CIVE1400: An Introduction to Fluid Mechanics

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

Real fluids

Flowing real fluids exhibit viscous effects, they:

- **"stick" to solid surfaces**
- **have stresses within their body.**

From earlier we saw this relationship between shear stress and velocity gradient:

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

The shear stress, τ, in a fluid is proportional to the velocity gradient - the rate of change of velocity across the flow.

For a "Newtonian" fluid we can write:

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

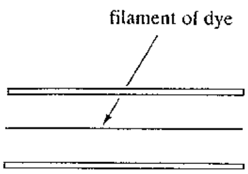
where μ, is coefficient of viscosity (or simply viscosity).

Here we look at the influence of forces due to momentum changes and viscosity in a moving fluid.

Laminar and turbulent flow

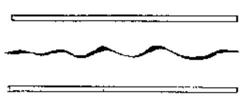
Injecting a dye into the middle of flow in a pipe, what would we expect to happen?

This



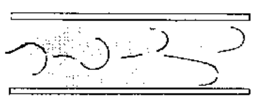
Laminar (viscous)

this



Transitional

or this



Turbulent

All three would happen - but for different flow rates.

Top: Slow flow
Middle: Medium flow
Bottom: Fast flow

Top:	**Laminar flow**
Middle:	**Transitional flow**
Bottom:	**Turbulent flow**

Laminar flow:

Motion of the fluid particles is very orderly all particles moving in straight lines parallel to the pipe walls.

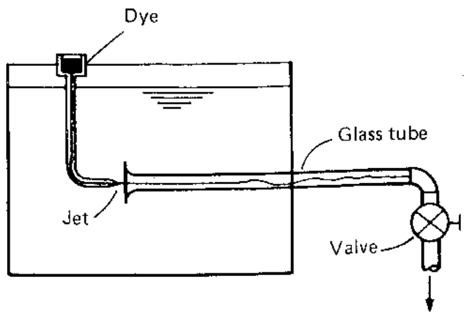
Turbulent flow:

Motion is, locally, completely random but the overall direction of flow is one way.

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

The was first investigated in the 1880s by Osbourne Reynolds in a classic experiment in fluid mechanics.

A tank arranged as below:



After many experiments he found this expression

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

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

This could be used to predict the change in flow type for any fluid.

This value is known as the Reynolds number, Re:

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

Laminar flow:	Re < 2000
Transitional flow:	2000 < Re < 4000
Turbulent flow:	Re > 4000

What are the units of Reynolds number?

We can fill in the equation with SI units:

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

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

It has no units!

A quantity with no units is known as a non-dimensional (or dimensionless) quantity.

(We will see more of these in the section on dimensional analysis.)

The Reynolds number, Re, is a non-dimensional number.

At what speed does the flow pattern change?

We use the Reynolds number in an example:

A pipe and the fluid flowing have the following properties:

water density	ρ = 1000 kg/m^3
pipe diameter	d = 0.5m
(dynamic) viscosity,	μ = 0.55x10^3 Ns/m^2

What is the MAXIMUM velocity when flow is laminar i.e. Re = 2000

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

What is the MINIMUM velocity when flow is turbulent i.e. Re = 4000

$$\mathrm{Re} = \frac{\rho u d}{\mu} = 4000$$
$$u = 0.0044\,m/s$$

In a house central heating system, typical pipe diameter = 0.015m,

limiting velocities would be, 0.0733 and 0.147m/s.

Both of these are very slow.

In practice laminar flow rarely occurs in a piped water system.

Laminar flow does occur in fluids of greater viscosity e.g. in bearing with oil as the lubricant.

<u>What does this abstract number mean</u>?

We can give the Re number a physical meaning.

This may help to understand some of the reasons for the changes from laminar to turbulent flow.

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

When inertial forces dominate (when the fluid is flowing faster and Re is larger) the flow is turbulent.

When the viscous forces are dominant (slow flow, low Re) they keep the fluid particles in line, the flow is laminar.

