream of water in order to estimate the force on a tall chimney of 1m diameter which is subject to wind of 33m/s. Calculate (A) the speed of the stream necessary to give dynamic similarity between the model and chimney, (b) the ratio of forces.

Chimney: ρ= 1.12kg/m^3 μ= 1610^{-6} kg/ms

Model: ρ= 1000kg/m^3 μ= 810^{-4} kg/ms

[11.55m/s, 0.057]

Draw up the table of values you have for each variable:

variable	Water	air
u	u_{water}	33m/s
F	F_{water}	F_{air}
ρ	1000 kg/m^3	1.12kg/m^3
μ	810^{-4} kg/ms	1610^{-6} kg/ms
d	0.16m	1m

Kinematic viscosity is dynamic viscosity over density = $\nu = \mu/\rho$.

The Reynolds number = $\mathrm{Re} = \frac{\rho u d}{\mu} = \frac{ud}{\nu}$

For dynamic similarity:

$$\mathrm{Re}_{water} = \mathrm{Re}_{air}$$
$$\frac{1000 u_{water} 0.16}{8 \times 10^{-4}} = \frac{1.12 \times 33 \times 1}{16 \times 10^{-6}}$$
$$u_{water} = 11.55 m/s$$

To obtain the ratio of forces we must obtain an expression for the force in terms of governing variables.

Choose the three recurring (governing) variables; u, d, ρ, F, μ.

From Buckinghams π theorem we have m-n = 5 - 3 = 2 non-dimensional groups.

$$\phi(u, d, \rho, \mu, F) = 0$$
$$\phi(\pi_1, \pi_2) = 0$$
$$\pi_1 = u^{a_1} d^{b_1} \rho^{c_1} \mu$$
$$\pi_2 = u^{a_2} d^{b_2} \rho^{c_2} F$$

As each π group is dimensionless then considering the dimensions, for the first group, π

$$M^0 L^0 T^0 = (LT^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} ML^{-1}T^{-1}$$

M] $0 = c_1 + 1$

$c_1 = -1$

L] $0 = a_1 + b_1 - 3c_1 - 1$

$-2 = a_1 + b_1$

T] $0 = -a_1 -$

1

$a_1 = -1$

$b_1 = -1$

$$\pi_1 = u^{-1} d^{-1} \rho^{-1} \mu$$
$$= \frac{\mu}{\rho u d}$$

i.e. the (inverse of) Reynolds number

And the second group π_2:

$$M^0 L^0 T^0 = \left(LT^{-1}\right)^{a_2} (L)^{b_2} \left(ML^{-3}\right)^{c_2} ML^{-1}T^{-2}$$

M] $0 = c_2 + 1$

$c_2 = -1$

L] $0 = a_2 + b_2 - 3c_2 - 1$

$-3 = a_2 + b_2$

T] $0 = -a_2 - 2$

$a_2 = -2$

$b_2 = -1$

$$\pi_2 = u^{-2} d^{-1} \rho^{-1} F$$
$$= \frac{F}{u^2 d \rho}$$

So the physical situation is described by this function of nondimensional numbers,

$$\phi(\pi_1, \pi_2) = \phi\left(\frac{\mu}{\rho u d}, \frac{F}{\rho d u^2}\right) = 0$$

For <u>dynamic similarity</u> these non-dimensional numbers are the same for the both water and air in the pipe.

$$\pi_{1_{air}} = \pi_{1_{water}}$$
$$\pi_{2_{air}} = \pi_{2_{water}}$$

To find the ratio of forces for the different fluids use π_2

$$\pi_{2\,air} = \pi_{2\,water}$$

$$\left(\frac{F}{\rho u^2 d}\right)_{air} = \left(\frac{F}{\rho u^2 d}\right)_{water}$$

$$\left(\frac{F}{\rho u^2 d}\right)_{air} = \left(\frac{F}{\rho u^2 d}\right)_{water}$$

$$\frac{F_{air}}{F_{water}} = \frac{1.12 \times 33^2 \times 1}{1000 \times 11.55^2 \times 0.16} = 0.057$$

4. If the resistance to motion, R, of a sphere through a fluid is a function of the density ρ and viscosity μ of the fluid, and the radius r and velocity u of the sphere, show that R is given by

$$R = \frac{\mu^2}{\rho} f\left(\frac{\rho u r}{\mu}\right)$$

Hence show that if at very low velocities the resistance R is proportional to the velocity u, then R = k $\mu r u$ where k is a dimensionless constant.

A fine granular material of specific gravity 2.5 is in uniform suspension in still water of depth 3.3m. Regarding the particles as spheres of diameter 0.002cm find how long it will take for the water to clear. Take k=6 π and μ =0.0013 kg/ms.

[218mins 39.3sec]

Choose the three recurring (governing) variables; u, r, ρ, R, μ.

From Buckinghams π theorem we have m-n = 5 - 3 = 2 non-dimensional groups.

$$\phi(u, r, \rho, \mu, R) = 0$$

$$\phi(\pi_1, \pi_2) = 0$$

$$\pi_1 = u^{a_1} r^{b_1} \rho^{c_1} \mu$$

$$\pi_2 = u^{a_2} r^{b_2} \rho^{c_2} R$$

As each π group is dimensionless then considering the dimensions, for the first group, π_1

$$M^0 L^0 T^0 = (LT^{-1})^{a_1} (L)^{b_1} (ML^{-3})^{c_1} ML^{-1}T^{-1}$$

M] $0 = c_1 + 1$

$c_1 = -1$

L] $0 = a_1 + b_1 - 3c_1 - 1$

$-2 = a_1 + b_1$

T] $0 = -a_1 -$

1

$a_1 = -1$

$b_1 = -1$

$$\pi_1 = u^{-1} r^{-1} \rho^{-1} \mu$$
$$= \frac{\mu}{\rho u r}$$

i.e. the (inverse of) Reynolds number

And the second group π_2:

$$M^0 L^0 T^0 = \left(LT^{-1}\right)^{a_2} (L)^{b_2} \left(ML^{-3}\right)^{c_2} ML^{-1}T^{-2}$$

M] $0 = c_2 + 1$

$c_2 = -1$

L] $0 = a_2 + b_2 - 3c_2 - 1$

$-3 = a_2 + b_2$

T] $0 = -a_2 - 2$

$a_2 = - 2$

$b_2 = -1$

$$\pi_2 = u^{-2} r^{-1} \rho^{-1} R$$
$$= \frac{R}{u^2 r \rho}$$

So the physical situation is described by this function of nondimensional numbers,

$$\phi(\pi_1, \pi_2) = \phi\left(\frac{\mu}{\rho u r}, \frac{R}{\rho r u^2}\right) = 0$$

or

$$\frac{R}{\rho r u^2} = \phi_1\left(\frac{\mu}{\rho u r}\right)$$

he question asks us to show $R = \frac{\mu^2}{\rho} f\left(\frac{\rho u r}{\mu}\right)$ or $\frac{R\rho}{\mu^2} = f\left(\frac{\rho u r}{\mu}\right)$

Multiply the LHS by the square of the RHS: (i.e. $\pi_2(1/\pi_1)^2$))

$$\frac{R}{\rho r u^2} \times \frac{\rho^2 u^2 r^2}{\mu^2} = \frac{R\rho}{\mu^2}$$

So

$$\frac{R\rho}{\mu^2} = f\left(\frac{\rho u r}{\mu}\right)$$

The question tells us that R is proportional to u so the function *f* must be a constant, *k*

$$\frac{R\rho}{\mu^2} = k\frac{\rho u r}{\mu}$$
$$R = \mu k r u$$

The water will clear when the particle moving from the water surface reaches the bottom.

At terminal velocity there is no acceleration - the force R = mg - upthrust.

From the question:

$\sigma = 2.5$ so $\rho = 2500 kg/m^3$ $\mu = 0.0013$ kg/ms k = 6π

r = 0.00001m depth = 3.3m

$$mg = \frac{4}{3}\pi 0.00001^3 \times 9.81 \times (2500 - 1000)$$
$$= 6.16 \times 10^{-11}$$

$$\mu k r u = 0.0013 \times 6\pi \times 0.00001 u = 6.16 \times 10^{-11}$$
$$u = 2.52 \times 10^{-4} m/s$$

$$t = \frac{3.3}{2.52 \times 10^{-4}} = 218 \text{min } 39.3 \text{sec}$$

UNIT IV

CENTRIFUGAL

PUMPS

A **centrifugal pump** is a rotodynamic pump that uses a rotating impeller to increase the pressure of a fluid. Centrifugal pumps are commonly used to move liquids through a piping system. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits into the downstream piping system. Centrifugal pumps are used for large discharge through smaller heads.

CENTRIFUGAL PUMPS

Modern process plants use powerful centrifugal pumps, primarily because of the following factors :

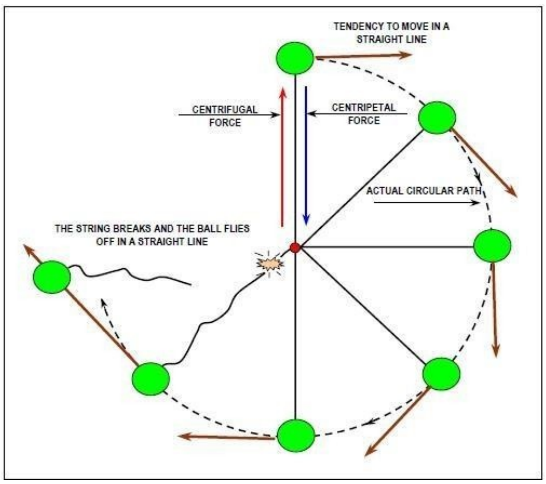
1. The low initial cost.
2. Low maintenance costs.
3. Simple in operation.
4. Ability to operate under a wide variety of conditions.
5. Give a smooth, continuous flow, free from pulsation.

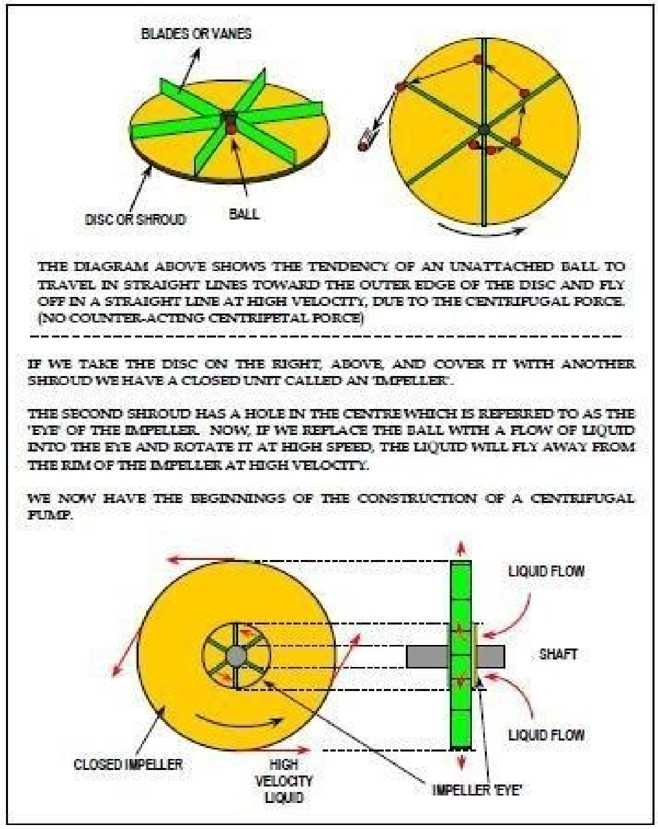
CENTRIFUGAL FORCE

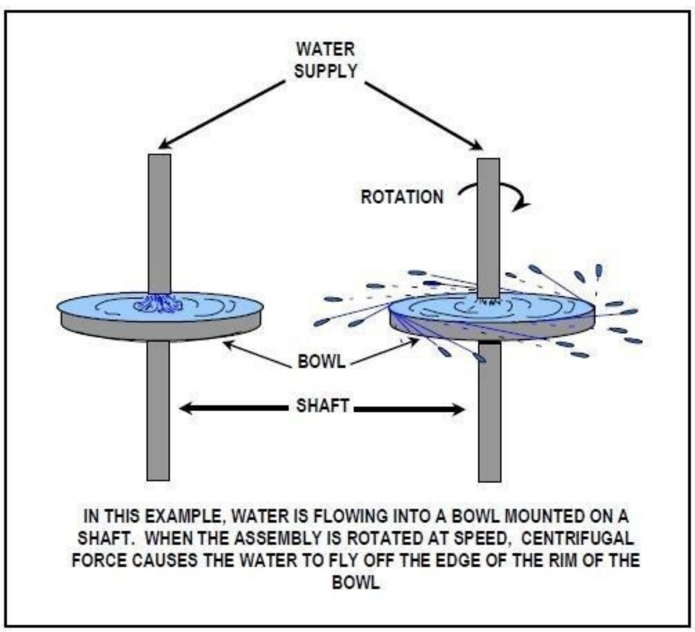
The word, ' centrifugal ' is derived from the latin language and is formed from two words 'centri' meaning 'centre' and 'fugal' meaning 'to fly away from'.

Centrifugal - 'to fly away from the centre'.

This is the force developed due to the rotation of a body - solid, liquid or gas. The force of rotation causes a body, or a fluid, to move away from the centre of rotation.







Parts of a Centrifugal Pump

A centrifugal pump is built up of two main parts:

1. THE ROTOR (or Rotating Element).
2. THE CASING (or Housing or Body).

The Rotor

One of the greatest advantages of a centrifugal pump is that it has very few moving parts which minimises mechanical problems and energy losses due to friction.
Other than the bearings, (and of course the driver), the only moving part in a centrifugal pump is the Rotor.
The Rotor (Rotating Element), is made up of the following main components :

1. THE IMPELLER(S) -Often called the 'Wheel(s)'. (In the centre of an impeller, is the 'EYE' which receives the inlet flow of liquid into the 'Vanes' of the impeller).
2. THE SHAFT -The impeller(s) is/are mounted on the shaft and enclosed by a casing.

The Impellers

These consist of wheel shaped elements containing 'Curved Vanes' at the centre of which is the liquid inlet called the 'EYE' of the impeller.

The wheel(s) is/are mounted on the shaft, (together called 'the Rotating Element' which is rotated at high speed. The liquid is thrown off the outer edge of the vanes, and more liquid flows into the eye to take its place. The speed of rotation of the wheel imparts kinetic energy to the liquid in the form of velocity which will be converted to pressure (potential) energy.

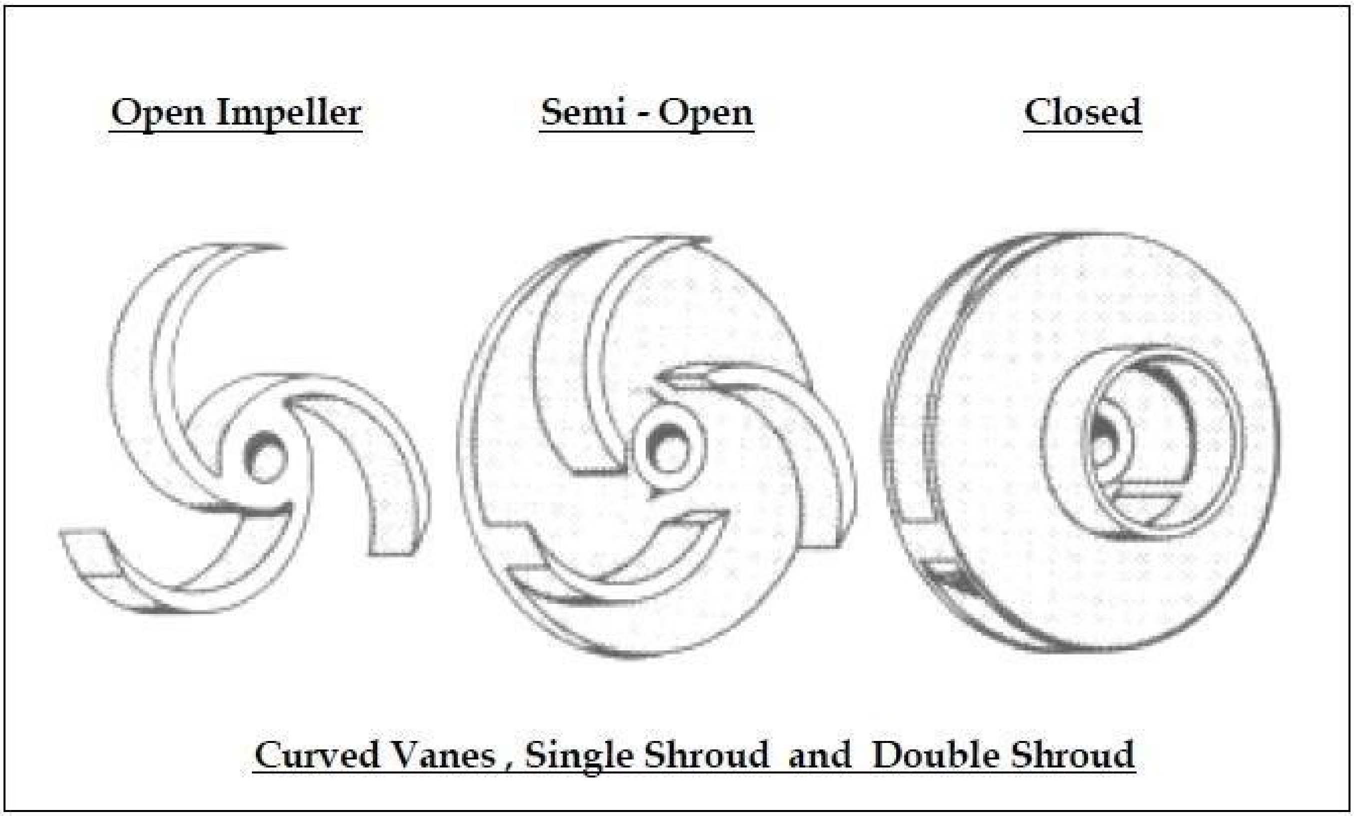
There are various types of impeller depending on the duty to be performed by the pump.

