INCOMPRESSIBLE FLUID FLOW

Viscous flow - Navier-Stoke's equation (Statement only) - Shear stress, pressure gradient relationship - laminar flow between parallel plates - Laminar flow through circular tubes (Hagen poiseulle's) - Hydraulic and energy gradient - flow through pipes - Darcy -weisback's equation - pipe roughness - friction factor-Moody's diagram-minor losses - flow through pipes in series and in parallel - power transmission - Boundary layer flows, boundary layer thickness, boundary layer separation - drag and lift coefficients.

Real fluids

The flow of real fluids exhibits viscous effect, that is, they tend to "stick" to solid surfaces and have stresses within their body.

You might remember from earlier in the course Newton's law of viscosity:

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

This tells us that the shear stress, τ , in a fluid is proportional to the velocity gradient - the rate of change of velocity across the fluid path. For a "Newtonian" fluid we can write:

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

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

In this lecture we shall look at how the forces due to momentum changes on the fluid and viscous forces compare and what changes take place.

Navier-Stokes Equations

Fluid dynamics is governed by conservation of mass, momentum and energy. For incompressible flow and in the absence of any body forces like gravity etc., the Navier-Stokes Equations are as follows

Conservation of mass:

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

X- momentum:

$$\frac{\partial \mathbf{u}}{\partial \mathbf{t}} + \mathbf{u} \frac{\partial \mathbf{u}}{\partial \mathbf{x}} + \mathbf{v} \frac{\partial \mathbf{u}}{\partial \mathbf{y}} + \mathbf{w} \frac{\partial \mathbf{u}}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial \mathbf{x}} + \frac{\mu}{\rho} \left(\frac{\partial^2 \mathbf{u}}{\partial x^2} + \frac{\partial^2 \mathbf{u}}{\partial y^2} + \frac{\partial^2 \mathbf{u}}{\partial z^2} \right)$$

Y- momentum:

$$\frac{\partial \mathbf{v}}{\partial t} + \mathbf{u} \frac{\partial \mathbf{v}}{\partial \mathbf{x}} + \mathbf{v} \frac{\partial \mathbf{v}}{\partial \mathbf{y}} + \mathbf{w} \frac{\partial \mathbf{v}}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial \mathbf{y}} + \frac{\mu}{\rho} \left(\frac{\partial^2 \mathbf{v}}{\partial \mathbf{x}^2} + \frac{\partial^2 \mathbf{v}}{\partial \mathbf{y}^2} + \frac{\partial^2 \mathbf{v}}{\partial z^2} \right)$$
Z- momentum:

$$\frac{\partial \mathbf{w}}{\partial t} + \mathbf{u} \frac{\partial \mathbf{w}}{\partial \mathbf{x}} + \mathbf{v} \frac{\partial \mathbf{w}}{\partial \mathbf{y}} + \mathbf{w} \frac{\partial \mathbf{w}}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \frac{\mu}{\rho} \left(\frac{\partial^2 \mathbf{w}}{\partial \mathbf{x}^2} + \frac{\partial^2 \mathbf{w}}{\partial \mathbf{y}^2} + \frac{\partial^2 \mathbf{w}}{\partial z^2} \right)$$

The first term of the left hand side is the term arising due to accumulation of momentum. For steady state this term will be zero. The rest of the terms are known as *convective terms*. The first term of right hand side is the *pressure term*. Rest of the term in right hand side is known as *diffusion term* and arises due to presence of *viscous force (shear force)*. This term becomes zero if the fluid is ideal (or irrotational).

Hence for *steady incompressible and irrotational flow* the Navier-Stokes equations become

Conservation of mass:

	$\frac{\partial}{\partial t}$	$\frac{\partial \mathbf{u}}{\partial \mathbf{x}} + \frac{\partial \mathbf{v}}{\partial \mathbf{y}} +$	$\frac{\partial w}{\partial z} = 0$)
X- momentum:		5		
	$u \frac{\partial u}{\partial x} +$	$v \frac{\partial u}{\partial y} + w$	$\frac{\partial \mathbf{u}}{\partial \mathbf{z}} = -$	$\frac{1}{\rho} \frac{\partial p}{\partial x}$
Y- momentum:				
	$u \frac{\partial v}{\partial x} +$	$-v \frac{\partial v}{\partial v} + w$	$\frac{\partial \mathbf{v}}{\partial \mathbf{z}} = -$	$-\frac{1}{\rho}\frac{\partial p}{\partial y}$
Z- momentum:		2		
	$u \frac{\partial w}{\partial x} +$	$v \frac{\partial w}{\partial y} + w$	$\frac{\partial w}{\partial z} =$	$-\frac{1}{\rho}\frac{\partial p}{\partial z}$
This is also known as Eular's	aquatio	- 		-

This is also known as Euler's equation.

Laminar and turbulent flow

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

This



Turbulent

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

The top situation is known as **turbulent** flow and the lower as **laminar** flow.



In laminar flow the motion of the particles of fluid is very orderly with all particles moving in straight lines parallel to the pipe walls.

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

The phenomenon was first investigated in the 1880s by Osbourne Reynolds in an experiment which has become a classic in fluid mechanics.

He used a tank arranged as above with a pipe taking water from the centre into which he injected a dye through a needle. After many experiments he saw that this expression

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

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

would help predict the change in flow type. If the value is less than about 2000 then flow is laminar, if greater than 4000 then turbulent and in between these then in the transition zone.



This value is known as the Reynolds number, Re:

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

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

What are the units of this Reynolds number? We can fill in the equation with SI units:

$$\rho = \text{kg/m}^3$$
, $u = \text{m/s}$, $d = \text{m}$, $\mu = \text{Ns/m}^2 = \text{kg/ms}$
 $Re = \frac{\rho u d}{\mu} = \frac{kg}{m^3} \frac{m}{s} \frac{m}{1} \frac{ms}{kg} = 1$

i.e. it has **no units**. A quantity that has no units is known as a **non-dimensional** (or dimensionless) quantity. Thus the Reynolds number, Re, is a non-dimensional number.