<u>Laminar flow</u>

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

<u>Transitional flow</u>

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

<u>Turbulent flow</u>

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

<u>Pressure loss due to friction in a pipeline</u>

Up to now we have considered ideal fluids: no energy losses due to friction

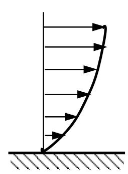
Because fluids are viscous, energy is lost by flowing fluids due to friction.

This must be taken into account.

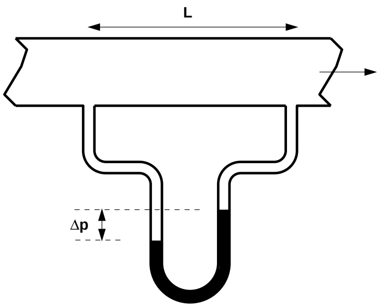
The effect of the friction shows itself as a pressure (or head) loss.

In a real flowing fluid shear stress slows the flow.

To give a velocity profile:



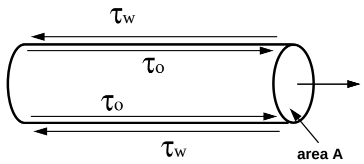
Attaching a manometer gives pressure (head) loss due to the energy lost by the fluid overcoming the shear stress.



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

How can we quantify this pressure loss in terms of the forces acting on the fluid?

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



τ_w is the mean shear stress on the boundary
Upstream pressure is p,
Downstream pressure falls by Δp to (p-Δp)

The <u>driving force</u> due to pressure

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

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

The <u>retarding force</u> is due to the shear stress

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

As the flow is in equilibrium,

driving force = retarding force

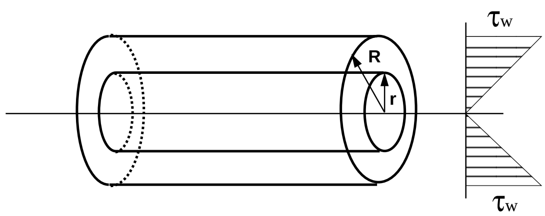
$$\Delta p \frac{\pi d^2}{4} = \tau_w \pi d L$$

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

Giving pressure loss in a pipe in terms of:

- **pipe diameter**
- **shear stress at the wall**

What is the variation of shear stress in the flow?



At the wall

$$\tau_w = \frac{R}{2}\frac{\Delta p}{L}$$

At a radius r

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

$$\tau = \tau_w \frac{r}{R}$$

A linear variation in shear stress.

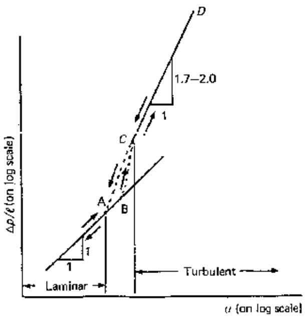
This is valid for:

- **steady flow**
- **laminar flow**
- **turbulent flow**

Shear stress and hence pressure loss varies with velocity of flow and hence with Re.

Many experiments have been done with various fluids measuring the pressure loss at various Reynolds numbers.

A graph of pressure loss and Re look like:



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

laminar $\Delta p \propto u$

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

Pressure loss during laminar flow in a pipe

In general the shear stress τ_w. is almost impossible to measure.

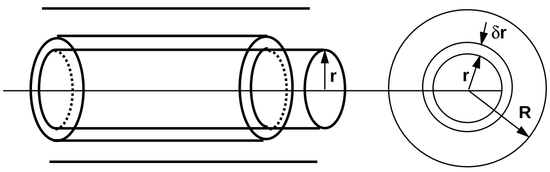
For laminar flow we can calculate a theoretical value for a given velocity, fluid and pipe dimension.

In laminar flow the paths of individual particles of fluid do not cross.

Flow is like a series of concentric cylinders sliding over each other.

And the stress on the fluid in laminar flow is entirely due to viscose forces.

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



The fluid is in equilibrium, shearing forces equal the pressure forces.

