Engineering Tripos 1B

Paper 4

Fluid Mechanics

Lecture 1 - Introduction

- Course overview
- The discrete and continuum descriptions of fluids
- Macroscopic properties of a fluid - Viscosity
- Fields and partial derivatives - The del operator

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

Course overview

Difficulty

1. Introduction
2. Vector calculus in fluid mechanics
3. Inviscid flows
4. Viscosity and viscous flows
5. Dimensional analysis
6. Pipe flow
7. Network analysis
8. Laminar boundary layers
9. Turbulent boundary layers
10. External flows. Lift and drag

Fundamentals

Internal flows

External flows

Applications

Online resources

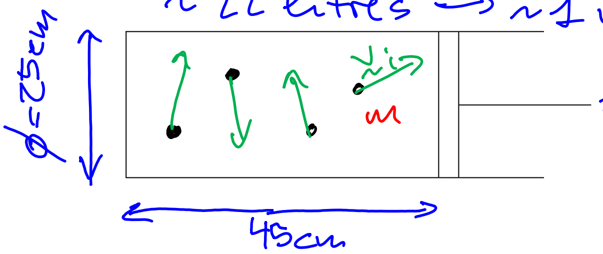
A collection of online resources for this course can be found in the moodle 1B-Thermofluis site. These include the completed handouts, videos and examples. The online version of the course prepared by Prof. Matthew Juniper is also an excellent resource and can be found at `http://learnfluidmechanics.org`.

1.1 The discrete and continuum descriptions of fluids

Watch videos on fluids at the molecular level on Moodle

Fluids are made up of a great number of molecules. These molecules attract and repel each other through diverse forces, and their motion is essentially governed by Newton's laws. One can picture a fluid as a collection of molecules moving and colliding against each other. In liquids, the molecules are in close contact with their neighbours, and they cannot move for too long before colliding. In gases, the molecules are normally well separated, which means that they move between one collision and the next with a *mean free path* that is much larger than the molecular diameter. Let us consider the gas contained in an average-sized piston. How many molecules would this system typically include?

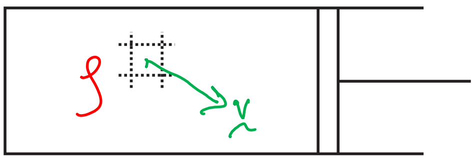
~22 litres → ~1 mole at STP → ~6×10^{23} molecules



= 600000000000000000000000

In any real-life situation it would be impossible to follow every molecule. Instead, we zoom out and look at the average properties of the fluid. Normally we do this *at a point in space*. For instance, we average[1] all the molecular velocities, $\mathbf{v}_i$, around a point in space (x, y, z) and say that the fluid there has a velocity $\mathbf{v}(x, y, z)$.

average molecular velocity ⇒ $\underset{\sim}{v}(x,y,z)$

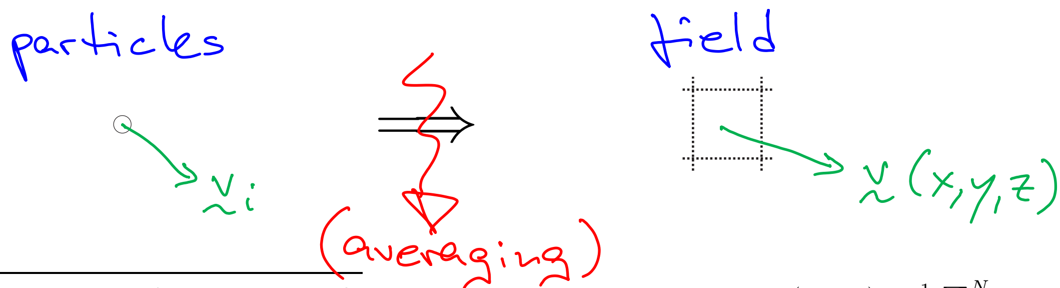


Also:

- n_V molecules per m^3
- m molecular mass

$n_V * m \equiv$ density $\rho(x,y,z)$

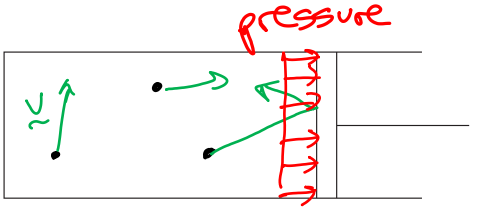
So now we can think of the fluid as a continuous lump of stuff—with no gaps!—and say that it has a certain velocity *field*.



[1]To be really careful we should define what we mean by 'average'. It is $\mathbf{v}(x, y, z) = \frac{1}{N}\sum_{i=1}^{N}\mathbf{v}_i$, where N is the number of molecules around that point in space and $\mathbf{v}_i$ is the velocity vector of each molecule.

1.2 Macroscopic properties of a fluid ρ, T, p

The properties of a fluid all arise from its molecular nature. The molecular mass, m, multiplied by the number of molecules in one metre cubed, n_V, gives the density, ρ. The temperature, T, is proportional to the average kinetic energy of the molecules, $mv_i^2/2$. For example, if we heat up a stationary gas, the *speeds* of all the molecules increase although, of course, their *velocity vectors* still average to zero because the gas remains stationary. To show how useful this concept is, let us look at the pressure on the piston face[2]:



1. momentum change on collision $\propto m v_i, \; i = 1, 2, \ldots$
2. number of collisions per unit area per unit time $\propto n_V v_i$

average

$$\text{pressure} = \frac{\text{force}}{\text{area}} = \frac{\text{average rate of change of momentum}}{\text{area}} = ① * ② =$$

$$= \text{constant} * m n_V v_i^2 = n_V * \text{constant} * m v_i^2 =$$

$$= \underbrace{m n_V}_{\text{density}} * \underbrace{\frac{2\,\text{constant}}{m}}_{\text{'gas constant'}} * \underbrace{\frac{1}{2} m v_i^2}_{\text{temperature}}$$

$$\underbrace{p = \rho * R * T}_{\text{eq. of state (equivalent to } pV = nR_gT)}$$

So by considering the molecular motion of the gas we can work out that $p = \rho RT$, where R is the gas constant. This is the *ideal gas equation of state*. In a similar way, macroscopic properties like viscosity, thermal conductivity, specific heat capacity etc. can all be worked out from the microscopic molecular motion.

The Knudsen number, Kn, is defined as the ratio of λ, the mean free path, to L, the size of the region we are considering. The continuum model only works when we average over very many molecules and very many collisions, so it requires the Knudsen number to be much less than 1. For instance, the continuum model breaks down for gases at extremely low pressures, or for flows in extremely small domains.

Try Q1 in Examples Paper 1

[2] For a more rigorous derivation of this, see Feynman's lectures on physics, section 39, classmark LA34 in the library.

Watch video on rarefied gas on Moodle

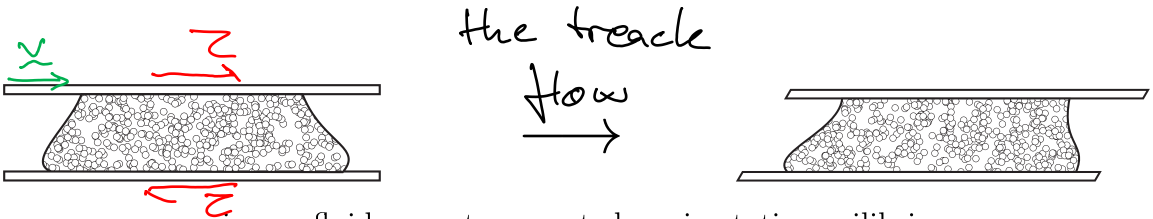
1.3 Resistance of a fluid to deformation - How fluids flow

One important macroscopic property of fluids is their viscosity, which characterises how they resist to deformation. The main difference between a solid and a fluid is that the relative position between molecules in a solid is fixed, while in a fluid molecules are relatively free to move around. Imagine holding a brick between the palms of your hands. When you move your right hand away from you and your left hand towards you, the forces from your hand are transmitted through the brick. This is because the molecules in the brick have defined positions. When displaced slightly the inter-molecular bonds resist in a spring-like manner. In static equilibrium, each layer of the brick experiences the same shear stress. τ

a solid brick can support shear in static equilibrium . . .

. . . because its molecules are held by rigid bonds

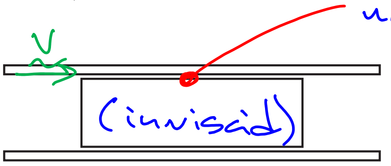
Now imagine that the brick is replaced with treacle. When you move your hands, the treacle flows into a new shape and then stops. In its new shape, at static equilibrium, there is no shear stress. This is because the molecules in the fluid do not have defined positions. When one layer is displaced with respect to another, the two flow over each other to accommodate the displacement. However, during the motion there *is* a shear stress, until statical equilibrium is reached. We will soon see that this shear is proportional to the rate of strain, dv_x/dy.



a viscous fluid cannot support shear in static equilibrium . . .

