2. Bernoulli's Equation

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

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

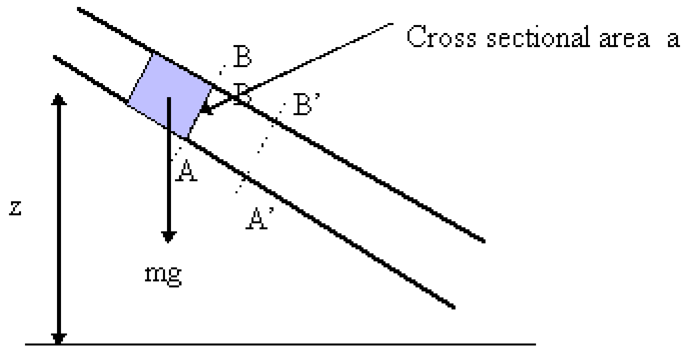
We see that from applying equal pressure or zero velocities we get the two equations from the section above. They are both just special cases of Bernoulli's equation.

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

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

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

The derivation of Bernoulli's Equation:



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

potential energy = mgz

---

potential energy per unit weight = z

---

kinetic energy = $\frac{1}{2}mu^2$

---

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

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

force on AB = pa

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

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

therefore

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

work done = force distance AA'

= $pa \times \frac{m}{\rho a} = \frac{pm}{\rho}$

---

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

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

Summing all of these energy terms gives

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

or

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

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

pressure head = $\frac{p}{\rho g}$

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

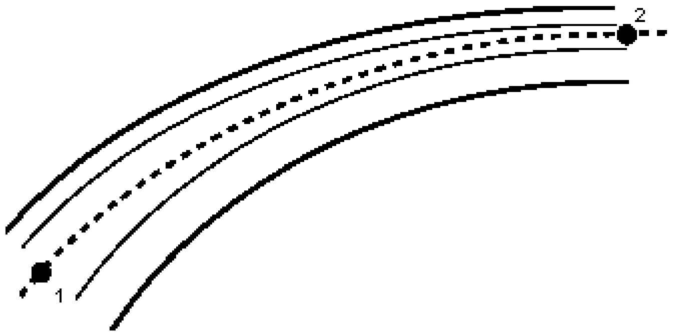
potential head = z

total head = H

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

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

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



Two points joined by a streamline

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

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:

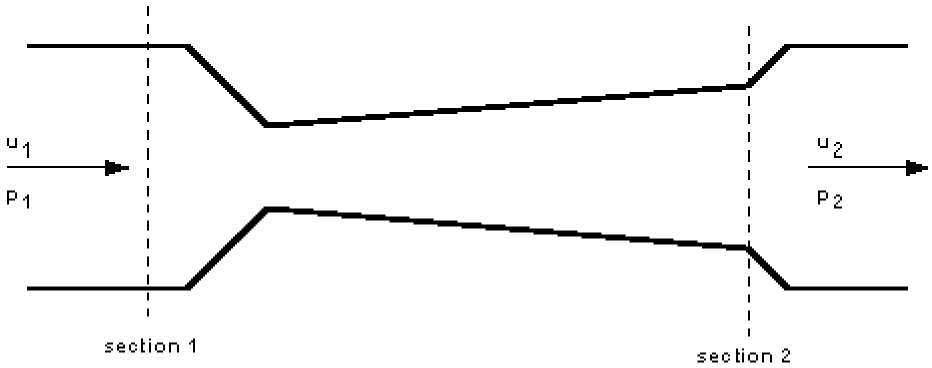
$$\begin{matrix}\text{Total} & & \text{Total} & & \text{Loss} & & \text{Work done} & & \text{Energy} \\ \text{energy per} & = & \text{energy per unit} & + & \text{per unit} & + & \text{per unit} & - & \text{supplied} \\ \text{unit weight at 1} & & \text{weight at 2} & & \text{weight} & & \text{weight} & & \text{per unit weight}\end{matrix}$$

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

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

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

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



A contracting expanding pipe

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

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

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

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

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

$$\begin{aligned} A_1 u_1 &= A_2 u_2 \\ u_2 &= \frac{A_1 u_1}{A_2} \\ u_2 &= \left(\frac{d_1}{d_2}\right)^2 u_1 \\ &= 7.8125 m/s \end{aligned}$$

So we can now calculate the pressure at section 2

$$\begin{aligned} p_2 &= 200000 - 17296.87 \\ &= 182703\ N/m^2 \\ &= 182.7\ kN/m^2 \end{aligned}$$

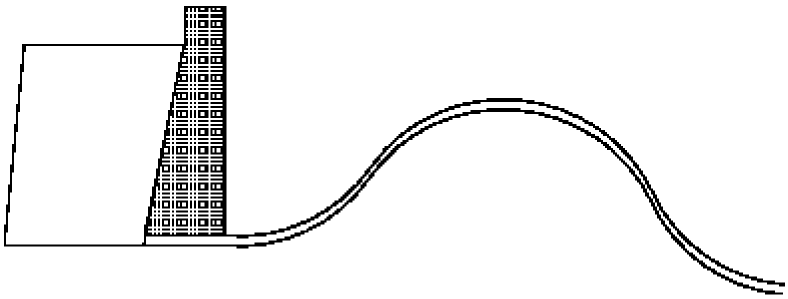
Notice how the velocity has increased while the pressure has decreased. The phenomenon - that pressure decreases as velocity increases - sometimes comes in very useful in engineering. (It is on this principle that carburettor in many car engines work - pressure reduces in a contraction allowing a small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

4. 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.



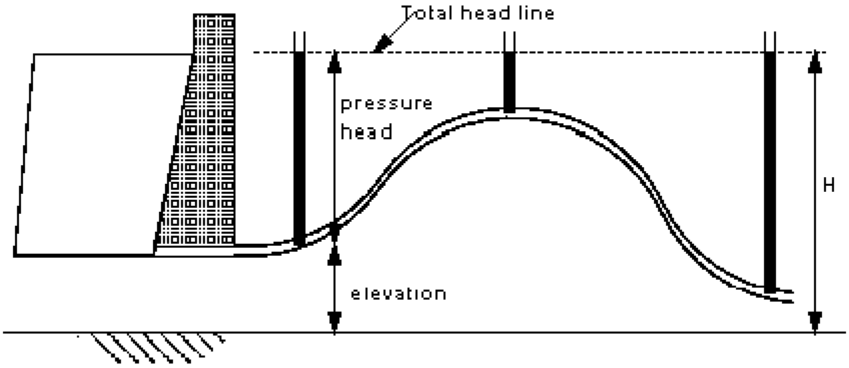
Reservoir feeding a pipe

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).



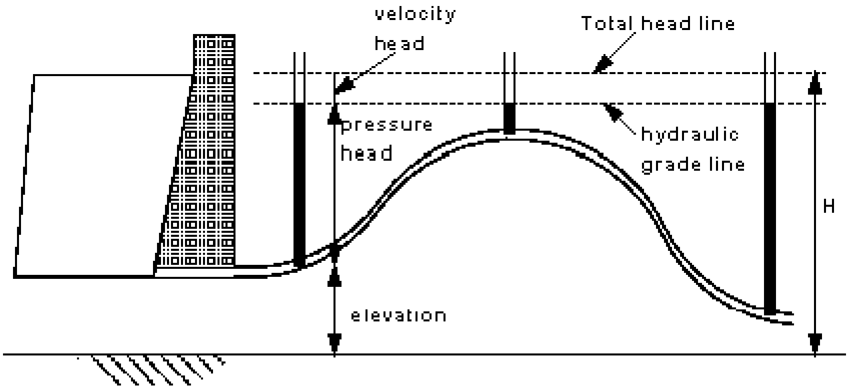
Piezometer levels with zero velocity

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 …

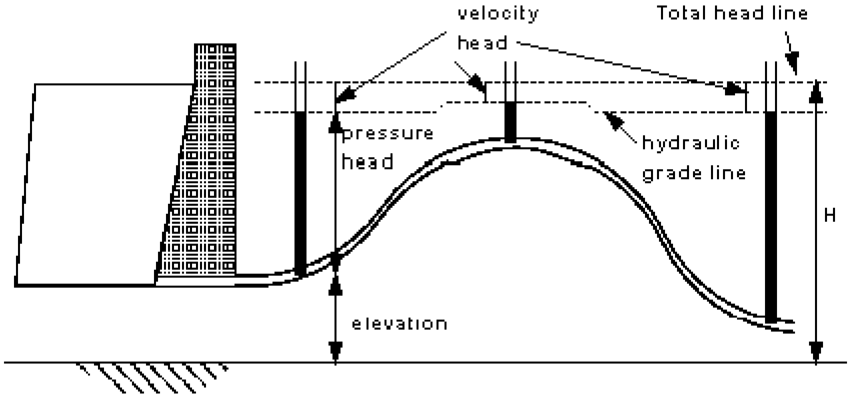


Piezometer levels when fluid is flowing

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

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

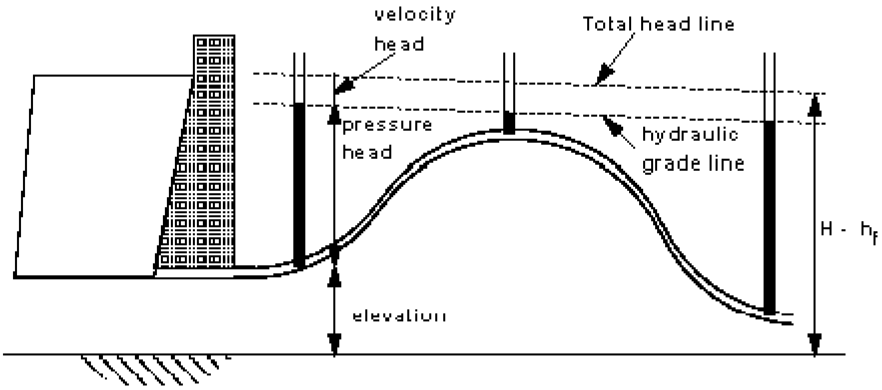
The velocity head at each point is now different. This is because the velocity is different at each point. By considering continuity we know that the velocity is different because the diameter of the pipe is different. Which pipe has the greatest diameter?

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.

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*.

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

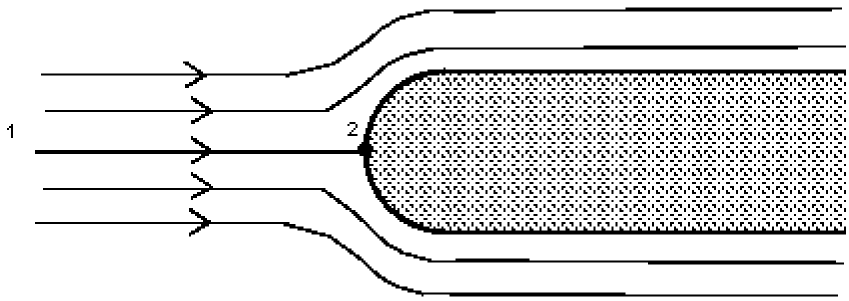
Go back to the main index page

Applications of the Bernoulli Equation

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

1. Pitot Tube

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



Streamlines around a blunt body

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

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

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

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

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

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

Dynamic pressure = $\frac{1}{2}\rho u_1^2$

or converting this to head (using $h = \frac{p}{\rho g}$)

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

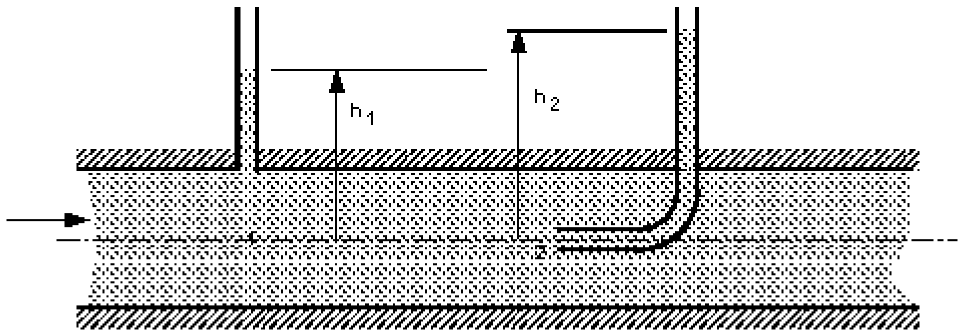
The total pressure is know as the *stagnation pressure* (or *total pressure*)

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 ,

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

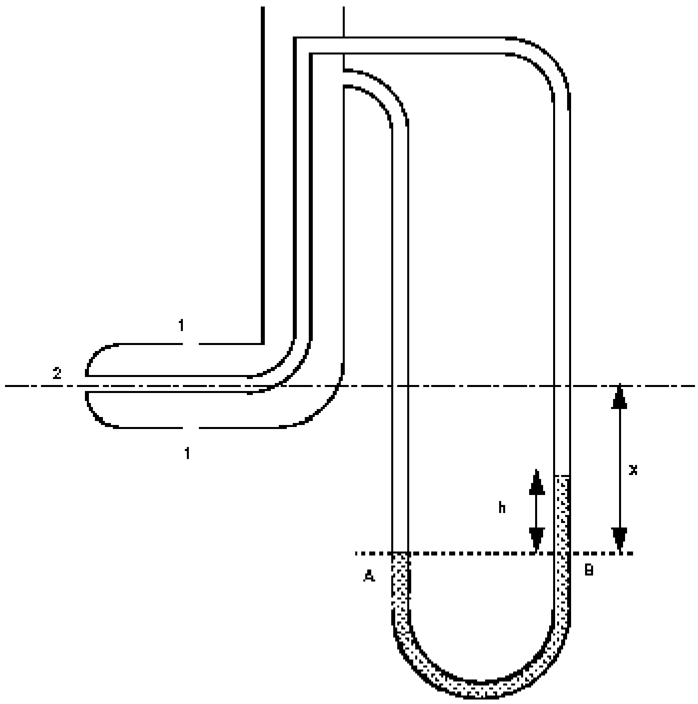
$$\rho g h_2 = \rho g h_1 + \frac{1}{2}\rho u_1^2$$