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

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

Newtons law of viscosity says $\tau = \mu \dfrac{du}{dy}$**,**

We are measuring from the pipe centre, so

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

Giving:

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

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

In an integral form this gives an expression for velocity,

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

The value of velocity at a point distance r from the centre

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

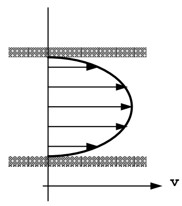
At *r = 0*, (the centre of the pipe), *u = u_{max}*, at *r = R* (the pipe wall) *u = 0*;

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

At a point r from the pipe centre when the flow is laminar:

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

This is a parabolic profile (of the form y = ax^2 + b) so the velocity profile in the pipe looks similar to



What is the discharge in the pipe?

The flow in an annulus of thickness δr

$$\delta Q = u_r A_{annulus}$$

$$A_{annulus} = \pi(r+\delta r)^2 - \pi r^2 \approx 2\pi r \delta r$$

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

$$Q = \frac{\Delta p}{L}\frac{\pi}{2\mu}\int_0^R\left(R^2 r - r^3\right)dr$$

$$= \frac{\Delta p}{L}\frac{\pi R^4}{8\mu} = \frac{\Delta p\,\pi d^4}{L128\mu}$$

So the discharge can be written

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

This is the Hagen-Poiseuille Equation for laminar flow in a pipe

To get pressure loss (head loss) in terms of the velocity of the flow, write pressure in terms of head loss h_f, i.e. $p = \rho g h_f$

Mean velocity:

$$u = Q / A$$

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

Head loss in a pipe with laminar flow by the Hagen-Poiseuille equation:

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

Pressure loss is directly proportional to the velocity when flow is laminar.

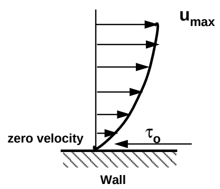
It has been validated many time by experiment.
It justifies two assumptions:

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

Boundary Layers

Recommended reading: Fluid Mechanics by Douglas J F, Gasiorek J M, and Swaffield J A. Longman publishers. Pages 327-332.

Fluid flowing over a stationary surface, e.g. the bed of a river, or the wall of a pipe, is brought to rest by the shear stress t_o
This gives a, now familiar, velocity profile:



Zero at the wall
A maximum at the centre of the flow.

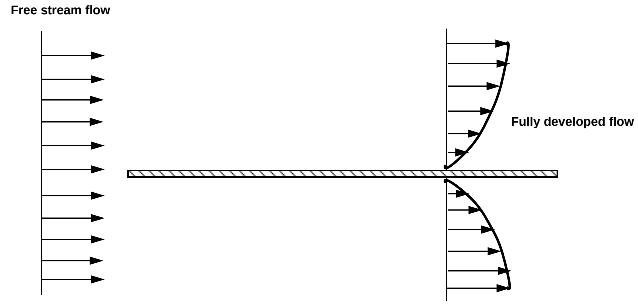
The profile doesn't just exit.
It is build up gradually.

Starting when it first flows past the surface e.g. when it enters a pipe.

Considering a flat plate in a fluid.

Upstream the velocity profile is uniform, This is known as free stream flow.

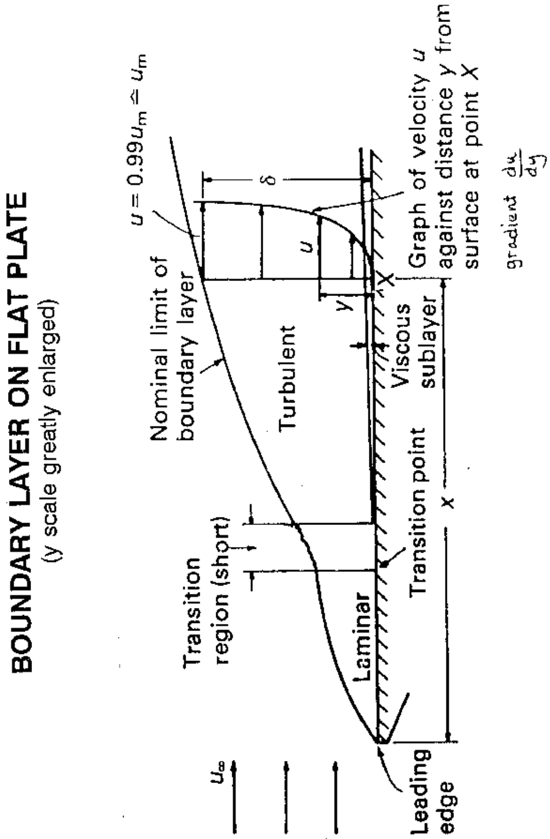
Downstream a velocity profile exists. This is known as fully developed flow.