. . . because the molecules are not held by rigid bonds

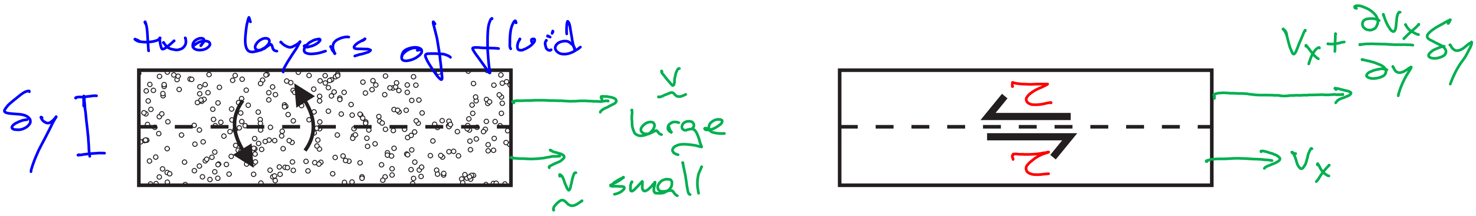
What would happen if we replaced the treacle with an inviscid fluid like those we considered in IA Fluids? An inviscid fluid has perfect slip. It cannot support any shear stress at all. It would be the perfect lubricant (if you could keep it in the right place).



no shear stress $\Rightarrow$ perfect slip (Euler eqs.)

1.4 Shear and viscosity

In most applications, there are too many molecules to consider each individually, so we consider the fluid as a continuum. The transfer of momentum by molecular motion then needs to be modelled in some way. Adjacent layers of fluid exchange momentum at a rate that is proportional to the velocity gradient. By Newton's second law ($\mathbf{f} = m\mathbf{a}$) the rate of change of momentum across a certain area is simply a force. When divided by the area, this is the shear stress τ:



In a certain time and over a certain area, some molecules swap places $\Rightarrow$ momentum exchange $\Rightarrow$ shear force

$$\tau = \text{shear stress} = \frac{\text{shear force}}{\text{area}} = \mu \frac{\partial v_x}{\partial y} \; ; \; \mu = \text{viscosity}$$

The coefficient of proportionality is the viscosity,[3] μ:

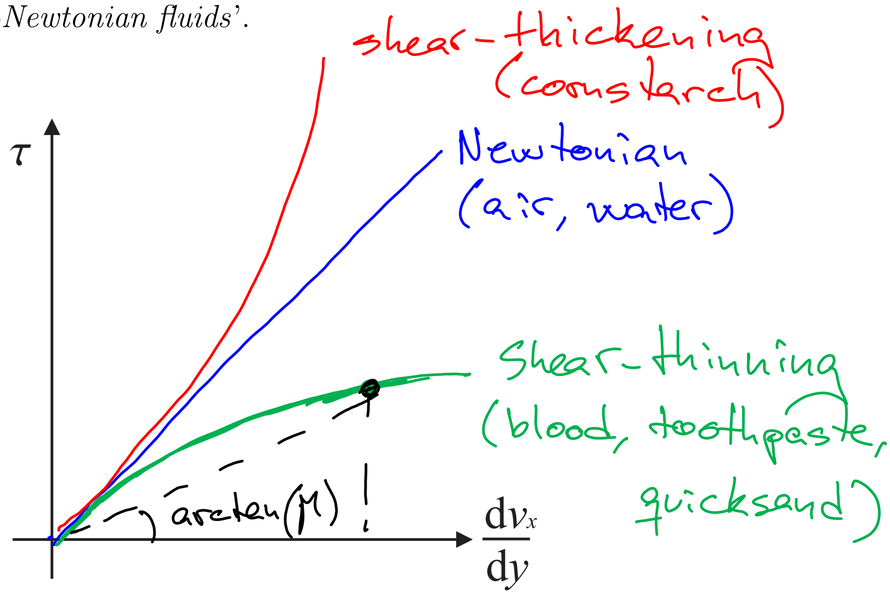
$$\tau = \mu \frac{dv_x}{dy} \qquad \mu = f(T)$$

This is described in "Random collisions, momentum transfer and viscosity" on the website.

Viscosity varies strongly with temperature because it is closely linked to molecular motion. In gases, viscosity increases with temperature because the average molecular speed increases and the momentum transfer per unit time therefore increases. However in liquids it decreases with increasing temperature, as you can see when you pour boiling water out of a kettle. This reflects the fact that the molecules in a liquid do not simply bounce off each other. Instead they form temporary bonds with each other which enhance the transfer of momentum. This bond energy becomes less significant compared with their kinetic energy as the temperature increases.

[3]In this particular case v_x is *only* a function of y so the partial derivative $\partial v_x/\partial y$ is equal to the ordinary derivative dv_x/dy. This is described in "Laminar viscous flow between flat plates" on the website.

For most fluids, the rate at which momentum diffuses is proportional to the velocity gradient, just as the rate at which heat diffuses is proportional to the temperature gradient. These fluids are called '*Newtonian fluids*'. If the molecules are long chains or the fluid contains small suspended solids, these can align or distort with the flow direction so the viscosity depends on the velocity gradient. These fluids are called '*non-Newtonian fluids*'.



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

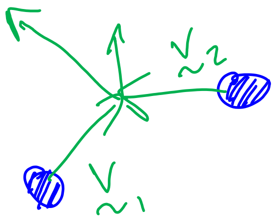
(Answer Q 5.1 on Examples Paper 1)

1.5 Flows beyond the scope of this course

We have just described how the viscosity of a fluid can vary depending on the flow conditions, temperature, etc. In this course we will concentrate on flows with constant viscosity. We will also assume that the density is constant, and that the continuum description holds. These are reasonable approximations for many hydrodynamic and aerodynamic flows at relatively low velocities, such as flows in pipes or around cars, submarines and low-speed aircraft, but we should note that they are not valid for every flow. High-speed aerodynamic flows where the compressibility of air is important, rarefied flows with very few molecules, or reacting and multiphase flows are beyond our scope, for instance.

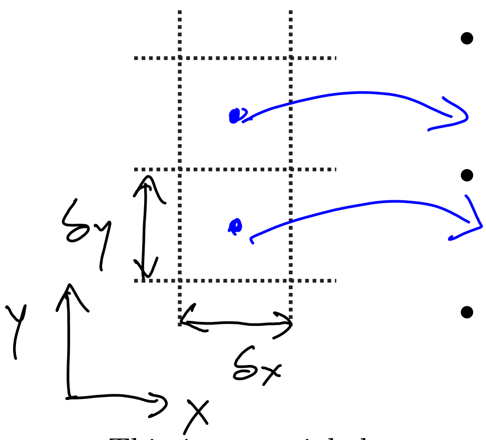
1.6 The continuum approach. Fields and partial derivatives

In the molecular description of a fluid, every molecule has a velocity vector, $\mathbf{v}_i$, and obeys Newton's laws of motion. The velocity is held *by the molecule* so we use *ordinary* derivatives such as $\mathrm{d}/\mathrm{d}t$. If we knew exactly how all the molecules started we could march forwards in time solving ordinary differential equations for each molecule. However, this is impractical for more than a few million molecules.



- $\mathbf{v}_i$ held by each molecule
- $\mathbf{F}_i = m \times \mathbf{a}_i = m * d\underset{\sim}{v}_i/dt$
- so we use ordinary derivatives, d/dt

Once we average over the very many molecules that occupy even a tiny region of space, we obtain a macroscopic, continuum picture of the flow. We can then think of *flow variables*, such as the velocity, defined at *points in space*, rather than for each discrete molecule. Instead of velocities assigned to each molecule, we now have a velocity *field* distributed in space. The description of the motion is no longer given by ordinary derivatives in time alone, as the velocity varies not only in time but also from one point to another. We therefore need to use *partial* derivatives, including the partial derivative with respect to time $\partial/\partial t$, but also the spatial derivatives $(\partial/\partial x, \partial/\partial y, \partial/\partial z)$. In this scenario, Newton's laws of motion still hold, but they adopt a slightly different form, as we will see in the coming lectures. In sum, by averaging over very many molecules and very many collisions, we exchange an enormous number of relatively simple *ordinary* differential equations for a few (but somewhat more complicated) *partial* differential equations, which describe the flow in terms of *fields*.