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

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

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

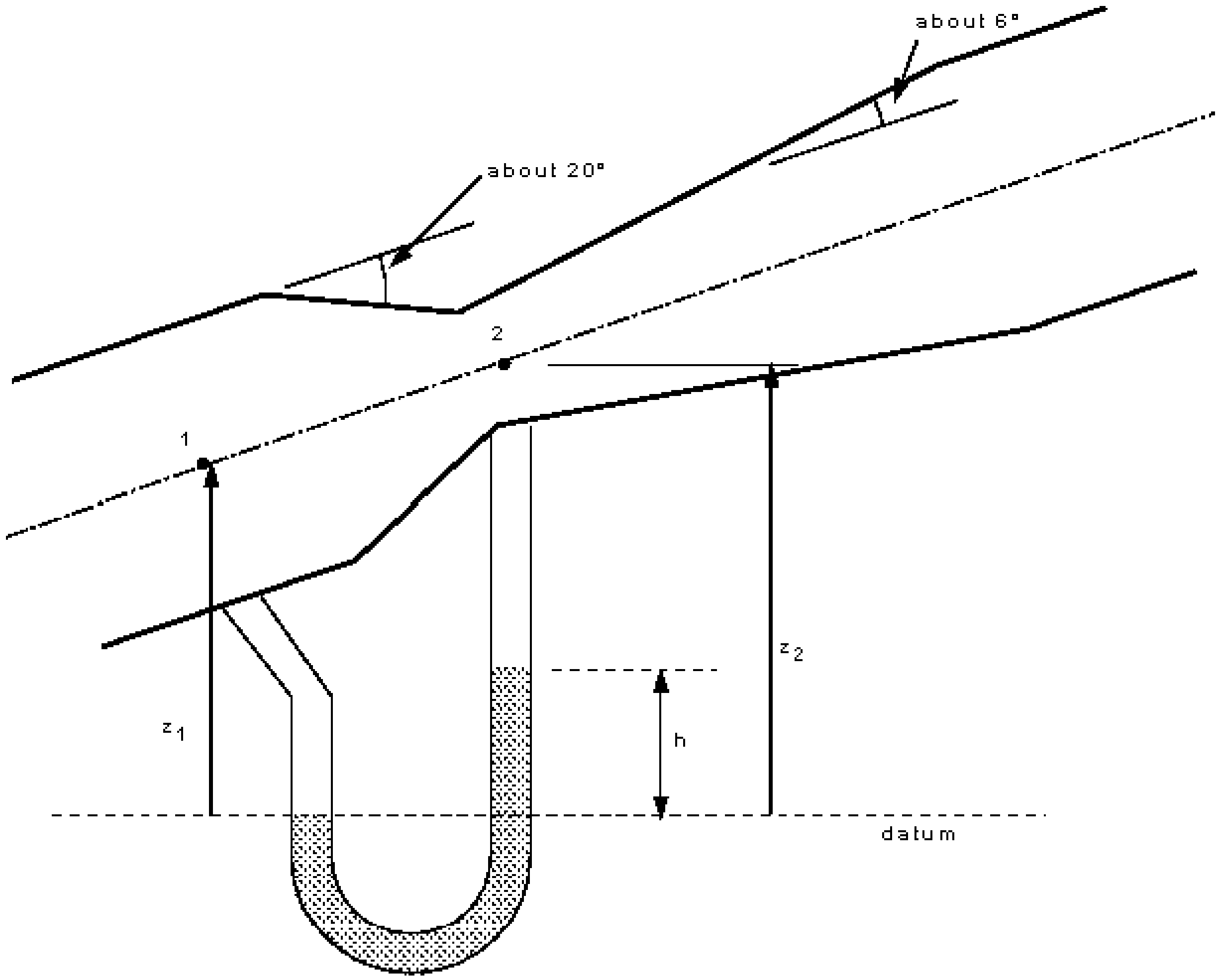
$$
\begin{aligned}
p_1 + hg(\rho_{man} - \rho) &= p_1 + \frac{\rho u_1^2}{2} \\
u_1 &= \sqrt{\frac{2gh(\rho_m - \rho)}{\rho}}
\end{aligned}
$$

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

3. Venturi Meter

The Venturi meter is a device for measuring discharge in a pipe. It consists of a rapidly converging section which increases the velocity of flow and hence reduces the

pressure. It then returns to the original dimensions of the pipe by a gently diverging 'diffuser' section. By measuring the pressure differences the discharge can be calculated. This is a particularly accurate method of flow measurement as energy loss are very small.



A Venturi meter

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

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

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

$$Q=u_1A_1=u_2A_2$$
$$u_2=\frac{u_1A_1}{A_2}$$

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::

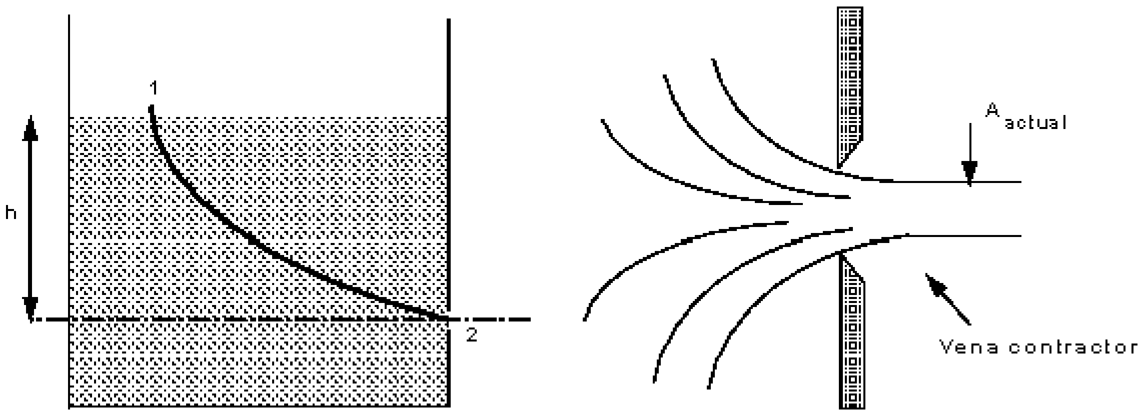
$$Q_{actual} = C_d A_1 A_2 \sqrt{\frac{2gh\left(\frac{\rho_{man}}{\rho} - 1\right)}{A_1^2 - A_2^2}}$$

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%.

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

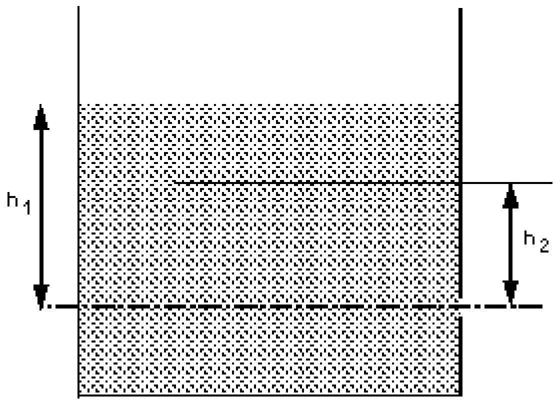
$$\begin{aligned} Q &= Au \\ Q_{actual} &= A_{actual} u_{actual} \\ &= C_c C_v A_{orifice} u_{theoretical} \\ &= C_d A_{orifice} u_{theoretical} \\ &= C_d A_{orifice} \sqrt{2gh} \end{aligned}$$

Where C_d is the **coefficient of discharge**, and $C_d = C_c C_v$

5. Time for a Tank to Empty

We now have an expression for the discharge out of a tank based on the height of water above the orifice. It would be useful to know how long it would take for the tank to empty.

As the tank empties, so the level of water will fall. We can get an expression for the time it takes to fall by integrating the expression for flow between the initial and final levels.



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

$$Q = Av$$
$$Q = -A\frac{\delta h}{\delta t}$$

(-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

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

1. Submerged Orifice

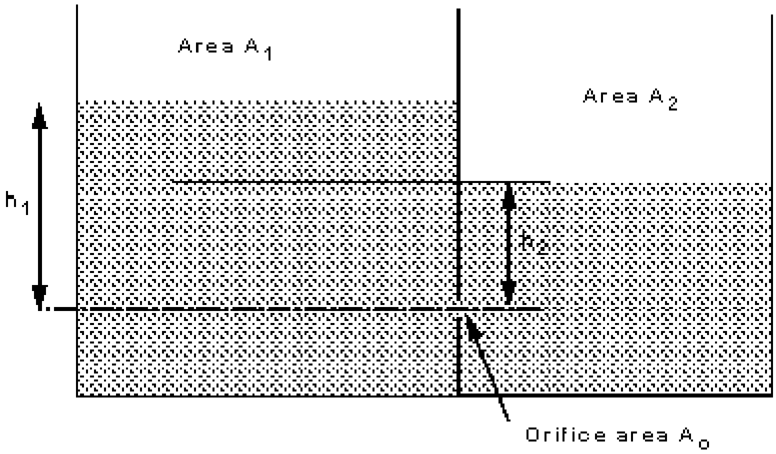
We have two tanks next to each other (or one tank separated by a dividing wall) and fluid is to flow between them through a submerged orifice. Although difficult to see, careful measurement of the flow indicates that the submerged jet flow behaves in a similar way to the jet in air in that it forms a 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

$$\begin{aligned} \frac{p_1}{\rho g} + \frac{u_1^2}{2g} + z_1 &= \frac{p_2}{\rho g} + \frac{u_2^2}{2g} + z_2 \\ 0 + 0 + h_1 &= \frac{\rho g h_2}{\rho g} + \frac{u_2^2}{2g} + 0 \\ u_2 &= \sqrt{2g(h_1 - h_2)} \end{aligned}$$

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

$$\begin{aligned} Q &= C_d A_o u \\ &= C_d A_o \sqrt{2g(h_1 - h_2)} \end{aligned}$$

6. Time for Equalisation of Levels in Two Tanks



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

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

Re arranging and integrating between the two levels we get

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

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

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

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

(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.

Flow Over Notches and Weirs

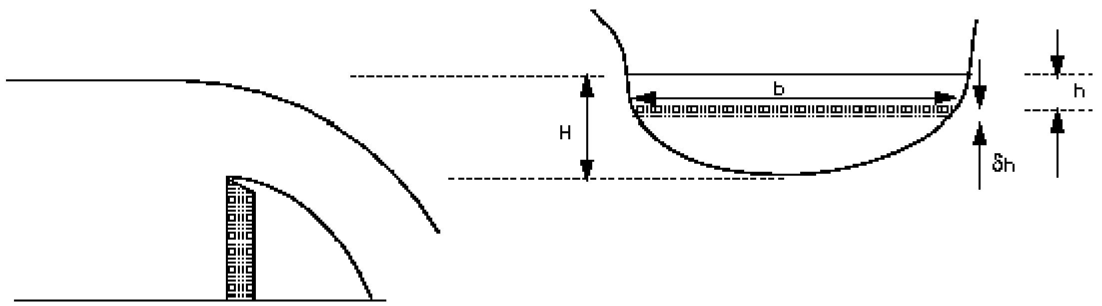
A notch is an opening in the side of a tank or reservoir which extends above the surface of the liquid. It is usually a device for measuring discharge. A weir is a notch on a larger scale - usually found in rivers. It may be sharp crested but also may have a substantial width in the direction of flow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

7. 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).

8. 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

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

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

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

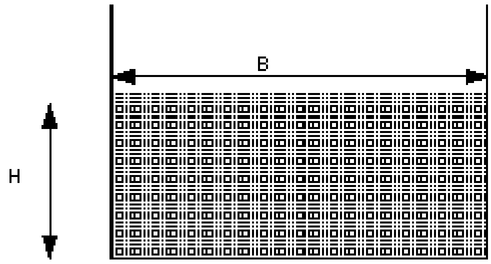
$$Q_{\text{theoretical}} = \sqrt{2g}\int_0^H bh^{1/2}\,dh$$

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.

9. 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

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

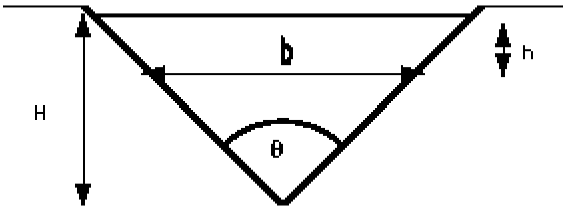
$$= \frac{2}{3}B\sqrt{2g}H^{3/2}$$

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

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

10. '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

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

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

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

Go back to the main index page

Go back to the main index page

Dynamics

Application of the Bernoulli Equation

4.1
In a vertical pipe carrying water, pressure gauges are inserted at points A and B where the pipe diameters are 0.15m and 0.075m respectively. The point B is 2.5m below A and when the flow rate down the pipe is 0.02 cumecs, the pressure at B is 14715 N/m^2 greater than that at A.

Assuming the losses in the pipe between A and B can be expressed as $k\frac{v^2}{2g}$ where *v* is

the velocity at A, find the value of k.
If the gauges at A and B are replaced by tubes filled with water and connected to a U-tube containing mercury of relative density 13.6, give a sketch showing how the levels in the two limbs of the U-tube differ and calculate the value of this difference in metres.
[k = 0.319, 0.0794m]

4.2

A Venturimeter with an entrance diameter of 0.3m and a throat diameter of 0.2m is used to measure the volume of gas flowing through a pipe. The discharge coefficient of the meter is 0.96.
Assuming the specific weight of the gas to be constant at 19.62 N/m^3, calculate the volume flowing when the pressure difference between the entrance and the throat is measured as 0.06m on a water U-tube manometer.
[0.816 m^3/s]

4.3
A Venturimeter is used for measuring flow of water along a pipe. The diameter of the Venturi throat is two fifths the diameter of the pipe. The inlet and throat are connected by water filled tubes to a mercury U-tube manometer. The velocity of flow along the pipe is found to be $2.5\sqrt{H}$ m/s, where H is the manometer reading in metres of mercury. Determine the loss of head between inlet and throat of the Venturi when H is 0.49m. (Relative density of mercury is 13.6).
[0.23m of water]

4.4
Water is discharging from a tank through a convergent-divergent mouthpiece. The exit from the tank is rounded so that losses there may be neglected and the minimum diameter is 0.05m.
If the head in the tank above the centre-line of the mouthpiece is 1.83m. a) What is the discharge?
b) What must be the diameter at the exit if the absolute pressure at the minimum area is to be 2.44m of water? c) What would the discharge be if the divergent part of the mouth piece were removed. (Assume atmospheric pressure is 10m of water).
[0.0752m, 0.0266m^3/s, 0.0118m^3/s]

4.5
A closed tank has an orifice 0.025m diameter in one of its vertical sides. The tank contains oil to a depth of 0.61m above the centre of the orifice and the pressure in the air space above the oil is maintained at 13780 N/m^2 above atmospheric. Determine the discharge from the orifice.
(Coefficient of discharge of the orifice is 0.61, relative density of oil is 0.9).
[0.00195 m^3/s]

4.6
The discharge of a Venturimeter was found to be constant for rates of flow exceeding a certain value. Show that for this condition the loss of head due to friction in the convergent parts of the meter can be expressed as KQ^2 *m* where K is a constant and Q is the rate of flow in cumecs.

