FLUID MECHANICS

(UTAB 2019/2020)

1. Fluids Mechanics and Fluid Properties

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 behaviour 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.

1.1 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.

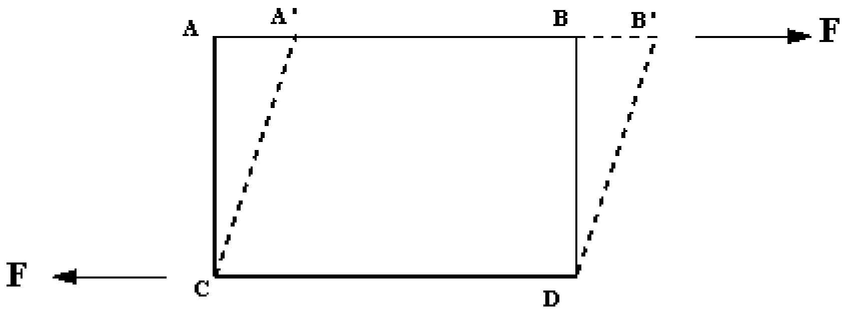
1.2 Fluids

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.



Shearing force, F, acting on a fluid element.

We can then say:

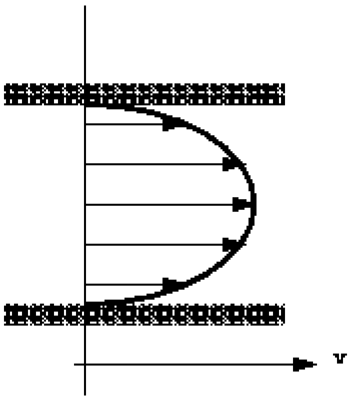
A Fluid is a substance which deforms continuously, or flows, when subjected to shearing forces.

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 the are acting.

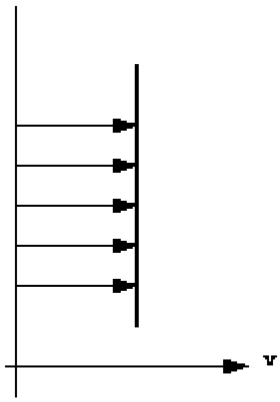
When a fluid is in motion shear stresses are developed if the particles of the fluid move relative to one another. When this happens adjacent particles have different velocities. If fluid velocity is the same at every point then there is no shear stress produced: the particles have zero *relative* velocity.

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:

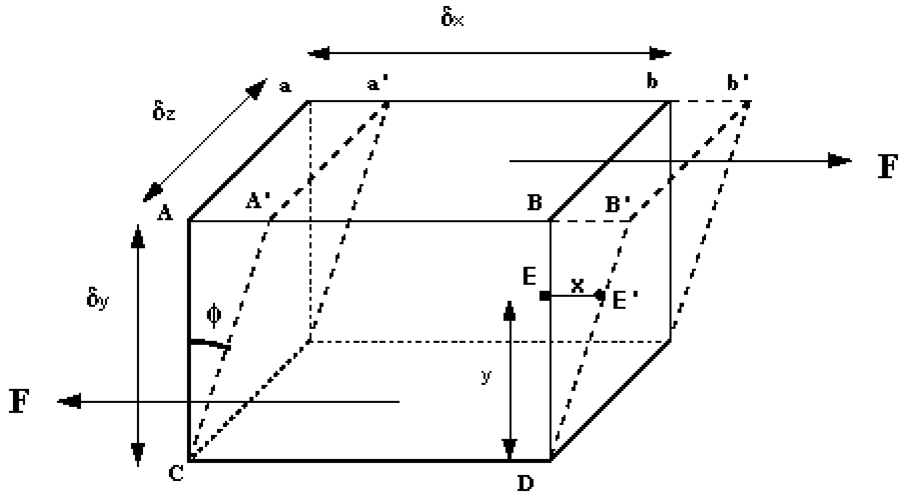


Velocity profile in uniform flow

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.

1.2.1 Newton's Law of Viscosity

How can we make use of these observations? 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 $A = \delta s \times \delta x$. We can thus calculate the *shear stress* which is equal to force per unit area i.e.

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

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

$$\text{shear strain } \phi = \frac{x}{y}$$

$$\text{rate of shear strain } = \frac{\phi}{t}$$

$$= \frac{x}{ty} = \frac{x}{t}\frac{1}{y}$$

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

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

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

This is known as **Newton's law of viscosity**.

1.2.2 Fluids vs. Solids

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

1. 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.
2. 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.

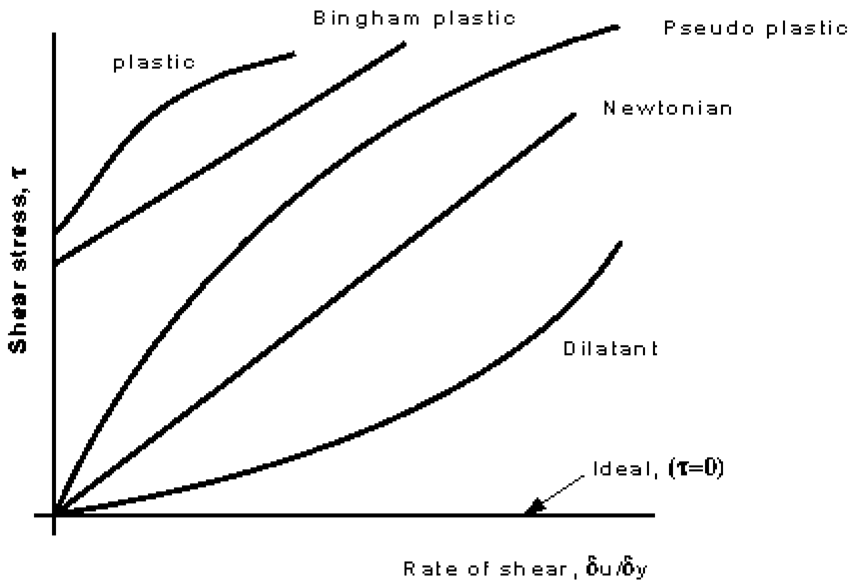
As you will have seen when looking at properties of solids, when the elastic limit is reached they seem to flow. They become plastic. They still do **not** meet the definition of true fluids as they will only flow after a certain minimum shear stress is attained.

1.2.3 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 δu/δy

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.

1.2.4 Liquids vs. 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.

1.3 Causes of Viscosity in Fluids

1.3.1 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 of a 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.

1.3.2 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 T°C, and μ_0 is the viscosity at temperature 0°C. 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.

1.4 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.)

1.4.1 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.

1.4.1.1 Mass Density

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

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

Dimensions: ML^{-3}

Typical values:

Water = 1000 $kg\,m^{-3}$, Mercury = 13546 $kg\,m^{-3}$ Air = 1.23 $kg\,m^{-3}$, Paraffin Oil = 800 $kg\,m^{-3}$.

(at pressure =1.013×10^{-5} $N\,m^{-2}$ and Temperature = 288.15 K.)

1.4.1.2 Specific Weight

Specific Weight ω, (sometimes γ, 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

$$\text{weight per unit volume} = \text{mass per unit volume} \times g$$

$$\omega = \rho g$$

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

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

Typical values:

Water =9814 $N\,m^{-3}$, Mercury = 132943 $N\,m^{-3}$, Air =12.07 $N\,m^{-3}$, Paraffin Oil =7851 $N\,m^{-3}$

1.4.1.3 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.

$$\sigma = \frac{\sigma_{substance}}{\sigma_{H_2O(at\,4°c)}}$$

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

Dimensions: 1.

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

1.4.2 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.

All fluids are viscous, "Newtonian Fluids" obey the linear relationship

given by Newton's law of viscosity. $\tau = \mu \frac{du}{dy}$, which we saw earlier.

where τ 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.

1.4.2.1 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.

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

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 $kg\,m^{-1}\,s^{-1}$.

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

2. Forces in Static Fluids

This section will study the forces acting on or generated by fluids at rest.

Objectives

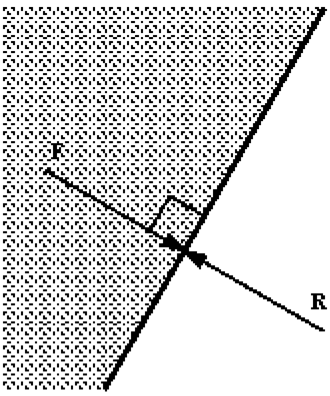
- Introduce the concept of pressure;
- Prove it has a unique value at any particular elevation;
- Show how it varies with depth according to the hydrostatic equation and
- Show how pressure can be expressed in terms of *head* of fluid.

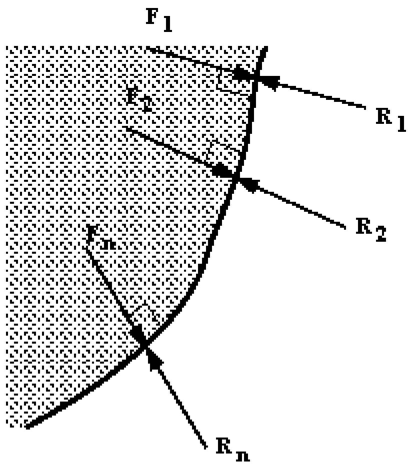
This understanding of pressure will then be used to demonstrate methods of pressure measurement that will be useful later with fluid in motion and also to analyse the forces on submerges surface/structures.

2.1 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.

2.2 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. $1Pa = 1\ N\,m^{-2}$)

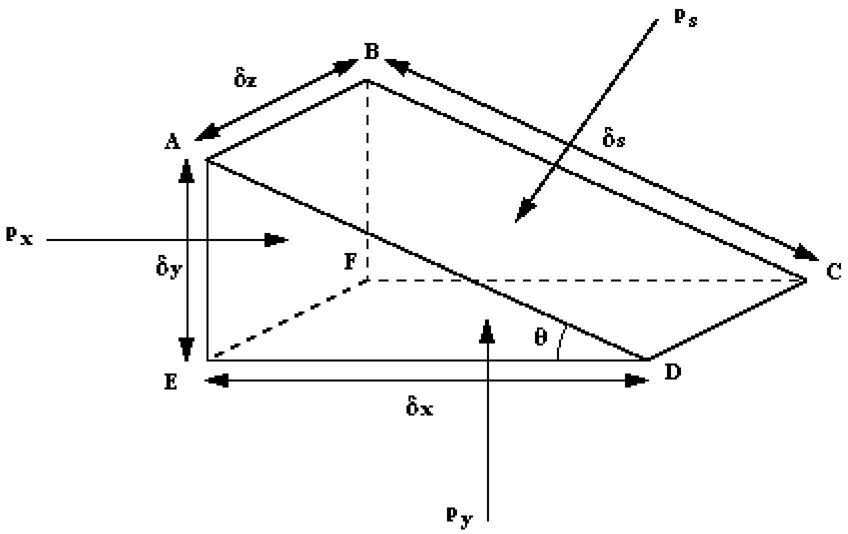
(Also frequently used is the alternative SI unit the *bar,* where $1bar = 10^5\ N\,m^{-2}$)

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

2.2.1 Pascal's Law for Pressure At A Point

(Proof that pressure acts equally in all directions.)

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,

$$\begin{aligned} F_{x_s} &= -p_s \times Area_{ABCD} \times \sin\theta \\ &= -p_s \delta s \, \delta z \frac{\delta y}{\delta s} \\ &= -p_s \delta y \, \delta z \end{aligned}$$

($\sin\theta = \delta y / \delta s$)

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,

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

Component of force due to p_s,

$$\begin{aligned} F_{y_s} &= -p_s \times Area_{ABCD} \times \cos\theta \\ &= -p_s \delta s \delta z \frac{\delta x}{\delta s} \\ &= -p_s \delta x \delta z \end{aligned}$$

($\cos\theta = \delta x / \delta s$)

Component of force due to p_x,

$$F_{y_x} = 0$$

Force due to gravity,

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

To be at rest (in equilibrium)

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

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

The element is small i.e. δx , δy and δz are small, and so $\delta x \delta y \delta z$ is very small and considered negligible, hence

$$p_y = p_s$$

thus

$$p_x = p_y = p_s$$

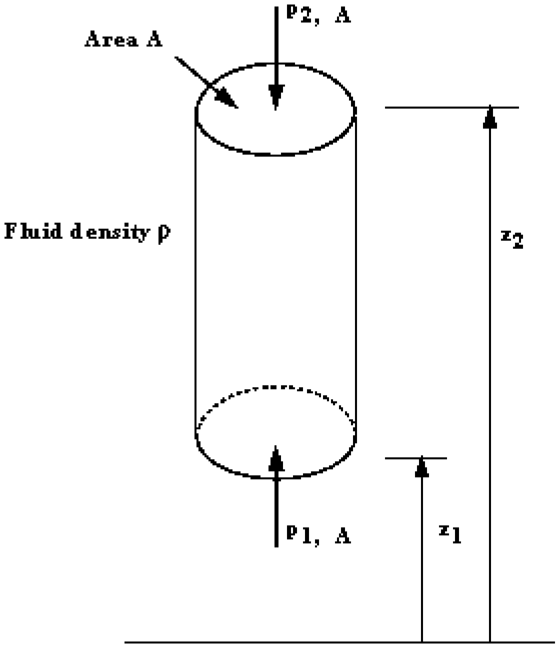
Considering the prismatic element again, p_s is the pressure on a plane at any angle θ, the x, y and z directions could be any orientation. The element is so small that it can be considered a point so the derived expression $p_x = p_y = p_s$. indicates that pressure at any point is the same in all directions.

(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.

2.2.2 *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

Force due to p_1 on A (upward) $= p_1 A$

Force due to p_2 on A (downward) $p_2 A$

Force due to weight of element (downward) mg

mass density volume $\rho g A(z_2 \quad z_1)$

Taking upward as positive, in equilibrium we have

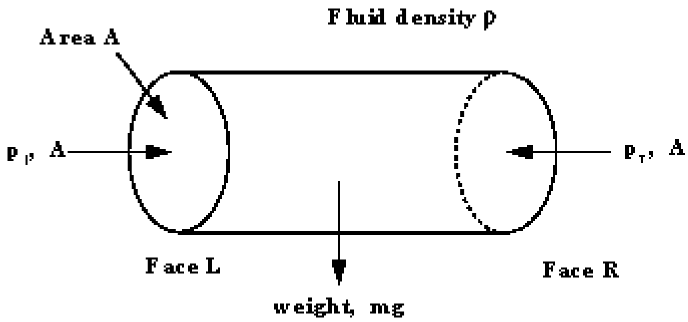
$$p_1 A - p_2 A - \rho g A(z_2 - z_1) = 0$$

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

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

2.2.3 *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_1 at the left hand end and pressure p_2 at the right hand end.



Horizontal elemental cylinder of fluid

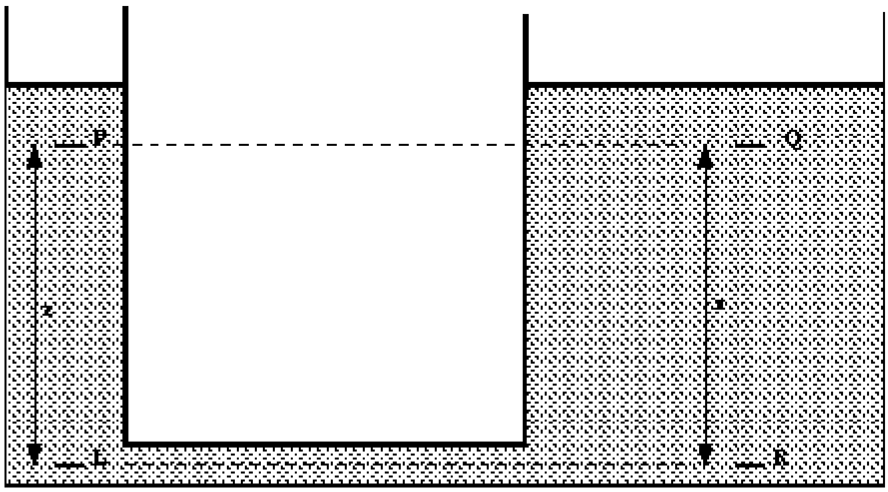
The fluid is at equilibrium so the sum of the forces acting in the x direction is zero.

$$p_l A = p_r A$$

$$p_l = p_r$$

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

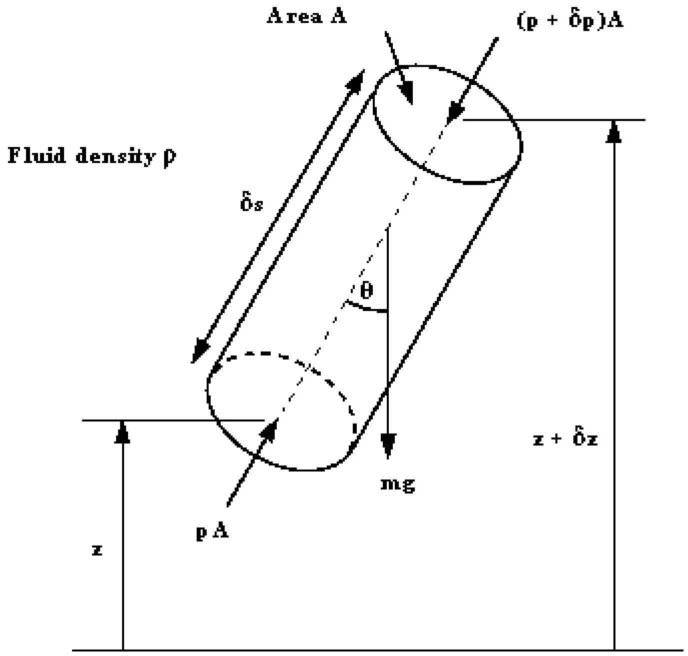
$$p_p + \rho g z = p_q + \rho g z$$

$$p_p = p_q$$

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

2.2.4 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

$$pA \quad \text{acting at right - angles to the end of the face at } z$$

$$\left(p + \delta p\right)A \quad \text{acting at right - angles to the end of the face at } z + \delta z$$

$$mg =$$

$$\rho A \delta s g$$

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

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

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

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

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

the weight of the element acting vertically down

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

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

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

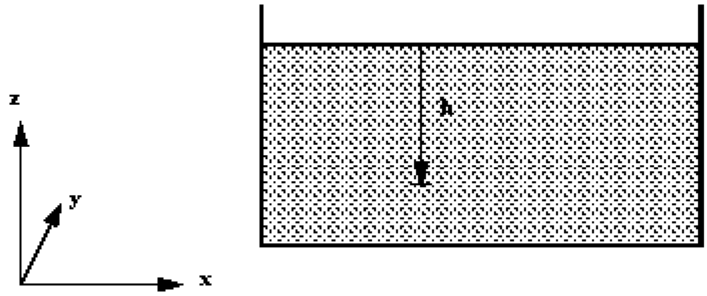
$$p_2 \quad p_1 \qquad \rho g(z_2 \quad z_1)$$

2.2.5 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, $p_{\text{atmospheric}}$. So

$$p = \rho g h + p_{\text{atmospheric}}$$

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

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

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

$$p_{\text{absolute}} = \rho gh + p_{\text{atmospheric}}$$

Absolute pressure = Gauge prssure + 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.

$$p = \rho gh$$

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 density, $\rho = 1000 kg\,m^{-3}$. Using $p = \rho gh$,

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

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

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

2.3 Pressure Measurement By Manometer

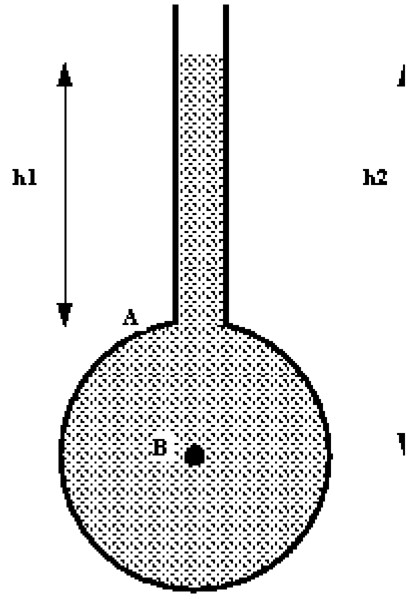
The relationship between pressure and head is used to measure pressure with a manometer (also know as a liquid gauge).

Objective:

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

2.3.1 The Piezometer Tube Manometer

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

$$p_A = \rho g h_1$$

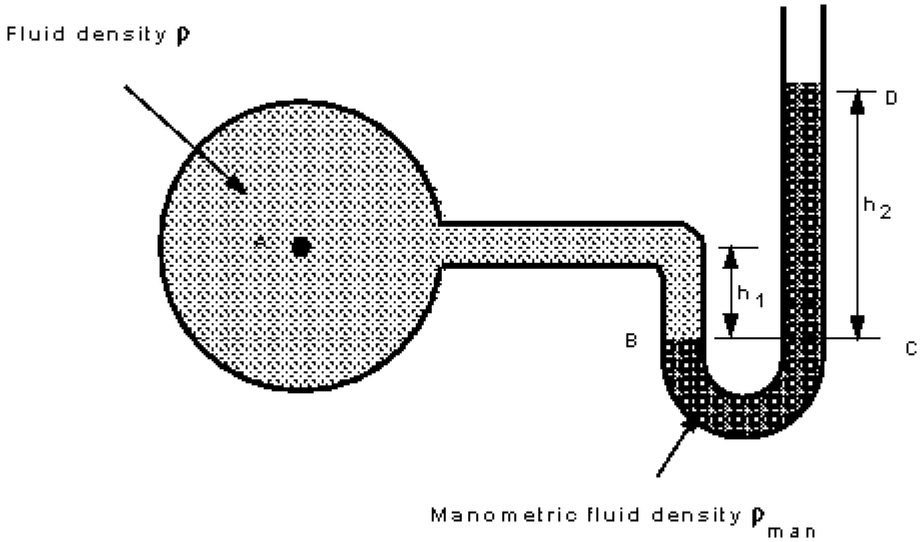
pressure at B = pressure due to column of liquid above B

$$p_B = \rho g h_2$$

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.