Some question we might ask:

How do we get to the fully developed state?
Are there any changes in flow as we get there?
Are the changes significant / important?

Understand this Boundary layer growth diagram.



Boundary layer thickness:

δ = distance from wall to where u = 0.99 $u_{mainstream}$

δ increases as fluid moves along the plate.
It reaches a maximum in fully developed flow.

The δ increase corresponds to a drag force increase on the fluid.

As fluid is passes over a greater length:

* **more fluid is slowed**
* **by friction between the fluid layers**
* **the thickness of the slow layer increases.**

Fluid near the top of the boundary layer drags the fluid nearer to the solid surface along.

The mechanism for this dragging may be one of two types:

First: <u>viscous forces</u>
(the forces which hold the fluid together)

When the boundary layer is thin:
<u>velocity gradient *du/dy*</u>, is <u>large</u>

by Newton's law of viscosity
<u>shear stress</u>, $\tau = \mu\ (du/dy)$, is <u>large</u>.

The force may be large enough to drag the fluid close to the surface.

As the boundary layer <u>thickens</u> velocity gradient <u>reduces</u> and shear stress <u>decreases</u>.

Eventually it is too small to drag the slow fluid along.

Up to this point the flow has been <u>laminar</u>.

Newton's law of viscosity has applied.

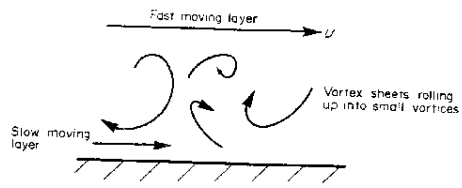
This part of the boundary layer is the <u>laminar boundary layer</u>

Second: <u>momentum transfer</u>

If the viscous forces were the only action the fluid would come to a rest.

Viscous shear stresses have held the fluid particles in a constant motion within layers.
Eventually they become too small to hold the flow in layers;

the fluid starts to rotate.



The fluid motion rapidly becomes turbulent.
Momentum transfer occurs between fast moving main flow and slow moving near wall flow.
Thus the fluid by the wall is kept in motion.
The net effect is an increase in momentum in the boundary layer.
This is the <u>turbulent boundary layer</u>.

Close to boundary velocity gradients are very large.
Viscous shear forces are large.
Possibly large enough to cause laminar flow.

This region is known as the laminar sub-layer.

This layer occurs within the turbulent zone it is next to the wall.
It is very thin – a few hundredths of a mm.

Surface roughness effect

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

In turbulent flow:
Roughness higher than laminar sub-layer: increases turbulence and energy losses.

In laminar flow:
Roughness has very little effect

Boundary layers in pipes
Initially of the laminar form.

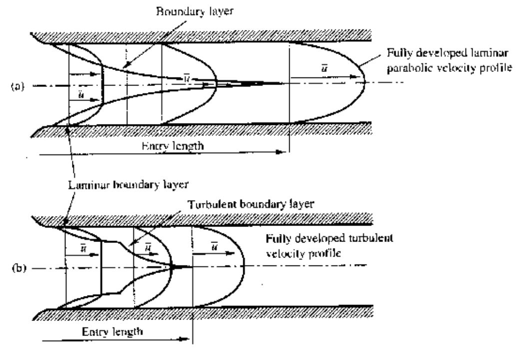
It changes depending on the ratio of inertial and viscous forces;

i.e. whether we have laminar (viscous forces high) or turbulent flow (inertial forces high).