1. The Open Impeller : This type consists of vanes attached to a central hub with no side wall or 'shroud'. It is used for pumping highly contaminated slurry type liquids.
2. Semi-Open Impeller : This type has the vanes attached to a wall or shroud on one side. It is used mainly for lightly contaminated and abrasive liquids and slurries.
3. Closed Impeller : This impeller has the vanes enclosed on both sides by a shroud and is the most efficient impeller, used for clean or very slightly contaminated liquids. Impellers can also be

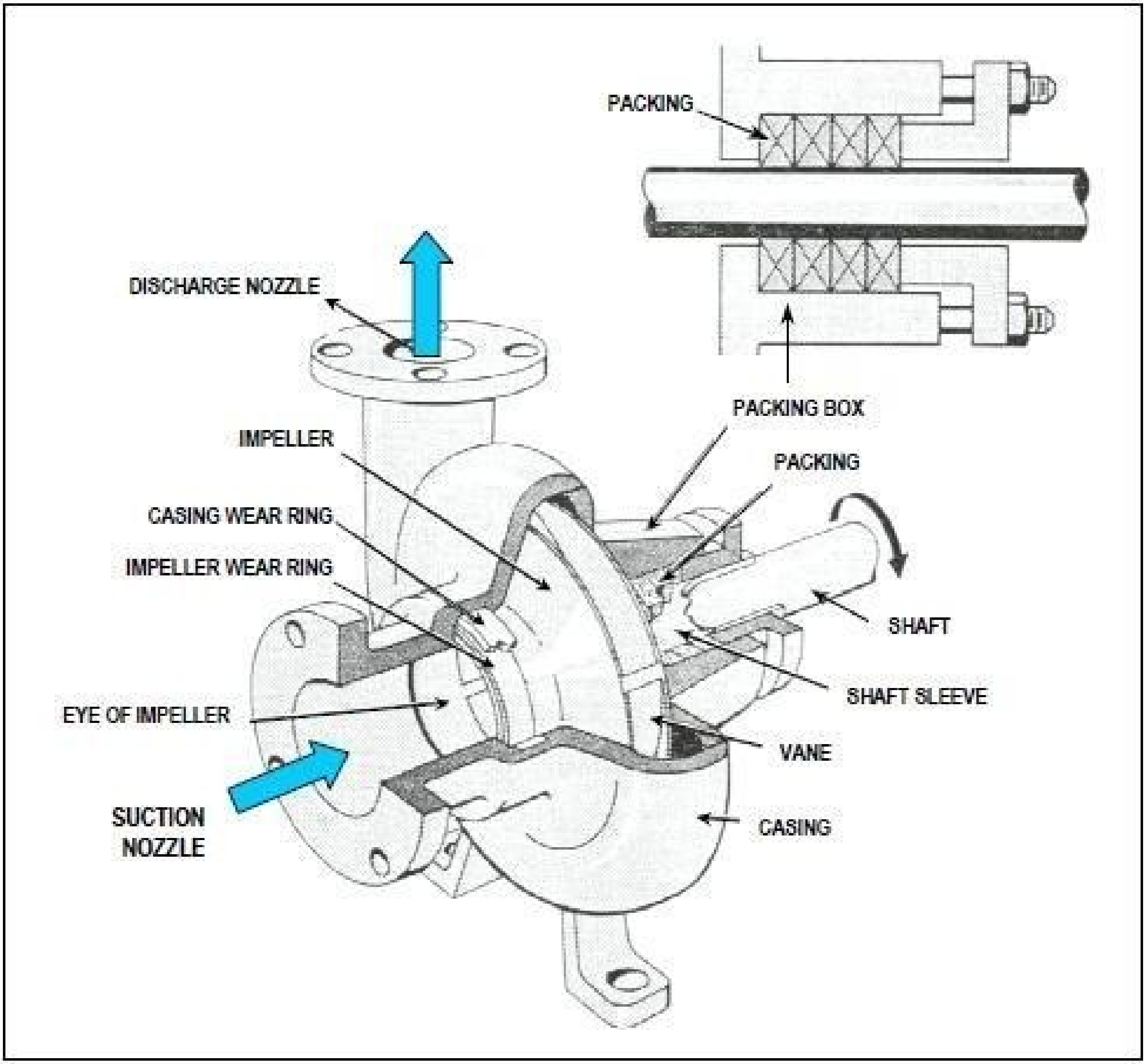
classified according to the vane curvature - i.e. 'Backward' curve used for high flow rate. 'Forward' curve for high liquid head and 'Straight' for either service.

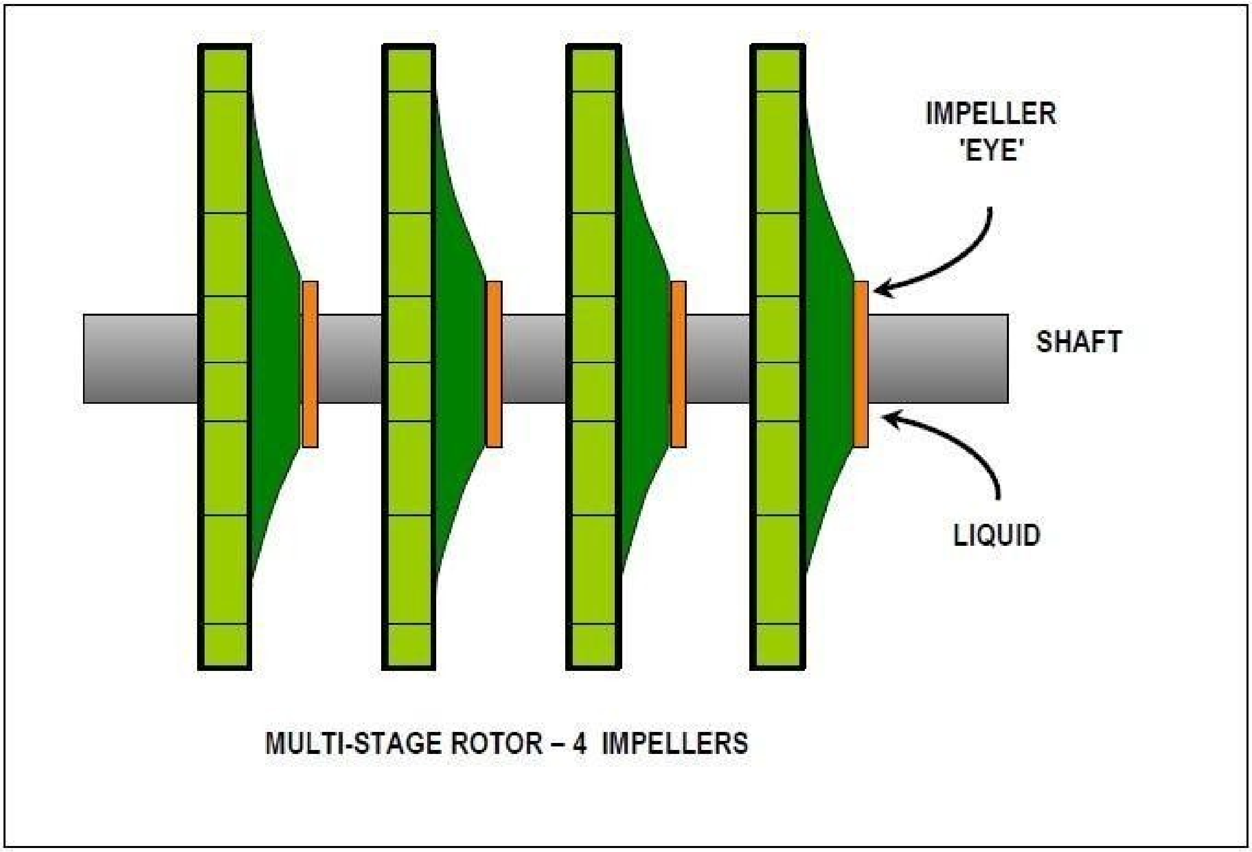
Types of Impeller

High power, high volume pumps are fitted with more than one impeller. This type is called a 'Multi-stage' pump and is actually a series of pumps mounted on the shaft within a single casing. The liquid leaving each impeller rim, is fed into the eye of the next wheel. In this way, the pressure is built up in stages through the pump. The more stages, the higher the discharge pressure. As liquids cannot be compressed and therefore no change in volume takes place, the impellers of a multi-stage pump are all the same size – (unlike those of a compressor).How the liquid is passed from stage to stage is discussed later in the notes on the casing.



How the liquid is passed from stage to stage is discussed later in the notes on the casing.





The Shaft

The Impeller(s) are mounted on this part of the pump which is then referred to as the 'Rotor' or rotating element which is coupled (connected) to the pump driver. The driver imparts the rotation to the rotor that is housed in the casing, supported by the bearings.

The shaft, due to the high speed of rotation, will tend to move :-

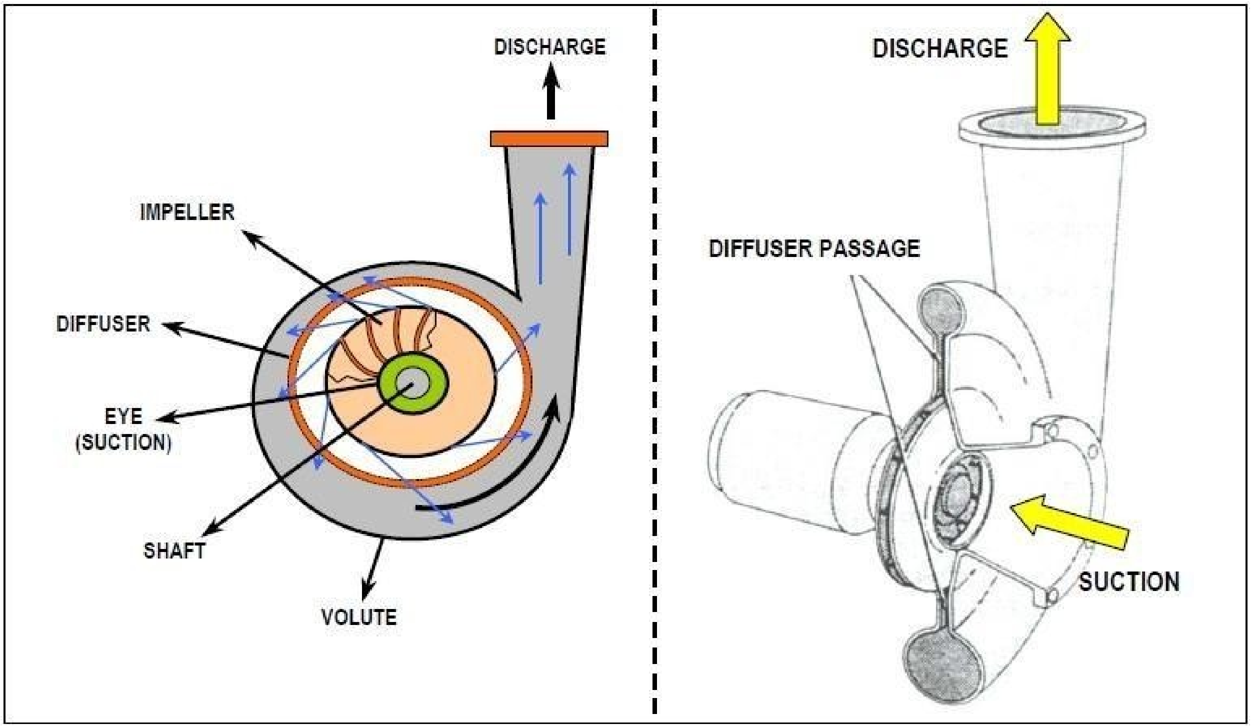
- Radially -movement across the shaft (Vibration) and,
- Axially -movement along the shaft (Thrust).

In order to minimise and control these movements, bearings are fitted (as discussed earlier).

The Casing

This is the stationary part of the pump and includes the :

1. Suction Nozzle(s) (or Port(s)).
2. Discharge Nozzle (or Port).
3. Bearings.
4. Seals.



The casing of a multi-stage centrifugal pump is very similar to that of a multi-stage compressor having diaphragms with diffusers & return passages. However, as liquids are non-compressible, the stages do not become progressively narrower.The 'Volute' casing. This is named from the spiral shape of the casing which is so constructed to act as a collector for the liquid as it leaves the outer edge of the impeller

vanes.

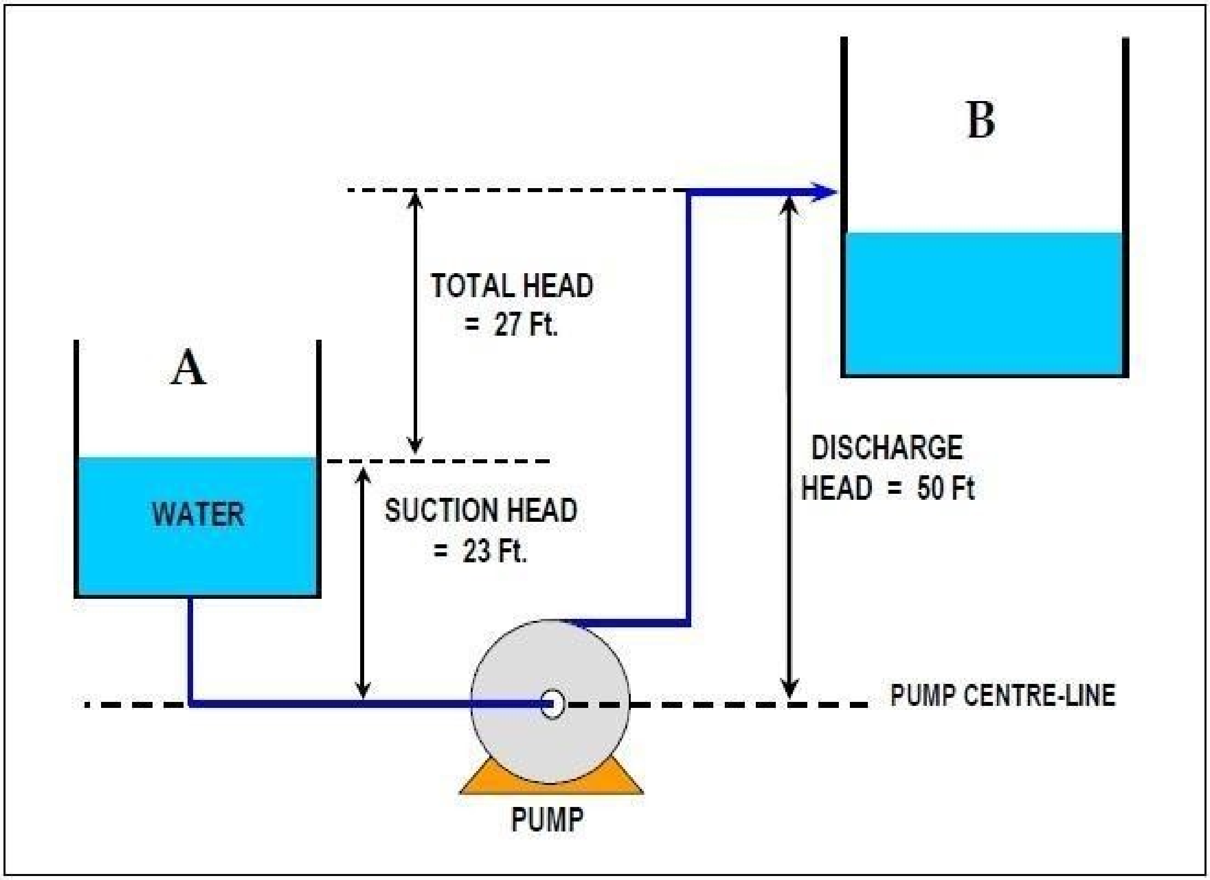
The liquid at this point is at high velocity. As the liquid enters the volute, the velocity is decreased. This causes an increase in pressure which is the objective of the pump. (Increased pressure is increased energy).

Prime Movers

The prime movers for pumps are the devices used to drive them - whether they are rotating machines or otherwise.

The types of prime mover used for modern pumps are :

1. Electric Motor.
2. Diesel (or petrol) engine.
3. Gas Turbine.
4. Steam Turbine.



CHARACTERISTICS OF CENTRIFUGAL PUMPS

Centrifugal pumps are specified by four characteristics.

Capacity:

This is defined as the quantity of liquid which is discharged from the pump in a given time. Capacity is expressed in 'm3/hr', 'gal/min', ..etc. The capacity of a pump is governed by the 'Head', the 'Speed' and the 'Size' of the pump.

Total Head:

The total head of a pump is the difference between the pump suction and discharge pressures - expressed in terms of metres or feet head :

Suction Head :

This is the vertical distance, in feet or metres, from the centreline of the pump to the level of liquid in the vessel from which the liquid is being pumped.

If the liquid level is above the pump centreline, the suction head is positive. If below the centreline, the suction head is negative.

Discharge Head:

Is the discharge pressure of the pump, expressed in feet or metres of liquid.

Total Head: = Discharge head -Suction head

Power:

This is the energy used by the pump in a given time. Its unit is 'Horsepower' (HP). 1 HP is equivalent to 0.746 kilowatt. (kW).

Efficiency:

This is a percentage measure of the pump's effectiveness in transferring the power used into energy added to the pumped liquid.

The formula for calculation of efficiency is:

Efficiency = (Output power)/(Input power)X 100%

Pumps in industry, usually operate at 70% to 80% efficiency.

The pump is taking suction from Tank 'A' and discharging to Tank 'B'. The Head (or height) of water in 'A' to the centre-line of the pump is 23 feet. This is called the 'SUCTION HEAD'.

The discharge line inlet to 'B' is 50 feet above the pump centre-line. This is the 'DISCHARGE HEAD'.

The 'TOTAL HEAD' is the difference between the two figures.

