CIVE1400: An Introduction to Fluid Mechanics

Unit 3: Fluid Dynamics

Dr P A Sleigh: P.A.Sleigh@leeds.ac.uk
Dr CJ Noakes: C.J.Noakes@leeds.ac.uk

January 2008

Module web site: **www.efm.leeds.ac.uk/CIVE/FluidsLevel1**

Fluid Dynamics

Objectives

1. Identify differences between:
   - steady/unsteady
   - uniform/non-uniform
   - compressible/incompressible flow

2. Demonstrate streamlines and stream tubes

3. Introduce the Continuity principle

4. Derive the Bernoulli (energy) equation

5. Use the continuity equations to predict pressure and velocity in flowing fluids

6. Introduce the momentum equation for a fluid

7. Demonstrate use of the momentum equation to predict forces induced by flowing fluids

Fluid dynamics:

The analysis of fluid in motion

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

Some fluid flow is very complex:
e.g.
- Spray 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

Fluid flow may be
classified under the following headings

uniform:
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 these four gives.

Steady uniform flow.
Conditions do not change with position in the stream or with time.
E.g. flow of water in a pipe of constant diameter at constant velocity.

Steady non-uniform flow.
Conditions change from point to point in the stream but do not change with time.
E.g. Flow in a tapering pipe with constant velocity at the inlet.

Unsteady uniform flow.
At a given instant in time the conditions at every point are the same, but will change with time.
E.g. A pipe of constant diameter connected to a pump pumping at a constant rate which is then switched off.

Unsteady non-uniform flow
Every condition of the flow may change from point to point and with time at every point.
E.g. Waves in a channel.

This course is restricted to Steady uniform flow
- the most simple of the four.

Compressible or Incompressible Flow?

All fluids are compressible - even water.
Density will change as pressure changes.

Under steady conditions
- provided that changes in pressure are *small - we* usually say the fluid is incompressible
- it has constant density.

Three-dimensional flow

In general fluid flow is three-dimensional.

Pressures and velocities 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.

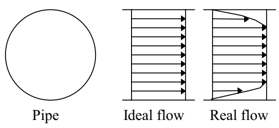
One dimensional flow:

Conditions 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.

But:

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.



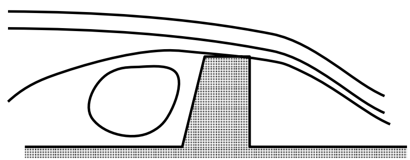
Should this be treated as two-dimensional flow?
Possibly - but it is only necessary if very high accuracy is required.

Two-dimensional flow

Conditions vary in the direction of flow and in one direction at right angles to this.

Flow patterns in two-dimensional flow can be shown by curved lines on a plane.

Below shows flow pattern over a weir.



In this course we will be considering:

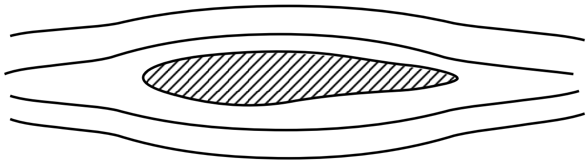
- steady
- incompressible
- one and two-dimensional flow

Streamlines

It is useful to visualise the flow pattern.
Lines joining points of equal velocity - velocity contours - can be drawn.

These lines are know as *streamlines*

Here are 2-D streamlines around a cross-section of an aircraft wing shaped body:



Fluid flowing past a solid boundary does not flow into or out of the solid surface.

Very close to a boundary wall the flow direction must be along the boundary.

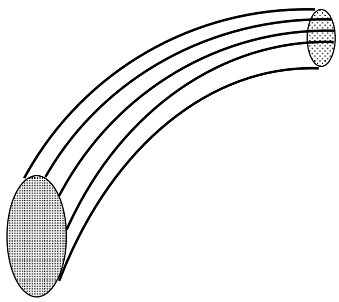
Some points about streamlines:

- *Close to a solid boundary, streamlines are parallel to that boundary*
- The direction of the streamline is the direction of the fluid velocity
- Fluid can not cross a streamline
- Streamlines can not cross each other
- Any particles starting on one streamline will stay on that same streamline
- In unsteady flow streamlines can change position with time
- In steady flow, the position of streamlines does not change.

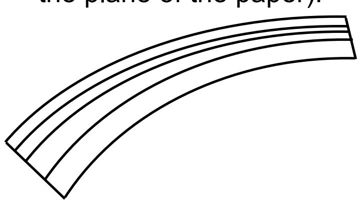
Streamtubes

A circle of points in a flowing fluid each has a streamline passing through it.

These streamlines make a tube-like shape known as a streamtube



In a two-dimensional flow the streamtube is flat (in the plane of the paper):



Some points about streamtubes

- The “walls” of a streamtube are streamlines.
- Fluid cannot flow across a streamline, so fluid cannot cross a streamtube “wall”.
- A streamtube is not like a pipe.
  Its “walls” move with the fluid.
- In unsteady flow streamtubes can change position with time
- In steady flow, the position of streamtubes does not change.