2.3.2 The “U”-Tube Manometer

Using a “U”-Tube enables the pressure of both liquids and gases to be measured with the same instrument. 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,

pressure at B = pressure at C

$$p_B = p_C$$

For the **left hand** arm

pressure at B = pressure at A + pressure due to height h_1 of fluid being measured

$$p_B = p_A + \rho g h_1$$

For the **right hand** arm

pressure at C = pressure at D + pressure due to height h_2 of manometric fluid

$$p_C = p_{\text{Amospheric}} + \rho_{\text{man}} g h_2$$

As we are measuring *gauge pressure* we can subtract $p_{\text{Atmospheric}}$ giving

$$p_B = p_C$$

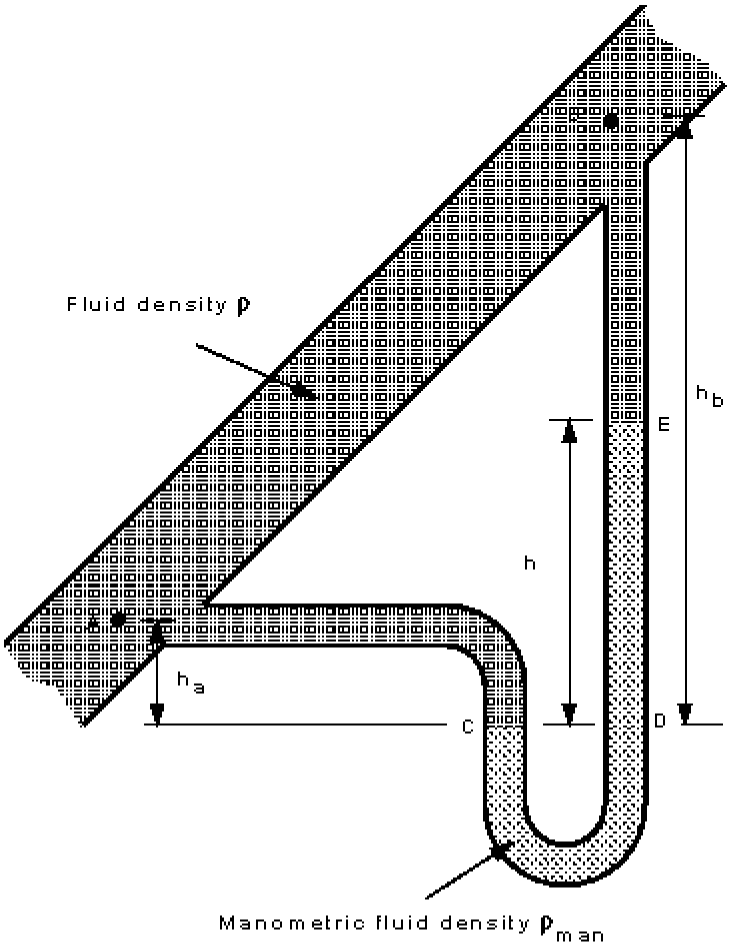
$$p_A = \rho_{\text{man}} g h_2 - \rho g h_1$$

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. $\rho_{\text{man}} >> \rho$. In this case the term $\rho g h_1$ can be neglected, and the gauge pressure give by

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

2.3.3 *Measurement Of Pressure Difference Using a "U"-Tube Manometer.*

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

pressure at C = pressure at C

$$p_C = p_D$$

$$p_C = p_A + \rho g h_a$$

$$p_D = p_B + \rho g\left(h_b - h\right) + \rho_{\text{man}} g h$$

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

Giving the pressure difference

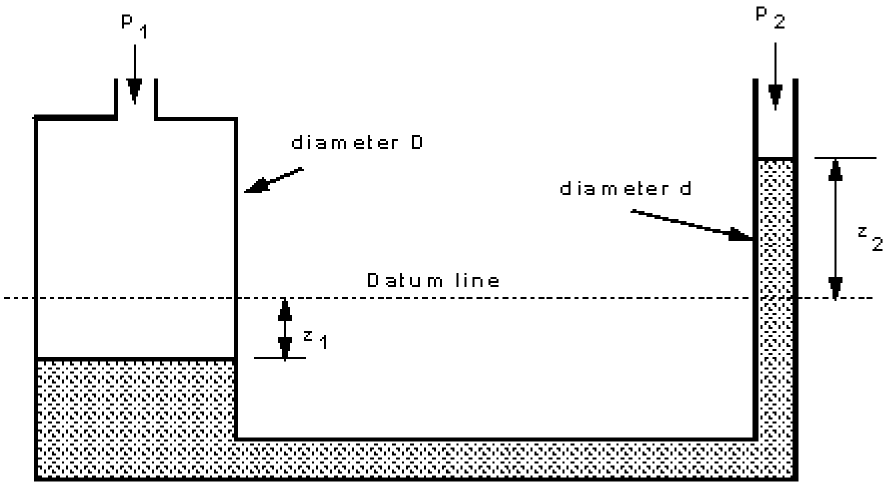
$$p_A - p_B = \rho g\left(h_b - h_a\right) + \left(\rho_{\text{man}} - \rho\right) g h$$

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

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

2.3.4 Advances to the "U" tube manometer.

The "U"-tube manometer has the disadvantage that the change in height of the liquid in both sides must be read. This can be avoided by making the diameter of one side very large compared to the other. In this case the side with the large area moves very little when the small area side move considerably more.



Assume the manometer is arranged as above to measure the pressure difference of a gas of (negligible density) and that pressure difference is $p_1 - p_2$. If the datum line indicates the level of the manometric fluid when the pressure difference is zero and the height differences when pressure is applied is as shown, the volume of liquid transferred from the left side to the right $= z_2 \times (\pi d^2 / 4)$

And the fall in level of the left side is

$$
\begin{aligned}
z_1 &= \frac{\text{Volume moved}}{\text{Area of left side}} \\
&= \frac{z_2(\pi d^2/4)}{\pi D^2/4} \\
&= z_2\left(\frac{d}{D}\right)^2
\end{aligned}
$$

We know from the theory of the "U" tube manometer that the height different in the two columns gives the pressure difference so

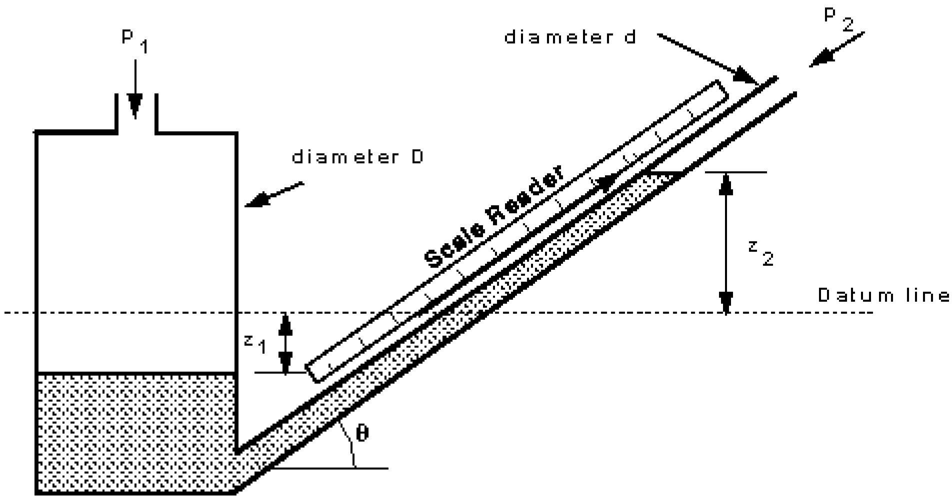
$$
\begin{aligned}
p_1 - p_2 &= \rho g\left[z_2 + z_2\left(\frac{d}{D}\right)^2\right] \\
&= \rho g z_2\left[1 + \left(\frac{d}{D}\right)^2\right]
\end{aligned}
$$

Clearly if D is very much larger than *d* then $(d/D)^2$ is very small so

$$p_1 - p_2 = \rho g z_2$$

So only one reading need be taken to measure the pressure difference.

If the pressure to be measured is very small then tilting the arm provides a convenient way of obtaining a larger (more easily read) movement of the manometer. The above arrangement with a tilted arm is shown in the figure below.



Tilted manometer.

The pressure difference is still given by the height change of the manometric fluid but by placing the scale along the line of the tilted arm and taking this reading large movements will be observed. The pressure difference is then given by

$$p_1 - p_2 = \rho g z_2$$
$$\rho g x \sin\theta$$

The sensitivity to pressure change can be increased further by a greater inclination of the manometer arm, alternatively the density of the manometric fluid may be changed.

2.3.5 Choice Of Manometer

Care must be taken when attaching the manometer to vessel, no burrs must be present around this joint. Burrs would alter the flow causing local pressure variations to affect the measurement.

Some disadvantages of manometers:

- Slow response - only really useful for very slowly varying pressures - no use at all for fluctuating pressures;
- For the "U" tube manometer two measurements must be taken simultaneously to get the h value. This may be avoided by using a tube with a much larger cross-sectional area on one side of the manometer than the other;
- It is often difficult to measure small variations in pressure - a different manometric fluid may be required - alternatively a sloping manometer may be employed; It cannot be used for very large pressures unless several manometers are connected in series;
- For very accurate work the temperature and relationship between temperature and ρ must be known;

Some advantages of manometers:

- They are very simple.
- No calibration is required - the pressure can be calculated from first principles

2.4 Forces on Submerged Surfaces in Static Fluids

We have seen the following features of statics fluids

- Hydrostatic vertical pressure distribution
- Pressures at any equal depths in a continuous fluid are equal
- Pressure at a point acts equally in all directions (Pascal's law).
- Forces from a fluid on a boundary acts at right angles to that boundary.

Objectives:

We will use these to analyse and obtain expressions for the forces on submerged surfaces. In doing this it should also be clear the difference between:

- Pressure which is a scalar quantity whose value is equal in all directions and,
- Force, which is a vector quantity having both magnitude and direction.

2.4.1 Fluid pressure on a surface

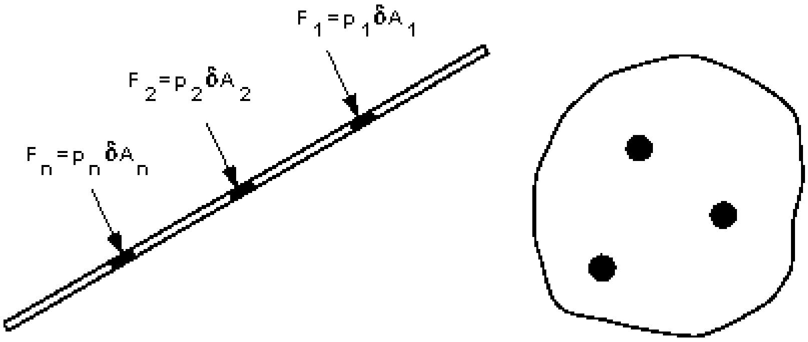
Pressure is defined as force per unit area. If a pressure p acts on a small area δA then the force exerted on that area will be

$$F = p\delta A$$

Since the fluid is at rest the force will act at right-angles to the surface.

General submerged plane

Consider the plane surface shown in the figure below. The total area is made up of many elemental areas. The force on each elemental area is always normal to the surface but, in general, each force is of different magnitude as the pressure usually varies.



We can find the total or ***resultant*** force, R, on the plane by summing up all of the forces on the small elements i.e.

$$R = p_1\delta A_1 + p_2\delta A_2 + \ldots\ldots p_n\delta A_n = \sum p\delta A$$

This resultant force will act through the centre of pressure, hence we can say

If the surface is a **plane** the force can be represented by one single **resultant force**, acting at right-angles to the plane through the **centre of pressure**.

Horizontal submerged plane

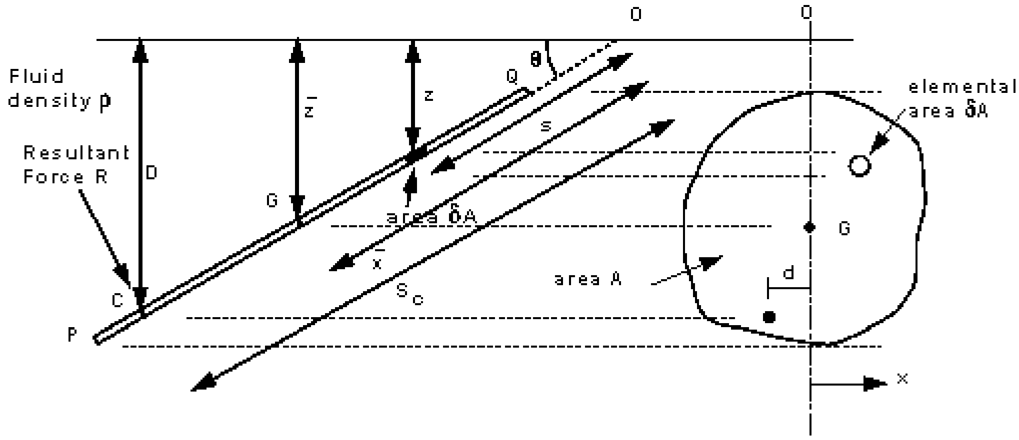
For a horizontal plane submerged in a liquid (or a plane experiencing uniform pressure over its surface), the pressure, *p*, will be equal at all points of the surface. Thus the resultant force will be given by

$$R = \text{pressure} \times \text{area of plane}$$
$$R = pA$$

Curved submerged surface

If the surface is curved, each elemental force will be a different magnitude and in different direction but still normal to the surface of that element. The resultant force can be found by resolving all forces into orthogonal co-ordinate directions to obtain its magnitude and direction. This will **always** be less than the sum of the individual forces, $\sum p\delta A$.

2.4.2 Resultant Force and Centre of Pressure on a submerged plane surface in a liquid.



This plane surface is totally submerged in a liquid of density ρ and inclined at an angle of θ to the horizontal. Taking pressure as zero at the surface and measuring down from the surface, the pressure on an element δA , submerged a distance *z*, is given by

$$p = \rho g z$$

and therefore the force on the element is

$$F = p\delta A = \rho g z \delta A$$

The resultant force can be found by summing all of these forces i.e.

$$R = \rho g \sum z \delta A$$

(assuming ρ and *g* as constant).

The term $\sum z\delta A$ is known as the *1st Moment of Area* of the plane PQ about the free surface. It is equal to $A\bar{z}$ i.e.

$$\sum z\delta A = A\bar{z}$$
$$= 1^{st} \text{ moment of area about the line of the free surface}$$

where *A* is the area of the plane and $\bar{z}$ is the depth (distance from the free surface) to the centroid, G. This can also be written in terms of distance from point O (as $\bar{z} = \bar{x}\sin\theta$)

$$\sum z\delta A = A\bar{x}\sin\theta$$
$$= 1^{st} \text{ Moment of area about a line through O} \times \sin\theta$$

Thus:

The resultant force on a plane

$$R = \rho g A \bar{z}$$

$$\rho g A \bar{x} \sin\theta$$

This resultant force acts at right angles to the plane through the centre of pressure, C, at a depth D. The moment of R about any point will be equal to the sum of the moments of the forces on all the elements δA of the plane about the same point. We use this to find the position of the centre of pressure.

It is convenient to take moments about the point where a projection of the plane passes through the surface, point O in the figure.

$$\text{Moment of } R \text{ about O} = \text{Sum of moments of force on all elements of } \delta A \text{ about O}$$

We can calculate the force on each elemental area:

$$\text{Force on } \delta A = \rho g z \delta A$$

$$\rho g\, s \quad \theta\, \delta A$$

$$\delta A \qquad = \rho g\, s \sin\theta\, \delta A \times s$$

$$= \rho g \sin\theta\, \delta A s^2$$

ρ, g and θ are the same for each element, so the total moment is

$$\text{Sum of moments of forces on all elemets of } \delta\text{A about O} = \rho g \sin\theta \sum s^2 \delta A$$

We know the resultant force from above $R = \rho g A \bar{x} \sin\theta$, which acts through the centre of pressure at C, so

$$\text{Moment of } R \text{ about O} = \rho g A \bar{x} \sin\theta\, S_c$$

Equating gives,

$$\rho g A \bar{x} \sin\theta\, S_c = \rho g \sin\theta \sum s^2 \delta A$$

Thus the position of the centre of pressure along the plane measure from the point O is:

$$S_c = \frac{\sum s^2 \delta A}{A\bar{x}}$$

It look a rather difficult formula to calculate - particularly the summation term. Fortunately this term is known as the *2nd Moment of Area* , I_o, of the plane about the axis through O and it can be easily calculated for many common shapes. So, we know:

$$\text{2nd moment of area about O} = I_o = \sum s^2 \delta A$$

And as we have also seen that $A\bar{x} = 1^{\text{st}}$ Moment of area about a line through O,

Thus the position of the centre of pressure along the plane measure from the point O is:

$$S_c = \frac{2^{\text{nd}} \text{ Moment of area about a line through O}}{1^{\text{st}} \text{ Moment of area about a line through O}}$$

and epth to the centre of pressure is

$$D = S_c \sin\theta$$

2.4.2.1 How do you calculate the 2nd moment of area?

To calculate the 2nd moment of area of a plane about an axis through O, we use the *parallel axis theorem* together with values of the 2nd moment of area about an axis though the centroid of the shape obtained from tables of geometric properties.

The *parallel axis theorem* can be written

$$I_o = I_{GG} + A\bar{x}^2$$

where I_{GG} is the 2nd moment of area about an axis though the centroid G of the plane.

Using this we get the following expressions for the position of the centre of pressure

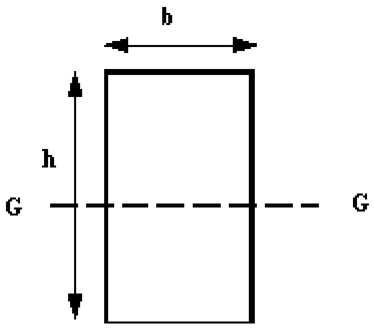
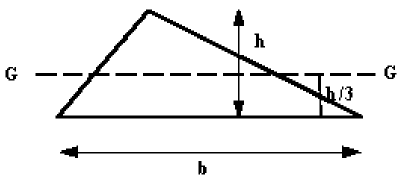
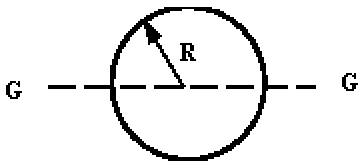
$$S_c = \frac{I_{GG}}{A\bar{x}} + \bar{x}$$

$$D = \sin\theta\left(\frac{I_{GG}}{A\bar{x}} + \bar{x}\right)$$

(In the examination the parallel axis theorem and the I_{GG} will be given)

2.4.2.2 *The second moment of area of some common shapes.*

The table blow given some examples of the 2nd moment of area about a line through the centroid of some common shapes.

Shape	Area A	2nd moment of area, I_{GG}, about an axis through the centroid
Rectangle 	bd	$\frac{bd^3}{12}$
Triangle 	$\frac{bd}{2}$	$\frac{bd^3}{36}$
Circle 	πR^2	$\frac{\pi R^4}{4}$
Semicircle 	$\frac{\pi R^2}{2}$	$0.1102R^4$

<u>Lateral position of Centre of Pressure</u>

If the shape is symmetrical the centre of pressure lies on the line of symmetry. But if it is not symmetrical its position must be found by taking moments about the line OG in the same way as we took moments along the line through O, i.e.

$$R \times d = \text{ Sum of the moments of the force on all elemnts of } \delta\text{A about OG}$$

$$= \sum \rho g z \delta A x$$

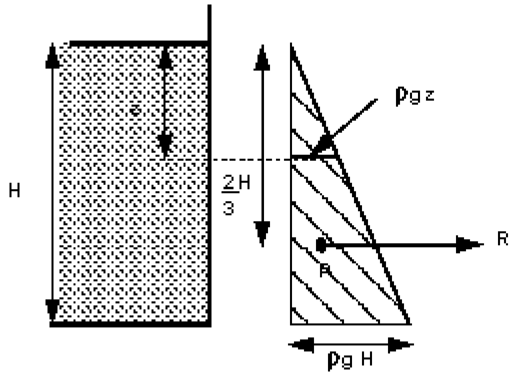
but we have $R = \rho g A \bar{z}$ so

$$d = \frac{\sum \delta A \, zx}{A\bar{z}}$$

2.4.3 Submerged vertical surface - Pressure diagrams

For vertical walls of constant width it is usually much easier to find the resultant force and centre of pressure. This is done graphically by means of a pressure diagram.

Consider the tank in the diagram below having vertical walls and holding a liquid of density ρ to a depth of H. To the right can be seen a graphical representation of the (gauge) pressure change with depth on one of the vertical walls. Pressure increases from zero at the surface linearly by $p = \rho g z$, to a maximum at the base of $p = \rho g H$.



Pressure diagram for vertical wall.

The area of this triangle represents the **resultant force per unit width** on the vertical wall, using SI units this would have units of Newtons per metre. So

$$\begin{aligned} \text{Area } &= \frac{1}{2} \times AB \times BC \\ &= \frac{1}{2} H \rho g H \\ &= \frac{1}{2} \rho g H^2 \end{aligned}$$

Resultant force per unit width

$$R = \frac{1}{2} \rho g H^2 \qquad (N/m)$$

The force acts through the centroid of the pressure diagram. For a triangle the centroid is at 2/3 its height, i.e. in the figure above the resultant force acts horizontally through the point $z = \frac{2}{3} H$.

For a vertical plane the depth to the centre of pressure is given by

$$D = \frac{2}{3} H$$

This can be checked against the previous method:

The resultant force is given by:

$$\begin{aligned} R &= \rho g A \bar{z} = \rho g A \bar{x} \sin\theta \\ &= \rho g (H \times 1) \frac{H}{2} \sin\theta \\ &= \frac{1}{2} \rho g H^2 \end{aligned}$$

and the depth to the centre of pressure by:

$$D = \sin\theta\left(\frac{I_o}{A\bar{x}}\right)$$

and by the parallel axis theorem (with width of 1)

$$\begin{aligned} I_o &= I_{GG} + A\bar{x}^2 \\ &= \frac{1 \times H^3}{12} + 1 \times H\left(\frac{H}{2}\right)^2 \\ &= \frac{H^3}{12} + \frac{H^3}{4} \\ &= \frac{H^3}{3} \end{aligned}$$

Giving depth to the centre of pressure

$$\begin{aligned} D &= \left(\frac{H^3/3}{H^2/2}\right) \\ &= \frac{2}{3}H \end{aligned}$$

These two results are identical to the pressure diagram method.
The same pressure diagram technique can be used when combinations of liquids are held in tanks (e.g. oil floating on water) with position of action found by taking moments of the individual resultant forces for each fluid. Look at the examples to examine this area further.

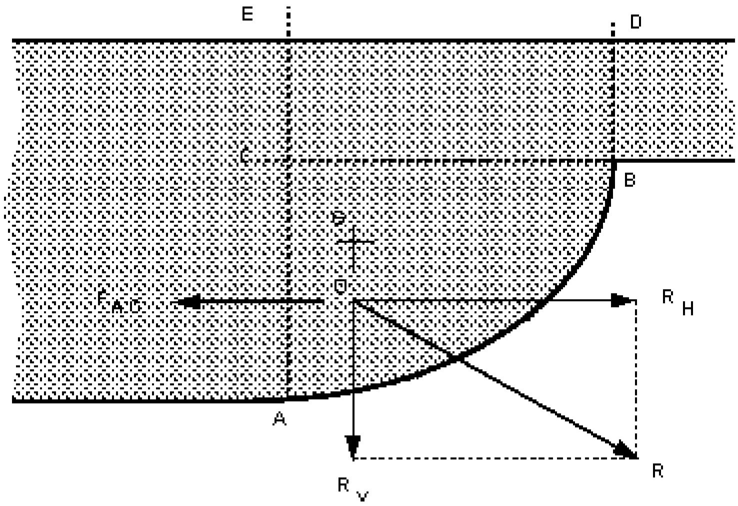
More complex pressure diagrams can be draw for non-rectangular or non-vertical planes but it is usually far easier to use the moments method.