- $\mathbf{v}$ defined at points in space, $\underset{\sim}{v}(x,y)$
- the average velocity in the top box is

  $\underset{\sim}{v}(x,y) + \dfrac{\partial \underset{\sim}{v}}{\partial y}\delta y$
- the average velocity in the bottom box is

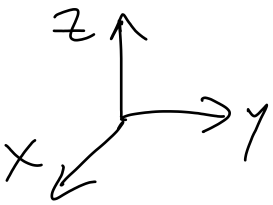
  $\underset{\sim}{v}(x,y)$
- so we use partial derivatives

  $\partial/\partial x, \partial/\partial y ; \partial/\partial t$

This is a crucial -but not at all straightforward- conceptual leap, and it took centuries for physicists and engineers to make it. In the words of Albert Einstein, "the application of Newton's mechanics to continuously distributed masses led inevitably to the discovery and application of partial differential equations, which in their turn first provided the language for the laws of the field-theory."

1.7 The del operator

As we have just discussed, when dealing with point masses, we use *ordinary* differential operators, such as $\mathrm{d}/\mathrm{d}t$, while when dealing with fields we need to use *partial* differential operators, such as $\partial/\partial t$, $\partial/\partial x$, $\partial/\partial y$ and $\partial/\partial z$. In space, partial differential operators are not very useful independently; they only give the change in one direction and, even worse, that direction depends on the choice of coordinate system.



$$\underset{\sim}{r}(x,y,z) \Rightarrow \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}$$

The real power of these partial differential operators arises when they are combined to form the *del* operator, which is given the symbol ∇ and is also called *nabla*. ∇ alone holds the full sense of *spatial derivative*, and it is invariant with respect to the coordinate system we choose. In Cartesian coordinates, the form of ∇ is very simple:

$$\nabla \equiv \hat{\mathbf{e}}_\mathbf{x}\frac{\partial}{\partial x} + \hat{\mathbf{e}}_\mathbf{y}\frac{\partial}{\partial y} + \hat{\mathbf{e}}_\mathbf{z}\frac{\partial}{\partial z} = \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix}$$

(shorthand in Cartesian coordinates)

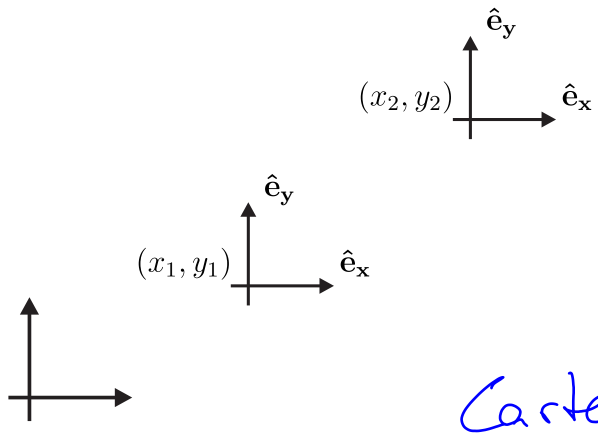
However, the fact that ∇ is independent of the coordinate system can make its expression in other coordinates very different. For instance, in cylindrical polars it is:

$$\nabla \equiv \hat{\mathbf{e}}_\mathbf{r}\frac{\partial}{\partial r} + \hat{\mathbf{e}}_\theta\frac{1}{r}\frac{\partial}{\partial \theta} + \hat{\mathbf{e}}_\mathbf{z}\frac{\partial}{\partial z}$$

!!

(no shorthand in curvilinear coordinates)

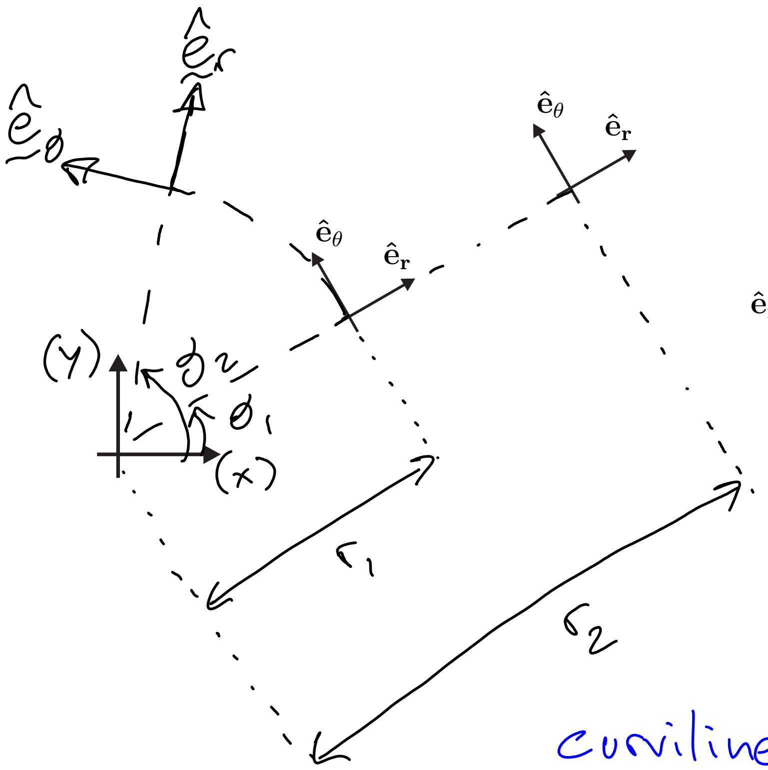
Furthermore, in a Cartesian coordinate system, the unit vectors are the same everywhere. This means that, when ∇ acts on a vector, we do not need to worry about the effect of ∇ on the unit vectors, because $\partial\hat{\mathbf{e}}_{\mathbf{x}}/\partial x$, $\partial\hat{\mathbf{e}}_{\mathbf{y}}/\partial x$ etc. are all zero. This is why the Cartesian shorthand works for ∇.



$\hat{\mathbf{e}}_{\mathbf{x}}$, $\hat{\mathbf{e}}_{\mathbf{y}}$ do not vary in space

Cartesian → (x,y)-invariant

In other coordinate systems, the unit vectors are not the same everywhere. This means that, when ∇ acts on a vector, its effect on the unit vectors must also be taken into account. We will encounter this in the worked example in Lecture 2 (section 2.5) and in Lecture 3 when differentiating along and across streamlines (section 3.5), and you can also find an example on the website.



$\hat{\mathbf{e}}_{\mathbf{r}}$, $\hat{\mathbf{e}}_\theta$
- do not vary with r
- vary with θ

curvilinear → ~~(r,θ)-invariant~~

Engineering Tripos 1B

Paper 4

Fluid Mechanics

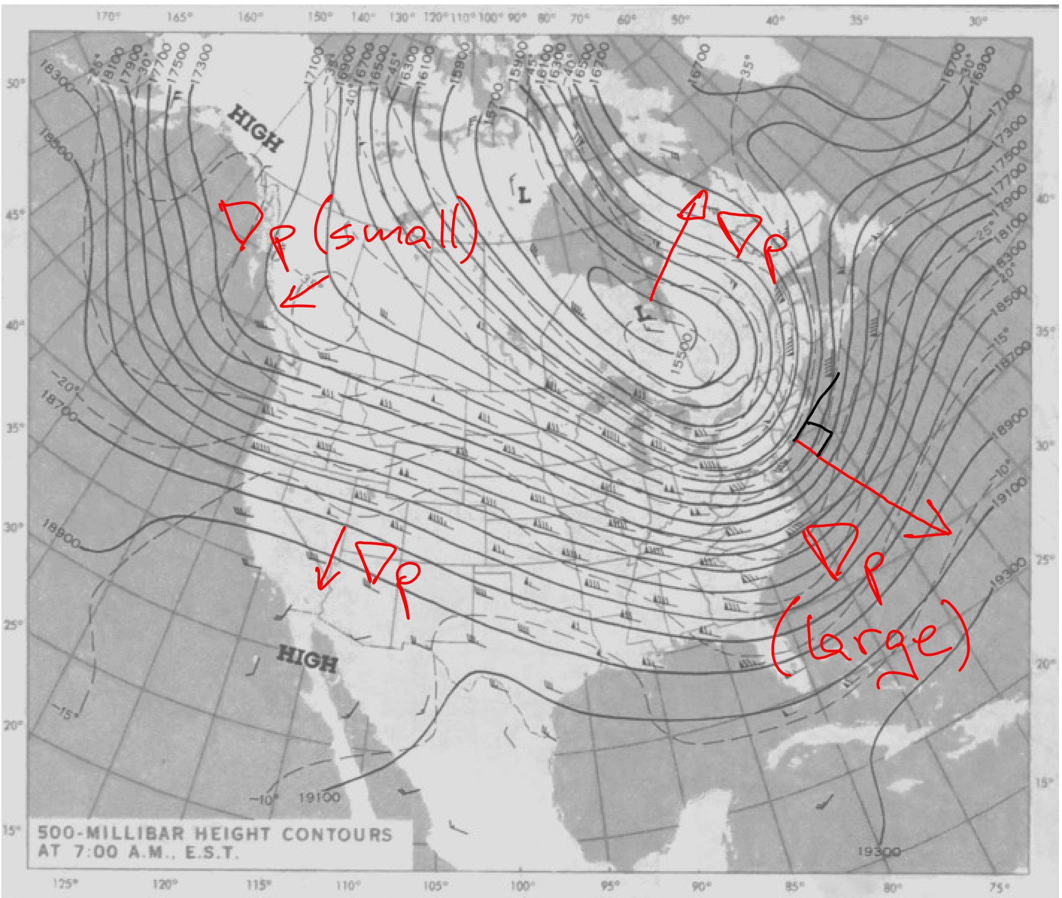
Lecture 2 - Vector calculus in fluid mechanics

- The gradient of a scalar field
- The divergence of a vector field - Conservation of mass - Incompressible flow
- The curl of a vector field - Vorticity
- Changes due to motion through a field - Advection

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

2.1 The gradient of a scalar field $\nabla(p)$

In Lecture 1, we have seen how ∇ encompasses all the spatial derivatives in a single vector-like, differential operator. When ∇ acts directly on a scalar field, it produces a vector that points in the direction of steepest increase of that scalar. Its magnitude equals the *gradient* in that direction. For example this weather map shows lines of constant pressure above the USA.