Obtain the value of K if the inlet and throat diameter of the Venturimeter are 0.102m and 0.05m respectively and the discharge coefficient is 0.96.
[K=1060]

4.7
A Venturimeter is to fitted in a horizontal pipe of 0.15m diameter to measure a flow of water which may be anything up to 240m^3/hour. The pressure head at the inlet for this flow is 18m above atmospheric and the pressure head at the throat must not be lower than 7m below atmospheric. Between the inlet and the throat there is an estimated frictional loss of 10% of the difference in pressure head between these points. Calculate the minimum allowable diameter for the throat.
[0.063m]

4.8
A Venturimeter of throat diameter 0.076m is fitted in a 0.152m diameter vertical pipe in which liquid of relative density 0.8 flows downwards. Pressure gauges are fitted to the inlet and to the throat sections. The throat being 0.914m below the inlet. Taking the coefficient of the meter as 0.97 find the discharge
a) when the pressure gauges read the same b)when the inlet gauge reads 15170 N/m^2 higher than the throat gauge.
[0.0192m^3/s, 0.034m^3/s]

Tank emptying

5.1
A reservoir is circular in plan and the sides slope at an angle of tan^{-1}(1/5) to the horizontal. When the reservoir is full the diameter of the water surface is 50m. Discharge from the reservoir takes place through a pipe of diameter 0.65m, the outlet being 4m below top water level. Determine the time for the water level to fall 2m assuming the discharge to be $0.75a\sqrt{2gH}$ cumecs where a is the cross sectional area of the pipe in m^2 and H is the head of water above the outlet in m.
[1325 seconds]

5.2
A rectangular swimming pool is 1m deep at one end and increases uniformly in depth to 2.6m at the other end. The pool is 8m wide and 32m long and is emptied through an orifice of area 0.224m^2, at the lowest point in the side of the deep end. Taking C_d for the orifice as 0.6, find, from first principles,
a) the time for the depth to fall by 1m b) the time to empty the pool completely.
[299 second, 662 seconds]

5.3
A vertical cylindrical tank 2m diameter has, at the bottom, a 0.05m diameter sharp edged orifice for which the discharge coefficient is 0.6.
a) If water enters the tank at a constant rate of 0.0095 cumecs find the depth of water above the orifice when the level in the tank becomes stable.
b) Find the time for the level to fall from 3m to 1m above the orifice when the inflow is turned off.
c) If water now runs into the tank at 0.02 cumecs, the orifice remaining open, find the

rate of rise in water level when the level has reached a depth of 1.7m above the orifice.
[a) 3.314m, b) 881 seconds, c) 0.252m/min]

5.4
A horizontal boiler shell (i.e. a horizontal cylinder) 2m diameter and 10m long is half full of water. Find the time of emptying the shell through a short vertical pipe, diameter 0.08m, attached to the bottom of the shell. Take the coefficient of discharge to be 0.8.
[1370 seconds]

5.5
Two cylinders standing upright contain liquid and are connected by a submerged orifice. The diameters of the cylinders are 1.75m and 1.0m and of the orifice, 0.08m. The difference in levels of the liquid is initially 1.35m. Find how long it will take for this difference to be reduced to 0.66m if the coefficient of discharge for the orifice is 0.605. (Work from first principles.)
[30.7 seconds]

5.6
A rectangular reservoir with vertical walls has a plan area of 60000m^3. Discharge from the reservoir take place over a rectangular weir. The flow characteristics of the weir is $Q = 0.678\ H^{3/2}$ cumecs where H is the depth of water above the weir crest. The sill of the weir is 3.4m above the bottom of the reservoir. Starting with a depth of water of 4m in the reservoir and no inflow, what will be the depth of water after one hour?
[3.98m]

Notches and weirs

6.1
Deduce an expression for the discharge of water over a right-angled sharp edged V-notch, given that the coefficient of discharge is 0.61.
A rectangular tank 16m by 6m has the same notch in one of its short vertical sides. Determine the time taken for the head, measured from the bottom of the notch, to fall from 15cm to 7.5cm.
[1399 seconds]

6.2
Derive an expression for the discharge over a sharp crested rectangular weir. A sharp edged weir is to be constructed across a stream in which the normal flow is 200 litres/sec. If the maximum flow likely to occur in the stream is 5 times the normal flow then determine the length of weir necessary to limit the rise in water level to 38.4cm above that for normal flow. $C_d=0.61$.
[1.24m]

6.3
Show that the rate of flow across a triangular notch is given by $Q=C_dKH^{5/2}$ cumecs, where C_d is an experimental coefficient, K depends on the angle of the notch, and H is the height of the undisturbed water level above the bottom of the notch in metres.

State the reasons for the introduction of the coefficient.
Water from a tank having a surface area of $10m^2$ flows over a 90 notch. It is found that the time taken to lower the level from 8cm to 7cm above the bottom of the notch is 43.5seconds. Determine the coefficient C_d assuming that it remains constant during his period.
[0.635]

6.4
A reservoir with vertical sides has a plan area of $56000m^2$. Discharge from the reservoir takes place over a rectangular weir, the flow characteristic of which is $Q=1.77BH^{3/2}$ m^3/s. At times of maximum rainfall, water flows into the reservoir at the rate of $9m^3/s$. Find a) the length of weir required to discharge this quantity if head must not exceed 0.6m; b) the time necessary for the head to drop from 60cm to 30cm if the inflow suddenly stops.
[10.94m, 3093seconds]

6.5
Develop a formula for the discharge over a 90 V-notch weir in terms of head above the bottom of the V.
A channel conveys 300 litres/sec of water. At the outlet end there is a 90 V-notch weir for which the coefficient of discharge is 0.58. At what distance above the bottom of the channel should the weir be placed in order to make the depth in the channel 1.30m? With the weir in this position what is the depth of water in the channel when the flow is 200 litres/sec?
[0.755m, 1.218m]

6.6
Show that the quantity of water flowing across a triangular V-notch of angle 2θ is
$Q = C_d \frac{8}{15} \tan\theta \sqrt{2g} H^{5/2}$. Find the flow if the measured head above the bottom of the V is 38cm, when θ=45 and C_d=0.6. If the flow is wanted within an accuracy of 2%, what are the limiting values of the head.
[$0.126m^3/s$, 0.377m, 0.383m]

Go back to the main index page

Go back to the main index page

---

1. The Momentum Equation And Its Applications

We have all seen moving fluids exerting forces. The lift force on an aircraft is exerted by the air moving over the wing. A jet of water from a hose exerts a force on whatever it hits. In fluid mechanics the analysis of motion is performed in the same

way as in solid mechanics - by use of Newton's laws of motion. Account is also taken for the special properties of fluids when in motion.

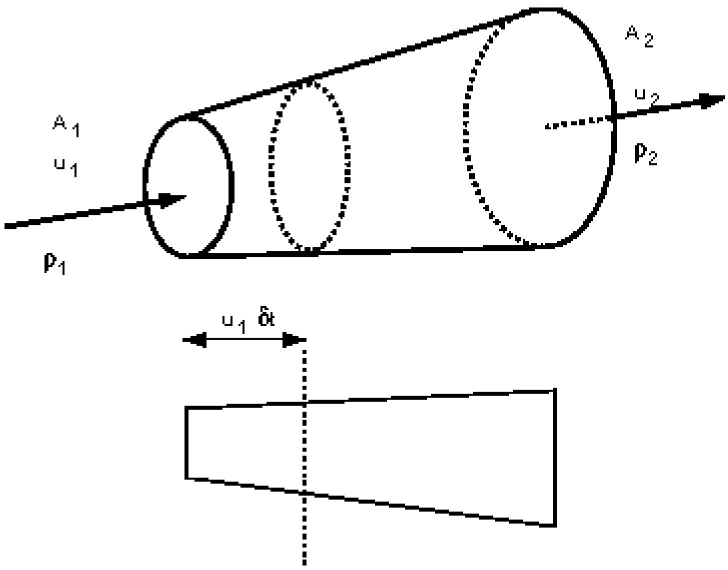
The momentum equation is a statement of Newton's Second Law and relates the sum of the forces acting on an element of fluid to its acceleration or rate of change of momentum. You will probably recognise the equation $F = ma$ which is used in the analysis of solid mechanics to relate applied force to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

Newton's 2nd Law can be written:

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

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

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



A streamtube in three and two-dimensions

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

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

this has mass,

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

and momentum

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

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

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

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

$$\text{Force} = \text{rate of change of momentum}$$
$$F = \frac{(\rho_2 A_2 u_2 \delta t\, u_2 - \rho_1 A_1 u_1 \delta t\, u_1)}{\delta t}$$

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

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

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

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

we can write

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

The rate at which momentum leaves face 1 is

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

The rate at which momentum enters face 2 is

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

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

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

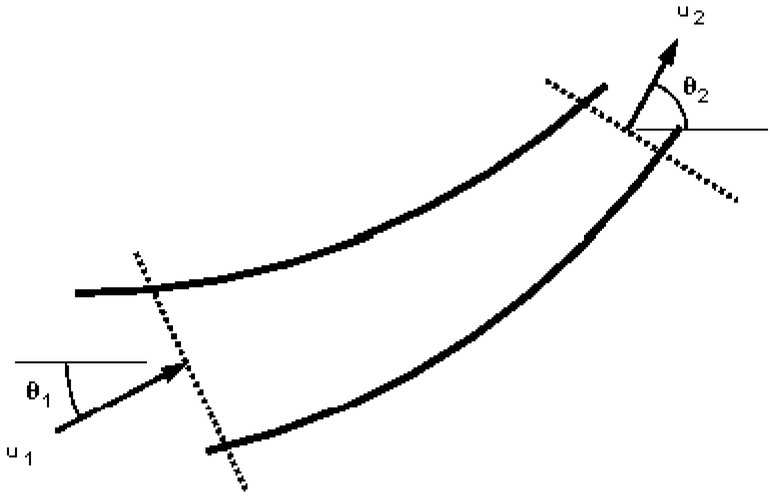
i.e.

$$\text{Force} = \text{rate of change of momentum}$$
$$F = \dot{m}(u_2 - u_1)$$
$$F = Q\rho(u_2 - u_1)$$

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

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

Consider the two dimensional system in the figure below:



Two dimensional flow in a streamtube

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

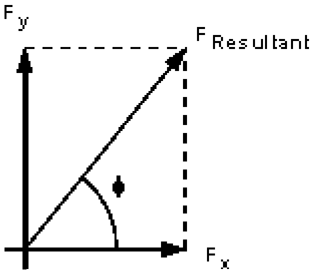
The force in the x-direction

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

And the force in the y-direction

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

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



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

And the angle which this force acts at is given by

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

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

In summary we can say:

$$\text{The total force } \textbf{exerted on} \text{ the fluid} = \text{rate of change of momentum through the control volume}$$

$$\begin{aligned} F &= \dot{m}\left(u_{out} - u_{in}\right) \\ &= \rho Q\left(u_{out} - u_{in}\right) \end{aligned}$$

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

$$R = -F_R$$

Go back to the main index page

Go back to the main index page

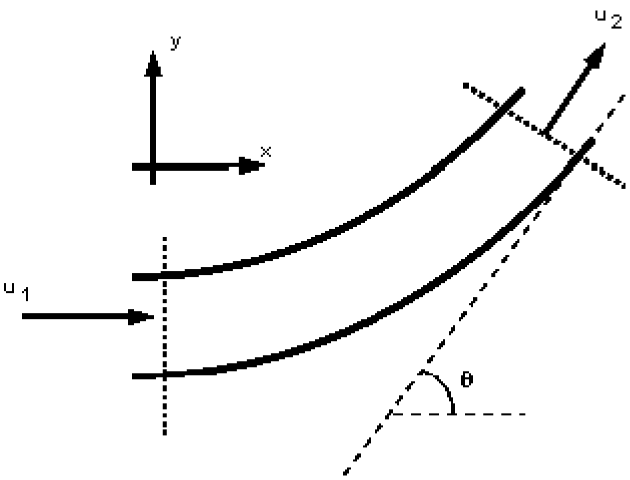
Application of the Momentum Equation

In this section we will consider the following examples:

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

1. The force due the flow around a pipe bend

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



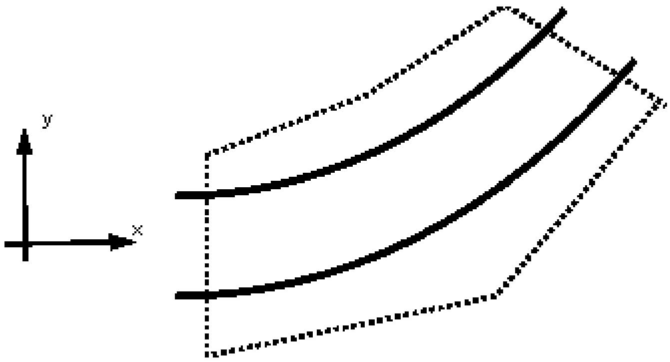
Flow round a pipe bend of constant cross-section

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 Control Volume



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

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>

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

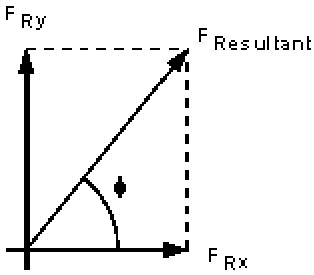
5 <u>Calculate the **body** force</u>

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 <u>Calculate the **resultant** force</u>