2.4.4 Resultant force on a submerged curved surface

As stated above, if the surface is curved the forces on each element of the surface will not be parallel and must be combined using some vectorial method.

It is most straightforward to calculate the horizontal and vertical components and combine these to obtain the resultant force and its direction. (This can also be done for all three dimensions, but here we will only look at one vertical plane).

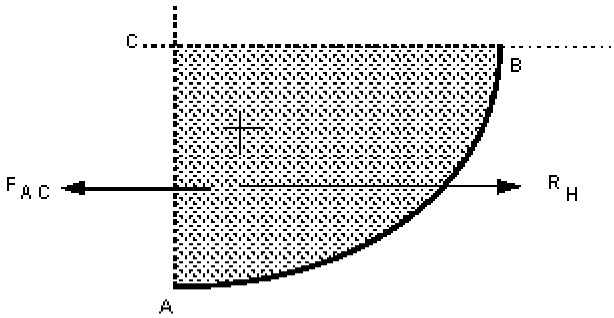
In the diagram below the liquid is resting on top of a curved base.



The element of fluid ABC is equilibrium (as the fluid is at rest).

<u>Horizontal forces</u>

Considering the horizontal forces, none can act on CB as there are no shear forces in a static fluid so the forces would act on the faces AC and AB as shown below.



We can see that the horizontal force on AC, F_{AC}, must equal and be in the opposite direction to the resultant force R_H on the curved surface.

As AC is the projection of the curved surface AB onto a vertical plane, we can generalise this to say

The resultant horizontal force of a fluid above a curved surface is: R_H = Resultant force on the projection of the curved surface onto a vertical plane.

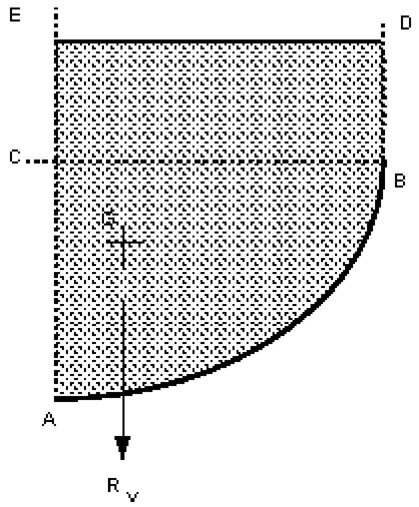
We know that the force on a vertical plane must act horizontally (as it acts normal to the plane) and that R_H must act through the same point. So we can say

R_H acts horizontally through the centre of pressure of the projection of the curved surface onto an vertical plane.

Thus we can use the pressure diagram method to calculate the position and magnitude of the resultant horizontal force on a two dimensional curved surface.

Vertical forces

The diagram below shows the vertical forces which act on the element of fluid above the curved surface.



There are no shear force on the vertical edges, so the vertical component can only be due to the weight of the fluid. So we can say

> The resultant vertical force of a fluid above a curved surface is:
>
> $$R_V = \text{Weight of fluid directly above the curved surface.}$$
>
> and it will act vertically downward through the centre of gravity of the mass of fluid.

Resultant force

The overall resultant force is found by combining the vertical and horizontal components vectorialy,

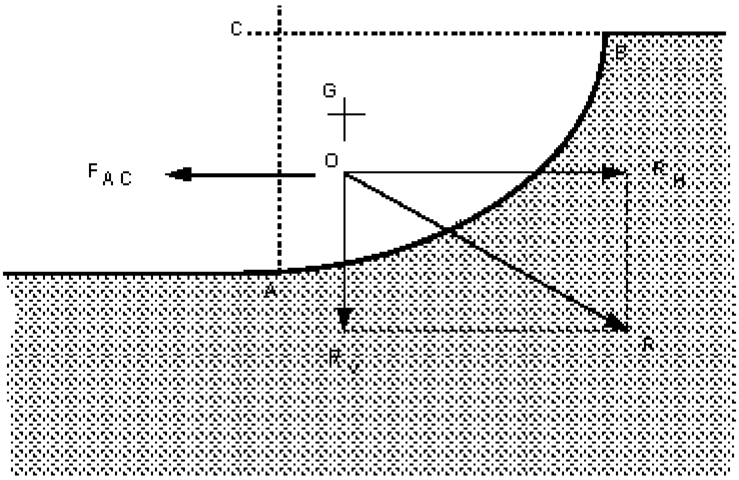
> Resultant force
>
> $$R = \sqrt{R_H^2 + R_V^2}$$

And acts through O at an angle of θ.

> The angle the resultant force makes to the horizontal is
>
> $$\theta = \tan^{-1}\left(\frac{R_V}{R_H}\right)$$

The position of O is the point of integration of the horizontal line of action of R_H and the vertical line of action of R_V.

What are the forces if the fluid is ***below*** *the curved surface*? This situation may occur or a curved sluice gate for example. The figure below shows a situation where there is a curved surface which is experiencing fluid pressure from below.

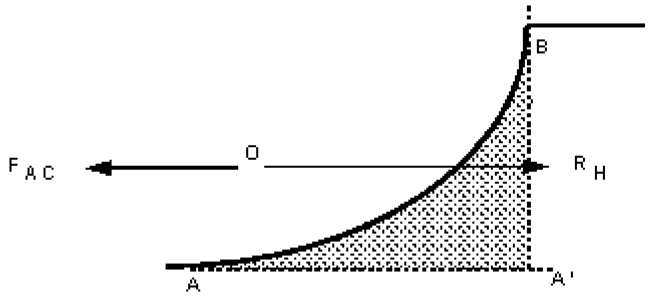


The calculation of the forces acting from the fluid below is very similar to when the fluid is above.

<u>Horizontal force</u>

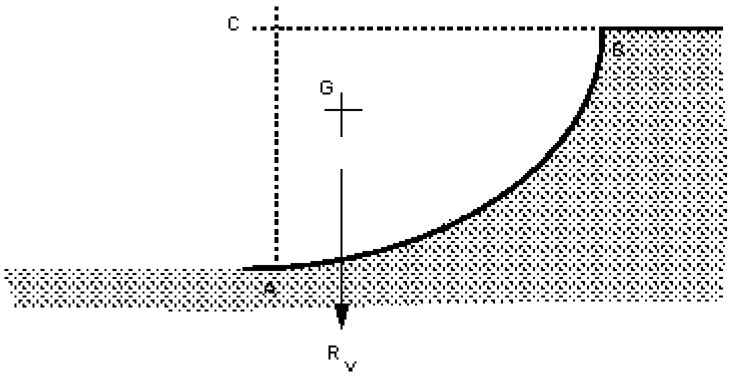
From the figure below we can see the only two horizontal forces on the area of fluid, which is in equilibrium, are the horizontal reaction force which is equal and in the opposite direction to the pressure force on the vertical plane A'B. The resultant horizontal force, R_H acts as shown in the diagram. Thus we can say:

> The resultant horizontal force of a fluid below a curved surface is:
>
> $$R_H = \text{Resultant force on the projection of the curved surface on a vertical plane}$$



<u>Vertical force</u>

The vertical force are acting are as shown on the figure below. If the curved surface were removed and the area it were replaced by the fluid, the whole system would be in equilibrium. Thus the force required by the curved surface to maintain equilibrium is equal to that force which the fluid above the surface would exert - i.e. the weight of the fluid.



Thus we can say:

The resultant vertical force of a fluid below a curved surface is:

R_v =Weight of the *imaginary* volume of fluid vertically above the curved surface.

The resultant force and direction of application are calculated in the same way as for fluids above the surface:

Resultant force

$$R = \sqrt{R_H^2 + R_V^2}$$

And acts through O at an angle of θ.

The angle the resultant force makes to the horizontal is

$$\theta = \tan^{-1}\left(\frac{R_V}{R_H}\right)$$

3. Fluid Dynamics

Objectives

- Introduce concepts necessary to analyse fluids in motion
- Identify differences between Steady/unsteady uniform/non-uniform compressible/incompressible flow
- Demonstrate streamlines and stream tubes
- Introduce the Continuity principle through conservation of mass and control volumes
- Derive the Bernoulli (energy) equation
- Demonstrate practical uses of the Bernoulli and continuity equation in the analysis of flow
- Introduce the momentum equation for a fluid
- Demonstrate how the momentum equation and principle of conservation of momentum is used to predict forces induced by flowing fluids

This section discusses the analysis of fluid in motion - fluid dynamics. The motion of fluids can be predicted in the same way as the motion of solids are predicted using the fundamental laws of physics together with the physical properties of the fluid.

It is not difficult to envisage a very complex fluid flow. Spray behind a car; waves on beaches; hurricanes and tornadoes or any other atmospheric phenomenon are all example of highly complex fluid flows which can be analysed with varying degrees of success (in some cases hardly at all!). There are many common situations which are easily analysed.

3.1 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 describe 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 type:

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.

If you imaging the flow in each of the above classes you may imagine that one class is more complex than another. And this is the case - *steady uniform flow* is by far the most simple of the four. You will then be pleased to hear that this course is restricted to only this class of flow. We will not be encountering any non-uniform or unsteady effects in any of the examples (except for one or two quasi-time dependent problems which can be treated at steady).

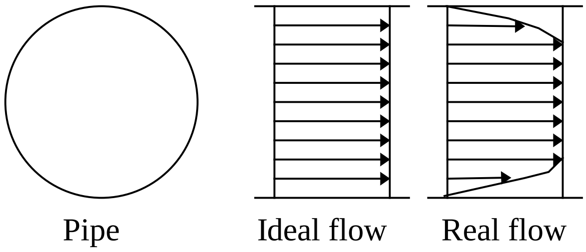
3.1.1 Compressible or Incompressible

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.

3.1.2 Three-dimensional flow

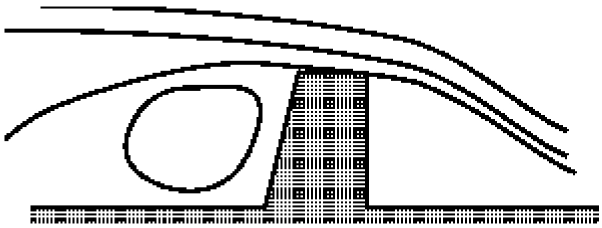
Although in general all fluids flow three-dimensionally, with pressures and velocities and other flow properties varying in all directions, in many cases the greatest changes only occur in two directions or even only in one. In these cases changes in the other direction can be effectively ignored making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

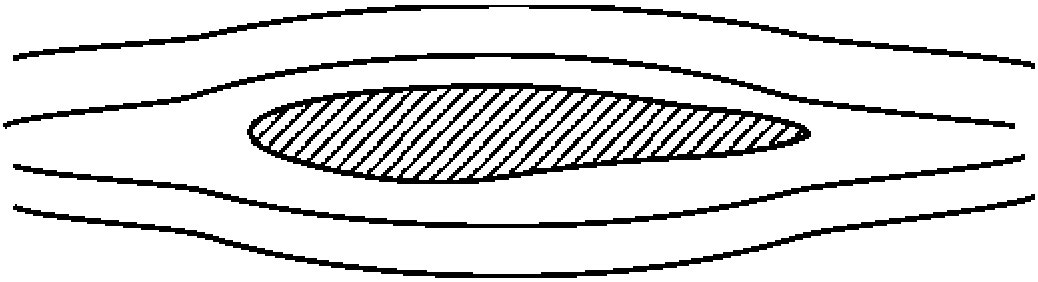


Two-dimensional flow over a weir.

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

3.1.3 Streamlines and streamtubes

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*

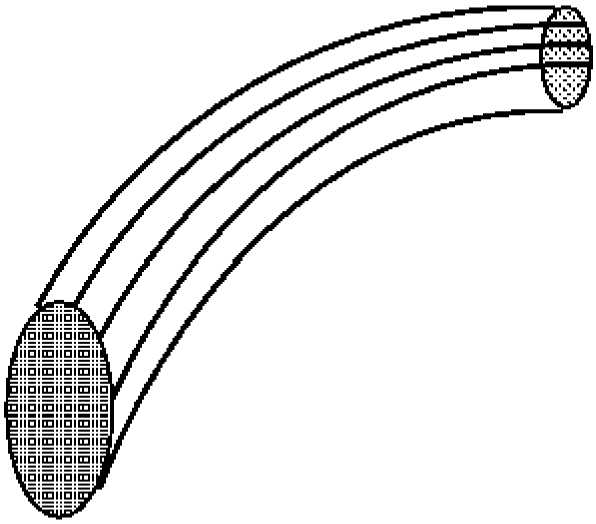
At all points the direction of the streamline is the direction of the fluid velocity: this is how they are defined. Close to the wall the velocity is parallel to the wall so the streamline is also parallel to the wall.

It is also important to recognise 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

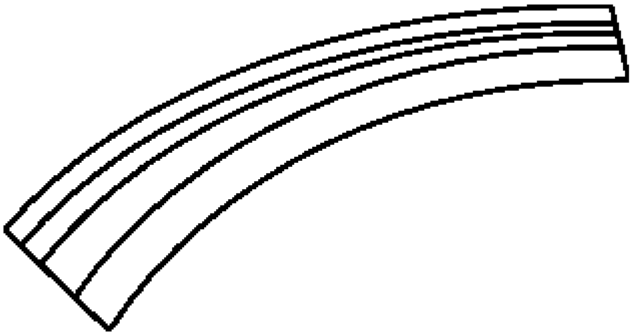
- Because the fluid is moving in the same direction as the streamlines, fluid can not cross a streamline.
- Streamlines can not cross each other. If they were to cross this would indicate two different velocities at the same point. This is not physically possible.
- The above point implies that any particles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

A useful technique in fluid flow analysis is to consider only a part of the total fluid in isolation from the rest. This can be done by imagining a tubular surface formed by streamlines along which the fluid flows. This tubular surface is known as a *streamtube*.



A Streamtube

And in a two-dimensional flow we have a streamtube which is flat (in the plane of the paper):



A two dimensional version of the streamtube

The “walls” of a streamtube are made of streamlines. As we have seen above, fluid cannot flow across a streamline, so fluid cannot cross a streamtube wall. The streamtube can often be viewed as a solid walled pipe. A streamtube is **not** a pipe - it differs in unsteady flow as the walls will move with time. And it differs because the “wall” is moving with the fluid

3.2 Flow rate.

3.2.1 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?

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

3.2.2 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:

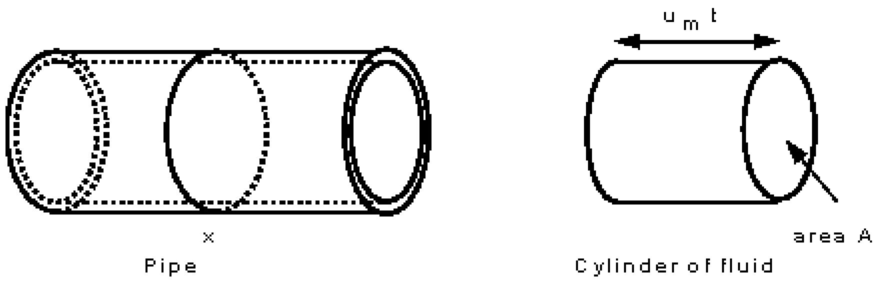
$$
\begin{aligned}
\text{discharge, } Q &= \frac{\text{volume of fluid}}{\text{time}} \\
&= \frac{\text{mass of fluid}}{\text{density} \times \text{time}} \\
&= \frac{\text{mass flow rate}}{\text{density}} \\
&= \frac{0.857}{850} \\
&= 0.001008\ m^3/s \quad (m^3\, s^{-1}) \\
&= 1.008 \times 10^{-3} m^3/s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

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 ($0.001008 m^3/s$ 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* $= 1.0 \times 10^{-3} m^3$). So the solution becomes $1.008\, l/s$. It is far easier to imagine 1 litre than $1.0 \times 10^{-3} m^3$. Units must always be checked, and converted if necessary to a consistent set before using in an equation.

3.2.3 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

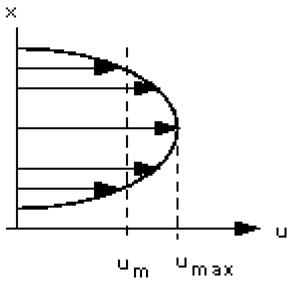
If the area of cross section of the pipe at point X is A, and the mean velocity here is u_m. During a time t, a cylinder of fluid will pass point X with a volume A×u_m×t. The volume per unit time (the discharge) will thus be

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

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.

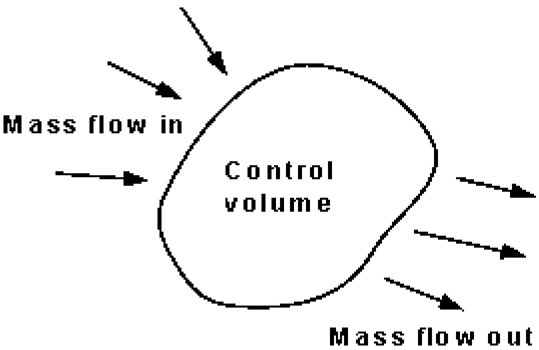


A typical velocity profile across a pipe

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

3.3 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.

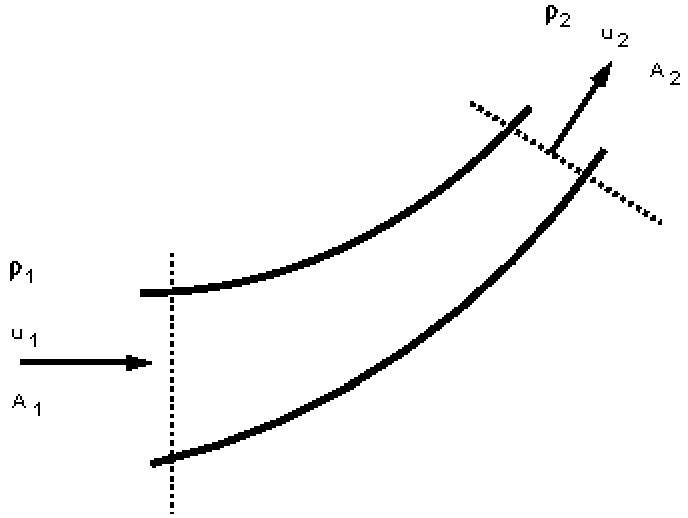
For any control volume the principle of *conservation of mass* says

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

mass entering per unit time at end 1 = 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

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

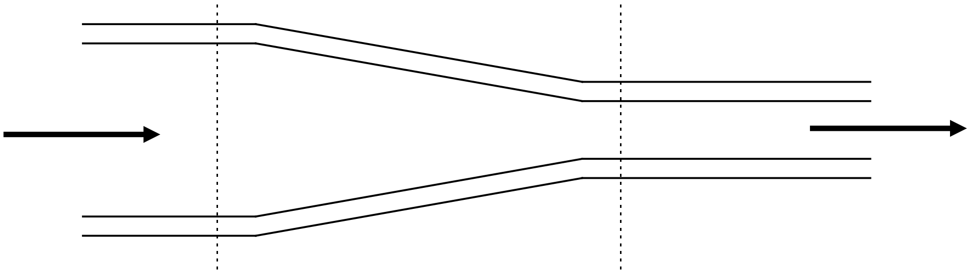
When the fluid can be considered incompressible, i.e. the density does not change, $\rho_1 = \rho_2 = \rho$ so (dropping the *m* subscript)

$A_1 u_1 = A_2 u_2 = Q$ 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:



Section 1 Section 2

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

Discharge at section 1 = Discharge at section 2

$$Q_1 = Q_2$$

$$A_1 u_1 = A_2 u_2$$

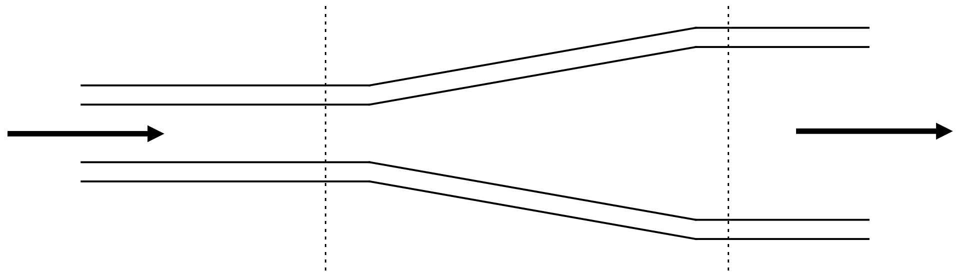
For example if the area $A_1 = 10 \times 10^{-3} m^2$ and $A_2 = 3 \times 10^{-3} m^2$ and the upstream mean velocity, $u_1 = 2.1 m/s$, then the downstream mean velocity can be calculated by

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

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,

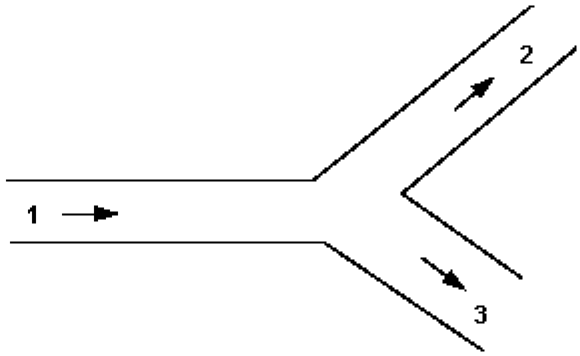


Section 1 Section 2

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

$$\rho_1 Q_1 = \rho_2 Q_2 + \rho_3 Q_3$$

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?