This is 50 -23 = 27 feet.

If the suction vessel is BELOW the pump centre line, the suction head will be a NEGATIVE figure.

Using the formula for Static Head Pressure, we can find the suction and discharge pressures of the pump. (Both tanks are at atmospheric pressure).

Suction pressure = 23 x 0.433 = 10 Psig. Discharge pressure = 50 x 0.433 = 21.7 Psig

If a liquid other than water is used, the Specific Gravity of the liquid must be included in the above formula to obtain the pressures.

E.g.

If we use an oil with S.G. of 0.88, the pressures would be: -

Suction pressure = 23 x 0.433 x 0.88 = 8.8 Psig.

Discharge pressure = 50 x 0.433 x 0.88 = 19.1 Psig

NET POSITIVE SUCTION HEAD REQUIRED

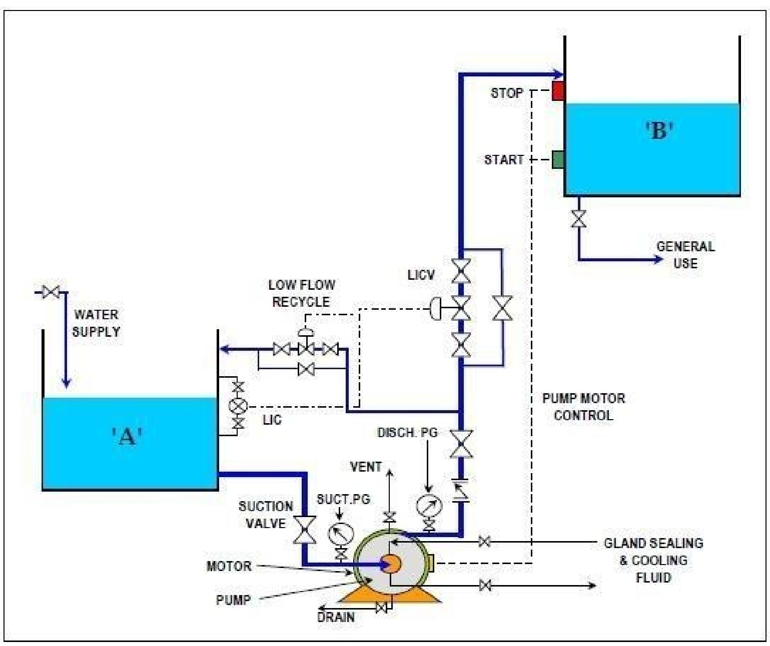
The pump manufacturer's specified margin of suction pressure above the boiling point of the liquid being pumped, is required to prevent cavitation. This pressure is called the 'Net Positive Suction Head' pressure (NPSH).

In order to ensure that a NPSH pressure is maintained, the Available NPSH should be higher than that required. The NPSH depends on the height and density of the liquid and the pressure above it.

The Valves of a Centrifugal Pump

The suction and discharge piping of a centrifugal pump, will generally have the following valve arrangements :

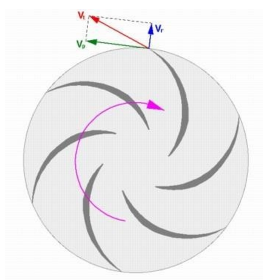
1. Suction Valve: Allows liquid to enter the pump.
2. Discharge Valve: Allows liquid to flow from the pump to other parts of the system.
3. Check or Non-Return Valve: In the discharge line -Prevents back-flow from discharge to suction through the pump.
4. Vent (priming) Valve: This is used to vent off air/gases from the pump before start-up.
5. Gauge Isolation Valves: Allows the replacement of pressure gauges on suction and discharge lines, the most important being the discharge pressure.
6. Gland Seal Valve: (where fitted). Controls the flow of cooling media to the pump gland cooling fluid.
7. Recycle Valve: This is a flowline valve which is used to recycle pumped liquid back to the suction side or to the suction vessel, in order to maintain a flow through the pump when the discharge valve, (and/or FCV), is closed. (Prevents heat build-up).
8. Drain Valve: Fitted on the bottom of the pump casing and used to drain the pump prior to maintenance work being done.



Working principle

A centrifugal pump works by converting kinetic energy into potential energy measurable as static fluid pressure at the outlet of the pump. This action is described by Bernoulli's principle.

With the mechanical action of an electric motor or similar, the rotation of the pump impeller imparts kinetic energy to the fluid through centrifugal force. The fluid is drawn from the inlet piping into the impeller intake eye and is accelerated outwards through the impeller vanes to the volute and outlet piping. As the fluid exits the impeller, if the outlet piping is too high to allow flow, the fluid kinetic energy is converted into static pressure. If the outlet piping is open at a lower level, the fluid will be released at greater speed.



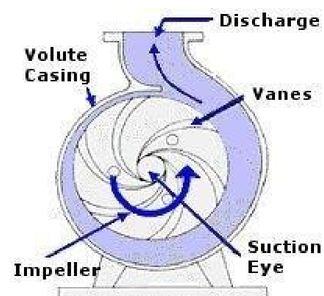


FIG 1

A centrifugal pump uses a spinning "impeller" which normally has backward-swept blade

CENTRIFUGAL PUMPS - FLOW & PRESSURE

In Parallel

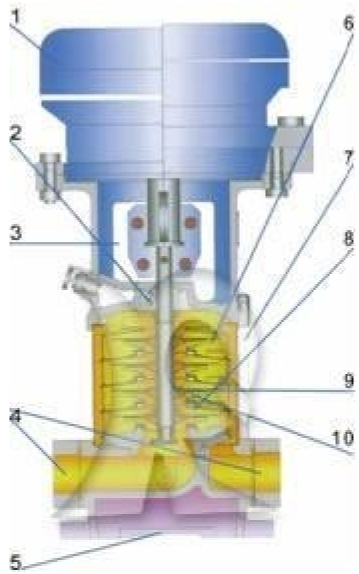
where extra flow is required, two or more pumps can be operated in 'parallel'. This means that the pumps all take suction from a common header and discharge into another common header. The number of pumps in the parallel line-up, depends on the system flow requirements.

In Series

where extra pressure is required, pumps may be operated in 'series'. Here, a pump takes suction from a vessel and discharges into the suction of another pump which then discharges into the system. The number of pumps lined up in series depends on the system pressure requirements.

Vertical centrifugal pumps

Vertical centrifugal pumps are also referred to as cantilever pumps. They utilize a unique shaft and bearing support configuration that allows the volute to hang in the sump while the bearings are outsideof the sump. This style of pump uses no stuffing box to seal the shaft but instead utilizes a "throttle Bushing". A common application for this style of pump is in a parts washer.



Construction:

Vertical multistage centrifugal pump, suitable for clean, watery liquids.

Equipped with ceramic wear-resistant liquid lubricated bearings.

Shaft sealing by means of mechanical seal.

Pump fulfils the latest safety regulations (CE-marking).

Sleeve sealing by means of O-rings.

Connections in-line with standard build-in sizes.

All hydraulic components such as shaft,

pumpfoot etc., manufactured of stainless steel AISI 304. Base plate and motor lantern made of cast iron.

Base plate protected by coating.

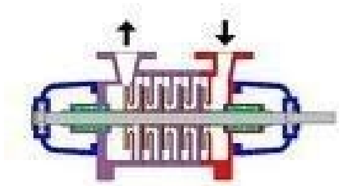
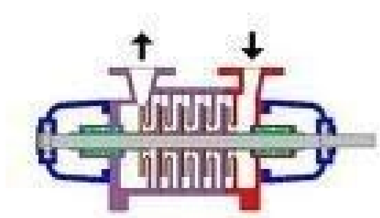
Product characteristics for Vertical Multistage Centrifugal Pump

1. Low noise: Install the model number of Y2 electric motor,the circulate is steady,and low noise .
2. Without leakage: Specially selected mechanical seal and science processes a craft,professional technical assurance,carry out the pursue of"without leakage"
3. Simple to demolish and install: Rigid couplings,spline shaft design,the weight ease 50% than the pumps,and the operability is better
4. Install is convenient: Enter of water and out of water are on the same level,the Lower to the tube road request,the the and tube road the s usage is more dependable,the stability is higher
5. The investment of pump - house - building is less: At the orginally foundation of vertical

6. multistage pump go forward to go an improvement,the the physical volume is small,covering area little
7. Avoid the second-pollution of the fluid matter: Inducer and shaft are made of stainless steel,overcome the weakness of the cast-iron pump,promise the quality of the fluid matter
8. Have no rust eclipse harassment: The use cylinder of stainless steel,better adapt various environment,never rusty
9. The life span is longer: The material hurtles of stainless steel are pressed,the weld impeller,the weight is light,balance good,circulate a stability
10. Support expenses low: The Adopt the water-lubrication-bearing design,Circulate more dependable,need not to lubriicate to maintain
11. Efficiently and economize on energy: The More excellent water power model,the smooth of over flow a parts,raised a machine efficiency consumedly

Multistage centrifugal pumps

A centrifugal pump containing two or more impellers is called a multistage centrifugal pump. The impellers may be mounted on the same shaft or on different shafts.If we need higher pressure at the outlet we can connect impellers in series.If we need a higher flow output we can connect impellers in parallel.All energy added to the fluid comes from the power of the electric or other motor force driving the impeller.


Pump Efficiency

Pump efficiency may be defined as the ratio of power added to the fluid in relation to the power required to drive the pump. Efficiency is never a single fixed value for any given centrifugal pump, it is a function of discharge and therefore also the operating head. The efficiency tends to increase with flow rate up to a point midway through the operating range (peak efficiency) and then declines as flow rates rise further, the resulting curve is almost parabolic in shape.

It is important that the system is designed in order that the pump will be operating at or close to its peak efficiency. This is particularly difficult for those systems with time variant flow rates, such difficulties may be overcome through the use of variable frequency drives to adjust the speed of the electric drive motor. Unless carefully designed, installed and monitored, pumps will be, or will become inefficient, wasting a lot of energy. Efficiency will decline over time due to wear of the impeller and hence should be regularly tested to ensure they are operating normally.

Single Stage Radial Flow Centrifugal Pump

Energy usage

The energy usage in a pumping installation is determined by the flow required, the height lifted and the length and friction characteristics of the pipeline. The power required to drive a pump (P_i), is defined simply using SI units by:

$$P_i = \rho\ g\ H\ Q\ /\ \eta$$

Where:

P_i is the input power required (W)

ρ is the fluid density (kg/m^3)

g is the standard acceleration of gravity (9.80665 m/s^2)

H is the energy Head added to the flow (m)

Q is the flow rate (m^3/s)

η is the efficiency of the pump plant as a decimal

The head added by the pump (H) is a sum of the static lift, the head loss due to friction and any losses due to valves or pipe bends all expressed in metres of fluid. Power is more commonly expressed as kilowatts (10^3 W) or horsepower (multiply kilowatts by 0.746). The value for the pump efficiency η may be stated for the pump itself or as a combined efficiency of the pump and motor system.

The **energy usage** is determined by multiplying the power requirement by the length of time the pump is operating.

CAVITATION

Cavitation is a problem condition which may develop while a centrifugal pump is operating. This occurs when a liquid boils inside the pump due to insufficient suction head pressure. Low suction head causes a pressure below that of vaporisation of the liquid, at the eye of the impeller.

The resultant gas which forms causes the formation and collapse of 'bubbles' within the liquid. This, because gases cannot be pumped together with the liquid, causes violent fluctuations of pressure within the pump casing and is seen on the discharge gauge. These sudden changes in pressure cause vibrations which can result in serious damage to the pump and, of course, cause pumping inefficiency.

To overcome cavitation:

1. Increase suction pressure if possible.
2. Decrease liquid temperature if possible.
3. Throttle back on the discharge valve to decrease flow-rate.
4. Vent gases off the pump casing.

AIR BINDING IN A CENTRIFUGAL PUMP

Air binding occurs when air is left in a pump casing due to improper venting, or, air collects when the pump is operating. The air, as it collects, forms a pocket around the impeller which forces liquid away from it. The impeller then spins in the air and heat begins to build up.

Symptoms of air binding:

1. Fluctuating pressure for a short time. The pressure may then stop jumping and fall quickly.
2. Overheating of the pump may take place shortly after air binding occurs.
3. An air-bound pump sounds quieter than

normal. To correct air binding:

1. Vent the pump during operation.
2. In some cases, the pump must be shut down and allowed to cool. The air must then be vented off.

POSITIVE DISPLACEMENT PUMPS

Positive-displacement pumps are another category of pumps. Types of positive-displacement pumps are reciprocating, metering, and rotary pumps. Positive-displacement pumps operate by forcing a fixed volume of fluid from the inlet pressure section of the pump into the discharge zone of the pump. These pumps generally tend to be larger than equal-capacity dynamic pumps. Positive-displacement pumps frequently are used in hydraulic systems at pressures ranging up to 5000 psi. A principal advantage of hydraulic power is the high power density (power per unit weight) that can be achieved. They also provide a fixed displacement per revolution and, within mechanical limitations, infinite pressure to move fluids. Positive displacement means that, when the pump piston or rotor moves, fluid moves and displaces the fluid ahead of it. Because of its operation, a positive displacement pump can build up a very high discharge pressure and, should a valve in the discharge system be closed for any reason, serious damage may result

- the cylinder head, the casing or other downstream equipment may rupture or the driver may stall and burn out.

A Positive Displacement pump must therefore be fitted with a safety relief system on the discharge side.

TYPES OF POSITIVE DISPLACEMENT PUMP

- ROTARY PUMPS
- RECIPROCATING (PISTON) PUMPS

Rotary Pumps

In Rotary pumps, movement of liquid is achieved by mechanical displacement of liquid produced by rotation of a sealed arrangement of intermeshing rotating parts within the pump casing.

THE GEAR PUMP

Construction and Operation:

In this pump, intermeshing gears or rotors, rotate in opposite directions, just like the gears in a vehicle or a watch mechanism. The pump rotors are housed in the casing or stator with a very small clearance between them and the casing. (The fluid being pumped will lubricate this small clearance and help prevent friction and therefore wear of the rotors and casing).

1. In this type of pump, only one of the rotors is driven. The intermeshing gears rotate the other rotor. As the rotors rotate, the liquid or gas, (this type of machine can also be used as a compressor), enters from the suction line and fills the spaces between the teeth of the gears and becomes trapped forming small 'Slugs' of fluid between the teeth.
2. The slugs are then carried round by the rotation of the teeth to the discharge side of the pump.
3. At this point, the gears mesh together and, as they do so, the fluid is displaced from each cavity by the intermeshing teeth.
4. Since the fluid cannot pass the points of near contact of the intermeshed teeth nor between the teeth and casing, it can only pass into the discharge line.
5. As the rotation continues, the teeth at the suction end are opened up again and the same amount of fluid will fill the spaces and the process repeated. The liquid at the discharge end is constantly being displaced (moved forward).

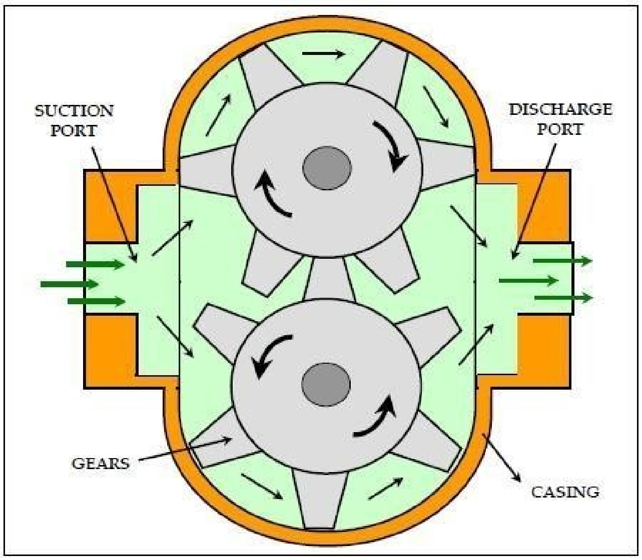
Thus gear pumps compel or force a fixed volume of fluid to be displaced for each revolution of the rotors giving the 'Positive Displacement' action of the pump.

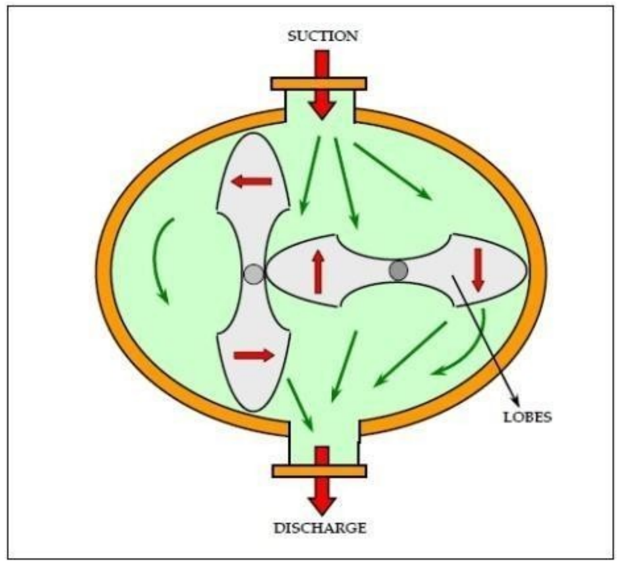
Gear pumps are generally operated at high speed and thus give a fairly pulse-free discharge flow and pressure. Where these pumps are operated at slower speeds, as in pumping viscous liquids, the output tends to pulsate due to the meshing of the teeth.