We can go through an example to discover at what velocity the flow in a pipe stops being laminar.

If the pipe and the fluid have the following properties:

Water density $\rho = 1000 \text{ kg/m}^3$ Pipe diameterd = 0.5 m

(Dynamic) viscosity $\mu = 0.55 \times 10^{-3} \text{ Ns/m}^2$

We want to know the maximum velocity when Re is 2000.

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

If this were a pipe in a house central heating system, where the pipe diameter is typically 0.015m, the limiting velocity for laminar flow would be, 0.0733 m/s.

Both of these are very slow. In practice it very rarely occurs in a piped water system - the velocities of flow are much greater. Laminar flow does occur in situations with fluids of greater viscosity - e.g. in bearing with oil as the lubricant.

At small values of Re above 2000 the flow exhibits small instabilities. At values of about 4000 we can say that the flow is truly turbulent. Over the past 100 years since this experiment, numerous more experiments have shown this phenomenon of limits of Re for many different Newtonian fluids - including gasses.

What does this abstract number mean?

We can say that the number has a physical meaning, by doing so it helps to understand some of the reasons for the changes from laminar to turbulent flow.

$$Re = \frac{\rho v d}{\mu} = \frac{inertia\ force}{viscous\ force}$$

It can be interpreted that when the inertial forces dominate over the viscous forces (when the fluid is flowing faster and Re is larger) then the flow is turbulent. When the viscous forces are dominant (slow flow, low Re) they are sufficient enough to keep all the fluid particles in line, then the flow is laminar. Reynolds number is essentially a means of comparing one flow with another and provided that the corresponding lengths and velocities are compared in two flows, the particular choices of length and velocity do not matter.

In summary:

Laminar flow

- Re < 2000
- 'low' velocity

- Dye does not mix with water
- Fluid particles move in straight lines
- Simple mathematical analysis possible
- Rare in practice in water systems.

Transitional flow

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

Turbulent flow

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

<u>Shear stress, pressure gradient relationship</u>.

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

In a pipe with a real fluid flowing, at the wall there is a shearing stress retarding the flow, as shown below.



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

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



We can do some analysis to express this loss in pressure in terms of the forces acting on the fluid.

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



The pressure at the upstream end is p and at the downstream end pressure has fallen by Δp to $(p - \Delta p)$. The driving force due to pressure (f = Pressure x area) can then be written as

Driving force = Pressure force at 1 – Pressure force at 2

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

The retarding force is that due to the shear stress by the walls

= shear stress x area over which it acts

$$= \tau_{\rm w} x$$
 area of pipe wall

$$= \tau_{\rm w} \pi dL$$

As the flow is in equilibrium

Driving force = retarding force

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

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

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



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



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

turbulent

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

Pressure loss during laminar flow in a pipe

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

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



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

We are in equilibrium, so the shearing forces on the cylinder equal the pressure forces. Pressure force at the upstream face of control volume = $p \pi r^2$

Pressure force at the downstream face of control volume = $\left(p + \frac{\partial p}{\partial x} \Delta x\right) \pi r^2$ The shear force acting on the control volume = $\tau \times 2\pi r \times \Delta x$ Hence,

$$p\pi r^{2} - \left(p + \frac{\partial p}{\partial x}\Delta x\right)\pi r^{2} - \tau \times 2\pi r \times \Delta x = 0$$

or

$$-\frac{\partial p}{\partial x}\,\Delta x\,\pi r^2 -\,\tau\,2\pi r\,\Delta x=0$$

or

Or

$$\tau = -\frac{\partial p}{\partial x}\frac{r}{2}$$

By Newton's law of viscosity we have $= \mu \frac{du}{dy}$, where y is the distance from the wall. As we are measuring from the pipe centre then y = R - r and dy = -dr, giving

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

Which can be combined with the equation above to give

$$-\frac{\partial p}{\partial x}\frac{r}{2} = -\mu \frac{du}{dr}$$
$$\frac{du}{dr} = \frac{\partial p}{\partial x}\frac{r}{2\mu}$$

Integrating we get

$$u = \frac{\partial p}{\partial x} \frac{1}{2\mu} \int r \, dr$$

The velocity at any point at a distance r from the centre is given by

$$u_r = \frac{\partial p}{\partial x} \frac{r^2}{4\mu} + 0$$

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

$$C = -\frac{\partial p}{\partial x} \frac{R^2}{4\mu}$$

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

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

Note that this is a parabolic profile and the velocity profile in the pipe looks similar to the figure below.



The maximum velocity is given by

$$u_{max} = -\frac{1}{4\mu} \frac{\partial p}{\partial x} R^2$$

What is the discharge in the pipe?

$$Q = \int_0^R dQ = \int_0^R u_r \times 2\pi r \, dr$$
$$= \int_0^R -\frac{1}{4\mu} \frac{\partial p}{\partial x} \left(R^2 - r^2\right) 2\pi r \, dr$$
$$= \frac{1}{4\mu} \left(-\frac{\partial p}{\partial x}\right) 2\pi \int_0^R (R^2 - r^2) \, r \, dr$$

 $Q = \frac{\pi}{8\mu} \left(-\frac{\partial P}{\partial x} \right) R^4$

or

Unit 3

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

The average velocity is given by

$$u_{av} = \frac{Q}{area} = \frac{\frac{\pi}{8\mu} \left(-\frac{\partial p}{\partial x}\right) R^2}{\pi R^2} = \frac{1}{8\mu} \left(-\frac{\partial p}{\partial x}\right) R^2$$

Hence,

$$\frac{u_{max}}{u_{ay}} = 2.0$$

We are interested in the pressure loss and want to relate this to the velocity of the flow.

$$u_{av} = \frac{1}{8\mu} \left(-\frac{\partial p}{\partial x} \right) R^2 \quad or \quad -\frac{\partial p}{\partial x} = \frac{8\mu u_{av}}{R^2}$$

Or

$$-\int_{2}^{1} dp = \int_{2}^{1} \frac{8\mu u_{av}}{R^{2}} dx$$

Or

$$-(p_1 - p_2) = \frac{8\mu u_{av}}{R^2} (x_1 - x_2)$$

Or

$$(p_1-p_2)=\frac{8\mu u_{av}}{R^2}L$$

i.e.

$$\Delta p = \frac{32\mu u_{av}}{D^2} L$$

In terms of head,

$$h_f = \frac{\Delta p}{\rho g} = \frac{32\mu u_{av}}{\rho g D^2} L$$

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

It has been validated many times by experiment.