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

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

$$R = -F_R$$

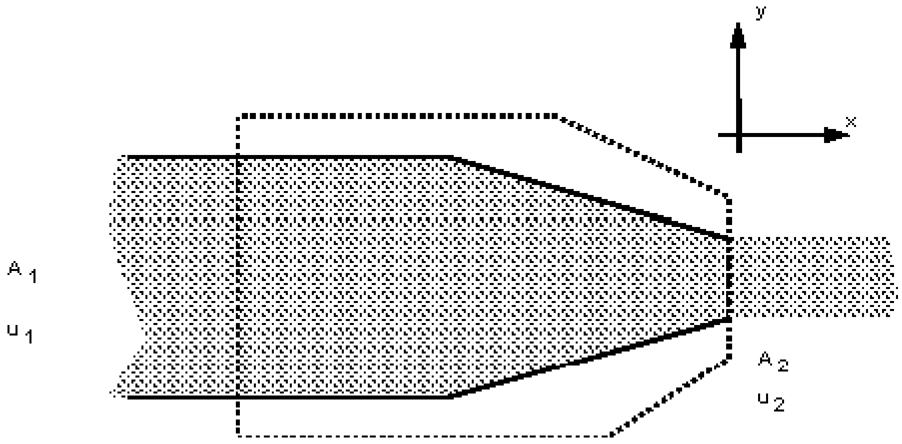
2. Force on a pipe nozzle

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 Calculate the **total** force

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

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

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

4 Calculate the **pressure** force

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

We use the Bernoulli equation to calculate the pressure

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

Is friction losses are neglected, $h_f = 0$

the nozzle is horizontal, $z_1 = z_2$

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

and with continuity gives

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

5 Calculate the **body** force

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

6 Calculate the **resultant** force

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

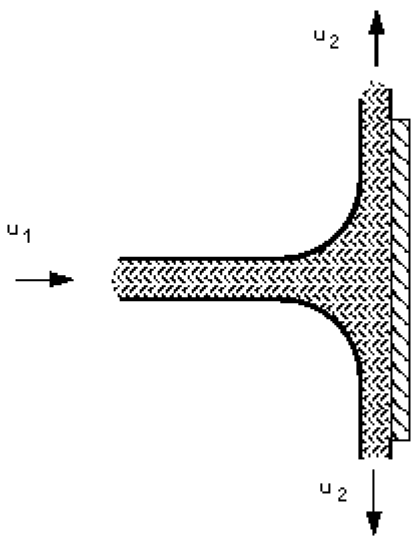
So the fireman must be able to resist the force of

$$R = -F_{T_x}$$

3. 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.



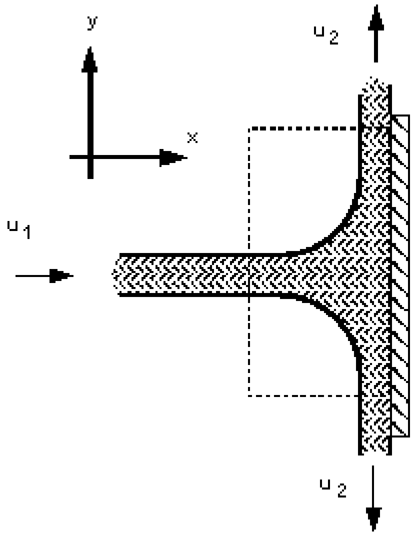
A perpendicular jet hitting a plane.

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

$$R = -F_{R_x}$$

Go back to the main index page

Go back to the main index page

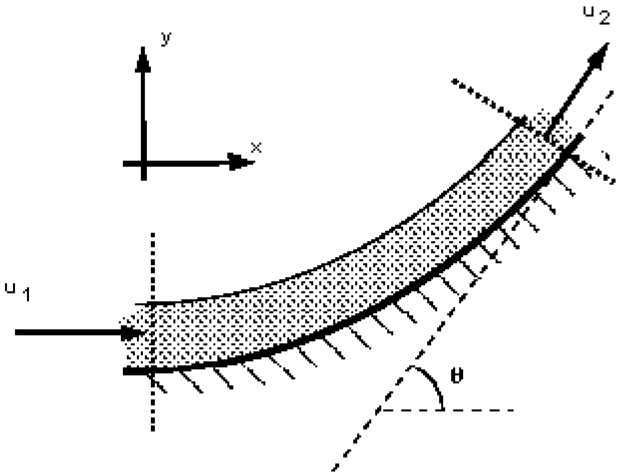
More Applications of the Momentum Equation

In this section we will consider the following examples:

1. Force due to flow round a curved vane.
2. A curved vane on a Pelton wheel turbine.
3. Impact of a jet on An angled plane surface.

1. 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 $u_1 = u_2 = \dfrac{Q}{A}$, so

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

and in the y-direction

$$\begin{aligned} F_{T_y} &= \rho Q\left(u_2 \sin\theta - 0\right) \\ &= \rho \frac{Q^2}{A} \end{aligned}$$

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

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

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

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 <u>Calculate the **resultant** force</u>

$$\begin{aligned} F_{T_x} &= F_{R_x} + F_{P_x} + F_{B_x} \\ F_{R_x} &= F_{T_x} \\ F_{T_y} &= F_{R_y} + F_{P_y} + F_{B_y} \\ F_{R_y} &= F_{T_y} \end{aligned}$$

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

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

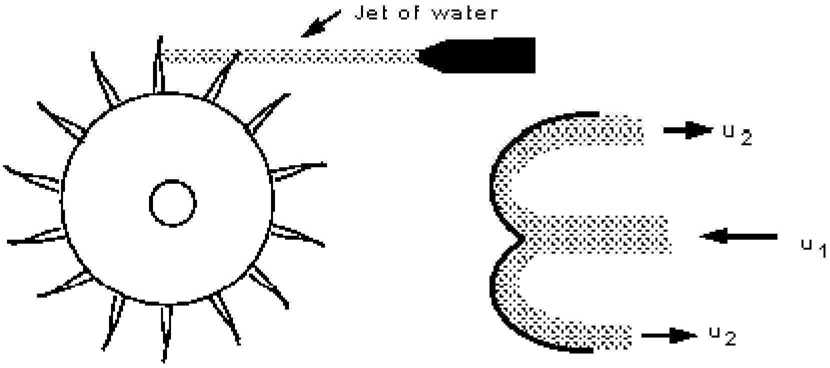
exerted on the fluid.

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

$$R = -F_R$$

• 2. 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.



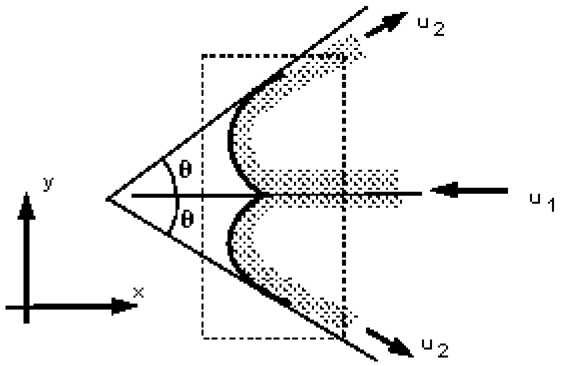
Pelton wheel arrangement and jet hitting cross-section of blade.

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 <u>Control volume and Co-ordinate axis are shown in the figure below.</u>



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

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

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

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

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

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

exerted on the fluid.

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

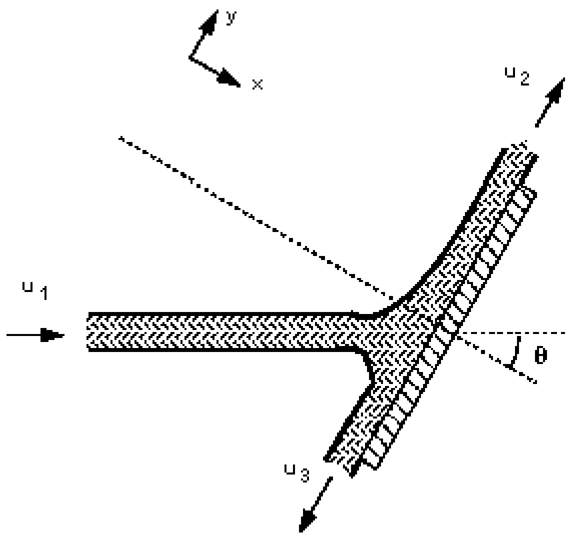
$$R = -F_{R_x}$$

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. Force due to a jet hitting an inclined plane

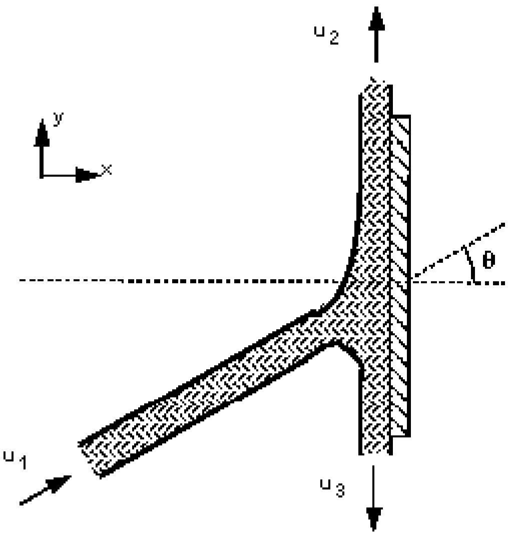
We have seen above the forces involved when a jet hits a plane at right angles. If the plane is tilted to an angle the analysis becomes a little more involved. This is demonstrated below.



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

$$Q_1 = Q_2 + Q_3$$
$$u_1A_1 = u_2A_2 + u_3A_3$$

so

$$A_1 = A_2 + A_3$$
$$Q_1 = A_1u$$
$$Q_2 = A_2u$$
$$Q_3 = (A_1 - A_2)u$$

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.

$$F_{T_x} = \rho\left((Q_2 u_{2_x} + Q_3 u_{3_x}) - Q_1 u_{1_x}\right)$$

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

All zero as the pressure is everywhere atmospheric.

1. <u>Calculate the **body** force</u>

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

4. <u>Calculate the **resultant** force</u>

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

exerted **on the fluid**.

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

$$\begin{aligned} R &= -F_{R_x} \\ &= \rho Q_1 u_1 \cos\theta \end{aligned}$$

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

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

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

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

as $u_2 = u_3 = u$

$$\begin{aligned} Q_2 &= Q_3\left(\frac{1+\sin\theta}{1-\sin\theta}\right) \\ Q_1 &= Q_3\left(\frac{1+\sin\theta}{1-\sin\theta}\right) + Q_3 \\ &= Q_3\left(1+\frac{1+\sin\theta}{1-\sin\theta}\right) \end{aligned}$$

So we know the discharge in each direction

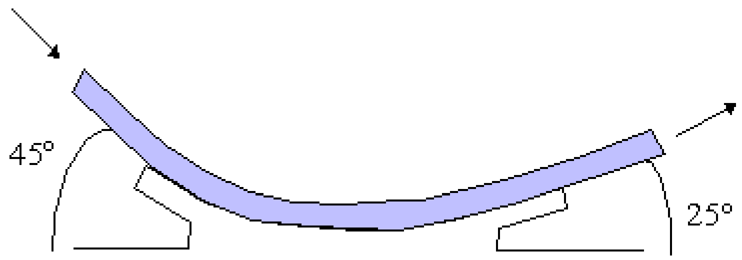
Go back to the main index page

Go back to the main index page

Dynamics

Application of the Momentum Equation

6.1
The figure below shows a smooth curved vane attached to a rigid foundation. The jet of water, rectangular in section, 75mm wide and 25mm thick, strike the vane with a velocity of 25m/s. Calculate the vertical and horizontal components of the force exerted on the vane and indicate in which direction these components act.
[Horizontal 233.4 N acting from right to left. Vertical 1324.6 N acting downwards]



6.2
A 600mm diameter pipeline carries water under a head of 30m with a velocity of 3m/s. This water main is fitted with a horizontal bend which turns the axis of the pipeline through 75 (i.e. the internal angle at the bend is 105). Calculate the resultant force on the bend and its angle to the horizontal.
[104.044 kN, 52 29']

6.3
A horizontal jet of water 210^3 mm^2 cross-section and flowing at a velocity of 15 m/s hits a flat plate at 60 to the axis (of the jet) and to the horizontal. The jet is such that there is no side spread. If the plate is stationary, calculate a) the force exerted on the plate in the direction of the jet and b) the ratio between the quantity of fluid that is deflected upwards and that downwards. (Assume that there is no friction and therefore no shear force.)
[338N, 3:1]

6.4
A 75mm diameter jet of water having a velocity of 25m/s strikes a flat plate, the normal of which is inclined at 30 to the jet. Find the force normal to the surface of the plate.
[2.39kN]

6.5
The outlet pipe from a pump is a bend of 45 rising in the vertical plane (i.e. and internal angle of 135). The bend is 150mm diameter at its inlet and 300mm diameter at its outlet. The pipe axis at the inlet is horizontal and at the outlet it is 1m higher. By neglecting friction, calculate the force and its direction if the inlet pressure is $100kN/m^2$ and the flow of water through the pipe is $0.3m^3/s$. The volume of the pipe is $0.075m^3$.
[13.94kN at 67 40' to the horizontal]

6.6
The force exerted by a 25mm diameter jet against a flat plate normal to the axis of the jet is 650N. What is the flow in m^3/s?
[0.018 m^3/s]

6.7
A curved plate deflects a 75mm diameter jet through an angle of 45. For a velocity in the jet of 40m/s to the right, compute the components of the force developed against the curved plate. (Assume no friction).
[R_x=2070N, R_y=5000N down]

6.8
A 45 reducing bend, 0.6m diameter upstream, 0.3m diameter downstream, has water flowing through it at the rate of 0.45m^3/s under a pressure of 1.45 bar. Neglecting any loss is head for friction, calculate the force exerted by the water on the bend, and its direction of application.
[R=34400N to the right and down, θ = 14]

Go back to the main index page

Go back to the main index page

---

1. 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:

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

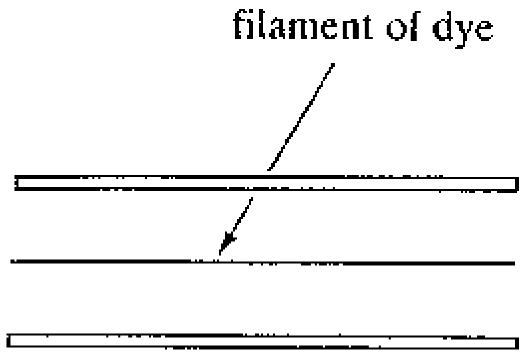
where the constant of proportionality, μ, is known as the coefficient of viscosity (or simply viscosity). We saw that for some fluids - sometimes known as exotic fluids - the value of μ changes with stress or velocity gradient. We shall only deal with Newtonian fluids.