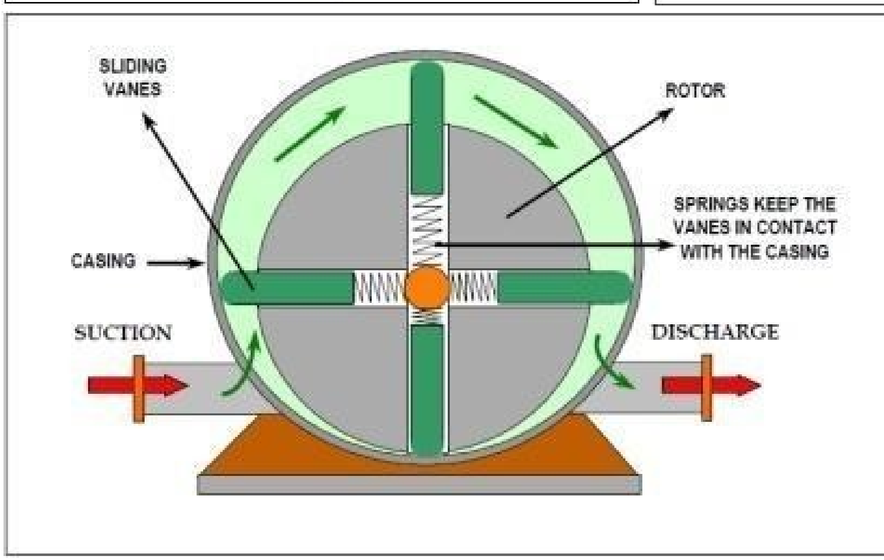
Any gas or air drawn into the pump with the liquid, will be carried through with the liquid and will not cause cavitation. This action of the pump means that it's a 'Self Priming' pump. The discharge pressure may however, fluctuate.

The output from this type of pump is directly proportional to the speed of operation. If the speed is doubled, the output will be doubled and the pressure will have very little effect. (*At higher pressures, due to the* *fine clearances between the teeth and between the casing and the rotors, a small leakage backto the suction side will occur resulting in a very small drop in actual flow rate. The higher the dischargepressure, the more likely that internal leakage will occur*).

Rotary pumps are widely used for viscous liquids and are self-lubricating by the fluid being pumped. This means that an external source of lubrication cannot be used as it would contaminate the fluid being pumped. However, if a rotary pump is used for dirty liquids or slurries, solid particles can get between the small clearances and cause wear of the teeth and casing. This will result in loss of efficiency and expensive repair or replacement of the pump.



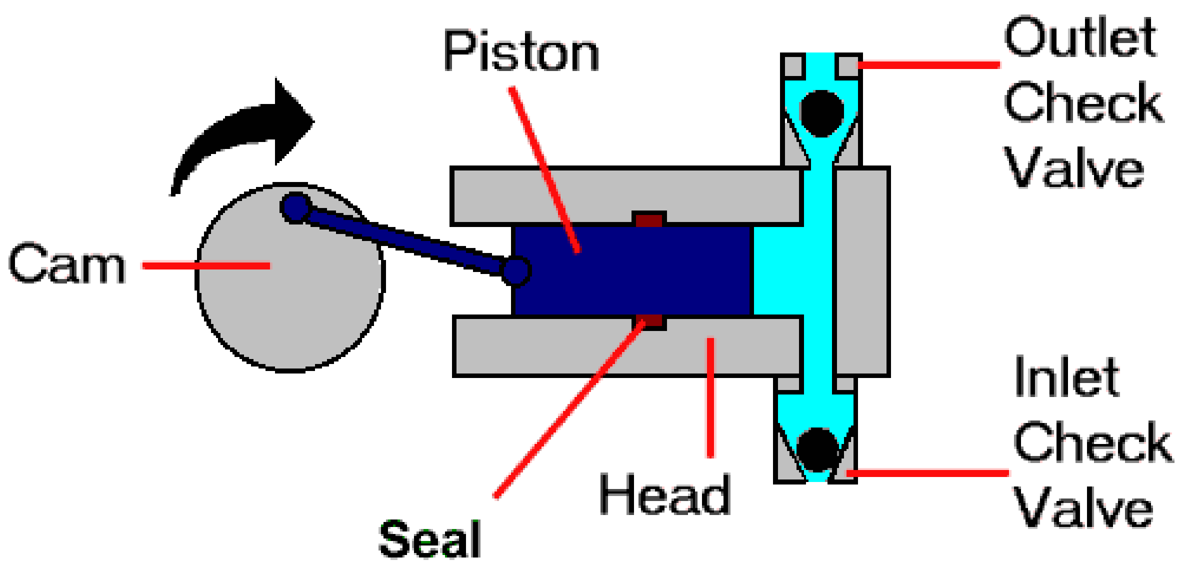




Reciprocating Pumps

In a reciprocating pump, a volume of liquid is drawn into the cylinder through the suction valve on the intake stroke and is discharged under positive pressure through the outlet valves on the discharge stroke. The discharge from a reciprocating pump is pulsating and changes changes only when the speed of the pump is changed. This is because the intake is always a constant volume. Often an air chamber is connected on the discharge side of the pump to provide a more even flow by evening out the pressure surges. Reciprocating pumps are often used for sludge and slurry.

One construction style of a reciprocating pump is the direct-acting steam pump. These consist of a steam cylinder end in line with a liquid cylinder end, with a straight rod conection between the steam piston and the pump piston or plunger. These pistons are double acting which means that each side pumps on every stroke. Another construction style is the power pump which convert rotary motion to low speed reciprocating motio using a speed reducing gear. The power pump can be either single or double-acting. A single- acting design discharges liquid only on one side of the piston or plunger. Only one suction and one discharge stroke per revolution of the crankshaft can occur. The double-acting design takes suction and discharges on both sides of the piston resulting in two suctions and discharges per crankshaft revolution.Power pumps are generally very efficient and can develop high pressures. These pumps do however tendto be expensive.



To 'Reciprocate' means 'To Move Backwards and Forwards'.
A 'Reciprocating' pump therefore, is one with a forward and backward operating action. The most simple reciprocating pump is the 'Bicycle Pump', which everyone at some time or other will have used to re-inflate their bike tyres. The name 'Bicycle PUMP' is not really the correct term because it causes compression.It is essentially a hand operated compressor and consists of a metal or plastic tube called a 'Cylinder' inside of which a hand-operated rod or 'Piston' is pushed back and forth. On the piston end, a special leather or rubber cup-shaped attachment is fixed.When the piston is pushed forward, (this is called a 'Stroke'), the cup flexes against the cylinder walls giving a seal to prevent air passing to the other side. As the pump handle is pushed, air pressure builds

up ahead of the cup and is forced (discharged) into the tyre through the tyre valve which also prevents air escaping when the pump is disconnected or when the piston is pulledback. When the pump handle is pulled back, (called the 'Suction' stroke), the cup relaxes and the backward motion causes air to pass between it and the cylinder wall to replace the air pushed into the tyre. This reciprocating action is repeated until the tyre is at the required pressure. Because the air is expelled from the pump during the forward stroke only, the pump is known as a 'Single Acting Reciprocating Pump'.

Single Acting Reciprocating Pumps

In industry, reciprocating pumps are of many sizes and designs. Their operation is similar to the bicycle pump described above.

An industrial reciprocating pump is constructed of metal and has the following main parts :

1. The cylinder

This is a metal tube-shaped casing (or body), which is generally fitted with a metal lining called a 'cylinder liner '. The liner is replaceable when it becomes worn and inefficient. The cylinder is also fitted with suction and discharge ports which contain special spring loaded valves to allow liquid to flow in one direction only - similar to check valves.

2. The piston

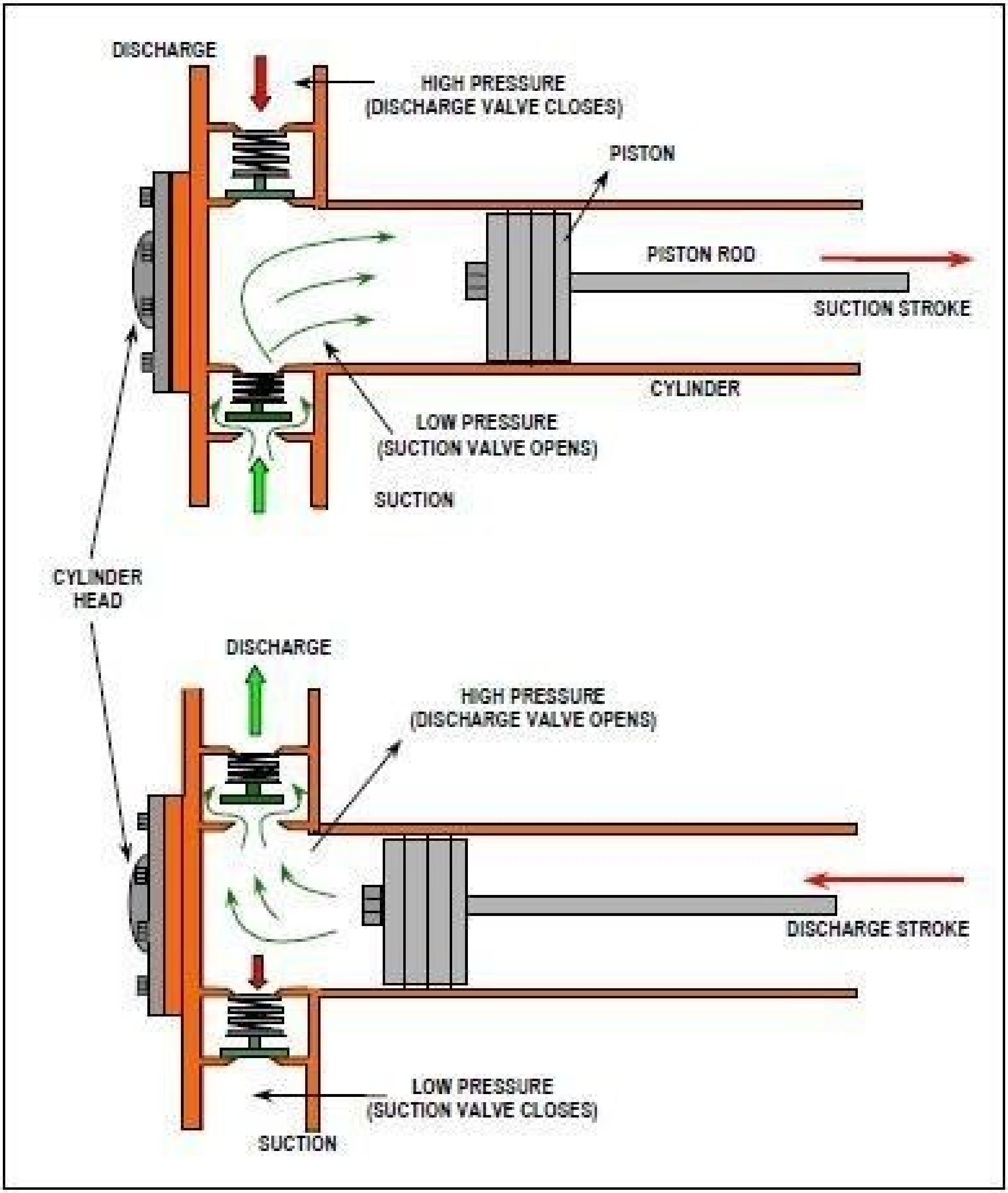
The piston consists of a metal drive rod connected to the piston head which is located inside the cylinder. The piston head is fitted with piston rings to give a seal against the cylinder lining and minimise internal leakage. The other end of the drive rod extends to the outside of the cylinder and is connected to the driver. (In the old days of piston pumps, the driver used to be (and still is in some cases), high pressure steam which was fed to a drive cylinder by a system of valves in a steam chest). Modern industries generally use high power electric motors, linkages and gearing to convert rotating motion into a reciprocating action.

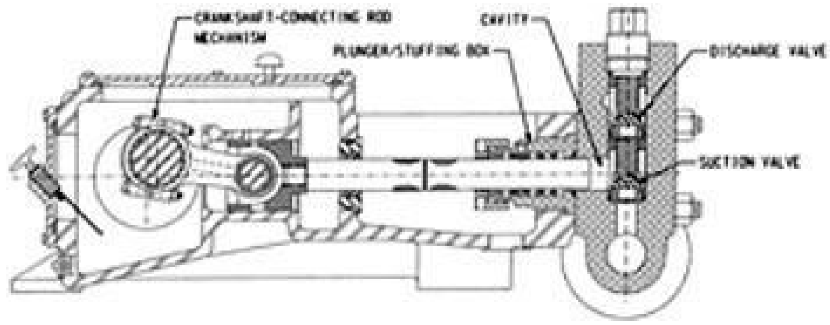
In a single acting pump, the backward stroke of the piston causes a suction which pulls in liquid through the inlet valve. (The same suction action keeps the discharge valve closed).

On the forward stroke, the increase in pressure generated by the piston, closes the inlet valve and opens the discharge valve. The liquid is displaced into the discharge system.

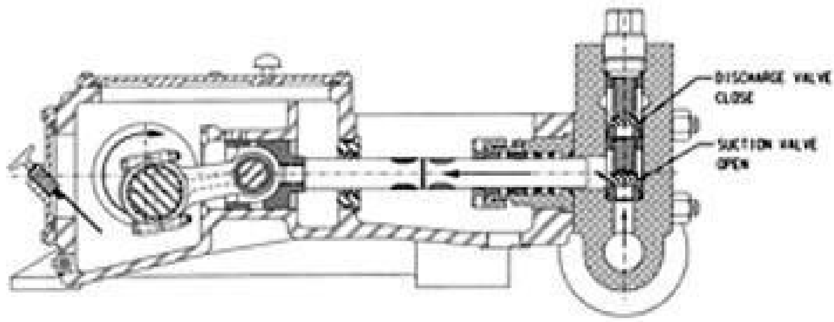
The flow from a reciprocating pump is uneven or pulsating. This can be undesirable in some applications. Flow can be smoothed out, but we will discuss this a little later.

Like the rotary pumps, because the action is positive displacement, a piston pump can generate very high pressure and therefore MUST NEVER be operated against a closed discharge system valve unless it is fitted with a safety relief system in order to prevent damage to the pump and/or the driver and/or other downstream equipment.

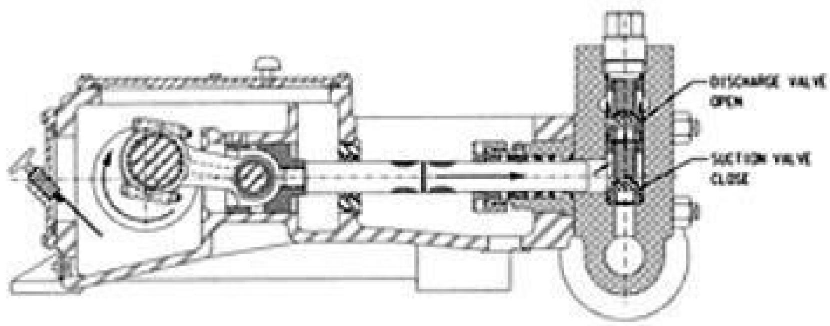




(A) TYPICAL RECIPROCATING PUMP(PLUNGER)



(B) SUCTION STROKE



(C) DISCHARGE STROKE

Double Acting Reciprocating Pumps

This type of pump operates in exactly the same way as the single acting with respect to its action. The difference is, that the cylinder has inlet and outlet ports at each end of the cylinder. As the piston moves forward, liquid is being drawn into the cylinder at the back end while, at the front end, liquid is being discharged. When the piston direction is reversed, the sequence is reversed.

With a double acting pump, the output pulsation is much less than the single acting.

In theory, a reciprocating pump will always deliver the same volume for each stroke regardless of discharge pressure. But, as discharge pressure is increased, there is more likelihood of internal leakage between the piston rings and the cylinder liner, or leaking internal valves, causing a decrease in output. A measure of this is known as the ' Volumetric Efficiency ' of the pump.

The amount of liquid which leaks internally is known as the ' Slip ' and, if the pump is in good condition, the slip should be below 1.0%. If slip is above 5.0%, the pump needs to be overhauled. However, at operating pressures, the amount of slip is relatively constant as long as wear is not rapid. The output therefore can still be classed as constant. This type of pump is useful for delivery of fixed quantities of liquid as used in metering or dosing operations.

The speed of a reciprocating pump is generally measured as ' Strokes per Minute '. This is the number of times the piston moves back and forth in one minute. Speed can also be measured as ' R.P.M.' of the drive motor.

As the cylinder(s) are of constant dimensions, the volume of liquid moved for each stroke, (discounting leakage described above), is the same and therefore the output per minute, hour or day ..etc can be calculated.

This type of pump operates in exactly the same way as the single acting with respect to its action. The difference is, that the cylinder has inlet and outlet ports at EACH END OF THE CYLINDER. As the piston moves forward, liquid is being drawn into the cylinder at the back end while, at the front end, liquid is being discharged. When the piston direction is reversed, the sequence is reversed.

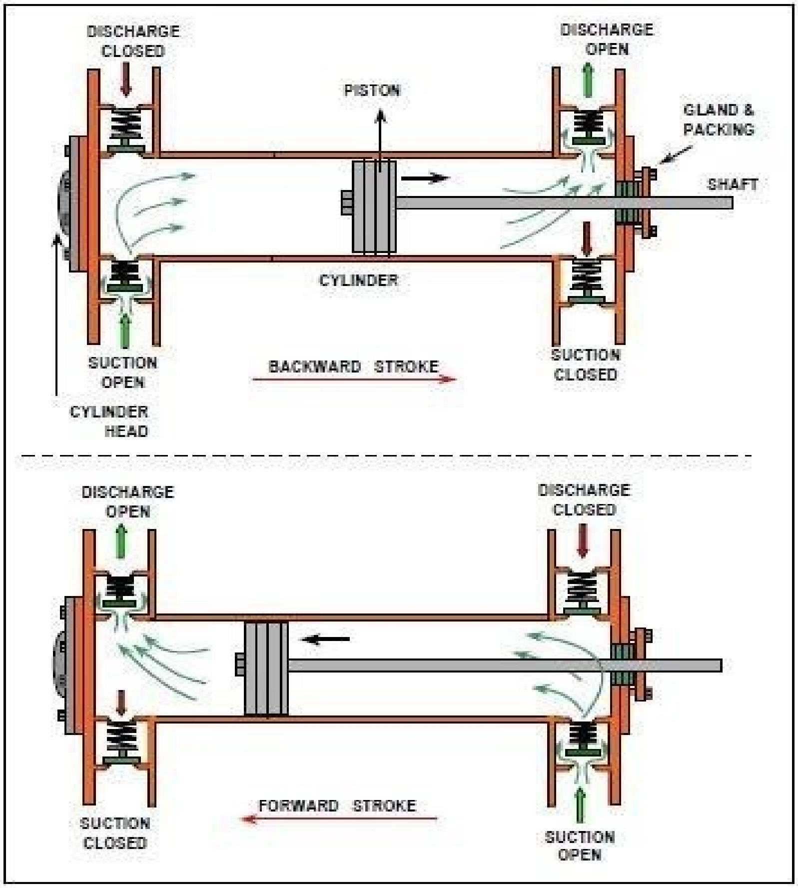
With a double acting pump, the output pulsation is much less than the single acting.