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

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

$$Q_2 = A_2 u_2$$
$$u_2 \quad 0\,936 m \quad s$$

$$Q_3 = A_3 u_3$$
$$u_3 \quad 0\,972\,m \quad s$$

3.4 The Bernoulli Equation - Work and Energy

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 $= \frac{1}{2} m v^2$

Gravitational potential energy $= m g h$

(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 $= m g h$

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

Final potential energy $= 0$

We know that

kinetic energy + potential energy = constant

so

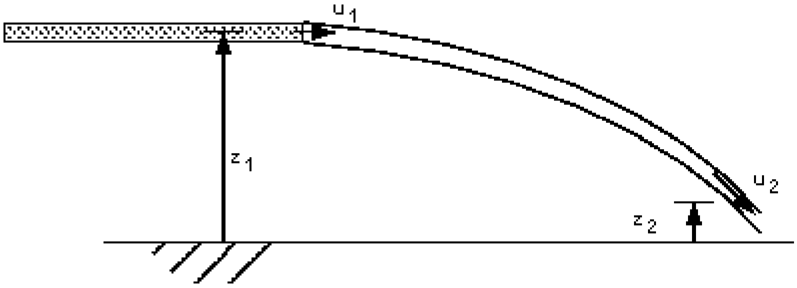
Initial kinetic energy + Initial potential energy = Final kinetic energy + Final potential energy

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

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_1. 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 that due to gravity. The sum of the kinetic and potential energies remains constant (as we neglect energy losses due to friction) so

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

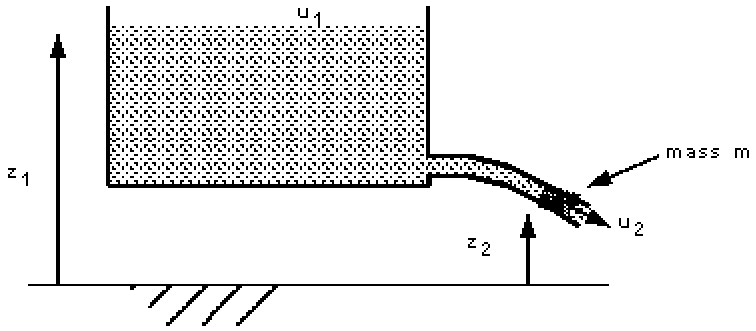
As m is constant this becomes

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

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.

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 ‘idealised reservoir’ in the figure below.



An idealised 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. Summarising

Initial kinetic energy $= 0$

Initial potential energy $= m\,g\,z_1$

Final kinetic energy $= \frac{1}{2}m u_2^2$

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

$$Q = Au$$

$$\pi \quad \frac{0\,015^2}{4} \quad 78\,0$$

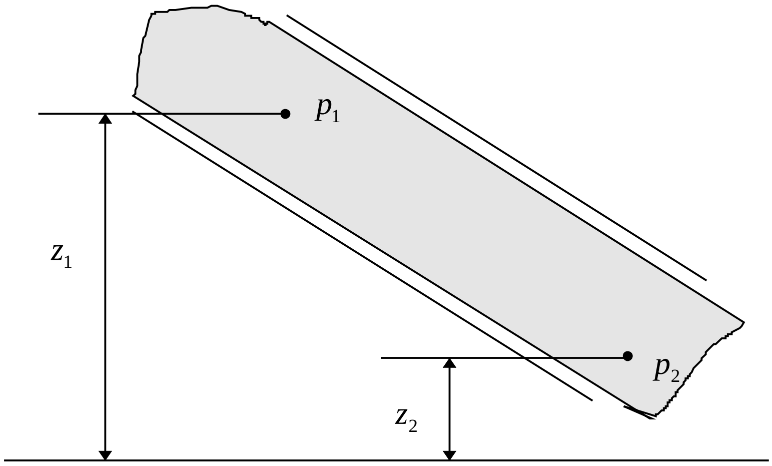
$$0\,01378 m^3 \quad s$$

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?

3.4.1 Bernoulli's Equation

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

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

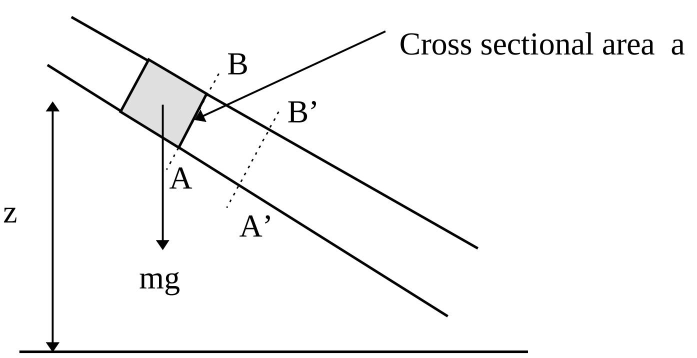
We see that from applying equal pressure or zero velocities we get the two equations from the section above. They are both just special cases of Bernoulli's equation.

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 $\times$ 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

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

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

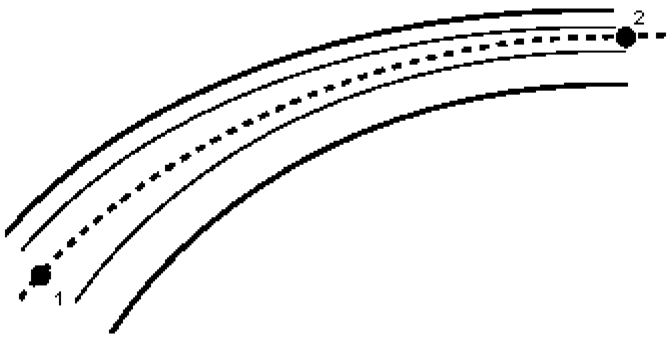
potential head = z

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

or

total head at 1 = total head at 2

or

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

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:

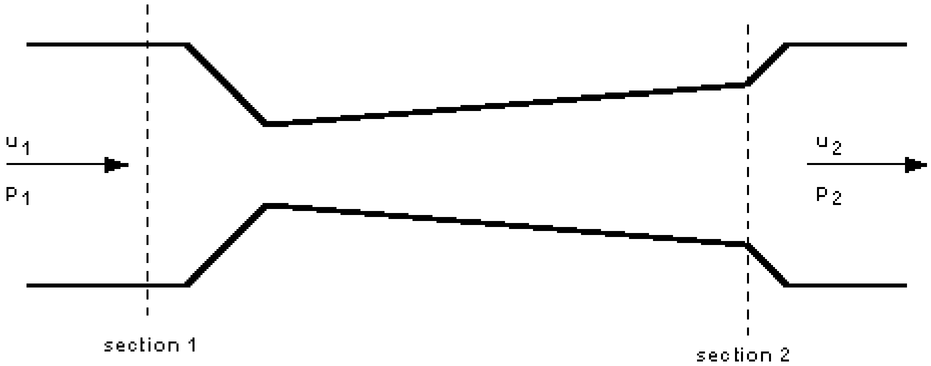
Total energy per unit weight at 1	=	Total energy per unit weight at 2	+	Loss per unit weight	+	Work done per unit weight	−	Energy supplied per unit weight

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

3.4.2 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 ρ = 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 into the tube is equal to the discharge out i.e.

$$A_1 u_1 = A_2 u_2$$

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

$$u_2 \quad \frac{d_1}{d_2} \quad ^2 \; u_1$$

$$7\,8125 m \quad s$$

$$= \qquad - \qquad .$$

$$= \qquad /$$

$$= \quad . \quad /$$

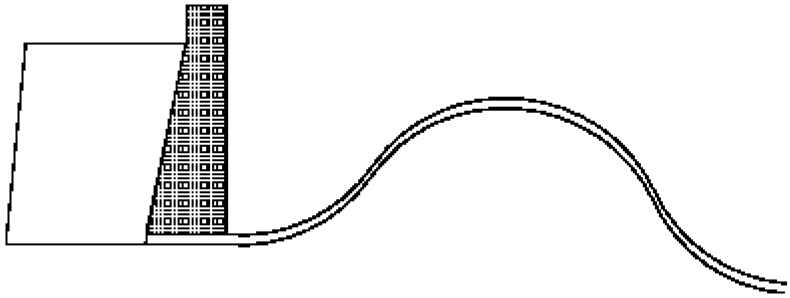
Notice how the velocity has increased while the pressure has decreased. The phenomenon - that pressure decreases as velocity increases - sometimes comes in very useful in engineering. (It is on this principle that carburettor in many car engines work - pressure reduces in a contraction allowing a small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

3.4.3 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.



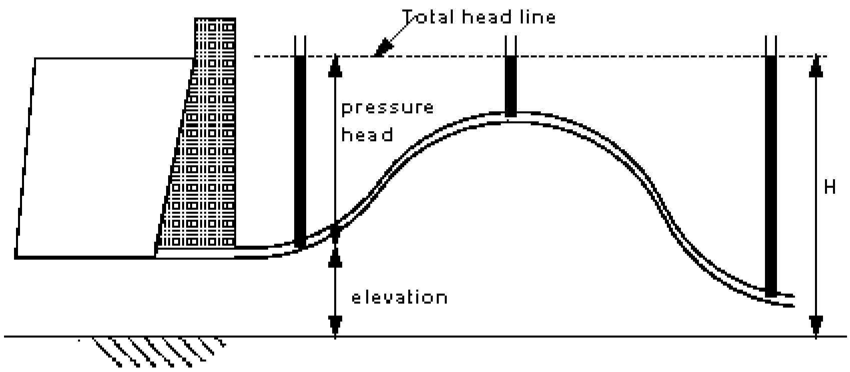
Reservoir feeding a pipe

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).



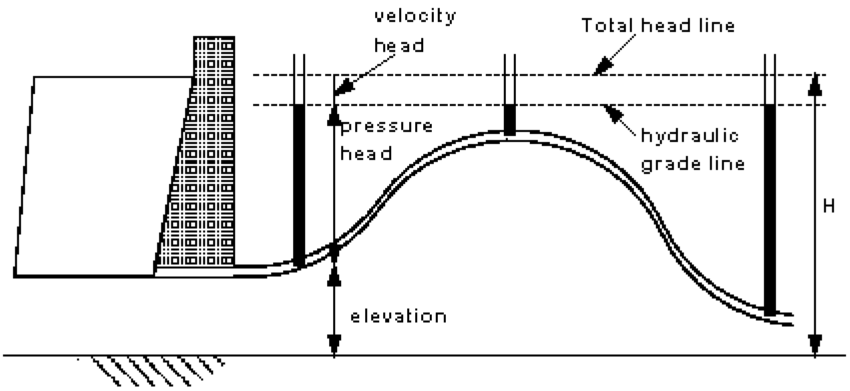
Piezometer levels with zero velocity

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 …

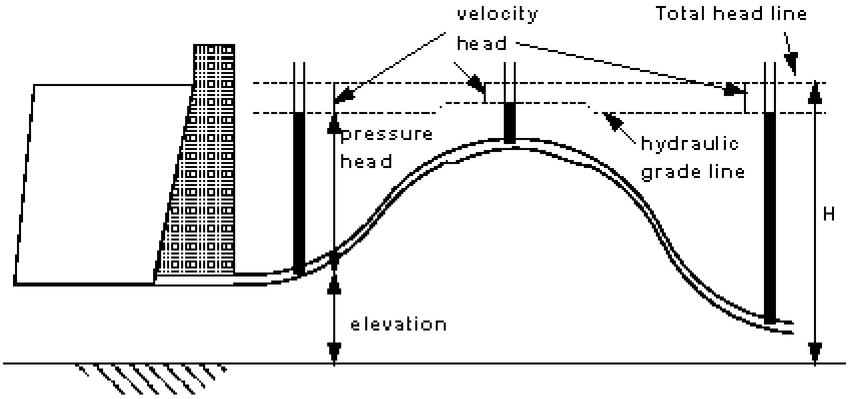


Piezometer levels when fluid is flowing

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

We see in this figure that the levels have reduced by an amount equal to the velocity head, $\frac{u^2}{2g}$. Now as the pipe is of constant diameter we know that the velocity is constant along the pipe so the velocity head is 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

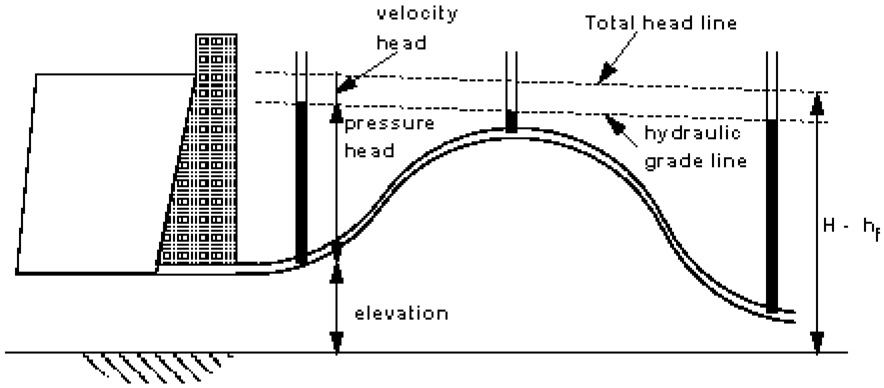
The velocity head at each point is now different. This is because the velocity is different at each point. By considering continuity we know that the velocity is different because the diameter of the pipe is different. Which pipe has the greatest diameter?

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.

3.4.4 Energy 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*.

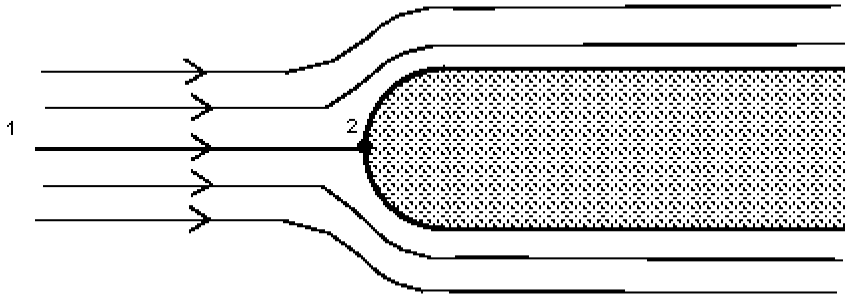
$$\frac{p_1}{\rho g} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{u_2^2}{2g} + z_2 + h_f$$

3.5 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.

3.5.1 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 = $\frac{1}{2g} u_1^2$

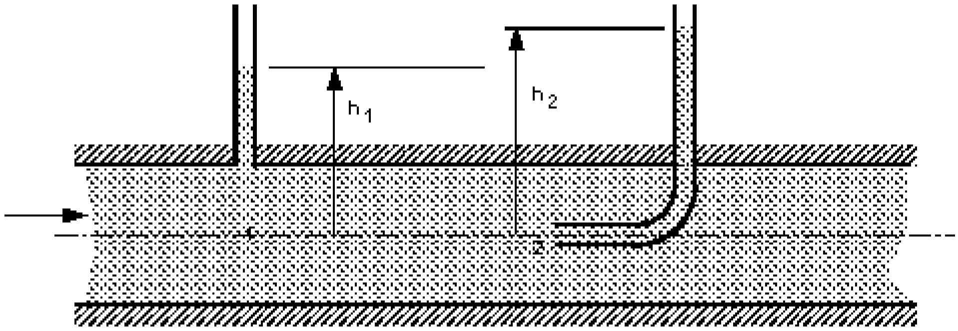
The total pressure is know as the *stagnation pressure* (or *total pressure*)

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 ,

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

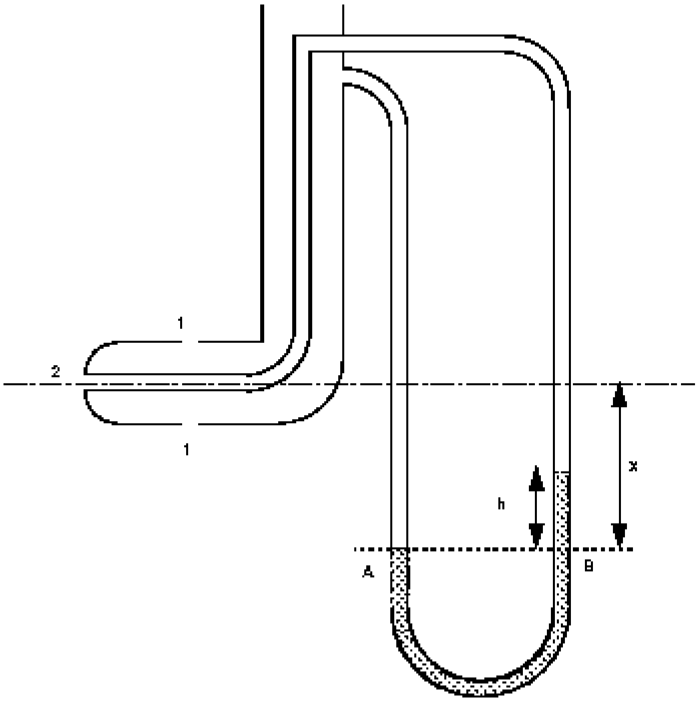
$$\rho g h_2 = \rho g h_1 + \frac{1}{2}\rho u_1^2$$

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

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

3.5.2 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 static tube 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, as before, 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,

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

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

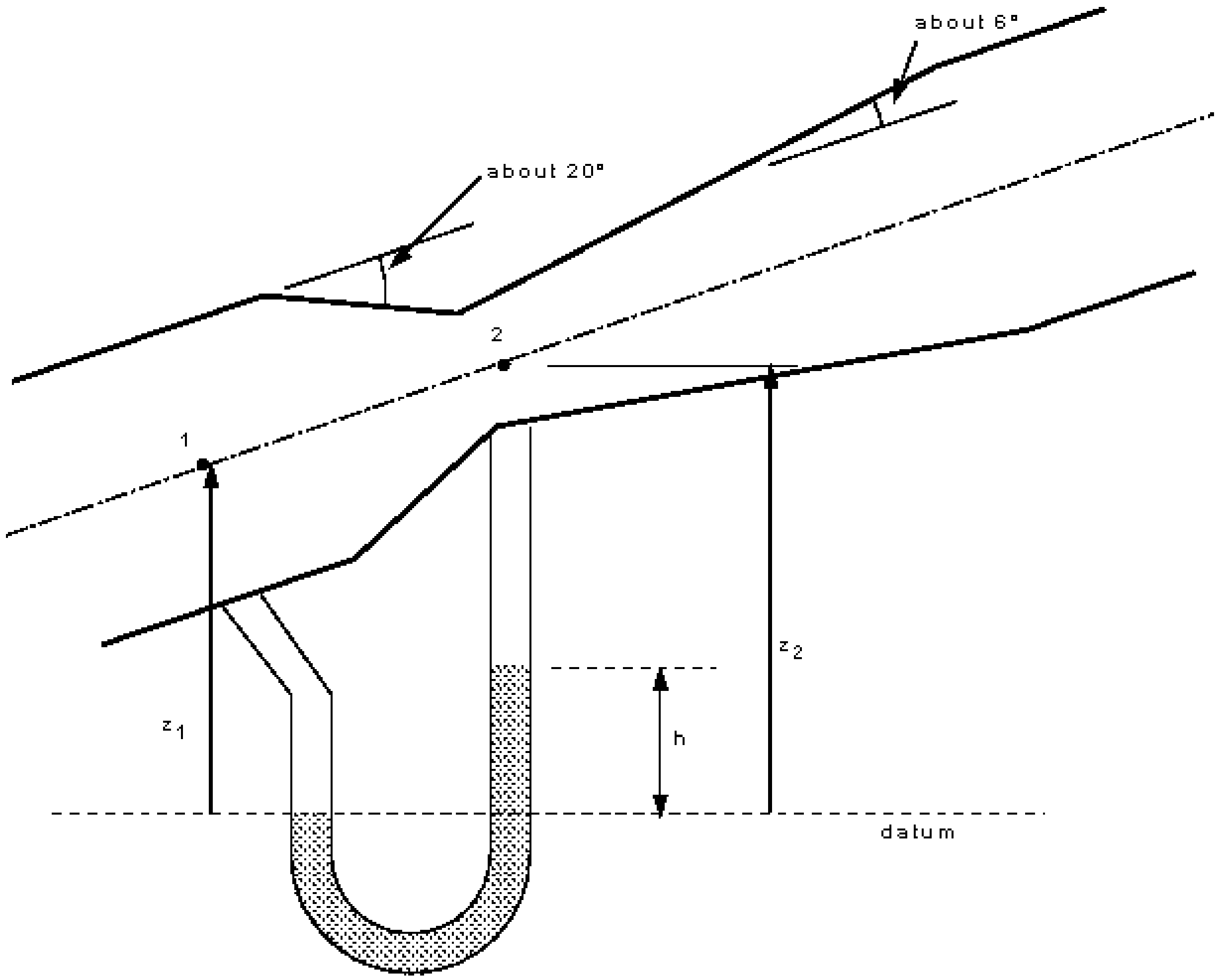
$$p_1 + hg(\rho_{man} - \rho) = p_1 + \frac{\rho u_1^2}{2}$$

$$u_1 = \sqrt{\frac{2gh(\rho_m - \rho)}{\rho}}$$

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.

3.5.3 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::

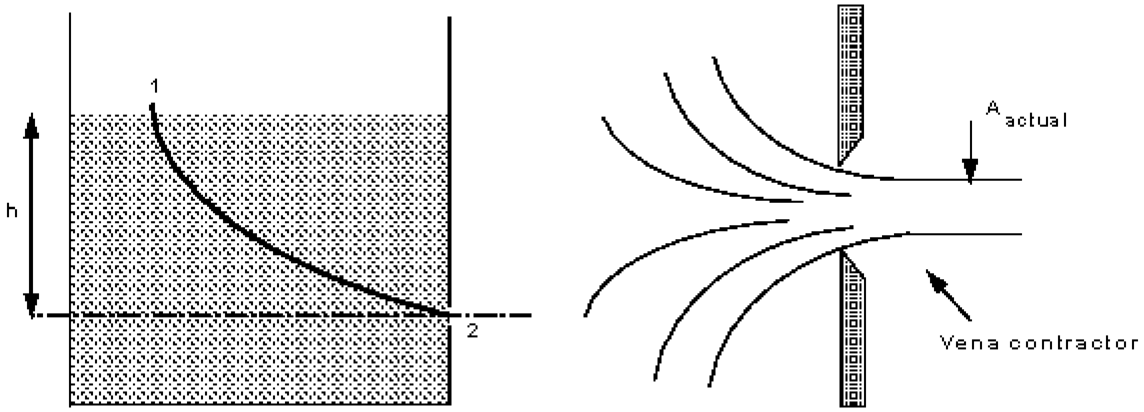
$$Q_{actual} = C_d A_1 A_2 \sqrt{\frac{2gh\left(\frac{\rho_{man}}{\rho} - 1\right)}{A_1^2 - A_2^2}}$$

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%.

3.5.4 Flow Through A Small Orifice

We are to consider the flow from a tank through a hole in the side close to the base. The general arrangement and a close up of the hole and streamlines are shown in the figure below



Tank and streamlines of flow out of the sharp edged orifice

The shape of the holes edges are as they are (sharp) to minimise frictional losses by minimising the contact between the hole and the liquid - the only contact is the very edge.

Looking at the streamlines you can see how they contract after the orifice to a minimum value when they all become parallel, at this point, the velocity and pressure are uniform across the jet. This convergence is called the *vena contracta*. (From the Latin 'contracted vein'). It is necessary to know the amount of contraction to allow us to calculate the flow.

We can predict the velocity at the orifice using the Bernoulli equation. Apply it along the streamline joining point 1 on the surface to point 2 at the centre of the orifice.