∇p is known as *"grad p"* because it gives the gradient of p at all points in the field.

$$\nabla p \;=\; \begin{bmatrix} \partial/\partial x \\ \partial/\partial y \\ \partial/\partial z \end{bmatrix} p \;=\; \begin{bmatrix} \partial p/\partial x \\ \partial p/\partial y \\ \partial p/\partial z \end{bmatrix} \Rightarrow \nabla p \text{ is a } \underline{\text{vector}}$$

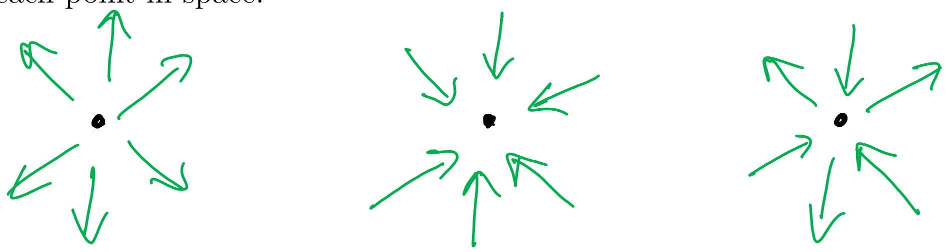
The vector ∇p is *orthogonal to the contour lines* and points in the direction in which the pressure increases. Its module $|\nabla p|$ is equal to the magnitude of the slope along ∇p.

2.2 The divergence of a vector field $\nabla \cdot (\underset{\sim}{v})$

The del operator ∇ can be applied on a vector field $\mathbf{a}$ through a dot product, $\nabla \cdot \mathbf{a}$. This is known as the *divergence* of the vector field $\mathbf{a}$. If we apply the divergence to the velocity field in Cartesian coordinates, the resulting *scalar field* is

$$\nabla \cdot \mathbf{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}. \Rightarrow \nabla \cdot \underset{\sim}{v} \text{ is a scalar}$$

The divergence of a vector field $\mathbf{a}$ produces a scalar field equal to the net flux of $\mathbf{a}$ out of each point in space.



$\nabla \cdot \underset{\sim}{v} > 0$ $\qquad$ $\nabla \cdot \underset{\sim}{v} < 0$ $\qquad$ $\nabla \cdot \underset{\sim}{v} = 0$

The divergence is a very important operator in fluid mechanics, as it appears in the equation that governs the conservation of mass for a flow field, the *continuity equation*. We will see this in more detail in the following sections.

A word of caution: for the expression for $\nabla \cdot \mathbf{v}$ in curvilinear coordinates things get a bit more complicated. For example, in cylindrical polars, and borrowing the expression for ∇ from section 1.7 of Lecture 1, we have

$$\nabla \cdot \mathbf{v} = \left(\hat{\mathbf{e}}_\mathbf{r} \frac{\partial}{\partial r} + \hat{\mathbf{e}}_\theta \frac{1}{r} \frac{\partial}{\partial \theta} + \hat{\mathbf{e}}_\mathbf{z} \frac{\partial}{\partial z} \right) \cdot (v_r \hat{\mathbf{e}}_\mathbf{r} + v_\theta \hat{\mathbf{e}}_\theta + v_z \hat{\mathbf{e}}_\mathbf{z}).$$

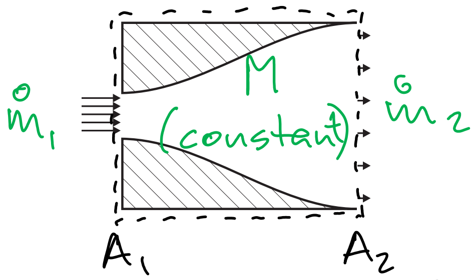
Try to work on the above expression, noting that the unit vectors now have some non-zero derivatives (but also that they are orthogonal to each other and certain products cancel out). You will see this in the Vector Calculus course. After some algebra, you should arrive to

$$\nabla \cdot \mathbf{v} = \frac{\partial v_r}{\partial r} + \frac{v_r}{r} + \frac{1}{r} \frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z}.$$

This is just as an example of how careful one needs to be when operating in curvilinear coordinates, especially because unit vectors have non-zero derivatives.

2.3 The law of conservation of mass

We have met the law of conservation of mass in 1A Fluids in its *integral form*. We could apply it to a control volume, for instance to the flow in a pipe:



$$\frac{dM}{dt} = \sum \mathring{m}_i = 0$$

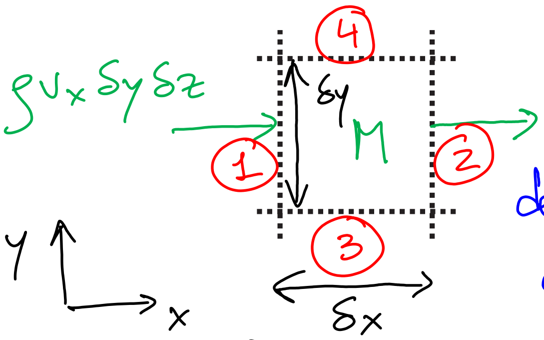
$$\Rightarrow \rho_1 V_1 A_1 = \rho_2 V_2 A_2$$

($A_i \equiv$ cross-sectional area)

We can also introduce the mass flux, defined as the flow of mass per unit area per unit time. For example, the mass flux at entry to the control volume above is:

$$\mathring{m}_1 / A_1 = \rho V_1 \qquad \text{(units } \mathrm{Kg/m^2 s}\text{)}$$

Let us now apply the same law to a small rectangular volume of space, with depth δz into the page. For simplicity we assume that $v_z = 0$.



$$\Big(\underbrace{\rho + \frac{\partial \rho}{\partial x}\delta x}_{\text{density change along 'x'}}\Big)\Big(\underbrace{V_x + \frac{\partial V_x}{\partial x}\delta x}_{\text{velocity change along 'x'}}\Big)\delta y \delta z$$

In a time δt, the change in mass, δM, is given by

$$\delta M = \left\{ \underbrace{\rho v_x \delta y \delta z}_{\text{in } 1} - \underbrace{\left(\rho + \frac{\partial \rho}{\partial x}\delta x\right)\left(v_x + \frac{\partial v_x}{\partial x}\delta x\right)\delta y \delta z}_{\text{out } 2} + \underbrace{\rho v_y \delta x \delta z}_{\text{in } 3} - \underbrace{\left(\rho + \frac{\partial \rho}{\partial y}\delta y\right)\left(v_y + \frac{\partial v_y}{\partial y}\delta y\right)\delta x \delta z}_{\text{out } 4} \right\} \delta t$$

$$\Rightarrow \frac{\delta M}{\delta t} = -\left(\frac{\partial(\rho v_x)}{\partial x} + \frac{\partial(\rho v_y)}{\partial y}\right)\delta x \delta y \delta z$$

But $M = \rho \delta x \delta y \delta z$ so the $\delta x \delta y \delta z$ cancels and, as δt tends to zero, we obtain:

$$\boxed{\frac{\partial \rho}{\partial t} = -\left(\frac{\partial(\rho v_x)}{\partial x} + \frac{\partial(\rho v_y)}{\partial y}\right) = -\begin{bmatrix} \partial/\partial x \\ \partial/\partial y \end{bmatrix} \cdot \begin{bmatrix} \rho v_x \\ \rho v_y \end{bmatrix} = -\nabla \cdot (\rho \underset{\sim}{v})}$$

"del dot mass flux"

This is the law of conservation of mass in *differential form*: the rate of change of mass per unit volume (density) is the net rate at which mass flows out of the volume. This method of deriving the formula is easy to understand physically but requires some messy maths. A more rigorous derivation uses Gauss' theorem, but is harder to visualize. You will see it in the Vector Calculus course.