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.

2. Laminar and turbulent flow

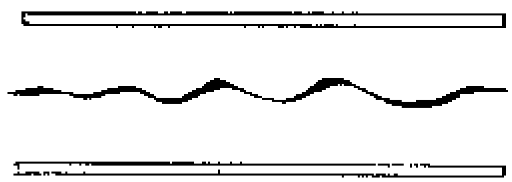
If we were to take a pipe of free flowing water and inject a dye into the middle of the stream, what would we expect to happen?

This



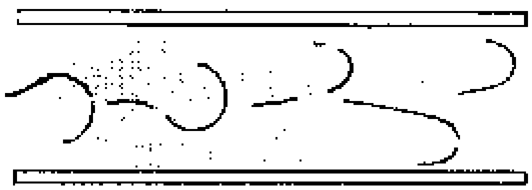
Laminar (viscous)

this



Transitional

or this



Turbulent

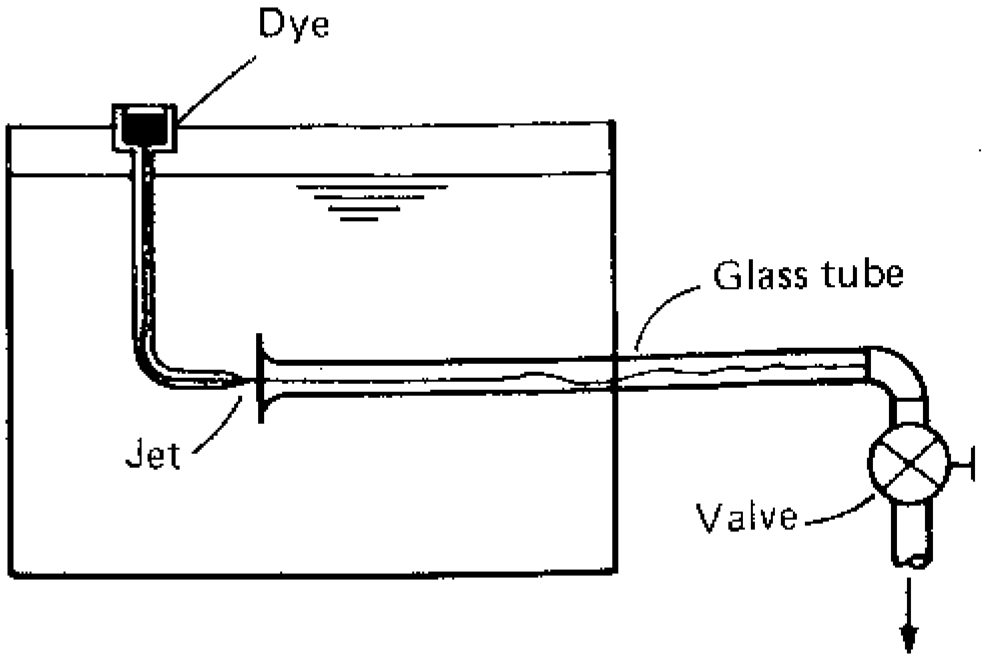
Actually both would happen - but for different flow rates. The top occurs when the fluid is flowing fast and the lower when it is flowing slowly.

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:

<u>Laminar flow</u>

- 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.

<u>Transitional flow</u>

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

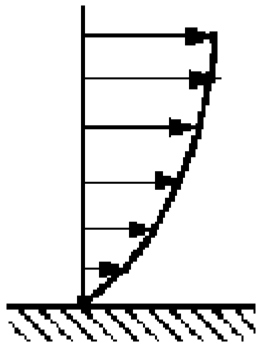
<u>Turbulent flow</u>

- 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.

3. Pressure loss due to friction in a pipeline.

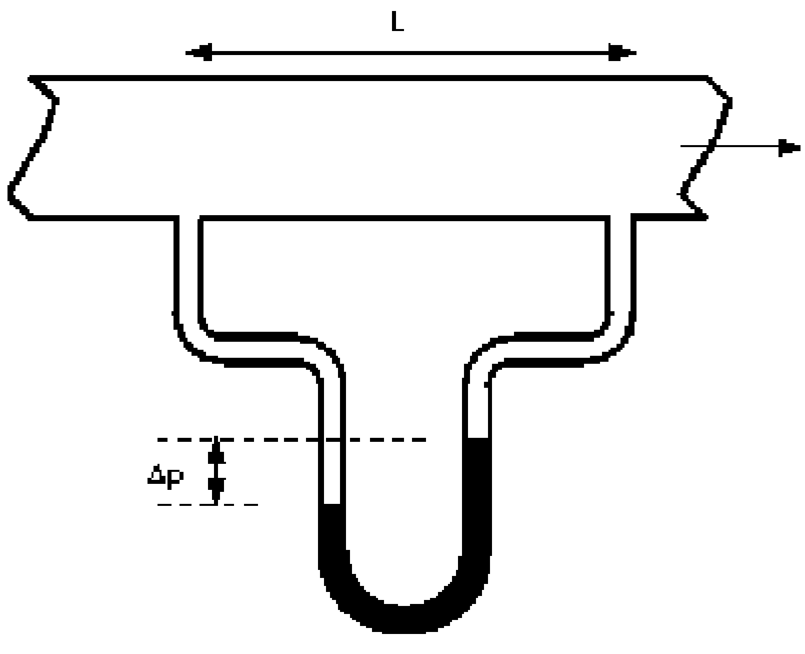
Up to this point on the course we have considered ideal fluids where there have been no losses due to friction or any other factors. In reality, because fluids are viscous, energy is lost by flowing fluids due to friction which must be taken into account. The effect of the friction shows itself as a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



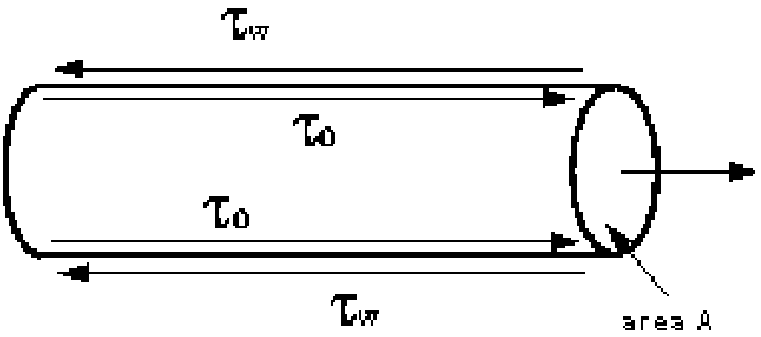
If a manometer is attached as the pressure (head) difference due to the energy lost by the fluid overcoming the shear stress can be easily seen.

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

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

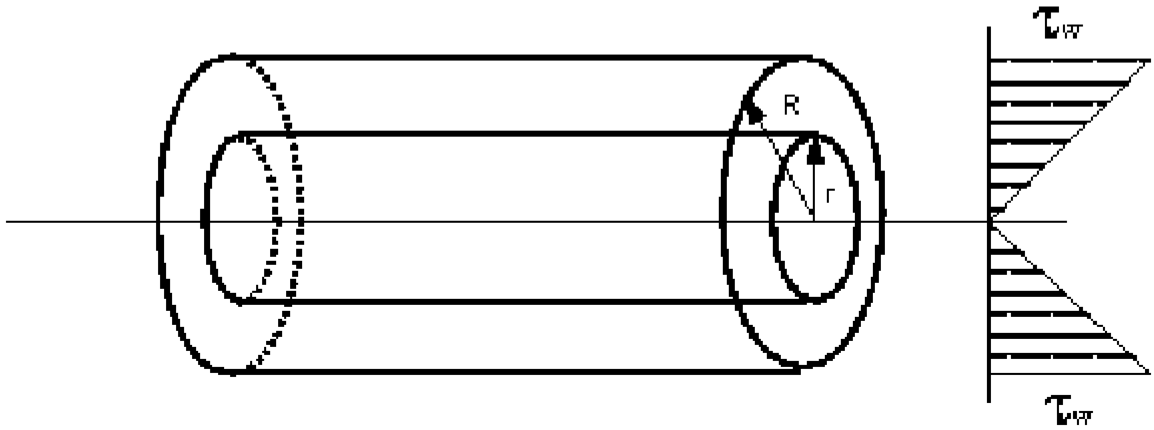
As the flow is in equilibrium,

driving force = retarding force

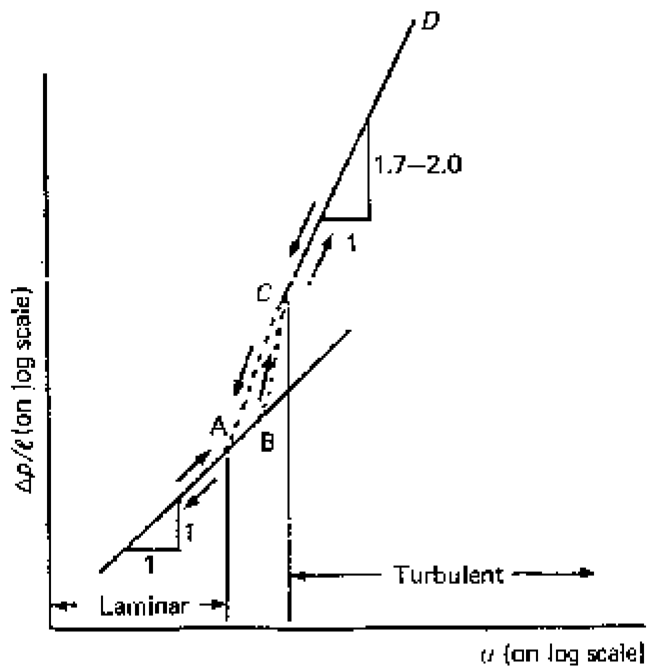
$$\Delta p \frac{\pi d^2}{4} = \tau_w \pi d L$$

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

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



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



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

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

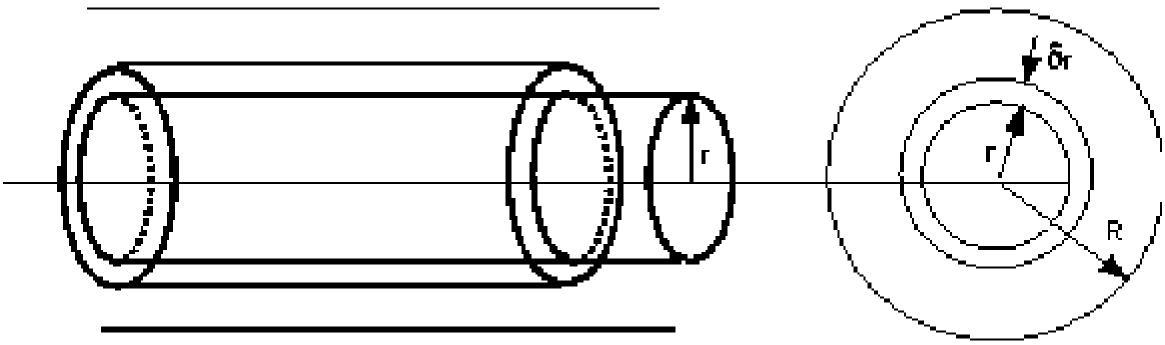
As these are empirical relationships, they help in determining the pressure loss but not in finding the magnitude of the shear stress at the wall τ_w on a particular fluid. If we knew τ_w we could then use it to give a general equation to predict the pressure loss.

4. Pressure loss during laminar flow in a pipe

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

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

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



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

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

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

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

Which can be combined with the equation above to give

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

In an integral form this gives an expression for velocity,

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

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

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

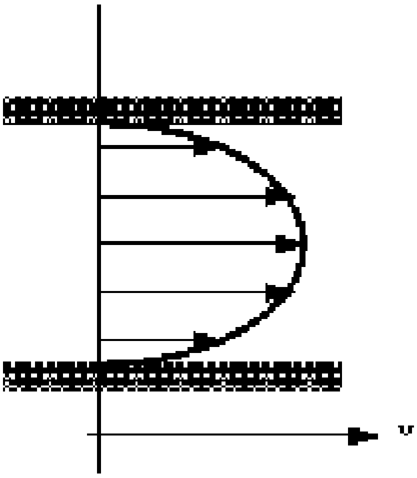
At $r = 0$, (the centre of the pipe), $u = u_{max}$, at $r = R$ (the pipe wall) $u = 0$, giving

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

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

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

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



What is the discharge in the pipe?

$$Q = u_m A$$

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

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

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

So the discharge can be written

$$Q = \frac{\Delta p \, d^2}{32 \mu L} \frac{\pi d^2}{4}$$
$$= \frac{\Delta p}{L} \frac{\pi d^2}{128 \mu}$$

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

We are interested in the pressure loss (head loss) and want to relate this to the velocity of the flow. Writing pressure loss in terms of head loss h_f, i.e. $p = \rho g h_f$

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

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.

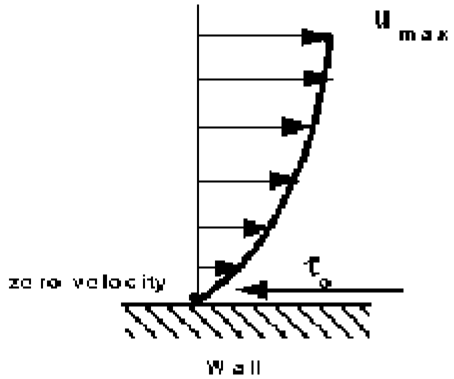
Go back to the main index page

Go back to the main index page

1. 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.