At the surface velocity is negligible ($u_1 = 0$) and the pressure atmospheric ($p_1 = 0$).At the orifice the jet is open to the air so again the pressure is atmospheric ($p_2 = 0$). If we take the datum line through the orifice then $z_1 = h$ and $z_2 = 0$, leaving

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

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

This is the theoretical value of velocity. Unfortunately it will be an over estimate of the real velocity because friction losses have not been taken into account. To incorporate friction we use the **coefficient of velocity** to correct the theoretical velocity,

$$u_{actual} = C_v u_{theoretical}$$

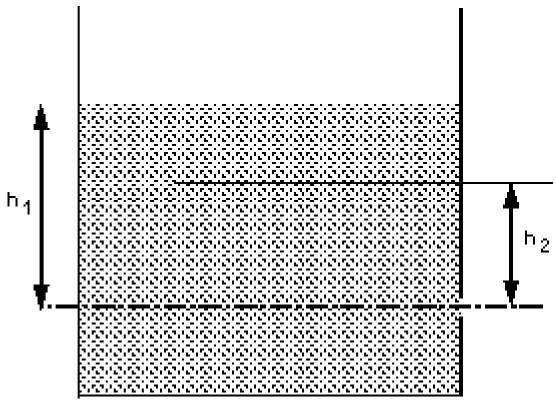
Each orifice has its own coefficient of velocity, they usually lie in the range(0.97 - 0.99)

To calculate the discharge through the orifice we multiply the area of the jet by the velocity. The actual area of the jet is the area of the vena contracta **not** the area of the orifice. We obtain this area by using a **coefficient of contraction** for the orifice

$$A_{actual} = C_c A_{orifice}$$

So the discharge through the orifice is given by

$$
\begin{aligned}
Q &= Au \\
Q_{actual} & \quad A_{actual} u_{actual} \\
& \quad C_c C_v A_{orifice} u_{theoretical} \\
& \quad C_d A_{orifice} u_{theoretical} \\
& \quad C_d A_{orifice} \sqrt{\ \ gh}
\end{aligned}
$$



Tank emptying from level h_1 to h_2.

The tank has a cross sectional area of *A*. In a time *dt* the level falls by *dh* or the flow out of the tank is

$$
\begin{aligned}
Q &= Av \\
Q & \quad A\frac{\delta h}{\delta t}
\end{aligned}
$$

(-ve sign as δh is falling)

Rearranging and substituting the expression for *Q* through the orifice gives

$$\delta t = \frac{-A}{C_d A_o \sqrt{2g}} \frac{\delta h}{\sqrt{h}}$$

This can be integrated between the initial level, h_1, and final level, h_2, to give an expression for the time it takes to fall this distance

$$
\begin{aligned}
t &= \frac{-A}{C_d A_o \sqrt{2g}} \int_{h_1}^{h_2} \frac{\delta h}{\sqrt{h}} \\
&= \frac{-A}{C_d A_o \sqrt{2g}} \left[2\sqrt{h}\right]_{h_1}^{h_2} \\
&= \frac{-2A}{C_d A_o \sqrt{2g}} \left[\sqrt{h_2} - \sqrt{h_1}\right]
\end{aligned}
$$

3.5.6 Submerged Orifice

We have two tanks next to each other (or one tank separated by a dividing wall) and fluid is to flow between them through a submerged orifice. Although difficult to see, careful measurement of the flow indicates that the submerged jet flow behaves in a similar way to the jet in air in that it forms a vena contracta below the surface. To determine the velocity at the jet we first use the Bernoulli equation to give us the ideal velocity. Applying Bernoulli from point 1 on the surface of the deeper tank to point 2 at the centre of the orifice, gives

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

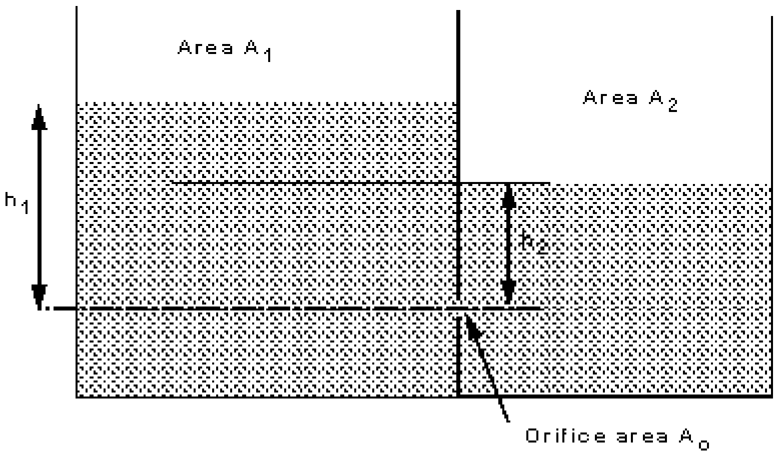
$$0+0+h_1=\frac{\rho g h_2}{\rho g}+\frac{u_2^2}{2g}+0$$

$$u_2=\sqrt{2g(h_1-h_2)}$$

i.e. the ideal velocity of the jet through the submerged orifice depends on the *difference* in head across the orifice. And the discharge is given by

$$Q=C_d A_o u$$

$$C_d A_o \sqrt{\;g\;h\;}\quad h$$



Two tanks of initially different levels joined by an orifice

By a similar analysis used to find the time for a level drop in a tank we can derive an expression for the change in levels when there is flow between two connected tanks.

Applying the continuity equation

$$Q=-A_1\frac{\delta h_1}{\delta t}=A_2\frac{\delta h_2}{\delta t}$$

$$Q\delta t=-A_1\delta h_1=A_2\delta h_2$$

Also we can write $-\delta h_1+\delta h_2=\delta h$

So

$$-A_1\delta h_1 = A_2\delta h_1 - A_2\delta h$$

$$\delta h_1 = \frac{A_2\delta h}{A_1 + A_2}$$

Then we get

$$Q\delta t = -A_1\delta h_1$$

$$C_d A_o\sqrt{2g(h_1 - h_2)}\;\delta t = \frac{A_1A_2}{A_1 + A_2}\delta h$$

Re arranging and integrating between the two levels we get

$$\delta t = \frac{A_1A_2}{(A_1 + A_2)C_d A_o\sqrt{2g}}\frac{\delta h}{\sqrt{h}}$$

$$t = \frac{A_1A_2}{(A_1 + A_2)C_d A_o\sqrt{2g}}\int_{h_{initial}}^{h_{final}}\frac{\delta h}{\sqrt{h}}$$

$$= \frac{2A_1A_2}{(A_1 + A_2)C_d A_o\sqrt{2g}}\left[\sqrt{h}\right]_{h_{initial}}^{h_{final}}$$

$$= \frac{2A_1A_2}{(A_1 + A_2)C_d A_o\sqrt{2g}}\left[\sqrt{h_{initial}} - \sqrt{h_{final}}\right]$$

(remember that *h* in this expression is the *difference* in height between the two levels (h_2 - h_1) to get the time for the levels to equal use $h_{initial} = h_1$ and $h_{final} = 0$).

Thus we have an expression giving the time it will take for the two levels to equal.

3.5.8 Flow Over Notches and Weirs

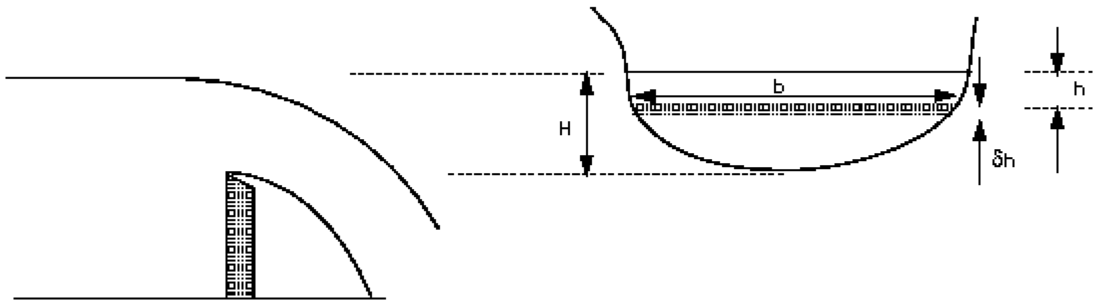
A notch is an opening in the side of a tank or reservoir which extends above the surface of the liquid. It is usually a device for measuring discharge. A weir is a notch on a larger scale - usually found in rivers. It may be sharp crested but also may have a substantial width in the direction of flow - it is used as both a flow measuring device and a device to raise water levels.

3.5.8.1 Weir Assumptions

We will assume that the velocity of the fluid approaching the weir is small so that kinetic energy can be neglected. We will also assume that the velocity through any elemental strip depends only on the depth below the free surface. These are acceptable assumptions for tanks with notches or reservoirs with weirs, but for flows where the velocity approaching the weir is substantial the kinetic energy must be taken into account (e.g. a fast moving river).

3.5.8.2 A General Weir Equation

To determine an expression for the theoretical flow through a notch we will consider a horizontal strip of width *b* and depth *h* below the free surface, as shown in the figure below.



Elemental strip of flow through a notch

$$\text{velocity through the strip,} \quad u = \sqrt{2gh}$$

$$\text{discharge through the strip,} \quad \delta Q = Au = b\delta h\sqrt{2gh}$$

integrating from the free surface, $h = 0$, to the weir crest, $h = H$ gives the expression for the total theoretical discharge

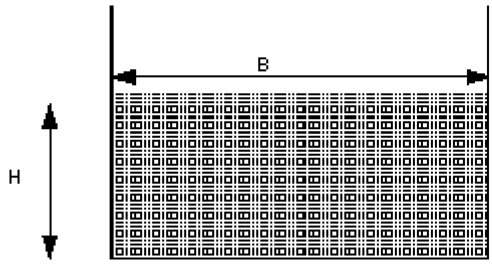
$$Q_{\text{theoretical}} = \sqrt{2g}\int_0^H bh^{1/2}\,dh$$

This will be different for every differently shaped weir or notch. To make further use of this equation we need an expression relating the width of flow across the weir to the depth below the free surface.

3.5.8.3 Rectangular Weir

For a rectangular weir the width does not change with depth so there is no relationship between *b* and depth *h*. We have the equation,

$$b = \text{constant} = B$$



A rectangular weir

Substituting this into the general weir equation gives

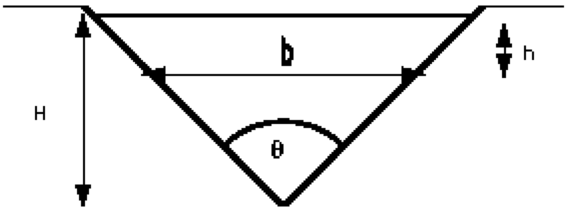
$$Q_{\text{theoretical}} = B\sqrt{2g}\int_0^H h^{1/2}\,dh$$
$$= \frac{2}{3}B\sqrt{2g}H^{3/2}$$

To calculate the actual discharge we introduce a coefficient of discharge, C_d, which accounts for losses at the edges of the weir and contractions in the area of flow, giving

$$Q_{\text{actual}} = C_d\,\frac{2}{3}B\sqrt{2g}H^{3/2}$$

3.5.8.4 'V' Notch Weir

For the "V" notch weir the relationship between width and depth is dependent on the angle of the "V".



"V" notch, or triangular, weir geometry.

If the angle of the "V" is θ then the width, *b*, a depth *h* from the free surface is

$$b = 2(H-h)\tan\left(\frac{\theta}{2}\right)$$

So the discharge is

$$Q_{\text{theoretical}} = 2\sqrt{2g}\,\tan\left(\frac{\theta}{2}\right)\int_0^H (H-h)h^{1/2}\,dh$$
$$= 2\sqrt{2g}\,\tan\left(\frac{\theta}{2}\right)\left[\frac{2}{5}Hh^{3/2} - \frac{2}{5}h^{5/2}\right]_0^H$$
$$= \frac{8}{15}\sqrt{2g}\,\tan\left(\frac{\theta}{2}\right)H^{5/2}$$

And again, the actual discharge is obtained by introducing a coefficient of discharge

$$Q_{\text{actual}} = C_d\,\frac{8}{15}\sqrt{2g}\,\tan\left(\frac{\theta}{2}\right)H^{5/2}$$

3.6 The Momentum Equation

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.

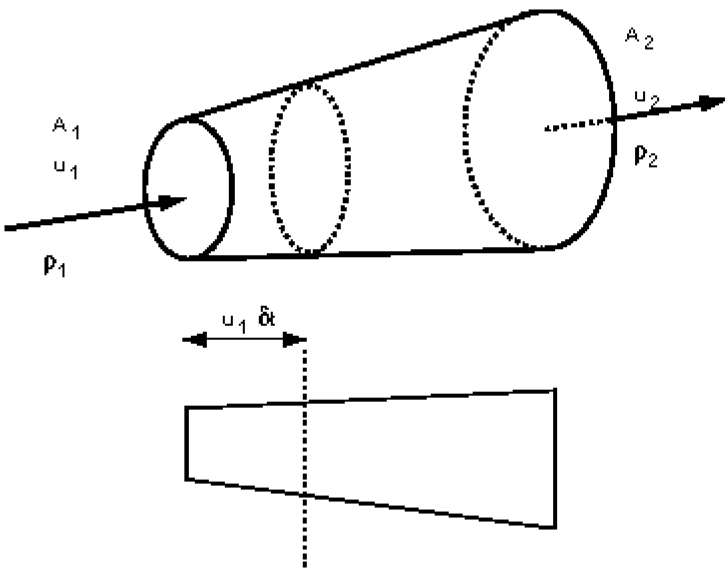
The momentum equation is a statement of Newton's Second Law and relates the sum of the forces acting on an element of fluid to its acceleration or rate of change of momentum. You will probably recognise the equation $F = ma$ which is used in the analysis of solid mechanics to relate applied force to 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.

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

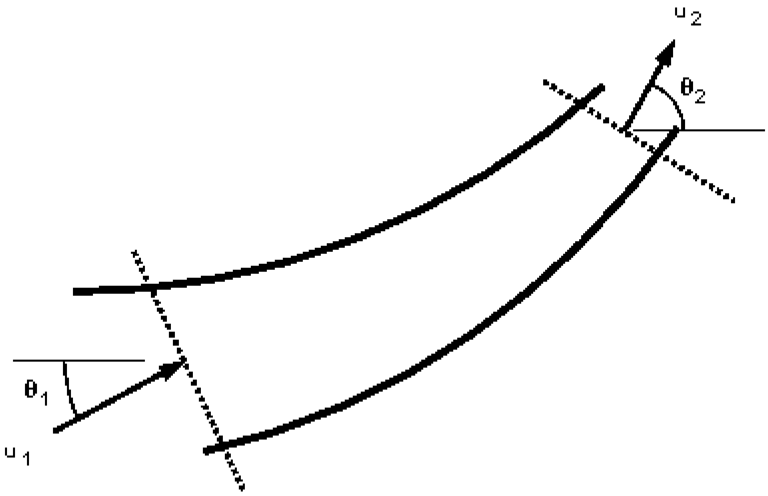
$$F = \dot{m}(u_2 - u_1)$$

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

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.

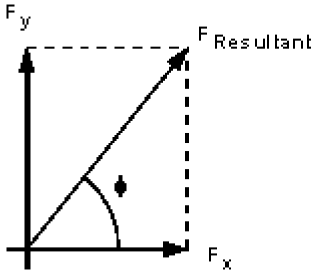
The force in the x-direction

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

And the force in the y-direction

$$\begin{aligned}
F_y &= \dot{m}\left(u_2 \sin\theta_2 - u_1 \sin\theta_1\right) \\
&= \dot{m}\left(u_{2_y} - u_{1_y}\right) \\
&= \rho Q\left(u_2 \sin\theta_2 - u_1 \sin\theta_1\right) \\
&= \rho Q\left(u_{2_y} - u_{1_y}\right)
\end{aligned}$$

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



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

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

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

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 volume

$F_B =$ 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

$$R = -F_R$$

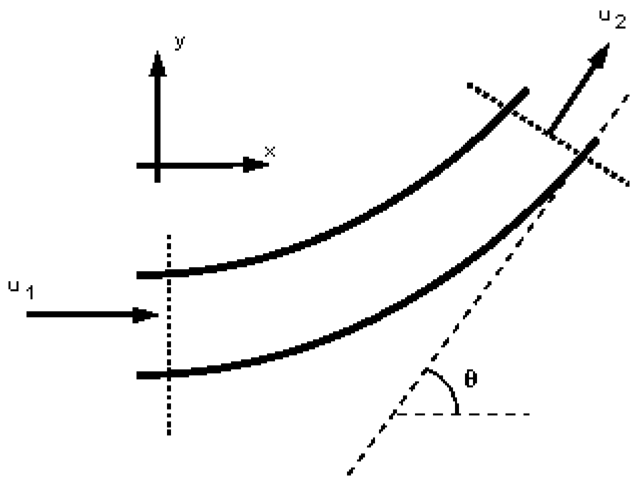
3.7 Application of the Momentum Equation

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.
4. Force due to flow round a curved vane.

3.7.1 The force due the flow around a pipe bend

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



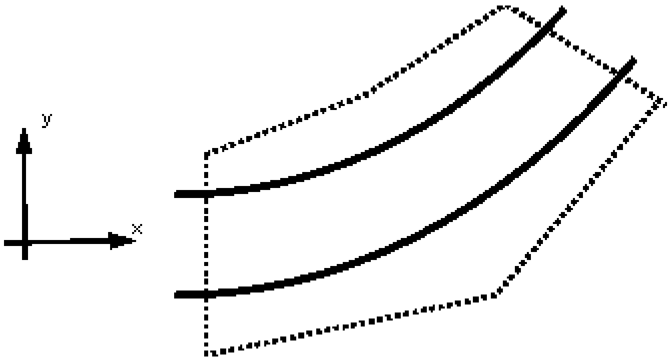
Flow round a pipe bend of constant cross-section

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:

1. Draw a control volume
2. Decide on co-ordinate axis system
3. Calculate the **total** force
4. Calculate the **pressure** force
5. Calculate the **body** force
6. Calculate the **resultant** force

1. <u>Control Volume</u>



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

2 <u>Co-ordinate axis system</u>

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.

3 <u>Calculate the **total** force</u>

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:

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

4 <u>Calculate the **pressure** force</u>

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

5 <u>Calculate the **body** force</u>

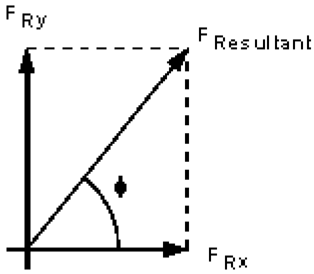
There are no body forces in the x or y directions. The only body force is that exerted by gravity (which acts into the paper in this example - a direction we do not need to consider).

6 Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{T_y} = F_{R_y} + F_{P_y} + F_{B_y}$$

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

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

$$R = -F_R$$

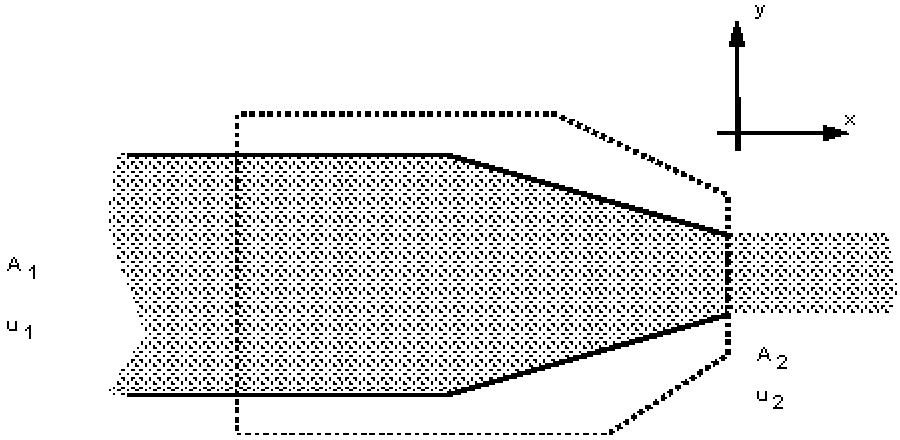
3.7.2 Force on a pipe nozzle

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:

1. Draw a control volume
2. Decide on co-ordinate axis system
3. Calculate the **total** force
4. Calculate the **pressure** force
5. Calculate the **body** force
6. Calculate the **resultant** force

1 & 2 Control volume and Co-ordinate axis are shown in the figure below.



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

3 <u>Calculate the **total** force</u>

$$F_T = F_{T\,x} = \rho Q\left(u_2 - u_1\right)$$

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

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

4 <u>Calculate the **pressure** force</u>

$$F_P = F_{P\,x} = \text{pressure force at 1 - pressure force at 2}$$

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

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

5 <u>Calculate the **body** force</u>

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.

6 Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{R_x} = F_{T_x} - F_{P_y} - 0$$

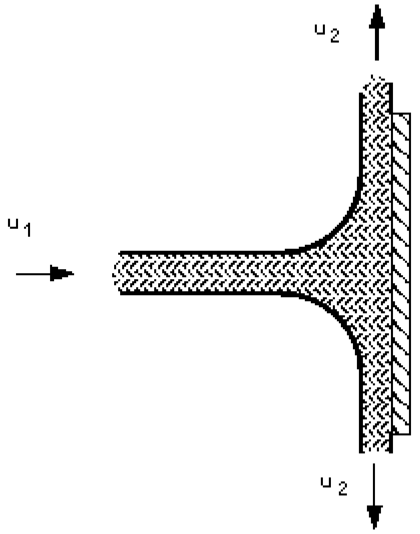
$$F_{T_x} = \rho Q^2 \left(\frac{1}{A_2} - \frac{1}{A_1} \right) - \frac{\rho Q^2}{2} \left(\frac{1}{A_2^2} - \frac{1}{A_1^2} \right)$$

So the fireman must be able to resist the force of

$$R = -F_{T_x}$$

3.7.3 Impact of a Jet on a Plane

We will first consider a jet hitting a flat plate (a plane) at an angle of 90°, as shown in the figure below. We want to find the reaction force of the plate i.e. the force the plate will have to apply to stay in the same position.

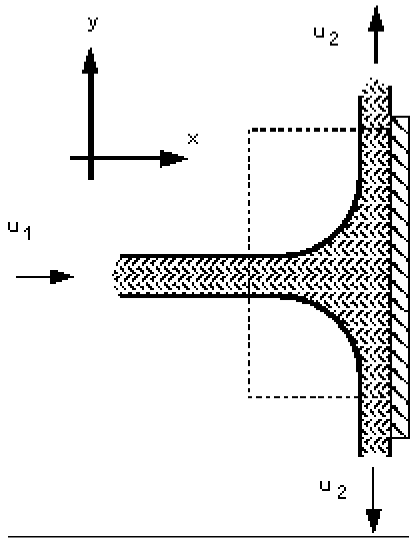


A perpendicular jet hitting a plane.