Flow rate

Mass flow rate

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

Volume flow rate - Discharge.

More commonly we use volume flow rate
Also know as *discharge.*

The symbol normally used for discharge is *Q*.

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

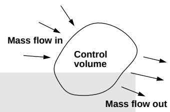
Discharge and mean velocity

Cross sectional area of a pipe is *A*
Mean velocity is u_m.

$$Q = Au_m$$

We usually drop the "m" and imply mean velocity.

Continuity



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

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

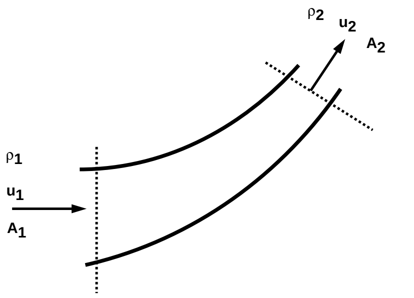
For steady flow

Mass entering = Mass leaving
per unit time per unit time

$$Q_1 = Q_2 = A_1u_1 = A_2u_2$$

Applying to a streamtube:

Mass enters and leaves only through the two ends
(it cannot cross the streamtube wall).



Mass entering = Mass leaving
per unit time per unit time

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

Or for steady flow,

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

This is the continuity equation.

In a real pipe (or any other vessel) we use the *mean* velocity and write

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

For incompressible, fluid $\rho_1 = \rho_2 = \rho$
(dropping the *m* subscript)

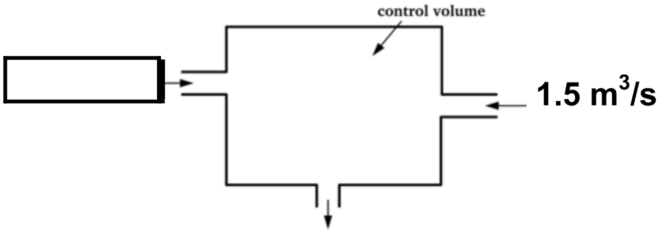
$$A_1 u_1 = A_2 u_2 = Q$$

This is the continuity equation most often used.

This equation is a very powerful tool.
It will be used repeatedly throughout the rest of this course.

Some example applications of Continuity

1. What is the outflow?

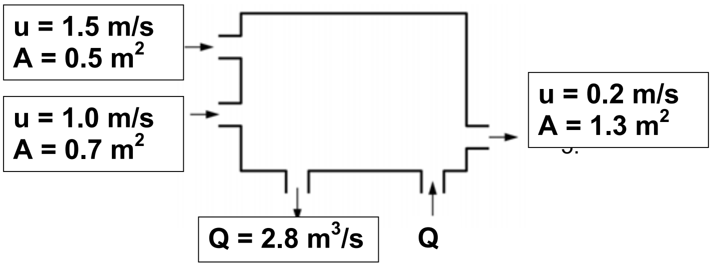


$Q_{in} = Q_{out}$

$1.5 + 1.5 = 3$

$Q_{out} = 3.0\ m^3/s$

2. What is the inflow?



Q = Area × Mean Velocity = Au

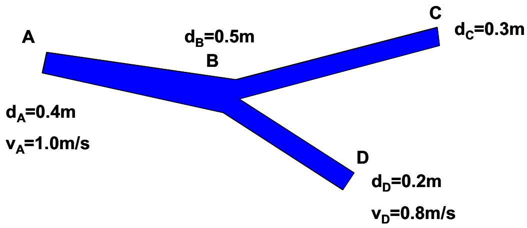
$Q + 1.5\times0.5 + 1\times0.7 = 0.2\times1.3 + 2.8$

$Q = 3.72\ m3/s$

Water flows in a circular pipe which increases in diameter from 400mm at point A to 500mm at point B. Then pipe then splits into two branches of diameters 0.3m and 0.2m discharging at C and D respectively.

If the velocity at A is 1.0m/s and at D is 0.8m/s, what are the discharges at C and D and the velocities at B and C?

