FLIUD KINEMATICS AND FLUID DYNAMICS 12

Fluid Kinematics - Flow visualization - lines of flow - types of flow - velocity field and acceleration - continuity equation (one and three dimensional differential forms)- Equation of streamline - stream function - velocity potential function - circulation - flow net – fluid dynamics - equations of motion - Euler's equation along a streamline - Bernoulli's equation – applications - Venturi meter, Orifice meter, Pitot tube - dimensional analysis - Buckingham's π theorem- applications - similarity laws and models.

Concepts of Fluid Flow

Objectives

- Introduce concepts necessary to analyse fluids in motion
- Identify differences between Steady/unsteady uniform/non-uniform compressible / incompressible flow
- Demonstrate streamlines and stream tubes
- Introduce the Continuity principle through conservation of mass and control volumes
- Derive the Euler's equation and Bernoulli (energy) equation
- Demonstrate practical uses of the Bernoulli and continuity equation in the analysis of flow

Fluid dynamics: The analysis of fluid in motion.

The motion of a fluid is usually complex. The study of static fluid (fluid at rest) was simplified by the absence of shear forces. But when a fluid flows over a solid surface or other boundary, whether stationary or moving, a velocity gradient is created at right angles to the boundary. The resulting change of velocity from layer to layer of fluid flowing parallel to the boundary gives rise to shear stresses in the fluid. The motion of the fluid particles is controlled by their inertia and the effect of the shear stresses exerted by the surrounding fluid. The resulting motion is not always easy to solve mathematically and it is often necessary to supplement theory by experiments.

Fluid motion can be predicted in the same way as the motion of solids by use of the fundamental laws of physics and the physical properties of the fluid.

When a force is applied, its behaviour can be predicted from Newton's laws, which state:

1. A body will remain at rest or in a state of uniform motion in a straight line until acted upon by an external force.
2. The rate of change of momentum of a body is proportional to the force applied and takes place in the direction of action of that force.
3. Action and reaction are equal and opposite.

Some fluid flow is very complex: e.g. flow behind a car; waves on beaches; hurricanes and tornadoes; any other atmospheric phenomenon

All can be analysed with varying degrees of success (in some cases hardly at all!).

There are many common situations which analysis gives very accurate predictions.

Flow Classification

- ***uniform flow**:* Flow conditions (velocity, pressure, cross-section or depth) are the same at every point in the fluid.
- ***non-uniform:*** Flow conditions are not the same at every point.
- ***steady**: Flow* conditions may differ from point to point but DO NOT change with time.
- ***unsteady**:* Flow conditions change with time at any point.

Fluid flowing under normal circumstances - a river for example - conditions vary from point to point we have non-uniform flow. If the conditions at one point vary as time passes then we have unsteady flow.

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

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

Frames of Reference

Whether a given flow is described as steady or unsteady will depend upon the situation of the observer. This is because the motion is relative and only can be described by a frame of reference - determined by observer.
Suppose there is a motion of fluid particles. If we observe the motion of fluid with respect to a reference system fixed relative to the particles then the frame of reference is known as Eulerian method of analysis. If the reference system moves with the particles then the system is known as Lagrangian method of analysis.

Compressible or Incompressible

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

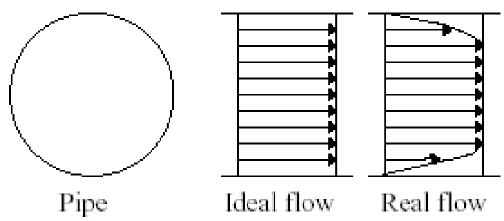
Three-dimensional flow

In general fluid flow is three-dimensional.
Pressures and velocities and other flow properties change in all directions.
In many cases the greatest changes only occur in two directions or even only in one.
Changes in the other direction can be effectively ignored making analysis much more simple.

Flow is *one dimensional* if the flow parameters (such as velocity, pressure, depth etc.) vary only in the direction of flow not across the cross-section.

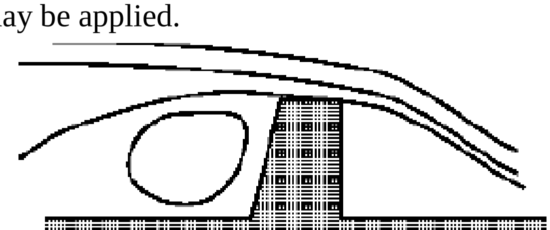
The flow may be unsteady with the parameters varying in time but not across the cross-section e.g. Flow in a pipe.

Note that since flow must be zero at the pipe wall - yet non-zero in the centre – there is a difference of parameters across the cross-section. Should this be treated as two-dimensional flow?
Possibly - but it is only necessary if very high accuracy is required. A correction factor is then usually applied.



One dimensional flow in a pipe.

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

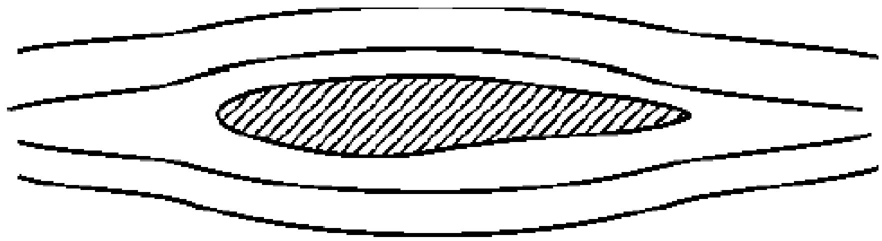


Two-dimensional flow over a weir.

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

Streamlines and streamtubes

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



Streamlines around a wing shaped body

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

- *Close to a solid boundary streamlines are parallel to that boundary*

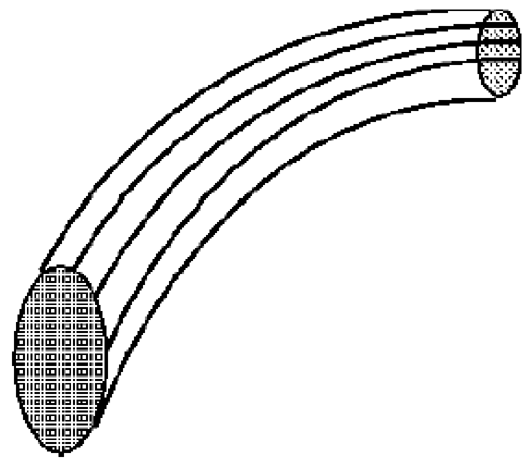
At all points the direction of the streamline is the direction of the fluid velocity: this is how they are defined. Close to the wall the velocity is parallel to the wall so the streamline is also parallel to the wall.

It is also important to recognise that the position of streamlines can change with time - this is the case in unsteady flow. In steady flow, the position of streamlines does not change.

Some things to know about streamlines

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

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



A Streamtube

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

A two dimensional version of the streamtube

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

Path line

If the individually particle of fluid is coloured, or otherwise rendered visible, it will describe a path line, which is the trace showing the position at successive intervals of time of a particle which started from a given point.