2.4 Incompressible flow vs. the equation of state of gases

The equation of state of a fluid gives the relationship between the density, ρ, the pressure, p, and the temperature, T. There is no universal equation of state that accurately models the properties of all fluids under all conditions. Instead, various different models are used. The most familiar model is that for an ideal gas: $\rho = p/(RT)$. A more sophisticated model is the Van der Waals equation of state, which accounts for the finite volume occupied by the molecules themselves. Depending on the problem, one model may be more suitable than another.

For liquids, the model $\rho =$ constant is generally a very good one. At low velocities and moderate temperature and pressure gradients, even gases can be approximated as having uniform density quite accurately. In the rest of the Fluid Mechanics course, we will assume $\rho =$ constant. Under this condition, the flow is called *incompressible*, and calculations become easier. For example, the continuity equation:

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot (\rho \mathbf{v})$$

$\cancel{p = \rho R T \Rightarrow \rho = f(p,T)}$

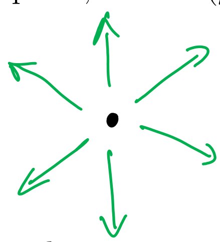
becomes simply

or

$$\boxed{\nabla \cdot \mathbf{v} = 0.} \Longleftarrow \rho = \text{constant}$$

2.5 Worked example - Uniform source flow. Source in a free stream flow

If we were to heat up a fluid at a point in space, perhaps with a laser, its density would drop at that point ($\partial\rho/\partial t$ negative), and the fluid would *diverge* away from that point, i.e. $\nabla \cdot (\rho \mathbf{v})$ would be positive. The point would become a flow *source*.



- mass diverges from source, $\nabla \cdot \underset{\sim}{v} > 0$ (elsewhere, $\nabla \cdot \underset{\sim}{v} = 0$)
- radial streamlines from source

In two dimensions, a uniform source flow could be obtained by injecting flow steadily at a point.

- *If the flow is (elsewhere than at the source) incompressible, use the continuity equation to show that the velocity is inversely proportional the distance r to the source, $v_r = C/r$*

The first thing to note is that, because of the axisymmetry of the flow, the velocity must be aligned with $\mathbf{r}$ and its magnitude depend only on r,

$$\mathbf{v} = v_r(r)\ \hat{\mathbf{e}}_{\mathbf{r}}.$$

It would then be sensible to use polar coordinates, so that the divergence, which must be zero, is given by the expression we found in section 2.2, which simplifies to

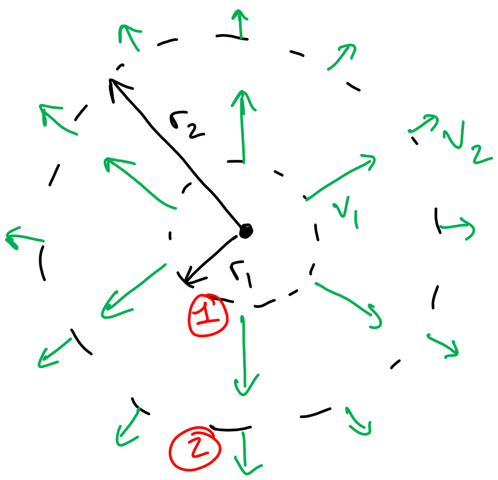
$$\nabla \cdot \mathbf{v} = \frac{\partial v_r}{\partial r} + \frac{v_r}{r} + \cancel{\frac{1}{r}\frac{\partial v_\theta}{\partial \theta}} + \cancel{\frac{\partial v_z}{\partial z}} = \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$$

Since the problem only depends on r, the partial derivatives $\partial/\partial r$ are actually absolute derivatives $\mathrm{d}/\mathrm{d}r$. The differential equation and its solution are

$$\frac{\mathrm{d}v_r}{\mathrm{d}r} = -\frac{v_r}{r} \qquad \Longrightarrow \qquad v_r(r) = \frac{C}{r}.$$

- *Choosing a suitable control volume, find the value of the constant of integration C for a volumetric flow rate Q at the source*

Once again, we can exploit the axisymmetry of the problem by choosing a volume with circular boundaries concentric with the source. The mass entering the volume through the inner boundary per unit time must be ρQ, and so must also be the mass leaving through the outer boundary.



-Mass flow rate through 1:

$$\int_0^{2\pi r_1} \rho\, v_1\, \mathrm{d}s_1 = 2\pi \rho v_1 r_1 = \rho Q$$

-Mass flow rate through 2:

$$\int_0^{2\pi r_2} \rho\, v_2\, \mathrm{d}s_2 = 2\pi \rho v_2 r_2 = \rho Q$$

Therefore,

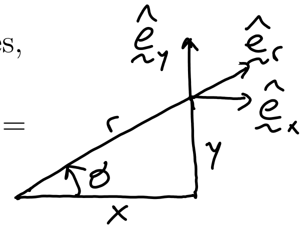
$$v_r(r) = \frac{C}{r} = \frac{Q}{2\pi r}.$$

Note that we could have used this control volume to show that v_r must be inversely proportional to r, but only because of the very easy symmetry of this particular case. The differential approach is in general more versatile.

- *Using Cartesian coordinates, check that the divergence is zero also in this coordinate system*

The aim of this question is to illustrate how ∇ is independent of the coordinate system, and that even if the expressions for the gradient or the divergence are more complicated in curvilinear coordinates, they may make the algebra of some problems simpler, provided certain symmetries exist.

First, we would need to express the velocity in Cartesian coordinates,



$$\mathbf{v} = \frac{Q}{2\pi r}\hat{\mathbf{e}}_{\mathbf{r}} = \frac{Q}{2\pi r}\cos\theta\,\hat{\mathbf{e}}_{\mathbf{x}} + \frac{Q}{2\pi r}\sin\theta\,\hat{\mathbf{e}}_{\mathbf{y}} = \frac{Q}{2\pi r}\frac{x}{r}\hat{\mathbf{e}}_{\mathbf{x}} + \frac{Q}{2\pi r}\frac{y}{r}\hat{\mathbf{e}}_{\mathbf{y}} =$$
$$= \frac{Q}{2\pi}\frac{x}{x^2+y^2}\hat{\mathbf{e}}_{\mathbf{x}} + \frac{Q}{2\pi}\frac{y}{x^2+y^2}\hat{\mathbf{e}}_{\mathbf{y}}.$$

From here, the differentiation is straightforward, albeit tedious. (Now think how you would obtain in Cartesian coordinates an expression for the divergence for a generic v_r, if you did not know how it depends on r, and if you could deduce from it that v_r must be inversely proportional to r).

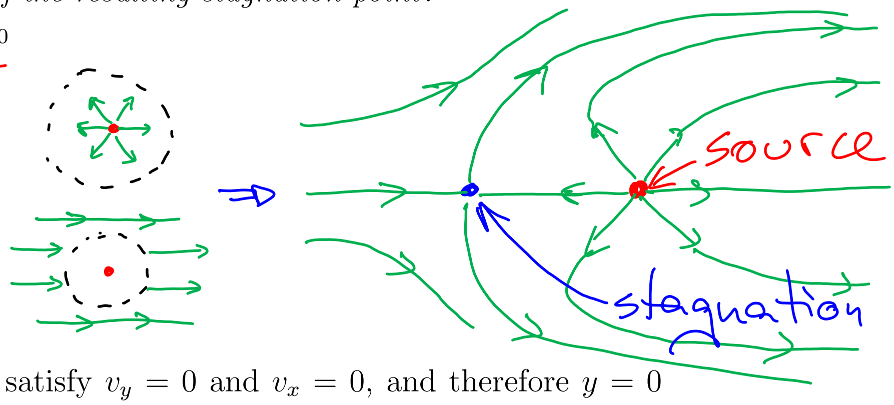
- *If the above source flow is superimposed with a free stream flow* $U\hat{\mathbf{e}}_{\mathbf{x}}$ ($\propto r^0$)*, the resulting flow is still incompressible (check this). Sketch the resulting streamlines. What would be the position of the resulting stagnation point?*

source flow $\sim r^{-1}$; free stream $\sim r^0$

$$\underset{\sim}{v} = \frac{Q}{2\pi r}\underset{\sim}{\hat{e}_r} + U\underset{\sim}{\hat{e}_x}$$