Use Reynolds number to determine which state.

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

Laminar flow:	**Re < 2000**
Transitional flow:	**2000 < Re < 4000**
Turbulent flow:	**Re > 4000**



Laminar flow: profile parabolic (proved in earlier lectures)
The first part of the boundary layer growth diagram.

Turbulent (or transitional),
Laminar and the turbulent (transitional) zones of the boundary layer growth diagram.

Length of pipe for fully developed flow is the entry length.

Laminar flow $\approx 120 \times$ diameter

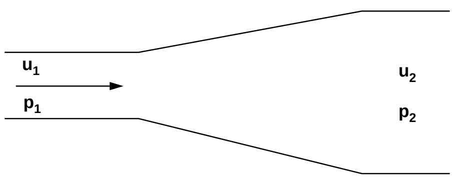
Turbulent flow $\approx 60 \times$ diameter

Boundary layer separation

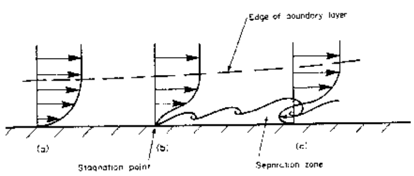
Divergent flows:
Positive pressure gradients.
Pressure increases in the direction of flow.

The fluid in the boundary layer has so little momentum that it is brought to rest, and possibly reversed in direction.

Reversal lifts the boundary layer.



$p_1 < p_2$ $u_1 > u_2$



This phenomenon is known as boundary layer separation.

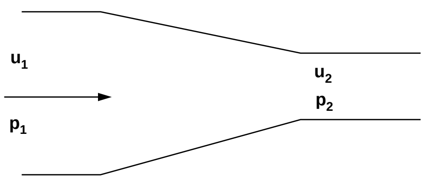
Boundary layer separation:

- **increases the turbulence**
- **increases the energy losses in the flow.**

Separating / divergent flows are inherently unstable

Convergent flows:

- **Negative pressure gradients**
- **Pressure decreases in the direction of flow.**
- **Fluid accelerates and the boundary layer is thinner.**



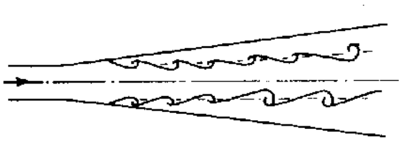
$p_1 > p_2$ $u_1 < u_2$

- **Flow remains stable**
- **Turbulence reduces.**
- **Boundary layer separation does not occur.**

Examples of boundary layer separation

A divergent duct or diffuser

velocity drop
(according to continuity)
pressure increase
(according to the Bernoulli equation).



Increasing the angle increases the probability of boundary layer separation.

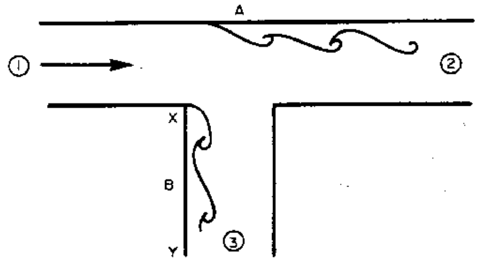
Venturi meter

Diffuser angle of about 6°

A balance between:

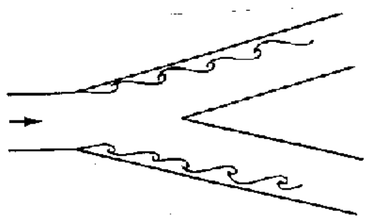
- **length of meter**
- **danger of boundary layer separation.**

Tee-Junctions

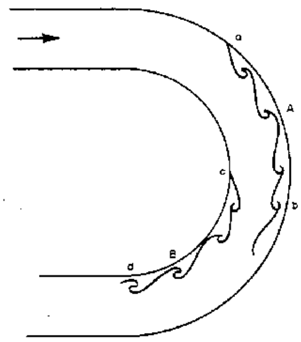


Assuming equal sized pipes),
Velocities at 2 and 3 are smaller than at 1.
Pressure at 2 and 3 are higher than at 1.
Causing the two separations shown

Y-Junctions
Tee junctions are special cases of the Y-junction.