Streak line or Filament line

If the flow pattern is made visible by injecting a stream of dye into a liquid or smoke into a gas, the result will be a streak line. It gives an instantaneous picture of the positions of all the particles which have passed through a particular point. Since the flow pattern may vary from moment to moment, a streak line will not necessarily be the same as a path line.

For steady flow, stream line, path line and streak line will be the same.

Flow rate.

Mass flow rate

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

$$\dot{m} = \frac{dm}{dt} = \frac{mass}{time\ taken\ to\ accumulate\ this\ mass}$$

Volume flow rate - Discharge.

More commonly we need to know the volume flow rate - this is more commonly known as *discharge.* (It is also commonly, but inaccurately, simply called flow rate). The symbol normally used for discharge is Q.
The discharge is the volume of fluid flowing per unit time. Multiplying this by the density of the fluid gives us the mass flow rate.

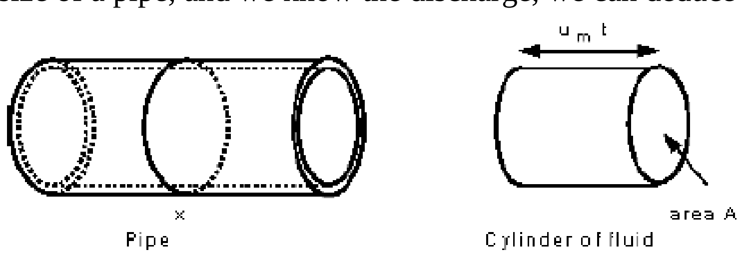
$$discharge, Q = \frac{Volume\ of\ fluid}{time}$$

An important aside about units should be made here:

As has already been stressed, we must always use a consistent set of units when applying values to equations. It would make sense therefore to always quote the values in this consistent set. This set of units will be the SI units. Unfortunately, and this is the case above, these actual practical values are very small or very large (0.001008m^3/s is very small). These numbers are difficult to imagine physically. In these cases it is useful to use *derived units*, and in the case above the useful derived unit is the litre. ($1 litre = 1.0 \times 10^{-3}\ m^3$). So the solution becomes 1008 *l/s*. It is far easier to imagine 1 litre than $1.0 \times 10^{-3}\ m^3$. Units must always be checked, and converted if necessary to a consistent set before using in an equation.

Discharge and mean velocity.

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



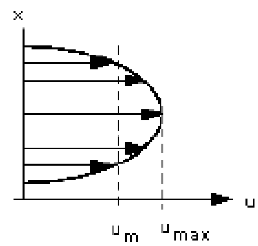
Discharge in a pipe

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

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

$$Q = A\, u_m$$

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



A typical velocity profile across a pipe

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

Velocity and Acceleration

Let **V** be the velocity vector and u, v, w are its component in x, y and z direction. So

$$u = f_1(x, y, z, t)$$
$$v = f_2(x, y, z, t)$$
$$w = f_3(x, y, z, t)$$

If a_x, a_y, a_z are the acceleration in x, y and z direction respectively, then by chain rule

$$a_x = \frac{du}{dt} = \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} + \frac{\partial u}{\partial z}\frac{dz}{dt} + \frac{\partial u}{\partial t} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}$$

$$a_y = \frac{dv}{dt} = \frac{\partial v}{\partial x}\frac{dx}{dt} + \frac{\partial v}{\partial y}\frac{dy}{dt} + \frac{\partial v}{\partial z}\frac{dz}{dt} + \frac{\partial v}{\partial t} = u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t}$$

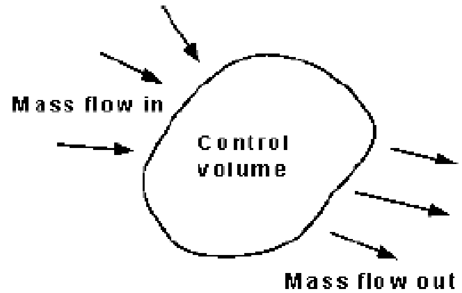
$$a_z = \frac{dw}{dt} = \frac{\partial w}{\partial x}\frac{dx}{dt} + \frac{\partial w}{\partial y}\frac{dy}{dt} + \frac{\partial w}{\partial z}\frac{dz}{dt} + \frac{\partial w}{\partial t} = u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t}$$

$\frac{\partial u}{\partial t}, \frac{\partial v}{\partial t}, \frac{\partial w}{\partial t}$ are called the local or temporal acceleration.

The other terms are known as convective acceleration.

Continuity

Matter cannot be created or destroyed - (it is simply changed in to a different form of matter). This principle is known as the *conservation of mass* and we use it in the analysis of flowing fluids. The principle is applied to fixed volumes, known as control volumes (or surfaces), like that in the figure below:



An arbitrarily shaped control volume.

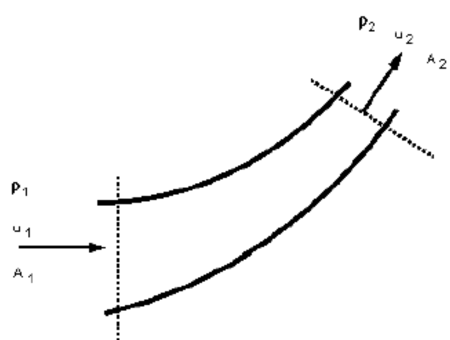
For any control volume the principle of *conservation of mass* says

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

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

For steady flow Mass entering per unit time = Mass leaving per unit time

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



A streamtube

We can write

Mass entering per unit time at end 1 = mass leaving per unit time at end 2

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

or in terms of mean velocities

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

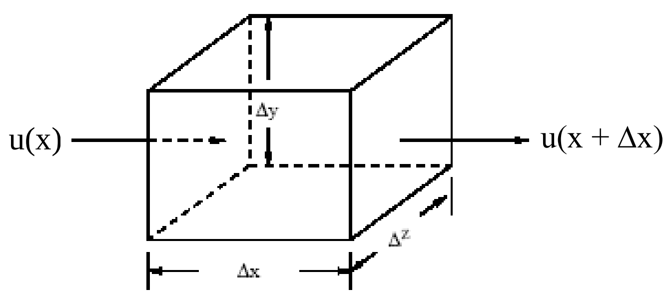
When the fluid can be considered incompressible, i.e. the density does not change, $\rho_1 = \rho_2 = \rho$ so (dropping the *m* subscript)

$$A_1\, u_1 = A_2\, u_2 = Q$$

This is the form of the continuity equation most often used.

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

Continuity Equation in Three Dimensions



Consider a control volume with sides Δx, Δy, Δz in x, y and z directions respectively. Considering the flow in x direction

Mass flow through the left face in unit time = $\rho u\, \Delta y \Delta z$.