Solution:
Draw diagram:



Make a table and fill in the missing values

Point	Velocity m/s	Diameter m	Area m²	Q m³/s
A	1.00	0.4	0.126	0.126
B	0.64	0.5	0.196	0.126
C	1.42	0.3	0.071	0.101
D	0.80	0.2	0.031	0.025

Lecture 9: The Bernoulli Equation
Unit 3: Fluid Dynamics

The Bernoulli equation is a statement of the principle of conservation of energy along a streamline

It can be written:

$$\frac{p_1}{\rho g} + \frac{u_1^2}{2g} + z_1 = \mathrm{H} = \text{Constant}$$

These terms represent:

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

These term all have units of length,
they are often referred to as the following:

pressure head = $\dfrac{p}{\rho g}$ velocity head = $\dfrac{u^2}{2g}$

potential head = z total head = H

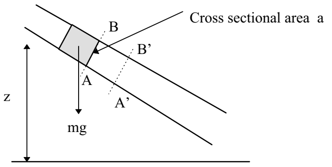
Restrictions in application of Bernoulli's equation:

- Flow is steady
- Density is constant (incompressible)
- Friction losses are negligible
- It 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

Pressure		Kinetic		Potential		Total
energy per	+	energy per	+	energy per	=	energy per
unit weight		unit weight		unit weight		unit weight

or

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

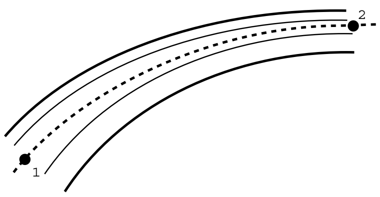
By the principle of conservation of energy, the total *energy* in the system does not change, thus the total *head* does not change. So the Bernoulli equation can be written

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

The Bernoulli equation is applied along

____

like that joining points 1 and 2 below.



total head at 1 = total head at 2

or

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

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

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

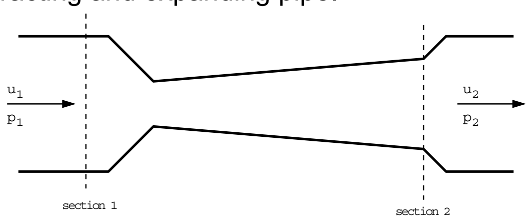
$$\frac{p_1}{\rho g} + \frac{u_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{u_2^2}{2g} + z_2 + h + w - q$$

Practical use of the Bernoulli Equation

The Bernoulli equation is often combined with the continuity equation to find velocities and pressures at points in the flow connected by a streamline.

Example:

Finding pressures and velocities within a contracting and expanding pipe.



A fluid, density ρ = 960 *kg/m*^{3} is flowing steadily through the above tube.

The section diameters are $d_1=100mm$ and $d_2=80mm$.

The gauge pressure at 1 is $p_1=200kN/m^2$

The velocity at 1 is $u_1=5m/s$.

The tube is horizontal ($z_1=z_2$)

What is the gauge pressure at section 2?

Apply the Bernoulli equation along a streamline joining section 1 with section 2.

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

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

Use the continuity equation to find u_2

$$A_1 u_1 = A_2 u_2$$

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

$$= 7.8125 m/s$$

So pressure at section 2

$$p_2 = 200000 - 17296.87$$

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

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

Note how

the velocity has increased

the pressure has decreased

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

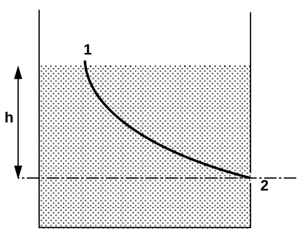
The Bernoulli equation is applicable to many situations not just the pipe flow.

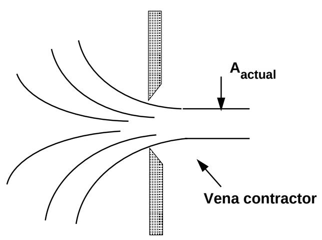
Here we will see its application to flow measurement from tanks, within pipes as well as in open channels.

Applications of Bernoulli: Flow from Tanks

Flow Through A Small Orifice

Flow from a tank through a hole in the side.





The edges of the hole are sharp to minimise frictional losses by minimising the contact between the hole and the liquid.

The streamlines at the orifice contract reducing the area of flow.

This contraction is called the *vena contracta*

The amount of contraction must be known to calculate the flow

Apply Bernoulli 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 theoretical value of velocity is an overestimate as friction losses have not been taken into account.

A coefficient of velocity is used to correct the theoretical velocity,

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

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

The discharge through the orifice

is
jet area × jet velocity

The area of the jet is the area of the vena contracta not the area of the orifice.

We use a *coefficient of contraction* to get the area of the jet

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

Giving discharge through the orifice:

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

C_d is the coefficient of discharge,
$C_d = C_c \times C_v$

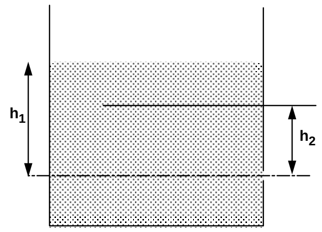
Time for the tank to empty

We have an expression for the discharge from the tank

$$Q = C_d A_o \sqrt{2gh}$$

We can use this to calculate how long it will take for level in the to fall

As the tank empties the level of water falls.
The discharge will also drop.



The tank has a cross sectional area of *A*.