In theory, a reciprocating pump will always deliver the same volume for each stroke regardless of discharge pressure. But, as discharge pressure is increased, there is more likelihood of internal leakage between the piston rings and the cylinder liner, or leaking internal valves, causing a decrease in output. A measure of this is known as the ' Volumetric Efficiency ' of the pump.

The amount of liquid which leaks internally is known as the ' Slip ' and, if the pump is in good condition, the slip should be below 1.0%. If slip is above 5.0%, the pump needs to be overhauled. However, at operating pressures, the amount of slip is relatively constant as long as wear is not rapid. The output therefore can still be classed as constant. This type of pump is useful for delivery of fixed quantities of liquid as used in metering or dosing operations.

The speed of a reciprocating pump is generally measured as ' Strokes per Minute '. This is the number of times the piston moves back and forth in one minute. Speed can also be measured as ' R.P.M.' of the drive motor.

As the cylinder(s) are of constant dimensions, the volume of liquid moved for each stroke, (discounting leakage described above), is the same and therefore the output per minute, hour or day ..etc can be calculated.



Metering Pumps

Metering pumps provide precision control of very low flow rates. Flow rates are generally less than 1/2 gallon per minute. They are usually used to control additives to the main flow stream. They are also called proportioning orcontrolled-volume pumps. Metering pumps are available in either a diaphragm orpacked plunger style, and are designed for clean service and dirty liquid can easily clog the valves and nozzle connections.

Variable Displacement Vane Pumps

One of the major advantages of the vane pump is that the design readily lends itself to become a variable displacement pump, rather than a fixed displacement pump such as a spur-gear (X-X) or a gerotor (I-X) pump. The centerline distance from the rotor to the eccentric ring is used to determine the pump's displacement. By allowing the eccentric ring to pivot or translate relative to the rotor, the displacement can be varied. It is even possible for a vane pump to pump in reverse if the eccentric ring moves far enough. However, performance cannot be optimized to pump in both directions. This can make for a very interesting hydraulic control oil pump.

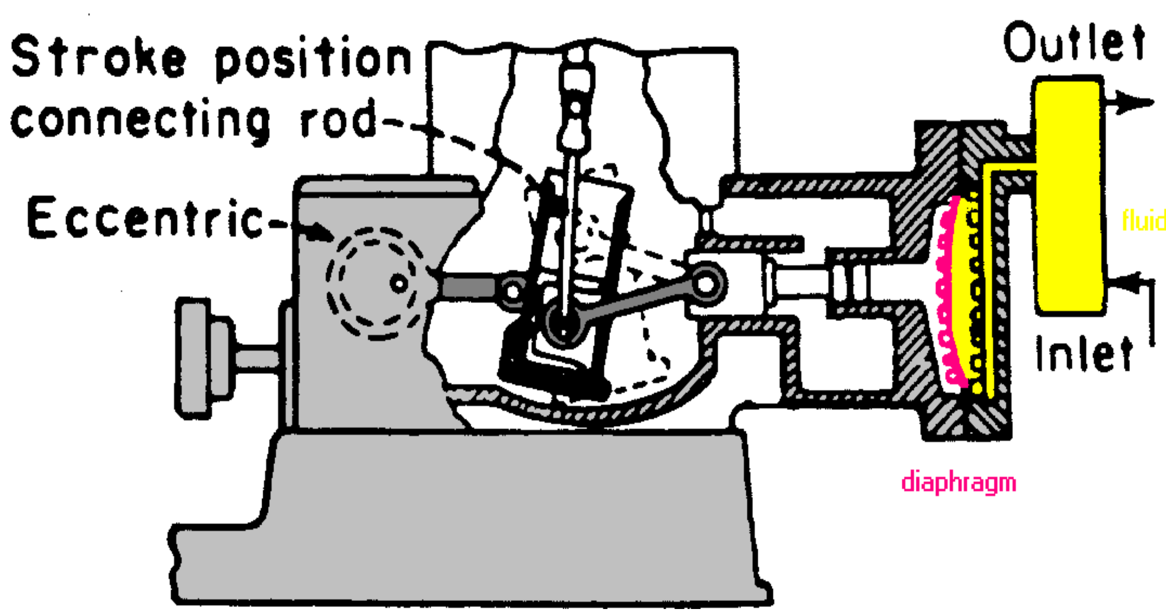
Variable displacement vane pumps are used as an energy savings device, and have been used in many applications, including automotive transmissions, for over 30 years.

Mono Pumps

Browse through the Moyno technical bulletins to see how the rotor turns inside the casing. This is calleda "progressing cavity". This pump handles solids beautifully. It is said that they can pump strawberries with little damage to each berry.

Diaphragm pump

The diaphragm pump is an offshoot of a plunger pump. Because of the risk that contamination could travel between the plunger and the cylinder, the diaphragm is safer for microbial processing. This applet is crude but shows how a reciprocating piston (plunger) pump works. The flywheel that moves the plunger can attach the arm to the plunger at various points to change the amplitude of the stroke and thusthe pumping rate. The pumping rate can also be changed with a different rotational speed, but variable speed motors or mechanical means of changing rpm are expensive.



Some Advantages of Piston Pumps

- Reciprocating pumps will deliver fluid at high pressure (High Delivery Head).
- They are 'Self-priming' - No need to fill the cylinders before starting.

Some Disadvantages of Piston Pumps

- Reciprocating pumps give a pulsating flow.
- The suction stroke is difficult when pumping viscous liquids.
- The cost of producing piston pumps is high. This is due to the very accurate sizes of the cylinders and pistons. Also, the gearing needed to convert the rotation of the drive motor into a reciprocating action involves extra equipment and cost.
- The close fitting moving parts cause maintenance problems, especially when the pump is handling fluids containing suspended solids, as the particles can get into the small clearances and cause severe wear. The piston pump therefore, should not be used for slurries.
- They give low volume rates of flow compared to other types of pump.

A single acting pump with One cylinder is called a ' Single-acting Simplex ' pump.

A double acting pump with One cylinder is called a ' Double-acting Simplex '.

MULTI-CYLINDER PUMPS

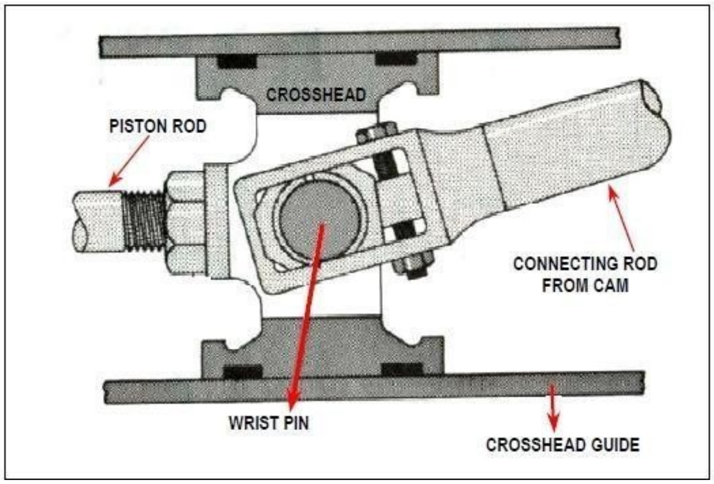
Where more than one cylinder is being driven by one driver, the arrangement is named according to the type and number of cylinders.

1. A Single-acting Duplex pump has TWO single acting cylinders.
2. A Double-acting Duplex pump has TWO double acting cylinders.
3. A Single-acting Triplex pump has THREE single acting cylinders.
4. A ' Double-acting Triplex ' pump has THREE double acting cylinders.

The more double-acting cylinders in a pump arrangement, driven by a single motor, the smoother and pulsation-free, is the output.

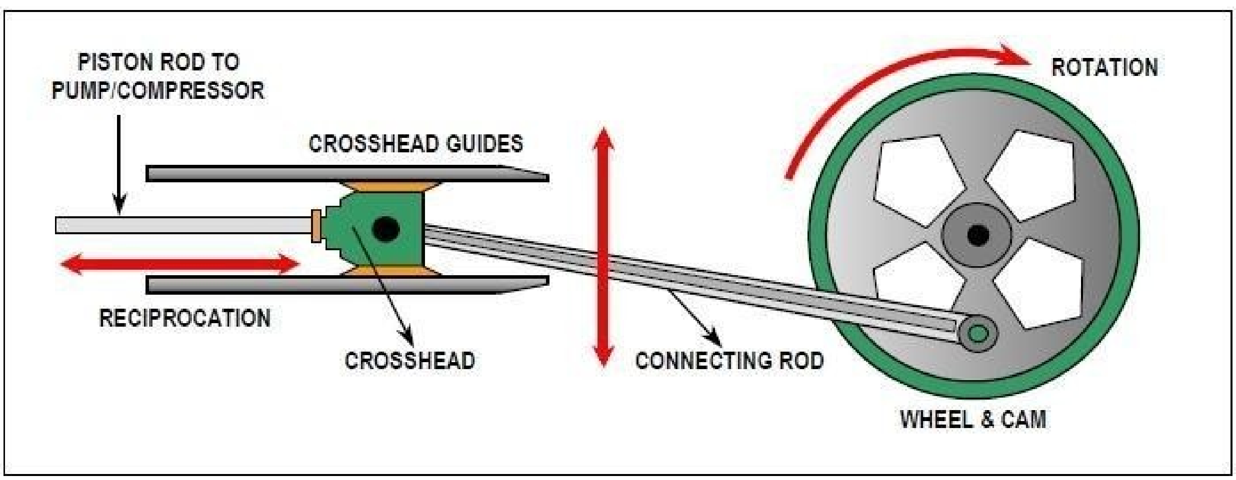
CONVERTING ROTATION INTO RECIPROCATION

The electric motor drives a fly-wheel or cam-shaft which is connected eccentrically to a connecting rod. The other end of the connecting rod is coupled to a 'Cross-head Gear' and 'Slide Assembly'. (This arrangement is the basis of the operation of the old Steam Engine drive cylinders and pistons).



As the motor rotates the fly-wheel or cam, the eccentrically mounted connecting rod rotates with it. This causes the rod to move up and down and backwards and forwards. The up and down motion cannot be transmitted to the pump shaft - it would not work. We do however, need the back and forth movement. The connecting rod goes to the cross-head gear which consists of a pivot inserted into the slide assembly. The pivot removes the up and down movement of the rod but allows the pump shaft to move back and forth.

The diagrams will explain the principle much more easily than words.



Comparing 4 Types of PD Pumps

Selection of a positive displacement (PD) rotary pump is not always an easy choice. There are four common types of PD pumps available: internal gear, external gear, timed lobe, and vane. Most PD pumps can be adapted to handle a wide range of applications, but some types are better suited than others for a given set of circumstances.

The first consideration in any application is pumping conditions. Usually the need for a PD pump is already determined, such as a requirement for a given amount of flow regardless of differential pressure, viscosity too high for a centrifugal pump, need for high differential pressure, or other factors. Inlet conditions, required flow rate, differential pressure, temperature, particle size in the liquid, abrasive characteristics, and corrosiveness of the liquid must be determined before a pump selection is made. A pump needs proper suction conditions to work well. PD pumps are self-priming, and it is often assumed that suction conditions are not important. But they are. Each PD pump has a minimum inlet pressure requirement to fill individual pump cavities. If these cavities are not completely filled, total pump flow is diminished. Pump manufacturers supply information on minimum inlet conditions required. If high lift or high vacuum inlet conditions exist, special attention must be paid to the suctionside of the pump.

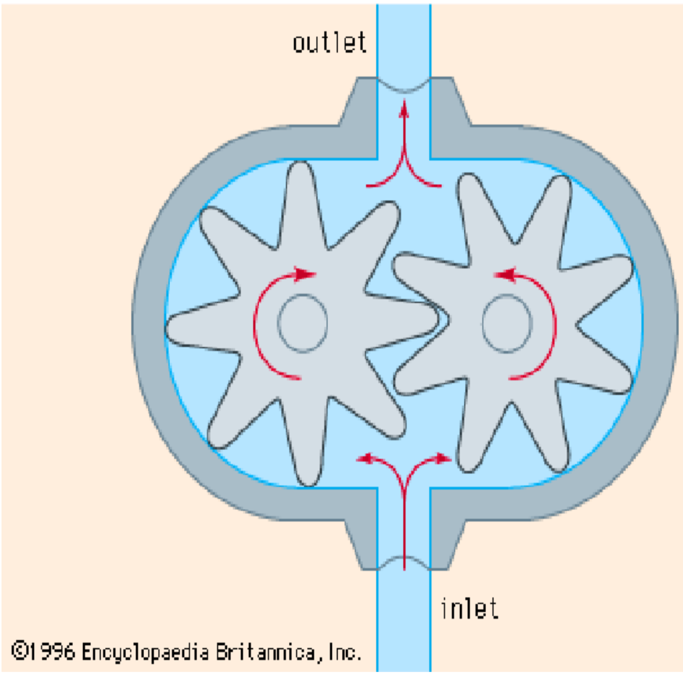
Rotary Pumps

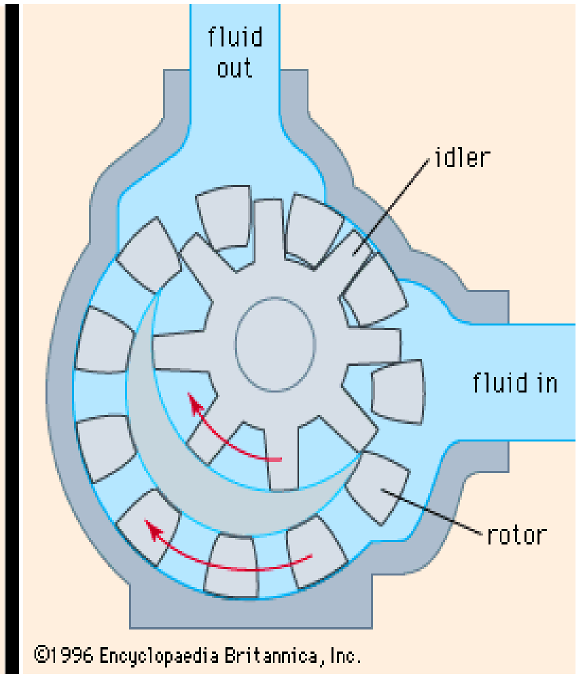
By definition, positive-displacement (PD) pumps displace a known quantity of liquid with each revolution of the pumping elements. This is done by trapping liquid between the pumping elements anda stationary casing. Pumping element designs include gears, lobes, rotary pistons, vanes, and screws.

PD pumps are found in a wide range of applications -- chemical-processing; liquid delivery; marine; biotechnology; pharmaceutical; as well as food, dairy, and beverage processing. Their versatility and popularity is due in part to their relatively compact design, high-viscosity performance, continuous flow regardless of differential pressure, and ability to handle high differential pressure.

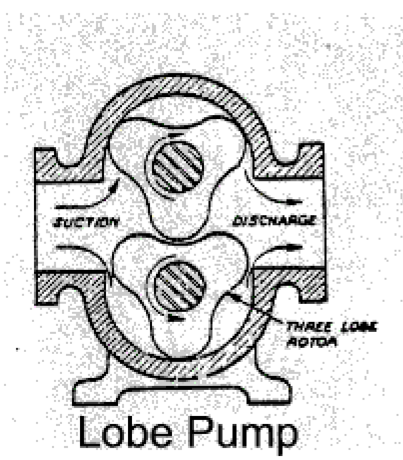
Pump School® is made up of information from a variety of sources including manufacturers, governmental agencies, industry trade organizations, and common PD industry knowledge.

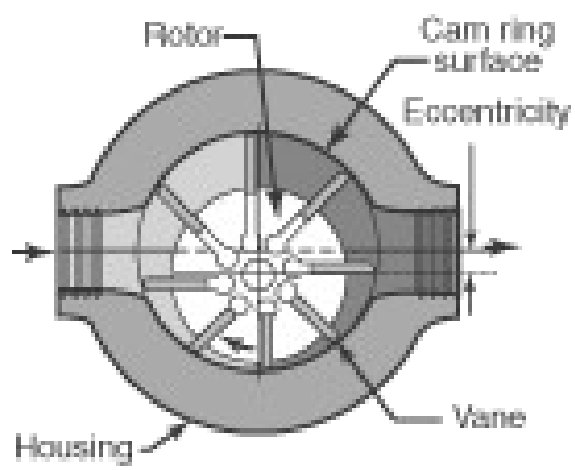
A rotary pump traps fluid in its closed casing and discharges a smooth flow. They can handle almost any liquid that does not contain hard and abrasive solids, including viscous liquids. They are also simple in design and efficient in handling flow conditions that are usually considered to low for economicapplication of centrifuges. Types of rotary pumps include cam-and-piston, internal-gear, lobular, screw, and vane pumps. Gear pumps are found in home heating systems in which the burners are fired by oil. Rotary pumps find wide use for viscous liquids. When pumping highly viscous fluids, rotary pumps must be operated at reduced speeds because at higher speeds the liquid cannot flow into the casing fast enough to fill it. Unlike a centrifugal pump, the rotary design will deliver a capacity that is not greatly affected by pressure variations on either the suction or discharge ends. In services where large changesin pressure are anticipated, the rotary design should be considered.