The analysis take the same procedure as above:

1. Draw a control volume
2. Decide on co-ordinate axis system
3. Calculate the **total** force
4. Calculate the **pressure** force
5. Calculate the **body** force
6. Calculate the **resultant** force

1 & 2 Control volume and Co-ordinate axis are shown in the figure below.



3 Calculate the **total** force

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

As the system is symmetrical the forces in the y-direction cancel i.e.

$$F_{T_y} = 0$$

4 Calculate the **pressure force.**

The pressure force is zero as the pressure at both the inlet and the outlets to the control volume are atmospheric.

5 Calculate the **body** force

As the control volume is small we can ignore the body force due to the weight of gravity.

6 Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{R_x} = F_{T_x} - 0 - 0$$
$$= -\rho Q u_{1_x}$$

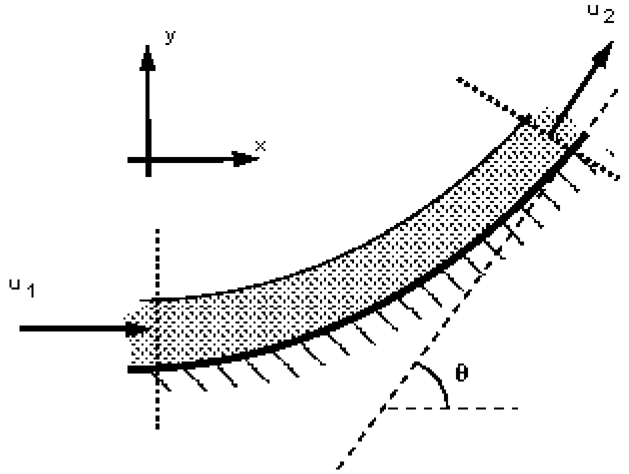
Exerted **on the fluid.**

The force **on the plane** is the same magnitude but in the opposite direction

$$R = -F_{R_x}$$

3.7.4 Force on a curved vane

This case is similar to that of a pipe, but the analysis is simpler because the pressures are equal - atmospheric , and both the cross-section and velocities (in the direction of flow) remain constant. The jet, vane and co-ordinate direction are arranged as in the figure below.



Jet deflected by a curved vane.

1 & 2 Control volume and Co-ordinate axis are shown in the figure above.

3 Calculate the **total** force in the x direction

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

but $u_1 = u_2 = \frac{Q}{A}$, so

$$F_{T_x} = -\rho \frac{Q^2}{A}(1 - \cos\theta)$$

and in the y-direction

$$F_{T_y} = \rho Q(u_2 \sin\theta - 0)$$

$$= \rho \frac{Q^2}{A}$$

4 Calculate the **pressure force.**

Again, the pressure force is zero as the pressure at both the inlet and the outlets to the control volume are atmospheric.

5 Calculate the **body** force

No body forces in the x-direction, $F_{B_x} = 0$.

In the y-direction the body force acting is the weight of the fluid. If V is the volume of the fluid on he vane then,

$$F_{B_x} = \rho g V$$

(This is often small is the jet volume is small and sometimes ignored in analysis.)

6 Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{R_x} = F_{T_x}$$

$$F_{T_y} = F_{R_y} + F_{P_y} + F_{B_y}$$
$$F_{R_y} = F_{T_y}$$

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

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

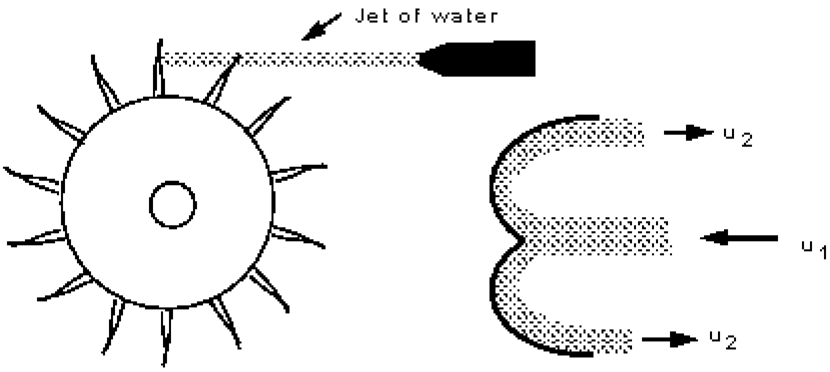
exerted on the fluid.

The force **on the vane** is the same magnitude but in the opposite direction

$$R = -F_R$$

3.7.5 Pelton wheel blade

The above analysis of impact of jets on vanes can be extended and applied to analysis of turbine blades. One particularly clear demonstration of this is with the blade of a turbine called the *pelton wheel*. The arrangement of a pelton wheel is shown in the figure below. A narrow jet (usually of water) is fired at blades which stick out around the periphery of a large metal disk. The shape of each of these blade is such that as the jet hits the blade it splits in two (see figure below) with half the water diverted to one side and the other to the other. This splitting of the jet is beneficial to the turbine mounting - it causes equal and opposite forces (hence a sum of zero) on the bearings.



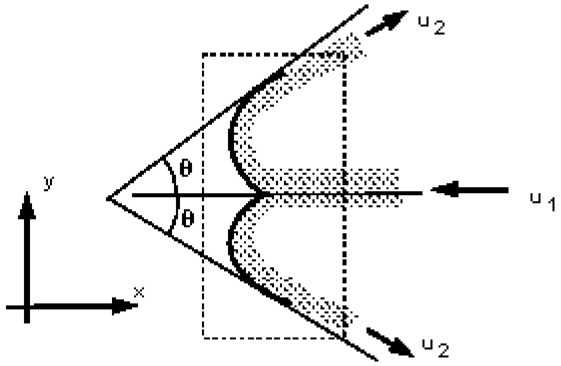
Pelton wheel arrangement and jet hitting cross-section of blade.

A closer view of the blade and control volume used for analysis can be seen in the figure below. Analysis again take the following steps:

1. Draw a control volume
2. Decide on co-ordinate axis system
3. Calculate the **total** force
4. Calculate the **pressure** force
5. Calculate the **body** force

6. Calculate the **resultant** force

1 & 2 Control volume and Co-ordinate axis are shown in the figure below.



3 Calculate the **total** force in the x direction

$$F_{T_x} = \rho\left(\frac{Q}{2}u_{2_x} + \frac{Q}{2}u_{2_x} - Qu_{1_x}\right)$$

$$u_{1_x} = -u_1$$
$$u_{2_x} = u_2 \cos\theta$$

$$F_{T_x} = \rho Q(u_2 \cos\theta + u_1)$$

and in the y-direction it is symmetrical, so

$$F_{T_y} = 0$$

4 Calculate the **pressure force.**

The pressure force is zero as the pressure at both the inlet and the outlets to the control volume are atmospheric.

5 Calculate the **body** force

We are only considering the horizontal plane in which there are no body forces.

6 Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{R_x} = F_{T_x} - 0 - 0$$
$$= \rho Q(u_2 \cos\theta + u_1)$$

exerted on the fluid.

The force **on the blade** is the same magnitude but in the opposite direction

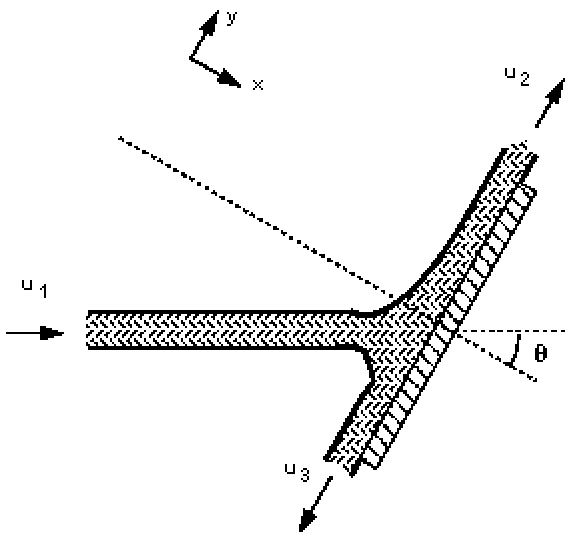
$$R = -F_{R_x}$$

So the blade moved in the x-direction.

In a real situation the blade is moving. The analysis can be extended to include this by including the amount of momentum entering the control volume over the time the blade remains there. This will be covered in the level 2 module next year.

3.7.6 Force due to a jet hitting an inclined plane

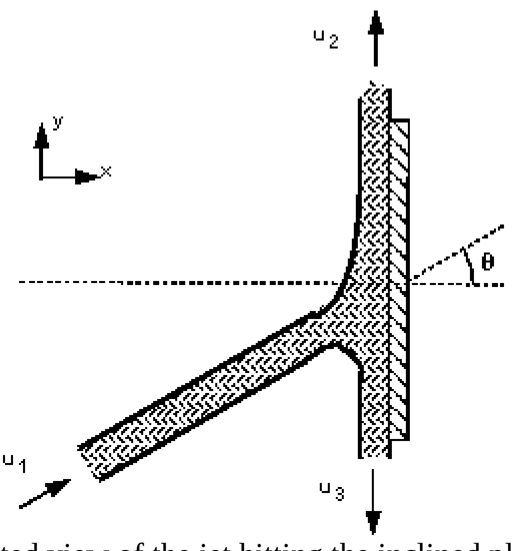
We have seen above the forces involved when a jet hits a plane at right angles. If the plane is tilted to an angle the analysis becomes a little more involved. This is demonstrated below.



A jet hitting an inclined plane.

(Note that for simplicity gravity and friction will be neglected from this analysis.)

We want to find the reaction force normal to the plate so we choose the axis system as above so that is normal to the plane. The diagram may be rotated to align it with these axes and help comprehension, as shown below



Rotated view of the jet hitting the inclined plane.

We do not know the velocities of flow in each direction. To find these we can apply Bernoulli equation

$$\frac{p_1}{\rho g}+\frac{u_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{u_2^2}{2g}+z_2=\frac{p_3}{\rho g}+\frac{u_3^2}{2g}+z_3$$

The height differences are negligible i.e. $z_1 = z_2 = z_3$ and the pressures are all atmospheric = 0. So

. $u_1 = u_2 = u_3 = u$

By continuity

$$Q_1 = Q_2 + Q_3$$

$$u_1 A_1 = u_2 A_2 + u_3 A_3$$

so

$$A_1 = A_2 + A_3$$

$$Q_1 = A_1 u$$

$$Q_2 = A_2 u$$

$$Q_3 = (A_1 - A_2)u$$

Using this we can calculate the forces in the same way as before.

1. Calculate the **total** force
2. Calculate the **pressure** force
3. Calculate the **body** force
4. Calculate the **resultant** force

1 Calculate the **total** force in the x-direction.

Remember that the co-ordinate system is normal to the plate.

$$F_{T\,x}=\rho\Big((Q_2 u_{2_x}+Q_3 u_{3_x})-Q_1 u_{1_x}\Big)$$

but $u_{2_x} = u_{3_x} = 0$ as the jets are parallel to the plate with no component in the x-direction.

$u_{1_x} = u_1 \cos\theta$, so

$$F_{T\,x} = -\rho Q_1 u_1 \cos\theta$$

2. Calculate the **pressure** force

All zero as the pressure is everywhere atmospheric.

3. Calculate the **body** force

As the control volume is small, hence the weight of fluid is small, we can ignore the body forces.

4. Calculate the **resultant** force

$$F_{T_x} = F_{R_x} + F_{P_x} + F_{B_x}$$
$$F_{R_x} = F_{T_x} - 0 - 0$$
$$= -\rho Q_1 u_1 \cos\theta$$

exerted **on the fluid**.

The force **on the plate** is the same magnitude but in the opposite direction

$$R = -F_{R_x}$$
$$= \rho Q_1 u_1 \cos\theta$$

We can find out how much discharge goes along in each direction on the plate. Along the plate, in the y-direction, the total force must be zero, $F_{T_y} = 0$.

Also in the y-direction: $u_{1_y} = u_1 \sin\theta,\ u_{2_y} = u_2,\ u_{3_y} = -u_3$, so

$$F_{T_y} = \rho\left((Q_2 u_{2_y} + Q_3 u_{3_y}) - Q_1 u_{1_y}\right)$$
$$F_{T_y} = \rho\left(Q_2 u_2 - Q_3 u_3 - Q_1 u_1 \sin\theta\right)$$

As forces parallel to the plate are zero,

$$0 = \rho A_2 u_2^2 - \rho A_3 u_3^2 - \rho A_1 u_1^2 \sin\theta$$

From above $u_1 = u_2 = u_3$

$$0 = A_2 - A_3 - A_1 \sin\theta$$

and from above we have $A_1 = A_2 + A_3$ so

$$0 = A_2 - A_3 - \left(A_2 + A_3\right)\sin\theta$$
$$= A_2(1 - \sin\theta) - A_3(1 + \sin\theta)$$
$$A_2 = A_3\left(\frac{1 + \sin\theta}{1 - \sin\theta}\right)$$

as $u_2 = u_3 = u$

$$Q_2 = Q_3\left(\frac{1 + \sin\theta}{1 - \sin\theta}\right)$$

$$Q_1 = Q_3\left(\frac{1 + \sin\theta}{1 - \sin\theta}\right) + Q_3$$
$$= Q_3\left(1 + \frac{1 + \sin\theta}{1 - \sin\theta}\right)$$

So we know the discharge in each direction

4. 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:

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

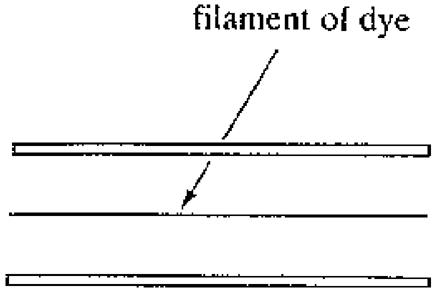
where the constant of proportionality, μ, is known as the coefficient of viscosity (or simply viscosity). We saw that for some fluids - sometimes known as exotic fluids - the value of μ changes with stress or velocity gradient. We shall only deal with Newtonian fluids.

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.

4.1 Laminar and turbulent flow

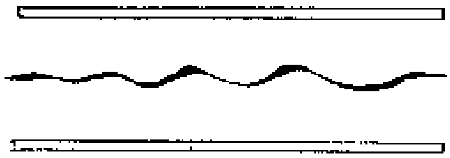
If we were to take a pipe of free flowing water and inject a dye into the middle of the stream, what would we expect to happen?

This



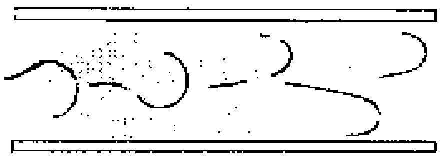
Laminar (viscous)

this



Transitional

or this



Turbulent

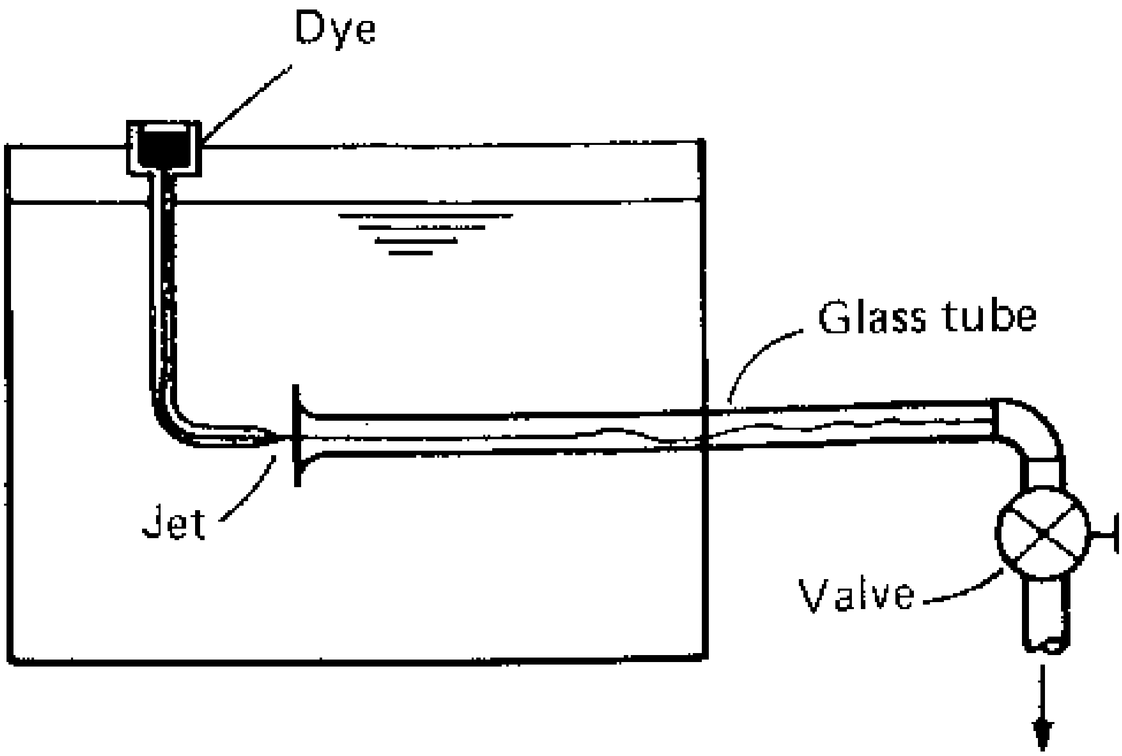
Actually both would happen - but for different flow rates. The top occurs when the fluid is flowing fast and the lower when it is flowing slowly.

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.

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$
$$= \frac{\text{inertial forces}}{\text{viscous forces}}$$

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.

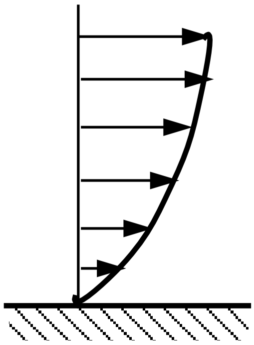
Turbulent flow

- 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.

4.2 Pressure loss due to friction in a pipeline.

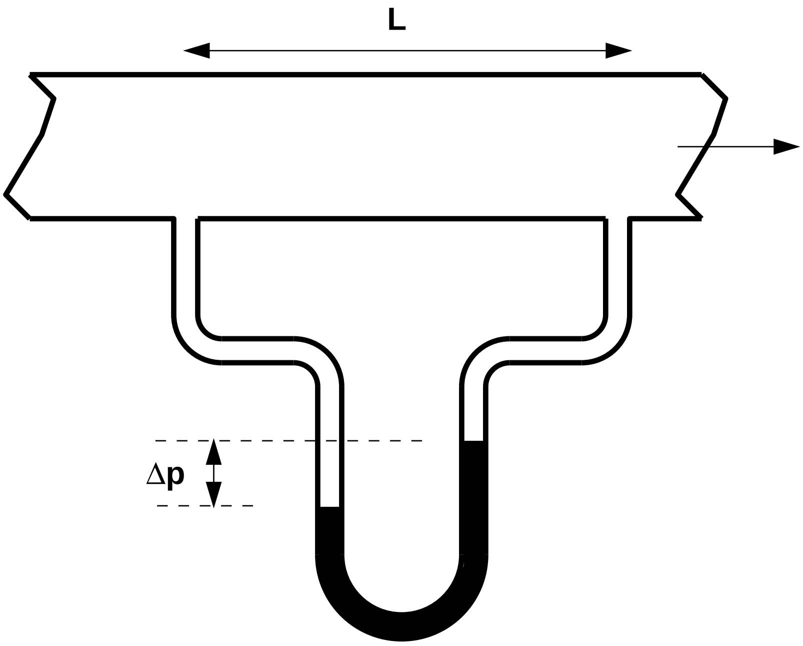
Up to this point on the course we have considered ideal fluids where there have been no losses due to friction or any other factors. In reality, because fluids are viscous, energy is lost by flowing fluids due to friction which must be taken into account. The effect of the friction shows itself as a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



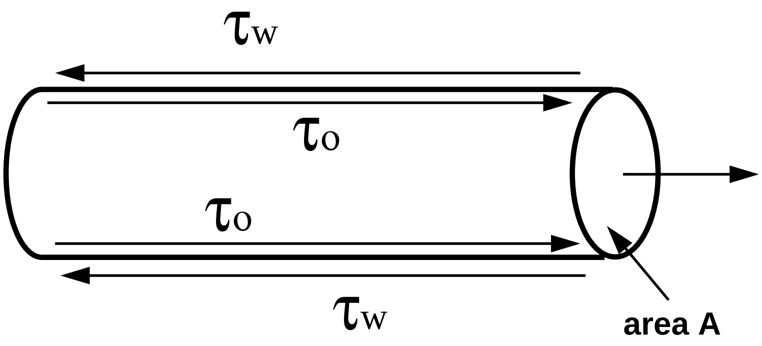
If a manometer is attached as the pressure (head) difference due to the energy lost by the fluid overcoming the shear stress can be easily seen.

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 Δp to (p-Δp).