In general both mass density ρ and velocity u will change in the x direction. So

Mass flow through the right face in unit time = $\left(\rho u + \frac{\partial}{\partial x}(\rho u)\Delta x \right)\Delta y\, \Delta z$

Thus,

Net outflow in unit time in x direction = $\frac{\partial}{\partial x}(\rho u)\Delta x\, \Delta y\, \Delta z$

Similarly,

Net outflow in unit time in y direction = $\frac{\partial}{\partial y}(\rho v)\Delta x\, \Delta y\, \Delta z$

and

Net outflow in unit time in z direction = $\frac{\partial}{\partial z}(\rho w)\Delta x\, \Delta y\, \Delta z$

Therefore,

Total net outflow in unit time = $\left(\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) \right)\Delta x\, \Delta y\, \Delta z$

Also,

Change of mass in control volume per unit volume = $\frac{\partial \rho}{\partial t}\Delta x\, \Delta y\, \Delta z$

As per continuity

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

or

Total out flow in unit time + Increase of mass in the control volume per unit time = 0

or

$$\frac{\partial \rho}{\partial t}\Delta x\,\Delta y\,\Delta z+\left(\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)\right)\Delta x\,\Delta y\,\Delta z=0$$

i.e.

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0$$

For steady flow

$$\frac{\partial \rho}{\partial t}=0$$

Hence, continuity equation becomes

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0$$

For incompressible flow density is constant and hence,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

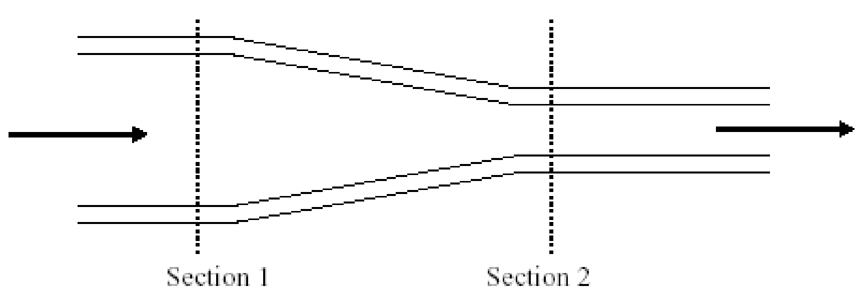
For two dimensions the equation further simplifies to

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

Some example applications

We can apply the principle of continuity to pipes with cross sections which change along their length.

Consider the diagram below of a pipe with a contraction:



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

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

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

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

Discharge at section 1 = Discharge at section 2

$$Q_1 = Q_2$$

$$A_1 u_1 = A_2 u_2$$

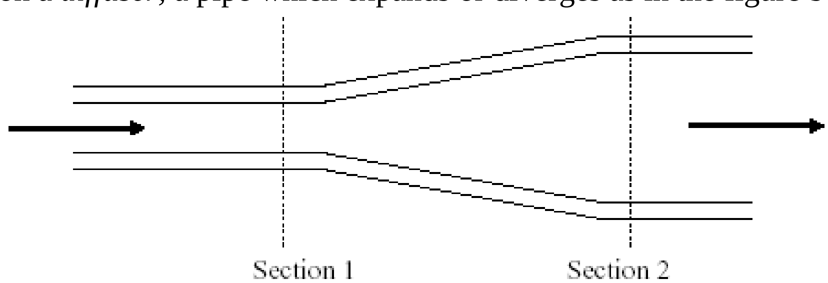
For example, if area $A_1 = 10\ m^2$ and $A_2 = 3\ m^2$ and the upstream velocity $u_1 = 2.1$ m/s, then the downstream mean velocity can be calculated by

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

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

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

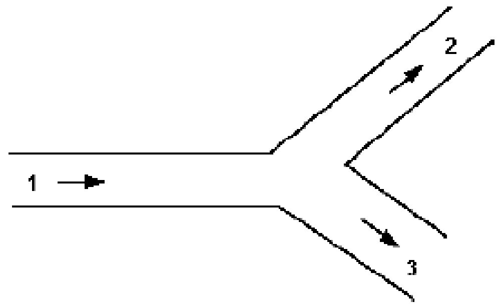
Now try this on a *diffuser*, a pipe which expands or diverges as in the figure below,



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

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

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



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

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

When the flow is incompressible $\rho_1 = \rho_2 = \rho_3 = \rho$

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

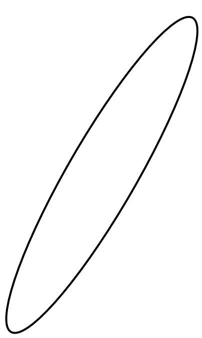
Example: If pipe 1 diameter = 50 mm, mean velocity = 2 m/s, pipe 2 diameter = 40 mm and takes 30% of total discharge, pipe 3 diameter = 60 mm. What are the values of discharge and mean velocity in each pipe?

Two-dimensional Ideal Flow

An ideal fluid is a purely hypothetical fluid which is assumed to have no viscosity and no compressibility. All real fluids posses viscosity and are in some degree compressible. Nevertheless there are many instances in which behaviour of real fluids quite closely approaches that of ideal fluid. Adjacent to the solid wall there is a thin layer in which viscosity effects are predominant and real fluid treatment in this thin layer is necessary. Outside this thin layer, viscous effects are negligible and flow is similar to that of inviscid fluid.

For two dimensional flow, let co-ordinate axes be OX and OY and let u = velocity component parallel to OX and v = velocity component parallel to OY. **q** is the velocity vector such that $q^2 = u^2 + v^2$.

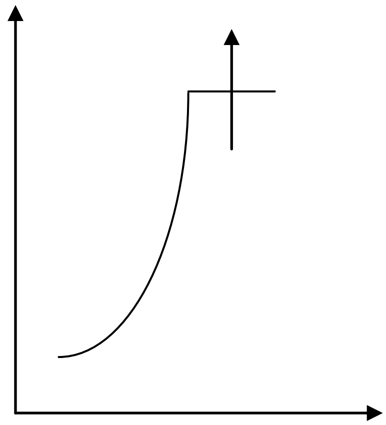
Stream Function



We shall now consider the motion of an incompressible fluid in two dimensions. The motion in all planes parallel to OXY is same as in this plane and there is no component of velocity perpendicular to this plane. Let us consider a layer of fluid lying between the plane OXY and a parallel plane at unit distance from it. Let A be a fixed point and P is any point in the plane. These points are joined by a pair of curves AMP and ANP, which together form a closed region. Since the fluid is incompressible, we have

Flux across ANP = flux across AMP

where we adopt the convection that the flux is positive in the sense from right to left across the curve. Now regard AMP is a fixed curve while the curve ANP is allowed to vary its situation. The flux across the curve ANP is fixed so



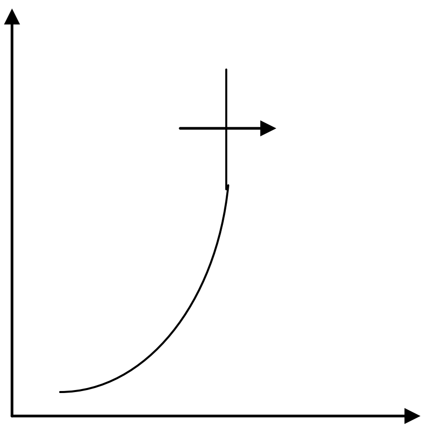
long as its end points A and P are given. Since, A is a fixed point, the flux across ANP is thus a function co-ordinates x,y of point P. This function is called the stream function and represented by $\psi(x,y)$. When the flow is steady ψ is independent of time but for unsteady flow it becomes a function of three variables x, y and t.