Bends



Two separation zones occur in bends as shown above.

$P_b > P_a$ causing separation.

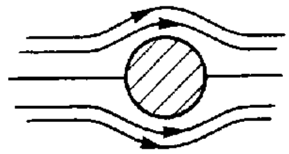
$P_d > P_c$ causing separation

Localised effect

Downstream the boundary layer reattaches and normal flow occurs.
Boundary layer separation is only local.
Nevertheless downstream of a junction / bend /valve etc. fluid will have lost energy.

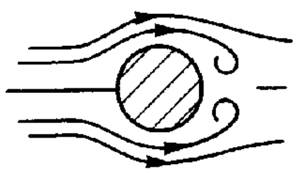
Flow past a cylinder

Slow flow, Re < 0.5 no separation:



$Re < 0{\cdot}5$

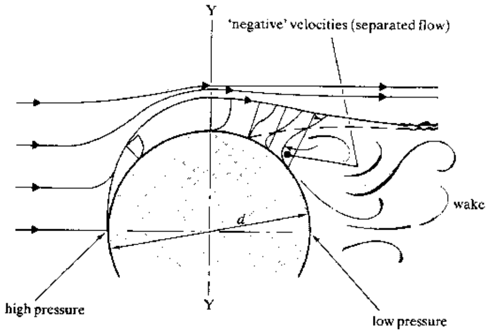
Moderate flow, Re < 70, separation vortices form.



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

Fast flow Re > 70
vortices detach alternately.
Form a trail of down stream.
Karman vortex trail or street.
(Easily seen by looking over a bridge)

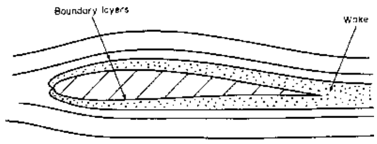
Causes whistling in power cables.
Caused Tacoma narrows bridge to collapse.
Frequency of detachment was equal to the bridge natural frequency.



Fluid accelerates to get round the cylinder
Velocity maximum at Y.
Pressure dropped.

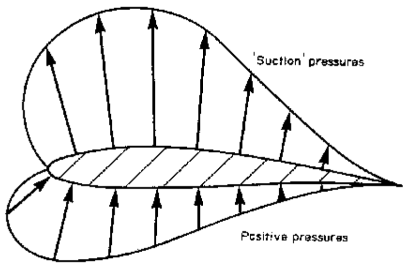
Adverse pressure between here and downstream.
Separation occurs

Aerofoil
Normal flow over a aerofoil or a wing cross-section.

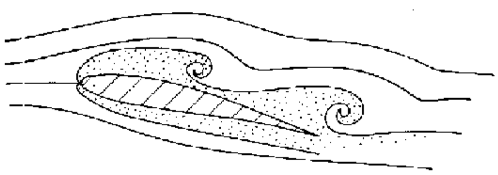


(boundary layers greatly exaggerated)

The velocity increases as air flows over the wing. The pressure distribution is as below so transverse lift force occurs.



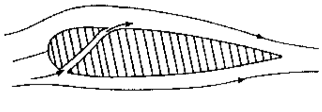
At too great an angle
boundary layer separation occurs on the top
Pressure changes dramatically.
This phenomenon is known as stalling.



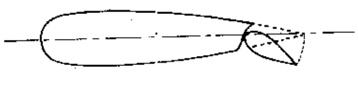
All, or most, of the 'suction' pressure is lost.
The plane will suddenly drop from the sky!

Solution:
Prevent separation.