- It justifies two assumptions:
- 1. fluid does not slip past a solid boundary

2. Newton's hypothesis.

Steady Laminar Flow between Parallel Planes (Plane Poiseuille Flow).

A flow is called parallel if only one velocity component is different from zero, all fluid particles moving in one direction only.

i.e. v = w = 0

From continuity, $\frac{\partial u}{\partial x} = 0$, i.e. the component of u does not depend on x. As the flow is laminar, there is no movement of fluid in any direction perpendicular to the flow and thus p varies only in the direction of flow.

i.e.
$$\frac{\partial p}{\partial y} = \frac{\partial p}{\partial z} = 0$$

Let us consider a small element with sides parallel to the coordinate axes. Let the lower face of the element be at a distance y from the lower plane and velocity be u. At the upper face of the element, at a distance $y + \delta y$ from the lower plane, the velocity is $u + \delta u$.



If δu is positive, the faster moving fluid just above the element exerts a force on the upper face. Similarly, the slower-moving fluid adjacent to the lower face tends to retard the element. Thus, there are stresses of magnitude τ on the lower face and $\tau + \delta \tau$ on the upper face. Let the piezometric pressure be p on the upstream face and $p + \delta p$ on the downstream face. Hence, by balancing the forces we get

$$\{p - (p + \delta p)\} \, \delta y + \{(\tau + \delta \tau) - \tau\} \, \delta x = 0 -\delta p \, \delta y + \delta \tau \, \delta x = 0$$

Or

$$\frac{\delta p}{\delta x} = \frac{\delta \tau}{\delta y}$$

In the limit, we get

$$\frac{dp}{dx} = \frac{\partial \tau}{\partial y} = \frac{\partial}{\partial y} \left(\mu \frac{\partial u}{\partial y} \right)$$

since the pressure varies only in the direction of flow $\frac{dp}{dx}$ is independent of y. Hence, integrating the equation with respect to y, we get

$$\frac{dp}{dx}y = \left(\mu\frac{\partial u}{\partial y}\right) + A$$

Or

$$\frac{dp}{dx}\frac{y^2}{2} = \mu u + Ay + B$$

Boundary conditions:

$$u = 0$$
 at $y = 0$ and $2b$

$$\frac{\partial u}{\partial y} = 0 \text{ at } y = b$$

Hence, we get B=0 and $A = \frac{dp}{dx} b$ Hence,

$$u = \frac{1}{\mu} \left[\frac{dp}{dx} \frac{y^2}{2} - \frac{dp}{dx} by \right]$$

Or

$$u = \frac{y^2}{2\mu} \left(\frac{dp}{dx}\right) \left[1 - 2\left(\frac{b}{y}\right)\right]$$

Hence, the velocity profile is parabolic and the velocity is maximum at the centre i.e. y = b.

$$u_{max} = -\frac{b^2}{2\mu} \left(\frac{dp}{dx}\right) = \frac{b^2}{2\mu} \left(-\frac{dp}{dx}\right)$$

Total discharge

$$Q = \int_{0}^{2b} u dy = \int_{0}^{2b} \frac{1}{2\mu} \left(\frac{dp}{dx}\right) (y^{2} - 2by) dy$$
$$= \frac{1}{2\mu} \left(\frac{dp}{dx}\right) \left[\frac{y^{3}}{3} - \frac{2by^{2}}{2}\right]_{0}^{2b}$$
$$= \frac{4b^{3}}{6\mu} \left(-\frac{dp}{dx}\right)$$

Hence,

$$u_{av} = \frac{Q}{A} = \frac{b^2}{3\mu} \left(-\frac{dp}{dx} \right)$$

$$\therefore \quad u_{max} = 1.5 u_{av}$$
$$\frac{dp}{dx} = -\frac{3}{2} \frac{Q\mu}{b^3} = -\frac{3\mu u_{av}}{b^2}$$

The pressure drop relation can be expressed in dimensionless form using friction factor and Reynolds number.

Friction factor,

$$f = \frac{\tau}{\frac{1}{2}\rho u^2} = \frac{-\left(-\frac{dp}{dx}\right)2b}{\frac{1}{2}\rho u^2} = \frac{6\mu u_{ab} \times 2b}{\rho b^2 u_{av}^2} = \frac{12\mu}{\rho b u_{av}}$$

Now,

$$Re = \frac{\rho u_{av} 2b}{\mu}$$

Hence,

$$fRe = \frac{12\mu}{\rho b u_{av}} \times \frac{\rho u_{av} 2b}{\mu} = 24$$

i.e. for a plane Poiseuille flow, f.Re = 24.

Head loss due to friction in a Pipe



Consider a small element of length L. On the left hand face the pressure is p_1 and on the right hand face pressure is p_2 . The shear stress acting on the element is τ . Balancing the forces on the element we get,

 $p_1A - p_2A - \tau Lp = 0$ where, P is the wetted perimeter.

or
$$A(p_1 - p_2) = \tau LP$$

or

$$\left(\frac{p_1 - p_2}{L}\right)A = \tau \times P$$

Dividing both side by $\frac{\pi d^2}{4}$ L, we get

$$\frac{\Delta p}{L} = \tau \frac{P}{A} = \frac{\tau}{m}$$
 , where $m = \frac{A}{P} = \frac{\frac{n}{4}d^2}{\pi d} = \frac{d}{4}$

Now the shear stress τ can be expressed in terms of skin friction c_f as

$$\tau = f\left(\frac{1}{2}\rho u^2\right)$$

Hence,

$$\Delta p = f \frac{L}{m} \left(\frac{1}{2} \rho u^2 \right) = 4f \frac{L}{d} \left(\frac{1}{2} \rho u^2 \right)$$

The quantity f is known as Dercy friction factor.