Let us calculate the increment $d\psi$ of stream function in passing from P to P' where PP' is parallel to OX and equal to infinitesimal distance dx. We may pass from fixed point A to P' by any path, so we choose the path AMPP'. The values of stream function at P' is thus

$$\Psi(x+dx,y) = \text{flux across AMP} + \text{flux across PP'}$$
$$= \psi(x,y) + v\,dx$$

For component of velocity normal to PP' is v. Hence we have

$$\Psi(x+dx,y) - \psi(x,y) = v\,dx$$



or $\frac{\partial \psi}{\partial x} dx = v dx$

i.e. $\frac{\partial \psi}{\partial x} = v$

If PP' is parallel to OY and equal to infinitesimal dy, then by similar argument,

$\Psi(x,y+dy) - \psi(x,y) = -u\,dx$

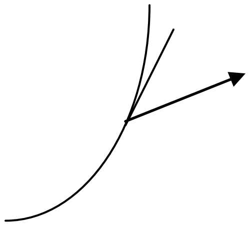
or $\frac{\partial \psi}{\partial y} = -u$

Thus, whenever we know Ψ, we may obtain the two velocity components of velocity from

$$u = -\frac{\partial \psi}{\partial y} \quad \text{and} \quad v = \frac{\partial \psi}{\partial x}$$

Physically, the stream function is the volume rate of flow per unit distance normal to the plane motion between a stream line in a fluid and an arbitrary base streamline. The stream function is constant along any streamline at the instant considered.

Flow along a curve, Circulation



APB is some curve joining points A and B and lying wholly within the fluid. Let PP' be an element of the curve of length ds and let θ be the angle between the tangent at P and the velocity of the fluid there at the instant considered. Then the flow from A to B along the curve is defined by the integral $\int_A^B q\cos\theta\, ds$. If we represent the flow from A to B as $\overrightarrow{AB}$ then

$$\overrightarrow{AB} = \int_A^B q \cos\theta \, ds$$

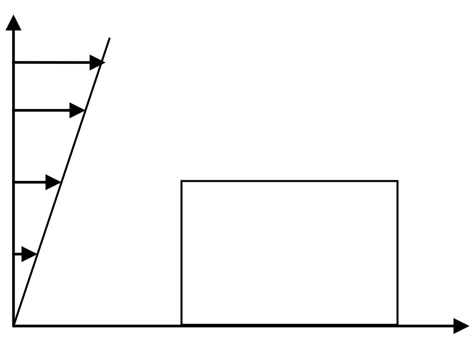
It is often convenient to represent the flow along a line ABCDE by symbol $\overrightarrow{ABCDE}$. Then by definition

$$\overrightarrow{ABCDE} = \overrightarrow{AB} + \overrightarrow{BC} + \overrightarrow{CD} + \overrightarrow{DE}$$

It is also evident that

$$\overrightarrow{AB} = -\overrightarrow{BA}$$

In general, AB has a definite value only when the curve joining A and B is specified. When A and B coincide, and the curve connecting them is accordingly a closed circuit, the flow is called the ***circulation*** in the circuit. The circulation in a given circuit may differ from zero while no particles of the fluid 'circulates' i.e. describes a closed curve. To illustrates this, let us consider the counterclockwise circulation in the rectangular circuit ABCD, when the velocity at the instant considered is given by
U = by, v = 0, w = 0



Then we have $\overrightarrow{AB}$ zero since the fluid is at rest on AB, and $\overrightarrow{BC}$ is zero since the component of velocity along BC is zero. On CD we have q = uh and cosθ = -1, since the positive sense of the tangent to the circuit is opposite to that of motion. Hence

$$\overrightarrow{CD} = -\,bkh$$

and $\overrightarrow{DA}$ is zero since the velocity is perpendicular to DA.

Finally the circulation is

$$\Gamma = \overrightarrow{AB} + \overrightarrow{BC} + \overrightarrow{CD} + \overrightarrow{DA} = -\,bkh\ .$$

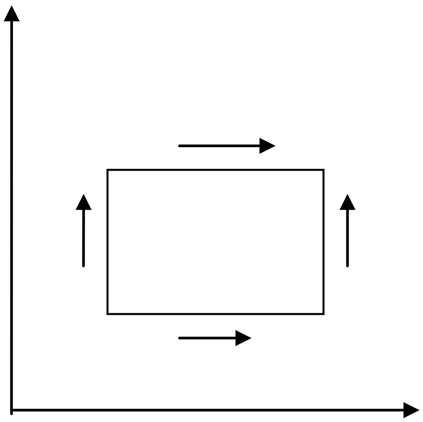
It is notable that the circulation is here proportional to the area of the circuit. This is always true for sufficiently small circuits lying in a given plane and enclosing a fixed point.

We shall now show that when a given circuit is divided into a pair of circuits, the circulation in the given circuit is equal to the sum of the circulations in pair of circuits. In the fig., the points A and B on the given circuit are joined by an arbitrary curve, which need not be the plane but must lie within the fluid. Let Γ be the circulation in the given circuit ADBCA while Γ_1 and Γ_2 are the circulations in the circuits ABCA and ADBA respectively, where positive sense of circulation is same for all circuits. Then

$$\Gamma_1 + \Gamma_2 = \left(\overrightarrow{AB} + \overrightarrow{BC} + \overrightarrow{CA}\right) + \left(\overrightarrow{AD} + \overrightarrow{DB} + \overrightarrow{BA}\right)$$
$$= \overrightarrow{AD} + \overrightarrow{DB} + \overrightarrow{BC} + \overrightarrow{CA} = \Gamma$$
$$\therefore \quad \Gamma_1 + \Gamma_2 = \Gamma$$

Evidently we can now carry on subdivision of the circuits as far as we please and we shall always have Γ equal to the sum of the circulations in the sub-circuits.

Vorticity



An example of the elementary circuit arising from the subdivision of a larger one we consider the elementary rectangle δx ×δy in size. The velocities along the sides have the average values shown, the arrows in each case indicating the direction.

Now $\overrightarrow{AB} = u\,\delta x$.

and $\overrightarrow{CD} = -\left(u + \dfrac{\partial u}{\partial y}\delta y\right)\delta x$

$$\therefore\ \overrightarrow{AB} + \overrightarrow{CD} = -\frac{\partial u}{\partial y}\delta x\,\delta y$$

Similarly, $\overrightarrow{BC} + \overrightarrow{DA} = \dfrac{\partial v}{\partial x}\delta x\,\delta y$

$$\therefore\ \Gamma = \overrightarrow{AB} + \overrightarrow{BC} + \overrightarrow{CD} + \overrightarrow{DA} = \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\delta x\,\delta y$$

The expression in the bracket is called the component of ***vorticity*** of the fluid along the normal to the plane of the circuit ABCDA and is represented by the symbol ξ (zeta). Now the vorticity at a point can be defined as the ratio of circulation round an infinitesimal circuit there to the area of the circuit (in case of two dimensional flow).

$$\therefore\ \text{Vorticity}\ ,\ \xi = \frac{\text{Circulation}}{\text{Area}} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\ .$$

It can be shown that the vorticity components are twice the rate of rotation of the fluid element.