1. **Engine intakes draws slow air from the boundary layer at the rear of the wing though small holes**
2. **Move fast air from below to top via a slot.**



3. **Put a flap on the end of the wing and tilt it.**



Examples:

Exam questions involving boundary layer theory are typically descriptive. They ask you to explain the mechanisms of growth of the boundary layers including how, why and where separation occurs. You should also be able to suggest what might be done to prevent separation.

Lectures 18 & 19: Dimensional Analysis
Unit 4: The Effect of the Boundary on Flow

Application of fluid mechanics in design makes use of experiments results.

Results often difficult to interpret.

Dimensional analysis provides a strategy for choosing relevant data.

Used to help analyse fluid flow

Especially when fluid flow is too complex for mathematical analysis.

Specific uses:

- help design experiments
- Informs which measurements are important
- Allows most to be obtained from experiment:

e.g. What runs to do. How to interpret.

It depends on the correct identification of variables

Relates these variables together

Doesn't give the complete answer

Experiments necessary to complete solution

Uses principle of dimensional homogeneity

It gives qualitative results which only become quantitative from experimental analysis.

Dimensions and units

Any physical situation
can be described by familiar properties.

e.g. length, velocity, area, volume, acceleration etc.

These are all known as dimensions.

Dimensions are of no use without a magnitude.

i.e. a standardised unit

e.g metre, kilometre, Kilogram, a yard etc.

Dimensions can be measured.

Units used to quantify these dimensions.

In dimensional analysis we are concerned with the nature of the dimension

i.e. its quality not its quantity.

The following common abbreviations are used:

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

Here we will use L, M, T and F (not Θ).

We can represent all the physical properties we are interested in with three:

L, T
and one of M or F

As either mass (M) of force (F) can be used to represent the other, i.e.

$$F = MLT^{-2}$$
$$M = FT^{2}L^{-1}$$

We will mostly use LTM:

This table 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 is only true if both sides have the same dimensions.

It must be dimensionally homogenous.

What are the dimensions of X?

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

$$L\,(LT^{-2})^{1/2}\,L^{3/2} = X$$

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

$$L^{3}\,T^{-1} = X$$

The powers of the individual dimensions must be equal on both sides.

(for L they are both 3, for T both -1).

Dimensional homogeneity can be useful for:

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

What exactly do we get from Dimensional Analysis?

A single equation,

Which relates all the physical factors of a problem to each other.

An example:

Problem: What is the force, F, on a propeller?

What *might* influence the 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

From this list we can write this equation:

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

or

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

ϕ and ϕ_1 are unknown functions.

Dimensional Analysis produces:

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

These groups are *dimensionless*.

ϕ will be determined by experiment.

These dimensionless groups help to decide what experimental measurements to take.

How do we get the dimensionless groups?

There are several methods.

We will use the strategic method based on:

Buckingham's π theorems.

There are two 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 problem is expressed:

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

Then this can also be expressed

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

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 appear in most of the π groups.

They have a large influence on the problem.

There is great freedom in choosing these.

Some rules which should be followed are

- There are n (= 3) repeating variables.
- In combination they must contain all of dimensions (M, L, T)
- The repeating variables must not form a dimensionless group.
- They do not have to appear in all π groups.
- The should be measurable in an experiment.
- They should be of major interest to the designer.

It is usually possible to take
ρ, u and d

This freedom of choice means:
many different π groups - 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 satisfies the criteria above.

They are:

- measurable,
- good design parameters
- contain all the dimensions M,L and T.

We can now form the three groups according to the 2nd theorem,

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

The π groups are all dimensionless,
i.e. they have dimensions $M^0L^0T^0$

We 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 dimensions

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

The powers for each dimension (M, L or T), the powers must be equal on each side.

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

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

for M: $0 = a_2$

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

$0 = b_2 + c_2$

for T: $0 = -b_2 - 1$

$b_2 = -1$

$c_2 = 1$

Giving π_2 as

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

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

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

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

for 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

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

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

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, extra, unimportant variables are chosen :

* Extra π groups will be formed
* Will have little effect on physical performance
* Should be identified during experiments

If an important variable is missed:

- A π group would be missing.
- Experimental analysis may miss significant behavioural changes.