Dercy-Weisbach Equation:-

$$\Delta p = 4f \frac{L}{d} \left(\frac{1}{2}\rho u^2\right)$$

i.e. pressure loss due to friction = $4f \frac{L}{d} \times dynamic pressure$. Dividing both sides by ρg we get

$$h_f = \frac{\Delta p}{\rho g} = 4f \frac{L}{d} \frac{u^2}{2g}$$

i.e. head loss due to friction = $4f \frac{L}{d} \times dynamic$ head.

The *pressure or head loss* is *proportional* to *pipe length* and *inversely proportional* to *diameter*. The constant or proportionality is called friction factor. Very important note:-

There is lot of disagreement about what is meant by "friction factor" and what symbol should be used to denote it. What is represented here by f is also denoted by λ by some authors. Also the head loss is expressed in some book as

$$h_f = f \frac{L}{d} \frac{u^2}{2g}$$

Be very wary of the definition. You should be able to distinguish it by the expression for friction factor in laminar flow: $\frac{64}{R_e}$ with the notation here i.e. if you use

$$h_f = f \frac{L}{d} \frac{u^2}{2g}$$

and $\frac{16}{Re}$ if you use

$$h_f = 4f \frac{L}{d} \frac{u^2}{2g}$$

Turbulent flow:

In turbulent flow there is no longer an explicit relationship between the mean shear stress τ and mean velocity gradient $\frac{du}{dr}$ because a far greater transfer of momentum arises from the net effect of random fluctuations than the relatively small viscous forces. Hence, to relate the head loss we require an empirical relation connecting the wall shear stress and the average velocity of the pipe.

Results of extensive experimentation led to the establishment of the following:

- 1. $h_f \propto L$
- 2. $h_f \propto u^2$
- 3. $h_f \propto \frac{1}{d}$
- 4. h_f depends on the surface roughness of pipe wall
- 5. h_f depends on the fluid density and viscosity
- 6. h_f is independent of pressure.

Expressed in a form suitable for dimensional analysis this implies that

$$f = \emptyset \left(u, d, \rho, \mu, \epsilon, \epsilon', \alpha \right)$$

where, ϵ =size of the wall roughness, ϵ' = is a measure of the spacing of roughness particles, α =is a form factor, a dimensionless parameter whose value depends on the shape of the roughness particles.

In general for rough pipe, dimensional analysis yields an expression

$$f = \emptyset_2\left(\frac{\rho u d}{\mu}, \frac{\epsilon}{d}, \frac{\epsilon'}{d}, \alpha\right)$$

Or in terms of Reynolds number

$$f = \emptyset_2\left(Re, \frac{\epsilon}{d}, \frac{\epsilon'}{d}, \alpha\right)$$

Expressions for friction factor f: Laminar flow (theory)

$$f = \frac{16}{\text{Re}}$$

Turbulent Flow (Smooth or rough pipe)

Nikuradse (1933) used sand grains to roughen pipe surfaces. He defined a relative roughness ϵ/d (or in some books as k_s/d . His experimental curves for the friction factor

showed 5 regions:

- 1. Laminar flow (Re < Recrit \approx 2000: roughness irrelevant)
- 2. Laminar-to-turbulent transition (2000< Re < 4000)
- 3. Smooth wall (f is a function of Re only)
- 4. Fully-rough walls (f is a function of roughness only)
- 5. Intermediate roughness (f is a function of both Re and ϵ/d)

For smooth pipe:

Blasius's correlation

$$f = \frac{0.079}{Re^{0.25}}$$

Prandtl correlation

$$\frac{1}{\sqrt{4f}} = 2.01 \log_{10} \frac{Re \sqrt{4f}}{2.51}$$

For Rough wall pipe:

Von Karman correlation

$$\frac{1}{\sqrt{4f}} = 2.0 \log_{10} \frac{3.7 d}{\epsilon}$$

For most of the commercial pipe, both roughness and Reynolds number are important. Colebrook and White combined smooth and roughness laws to obtain the following formula known as Colebrook-White formula

$$\frac{1}{\sqrt{4f}} = -2.0 \log_{10} \left(\frac{\epsilon}{3.7d} + \frac{2.51}{Re\sqrt{4f}}\right)$$

This is the main formula for friction factor. The main difficulty is that f appears both the sides of the equation and hence has to be determined iteratively.

Moody Chart

Graphical solution of the Colebrook-White relation.



Minor loses

Pipeline systems are subject to two types of losses:

- *frictional losses*, also called *major losses*, due to wall, contributing a continuous fall in head over a large distance;
- *minor loses* due to abrupt changes in geometry; e.g. pipe junctions, bends, valves, fittings of all kind.

Each type of loss can be quantified using a *loss coefficient K*, the ratio of pressure loss to dynamic pressure (or head loss to dynamic head):

Pressure loss =
$$K\left(\frac{1}{2}\rho u^2\right)$$
 or head loss = $K\frac{u^2}{2g}$

Pipeline friction is just one type of loss, for which $K = f \frac{L}{d}$

In long pipelines the minor losses may be neglected in comparison with the friction loss. But for a short pipeline, these minor losses actually outweigh the friction loss. The minor losses arise from sudden changes of velocity (either in magnitude or direction). These changes generate large-scale turbulence in which energy is dissipated as heat. The total head loss in a pipeline may be calculated as the sum of normal friction for the pipe considered and additional losses (i.e. minor losses). Loss due to Sudden Expansion:



Let us consider a fluid control volume BCDEFG as shown in the figure. The net force acting towards the right on the control volume is