Irrotational Flow

If the vorticity is zero at all points in a region, then the flow in the region is said to be ***irrotational***. Hence, irrotational flow means that there is zero angular velocity of the fluid element about their centre. Since the fluid has zero viscosity, no tangential or shear stresses may be applied to the fluid elements. The pressure forces act through the centre of the elements and can cause no rotation; therefore no torque may be applied to the fluid elements. If a fluid element is initially at rest, it cannot be set in rotation; if it is rotating, the rotation cannot be changed.

For 2 dimensional flow to be irrotational we have

$$\xi = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0$$

Now $\quad u = -\dfrac{\partial \psi}{\partial y} \quad$ and $\quad v = \dfrac{\partial \psi}{\partial x}$

So putting the values of u and v we get

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0 \qquad \text{i.e.} \quad \nabla^2 \psi = 0$$

Velocity Potential

It is convenient to introduce a new function φ, which is the velocity potential such that

$$-\frac{\partial \varphi}{\partial x} = u \qquad \text{(negative sign by convention).}$$

$$-\frac{\partial \varphi}{\partial y} = v$$

$$-\frac{\partial \varphi}{\partial z} = w$$

Now $\frac{\partial u}{\partial y} = -\frac{\partial^2 \varphi}{\partial x \, \partial y}$

$$\frac{\partial v}{\partial x} = -\frac{\partial^2 \varphi}{\partial x \, \partial y}$$

$\therefore \quad \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = -\frac{\partial^2 \varphi}{\partial x \, \partial y} + \frac{\partial^2 \varphi}{\partial x \, \partial y} = 0$

Hence for velocity potential to exist, the flow must be irrotational.
Now from the continuity equation for incompressible fluid

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

or $\frac{\partial^2 \varphi}{\partial x^2} + \frac{\partial^2 \varphi}{\partial y^2} + \frac{\partial^2 \varphi}{\partial z^2} = 0$

or $\nabla^2 \varphi = 0$

This is Laplace equation. Any function which satisfies Laplace equation is called a harmonic function. Hence we can see that the velocity potential of an incompressible flow is necessarily a harmonic function.
For two dimensional flow, the equation of continuity in terms of velocity potential for incompressible flow becomes

$$\frac{\partial^2 \varphi}{\partial x^2} + \frac{\partial^2 \varphi}{\partial y^2} = 0$$

Since the flow is irrotational we have $\xi = 0$
The stream function for two dimensional flow can written as

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = 0$$

Thus both the velocity potential and stream functions are here harmonic functions.

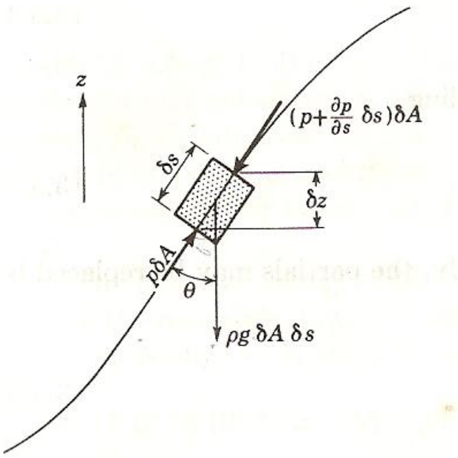
Flownet

In two-dimensional flow, the flownet is of great benefit. The line given by $\varphi(x,y)$ = constant is called an ***equipotential line***. It is a line along which the value of φ (the velocity potential) does not change.
Let u_s be the component of fluid velocity in the direction of length ds with direction cosines l,m,n. Then

$$u_s = lu + mv + nw = \left(l\frac{\partial\varphi}{\partial x} + m\frac{\partial\varphi}{\partial y} + n\frac{\partial\varphi}{\partial z} \right) = -\frac{\partial\varphi}{\partial s}$$

Now let ds lie in a surface on which φ is constant. Then $\frac{\partial\varphi}{\partial s}$ is zero and this proves that there is no velocity component tangent to equipotential line. Therefore, the velocity vector must be everywhere normal to an equipontetial line. The line ψ (x,y) = constant is a streamline and is everywhere tangent to velocity vector. Streamline and equipotential lines are therefore orthogonal i.e. they intersect at right angles. A flownet is composed of a family of equipotential lines and a corresponding family of stream lines with the constant varying in arithmetical progression. In steady flow when boundaries are stationary, the boundaries are themselves become part of the flownet, as they are streamlines.

Euler Equation of motion in a a Streamline



Assumptions:-

1. Motion along a streamline
2. Frictionless fluid, and
3. Steady flow

Consider a prismatic element of mass $\rho\delta A\delta s$ is moving in positive s direction. Since the fluid is frictionless, no shear force is acting on the fluid.
Force on the upstream face = $p\,\delta A$

Force on the downstream face = $\left(p + \frac{\partial p}{\partial s}\delta s\right)\delta A$

Body force acting in s direction = $\rho g\delta A\delta s\cos\theta$.

From Newton's second law

$$\Sigma F = \delta m\, a_s \quad \text{we get}$$

$$p\,\delta A - \left(p + \frac{\partial p}{\partial s}\delta s\right)\delta A - \rho g\,\delta A\,\delta s\cos\theta = \rho\delta A\,\delta s\, a_s$$

where a_s is the acceleration of the fluid particle. Dividing throughout by $\rho\delta A\delta s$ we get

$$\frac{1}{\rho}\frac{\partial p}{\partial s} + g\cos\theta + a_s = 0$$

Now, $\cos\theta = \frac{\delta z}{\delta s} = \frac{\partial z}{\partial s}$ and $v = v(s,t)$

$$dv = \frac{\partial v}{\partial s}ds + \frac{\partial v}{\partial t}dt$$

$$a_s = \frac{dv}{dt} = \frac{\partial v}{\partial s}\frac{ds}{dt} + \frac{\partial v}{\partial t} = v\frac{\partial v}{\partial s} + \frac{\partial v}{\partial t}$$

$$\frac{1}{\rho}\frac{\partial p}{\partial s} + g\frac{\partial z}{\partial s} + v\frac{\partial v}{\partial s} + \frac{\partial v}{\partial t} = 0$$

Since for steady flow $\frac{\partial v}{\partial t} = 0$

We get,
$$\frac{1}{\rho}\frac{\partial p}{\partial s} + v\frac{\partial v}{\partial s} + g\frac{\partial z}{\partial s} = 0$$

Since p, z and v are functions of s only, we can write

$$\frac{dp}{\rho} + vdv + gdz = 0$$

This is the Euler's equation of motion in a streamline.