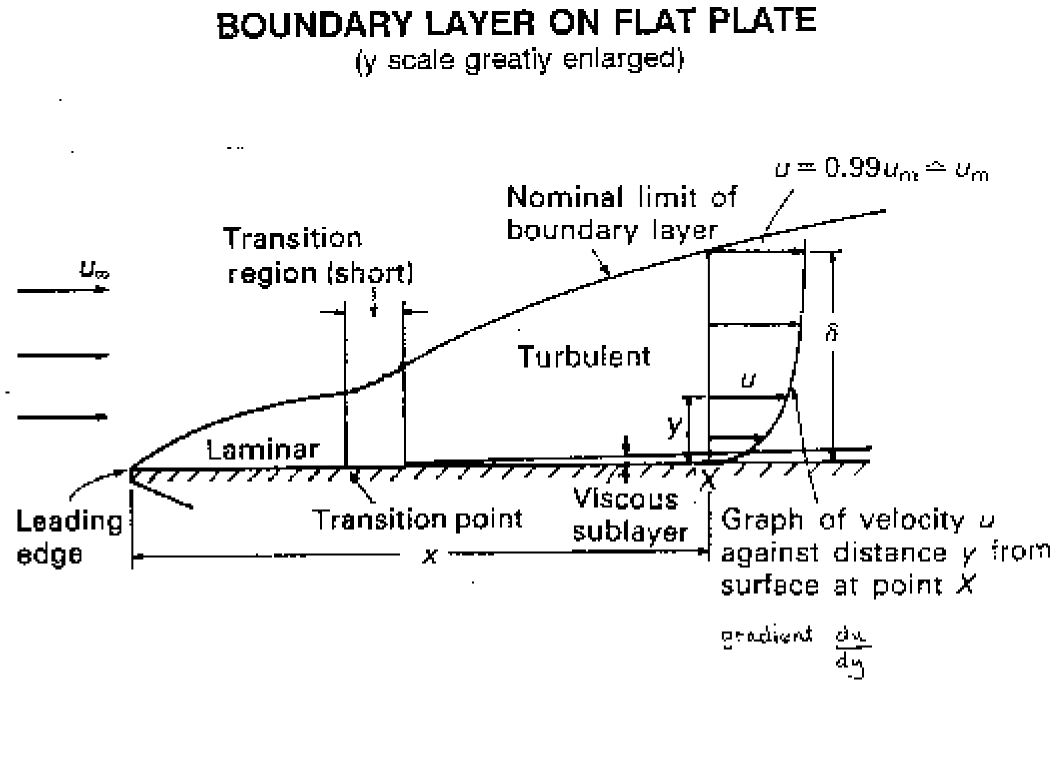
Looking at this two-dimensionally we get the above velocity profile from the wall to the centre of the flow.

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.

2. 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.

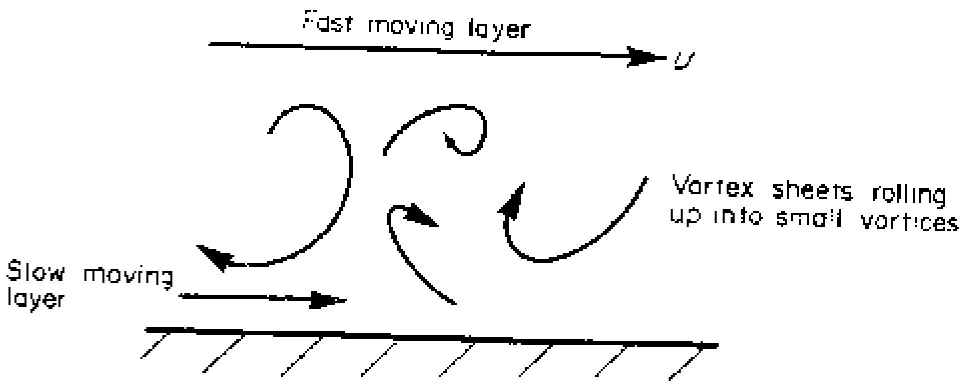
The fluid near the top of the boundary layer is dragging the fluid nearer to the solid surface along. The mechanism for this dragging may be one of two types:

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.

3. 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. 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.

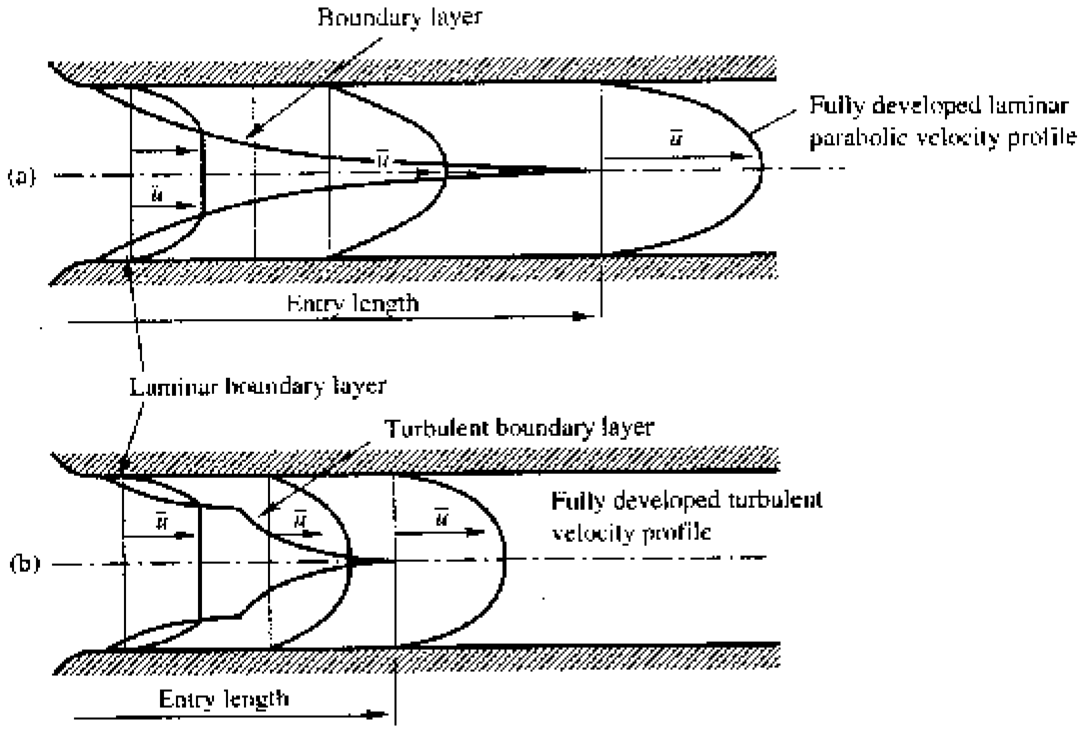
$$\mathrm{Re} = \frac{\rho u d}{\mu}$$

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

Laminar flow: Re < 2000

Transitional flow: 2000 < Re < 4000

Turbulent flow: 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 120 diameter

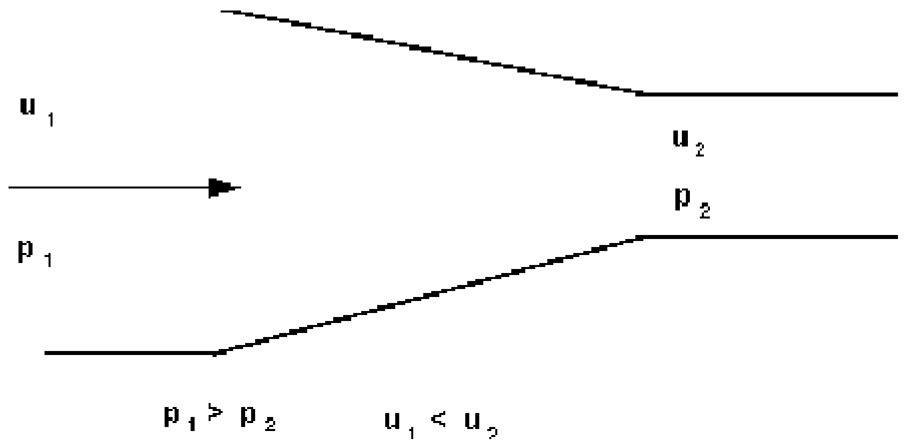
Turbulent flow entry length 60 diameter

5. Boundary layer separation

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.

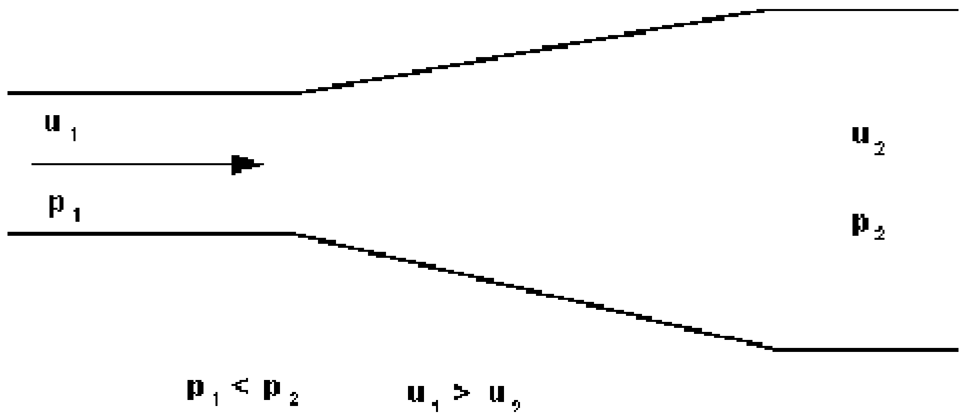
This is the case for *convergent* flows.

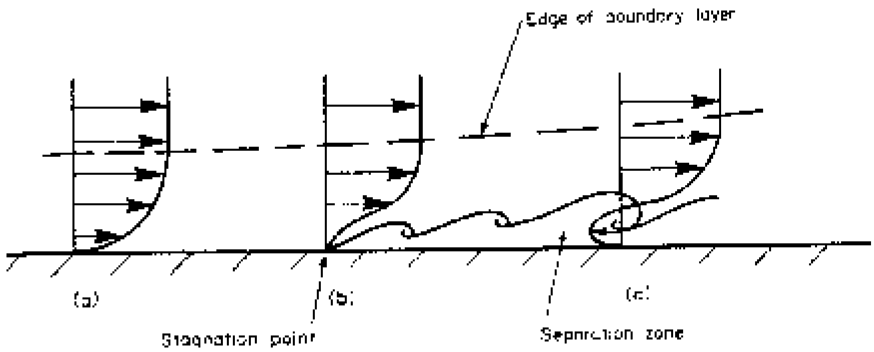


The accelerating fluid maintains the fluid close to the wall in motion. Hence the flow remains stable and turbulence reduces. Boundary layer separation does not occur.

Divergent flows: Positive pressure gradients

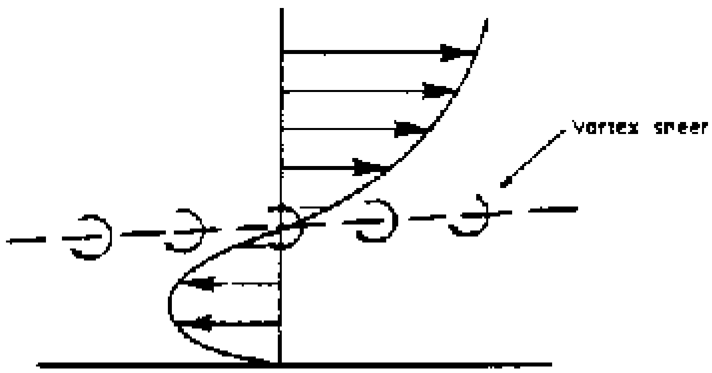
When the pressure increases in the direction of flow the situation is very different. Fluid outside the boundary layer has enough momentum to overcome this pressure which is trying to push it backwards. The fluid within the boundary layer has so little momentum that it will very quickly be brought to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





This phenomenon is known as **boundary layer separation**.

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.

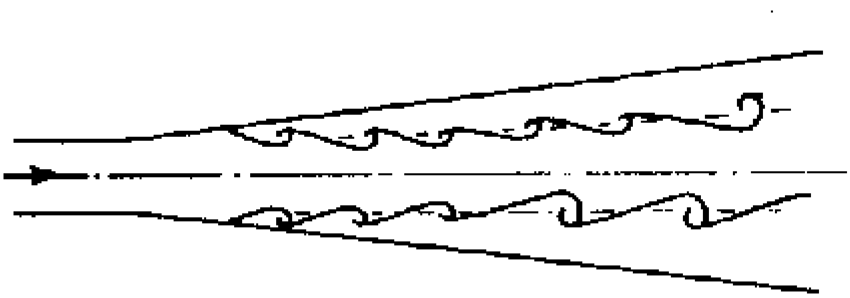


This boundary layer separation and increase in the turbulence because of the vortices results in very large energy losses in the flow.

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

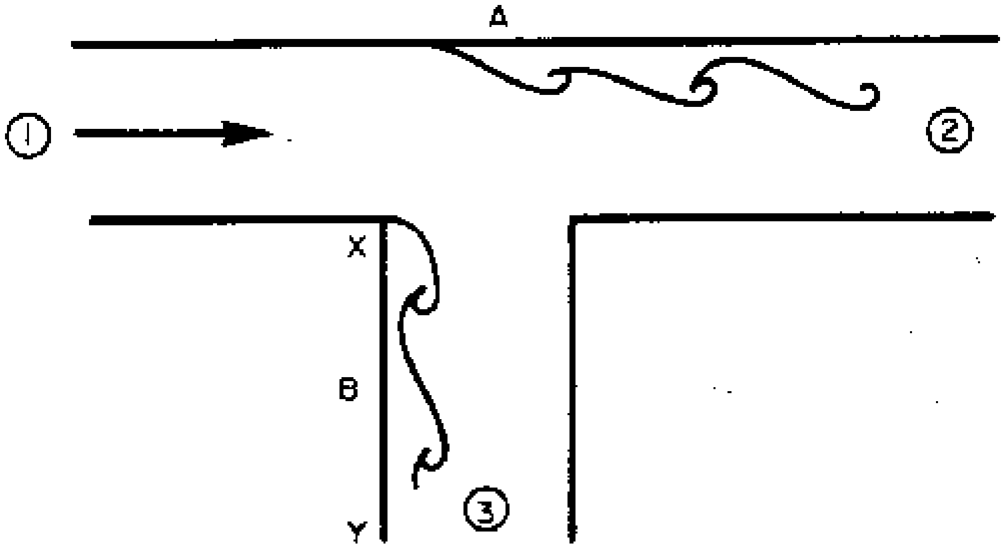
6. Examples of boundary layer separation
A divergent duct or diffuser

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.