 $p_1A_1 + p'(A_2 - A_1) - p_2A_2$ where p' represents the mean pressure of the eddying fluid over the annular face GD. We assume p' is equal to p_1 . The net force is thus $(p_1 - p_2)A_2$. From Newton's second

law, the net force equal to rate of change of momentum, i.e. $(p_1 - p_2)A_2 = \rho Q(u_2 - u_1)$

$$(p_1 - p_2) = \rho \frac{Q}{A_2}(u_2 - u_1) = \rho u_2(u_2 - u_1)$$

From the energy equation for a constant density fluid we have

$$\frac{p_1}{\gamma} + \frac{u_1^2}{2g} + z = \frac{p_2}{\gamma} + \frac{u_2^2}{2g} + z + h_1$$

or

 $h_{l} = \frac{p_{1} - p_{2}}{\gamma} + \frac{u_{1}^{2} - u_{2}^{2}}{2g} = \frac{u_{2}(u_{2} - u_{1})}{g} + \frac{u_{1}^{2} - u_{2}^{2}}{2g} = \frac{(u_{1} - u_{2})^{2}}{2g}$ From continuity $A_1u_1 = A_2u_2$ Hence,

$$h_{l} = \frac{u_{1}^{2}}{2g} \left(1 - \frac{A_{1}}{A_{2}}\right)^{2} = \frac{u_{2}^{2}}{2g} \left(\frac{A_{2}}{A_{1}} - 1\right)^{2}$$

or

$$h_l = K \; rac{u_1^2}{2g}$$
 where $K = \left(1 - rac{A_1}{A_2}\right)^2$

Exit loss:

If $A_2 \rightarrow \infty$, then K becomes equal to 1 and the above equation becomes



$$h_l = K \ \frac{u_1^2}{2g}$$

This happens at the outlet of a submerged pipe discharged into a large reservoir as shown in the figure.

Loss due to Sudden Contraction:

The loss due to sudden contraction is given by

$$h_{l} = \frac{u_{2}^{2}}{2g} \left(\frac{A_{2}}{A_{c}} - 1\right)^{2} = \frac{u_{2}^{2}}{2g} \left(\frac{1}{C_{c}} - 1\right)^{2} = K \frac{u_{2}^{2}}{2g}$$

Where A_c is the area of vena-contracta and C_c is the contraction coefficient. The values of loss coefficients are given in the following table.

d_2/d_1	0	0.2	0.4	0.6	0.8	1.0
K	0.5	0.45	0.38	0.28	0.14	0

Entry loss:

If $A_1 \rightarrow \infty$, then K becomes equal to 0.5 and the above equation becomes



$$h_l = 0.5 \ \frac{u_1^2}{2g}$$

2gThis is limiting case corresponds to a flow from large reservoir into a sharp-edge pipe as shown in the figure.

Equivalent Length for Pipe Fitting Loss Calculation

The loss coefficient K may also be defined in terms of an equivalent length of straight pipe, of same diameter as that including the fitting, that would result in the same frictional loss as that incurred by flow separation through fitting. That is

$$h_f = 4f \frac{L_e}{d} \frac{u^2}{2g} = K \frac{u^2}{2g}$$

where L_e is the equivalent length of pipe that would yield a friction loss equivalent to the particular fitting.

Thus

$$L_e = \frac{Kd}{4f}$$

where f is known. L_e can be expressed as 'n diameters' i.e. $n = \frac{L_e}{d}$. The value of L_e thus depends on the value of f and therefore on Reynolds number and the roughness of the pipe.

Hence total pressure drop through a pipeline of length L and diameter d can be expressed as

$$h_f = 4f \, \frac{(L+L_e)}{d} \, \frac{u^2}{2g}$$

Typical values of loss coefficients are given below.

Commercial pipe fittings (a	pproxin	nate
Fitting	K	
Globe valve	10	
Gate valve – wide open	0.2	
Gate valve – ½ open	5.6	
Pump foot valve	1.5	
90° elbow	0.9	
45° elbow	0.4	
side outlet of T-junction	1.8	

Entry/exit losses		
Configuration	K	
Bell-mouthed entry	0	
Abrupt entry	0.5	
Protruding entry	1.0	
Bell-mouthed exit	0.2	
Abrupt enlargement	0.5	
Exit to atmosphere*	1.0	

Pipeline Calculation

The objective is to establish the relationship between available head and quantity of flow. Available head = sum of head losses along the pipe.

Available head is the overall drop in head from start to end of pipe (often the difference between still-water levels), sometimes supplemented by additional pumping head. Head losses are proportional to the dynamic head $u^2/_{2,g}$. Fluid then flows through the pipe at

precisely the right velocity u (or discharge Q) so that the above criteria is satisfied.

Pipe parameters are illustrated below. Although a reservoir is indicated at each end of the pipe, this is simply a diagrammatic way of saying "a point at which the total head is known".



Typical pipeline problems are: given two of the following parameters, find the third.

Head loss: h Quantity of flow: Q Diameter: d

Other parameters: length *L*, roughness ϵ , kinematic viscosity v ad minor loss coefficient *K*.

Calculation involve:

(1) Head losses

e.g. with friction factor f and minor loss coefficient K:

$$h = \left(4f\frac{L}{d} + K\right)\frac{u^2}{2g}$$

(2) Expression for loss coefficients:

e.g. friction losses f and minor loss coefficient K which can be obtained from the methods described earlier.

Pressure Head, Velocity Head, Potential Head and Total Head.

By looking again at the example of the reservoir with which feeds a pipe we will see how these different *heads* relate to each other.

Consider the reservoir below feeding a pipe which changes diameter and rises (in reality it may have to pass over a hill) before falling to its final level.



Reservoir feeding a pipe

To analyses the flow in the pipe we apply the Bernoulli equation along a streamline from point 1 on the surface of the reservoir to point 2 at the outlet nozzle of the pipe. And we know that the *total energy per unit weight* or the *total head* does not change - it is **constant** - along a streamline. But what is this value of this constant? We have the Bernoulli equation

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

We can calculate the total head, H, at the reservoir, $p_1 = 0$ as this is atmospheric and atmospheric gauge pressure is zero, the surface is moving very slowly compared to that in the pipe so $u_1 = 0$, so all we are left with is *total head* = $H = z_1$ the elevation of the reservoir.

A useful method of analysing the flow is to show the pressures graphically on the same diagram as the pipe and reservoir. In the figure above the *total head* line is shown. If we attached piezometers at points along the pipe, what would be their levels when the pipe nozzle was closed? (Piezometers, as you will remember, are simply open ended vertical tubes filled with the same liquid whose pressure they are measuring).