In a time δt the level falls by δh
The flow out of the tank is

$$Q = Au$$

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

(-ve sign as δh is falling)

This *Q* is the same as the flow out of the orifice so

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

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

Integrating between the initial level, h_1, and final level, h_2, gives the time it takes to fall this height

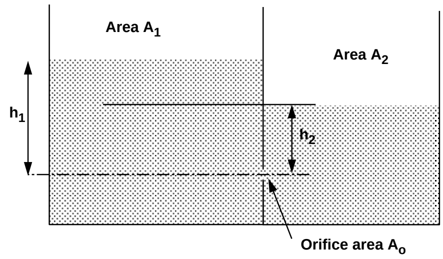
$$t = \frac{-A}{C_d A_o \sqrt{2g}} \int_{h_1}^{h_2} \frac{\delta h}{\sqrt{h}}$$

$$\left(\int \frac{1}{\sqrt{h}} = \int h^{-1/2} = 2h^{1/2} = 2\sqrt{h} \right)$$

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

Submerged Orifice

What if the tank is feeding into another?



Apply Bernoulli from point 1 on the surface of the deeper tank to point 2 at the centre of the orifice,

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

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

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

And the discharge is given by

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

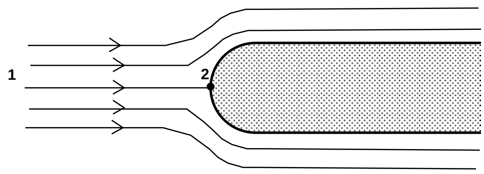
So the discharge of the jet through the submerged orifice depends on the *difference* in head across the orifice.

Lecture 10: Flow Measurement Devices
Unit 3: Fluid Dynamics

Pitot Tube

The Pitot tube is a simple velocity measuring device.

Uniform velocity flow hitting a solid blunt body, has streamlines similar to this:



Some move to the left and some to the right.
The centre one hits the blunt body and stops.

At this point (2) velocity is zero

The fluid does not move at this one point.
This point is known as the *stagnation point*.

Using the Bernoulli equation we can calculate the pressure at this point.

Along the central streamline at 1: velocity u_1 , pressure p_1
At the stagnation point (2): $u_2 = 0$. (Also $z_1 = z_2$)

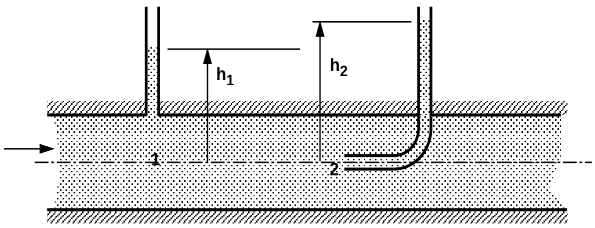
$$\frac{p_1}{\rho} + \frac{u_1^2}{2} = \frac{p_2}{\rho}$$

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

How can we use this?

The blunt body does not have to be a solid.
It could be a static column of fluid.

Two piezometers, one as normal and one as a Pitot tube within the pipe can be used as shown below to measure velocity of flow.



We have the equation for p_2 ,

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

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

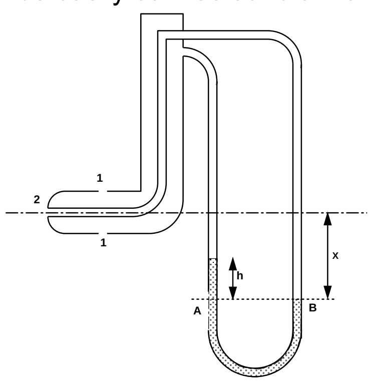
$$u = \sqrt{2g(h_2 - h_1)}$$

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

Pitot Static Tube

The necessity of two piezometers makes this arrangement awkward.

The *Pitot static* tube combines the tubes and they can then be easily connected to a manometer.



[Note: the diagram of the Pitot tube is not to scale. In reality its diameter is very small and can be ignored i.e. points 1 and 2 are considered to be at the same level]

The holes on the side connect to one side of a manometer, while the central hole connects to the other side of the manometer

Using the theory of the manometer,

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

We know that $p_2 = p_1 + \frac{1}{2}\rho u_1^2$, giving

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

The Pitot/Pitot-static is:

- Simple to use (and analyse)
- Gives velocities (not discharge)
- May block easily as the holes are small.

Pitot-Static Tube Example

A pitot-static tube is used to measure the air flow at the centre of a 400mm diameter building ventilation duct.