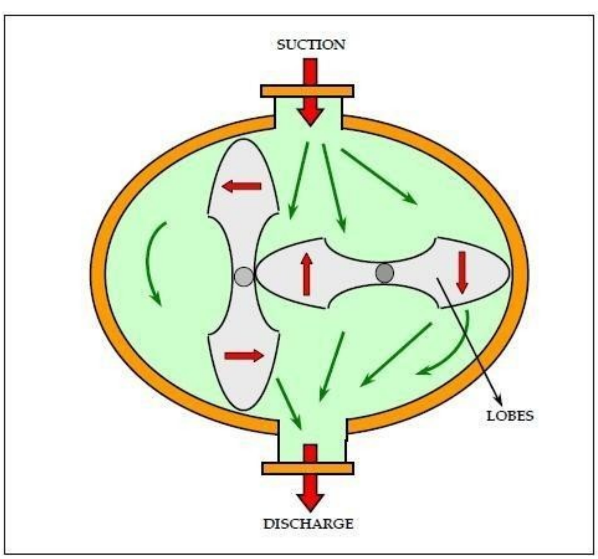


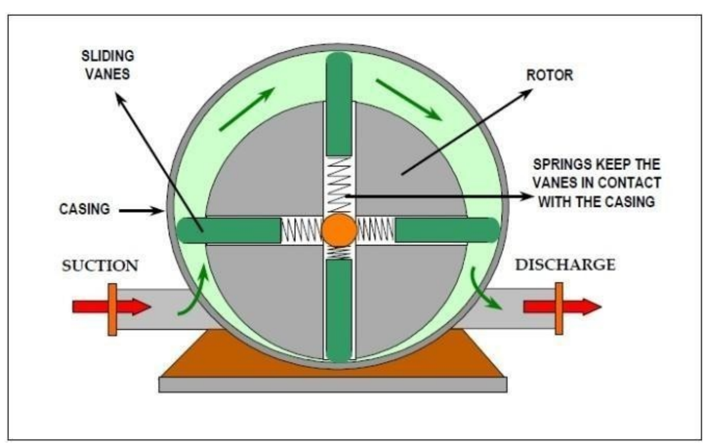


External gear Pump **Internal gear Pump**









Advantages of Rotary Pumps

- They can deliver liquid to high pressures. • Self - priming.
- Give a relatively smooth output, (especially at high speed). • Positive Acting.
- Can pump viscous liquids.

Disadvantages of Rotary Pumps

- More expensive than centrifugal pumps.
- Should not be used for fluids containing suspended solids. • Excessive wear if not pumping viscous material.
- Must NEVER be used with the discharge closed.

A rotary vane pump is a positive-displacement pump that consists of vanes mounted to a rotor that rotates inside of a cavity. In some cases these vanes can be variable length and/or tensioned to maintain contact with the walls as the pump rotates

Types

The simplest vane pump is a circular rotor rotating inside of a larger circular cavity. The centers of these two circles are offset, causing eccentricity. Vanes are allowed to slide into and out of the rotor and seal on all edges, creating vane chambers that do the pumping work. On the intake side of the pump, the vane chambers are increasing in volume. These increasing volume vane chambers are filled with fluid forced in by the inlet pressure. Often this inlet pressure is nothing more than pressure from the atmosphere. On the discharge side of the pump, the vane chambers are decreasing in volume, forcing fluid out of the pump. The action of the vane drives out the same volume of fluid with each rotation. Multistage rotary vane vacuum pumps can attain pressures as low as 10^{-3} mbar (0.1 Pa).

Uses

Common uses of vane pumps include high pressure hydraulic pumps and automotive uses including, supercharging, power steering and automatic transmission pumps. Pumps for mid-range pressures include applications such as carbonators for fountain soft drink dispensers and espresso coffee machines.They are also often used as vacuum pumps for providing braking assistance (through a braking booster) in large trucks, and in most light aircraft to drive gyroscopic flight instruments, the attitude indicator andheading indicator. Furthermore, vane pumps can be used in low-pressure gas applications such as secondary air injection for auto exhaust emission control, and in vacuum applications including evacuating refrigerant lines in air conditioners, and laboratory freeze dryers, extensively insemiconductor low pressure chemical vapor deposition systems, and vacuum experiments in physics. In this application the pumped gas and the oil are mixed within the pump, but must be separated externally.Therefore the inlet and the outlet have a large chamber – maybe with swirl – where the oil drops fall out of the gas. The inlet has a venetian blind cooled by the room air (the pump is usually 40 K hotter) to condense cracked pumping oil and water, and let it drop back into the inlet. It eventually exits through the outlet.

Internal Gear Pump Overview

Internal gear pumps are exceptionally versatile. While they are often used on thinliquids such as solvents and fuel oil, they excel at efficiently pumping thick liquidssuch as asphalt, chocolate, and adhesives. The useful viscosity range of an internal gear pump is from 1cPs to over 1,000,000cP.

In addition to their wide viscosity range, the pump has a wide temperature range as well, handling liquids up to 750°F / 400°C. This is due to the single point of end clearance (the distance between the ends of the rotor gear teeth and the head of the pump). This clearance is adjustable to accommodate high temperature, maximize efficiency for handling high viscosity liquids, and to accommodate for wear. The internal gear pump is non-pulsing, self-priming, and can run dry for short periods. They're also bi-rotational, meaning that the same pump can be used to load and unload vessels. Because internal gear pumps have onlytwo moving parts, they are reliable, simple to operate, and easy to maintain.

How Internal Gear Pumps Work

1. Liquid enters the suction port between the rotorlarge exterior gear) and idler (small interior gear) teeth. The arrows indicate the direction of the pump and liquid.

2. Liquid travels through the pump between theeeth of the "gear-within-a-gear" principle.
The crescent shape divides the liquid and acts as a seal between the suction and discharge ports.

3. The pump head is now nearly flooded, just prior to forcing the liquid out of the discharge port. Intermeshing gears of the idler and rotor form locked pockets for the liquid which assures volume control.

4. Rotor and idler teeth mesh completely to form a seal equidistant from the discharge and suction ports. This seal forces the liquid out of the discharge port.

Advantages

- Only two moving parts
- Usually requires moderate speeds
- Only one stuffing box
- Non-pulsating discharge
- Excellent for high-viscosity liquids
- Constant and even discharge regardless of pressure conditions
- Operates well in either direction
- Can be made to operate with one direction of flow with either rotation
- Low NPSH required
- Single adjustable end clearance
- Easy to maintain
- Flexible design offers application customization

Disadvantages

- Medium pressure limitations
- One bearing runs in the product pumped
- Overhung load on shaft bearing

Applications

Common internal gear pump applications include, but are not limited to:

- All varieties of fuel oil and lube oil
- Resins and Polymers
- Alcohols and solvents
- Asphalt, Bitumen, and Tar
- Polyurethane foam (Isocyanate and polyol)
- Food products such as corn syrup, chocolate, and peanut butter
- Paint, inks, and pigments
- Soaps and surfactants
- Glycol

Materials of Construction / Configuration Options

- **Externals (head, casing, bracket)** - Cast iron, ductile iron, steel, stainless steel, Alloy 20, and higher alloys.

- **Internals (rotor, idler)** - Cast iron, ductile iron, steel, stainless steel, Alloy 20, and higher alloys.
- **Bushing** - Carbon graphite, bronze, silicon carbide, tungsten carbide, ceramic, colomony, and other specials materials as needed.
- **Shaft Seal** - Lip seals, component mechanical seals, industry-standard cartridge mechanical seals, gas barrier seals, magnetically-driven pumps.
- **Packing** - Impregnated packing, if seal not required.

External Gear Pump Overview

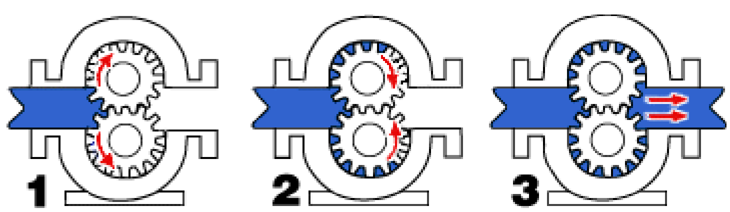
External gear pumps are a popular pumping principle and are often used as lubrication pumps in machine tools, in fluid power transfer units, and as oil pumps in engines.

External gear pumps can come in single or double (two sets of gears) pump configurations with spur (shown), helical, and herringbone gears. Helical and herringbone gears typically offer a smoother flow than spur gears, although all gear types are relatively smooth. Large-capacity external gear pumps typically use helical or herringbone gears. Small external gear pumps usually operate at 1750 or 3450 rpm and larger models operate at speeds up to 640 rpm. External gear pumps have close tolerances and shaft support on both sides of the gears. This allows them to run to pressures beyond 3,000 PSI / 200 BAR, making them well suited for use in hydraulics. With four bearings in the liquid and tight tolerances, they are not well suited to handling abrasive or extreme high temperature applications.

Tighter internal clearances provide for a more reliable measure of liquid passing through a pump and for greater flow control. Because of this, external gear pumps are popular for precise transfer and metering applications involving polymers, fuels, and chemical additives.

How External Gear Pumps Work

External gear pumps are similar in pumping action to internal gear pumps in that two gears come into and out of mesh to produce flow. However, the external gear pump uses two



< Click Here to Animate >

identical gears rotating against each other -- one gear is driven by a motor and it in turn drives the other gear. Each gear is supported by a shaft with bearings on both sides of the gear.

1. As the gears come out of mesh, they create expanding volume on the inlet side of the pump. Liquid flows into the cavity and is trapped by the gear teeth as they rotate.

2. Liquid travels around the interior of the casing in the pockets between the teeth and the casing -- it does not pass between the gears.

3. Finally, the meshing of the gears forces liquid through the outlet port under pressure.

Because the gears are supported on both sides, external gear pumps are quiet-running and are routinely used for high-pressure applications such as hydraulic applications. With no overhung bearing loads, the rotor shaft can't deflect and cause premature wear.

Advantages

- **High speed**
- **High pressure**
- **No overhung bearing loads**
- **Relatively quiet operation**
- **Design accommodates wide variety of materials**

Disadvantages

- **Four bushings in liquid area**
- **No solids allowed**
- **Fixed End Clearances**

Applications

Common external gear pump applications include, but are not limited to:

- **Various fuel oils and lube oils**
- **Chemical additive and polymer metering**

- **Chemical mixing and blending (double pump)**
- **Industrial and mobile hydraulic applications (log splitters, lifts, etc.)**
- **Acids and caustic (stainless steel or composite construction)**
- **Low volume transfer or application**

Lobe Pump Overview

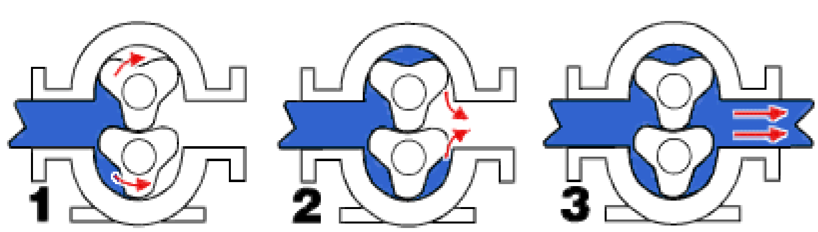
Lobe pumps are used in a variety of industries including, pulp and paper, chemical, food, beverage, pharmaceutical, and biotechnology. They are popular in these diverse industries because they offer superb sanitary qualities, high efficiency, reliability, corrosion resistance, and good clean-in-place and sterilize-in-place (CIP/SIP) characteristics.

These pumps offer a variety of lobe options including single, bi-wing, tri-lobe (shown), and multi-lobe. Rotary lobe pumps are non-contacting and have large pumping chambers, allowing them to handle solids such as cherries or olives without damage. They are also used to handle slurries, pastes, and a wide variety of other liquids. If wetted, they offer self-priming performance. A gentle pumping action minimizes product degradation. They also offer reversible flows and can operate dry for long periods of time. Flow is relatively independent of changes in process pressure, so output is constant and continuous.

Rotary lobe pumps range from industrial designs to sanitary designs. The sanitary designs break down further depending on the service and specific sanitary requirements. These requirements include 3-A, EHEDG, and USDA. The manufacturer can tell you which certifications, if any, their rotary lobe pump meets.

How Lobe Pumps Work

Lobe pumps are similar to external gear pumps in operation in that fluid flows around the interior of the casing. Unlike external gear pumps, however, the lobes do not make contact. Lobe contact is



< Click Here to Animate >

prevented by external timing gears located in the gearbox. Pump shaft support bearings are located in the gearbox, and since the bearings are out of the pumped liquid, pressure is limited by bearing location and shaft deflection.

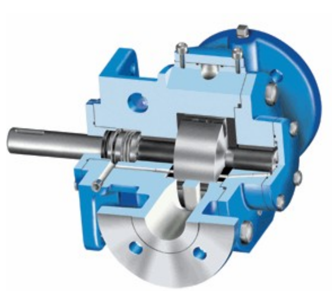
1. As the lobes come out of mesh, they create expanding volume on the inlet side of the pump. Liquid flows into the cavity and is trapped by the lobes as they rotate.
2. Liquid travels around the interior of the casing in the pockets between the lobes and the casing -- it does not pass between the lobes.
3. Finally, the meshing of the lobes forces liquid through the outlet port under pressure.

Lobe pumps are frequently used in food applications because they handle solids without damaging the product. Particle size pumped can be much larger in lobe pumps than in other PD types. Since the lobes do not make contact, and clearances are not as close as in other PD pumps, this design handles low viscosity liquids with diminished performance. Loading characteristics are not as good as other designs, and suction ability is low. High-viscosity liquids require reduced speeds to achieve satisfactory performance. Reductions of 25% of rated speed and lower are common with high-viscosity liquids.

Advantages

- Pass medium solids
- No metal-to-metal contact
- Superior CIP/SIP capabilities
- Long term dry run (with lubrication to seals)
- Non-pulsating discharge

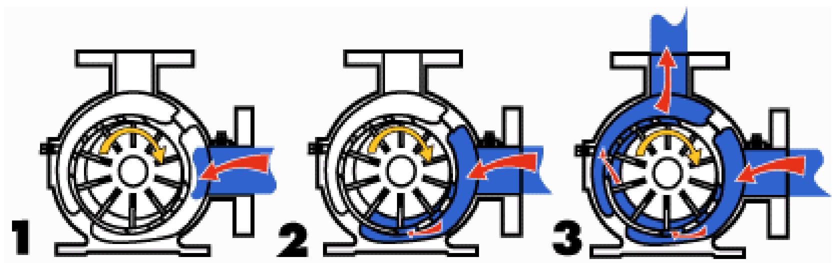
Vane Pump Overview



While vane pumps can handle oderate viscosity liquids, they excel at handling low viscosity liquids such as LP gas (propane), ammonia, solvents, alcohol, fuel oils, gasoline, and refrigerants. Vane pumps have no internal metal-to-metal contact and self-compensate for wear,enabling them to maintain peak performance on these non-lubricatingliquids. Though efficiency drops quickly, they can be used up to 500 cPs / 2,300 SSU.

flexible vane, swinging vane, rolling vane, and external vane. Vane pumps are noted for their dry priming, ease of maintenance, and good suction characteristics over the life of the pump. Moreover, vanes can usually handle fluid temperatures from -32°C / -25°F to 260°C / 500°F and differential pressures to 15 BAR / 200 PSI (higher for hydraulic vane pumps).

Each type of vane pump offers unique advantages. For example, external vane pumps can handle large solids. Flexible vane pumps, on the other hand, can only handle small solids but create good vacuum. Sliding vane pumps can run dry for short periods of time and handle small amounts of vapor.



< Click Here to Animate >

How Vane Pumps Work

Despite the different configurations, most vane pumps operate under the same general principle described below.

1. A slotted rotor is eccentrically supported in a cycloidal cam. The rotor is located close to the wall of the cam so a crescent-shaped cavity is formed. The rotor is sealed into the cam by two sideplates. Vanes or blades fit within the slots of the impeller. As the rotor rotates (yellow arrow) and fluid enters the pump, centrifugal force, hydraulic pressure, and/or pushrods push the vanes to the walls of the housing. The tight seal among the vanes, rotor, cam, and sideplate is the key to the good suction characteristics common to the vane pumping principle.

2. The housing and cam force fluid into the pumping chamber through holes in the cam (small red arrow on the bottom of the pump). Fluid enters the pockets created by the vanes, rotor, cam, and sideplate.

where it is squeezed through discharge holes of the cam as the vane approaches the point of the crescent (small red arrow on the side of the pump). Fluid then exits the discharge port.

Advantages

- Handles thin liquids at relatively higher pressures
- Compensates for wear through vane extension
- Sometimes preferred for solvents, LPG
- Can run dry for short periods
- Can have one seal or stuffing box
- Develops good vacuum

Disadvantages

- Can have two stuffing boxes
- Complex housing and many parts
- Not suitable for high pressures
- Not suitable for high viscosity
- Not good with abrasives

Applications

- Aerosol and Propellants
- Aviation Service - Fuel Transfer, Deicing
- Auto Industry - Fuels, Lubes, Refrigeration Coolants
- Bulk Transfer of LPG and NH_3
- LPG Cylinder Filling
- Alcohols
- Refrigeration - Freons, Ammonia
- Solvents
- Aqueous solutions

UNIT V

Turbines

They all use turbines—machines that capture energy from a moving liquid or gas. In a sandcastle windmill, the curved blades are designed to catch the wind's energy so they flutter and spin. In an ocean liner or a jet, hot burning gas is used to spin metal blades at high speed—capturing energy that's used to power the ship's propeller or push the plane through the sky. Turbines also help us make the vast majority of our electricity: turbines driven by steam are used in virtually every major power plant, while wind and water turbines help us to produce renewable energy. Wherever energy's being harnessed for human needs, turbines are usually somewhere nearby.