Piezometer levels with zero velocity

As you can see in the above figure, with zero velocity all of the levels in the piezometers are equal and the same as the total head line. At each point on the line, when u = 0

$$\frac{p}{\gamma} + z = H$$

The level in the piezometer is the pressure head and its value is $\frac{p}{\gamma}$ or $\frac{p}{\rho g}$.

What would happen to the levels in the piezometers (pressure heads) if the if water was flowing with velocity = u? We know from earlier examples that as velocity increases so pressure falls ...



Piezometer levels when fluid is flowing

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

We see in this figure that the levels have reduced by an amount equal to the velocity head $u^2/2g$. Now as the pipe is of constant diameter we know that the velocity is constant along the pipe, so the velocity head is constant and represented graphically by the horizontal line shown (this line is known as the hydraulic grade line).

What would happen if the pipe were not of constant diameter? Look at the figure below where the pipe from the example above is replaced be a pipe of three sections with the middle section of larger diameter



Piezometer levels and velocity heads with fluid flowing in varying diameter pipes

The velocity head at each point is now different. This is because the velocity is different at each point. By considering continuity we know that the velocity is different because the diameter of the pipe is different. Which pipe has the greatest diameter? Pipe 2, because the velocity, and hence the velocity head, is the smallest.

This graphical representation has the advantage that we can see at a glance the pressures in the system. For example, where along the whole line is the lowest pressure head? It is where the hydraulic grade line is nearest to the pipe elevation i.e. at the highest point of the pipe.

Energy and Hydraulic Grade Line

Energy grade line or Total Energy Line and hydraulic grade line are the graphical means of portraying the energy changes in the pipe lines.

Three elevations may be drawn:

Pipe centerline	Z	geometric height
Hydraulic grade line (HGL)	$\frac{p}{\rho g} + z$	piezometric height
Energy grade line (EGL)	$\frac{p}{\rho g} + \frac{u^2}{2g} + Z$	total head.

where *p* is the gauge pressure.

Energy losses due to friction.

In a real pipe line there are energy losses due to friction - these must be taken into account as they can be very significant. How would the pressure and hydraulic grade lines change with friction? Going back to the constant diameter pipe, we would have a pressure situation like this shown below



Hydraulic Grade line and Total head lines for a constant diameter pipe with friction

How can the total head be changing? We have said that the total head - or total energy per unit weight – is constant. We are considering energy conservation, so if we allow for an amount of energy to be lost due to friction the total head will change. We have seen the equation for this before. But here it is again with the energy loss due to friction written as a *head* and given the symbol h_f . This is often known as the *head loss due to friction*.

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

Energy Grade Line (EGL) or Total Energy Line (TEL)

- Shows the change in *total head* along the pipeline
- Starts and ends at still-water levels
- Small discontinuities correspond to entry loss, exit loss or other minor losses. Steady downward slope reflects pipe friction (slope change if pipe radius changes; Large discontinuities correspond to turbines (loss of head) or pump (gain of head).
- The EGL represents the maximum height to which water may be delivered.

Hydraulic Grade line (HGL)

- Shows the change in piezometric head along the pipeline.
- For pipe flow the HGL lies a distance p/pg above the pipe centerline. Thus, the difference between pipe elevation and hydraulic grade line gives the static pressure p. If the HGL drops below pipe elevation, this means negative gauge pressures (i.e. less than atmospheric). A HGL more than p_{atm}/pg (≈ 10 m of water) below the pipeline is impossible.
- The HGL is the height to which the liquid would rise in a piezometer tube.

The EGL is always the dynamic head $u^2/_{2g}$ above the HGL. For uniform pipes (constant

u), the two grade lines are parallel.

Illustrations





Examples:



Two reservoirs A and B are connected by pipeline 1 and 2 as shown in the figure. The level difference between the liquid for reservoirs A and B is H. Let u be the velocity, L the length, d the diameter of the pipe. The liquid will flow from reservoir A to B due to the level difference. Since the driving head is H, this should be equal to all the losses in the pipes.

The losses consists of

a) Entry loss between reservoir A to pipe 1 which is equal to

$$0.5 \ \frac{u_1^2}{2g}$$

b) Friction loss in pipe 1 which can be written as

$$4f \ \frac{L_1}{d_1} \ \frac{u_1^2}{2g}$$

c) Loss due to sudden expansion in pipe 1 and 2

$$\frac{(u_1-u_2)^2}{2g}$$

d) Friction loss in pipe 2

e) Exit loss to Reservoir B

$$4f \frac{L_2}{d_2} \frac{u_2^2}{2g}$$
$$\frac{u_2^2}{2g}$$

The total loss consists is the sum of all the losses mentioned above and equals to H. Since u_1 and u_2 are related by $A_1u_1 = A_2u_2$, either u_1 or u_2 and hence the discharge Q may be determined if H, pipe lengths, diameters and friction factors are known.

Problem on pipes in series

Problem 1: Two reservoirs A and B have a difference level of 9 m and are connected by pipeline over the first part, which is 15 m long and then 250 mm diameter for the remaining 45 m length. The entrance to and exit from the pipes are sharp and the change in section between pipe 1 and 2 is sudden. The friction coefficient f is 0.01 for both pipes. List the losses of head and calculate system flow rate and hydraulic gradient.