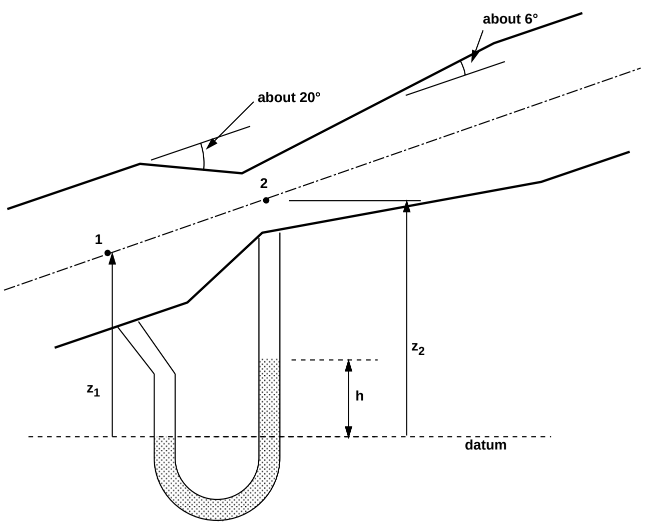
If the height measured on the attached manometer is 10 mm and the density of the manometer fluid is 1000 kg/m^3, determine the volume flow rate in the duct. Assume that the density of air is 1.2 kg/m^3.

Venturi Meter

The Venturi meter is a device for measuring **discharge** in a pipe.

It is 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.



Apply Bernoulli along the streamline from point 1 to point 2

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

By continuity

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

Substituting and rearranging gives

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

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

The theoretical (ideal) discharge is u×A.

Actual discharge takes into account the losses due to friction, we include a coefficient of discharge ($C_d \approx 0.9$)

$$Q_{ideal} = u_1 A_1$$

$$Q_{actual} = C_d Q_{ideal} = C_d u_1 A_1$$

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

In terms of the manometer readings

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

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

Giving

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

This expression does not include any elevation terms. (z_1 or z_2)

When used with a manometer
The Venturimeter can be used without knowing its angle.

Venturimeter design:

- The diffuser assures a gradual and steady deceleration after the throat. So that pressure rises to something near that before the meter.
- The angle of the diffuser is usually between 6 and 8 degrees.
- Wider and the flow might separate from the walls increasing energy loss.
- If the angle is less 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%.
- Care must be taken when connecting the manometer so that no burrs are present.

Venturimeter Example

A venturimeter is used to measure the flow of water in a 150 mm diameter pipe. The throat diameter of the venturimeter is 60 mm and the discharge coefficient is 0.9. If the pressure difference measured by a manometer is 10 cm mercury, what is the average velocity in the pipe?

Assume water has a density of 1000 kg/m^3 and mercury has a *relative* density of 13.6.

Lecture 11: Notches and Weirs
Unit 3: Fluid Dynamics

- A notch is an opening in the side of a tank or reservoir.
- It is a device for measuring discharge
- A weir is a notch on a larger scale - usually found in rivers.
- It is used as both a discharge measuring device and a device to raise water levels.
- There are many different designs of weir.
- We will look at sharp crested weirs.

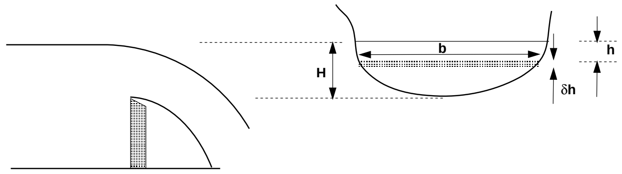
Weir Assumptions

- velocity of the fluid approaching the weir is small so we can ignore kinetic energy.
- The velocity in the flow depends only on the depth below the free surface. $u = \sqrt{2gh}$

These assumptions are fine for tanks with notches or reservoirs with weirs, in rivers with high velocity approaching the weir is substantial the kinetic energy must be taken into account

A General Weir Equation

Consider a horizontal strip of width *b,* depth *h* below the free surface



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

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 total theoretical discharge

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

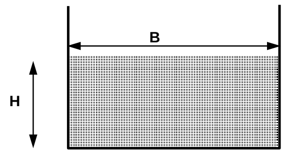
This is different for every differently shaped weir or notch.

We need an expression relating the width of flow across the weir to the depth below the free surface.

Rectangular Weir

The width does not change with depth so

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



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 get the actual discharge we introduce a coefficient of discharge, C_d, to account for losses at the edges of the weir and contractions in the area of flow,

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

Rectangular Weir Example

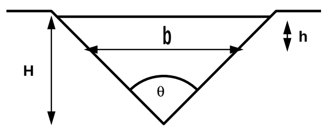
Water enters the Millwood flood storage area via a rectangular weir when the river height exceeds the weir crest. For design purposes a flow rate of 162 litres/s over the weir can be assumed

1. Assuming a height over the crest of 20cm and Cd=0.2, what is the necessary width, B, of the weir?

2. What will be the velocity over the weir at this design?

'V' Notch Weir

The relationship between width and depth is dependent on the angle of the "V".