$\Rightarrow$ near the source: $r^{-1} \gg r^0$

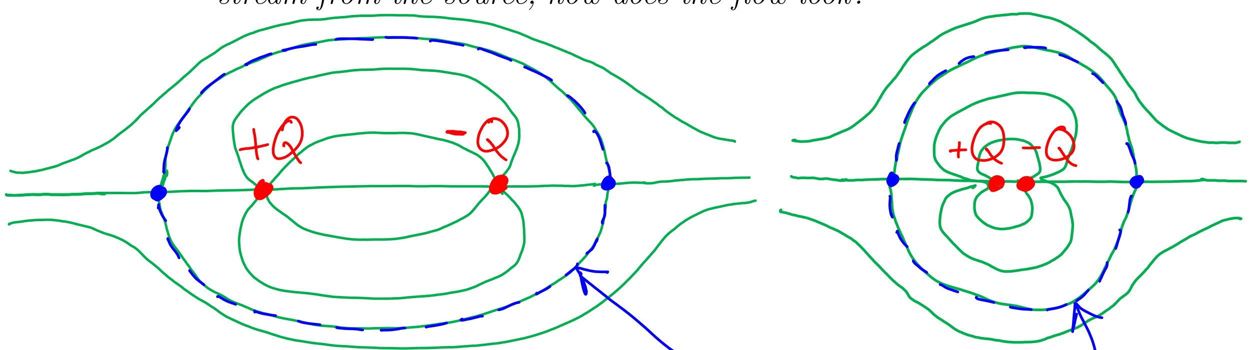
$\Rightarrow$ far from source: $r^{-1} \ll r^0$



The stagnation point needs to satisfy $v_y = 0$ and $v_x = 0$, and therefore $y = 0$ and

$$U + \frac{Q}{2\pi}\frac{x}{x^2+y^2} = 0, \qquad \text{or} \qquad x = \frac{Q}{2\pi U}$$

- *If we now superimpose to the above -uniform free stream plus source- a sink flow of flow rate $-Q$ with the sink located along the x axis a distance d downstream from the source, how does the flow look?*



large d small d

Note that the streamlines connecting the stagnation points do not allow flow through. This property is sometimes exploited to represent solid obstacles as combinations of sinks and sources. When the distance d between the source and the sink becomes vanishingly small, the aggregate is called a *doublet*.

Watch the videos on section 2.7 in Moodle

2.6 The curl of a vector field. Vorticity $\nabla\times(\underset{\sim}{v})$

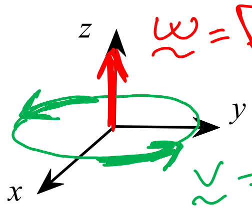
Just like ∇ can be dotted with a vector $\mathbf{a}$ to obtain the divergence, it can also be crossed to obtain the *curl* $\nabla\times\mathbf{a}$. The curl is also very important in fluid mechanics, because the curl of the velocity is the *vorticity* $\boldsymbol{\omega}$,

$$\boldsymbol{\omega} = \nabla\times\mathbf{v}$$

The vorticity turns out to be *twice the angular velocity* of the fluid. It has a key role in flow dynamics and is intimately related to dissipation processes, but in this course we will only be concerned with one aspect of it. In IA Fluids you learned how to apply Bernoulli's equation along streamlines. This is a conservative approach, but it isn't quite the whole story. In Lecture 3 we will see that Bernoulli's equation can be applied across streamlines, but only when there is zero vorticity between them. We will only be concerned with vorticity insofar as to whether it is zero, and therefore Bernoulli can be applied across streamlines. For illustrative purposes, let us look at the expression of vorticity for a two-dimensional flow.

$$\boldsymbol{\omega} = \nabla\times\mathbf{v} = \begin{bmatrix}\partial/\partial x\\ \partial/\partial y\\ \cancel{\partial/\partial z}\end{bmatrix}\times\begin{bmatrix}v_x\\ v_y\\ 0\end{bmatrix} = \begin{bmatrix}0\\ 0\\ \dfrac{\partial v_y}{\partial x}-\dfrac{\partial v_x}{\partial y}\end{bmatrix}$$

(2D flow)



$\underset{\sim}{\omega} = \nabla\times\underset{\sim}{v} = \begin{bmatrix}0\\0\\\omega_z\end{bmatrix}$

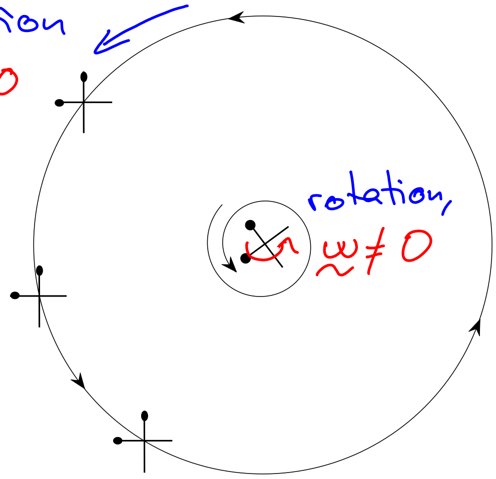
$\underset{\sim}{v} = (v_x, v_y, 0)$

Note that, just like the angular velocity, the vorticity is oriented along the local axis of rotation of the flow.

It is also important to note that fluids can move along curved paths without actually rotating. For instance, in a plughole vortex, only the fluid near the centre has non-zero vorticity. The fluid particles away from the centre move along circular paths but keep facing in the same direction. You can test this by putting a matchstick cross on the water surface.

$\underset{\sim}{v}\neq 0$, but no rotation

$\underset{\sim}{\omega}=0$



rotation, $\underset{\sim}{\omega}\neq 0$

Over the plughole:

- the fluid spins quickly (high shear);
- viscous forces are strong;
- the flow has non-zero vorticity.

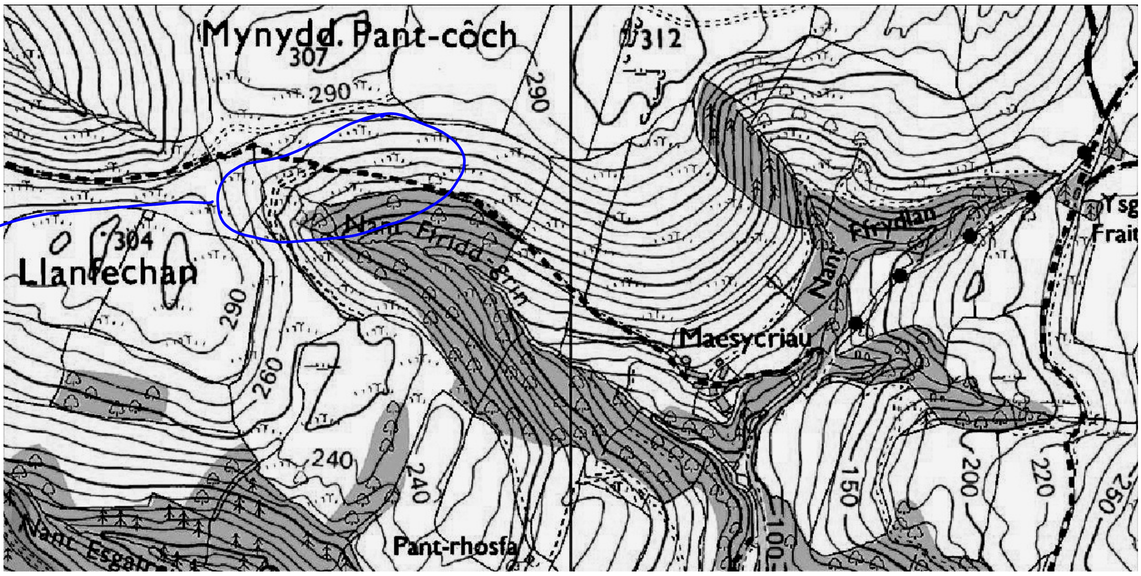
Away from the plughole the cross keeps pointing in the same direction

- low shear;
- viscous forces are weak;
- the flow has zero vorticity. $\Longrightarrow$ The flow is *irrotational*

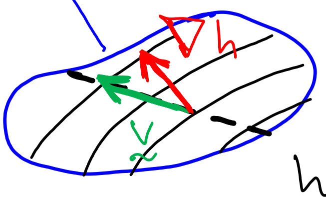
Watch the video on vorticity and irrotational flow in Moodle

2.7 Changes due to motion through a field. Advection

The contour lines on this map of Snowdonia show average height above sea level.



At every point (x, y), this two-dimensional scalar *height field* has a single value of $h(x, y)$. As you walk along a path at a certain velocity $\mathbf{v}$ at what rate does your height change?



$$\frac{dh}{dt} = \frac{dh}{dx}\frac{dx}{dt} = (\nabla h \cdot \cos\theta)(v) = [\underset{\sim}{v} \cdot \nabla] h$$

held by traveler

held by the field

The operator $\underbrace{(\underset{\sim}{v} \cdot \nabla)}_{\text{advection}}$ gives the variation (of h) following $\mathbf{v}$.

The rate of change of your height is given by $\mathbf{v} \cdot \nabla h$. We can write this as $\mathbf{v} \cdot (\nabla h)$ or as $(\mathbf{v} \cdot \nabla)h$. These are equivalent but the second version becomes more convenient later because $(\mathbf{v} \cdot \nabla)$ is a *scalar operator* that acts on anything to its right *preserving its dimension.* In this case it acts on h, but it may act on any field variable.