Solution:

(iv)

- a) Head losses:
- (i) Entry loss to pipe 1:

$$h_1 = 0.5 \ \frac{u_1^2}{2g}$$

(ii) Friction loss to pipe 1:

$$h_{f1} = 4f \ \frac{L_1}{d_1} \frac{u_1^2}{2g} = 4 \times 0.01 \times \frac{15}{0.2} \times \frac{u_1^2}{2g} = 3 \ \frac{u_1^2}{2g}$$

(iii) Loss due to sudden expansion:

$$h_2 = \frac{(u_1 - u_2)^2}{2g} = \left(1 - \frac{A_1}{A_2}\right)^2 \frac{u_1^2}{2g} = 0.1296 \frac{u_1^2}{2g}$$

Friction loss in pipe 2

$$h_{f2} = 4f \ \frac{L_2}{d_2} \ \frac{u_2^2}{2g} = 4 \times 0.01 \times \frac{45}{0.25} \times \left(\frac{A_1}{A_2}\right)^2 \ \times \ \frac{u_1^2}{2g} = 2.949 \ \frac{u_1^2}{2g}$$

(v) Exit loss:

$$h_3 = \frac{u_2^2}{2g} = \left(\frac{A_1}{A_2}\right)^2 \times \frac{u_1^2}{2g} = 0.4096 \times \frac{u_1^2}{2g}$$

Hence, total head loss = $h_1 + h_{f1} + h_2 + h_{f2} + h_3$ Or

$$9 = (0.5 + 3 + 0.1296 + 2.929 + 0.4096) \frac{u_1^2}{2g} = 6.9682 \frac{u_1^2}{2 \times 9.806} = 0.3553 u_1^2$$

Or $u_1 = 5.033$ m/s Hence Volume flow rate is given by $Q = A_1 u_1 = \frac{\pi}{2} (0.2)^2 \times 10^{-10}$

$$Q = A_1 u_1 = \frac{\pi}{4} (0.2)^2 \times 5.033 = 0.1581 \, m^3 / s$$

Hence,

$h_1 = 0.6458 m;$	$h_2 = 0.1674 m;$	$h_3 = 0.529 m$
h _{f1} = 3.8748 m;	$h_{f2} = 3.809 \text{ m}$	





Water discharges from reservoir A through a 100 mm diameter pipe 15 m long which rises to its highest point B, 1.5 m above the free surface of the reservoir, and discharges direct to atmosphere C, 4 m below the free surface at A. The length of pipe L_1 from A to B is 5 m and length of pipe L_2 from B to C is 10 m. Both entrance and exit are sharp and

the value of f is 0.08. Calculate a) mean velocity of water leaving pipe at C and b) the pressure in the pipe at B.

Solution:

a) Determination of velocity u

Total energy at A = Total energy at C + losses

$$\frac{p_A}{\gamma} + \frac{u_A^2}{2g} + Z_A = \frac{p_C}{\gamma} + \frac{u_C^2}{2g} + Z_C + losses$$

Now at both A and C, the pressure is atmospheric and hence

$$p_A = p_C = 0$$
 (gauge pressure)

Velocity at A, $u_A = 0$ Hence,

$$Z_A = \frac{u_C^2}{2g} + Z_C + losses$$

Or

$$Z_A - Z_C = \frac{u_C^2}{2g} + losses$$

Entry loss:

$$h_1 = 0.5 \ \frac{u_C^2}{2g}$$

Pipe friction loss:

$$h_f = 4f \; \frac{(L_1 + L_2)}{d} \; \frac{u_c^2}{2g}$$

There is no exit loss because the water is discharged into atmosphere without any change of cross-section.

Hence,

$$Z_A - Z_C = \frac{u_C^2}{2g} \left(1 + 0.5 + 4 f \frac{L_1 + L_2}{d} \right)$$

Or

$$4 = \frac{u_c^2}{2g} \left(1 + 0.5 + 4 \times 0.08 \times \frac{5 + 10}{0.1} \right)$$

u_c = 1.26 m/s.

Or

b) To find the pressure at B, P_B

$$\frac{p_A}{\gamma} + \frac{u_A^2}{2g} + Z_A = \frac{p_B}{\gamma} + \frac{u_B^2}{2g} + Z_B + losses$$

Or

$$Z_A = \frac{p_B}{\gamma} + \frac{u_B^2}{2g} + Z_B + 0.5 \frac{u_B^2}{2g} + 4f \frac{L_1}{d} \frac{u_B^2}{2g}$$

Since the pipe diameter is same, $u_B = u_C = u = 1.26$ m/s

Hence,

$$p_B = \gamma (Z_A - Z_B) - \frac{\gamma u^2}{2g} \left(1 + 0.5 + 4f \frac{L_1}{d} \right)$$

= - 28.58 kN/m² i.e 28.58 kN/m² below atmospheric pressure

Problem on parallel pipes

Two sharp-edged pipes of diameter $d_1 = 50$ mm and $d_2 = 100$ mm each of length 100 m are connected in parallel between two reservoirs which have a difference of level h = 10 m. If Darcy coefficient f = 0.008 for each pipe, calculate: a) the rate of flow through each pipe, b) the diameter d of a single pipe of 100 m long which would give the same flow if it was substituted for the original two pipes.



For flow in pipe 1

$$\frac{p_A}{\gamma} + \frac{u_A^2}{2g} + z_A = \frac{p_B}{\gamma} + \frac{u_B^2}{2g} + z_B + \left(0.5\frac{u_1^2}{2g} + 4f\frac{L_1}{d_1}\frac{u_1^2}{2g} + \frac{u_1^2}{2g}\right)$$

Since $p_A = p_B$ = atmospheric pressure, we can neglect these terms and if reservoirs are lerge u_A and u_B are negligible. Hence,

$$z_A - z_B = \left(1.5 + 4f \frac{L_1}{d_1}\right) \frac{u_1^2}{2g}$$

Or

$$10 = \left(1.5 + \frac{4 \times 0.008 \times 100}{0.05}\right) \frac{u_1^2}{2g}$$

Hence, $u_1 = 1.731 \text{ m/s}$

$$Q_1 = \frac{\pi}{4} (d_1)^2 \times u_1 = \frac{\pi}{4} (0.05)^2 \times 1.731 = 0.003 \frac{m^3}{s}$$

Similarly for pipe 2

or

$$10 = \left(1.5 + \frac{4 \times 0.008 \times 100}{0.1}\right) \frac{u_2^2}{2g}$$