Initial choice of variables
should be done with great care.

Manipulation of the π groups

Once identified the π groups can be changed.
The number of groups does not change.
Their appearance may change drastically.

Taking the defining equation as:

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

The following changes are permitted:

i. Combination of exiting groups by multiplication or division to form a new group to 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. Reciprocal of any group is valid.

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

iii. A group may be raised to any power.

$$\phi ((\pi_1)^2, (\pi_2)^{1/2}, (\pi_3)^3 \ldots\ldots\ldots \pi_{m-n}) = 0$$

iv. Multiplied by a constant.

v. Expressed as a function of the other groups

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

In general the defining equation could look like

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

An Example

Q. If we have a function describing a problem:

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

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

Ans.

Dimensional analysis using Q, ρ, d will result in:

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

The reciprocal of square root of π_2:

$$\frac{1}{\sqrt{\pi_2}} = \frac{\rho^{1/2} Q}{d^2 p^{1/2}} = \pi_{2a},$$

Multiply π_1 by this new group:

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

then we can say

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

or

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

Common π groups

Several groups will appear again and again.

These often have names.

They can be related to physical forces.

Other common non-dimensional numbers or (π groups):

Reynolds number:

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$ inertial, viscous force ratio

Euler number:

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

Froude number:

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

Weber number:

$$\mathrm{We} = \frac{\rho u d}{\sigma}$$ inertial, surface tension force ratio

Mach number:

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

Similarity

Similarity is concerned with how to transfer measurements from models to the full scale.

Three types of similarity which exist between a model and prototype:

Geometric similarity:

The ratio of all corresponding dimensions in the model and prototype are equal.

For lengths

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

λ_L is the scale factor for length.

For areas

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

All corresponding angles are the same.

Kinematic similarity

The similarity of time as well as geometry.
It exists if:

i. the paths of particles are geometrically similar
ii. the ratios of the velocities of are similar

Some useful ratios are:

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

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

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

A consequence is that streamline patterns are the same.

Dynamic similarity

If geometrically and kinematically similar and the ratios of all forces 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 π group is the same for model and prototype.

The controlling π group is usually Re. So Re is the same for model and prototype:

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

It is possible another group is dominant. In open channel i.e. river Froude number is often taken as dominant.

Modelling and Scaling Laws

Measurements taken from a model needs a scaling law applied to predict the values in the prototype.

An example:

For resistance R, of a body moving through a fluid.
R, is dependent on the following:

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

So

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

Taking ρ, u, l as repeating variables gives:

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

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

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

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

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

W can go no further without some assumptions.

Assuming dynamic similarity, so Reynolds number are the same for both the model and prototype:

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

so

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

i.e. a scaling law for resistance force:

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

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?

Dynamic similarity so Reynolds numbers equal:

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

The model velocity should be

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

Both the model and prototype are in water then,
$\mu_m = \mu_p$ and $\rho_m = \rho_p$ so

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

This is a very high velocity.

This is one reason why model tests are not always done at exactly equal Reynolds numbers.

A wind tunnel could have been used so the values of the ρ and μ ratios would be used in the above.

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?

Earlier we derived an equation for resistance on a body moving through air:

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

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

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

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

For an ideal gas is $p = \rho RT$ so 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

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

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

And the ratio of forces is

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

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

So the drag force on the prototype will be

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

Geometric distortion in river models

For practical reasons it is difficult to build a geometrically similar model.

A model with suitable depth of flow will often be far too big - take up too much floor space.

Keeping Geometric Similarity result in:

- depths and become very difficult to measure;
- the bed roughness becomes impracticably small;
- laminar flow may occur - (turbulent flow is normal in rivers.)

Solution: Abandon geometric similarity.

Typical values are
1/100 in the vertical and 1/400 in the horizontal.

Resulting in:

- Good overall flow patterns and discharge
- local detail of flow is not well modelled.

The Froude number (Fn) is taken as dominant.
Fn can be the same even for distorted models.