The width, *b*, a depth *h* from the free surface is

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

So the discharge is

$$Q_{\text{theoretical}} = 2\sqrt{2g}\tan\left(\frac{\theta}{2}\right)\int_0^H (H-h)h^{1/2}\,dh$$

$$= 2\sqrt{2g}\tan\left(\frac{\theta}{2}\right)\left[\frac{2}{3}Hh^{3/2} - \frac{2}{5}h^{5/2}\right]_0^H$$

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

The actual discharge is obtained by introducing a coefficient of discharge

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

‘V’ Notch Weir Example

Water is flowing over a 90° ‘V’ Notch weir into a tank with a cross-sectional area of 0.6m^2. After 30s the depth of the water in the tank is 1.5m.

If the discharge coefficient for the weir is 0.8, what is the height of the water above the weir?

Lecture 12: The Momentum Equation
Unit 3: Fluid Dynamics

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.

The analysis of motion is as in solid mechanics: by use of Newton’s laws of motion.

The **Momentum equation** is a statement of Newton’s Second Law

It relates the sum of the forces to the acceleration or rate of change of momentum.

From solid mechanics you will recognise

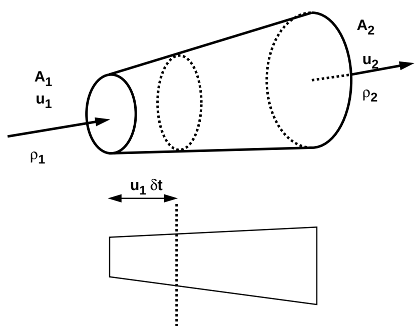
F = ma

What mass of moving fluid we should use?

We use a different form of the equation.

Consider a streamtube:

And assume *steady non-uniform* flow



In time δt a volume of the fluid moves from the inlet a distance $u_1\delta t$, so

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

The mass entering,

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

And momentum

$$\begin{aligned}\text{momentum entering stream tube} &= \text{mass} \times \text{velocity}\\ &= \rho_1 A_1 u_1 \delta t\, u_1\end{aligned}$$

Similarly, at the exit, we get the expression:

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

By Newton's 2nd Law.

Force = 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

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

An alternative derivation

From conservation of mass

mass into face 1 = 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 enters face 1 is

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

The rate at which momentum leaves face 2 is

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

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

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

So

Force = rate of change of momentum

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

So we have these two expressions,
either one is known as the momentum equation

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

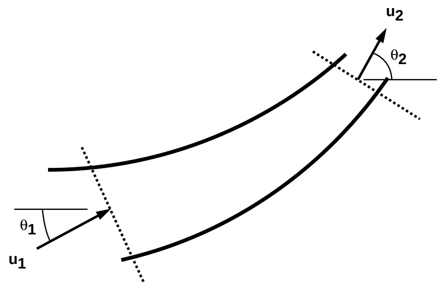
$$F = Q\rho(u_2 - u_1)$$

The Momentum equation.

This force acts on the fluid in the direction of the flow of the fluid.

The previous analysis assumed the inlet and outlet velocities in the same direction
i.e. a one dimensional system.

What happens when this is not the case?



We consider the forces by resolving in the directions of the co-ordinate axes.

The force in the x-direction

$$F_x = \dot{m}(u_2 \cos\theta_2 - u_1 \cos\theta_1)$$
$$= \dot{m}(u_{2x} - u_{1x})$$

or

$$F_x = \rho Q(u_2 \cos\theta_2 - u_1 \cos\theta_1)$$
$$= \rho Q(u_{2x} - u_{1x})$$

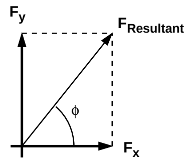
And the force in the y-direction

$$F_y = \dot{m}\left(u_2 \sin\theta_2 - u_1 \sin\theta_1\right)$$
$$= \dot{m}\left(u_{2_y} - u_{1_y}\right)$$

or

$$F_y = \rho Q\left(u_2 \sin\theta_2 - u_1 \sin\theta_1\right)$$
$$= \rho Q\left(u_{2_y} - u_{1_y}\right)$$

The resultant force can be found by combining these components



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

And the angle of this force

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

In summary we can say:

Total force on the fluid	=	rate of change of momentum through the control volume

$$F = \dot{m}\left(u_{\text{out}} - u_{\text{in}}\right)$$

or

$$F = \rho Q\left(u_{\text{out}} - u_{\text{in}}\right)$$

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

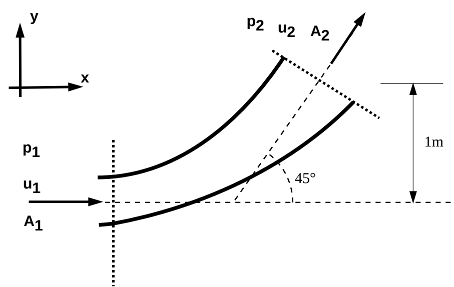
$$R = -F_R$$

Application of the Momentum Equation

Forces on a Bend

Consider a converging or diverging pipe bend lying in the vertical or horizontal plane turning through an angle of θ.

Here is a diagram of a diverging pipe bend.



Why do we want to know the forces here?