!!!

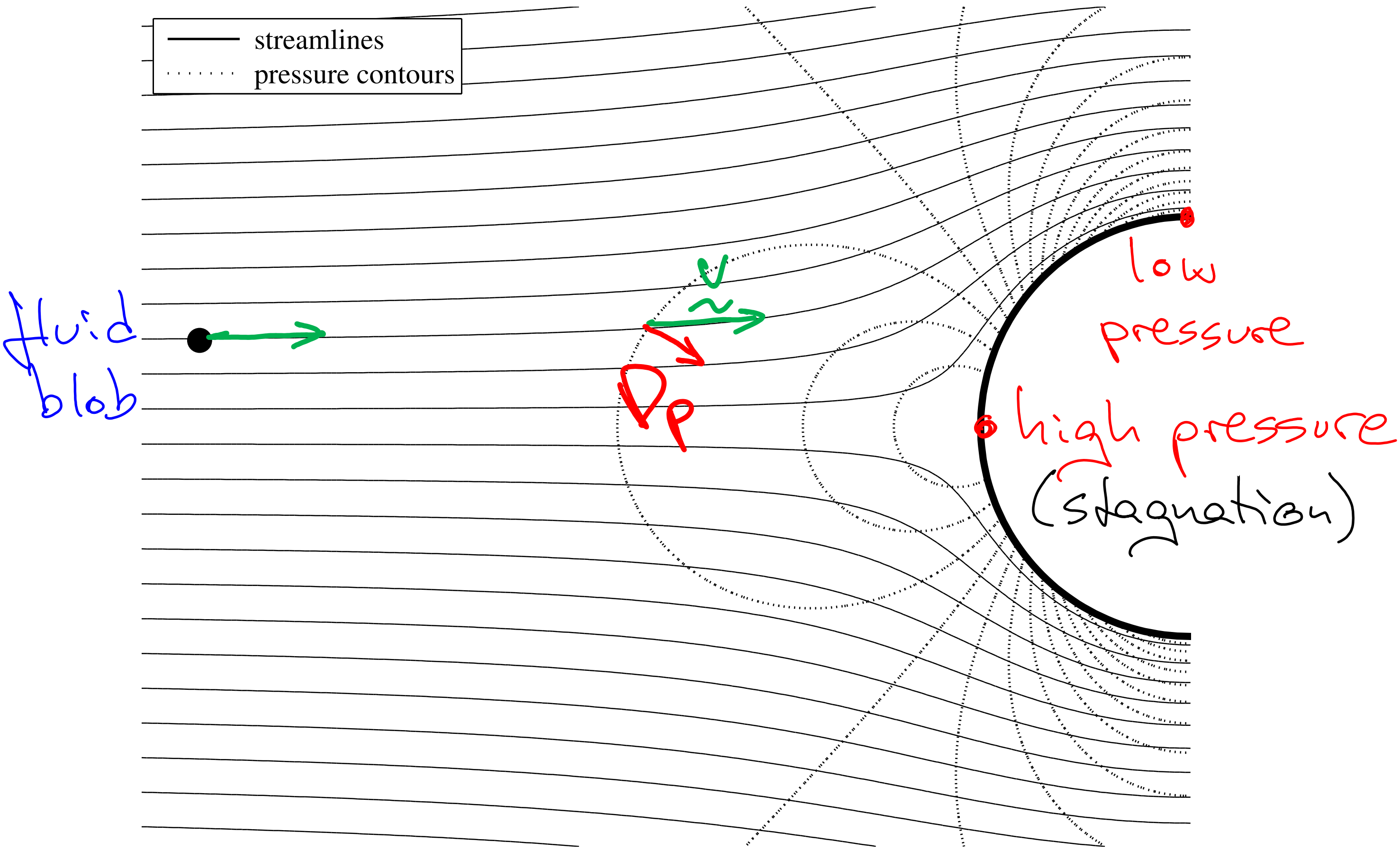
$$(\mathbf{v} \cdot \nabla) = \left(\begin{bmatrix} v_x \\ v_y \\ v_z \end{bmatrix} \cdot \begin{bmatrix} \partial/\partial x \\ \partial/\partial y \\ \partial/\partial z \end{bmatrix} \right) = \left(v_x \frac{\partial}{\partial x} + v_y \frac{\partial}{\partial y} + v_z \frac{\partial}{\partial z} \right) \Rightarrow \text{operator}$$

Note that $(\mathbf{v} \cdot \nabla)$ is *not* the same as $(\nabla \cdot \mathbf{v})$,

$$(\nabla \cdot \mathbf{v}) = \left(\begin{bmatrix} \partial/\partial x \\ \partial/\partial y \\ \partial/\partial z \end{bmatrix} \cdot \begin{bmatrix} v_x \\ v_y \\ v_z \end{bmatrix} \right) = \left(\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \right) \Rightarrow \text{divergence of vector (result is scalar)}$$

Answer Q 52 in Examples Paper 1

The streamlines and the pressure field in a steady flow around a cylinder are shown below. Let us follow a 'blob' of fluid, or *fluid particle*, through this pressure field. The fluid particle is big enough that it contains many billions of molecules, so that we can average the molecular motion and speak meaningfully about the fluid particle's velocity and pressure. But it is small enough that we can consider it as being at a single point in the field. As part of the fluid, the particle must move along a streamline. At what rate does its pressure change?



$$\frac{dp}{dt} = (\underset{\sim}{v}) \cdot (\nabla p) = (\underset{\sim}{v} \cdot \nabla) p$$

held by fluid blob

held by the field

The rate of change of the fluid particle's pressure is given by $(\mathbf{v} \cdot \nabla)p$.

Engineering Tripos 1B

Paper 4

Fluid Mechanics

Lecture 3 - Inviscid flow

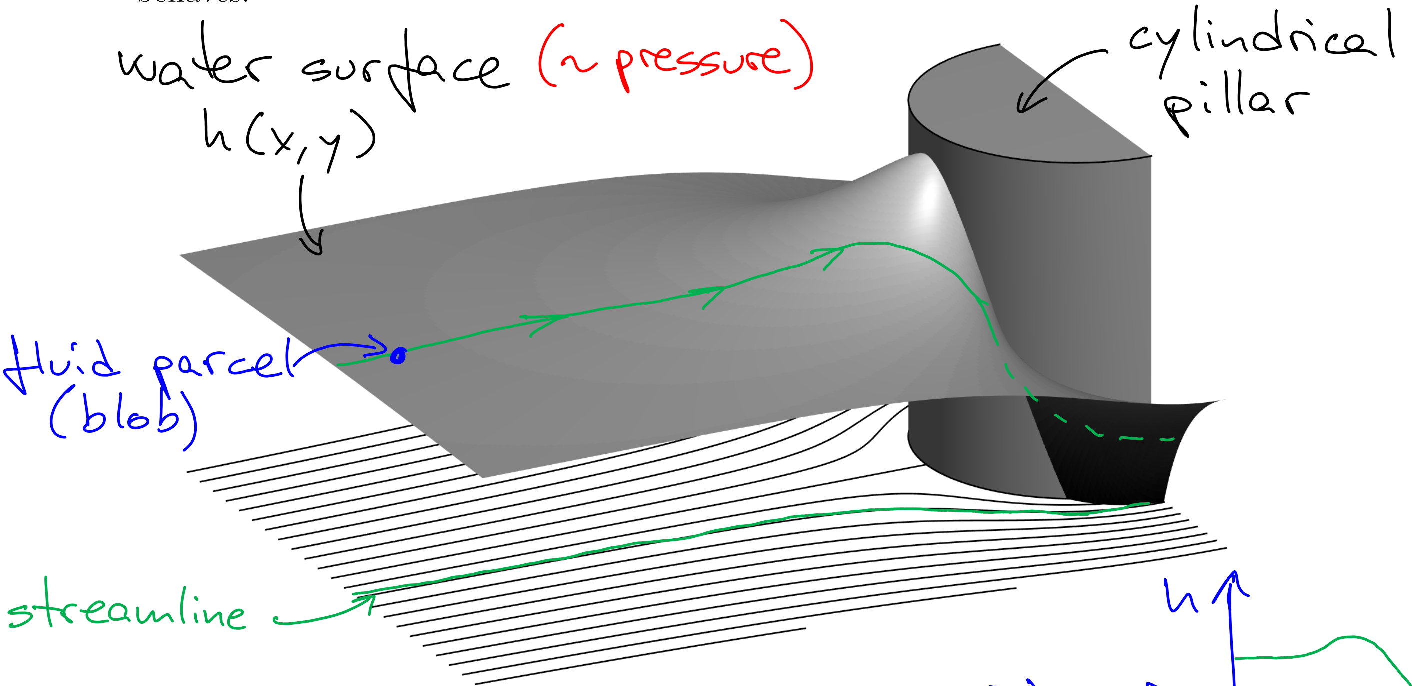
- The material derivative
- Newton's second law ($\mathbf{f} = m\mathbf{a}$) applied to a fluid: Euler's equations
- Euler's equations integrated along a streamline: Bernoulli's equation
- Bernoulli's equation and streamline curvature
- Determining the pressure field from a flow's streamlines

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

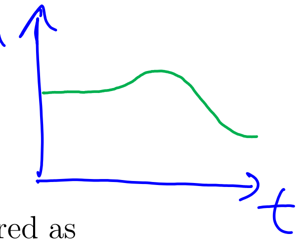
Watch the video on the water flow past a cylinder in Moodle

3.1 The material derivative

Let us recall the steady flow around a cylinder from the end of last lecture. When water flows steadily past a cylindrical obstacle, its height follows the same pattern as that of the pressure deeper within, which helps us visualize how the pressure behaves.