 $z_A - z_B = \left(1.5 + 4f \frac{L_2}{d_2}\right) \frac{u_2^2}{2g}$

Or $u_2 = 2.42 \text{ m/s}$

$$Q_2 = \frac{\pi}{4} (d_2)^2 \times u_2 = \frac{\pi}{4} (0.1)^2 \times 2.42 = 0.19 \frac{m^3}{s}$$

b) Replacing by equivalent single pipe

$$Q = Q_1 + Q_2 = 0.0034 + 0.019 \ 0.0224 \ \text{m}^3/\text{s}$$

$$z_A - z_B = \left(1.5 + 4f \frac{L}{d}\right) \frac{u^2}{2g}$$

$$10 = \left(1.5 + \frac{4 \times 0.008 \times 100}{d}\right) \frac{u^2}{2g}$$
Now
$$Q = A \ u = \frac{\pi}{4} \ d^2 \times u$$

Or

$$u = \frac{4 Q}{\pi d^2} = \frac{4 \times 0.0224}{\pi d^2} = \frac{0.02852}{\pi d^2}$$

Therefore,

$$10 = \left(1.5 + \frac{4 \times 0.008 \times 100}{d}\right) \frac{(0.02852)^2}{2gd^4}$$

 $0r \qquad 241212d^5 - 1.5 d - 3.2 = 0$

Approximate answer can be obtained by neglecting the second term.

Hence, $241212 d^5 = 3.2$ or d = 0.1058 m

Boundary Layers

When a fluid flows over a stationary surface, e.g. the bed of a river, or the wall of a pipe, the fluid touching the surface is brought to rest by the shear stress τ_0 at the wall. The velocity increases from the wall to a maximum in the main stream of the flow.

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

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



If we consider a flat plate in the middle of a fluid, we will look at the build up of the velocity profile as the fluid moves over the plate. Upstream the velocity profile is uniform, (free stream flow) a long way downstream we have the velocity profile we have talked about above. This is the known as **fully developed flow**. But how do we get to that state? This region, where there is a velocity profile in the flow due to the shear stress at the wall, we call the **boundary layer**. The stages of the formation of the boundary layer are shown in the figure below:



Notes on Fluid Mechanics by Dr. D.G. Roychowdhury

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

Boundary layer thickness, δ = distance from wall to point where u = 0.99 umainstream

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

Correspondingly, the drag force D on the fluid due to shear stress τ_0 at the wall increases from zero at the start of the plate to a maximum in the fully developed flow region where it remains constant. We can calculate the magnitude of the drag force by using the momentum equation.

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

Formation of the boundary layer

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

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

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

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

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

The viscous shear stresses have held the fluid particles in a constant motion within layers. They become small as the boundary layer increases in thickness and the velocity gradient gets smaller. Eventually they are no longer able to hold the flow in layers and the fluid starts to rotate. This causes the fluid motion to rapidly become turbulent. Fluid from the fast moving region moves to the slower zone transferring momentum and thus maintaining the fluid by the wall in motion. Conversely, slow moving fluid moves to the faster moving region slowing it down. The net effect is an increase in momentum in the boundary layer. We call the part of the boundary layer the **turbulent boundary layer**.

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



Surface roughness effect

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

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

In laminar flow the height of roughness has very little effect

Boundary layers in pipes

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

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



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

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

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

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

Laminar flow entry length $\approx 120 \times \text{diameter}$

Turbulent flow entry length $\approx 60 \times \text{diameter}$

Some basic formula used in Boundary layer

a) For laminar boundary layer:

The boundary layer thickness δ can be obtained from

$$\frac{\delta}{x} = \frac{5}{\sqrt{Re_x}} \qquad \text{where, } Re_x = \frac{\rho U_\infty x}{\mu}$$

Where x = distance from the leading edge.

The skin friction coefficient or drag coefficient can be obtained from

$$C_D = \frac{1.328}{\sqrt{Re_l}}$$
 where, $Re_l = \frac{\rho U_{\infty}L}{\mu}$

Where L = Length of the plate.

The drag on the plate of length L and width b can be obtained from

$$F_D = C_D \times \frac{1}{2} \rho A U_{\infty}^2 = \frac{1.328}{\sqrt{Re_l}} \times \frac{1}{2} \rho \times b \times L \times U_{\infty}^2$$

Note : for estimating the drag force from both sides of the plate, the value obtained from the above equation has to be multiplied by 2.

b) For turbulent boundary layer: If the Reynolds number is greater than equal to 5×10^5 , then it is turbulent boundary layer.

The boundary layer thickness δ can be obtained from

$$\frac{\delta}{x} = \frac{0.37}{(Re_x)^{0.2}} \qquad \text{where, } Re_x = \frac{\rho U_{\infty} x}{\mu}$$

The skin friction coefficient or drag coefficient can be obtained from

$$C_D = \frac{0.072}{(Re_l)^{0.2}} \qquad where, Re_l = \frac{\rho U_{\infty} L}{\mu}$$

Where L = Length of the plate.

The drag on the plate of length L and width b can be obtained from

$$F_D = C_D \times \frac{1}{2} \rho A U_{\infty}^2 = \frac{0.072}{(Re_l)^{0.2}} \times \frac{1}{2} \rho \times b \times L \times U_{\infty}^2$$

Boundary layer separation

Convergent flows: Negative pressure gradients

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

This is the case for *convergent* flows.



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

Divergent flows: Positive pressure gradients

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





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

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



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

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

A divergent duct or diffuser

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



Increasing the angle of the diffuser increases the probability of boundary layer separation. In a Venturi meter it has been found that an angle of about 6° provides the optimum

balance between length of meter and danger of boundary layer separation which would cause unacceptable pressure energy losses.

Tee-Junctions

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



Y-Junctions

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



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

Bends

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

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



Flow past a cylinder

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



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



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



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

Unit 3

Aerofoil

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



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



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



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

Calculation of Drag and Lift Force

The drag force is given by

$$F_D = C_D \times \frac{1}{2} \rho A U_{\infty}^2$$

The Lift force is given by

$$F_L = C_L \times \frac{1}{2} \rho A U_{\infty}^2$$

Where, $C_D = \text{Coefficient of drag, and}$ $C_L = \text{Coefficient of lift.}$

The resultant force is given by

$$F_R = \sqrt{F_D^2 + F_L^2}$$

The Power required

$$P = \frac{Force in the direction of motion \times velocity}{1000} kW = \frac{F_D \times U_{\infty}}{1000}$$