Bernoulli's Equation

The Euler's equation can be integrated provided relation between p and ρ is known. For a constant density ρ is constant and hence integrating we get,

$$\frac{p}{\rho} + \frac{v^2}{2} + gz = const.$$

Since, γ = ρg we get,

$$\frac{p_1}{\gamma} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\gamma} + \frac{v_2^2}{2g} + z_2 = const.$$

The each term of Bernoulli's equation can be interpreted as a form of energy.

$\frac{p}{\rho} \rightarrow$ Flow work or flow energy per unit mass.

$\frac{v^2}{2} \rightarrow$ Kinetic energy per unit mass.

$gz \rightarrow$ Potential energy per unit mass.

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

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 = const.$$

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

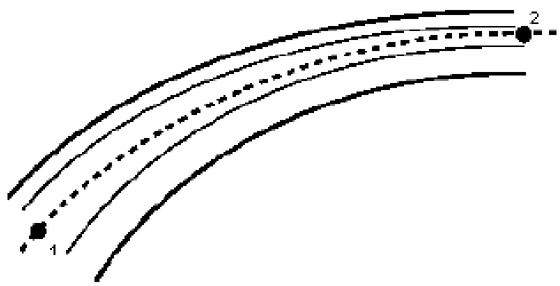
- Flow is steady;
- Density is constant (which also means the fluid is incompressible);
- Friction losses are negligible.
- The equation relates the states at two points along a single streamline, (not conditions on two different streamlines).

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.

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

$$\frac{p}{\rho g} + \frac{v^2}{2g} + z = H = const.$$

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

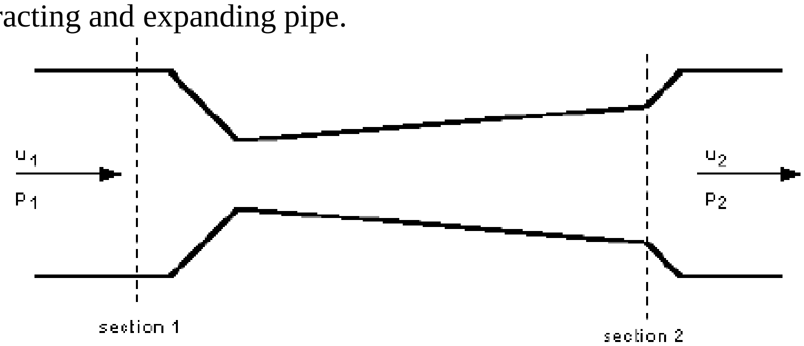
Or

$$\frac{p_1}{\rho g}+\frac{v_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{v_2^2}{2g}+z_2$$

This equation assumes no energy losses (e.g. from friction) or energy gains (e.g. from a pump) along the streamline.

An example of the use of the Bernoulli equation.

When the Bernoulli equation is combined with the continuity equation the two can be used to find velocities and pressures at points in the flow connected by a streamline.
Here is an example of using the Bernoulli equation to determine pressure and velocity at within a contracting and expanding pipe.



A contracting expanding pipe

A fluid of constant density ρ = 960 kg/m^3 is flowing steadily through the above tube. The diameters at the sections are d_1 =100mm and d_2 =80mm. The gauge pressure at 1 is p_1= 200 kN/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's equation becomes

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

But we do not know the value of u_2.We can calculate this from the continuity equation

$$A_1 u_1 = A_2 u_2$$

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

$$= 7.8125 \text{ m/s.}$$

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).

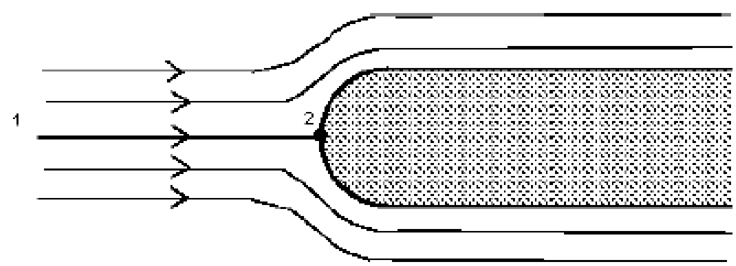
Here we have used both the Bernoulli equation and the Continuity principle together to solve the problem. Use of this combination is very common. We will be seeing this again frequently throughout the rest of the course.

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.

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 $z_1 = z_2$.

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

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

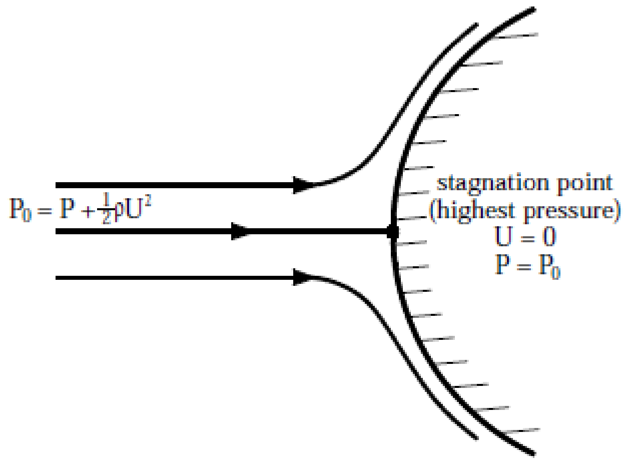
$$p_2 = p_1 + \frac{1}{2}\rho u_1^2$$

This increase in pressure which brings 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 = $\frac{u_1^2}{2g}$



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

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

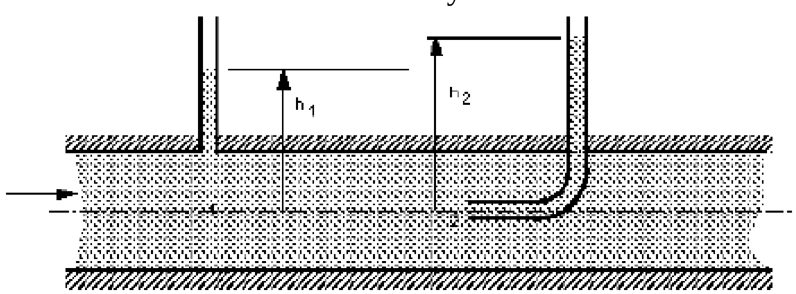
or in terms of head

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

Energy losses due to friction and the change in pressure imparted by pumps are often specified in terms of head. For pumps the rate of working i.e. ***power*** is given by

power = ρgQH

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,

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

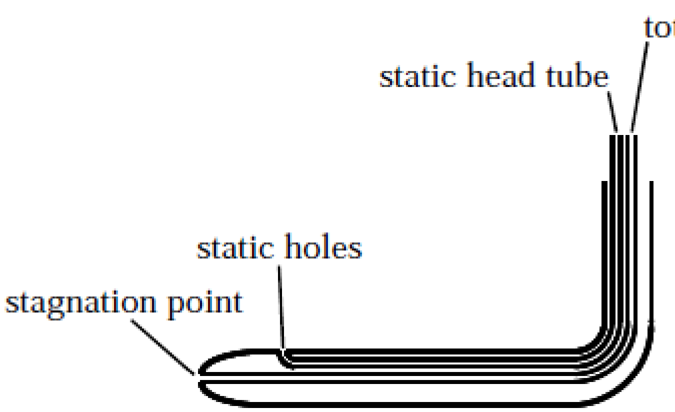
$$\rho g h_2 = \rho g h_1 + \frac{1}{2}\rho u_1^2$$