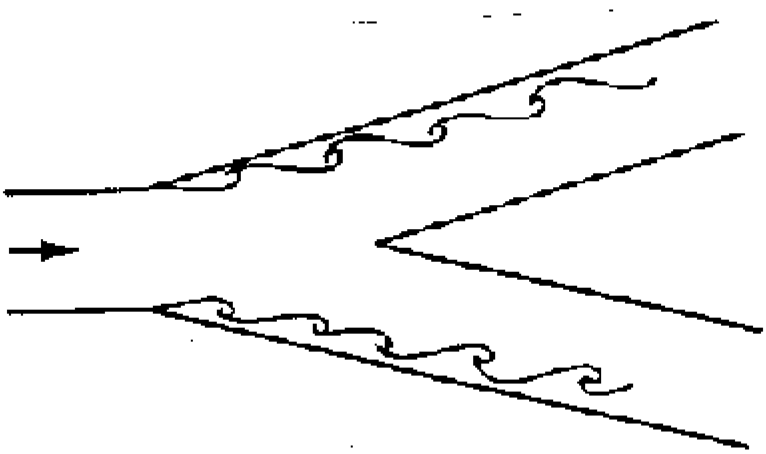
Tee-Junctions



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.

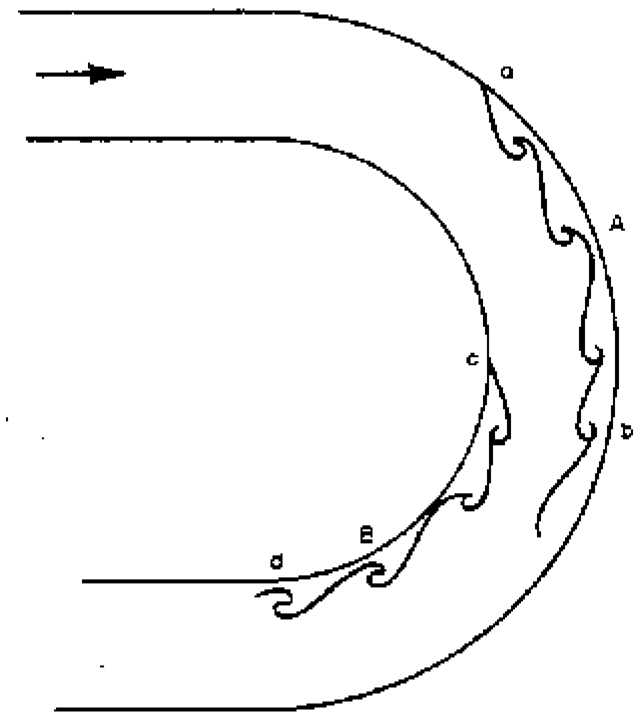
Y-Junctions

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.

Bends

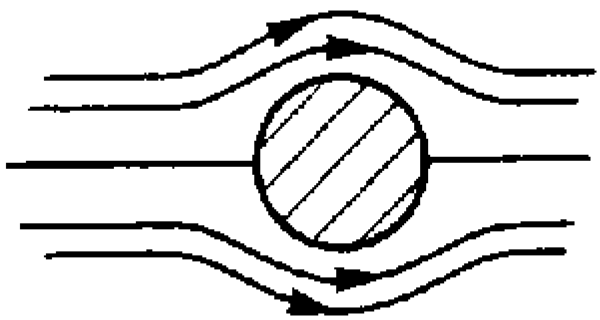


Two separation zones occur in bends as shown above. The pressure at b must be greater than at a as it must provide the required radial acceleration for the fluid to get round the bend. There is thus an adverse pressure gradient between a and b so separation may occur here.

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.

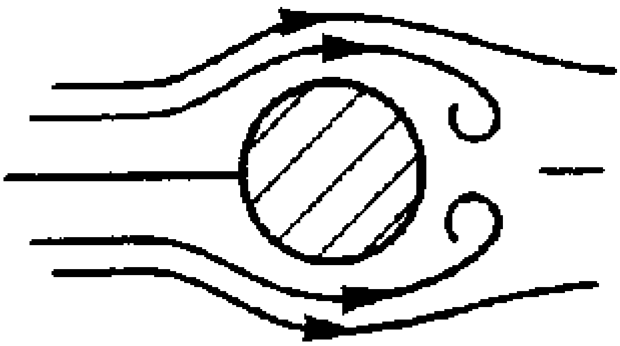
Flow past a cylinder

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.



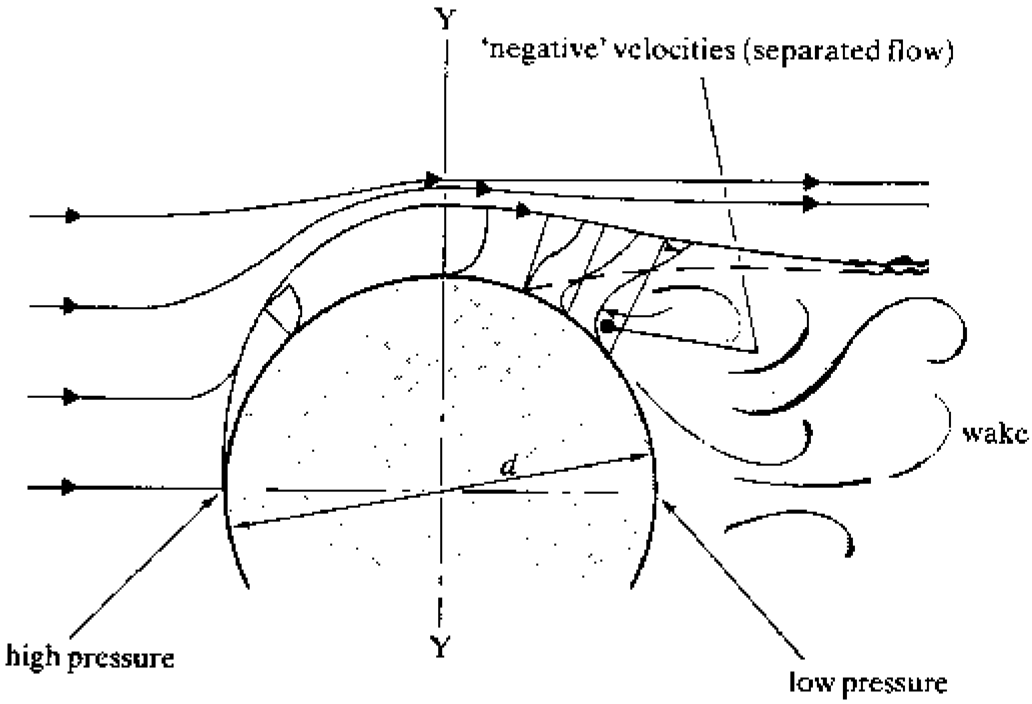
$Re < 0{\cdot}5$

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.



$0{\cdot}5 < Re < 70$

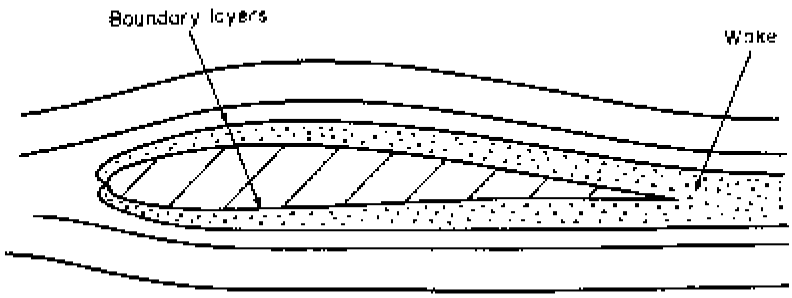
Above a Re of 70 the ends of the separated zones curl up into vortices and detach alternately from each side forming a trail of vortices on the down stream side of the cylinder. This trial in known as a **Karman vortex trail** or **street**. This vortex trail can easily be seen in a river by looking 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.)



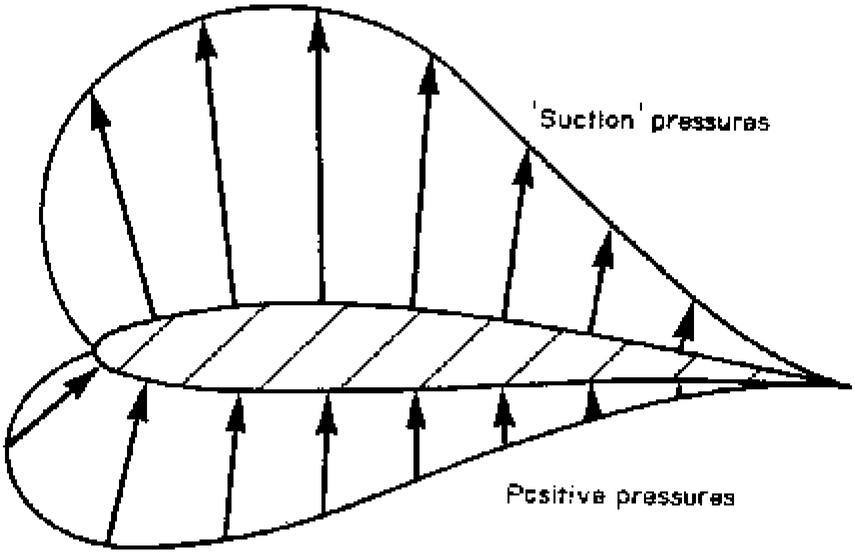
Looking at the figure above, the formation of the separation occurs as the fluid accelerates from the centre to get round the cylinder (it must accelerate as it has further to go than the surrounding fluid). It reaches a maximum at Y, where it also has also dropped in pressure. The adverse pressure gradient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

Aerofoil

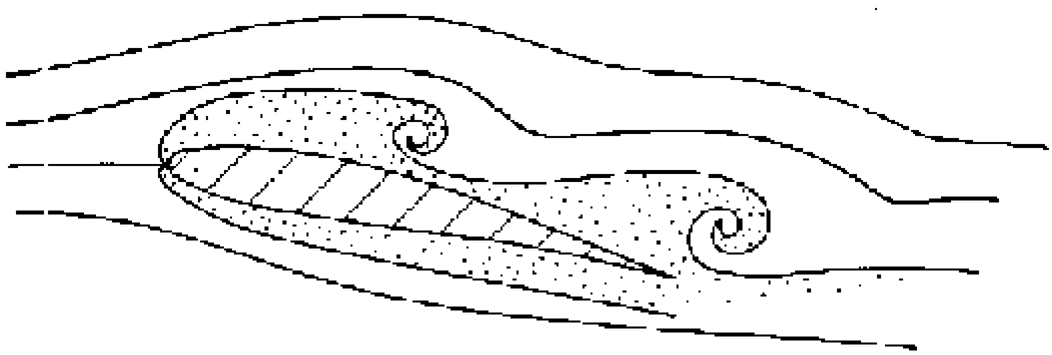
Normal flow over a aerofoil (a wing cross-section) is shown in the figure below with the boundary layers greatly exaggerated.



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



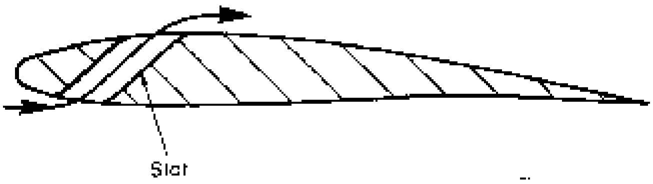
If the angle of the wing becomes too great and boundary layer separation occurs on the top of the aerofoil the pressure pattern will change dramatically. This phenomenon is known as **stalling**.



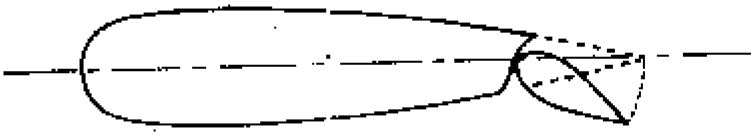
When stalling occurs, all, or most, of the 'suction' pressure is lost, and the plane will suddenly drop from the sky! The only solution to this is to put the plane into a dive to regain the boundary layer. A transverse lift force is then exerted on the wing which gives the pilot some control and allows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



Go back to the main index page

Go back to the main index page

1. 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.

2. Dimensions and units

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

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

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

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

length = L

mass = M

time = T

force = F

temperature = Θ

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

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

Quantity	SI Unit	.	Dimension
velocity	m/s	ms^{-1}	LT^{-1}
acceleration	m/s^2	ms^{-2}	LT^{-2}
force	N kg m/s^2	kg ms^{-2}	$M LT^{-2}$
energy (or work)	Joule 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}

1. 3. 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

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

i.e. the units are consistent.

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

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

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

This property of dimensional homogeneity can be useful for:

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

4. 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. 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.

---

6. Choice of repeating variables

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

Some rules which should be followed are

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

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

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

7. An example

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

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

n = 3 and m = 6

There are m - n = 3 π groups, so

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

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

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

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

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

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

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 $\pi_2 = \rho^{a_2}u^{b_2}d^{c_2}N$

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

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

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

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,

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

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

This may also be written:

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

8. 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.

9. 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: $\phi(\pi_1 , \pi_2 , \pi_3 \dots\dots\dots \pi_{m-n}) = 0$

Then the following manipulations are permitted:

1. Any number of groups can be combined by multiplication or division to form a new group which replaces one of the existing. E.g. π_1 and π_2 may be combined to form $\pi_{1a} = \pi_1 / \pi_2$ so the defining equation becomes $\phi(\pi_{1a} , \pi_2 , \pi_3 \dots\dots\dots \pi_{m-n}) = 0$

2. 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.
3. 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.
4. Any dimensionless group may be multiplied by a constant.
5. Any group may be expressed as a function of the other groups, e.g. $\pi_2 = \phi(\pi_1, \pi_3 \ldots\ldots\ldots \pi_{m-n})$

In general the defining equation could look like

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

10. Common π groups

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

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

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

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

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

Mach number $\mathrm{Mn} = \dfrac{u}{c}$ Local velocity, local velocity of sound ratio

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

Go back to the main index page

Go back to the main index page

1. 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.

2. 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.

3. Kinematic similarity

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

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

Some useful ratios are:

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

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.