As the fluid changes direction a force will act on the bend.

This force can be very large in the case of water supply pipes. The bend must be held in place to prevent breakage at the joints.

We need to know how much force a support (thrust block) must withstand.

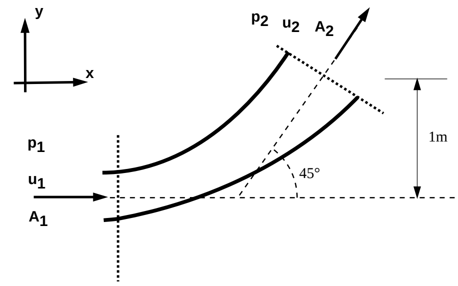
Step in Analysis:

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

An Example of Forces on a Bend

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.
[13.95kN at 67° 39' to the horizontal]

1&2 Draw the control volume and the axis system



p_1 = 100 kN/m^2,
Q = 0.3 m^3/s
θ = 45°

d_1 = 0.15 m d_2 = 0.3 m

A_1 = 0.177 m^2 A_2 = 0.0707 m^2

3 Calculate the total force
in the x direction

$$F_{Tx} = \rho Q(u_{2x} - u_{1x})$$
$$= \rho Q(u_2 \cos\theta - u_1)$$

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

$$u_1 = \frac{0.3}{\pi(0.15^2/4)} = 16.98\,m/s$$
$$u_2 = \frac{0.3}{0.0707} = 4.24\,m/s$$

$$F_{Tx} = 1000 \times 0.3(4.24\cos 45 - 16.98)$$
$$= -4193.68\,N$$

and in the y-direction

$$F_{Ty} = \rho Q(u_{2y} - u_{1y})$$
$$= \rho Q(u_2 \sin\theta - 0)$$
$$= 1000 \times 0.3(4.24\sin 45)$$
$$= 899.44\,N$$

4 Calculate the pressure force.

$$F_P = \text{pressure force at 1 - pressure force at 2}$$
$$\theta_1 = 0, \qquad \theta_2 = \theta$$

$$F_{Px} = p_1 A_1 \cos 0 - p_2 A_2 \cos\theta = p_1 A_1 - p_2 A_2 \cos\theta$$

$$F_{Py} = p_1 A_1 \sin 0 - p_2 A_2 \sin\theta = -p_2 A_2 \sin\theta$$

We know pressure at the inlet but not at the outlet

we can use the Bernoulli equation to calculate this unknown pressure.

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

where h_f is the friction loss
In the question it says this can be ignored, h_f=0

The height of the pipe at the outlet is 1m above the inlet.
Taking the inlet level as the datum:
z_1 = 0 z_2 = 1m

So the Bernoulli equation becomes:

$$\frac{100000}{1000\times 9.81}+\frac{16.98^2}{2\times 9.81}+0=\frac{p_2}{1000\times 9.81}+\frac{4.24^2}{2\times 9.81}+1.0$$

$$p_2 = 225361.4\ N/m^2$$

$$F_{P\,x} = 100000\times 0.0177 - 225361.4\cos 45\times 0.0707$$
$$= 1770 - 11266.34 = -9496.37\,kN$$

$$F_{P\,y} = -225361.4\sin 45\times 0.0707$$
$$= -11266.37$$

5 Calculate the body force

The only body force is the force due to gravity. That is the weight acting in the -ve y direction.

$$F_{B\,y} = -\rho g\times volume$$
$$= -1000\times 9.81\times 0.075$$
$$= -12901.56\,N$$

There are no body forces in the x direction,

$$F_{B\,x} = 0$$

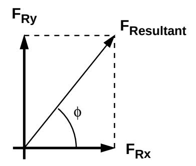
6 Calculate the resultant force

$$F_{T\,x} = F_{R\,x} + F_{P\,x} + F_{B\,x}$$
$$F_{T\,y} = F_{R\,y} + F_{P\,y} + F_{B\,y}$$

$$F_{R\,x} = F_{T\,x} - F_{P\,x} - F_{B\,x}$$
$$= -4193.6 + 9496.37$$
$$= 5302.7\,N$$

$$F_{R\,y} = F_{T\,y} - F_{P\,y} - F_{B\,y}$$
$$= 899.44 + 11266.37 + 735.75$$
$$= 12901.56\,N$$

And the resultant force on the fluid is given by



$$F_R = \sqrt{F_{R\,x}^2 - F_{R\,y}^2}$$
$$= \sqrt{5302.7^2 + 12901.56^2}$$
$$= 13.95\,kN$$

And the direction of application is

$$\phi = \tan^{-1}\left(\frac{F_{R\,y}}{F_{R\,x}}\right)$$
$$= \tan^{-1}\left(\frac{12901.56}{5302.7}\right)$$
$$= 67.66^\circ = 67^\circ\ 39'$$

The force on the bend is the same magnitude but in the opposite direction

$$R = -F_R = -13.95\,kN$$

Lecture 13: Design Study 2

See Separate Handout

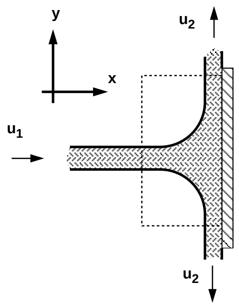
Lecture 14: Momentum Equation Examples
Unit 3: Fluid Dynamics

Impact of a Jet on a Plane

A jet hitting a flat plate (a plane) at an angle of 90°

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.