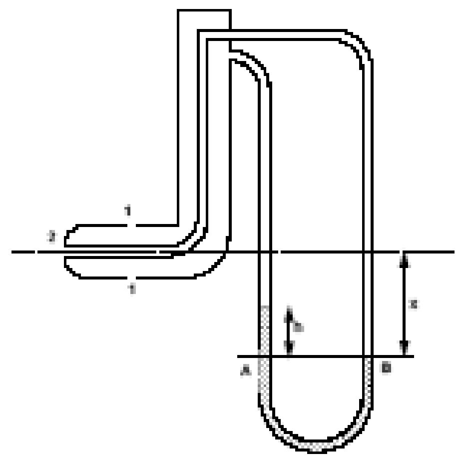
$$u = \sqrt{2g(h_2 - h_1)}$$

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

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)*.





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

$$p_B = p_2 + \rho g X$$

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

Now,

$$p_A = p_B$$

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

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

$$p_1 + hg(\rho_{man} - \rho) = p_1 + \frac{1}{2}\rho u_1^2$$

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

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

Venturi Meter

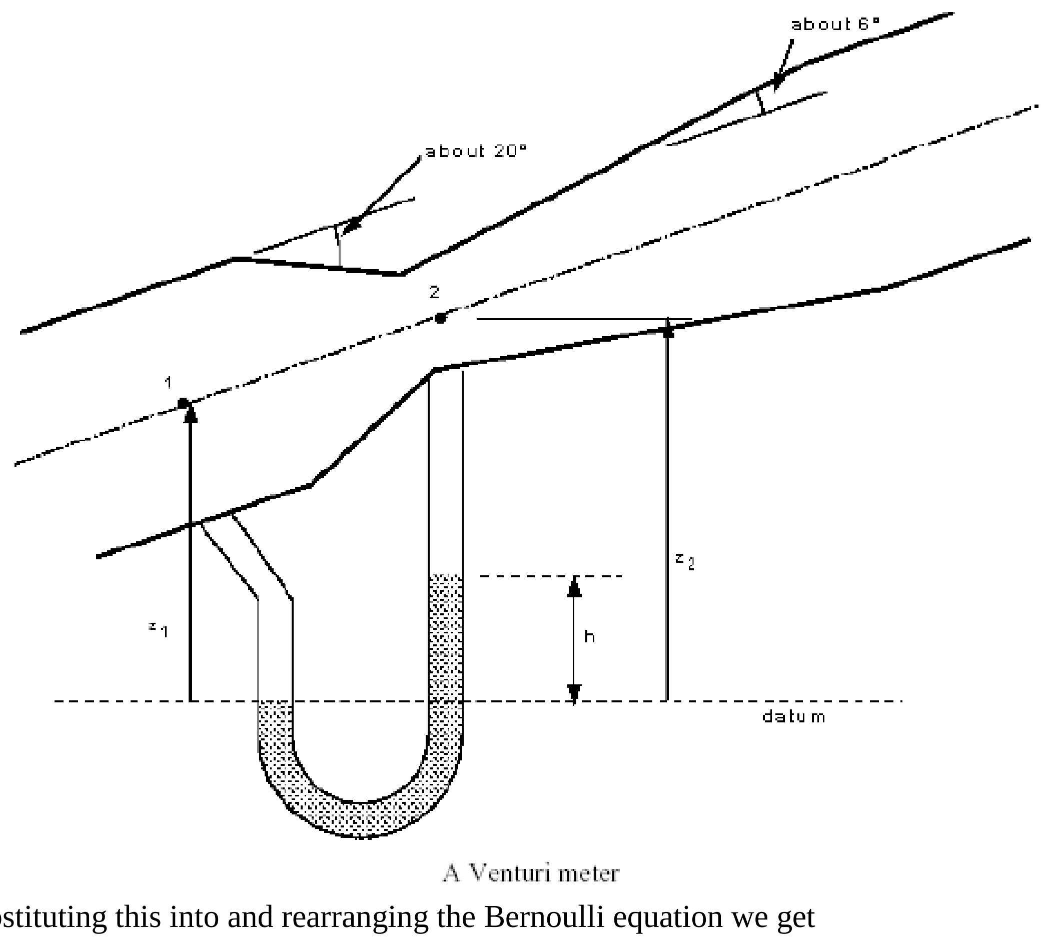
The Venturi meter is a device for measuring discharge in a pipe. It consists of a rapidly converging section which increases the velocity of flow and hence reduces the pressure. It then returns to the original dimensions of the pipe by a gently diverging 'diffuser' section. By measuring the pressure differences the discharge can be calculated. This is a particularly accurate method of flow measurement as energy losses are very small.
Applying Bernoulli's theorem along the streamline from point 1 to point 2 in the narrow throat of the Venturi meter, we have

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

By using the continuity equation we can eliminate velocity u_2

$$Q = u_1 A_1 = u_2 A_2$$

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



A Venturi meter

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 into 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 manometer readings

$$p_1 + \rho g z_1 = p_2 + \rho_{man}\, gh + \rho g(z_2 - h)$$

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

Thus the discharge can be expressed in terms of the manometer readings as:

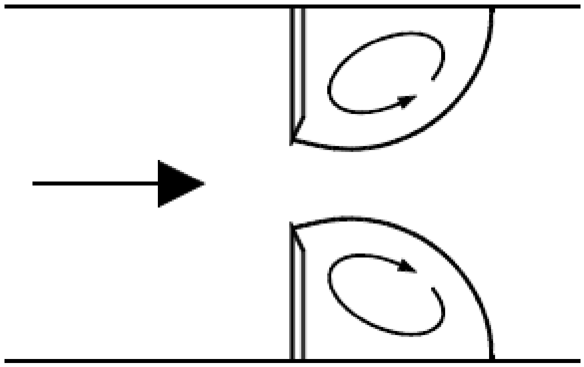
$$Q_{actual} = C_d A_1 A_2 \sqrt{\frac{2g\, h\left(\frac{\rho_{man}}{\rho} - 1\right)}{A_1^2 - A_2^2}}$$

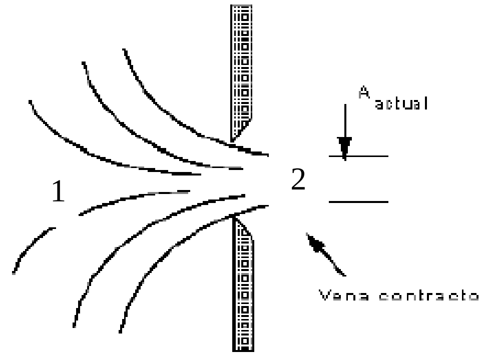
Notice how this expression does not include any terms for the elevation or orientation (z_1 or z_2) of the Venturi meter. This means that the meter can be at any convenient angle to function.