A **turbine** is a rotary engine that extracts energy from a fluid flow and converts it into useful work. The simplest turbines have one moving part, a rotor assembly, which is a shaft or drum with blades attached. Moving fluid acts on the blades, or the blades react to the flow, so that they move and impart rotational energy to the rotor. Early turbine examples are windmills and water wheels.

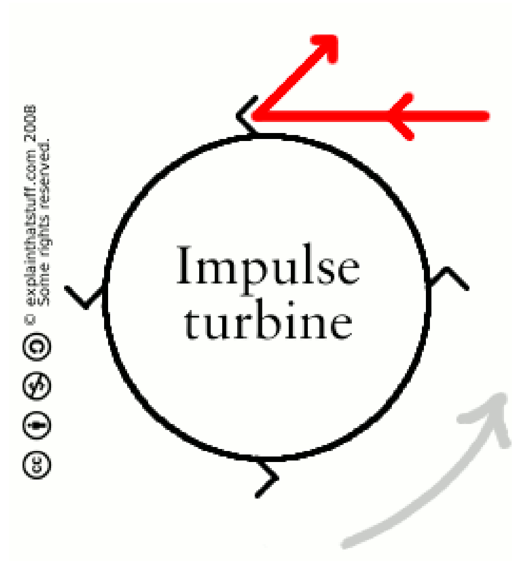
Gas, steam, and water turbines usually have a casing around the blades that contains and controls the working fluid. Credit for invention of the steam turbine is given both to the British Engineer Sir Charles Parsons (1854–1931), for invention of the reaction turbine and to Swedish Engineer Gustaf de Laval (1845–1913), for invention of the impulse turbine. Modern steam turbines frequently employ both reaction and impulse in the same unit, typically varying the degree of reaction and impulse from the blade root to its periphery.

Impulse and reaction turbines

Turbines work in two different ways described as impulse and reaction—terms that are often very confusingly described (and sometimes completely muddled up) when people try to explain them. So what's the difference?

Impulse turbines

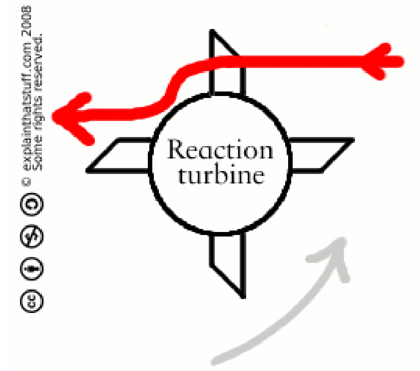
In an impulse turbine, a fast-moving fluid is fired through a narrow nozzle at the turbine blades to make them spin around. The blades of an impulse turbine are usually bucket-shaped so they catch the fluid and direct it off at an angle or sometimes even back the way it came (because that gives the most efficient transfer of energy from the fluid to the turbine). In an impulse turbine, the fluid is forced to hit the turbine at high speed. Imagine trying to make a wheel like this turn around by kicking soccer balls into its paddles. You'd need the balls to hit hard and bounce back well to get the wheel spinning—and those constant energy impulses are the key to how it works.



Reaction turbines

In a reaction turbine, the blades sit in a much larger volume of fluid and turn around as the fluid flows past them. A reaction turbine doesn't change the direction of the fluid flow as drastically as an impulse turbine: it simply spins as the fluid pushes through and past its blades.

If an impulse turbine is a bit like kicking soccer balls, a reaction turbine is more like swimming—in reverse. Let me explain! Think of how you do freestyle (front crawl) by hauling your arms through the water, starting with each hand as far in front as you can reach and ending with a "follow through" that throws your arm well behind you. What you're trying to achieve is to keep your hand and forearm pushing against the water for as long as possible, so you transfer as much energy as you can in each stroke. A reaction turbine is using the same idea in reverse: imagine fast-flowing water moving past you so it makes your arms and legs move and supplies energy to your body! With a reaction turbine, you want the water to touch the blades smoothly, for as long as it can, so it gives up as much energy as possible. The water isn't hitting the blades and bouncing off, as it does in an impulse turbine: instead, theblades are moving more smoothly, "going with the flow".



Turbines in action

Broadly speaking, we divide turbines into four kinds according to the type of fluid that drives them: water, wind, steam, and gas. Although all four types work in essentially the same way—spinning aroundas the fluid moves against them—they are subtly different and have to be engineered in very different ways. Steam turbines, for example, turn incredibly quickly because steam is produced under high- pressure. Wind turbines that make electricity turn relatively slowly (mainly for safety reasons), so they need to be huge to capture decent amounts of energy. Gas turbines need to be made from specially resilient alloys because they work at such high temperatures. Water turbines are often very big because they have to extract energy from an entire river, dammed and diverted to flow past them.

Water turbines

Water wheels, which date back over 2000 years to the time of the ancient Greeks, were the original water turbines. Today, the same principle is used to make electricity in hydroelectric power plants. The basic idea of hydroelectric power is that you dam a river to harness its energy. Instead of the river flowing freely downhill from its hill or mountain source toward the sea, you make it fall through aheight (called a head) so it picks up speed (in other words, so its potential energy is converted to kinetic energy), then channel it through a pipe called a penstock past a turbine and generator. Hydroelectricity is effectively a three-step energy conversion:

- The river's original potential energy (which it has because it starts from high ground) is turned into kinetic energy when the water falls through a height.
- The kinetic energy in the moving water is converted into mechanical energy by a water turbine.
- The spinning water turbine drives a generator that turns the mechanical energy into electrical energy.

Different kinds of water turbine are used depending on the geography of the area, how much water is available (the flow), and the distance over which it can be made to fall (the head). Some hydroelectric plants use bucket-like impulse turbines (typically Pelton wheels); others use Francis, Kaplan, or Deriaz reaction turbines. The type of turbine is chosen carefully to extract the maximum amount of energy from the water.

Steam turbines

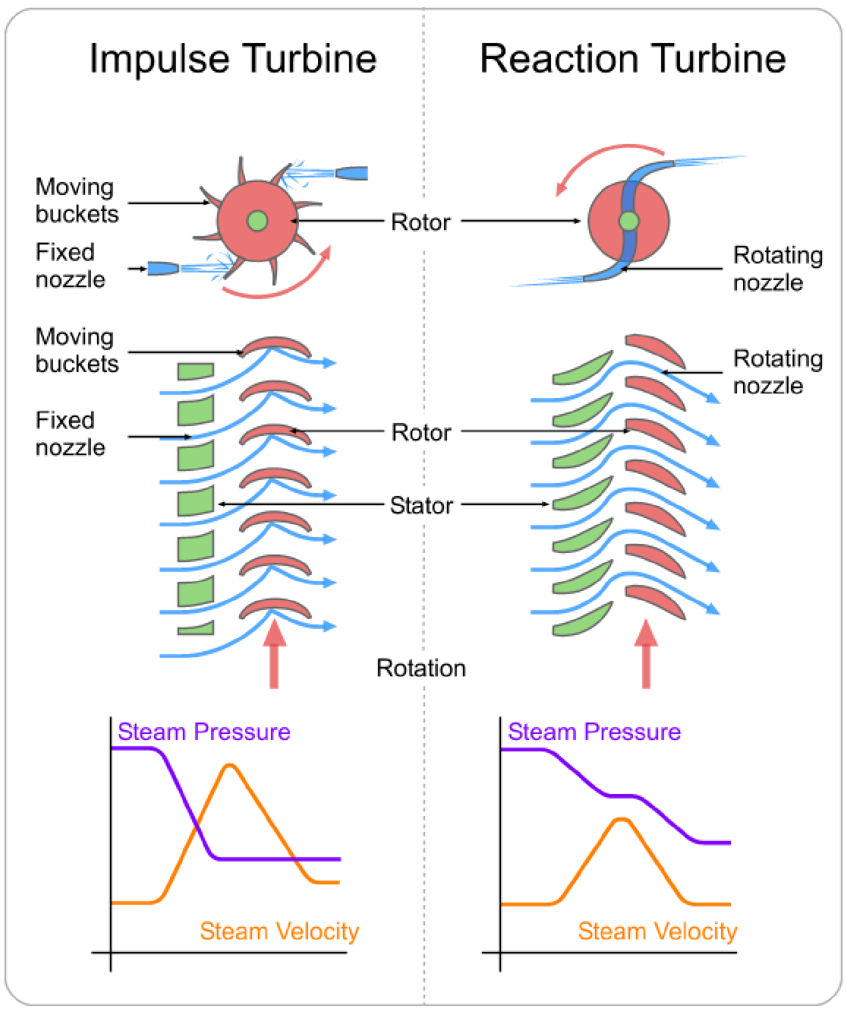
Steam turbines evolved from the steam engines that changed the world in the 18th and 19th centuries. A steam engine burns coal on an open fire to release the heat it contains. The heat is used to boil water and make steam, which pushes a piston in a cylinder to power a machine such as a railroad locomotive. This is quite inefficient (it wastes energy) for a whole variety of reasons. A much better design takes the steam and channels it past the blades of a turbine, which spins around like a propeller and drives the machine as it goes.

Steam turbines were pioneered by British engineer Charles Parsons (1854–1931), who used them to power a famously speedy motorboat called Turbinia in 1889. Since then, they've been used in many different ways. Virtually all power plants generate electricity using steam turbines. In a coal-fired plant, coal is burned in a furnace and used to heat water to make steam that spins high-speed turbines connected to electricity generators. In a nuclear power plant, the heat that makes the steam comes from atomic reactions.

Unlike water and wind turbines, which place a single rotating turbine in the flow of liquid or gas, steam turbines have a whole series of turbines (each of which is known as a stage) arranged in a sequence inside what is effectively a closed pipe. As the steam enters the pipe, it's channelled past each stage in turn so progressively more of its energy is extracted. If you've ever watched a kettle boiling, you'll know that steam expands and moves very quickly if it's directed through a nozzle. For that reason, steam turbines turn at very high speeds—many times faster than wind or water turbines.

Gas turbines

Airplane jet engines are a bit like steam turbines in that they have multiple stages. Instead of steam, they're driven by a mixture of the air sucked in at the front of the engine and the incredibly hot gases made by burning huge quantities of kerosene (petroleum-based fuel). Somewhat less powerful gas turbine engines are also used in modern railroad locomotives and industrial machines. See our article on jet engines for more details.



A device similar to a turbine but operating in reverse, i.e., driven, is a compressor or pump. The axial compressor in many gas turbine engines is a common example. Here again, both reaction and impulse are employed and again, in modern axial compressors, the degree of reaction and impulse will typically vary from the blade root to its periphery.

Claude Burdin coined the term from the Latin *turbo*, or vortex, during an 1828 engineering competition. Benoit Fourneyron, a student of Claude Burdin, built the first practical water turbine.

A working fluid contains potential energy (pressure head) and kinetic energy (velocity head). The fluid may be compressible or incompressible. Several physical principles are employed by turbines to collect this energy:

Impulse turbines

These turbines change the direction of flow of a high velocity fluid or gas jet. The resulting impulse spins the turbine and leaves the fluid flow with diminished kinetic energy. There is no pressure changeof the fluid or gas in the turbine rotor blades (the moving blades), as in the case of a steam or gas turbine, all the pressure drop takes place in the stationary blades (the nozzles).

Before reaching the turbine, the fluid's *pressure head* is changed to *velocity head* by accelerating the fluid with a nozzle. Pelton wheels and de Laval turbines use this process exclusively. Impulse turbines do not require a pressure casement around the rotor since the fluid jet is created by the nozzle prior to reaching the blading on the rotor. Newton's second law describes the transfer of energy for impulse turbines.

Reaction turbines

These turbines develop torque by reacting to the gas or fluid's pressure or mass. The pressure of the gas or fluid changes as it passes through the turbine rotor blades. A pressure casement is needed to contain the working fluid as it acts on the turbine stage(s) or the turbine must be fully immersed in the fluid flow(such as with wind turbines). The casing contains and directs the working fluid and, for water turbines, maintains the suction imparted by the draft tube. Francis turbines and most steam turbines use this concept. For compressible working fluids, multiple turbine stages are usually used to harness the expanding gas efficiently. Newton's third law describes the transfer of energy for reaction turbines.

In the case of steam turbines, such as would be used for marine applications or for land-based electricity generation, a Parsons type reaction turbine would require approximately double the number of blade rows as a de Laval type impulse turbine, for the same degree of thermal energy conversion. Whilst this makes the Parsons turbine much longer and heavier, the overall efficiency of a reaction turbine isslightly higher than the equivalent impulse turbine for the same thermal energy conversion.

Steam turbines and later, gas turbines developed continually during the 20th Century, continue to do so and in practice, modern turbine designs will use both reaction and impulse concepts to varying degrees whenever possible. Wind turbines use an airfoil to generate lift from the moving fluid and impart it to the rotor (this is a form of reaction). Wind turbines also gain some energy from the impulse of the wind, by deflecting it at an angle. Crossflow turbines are designed as an impulse machine, with a nozzle, but inlow head applications maintain some efficiency through reaction, like a traditional water wheel. Turbines with multiple stages may utilize either reaction or impulse blading at high pressure. Steam Turbines were traditionally more impulse but continue to move towards reaction designs similar to thoseused in Gas Turbines. At low pressure the operating fluid medium expands in volume for small reductions in pressure. Under these conditions (termed Low Pressure Turbines) blading becomes strictly a reaction type design with the base of the blade solely impulse. The reason is due to the effect of the rotation speed for each blade. As the volume increases, the blade height increases, and the base of the blade spins at a slower speed relative to the tip. This change in speed forces a designer to change from impulse at the base, to a high reaction style tip.

Classical turbine design methods were developed in the mid 19th century. Vector analysis related the fluid flow with turbine shape and rotation. Graphical calculation methods were used at first. Formulaefor the basic dimensions of turbine parts are well documented and a highly efficient machine can be reliably designed for any fluid flow condition. Some of the calculations are empirical or 'rule of thumb' formulae, and others are based on classical mechanics. As with most engineering calculations, simplifying assumptions were made.

Velocity triangles can be used to calculate the basic performance of a turbine stage. Gas exits the stationary turbine nozzle guide vanes at absolute velocity V_{a1}. The rotor rotates at velocity U. Relative tothe rotor, the velocity of the gas as it impinges on the rotor entrance is V_{r1}. The gas is turned by the rotor and exits, relative to the rotor, at velocity V_{r2}. However, in absolute terms the rotor exit velocity is V_{a2}. The velocity triangles are constructed using these various velocity vectors. Velocity triangles can be constructed at any section through the blading (for example: hub , tip, midsection and so on) but are usually shown at the mean stage radius. Mean performance for the stage can be calculated from the velocity triangles, at this radius, using the Euler equation:

Types of turbines

Steam turbines are used for the generation of electricity in thermal power plants, such as plants using coal, fuel oil or nuclear power. They were once used to directly drive mechanical devices such as ships' propellors (eg the Turbinia), but most such applications now use reduction gears or an intermediate electrical step, where the turbine is used to generate electricity, which then powers an electric motor connected to the mechanical load. Turbo electric ship machinery was particularly popular in the period immediately before and during WWII, primarily due to a lack of sufficient gear-cutting facilities in US and UK shipyards.

Gas turbines are sometimes referred to as turbine engines. Such engines usually feature an inlet, fan, compressor, combustor and nozzle (possibly other assemblies) in addition to one or more turbines.

Transonic turbine. The gasflow in most turbines employed in gas turbine engines remains subsonic throughout the expansion process. In a transonic turbine the gasflow becomes supersonic as it exits the nozzle guide vanes, although the downstream velocities normally become subsonic. Transonic turbines operate at a higher pressure ratio than normal but are usually less efficient and uncommon.

Contra-rotating turbines. With axial turbines, some efficiency advantage can be obtained if a downstream turbine rotates in the opposite direction to an upstream unit. However, the complication can be counter-productive. A contra-rotating steam turbine, usually known as the Ljungström turbine, was originally invented by Swedish Engineer Fredrik Ljungström (1875–1964), in Stockholm and in partnership with his brother Birger Ljungström he obtained a patent in 1894. The design is essentially a multi-stage radial turbine (or pair of 'nested' turbine rotors) offering great efficiency, four times as large heat drop per stage as in the reaction (Parsons) turbine, extremely compact design and the type met particular success in backpressure power plants. However, contrary to other designs, large steam volumes are handled with difficulty and only a combination with axial flow turbines (DUREX) admits the turbine to be built for power greater than ca 50 MW. In marine applications only about 50 turbo-electric units were ordered (of which a considerable amount were finally sold to land plants) during 1917-19, and during 1920-22 a few turbo-mechanic not very successful units were sold[1]. Only a few turbo-electric marine plants were still in use in the late 1960s (ss Ragne, ss Regin) while most land plants remain in use 2010.

Statorless turbine. Multi-stage turbines have a set of static (meaning stationary) inlet guide vanes that direct the gasflow onto the rotating rotor blades. In a statorless turbine the gasflow exiting an upstream rotor impinges onto a downstream rotor without an intermediate set of stator vanes (that rearrange the pressure/velocity energy levels of the flow) being encountered.

Ceramic turbine. Conventional high-pressure turbine blades (and vanes) are made from nickelbased alloys and often utilise intricate internal air-cooling passages to prevent the metal fromoverheating. In recent years, experimental ceramic blades have been manufactured and tested in gas turbines, with a view to increasing Rotor Inlet Temperatures and/or, possibly, eliminating aircooling.Ceramic blades are more brittle than their metallic counterparts, and carry a greater risk of catastrophic blade failure. This has tended to limit their use in jet engines and gas turbines, to the stator (stationary) blades.

low-pressure turbine, with lacing wires. These are wires which pass through holes drilled in the blades at suitable distances from the blade root and the wires are usually brazed to the blades at the point where they pass through. The lacing wires are designed to reduce blade flutter in the central part of the blades. The introduction of lacing wires substantially reduces the instances of blade failure in large or low-pressure turbines.