4. Dynamic similarity

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

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

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

5. 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:

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

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

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

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

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

Leading to π_1 as

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

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

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

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

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

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

This assumption then allows the equation following to be written

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

Which gives this scaling law for resistance force:

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

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

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

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

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

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

Substituting this into the scaling law for resistance gives

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

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

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

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

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

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

6. 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

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

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

$$\rho_m = 20\rho_p$$

So the model velocity is found to be

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

$$u_m = 250km/h$$

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

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

7. 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.

8. Geometric distortion in river models

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

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

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

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

Go back to the main index page

Go back to the main index page

Pressure and Manometers

1.1
What will be the (a) the gauge pressure and (b) the absolute pressure of water at depth 12m below the surface? ρ_{water} = 1000 kg/m^3, and p $_{atmosphere}$ = 101kN/m^2.
[117.72 kN/m^2, 218.72 kN/m^2]

1.2
At what depth below the surface of oil, relative density 0.8, will produce a pressure of 120 kN/m^2? What depth of water is this equivalent to?
[15.3m, 12.2m]

1.3
What would the pressure in kN/m^2 be if the equivalent head is measured as 400mm of (a) mercury γ=13.6 (b) water (c) oil specific weight 7.9 kN/m^3 (d) a liquid of density

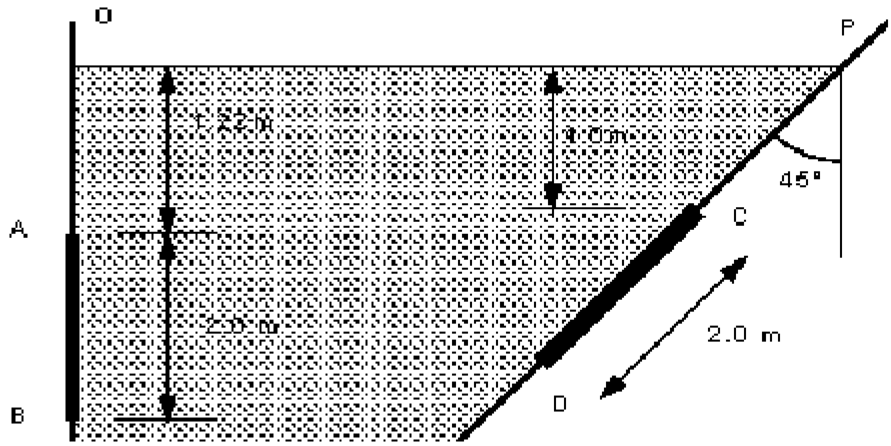
520 kg/m^3?
[53.4 kN/m^2, 3.92 kN/m^2, 3.16 kN/m^2, 2.04 kN/m^2]

1.4
A manometer connected to a pipe indicates a negative gauge pressure of 50mm of mercury. What is the absolute pressure in the pipe in Newtons per square metre if the atmospheric pressure is 1 bar?
[93.3 kN/m^2]

1.5
What height would a water barometer need to be to measure atmospheric pressure of 1 bar?
[>10.19m]

1.6
An inclined manometer is required to measure an air pressure of 3mm of water to an accuracy of +/- 3%. The inclined arm is 8mm in diameter and the larger arm has a diameter of 24mm. The manometric fluid has density 740 kg/m^3 and the scale may be read to +/- 0.5mm.
What is the angle required to ensure the desired accuracy may be achieved?
[7.6]

1.7
Determine the resultant force due to the water acting on the 1m by 2m rectangular area AB shown in the diagram below.
[43 560 N, 2.37m from O



1.8
Determine the resultant force due to the water acting on the 1.25m by 2.0m triangular area CD shown in the figure above. The apex of the triangle is at C.
[23.810^3N, 2.821m from P]

<u>Forces on submerged surfaces</u>

2.1
Obtain an expression for the depth of the centre of pressure of a plane surface wholly submerged in a fluid and inclined at an angle to the free surface of the liquid.

A horizontal circular pipe, 1.25m diameter, is closed by a butterfly disk which rotates about a horizontal axis through its centre. Determine the torque which would have to be applied to the disk spindle to keep the disk closed in a vertical position when there is a 3m head of fresh water above the axis.
[1176 Nm]

2.2
A dock gate is to be reinforced with three horizontal beams. If the water acts on one side only, to a depth of 6m, find the positions of the beams measured from the water surface so that each will carry an equal load. Give the load per meter.
[58 860 N/m, 2.31m, 4.22m, 5.47m]

2.3
The profile of a masonry dam is an arc of a circle, the arc having a radius of 30m and subtending an angle of 60 at the centre of curvature which lies in the water surface. Determine (a) the load on the dam in N/m length, (b) the position of the line of action to this pressure.
[4.28 10^6 N/m length at depth 19.0m]

2.4
The arch of a bridge over a stream is in the form of a semi-circle of radius 2m. the bridge width is 4m. Due to a flood the water level is now 1.25m above the crest of the arch. Calculate (a) the upward force on the underside of the arch, (b) the horizontal thrust on one half of the arch.
[263.6 kN, 176.6 kN]

2.5
The face of a dam is vertical to a depth of 7.5m below the water surface then slopes at 30 to the vertical. If the depth of water is 17m what is the resultant force per metre acting on the whole face?
[1563.29 kN]

2.6
A tank with vertical sides is square in plan with 3m long sides. The tank contains oil of relative density 0.9 to a depth of 2.0m which is floating on water a depth of 1.5m. Calculate the force on the walls and the height of the centre of pressure from the bottom of the tank.
[165.54 kN, 1.15m]

Application of the Bernoulli Equation

3.1
In a vertical pipe carrying water, pressure gauges are inserted at points A and B where the pipe diameters are 0.15m and 0.075m respectively. The point B is 2.5m below A and when the flow rate down the pipe is 0.02 cumecs, the pressure at B is 14715 N/m^2 greater than that at A.

Assuming the losses in the pipe between A and B can be expressed as $k\frac{v^2}{2g}$ where *v* is the velocity at A, find the value of *k*.

If the gauges at A and B are replaced by tubes filled with water and connected to a U-tube containing mercury of relative density 13.6, give a sketch showing how the levels in the two limbs of the U-tube differ and calculate the value of this difference in metres.
[k = 0.319, 0.0794m]

3.2
A Venturimeter with an entrance diameter of 0.3m and a throat diameter of 0.2m is used to measure the volume of gas flowing through a pipe. The discharge coefficient of the meter is 0.96.
Assuming the specific weight of the gas to be constant at 19.62 N/m^3, calculate the volume flowing when the pressure difference between the entrance and the throat is measured as 0.06m on a water U-tube manometer.
[0.816 m^3/s]

3.3
A Venturimeter is used for measuring flow of water along a pipe. The diameter of the Venturi throat is two fifths the diameter of the pipe. The inlet and throat are connected by water filled tubes to a mercury U-tube manometer. The velocity of flow along the pipe is found to be $2.5\sqrt{H}$ m/s, where H is the manometer reading in metres of mercury. Determine the loss of head between inlet and throat of the Venturi when H is 0.49m. (Relative density of mercury is 13.6).
[0.23m of water]

3.4
Water is discharging from a tank through a convergent-divergent mouthpiece. The exit from the tank is rounded so that losses there may be neglected and the minimum diameter is 0.05m.
If the head in the tank above the centre-line of the mouthpiece is 1.83m. a) What is the discharge?
b) What must be the diameter at the exit if the absolute pressure at the minimum area is to be 2.44m of water? c) What would the discharge be if the divergent part of the mouth piece were removed. (Assume atmospheric pressure is 10m of water).
[0.0752m, 0.0266m^3/s, 0.0118m^3/s]

3.5
A closed tank has an orifice 0.025m diameter in one of its vertical sides. The tank contains oil to a depth of 0.61m above the centre of the orifice and the pressure in the air space above the oil is maintained at 13780 N/m^2 above atmospheric. Determine the discharge from the orifice.
(Coefficient of discharge of the orifice is 0.61, relative density of oil is 0.9).
[0.00195 m^3/s]

3.6
The discharge coefficient of a Venturimeter was found to be constant for rates of flow exceeding a certain value. Show that for this condition the loss of head due to friction in the convergent parts of the meter can be expressed as KQ^2 m where K is a constant and Q is the rate of flow in cumecs.
Obtain the value of K if the inlet and throat diameter of the Venturimeter are 0.102m

and 0.05m respectively and the discharge coefficient is 0.96.
[K=1060]

3.7
A Venturimeter is to fitted in a horizontal pipe of 0.15m diameter to measure a flow of water which may be anything up to 240m^3/hour. The pressure head at the inlet for this flow is 18m above atmospheric and the pressure head at the throat must not be lower than 7m below atmospheric. Between the inlet and the throat there is an estimated frictional loss of 10% of the difference in pressure head between these points. Calculate the minimum allowable diameter for the throat.
[0.063m]

3.8
A Venturimeter of throat diameter 0.076m is fitted in a 0.152m diameter vertical pipe in which liquid of relative density 0.8 flows downwards. Pressure gauges are fitted to the inlet and to the throat sections. The throat being 0.914m below the inlet. Taking the coefficient of the meter as 0.97 find the discharge
a) when the pressure gauges read the same b)when the inlet gauge reads 15170 N/m^2 higher than the throat gauge.
[0.0192m^3/s, 0.034m^3/s]

Tanks emptying

4.1
A reservoir is circular in plan and the sides slope at an angle of tan^{-1}(1/5) to the horizontal. When the reservoir is full the diameter of the water surface is 50m. Discharge from the reservoir takes place through a pipe of diameter 0.65m, the outlet being 4m below top water level. Determine the time for the water level to fall 2m assuming the discharge to be $0.75a\sqrt{2gH}$ cumecs where a is the cross sectional area of the pipe in m^2 and H is the head of water above the outlet in m.
[1325 seconds]

4.2
A rectangular swimming pool is 1m deep at one end and increases uniformly in depth to 2.6m at the other end. The pool is 8m wide and 32m long and is emptied through an orifice of area 0.224m^2, at the lowest point in the side of the deep end. Taking C_d for the orifice as 0.6, find, from first principles,
a) the time for the depth to fall by 1m b) the time to empty the pool completely.
[299 second, 662 seconds]

4.3
A vertical cylindrical tank 2m diameter has, at the bottom, a 0.05m diameter sharp edged orifice for which the discharge coefficient is 0.6.
a) If water enters the tank at a constant rate of 0.0095 cumecs find the depth of water above the orifice when the level in the tank becomes stable.
b) Find the time for the level to fall from 3m to 1m above the orifice when the inflow is turned off.
c) If water now runs into the tank at 0.02 cumecs, the orifice remaining open, find the rate of rise in water level when the level has reached a depth of 1.7m above the

orifice.
[a) 3.314m, b) 881 seconds, c) 0.252m/min]

4.4
A horizontal boiler shell (i.e. a horizontal cylinder) 2m diameter and 10m long is half full of water. Find the time of emptying the shell through a short vertical pipe, diameter 0.08m, attached to the bottom of the shell. Take the coefficient of discharge to be 0.8.
[1370 seconds]

4.5
Two cylinders standing upright contain liquid and are connected by a submerged orifice. The diameters of the cylinders are 1.75m and 1.0m and of the orifice, 0.08m. The difference in levels of the liquid is initially 1.35m. Find how long it will take for this difference to be reduced to 0.66m if the coefficient of discharge for the orifice is 0.605. (Work from first principles.)
[30.7 seconds]

4.6
A rectangular reservoir with vertical walls has a plan area of 60000m^3. Discharge from the reservoir take place over a rectangular weir. The flow characteristics of the weir is $Q = 0.678\ H^{3/2}$ cumecs where H is the depth of water above the weir crest. The sill of the weir is 3.4m above the bottom of the reservoir. Starting with a depth of water of 4m in the reservoir and no inflow, what will be the depth of water after one hour?
[3.98m]

Notches and weirs

5.1
Deduce an expression for the discharge of water over a right-angled sharp edged V-notch, given that the coefficient of discharge is 0.61.
A rectangular tank 16m by 6m has the same notch in one of its short vertical sides. Determine the time taken for the head, measured from the bottom of the notch, to fall from 15cm to 7.5cm.
[1399 seconds]

5.2
Derive an expression for the discharge over a sharp crested rectangular weir. A sharp edged weir is to be constructed across a stream in which the normal flow is 200 litres/sec. If the maximum flow likely to occur in the stream is 5 times the normal flow then determine the length of weir necessary to limit the rise in water level to 38.4cm above that for normal flow. $C_d=0.61$.
[1.24m]

5.3
Show that the rate of flow across a triangular notch is given by $Q=C_dKH^{5/2}$ cumecs, where C_d is an experimental coefficient, K depends on the angle of the notch, and H is the height of the undisturbed water level above the bottom of the notch in metres. State the reasons for the introduction of the coefficient.

Water from a tank having a surface area of $10m^2$ flows over a 90 notch. It is found that the time taken to lower the level from 8cm to 7cm above the bottom of the notch is 43.5seconds. Determine the coefficient C_d assuming that it remains constant during his period.
[0.635]

5.4
A reservoir with vertical sides has a plan area of $56000m^2$. Discharge from the reservoir takes place over a rectangular weir, the flow characteristic of which is $Q=1.77BH^{3/2}$ m^3/s. At times of maximum rainfall, water flows into the reservoir at the rate of $9m^3/s$. Find a) the length of weir required to discharge this quantity if head must not exceed 0.6m; b) the time necessary for the head to drop from 60cm to 30cm if the inflow suddenly stops.
[10.94m, 3093seconds]

5.5
Develop a formula for the discharge over a 90 V-notch weir in terms of head above the bottom of the V.
A channel conveys 300 litres/sec of water. At the outlet end there is a 90 V-notch weir for which the coefficient of discharge is 0.58. At what distance above the bottom of the channel should the weir be placed in order to make the depth in the channel 1.30m? With the weir in this position what is the depth of water in the channel when the flow is 200 litres/sec?
[0.755m, 1.218m]

5.6
Show that the quantity of water flowing across a triangular V-notch of angle 2θ is
$$Q = C_d \frac{8}{15}\tan\theta\sqrt{2g}H^{5/2}$$
. Find the flow if the measured head above the bottom of the V is 38cm, when θ=45 and C_d=0.6. If the flow is wanted within an accuracy of 2%, what are the limiting values of the head.
[$0.126m^3/s$, 0.377m, 0.383m]