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



3 Calculate the total force

In the x-direction

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

The system is symmetrical

the forces in the y-direction cancel.

$$F_{T_y} = 0$$

4 Calculate the pressure force.

The pressures at both the inlet and the outlets to the control volume are atmospheric.

The pressure force is zero

$$F_{P_x} = F_{P_y} = 0$$

5 Calculate the body force

As the control volume is small

we can ignore the body force due to gravity.

$$F_{B_x} = F_{B_y} = 0$$

6 Calculate the resultant force

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

Exerted on the fluid.

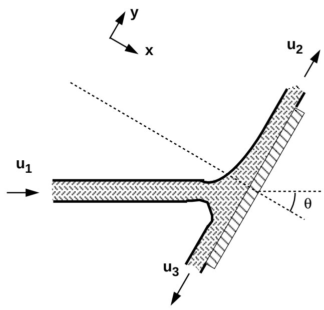
The force on the plane is the same magnitude but in the opposite direction

$$R = -F_{R_x}$$

If the plane were at an angle

the analysis is the same.

But it is usually most convenient to choose the axis system normal to the plate.

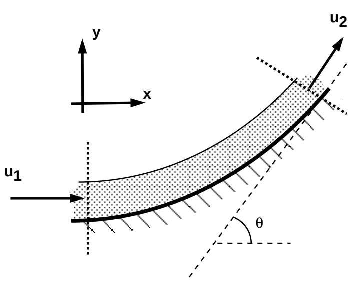


Force on a curved vane

This case is similar to that of a pipe, but the analysis is simpler.

Pressures at ends are equal at atmospheric

Both the cross-section and velocities (in the direction of flow) remain constant.



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

3 Calculate the total force

in the x direction

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

by continuity $u_1 = u_2 = \dfrac{Q}{A}$, so

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

and in the y-direction

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

4 Calculate the pressure force.

The pressure at both the inlet and the outlets to the control volume is atmospheric.

$$F_{P_x} = F_{P_y} = 0$$

5 Calculate the body force

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

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

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

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

6 Calculate the resultant force

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

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

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

$$F_{R_y} = F_{T_y} = \rho \frac{Q^2}{A}$$

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

$$R = -F_R$$

SUMMARY

The Momentum equation
is a statement of Newton's Second Law

For a fluid of constant density,

Total force on the fluid = rate of change of momentum through the control volume

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

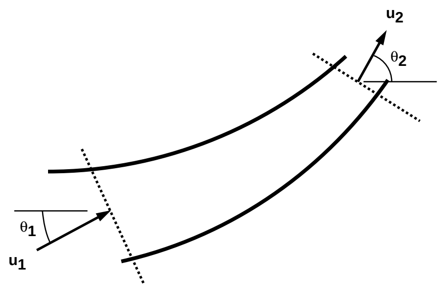
This force acts on the fluid
in the direction of the velocity of fluid.

This is the total force F_T where:

$F_T = F_R + F_B + F_P$

F_R = External force on the fluid from any solid body touching the control volume
F_B = Body force on the fluid body (e.g. gravity)
F_P = Pressure force on the fluid by fluid pressure outside the control volume

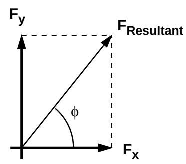
We work with components of the force:



$$F_x = \rho Q(u_{2x} - u_{1x}) = \rho Q(u_2\cos\theta_2 - u_1\cos\theta_1)$$

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

The resultant force can be found by combining these components



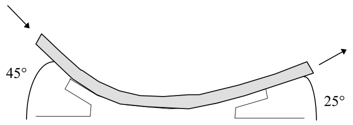
$$F_{\text{resultant}} = \sqrt{F_x^2 + F_y^2}$$

And the angle this force acts:

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

Lecture 15: Calculations
Unit 3: Fluid Dynamics

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.



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.

3. 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.

4. In an experiment a jet of water of diameter 20mm is fired vertically upwards at a sprung target that deflects the water at an angle of 120° to the horizontal in all directions. If a 500g mass placed on the target balances the force of the jet, was is the discharge of the jet in litres/s?

5. Water is being fired at 10 m/s from a hose of 50mm diameter into the atmosphere. The water leaves the hose through a nozzle with a diameter of 30mm at its exit. Find the pressure just upstream of the nozzle and the force on the nozzle.