Shroudless turbine. Modern practice is, wherever possible, to eliminate the rotor shrouding, thus reducing the centrifugal load on the blade and the cooling requirements.

Bladeless turbine uses the boundary layer effect and not a fluid impinging upon the blades as in a conventional turbine.

Water turbines

Pelton turbine, a type of impulse water turbine.

Francis turbine, a type of widely used water turbine.

Kaplan turbine, a variation of the Francis Turbine.

Wind turbine. These normally operate as a single stage without nozzle and interstage guide vanes. An exception is the Éolienne Bollée, which has a stator and a rotor, thus being a true turbine.

Kaplan turbine

The **Kaplan turbine** is a propeller-type water turbine which has adjustable blades. It was developed in 1913 by the Austrian professor Viktor Kaplan, who combined automatically-adjusted propeller blades with automatically-adjusted wicket gates to achieve efficiency over a wide range of flow and water level. The Kaplan turbine was an evolution of the Francis turbine. Its invention allowed efficient power production in low-head applications that was not possible with Francis turbines.

Kaplan turbines are now widely used throughout the world in high-flow, low-head power production.

The Kaplan turbine is an inward flow reaction turbine, which means that the working fluid changes pressure as it moves through the turbine and gives up its energy. The design combines radial and axial features.

The inlet is a scroll-shaped tube that wraps around the turbine's wicket gate. Water is directed tangentially through the wicket gate and spirals on to a propeller shaped runner, causing it to spin.

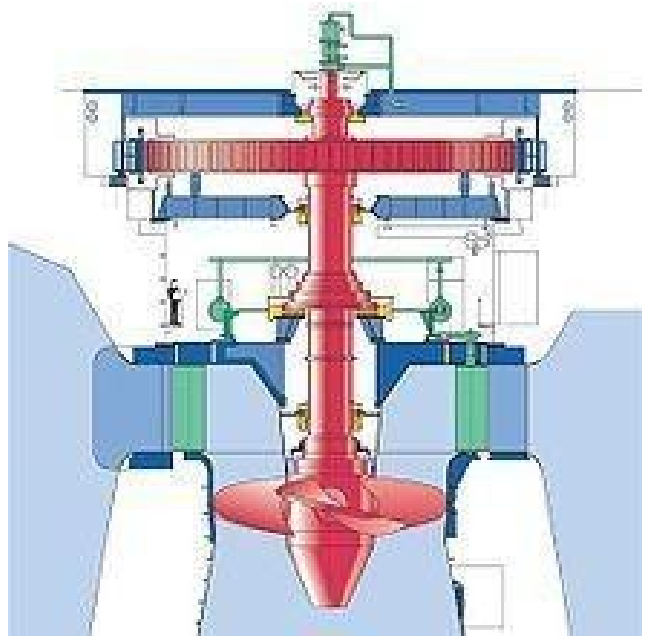
The outlet is a specially shaped draft tube that helps decelerate the water and recover kinetic energy.

The turbine does not need to be at the lowest point of water flow as long as the draft tube remains full of water. A higher turbine location, however, increases the suction that is imparted on the turbine blades by the draft tube. The resulting pressure drop may lead to cavitation.

Variable geometry of the wicket gate and turbine blades allow efficient operation for a range of flow conditions. Kaplan turbine efficiencies are typically over 90%, but may be lower in very low head applications.

Current areas of research include CFD driven efficiency improvements and new designs that raise survival rates of fish passing through.

Because the propeller blades are rotated by high-pressure hydraulic oil, a critical element of Kaplan design is to maintain a positive seal to prevent emission of oil into the waterway. Discharge of oil into rivers is not permitted.



Applications

Kaplan turbines are widely used throughout the world for electrical power production. They cover the lowest head hydro sites and are especially suited for high flow conditions.

Inexpensive micro turbines are manufactured for individual power production with as little as two feet of head.Kaplan turbine is low head turbine.

Large Kaplan turbines are individually designed for each site to operate at the highest possible efficiency, typically over 90%. They are very expensive to design, manufacture and install, but operate for decades.

Variations

The Kaplan turbine is the most widely used of the propeller-type turbines, but several other variations exist:

Propeller turbines have non-adjustable propeller vanes. They are used in where the range of head is not large. Commercial products exist for producing several hundred watts from only a few feet of head. Larger propeller turbines produce more than 100 MW.

Bulb or Tubular turbines are designed into the water delivery tube. A large bulb is centered in the water pipe which holds the generator, wicket gate and runner. Tubular turbines are a fully axial design, whereas Kaplan turbines have a radial wicket gate.

Pit turbines are bulb turbines with a gear box. This allows for a smaller generator and bulb.

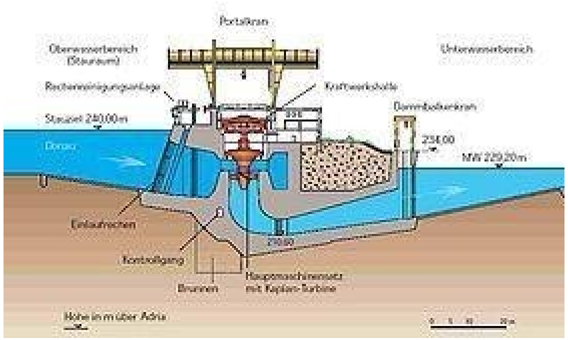
Straflo turbines are axial turbines with the generator outside of the water channel, connected to the periphery of the runner.

S- turbines eliminate the need for a bulb housing by placing the generator outside of the water channel.

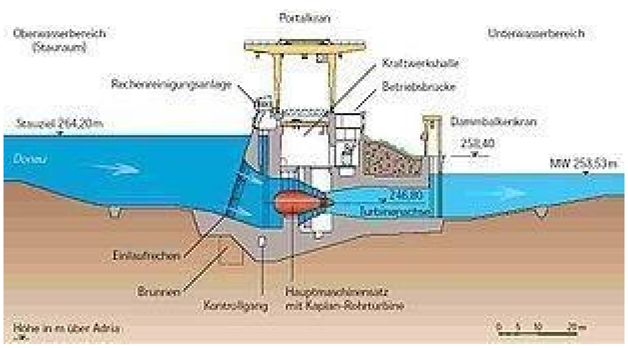
This is accomplished with a jog in the water channel and a shaft connecting the runner and generator.

VLH turbine an open flow, very low head "kaplan" turbine slanted at an angle to the water flow. It has a large diameter, is low speed using a permanent magnet alternator with electronic power regulation and is very fish friendly (<5% mortality). VLH Turbine

Tyson turbines are a fixed propeller turbine designed to be immersed in a fast flowing river, either permanently anchored in the river bed, or attached to a boat or barge.



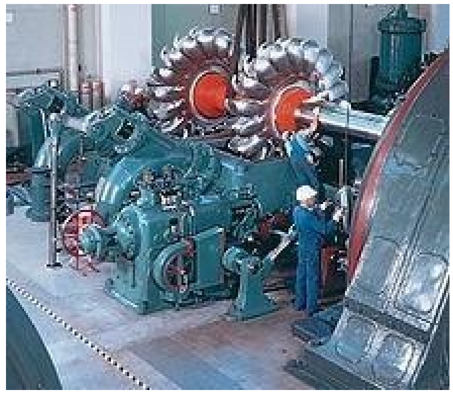
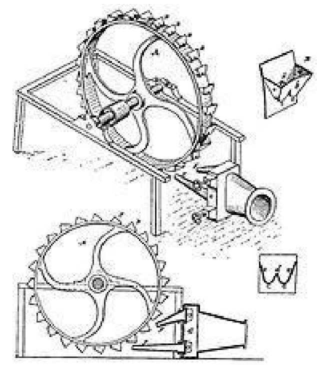
Vertical Kaplan Turbine (courtesy VERBUND-Austrian Hydro Power).



Horizontal Bulb turbine. (courtesy VERBUND-Austrian Hydro Power).

Pelton wheel

The **Pelton wheel** is among the most efficient types of water turbines. It was invented by Lester Allan Pelton in the 1870s. The Pelton wheel extracts energy from the impulse (momentum) of moving water, as opposed to its weight like traditional overshot water wheel. Although many variations of impulse turbines existed prior to Pelton's design, they were less efficient than Pelton's design; the water leaving these wheels typically still had high speed, and carried away much of the energy. Pelton' paddle geometry was designed so that when the rim runs at ½ the speed of the water jet, the water leaves the wheel with very little speed, extracting almost all of its energy, and allowing for a very efficient turbine.

The water flows along the tangent to the path of the runner. Nozzles direct forceful streams of water against a series of spoon-shaped buckets mounted around the edge of a wheel. As water flows into the bucket, the direction of the water velocity changes to follow the contour of the bucket. When the water-jet contacts the bucket, the water exerts pressure on the bucket and the water is decelerated as it does a "u-turn" and flows out the other side of the bucket at low velocity. In the process, the water's momentumis transferred to the turbine. This "impulse" does work on the turbine. For maximum power and efficiency, the turbine system is designed such that the water-jet velocity is twice the velocity of the bucket. A very small percentage of the water's original kinetic energy will still remain in the water; however, this allows the bucket to be emptied at the same rate it is filled, (see conservation of mass), thus allowing the water flow to continue uninterrupted. Often two buckets are mounted side-by-side, thus splitting the water jet in half (see photo). This balances the side-load forces on the wheel, and helps to ensure smooth, efficient momentum transfer of the fluid jet to the turbine wheel.

Because water and most liquids are nearly incompressible, almost all of the available energy is extracted in the first stage of the hydraulic turbine. Therefore, Pelton wheels have only one turbine stage, unlike gas turbines that operate with compressible fluid.

Applications

Pelton wheels are the preferred turbine for hydro-power, when the available water source has relatively high hydraulic head at low flow rates. Pelton wheels are made in all sizes. There exist multi-ton Pelton wheels mounted on vertical oil pad bearings in hydroelectric plants. The largest units can be up to 200 megawatts. The smallest Pelton wheels are only a few inches across, and can be used to tap power from mountain streams having flows of a few gallons per minute. Some of these systems utilize household plumbing fixtures for water delivery. These small units are recommended for use with thirty meters or more of head, in order to generate significant power levels. Depending on water flow and design, Pelton wheels operate best with heads from 15 meters to 1,800 meters, although there is no theoretical limit.

The Pelton wheel is most efficient in high head applications (see the "Design Rules" section). Thus, more power can be extracted from a water source with high-pressure and low-flow than from a source with low-pressure and high-flow, even though the two flows theoretically contain the same power. Also a comparable amount of pipe material is required for each of the two sources, one requiring a long thin pipe, and the other a short wide pipe.

Design rules

Specific speed

Main article: Specific speed

The specific speed n_s of a turbine dictates the turbine's shape in a way that is not related to its size. This allows a new turbine design to be scaled from an existing design of known performance. The specific speed is also the main criterion for matching a specific hydro-electric site with the correct turbine type.

The formula suggests that the Pelton turbine is most suitable for applications with relatively high hydraulic head, due to the 5/4 exponent being greater than unity, and given the characteristically low specific speed of the Pelton[1].

Energy and initial jet velocity

In the ideal (frictionless) case, all of the hydraulic potential energy ($E_p = mgh$) is converted into kinetic energy ($E_k = mv2/2$) (see Bernoulli's principle). Equating these two equations and solving for the initial jet velocity (V_i) indicates that the theoretical (maximum) jet velocity is $V_i = \sqrt{(2gh)}$. For simplicity, assume that all of the velocity vectors are parallel to each other. Defining the velocity of the wheel runner as: (u), then as the jet approaches the runner, the initial jet velocity relative to the runner is: ($V_i - u$).[1]

Final jet velocity

Assuming that the jet velocity is higher than the runner velocity, if the water is not to become backed-up in runner, then due to conservation of mass, the mass entering the runner must equal the mass leaving the runner. The fluid is assumed to be incompressible (an accurate assumption for most liquids). Also it is assumed that the cross-sectional area of the jet is constant. The jet *speed* remains constant relative to the runner. So as the jet recedes from the runner, the jet velocity relative to the runner is: $-(V_i - u)$ =

$-V_i + u$. In the standard reference frame (relative to the earth), the final velocity is then: $V_f = (-V_i + u) + u = -V_i + 2u$.

Optimal wheel speed

We know that the ideal runner speed will cause all of the kinetic energy in the jet to be transferred to the wheel. In this case the final jet velocity must be zero. If we let $-V_i + 2u = 0$, then the optimal runner speed will be $u = V_i /2$, or half the initial jet velocity.

Torque

By newton's second and third laws, the force F imposed by the jet on the runner is equal but opposite to the rate of momentum change of the fluid, so:

$$F = -m(V_f - V_i) = -\rho Q[(-V_i + 2u) - V_i] = -\rho Q(-2V_i + 2u) = 2\rho Q(V_i - u)$$

where (ρ) is the density and (Q) is the volume rate of flow of fluid. If (D) is the wheel diameter, the torque on the runner is: $T = F(D/2) = \rho QD(V_i - u)$. The torque is at a maximum when the runner is stopped (i.e. when $u = 0$, $T = \rho QDV_i$). When the speed of the runner is equal to the initial jet velocity,the torque is zero (i.e. when $u = V_i$, then $T = 0$). On a plot of torque versus runner speed, the torque curveis straight between these two points, ($0, pQDV_i$) and (V_i, 0).[1]

Power

The power $P = Fu = T\omega$, where ω is the angular velocity of the wheel. Substituting for F, we have $P = 2\rho Q(V_i - u)u$. To find the runner speed at maximum power, take the derivative of P with respect to u and set it equal to zero, [$dP/du = 2\rho Q(V_i - 2u)$]. Maximum power occurs when $u = V_i /2$. $P_{max} = \rho QV_i^2/2$. Substituting the initial jet power $V_i = \sqrt{(2gh)}$, this simplifies to $P_{max} = \rho ghQ$. This quantity exactly equals the kinetic power of the jet, so in this ideal case, the efficiency is 100%, since all the energy in the jet is converted to shaft output.

Efficiency

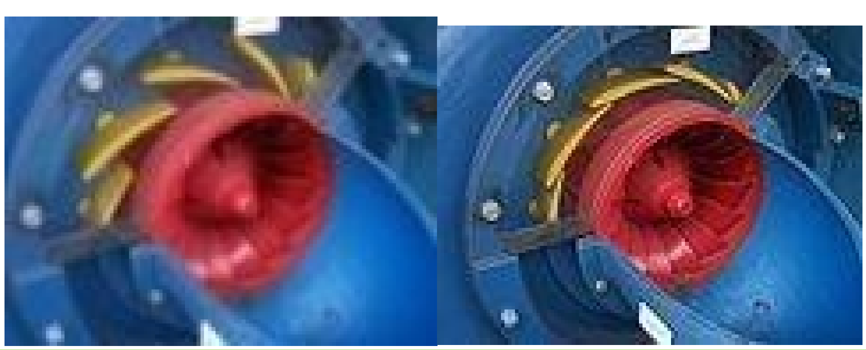
A wheel power divided by the initial jet power, is the turbine efficiency, $\eta = 4u(V_i - u)/V_i^2$. It is zero for $u = 0$ and for $u = V_i$. As the equations indicate, when a real Pelton wheel is working close to maximum efficiency, the fluid flows off the wheel with very little residual velocity.[1] Apparently, this basic theory does *not* suggest that efficiency will vary with hydraulic head, and further theory is required to show this.

Francis turbine

The **Francis turbine** is a type of water turbine that was developed by James B. Francis in Lowell, MA. It is an inward-flow reaction turbine that combines radial and axial flow concepts.

Francis turbines are the most common water turbine in use today. They operate in a head range of ten meters to several hundred meters and are primarily used for electrical power production.

The Francis turbine is a *reaction turbine*, which means that the working fluid changes pressure as it moves through the turbine, giving up its energy. A casement is needed to contain the water flow. The turbine is located between the high-pressure water source and the low-pressure water exit, usually at the base of a dam.



The inlet is spiral shaped. Guide vanes direct the water tangentially to the turbine wheel, known as a runner. This radial flow acts on the runner's vanes, causing the runner to spin. The guide vanes (or wicket gate) may be adjustable to allow efficient turbine operation for a range of water flow conditions. As the water moves through the runner, its spinning radius decreases, further acting on the runner. For an analogy, imagine swinging a ball on a string around in a circle; if the string is pulled short, the ball spins faster due to the conservation of angular momentum. This property, in addition to the water's pressure, helps Francis and other inward-flow turbines harness water energy efficiently.

Water wheels have been used historically to power mills of all types, but they are inefficient. Nineteenth-century efficiency improvements of water turbines allowed them to compete with steam engines (wherever water was available). In 1848 James B. Francis, while working as head engineer of the Locks and Canals company in the water-powered factory city of Lowell, Massachusetts, improved on these designs to create a turbine with90% efficiency. He applied scientific principles and testing methods to produce a very efficient turbine design. More importantly, his mathematical and graphical calculation methods improved turbine design and engineering. His analytical methods allowed confident design of high efficiency turbines to exactly match a site's flow conditions.

The Francis turbine is a *reaction turbine*, which means that the working fluid changes pressure as it moves through the turbine, giving up its energy. A casement is needed to contain the water flow. The turbine is located between the high-pressure water source and the low-pressure water exit, usually at thebase of a dam. The inlet is spiral shaped. Guide vanes direct the water tangentially to the turbine wheel, known as a*runner*. This radial flow acts on the runner's vanes, causing the runner to spin. The guide vanes (orwicket gate) may be adjustable to allow efficient turbine operation for a range of water flow conditions. As the water moves through the runner, its spinning radius decreases, further acting on the runner. Foran analogy, imagine swinging a ball on a string around in a circle; if the string is pulled short, the ballspins faster due to the conservation of angular momentum. This property, in addition to the water'spressure, helps Francis and other inward- flow turbines harness water energy efficiently.