The driving force due to pressure (F = Pressure x Area) can then be written

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

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

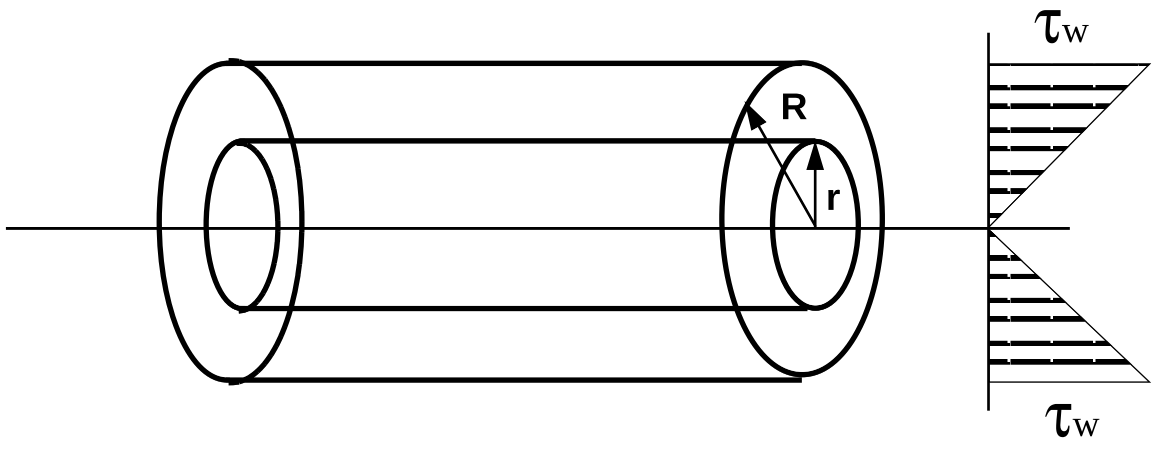
As the flow is in equilibrium,

driving force = retarding force

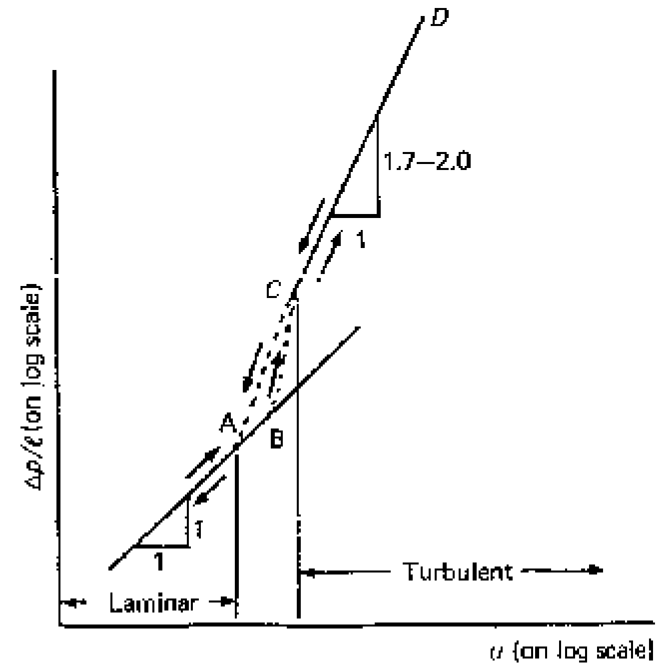
$$\Delta p \frac{\pi d^2}{4} = \tau_w \pi d L$$

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

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 $\Delta \propto$

turbulent or

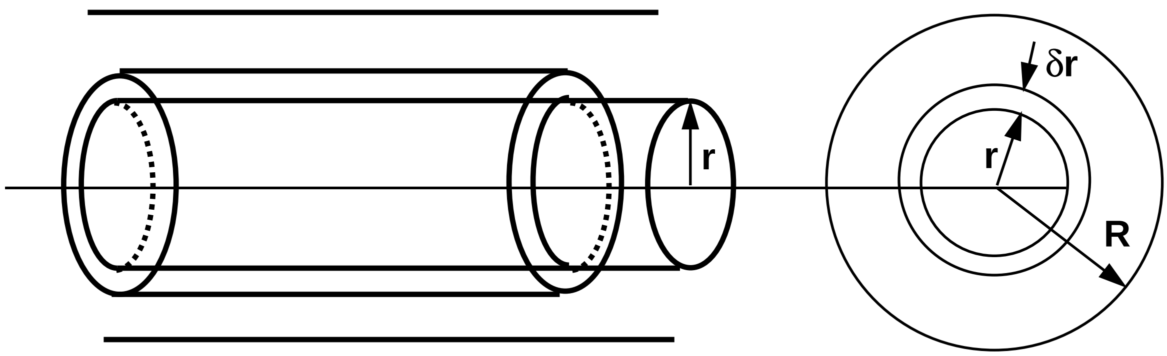
τ_w on a particular fluid. If we knew τ_w we could then use it to give a general equation to predict the pressure loss.

4.3 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.

$p \quad u$

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

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

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

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

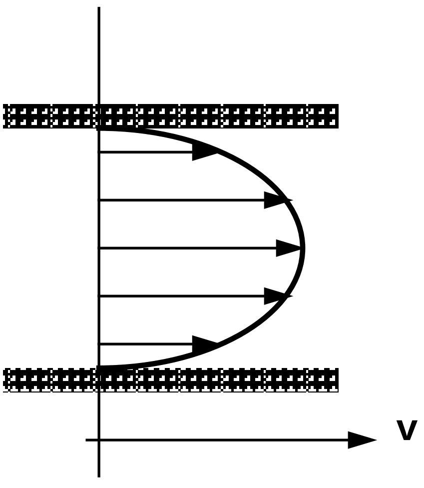
At $r = 0$, (the centre of the pipe), $u = u_{max}$, at $r = R$ (the pipe wall) $u = 0$, giving

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

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?

$$Q = u_m A$$

$$u_m \qquad \overset{R}{} \; u_r \; dr$$

$$\frac{p}{L}\frac{}{\mu} \overset{R}{} \left(R \qquad r\;\right) dr$$

$$\frac{p}{L}\frac{R}{\mu} \qquad \frac{p\,d}{\mu L}$$

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. $\quad p = \rho g h_f$

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

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

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

It has been validated many time by experiment.

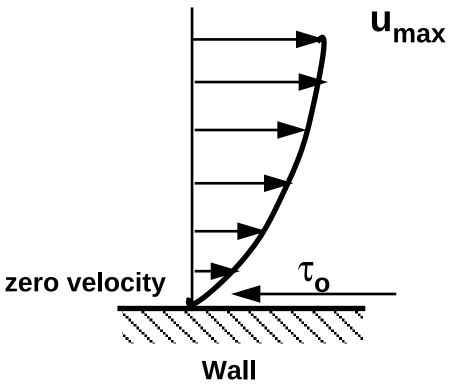
It justifies two assumptions:

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

4.4 Boundary Layers

(Recommended extra reading for this section: Fluid Mechanics by Douglas J F, Gasiorek J M, and Swaffield J A. Longman publishers. Pages 327-332.)

When a fluid flows over a stationary surface, e.g. the bed of a river, or the wall of a pipe, the fluid touching the surface is brought to rest by the shear stress τ_o at the wall. The velocity increases from the wall to a maximum in the main stream of the flow.



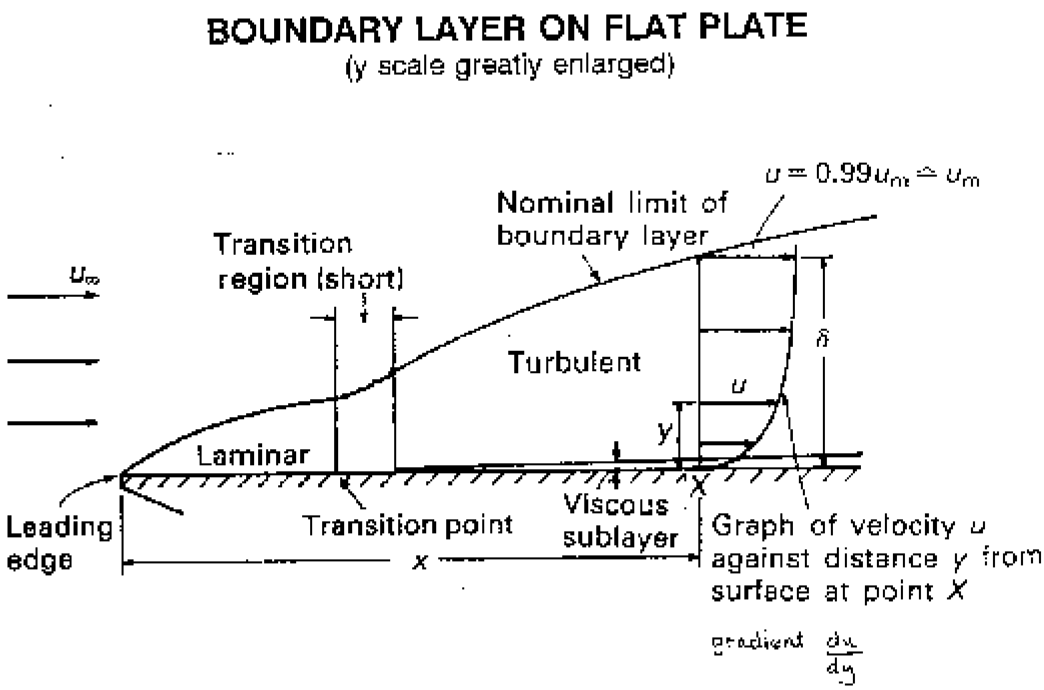
Looking at this two-dimensionally we get the above velocity profile from the wall to the centre of the flow.

This profile doesn't just exit, it must build up gradually from the point where the fluid starts to flow past the surface - e.g. when it enters a pipe.

If we consider a flat plate in the middle of a fluid, we will look at the build up of the velocity profile as the fluid moves over the plate.

Upstream the velocity profile is uniform, (free stream flow) a long way downstream we have the velocity profile we have talked about above. This is the known as **fully developed flow**. But how do we get to that state?

This region, where there is a velocity profile in the flow due to the shear stress at the wall, we call the **boundary layer**. The stages of the formation of the boundary layer are shown in the figure below:



We define the thickness of this boundary layer as the distance from the wall to the point where the velocity is 99% of the “free stream” velocity, the velocity in the middle of the pipe or river.

boundary layer thickness, δ = distance from wall to point where $u = 0.99\ u_{mainstream}$

The value of δ will increase with distance from the point where the fluid first starts to pass over the boundary - the flat plate in our example. It increases to a maximum in fully developed flow.

Correspondingly, the drag force D on the fluid due to shear stress τ_o at the wall increases from zero at the start of the plate to a maximum in the fully developed flow region where it remains constant. We can calculate the magnitude of the drag force by using the momentum equation. But this complex and not necessary for this course.

Our interest in the boundary layer is that its presence greatly affects the flow through or round an object. So here we will examine some of the phenomena associated with the boundary layer and discuss why these occur.

4.4.1 Formation of the boundary layer

Above we noted that the boundary layer grows from zero when a fluid starts to flow over a solid surface. As is passes over a greater length more fluid is slowed by friction between the fluid layers close to the boundary. Hence the thickness of the slower layer increases.

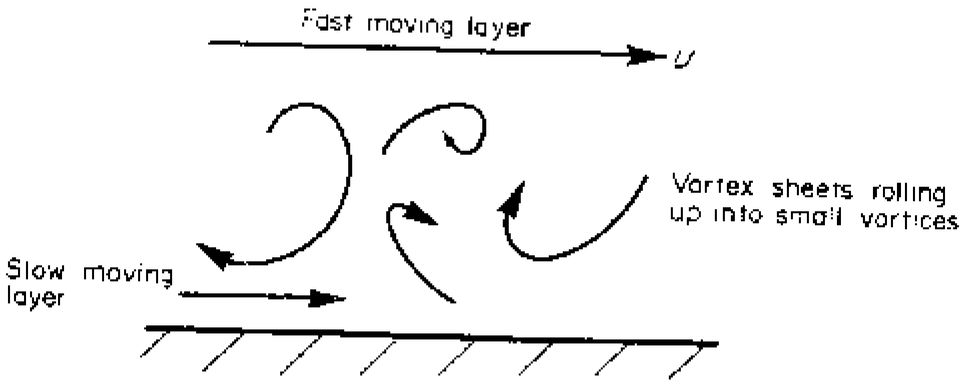
The fluid near the top of the boundary layer is dragging the fluid nearer to the solid surface along. The mechanism for this dragging may be one of two types:

The first type occurs when the normal viscous forces (the forces which hold the fluid together) are large enough to exert drag effects on the slower moving fluid close to the solid boundary. If the boundary layer is thin then the velocity gradient normal to the surface, *(du/dy)*, is large so by Newton’s law of viscosity the shear stress, $\tau = \mu\,(du/dy)$, is also large. The corresponding force may then be large enough to exert drag on the fluid close to the surface.

As the boundary layer thickness becomes greater, so the velocity gradient become smaller and the shear stress decreases until it is no longer enough to drag the slow fluid near the surface along. If this viscous force was the only action then the fluid would come to a rest.

It, of course, does not come to rest but the second mechanism comes into play. Up to this point the flow has been **laminar** and Newton’s law of viscosity has applied. This part of the boundary layer is known as the **laminar boundary layer**

The viscous shear stresses have held the fluid particles in a constant motion within layers. They become small as the boundary layer increases in thickness and the velocity gradient gets smaller. Eventually they are no longer able to hold the flow in layers and the fluid starts to rotate.



This causes the fluid motion to rapidly becomes turbulent. Fluid from the fast moving region moves to the slower zone transferring momentum and thus maintaining the fluid by the wall in motion. Conversely, slow moving fluid moves to the faster moving region slowing it down. The net effect is an increase in momentum in the boundary layer. We call the part of the boundary layer the **turbulent boundary layer**.

At points very close to the boundary the velocity gradients become very large and the velocity gradients become very large with the viscous shear forces again becoming large enough to maintain the fluid in laminar motion. This region is known as the **laminar sub-layer**. This layer occurs within the turbulent zone and is next to the wall and very thin – a few hundredths of a mm.

4.4.2 Surface roughness effect

Despite its thinness, the laminar sub-layer can play a vital role in the friction characteristics of the surface.

This is particularly relevant when defining pipe friction - as will be seen in more detail in the level 2 module. In **turbulent** flow if the height of the roughness of a pipe is greater than the thickness of the laminar sub-layer then this increases the amount of turbulence and energy losses in the flow. If the height of roughness is less than the thickness of the laminar sub-layer the pipe is said to be smooth and it has little effect on the boundary layer.

In **laminar** flow the height of roughness has very little effect

4.4.3 Boundary layers in pipes

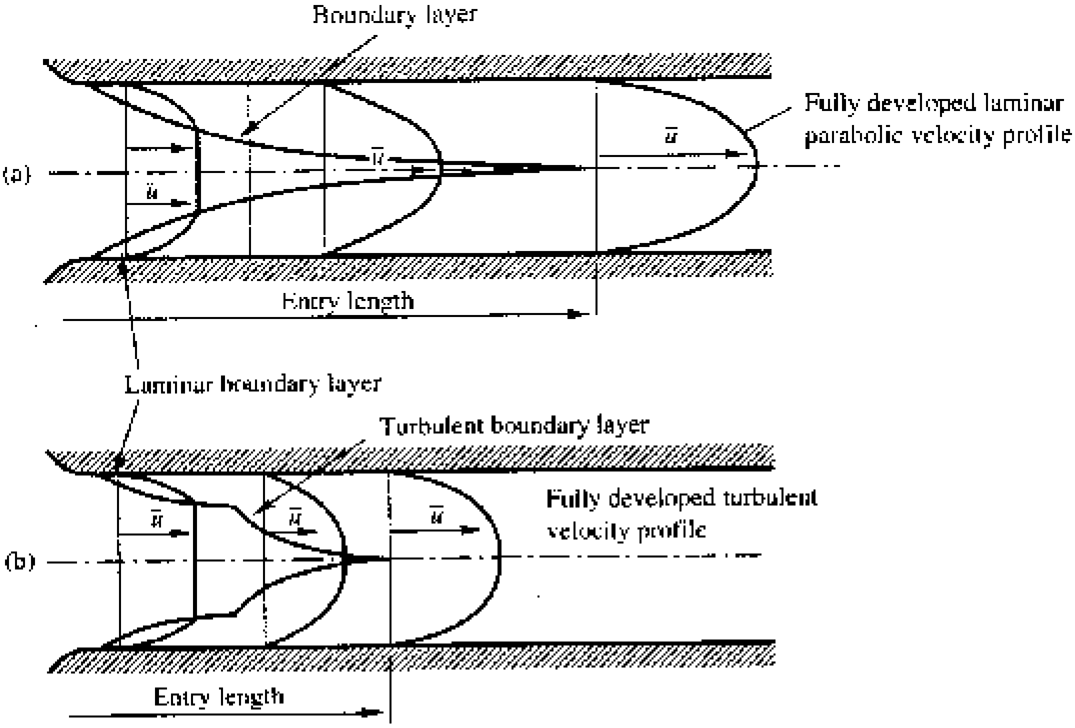
As flow enters a pipe the boundary layer will initially be of the laminar form. This will change depending on the ration of inertial and viscous forces; i.e. whether we have laminar (viscous forces high) or turbulent flow (inertial forces high).

From earlier we saw how we could calculate whether a particular flow in a pipe is laminar or turbulent using the Reynolds number.

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

(ρ = *density* u = *velocity* μ = *viscosity* d = *pipe diameter*)

Laminar flow:	$\mathrm{Re} < 2000$
Transitional flow:	$2000 < \mathrm{Re} < 4000$
Turbulent flow:	$\mathrm{Re} > 4000$



If we only have laminar flow the profile is parabolic – as proved in earlier lectures – as only the first part of the boundary layer growth diagram is used. So we get the top diagram in the above figure.

If turbulent (or transitional), both the laminar and the turbulent (transitional) zones of the boundary layer growth diagram are used. The growth of the velocity profile is thus like the bottom diagram in the above figure.

Once the boundary layer has reached the centre of the pipe the flow is said to be **fully developed**. (Note that at this point the whole of the fluid is now affected by the boundary friction.)

The length of pipe before fully developed flow is achieved is different for the two types of flow. The length is known as the **entry length**.

Laminar flow entry length $\approx 120 \times$ diameter

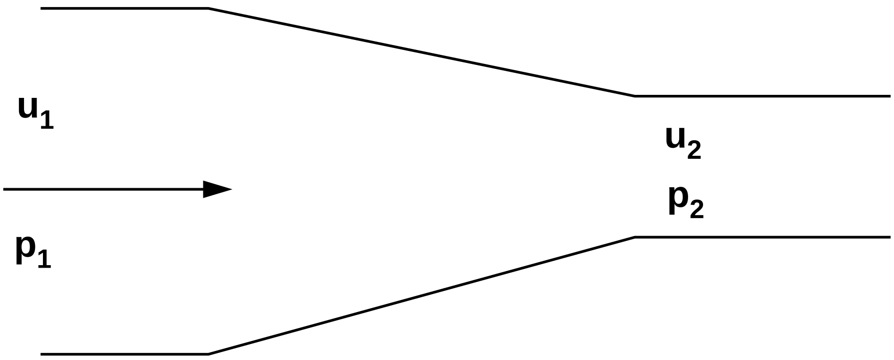
Turbulent flow entry length $\approx 60 \times$ diameter

4.4.4 Boundary layer separation

4.4.4.1 Convergent flows: Negative pressure gradients

If flow over a boundary occurs when there is a pressure decrease in the direction of flow, the fluid will accelerate and the boundary layer will become thinner.

This is the case for *convergent* flows.

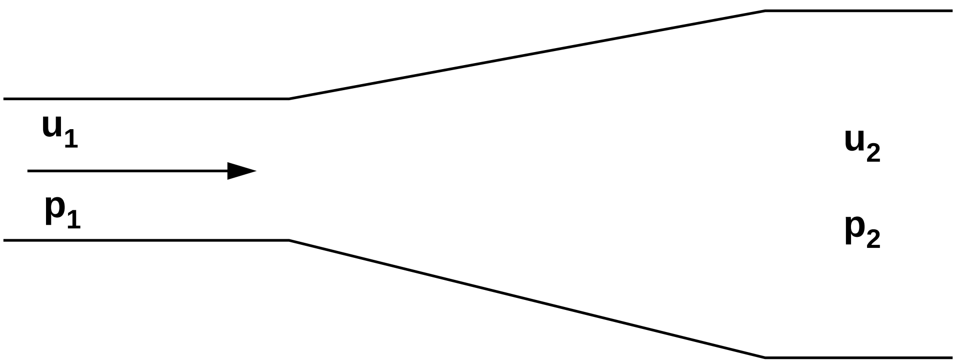


$p_1 > p_2 \qquad u_1 < u_2$

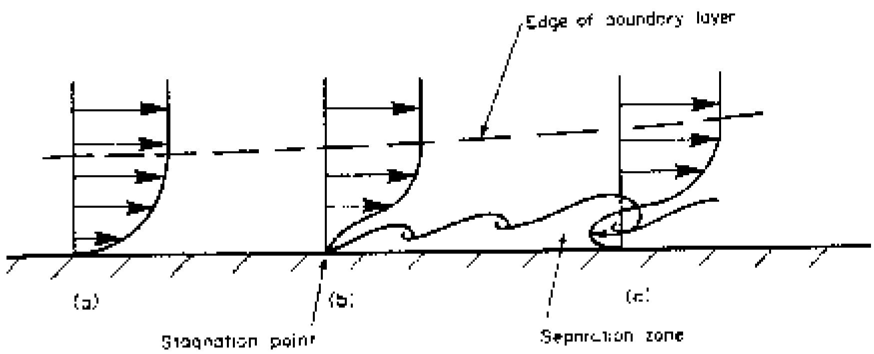
The accelerating fluid maintains the fluid close to the wall in motion. Hence the flow remains stable and turbulence reduces. Boundary layer separation does not occur.

4.4.4.2 Divergent flows: Positive pressure gradients

When the pressure increases in the direction of flow the situation is very different. Fluid outside the boundary layer has enough momentum to overcome this pressure which is trying to push it backwards. The fluid within the boundary layer has so little momentum that it will very quickly be brought to rest, and possibly reversed in direction. If this reversal occurs it lifts the boundary layer away from the surface as shown below.

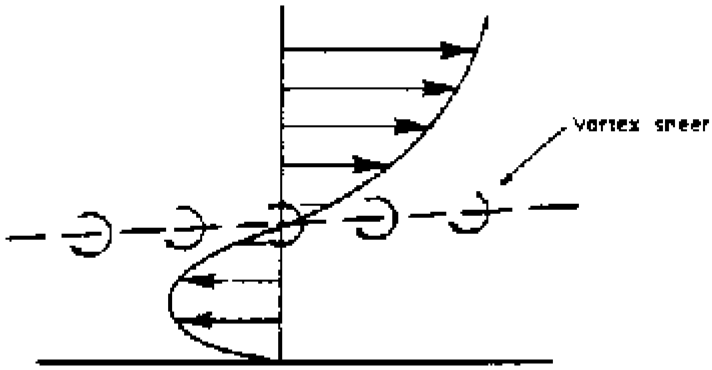


$p_1 < p_2 \qquad u_1 > u_2$



This phenomenon is known as **<u>boundary layer separation</u>**.

At the edge of the separated boundary layer, where the velocities change direction, a line of vortices occur (known as a vortex sheet). This happens because fluid to either side is moving in the opposite direction.

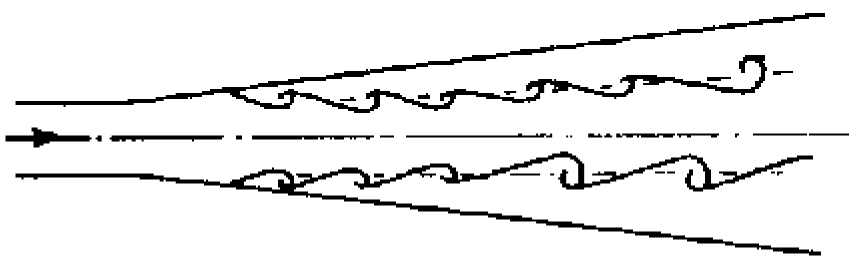


This boundary layer separation and increase in the turbulence because of the vortices results in very large energy losses in the flow.

These separating / divergent flows are inherently unstable and far more energy is lost than in parallel or convergent flow.