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%.

Flow Through Orifice in a Pipe

We are to consider the flow through square-edge orifice in a pipe. 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 to the jet at its vena contracta, point 2.

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

Now $z_1 = z_2$. So, we get

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

By using continuity equation we get $Q = A_1u_1 = A_2u_2$

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

So we get, $$u_2 = \sqrt{\frac{2(p_1 - p_2)/\rho}{\left[1 - \left(\frac{A_2}{A_1}\right)^2\right]}}$$

The actual area of the jet i.e. the area of the vena contracta A_2 is not known whereas the area of the orifice opening A_0 is known. Hence, it is customary to express A_2 in terms of A_0. We obtain this area by using a **coefficient of contraction, C_c** for the orifice which is the area of jet at vena contracta to area of orifice. So we get

$$u_2 = \sqrt{\frac{2(p_1 - p_2)/\rho}{\left[1 - C_c^2\left(\frac{A_0}{A_1}\right)^2\right]}}$$

So the discharge $Q = A_2 u_2 = C_c A_0 \sqrt{\dfrac{2(p_1 - p_2)/\rho}{\left[1 - C_c^2 \left(\dfrac{A_0}{A_1}\right)^2\right]}} = C_c A_0 \sqrt{\dfrac{2gh}{\left[1 - C_c^2 \left(\dfrac{A_0}{A_1}\right)^2\right]}}$

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

$$\therefore \qquad Q = \frac{C_v C_c A_0 \sqrt{2gh}}{\left[1 - C_c^2 \left(\dfrac{A_0}{A_1}\right)^2\right]^{\frac{1}{2}}} = \frac{C_d A_0 \sqrt{2gh}}{\left[1 - C_c^2 \left(\dfrac{A_0}{A_1}\right)^2\right]^{\frac{1}{2}}}$$

Where C_d is the coefficient of discharge = $\dfrac{\textit{Actual discharge}}{\textit{Theoretical Discharge}} = C_c \times C_v$

Dimensional Analysis

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

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

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

Dimensions and units

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

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

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

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

length	= L
mass	= M
time	= T
force	= F
temperature	= Θ

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

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

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

Dimensional Homogeneity

Any equation describing a physical situation will only be true if both sides have the same dimensions.

That is it must be **dimensionally homogenous**.

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

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

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

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

i.e. the units are consistent.

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

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

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

This property of dimensional homogeneity can be useful for:

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

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

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

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

1. obtain these powers

2. form the variables into several dimensionless groups

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

Buckingham's π theorems

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

This will be outlined below.

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

> 1st π theorem:
>
> A relationship between **m** variables (physical properties such as velocity, density etc.) can be expressed as a relationship between **m-n** *non-dimensional* groups of variables (called π groups), where **n** is the number of fundamental dimensions (such as mass, length and time) required to express the variables.

So if a physical problem can be expressed:

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

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

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

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

> 2nd π theorem
>
> Each π group is a function of **n** *governing* or *repeating variables* plus one of the remaining variables.

Choice of repeating variables

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

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

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

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

An example

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

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

n = 3 and m = 6

There are m - n = 3 π groups, so

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

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

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

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

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

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

And in terms of dimensions

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

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

for M: $0 = a_1 + 1$

$a_1 = -1$

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

$0 = 4 + b_1 + c_1$

for T: $0 = -b_1 - 2$

$b_1 = -2$

$c_1 = -4 - b_1 = -2$

Giving π_1 as

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

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

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

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

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

for M: $0 = a_2$

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

$0 = b_2 + c_2$

for T: $0 = -b_2 - 1$

$b_2 = -1$

$c_2 = 1$

Giving π_2 as

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

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

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

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

for M: $0 = a_3 + 1$

$a_3 = -1$

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

$b_3 + c_3 = -2$

for T: $0 = -b_3 - 1$

$b_3 = -1$

$c_3 = -1$

Giving π_3 as

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

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

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

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

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

This may also be written:

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

Wrong choice of physical properties.

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

Manipulation of the π groups

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

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

Then the following manipulations are permitted:

i. Any number of groups can be combined by multiplication or division to form a new group which replaces one of the existing. e.g. π_1 and π_2 may be combined to form $\pi_{1a} = \pi_1 / \pi_2$ so the defining equation becomes

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

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

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

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

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

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

In general the defining equation could look like

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

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

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} $\quad$ u: LT^{-1}

d: L $\quad$ 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] $\quad 0 = c_1 + 1$

$\quad c_1 = -1$

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

$\quad -2 = 3a_1 + b_1$

T] $\quad 0 = -a_1 - 1$

$\quad a_1 = -1$

$\quad b_1 = 1$

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

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

And the second group π_2 :

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

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

M] $\quad 0 = c_2 + 1$

$\quad c_2 = -1$

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

$\quad -2 = 3a_2 + b_2$

T] $\quad 0 = -a_2 - 2$

$\quad a_2 = -2$

$\quad b_2 = 4$

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

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

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

or

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

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

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

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

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

then we can say

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

or

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

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.

Geometric similarity

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

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

where λ_L is the scale factor for length.

For area

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

All corresponding angles are the same.

Kinematic similarity

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

i. If the paths of moving particles are geometrically similar

ii. If the rations of the velocities of particles are similar

Some useful ratios are:

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

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

Discharge $$\frac{Q_m}{Q_p} = \frac{L_m^3 / T_m}{L_p^3 / T_p} = \frac{\lambda_L^3}{\lambda_T} = \lambda_Q$$

This has the consequence that streamline patterns are the same.

Dynamic similarity

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

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

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

Models

When a hydraulic structure is build it undergoes some analysis in the design stage. Often the structures are too complex for simple mathematical analysis and a hydraulic model is build. Usually the model is less than full size but it may be greater. The real structure is known as the prototype. The model is usually built to an exact geometric scale of the prototype but in some cases - notably river model - this is not possible. Measurements can be taken from the model and a suitable scaling law applied to predict the values in the prototype.
To illustrate how these scaling laws can be obtained we will use the relationship for resistance of a body moving through a fluid.

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

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

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

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

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

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

Leading to π_1 as

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

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

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 $\pi_{2a.}$

So the defining equation for resistance to motion is

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

We can write

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

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

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

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

and for the prototype

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

Dividing these two equations gives

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

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

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

This assumption then allows the equation following to be written

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

Which gives this scaling law for resistance force:

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

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

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

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

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

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

Substituting this into the scaling law for resistance gives

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

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

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

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

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

Dynamically similar model examples

Example 1

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

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

$$Re_m = Re_p$$

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

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:

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

For dynamic similarity $Re_m = Re_p$, so

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

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

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

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

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

$$\rho_m = 20\rho_p$$

So the model velocity is found to be

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

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

The ratio of forces is found from

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

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

So the drag force on the prototype will be

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