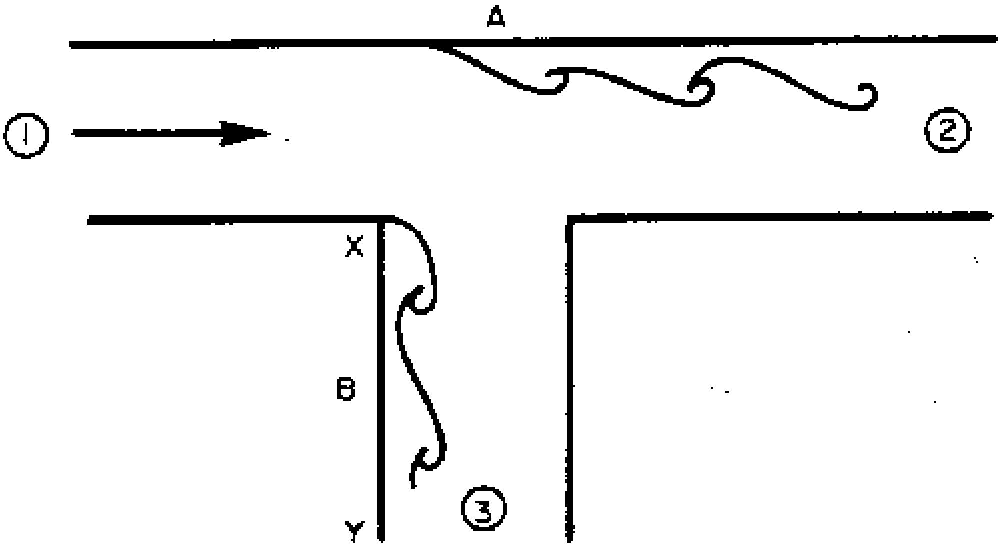
4.4.4.3 A divergent duct or diffuser

The increasing area of flow causes a velocity drop (according to continuity) and hence a pressure rise (according to the Bernoulli equation).



Increasing the angle of the diffuser increases the probability of boundary layer separation. In a Venturi meter it has been found that an angle of about 6° provides the optimum balance between length of meter and danger of boundary layer separation which would cause unacceptable pressure energy losses.

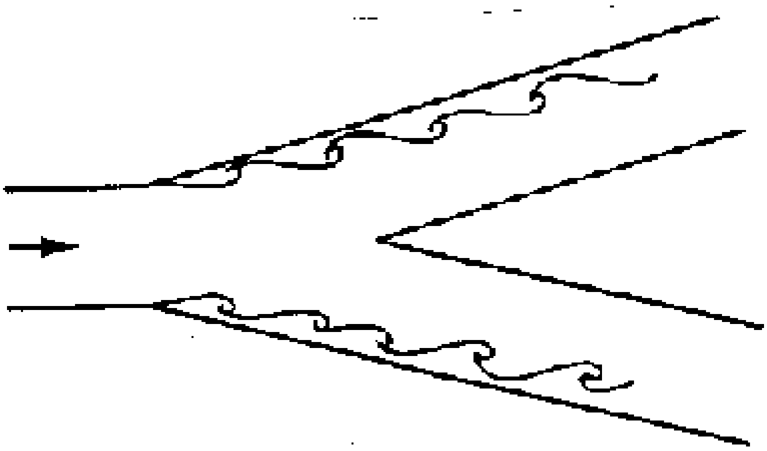
4.4.4.4 Tee-Junctions



Assuming equal sized pipes, as fluid is removed, the velocities at 2 and 3 are smaller than at 1, the entrance to the tee. Thus the pressure at 2 and 3 are higher than at 1. These two adverse pressure gradients can cause the two separations shown in the diagram above.

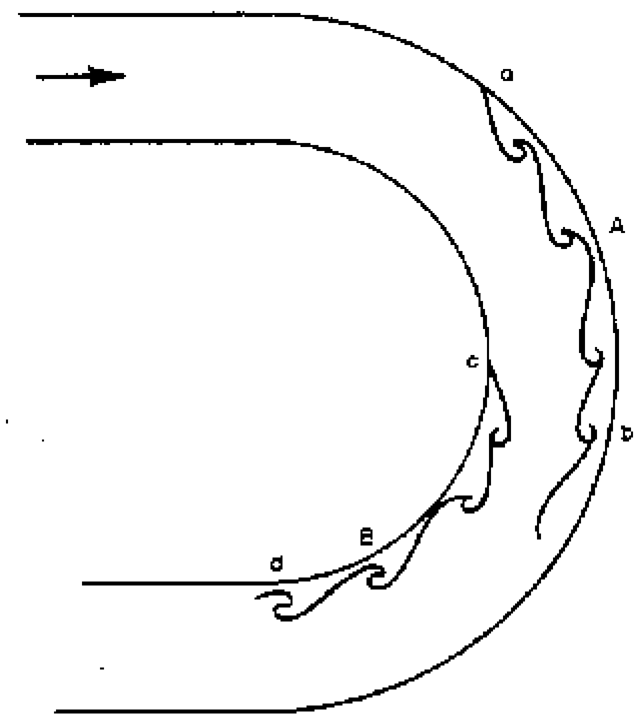
4.4.4.5 Y-Junctions

Tee junctions are special cases of the Y-junction with similar separation zones occurring. See the diagram below.



Downstream, away from the junction, the boundary layer reattaches and normal flow occurs i.e. the effect of the boundary layer separation is only local. Nevertheless fluid downstream of the junction will have lost energy.

4.4.4.6 Bends

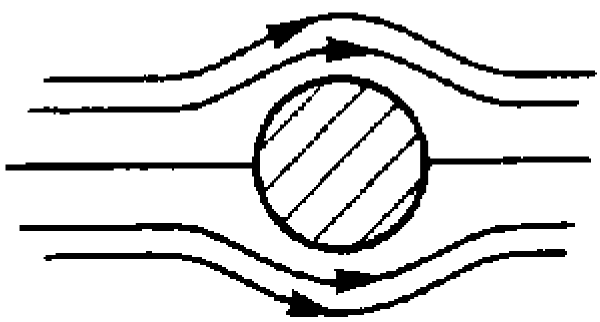


Two separation zones occur in bends as shown above. The pressure at b must be greater than at a as it must provide the required radial acceleration for the fluid to get round the bend. There is thus an adverse pressure gradient between a and b so separation may occur here.

Pressure at c is less than at the entrance to the bend but pressure at d has returned to near the entrance value - again this adverse pressure gradient may cause boundary layer separation.

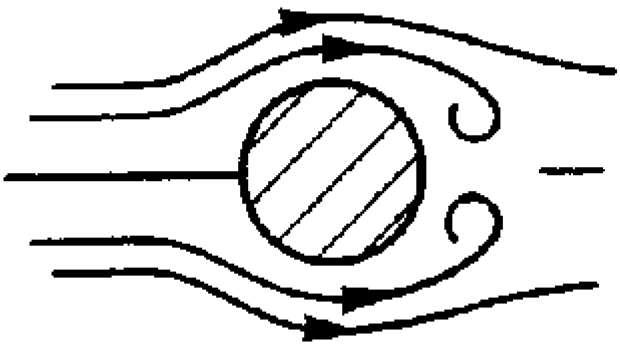
4.4.4.7 Flow past a cylinder

The pattern of flow around a cylinder varies with the velocity of flow. If flow is very slow with the Reynolds number (ρ v diameter/μ) less than 0.5, then there is no separation of the boundary layers as the pressure difference around the cylinder is very small. The pattern is something like that in the figure below.

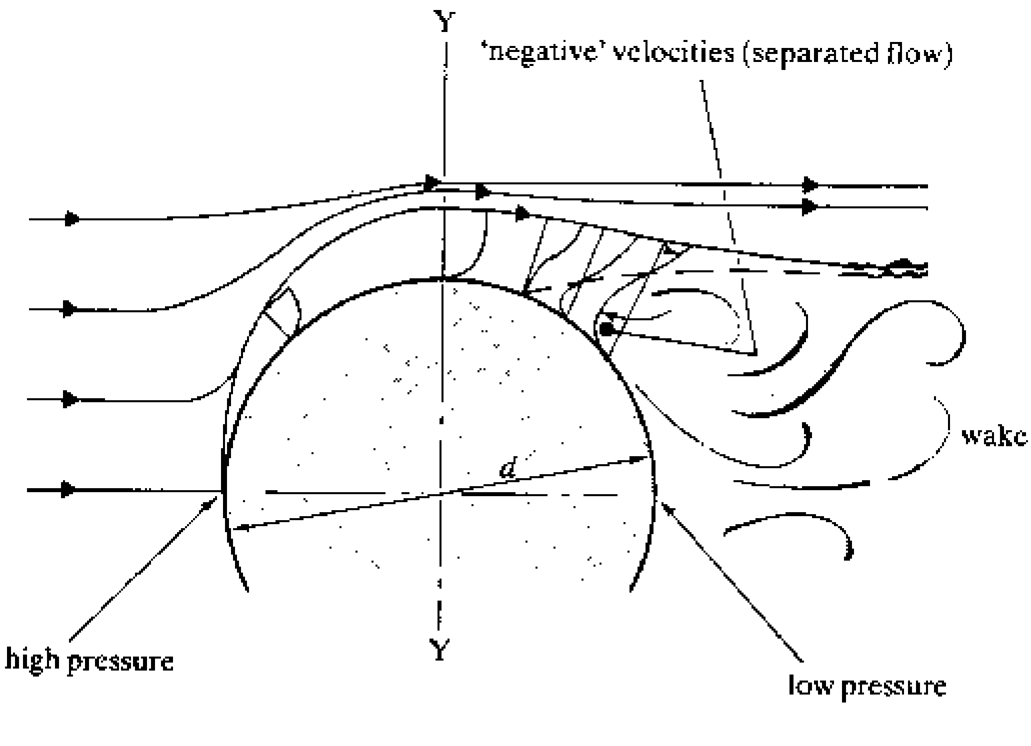


$Re < 0{\cdot}5$

If 2 < Re < 70 then the boundary layers separate symmetrically on either side of the cylinder. The ends of these separated zones remain attached to the cylinder, as shown below.



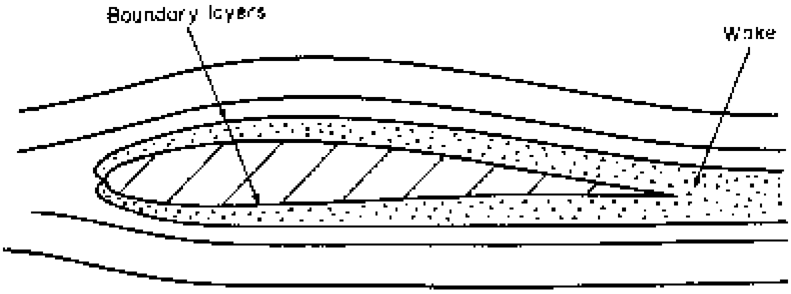
Above a Re of 70 the ends of the separated zones curl up into vortices and detach alternately from each side forming a trail of vortices on the down stream side of the cylinder. This trial in known as a **Karman vortex trail** or **street**. This vortex trail can easily be seen in a river by looking over a bridge where there is a pier to see the line of vortices flowing away from the bridge. The phenomenon is responsible for the whistling of hanging telephone or power cables. A more significant event was the famous failure of the Tacoma narrows bridge. Here the frequency of the alternate vortex shedding matched the natural frequency of the bridge deck and resonance amplified the vibrations until the bridge collapsed. (The frequency of vortex shedding from a cylinder can be predicted. We will not try to predict it here but a derivation of the expression can be found in many fluid mechanics text books.)



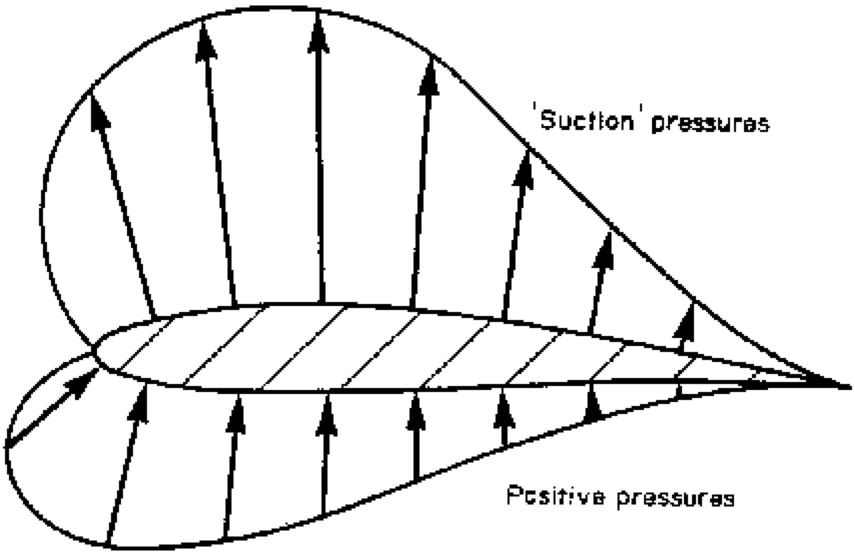
Looking at the figure above, the formation of the separation occurs as the fluid accelerates from the centre to get round the cylinder (it must accelerate as it has further to go than the surrounding fluid). It reaches a maximum at Y, where it also has also dropped in pressure. The adverse pressure gradient between here and the downstream side of the cylinder will cause the boundary layer separation if the flow is fast enough, (Re > 2.)

4.4.4.8 Aerofoil

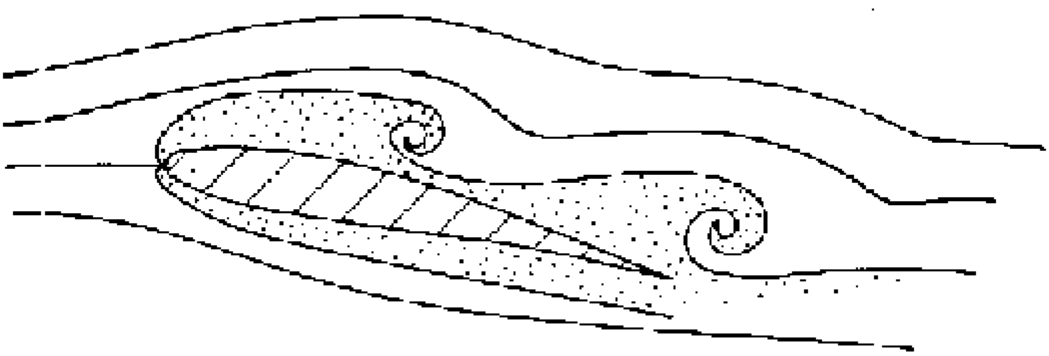
Normal flow over a aerofoil (a wing cross-section) is shown in the figure below with the boundary layers greatly exaggerated.



The velocity increases as air it flows over the wing. The pressure distribution is similar to that shown below so transverse lift force occurs.



If the angle of the wing becomes too great and boundary layer separation occurs on the top of the aerofoil the pressure pattern will change dramatically. This phenomenon is known as **stalling**.

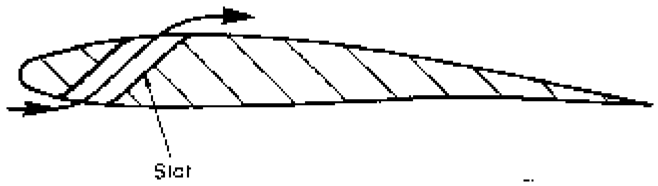


When stalling occurs, all, or most, of the 'suction' pressure is lost, and the plane will suddenly drop from the sky! The only solution to this is to put the plane into a dive to regain the boundary layer. A transverse lift force is then exerted on the wing which gives the pilot some control and allows the plane to be pulled out of the dive.

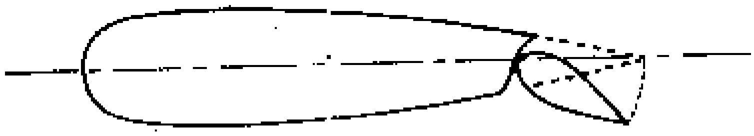
Fortunately there are some mechanisms for preventing stalling. They all rely on preventing the boundary layer from separating in the first place.

1 Arranging the engine intakes so that they draw slow air from the boundary layer at the rear of the wing though small holes helps to keep the boundary layer close to the wing. Greater pressure gradients can be maintained before separation take place.

2 Slower moving air on the upper surface can be increased in speed by bringing air from the high pressure area on the bottom of the wing through slots. Pressure will decrease on the top so the adverse pressure gradient which would cause the boundary layer separation reduces.



3 Putting a flap on the end of the wing and tilting it before separation occurs increases the velocity over the top of the wing, again reducing the pressure and chance of separation occurring.



5. 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.

5.1 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 <u>dimensions</u>.

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 <u>not</u> 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 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}

5.2 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^3s^{-1}. The units of the right hand side must be the same. Writing the equation with only the SI units gives

$$m^3 s^{-1} = m\left(m\,s^{-2}\right)^{1/2} m^{3/2}$$
$$= m^3 s^{-1}$$

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,

$$L^3 T^{-1} = L\left(L\,T^{-2}\right)^{1/2} L^{3/2}$$
$$= L^3 T^{-1}$$

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).

5.3 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:

diameter, d

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

fluid density, ρ

revolutions per second, N

fluid viscosity, μ

Before we do any analysis we can write this equation:

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

or

$$0 = \phi_1 (F, d, u, \rho, N, \mu)$$

where ϕ and ϕ_1 are unknown functions.

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

$$F = K\, d^m\, u^p\, \rho^\theta\, N^r\, \mu^\sigma$$

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.

5.4 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.

5.5 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

i. From the 2nd theorem there can be **n** (= 3) repeating variables.

ii. 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).

iii. A combination of the repeating variables must not form a dimensionless group.

iv. The repeating variables do not have to appear in all π groups.

v. 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.

5.6 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,

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

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 $1 = \left(kg\,m^{-3}\right)^{a_1} \left(m\,s^{-1}\right)^{b_1} (m)^{c_1}\, kg\,m\,s^{-2}$

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, $\pi_3 = \rho^{a_3} u^{b_3} d^{c_3} \mu$

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

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,

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

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

This may also be written:

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

5.6.1 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.

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

Then the following manipulations are permitted:

i. 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

$$\phi(\pi_{1a}, \pi_2, \pi_3 \ldots\ldots\ldots \pi_{m-n}) = 0$$

ii. The reciprocal of any dimensionless group is valid. So $\phi(\pi_1, 1/\pi_2, \pi_3 \ldots\ldots\ldots 1/\pi_{m-n}) = 0$ is valid.

iii. Any dimensionless group may be raised to any power. So $\phi((\pi_1)^2, (\pi_2)^{1/2}, (\pi_3)^3 \ldots\ldots\ldots \pi_{m-n}) = 0$ is valid.

iv. Any dimensionless group may be multiplied by a constant.

v. Any group may be expressed as a function of the other groups, e.g.

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

In general the defining equation could look like

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

5.8 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	$\mathrm{Re} = \frac{\rho ud}{\mu}$	inertial, viscous force ratio

Euler number	$\mathrm{En} = \frac{p}{\rho u^2}$	pressure, inertial force ratio
Froude number	$\mathrm{Fn} = \frac{u^2}{gd}$	inertial, gravitational force ratio
Weber number	$\mathrm{We} = \frac{\rho u d}{\sigma}$	inertial, surface tension force ratio
Mach number	$\mathrm{Mn} = \frac{u}{c}$	Local velocity, local velocity of sound ratio

5.9 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 variables

ρ:	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:

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

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 $ML^{-1}T^{-2}$)

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

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

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

5.10 Similarity

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.

5.10.1 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.

5.10.2 Kinematic similarity

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

i. If the paths of moving particles are geometrically similar
ii. 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.

5.10.3 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 $$\frac{F_{\mathrm{m}}}{F_{\mathrm{p}}} = \frac{M_m a_m}{M_p a_p} = \frac{\rho_m L_m^3}{\rho_p L_p^3} \times \frac{\lambda_L}{\lambda_T^2} = \lambda_\rho \lambda_L^2 \left(\frac{\lambda_L}{\lambda_T}\right)^2 = \lambda_\rho \lambda_L^2 \lambda_u^2$$

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

5.11 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 is not possible. Measurements can be taken from the model and a suitable scaling law applied to predict the values in the prototype.

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:

ρ: ML^{-3} $\quad$ *u:* LT^{-1} $\quad$ *l:(length)* L $\quad$ μ: $ML^{-1}T^{-1}$

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

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

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

Leading to π_1 as

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

Notice how $1/\pi_2$ 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

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

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.

5.11.1 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 $\rho_m = \rho_p$ so

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

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 ρ and μ ratios need to be used in the above equation.

Example 2

A model aeroplane is built at 1/10 scale and is to be tested in a wind tunnel operating at a pressure of 20 times atmospheric. The aeroplane will fly at 500km/h. At what speed should the wind tunnel operate to give dynamic similarity between the model and prototype? If the drag measure on the model is 337.5 N what will be the drag on the plane?

From earlier we derived the equation for resistance on a body moving through air:

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

For dynamic similarity $\mathrm{Re}_m = \mathrm{Re}_p$, so

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

The value of μ does not change much with pressure so $\mu_m = \mu_p$

The equation of state for an ideal gas is $p = \rho RT$. As temperature is the same then the density of the air in the model can be obtained from

$$\frac{p_m}{p_p} = \frac{\rho_m RT}{\rho_p RT} = \frac{\rho_m}{\rho_p}$$

$$\frac{20 p_p}{p_p} = \frac{\rho_m}{\rho_p}$$

$$\rho_m = 20\rho_p$$

So the model velocity is found to be

$$u_m = u_p \frac{1}{20} \frac{1}{1/10} = 0.5 u_p$$

$$u_m = 250\, km/h$$

The ratio of forces is found from

$$\frac{R_m}{R_p} = \frac{\left(\rho u^2 l^2\right)_m}{\left(\rho u^2 l^2\right)_p}$$

$$\frac{R_m}{R_p} = \frac{20}{1} \frac{(0.5)^2}{1} \frac{(0.1)^2}{1} = 0.05$$

So the drag force on the prototype will be

$$R_p = \frac{1}{0.05} R_m = 20 \times 337.5 = 6750 N$$

5.11.2 Models with free surfaces - rivers, estuaries etc.

When modelling rivers and other fluid with free surfaces the effect of gravity becomes important and the major governing non-dimensional number becomes the Froude (Fn) number. The resistance to motion formula above would then be derived with g as an extra dependent variables to give an extra π group. So the defining equation is:

$$\phi\,(R, \rho, u, l, \mu, g\,) = 0$$

From which dimensional analysis gives:

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

$$R = \rho u^2 l^2 \phi(\mathrm{Re}, \mathrm{Fn})$$

Generally the prototype will have a very large Reynolds number, in which case slight variation in Re causes little effect on the behaviour of the problem. Unfortunately models are sometimes so small and the Reynolds numbers are large and the viscous effects take effect. This situation should be avoided to achieve correct results. Solutions to this problem would be to increase the size of the model - or more difficult - to change the fluid (i.e. change the viscosity of the fluid) to reduce the Re.

5.11.3 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:

i. accurate depths and depth changes become very difficult to measure;

ii. the bed roughness of the channel becomes impracticably small;

iii. 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.