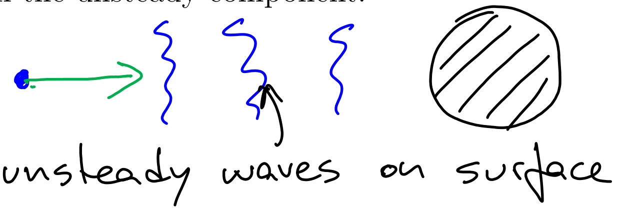
change in height of the blob as it moves $\equiv (\underset{\sim}{v} \cdot \nabla) h \Rightarrow$



Let us focus on a small 'blob' of fluid, small enough that it can be considered as a point travelling through the fluid field on the macroscopic level–yet containing a very large number of molecules on the microscopic level. This is known as a *fluid parcel* or *fluid particle*. We know from section 2.7 that the change in pressure experienced by a fluid particle as it moves with the flow is

$$\frac{\mathrm{d}p}{\mathrm{d}t} = (\mathbf{v} \cdot \nabla)p$$

When the flow is *unsteady*, the pressure field changes with time and we need to add on the unsteady component:



$\Rightarrow$ pressure field changes with time, $\partial p/\partial t \neq 0$

$\Rightarrow$ add it on

$$\frac{\mathrm{d}p}{\mathrm{d}t} = \left.\frac{\partial p}{\partial t}\right|_{(x,y,z)=\text{constant}} + (\mathbf{v} \cdot \nabla)p$$

as held by the fluid parcel

at a point in space, as held by the field

This is one of the most powerful concepts in fluid mechanics. Gabriel Stokes found himself using this derivative 'following the blob' so much that he gave it a name, the *material derivative*, and its own notation:

$$\frac{D}{Dt} = \left(\frac{\partial}{\partial t} + \mathbf{v}\cdot\nabla\right)$$

It is coordinate-free (i.e. the definition does not depend on the coordinate system being used). Being an operator, it is hungry for something to differentiate. For instance, we can use it to find the acceleration of a fluid particle by differentiating its velocity:

when looking at blobs

when looking at points in space

scalar operator! (advection)

$$\underbrace{\frac{\mathrm{d}\mathbf{v}}{\mathrm{d}t}}_{\text{acceleration of fluid particle}} = \frac{D\mathbf{v}}{Dt} = \underbrace{\frac{\partial \mathbf{v}}{\partial t}}_{\text{rate of change of } \mathbf{v} \text{ at a fixed point in space } (x,y,z)} + \underbrace{(\mathbf{v}\cdot\nabla)\mathbf{v}}_{\text{change in } \mathbf{v} \text{ due to the motion through the field (with velocity } \mathbf{v}!)}$$

We refer to the description of fluids following its particles as a *Lagrangian description*, and to its description looking at fixed points in space as the *Eulerian description*.

Lagrangian description	Eulerian description
Following fluid particles	At fixed points in space
$\frac{\mathrm{d}}{\mathrm{d}t}$	$\frac{D}{Dt}\left(=\frac{\partial}{\partial t} + \mathbf{v}\cdot\nabla\right)$

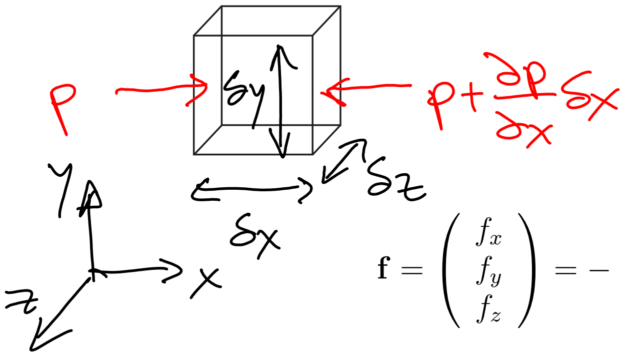
- Watch the video on the Lagrangian and Eulerian descriptions in Moodle
- Answer Q2 in Examples Paper 1

3.2 Newton's second law: $\mathbf{f} = m\mathbf{a}$

If we put a neutrally buoyant solid cube into a fluid flow and ignore all the viscous forces, we can work out the net force on the cube by considering the pressure on each face:

$$f_x = p\,\delta y\,\delta z - \left(p + \frac{\partial p}{\partial x}\delta x\right)\delta y\,\delta z$$

$$= \frac{\partial p}{\partial x}\,\delta y\,\delta z$$



$$\mathbf{f} = \begin{pmatrix} f_x \\ f_y \\ f_z \end{pmatrix} = -\begin{pmatrix} \partial p/\partial x \\ \partial p/\partial y \\ \partial p/\partial z \end{pmatrix} \delta x\delta y\delta z = -\nabla p \quad \delta x\delta y\delta z$$

Now we write $\mathbf{f} = m\mathbf{a}$ for the cube:

$$\overbrace{-\nabla p\,\delta x\,\delta y\,\delta z}^{f} = \overbrace{\rho\,\delta x\,\delta y\,\delta z}^{m}\,\overbrace{\frac{d\underset{\sim}{v}}{dt}}^{a}$$

$$-\nabla p = \rho\,\frac{dv}{dt}$$

We could do this for an imaginary cube, that is, for a fluid particle. Similarly, we would find that the particle is being accelerated or decelerated by the pressure gradients in the flow. However, we know how to express the acceleration of a fluid particle in terms of the fluid's velocity field because we have worked it out in Section 3.1. We can combine all this to get:

$$-\nabla p = \rho\,\frac{D\underset{\sim}{v}}{Dt}$$ held by the field

$$\Rightarrow \underbrace{\rho}_{\text{mass per unit volume}}\underbrace{\left(\frac{\partial \underline{v}}{\partial t} + \underline{v}\cdot\nabla\underline{v}\right)}_{\text{acceleration}} = \underbrace{-\nabla p}_{\text{force per unit volume}}$$

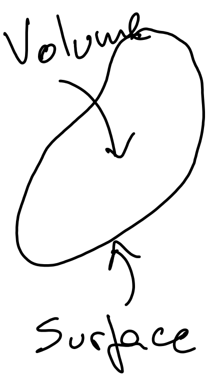
EULER EQS,

$\rho, \underset{\sim}{v}, p$ are <u>all</u> field variables

These are the *Euler equations*, which become the *Navier-Stokes equations* when viscous terms are included (see p9 of the databook, where it is shown in its steady form and called by its other name: the *momentum equation*). Euler's equations are simply $\mathbf{f} = m\mathbf{a}$ applied to an inviscid fluid. Remember that there is one equation for each dimension. For example in 3D cartesian coordinates there is one equation in the x-direction, one in the y-direction and one in the z-direction.

Link to 1st year Momentum Equation

In the 1st year you derived the Steady Flow Momentum Equation (SFME) and the more general Momentum Equation (p7 of databook) by considering a control volume:



$$\underbrace{\frac{\mathrm{d}}{\mathrm{d}t}\int_{Vol}\rho\mathbf{v}\ dV}_{\text{rate of change of momentum in the volume}} + \underbrace{\int_{Surf}\rho\mathbf{v}(\mathbf{v}\cdot\mathbf{dA})}_{\text{net rate at which momentum enters the volume}} = \underbrace{\mathbf{F}}_{\text{body forces (e.g. gravity)}} - \underbrace{\int_{Surf} p\ \mathbf{dA}}_{\text{pressure forces on the surface}} \qquad (1)$$

Euler's equations are the same thing, but in vector notation and differential form:

$$\frac{\partial\rho\mathbf{v}}{\partial t} + \rho(\mathbf{v}\cdot\nabla)\mathbf{v} = \text{body forces} - \nabla p \qquad (2)$$

valid at each and every point in the fluid

While the Steady Flow Momentum Equation gives information on the fluid variables only in an integral sense, overall in the control volume, the Euler equations give information at each point in space. Once you know a bit more vector calculus, you will be able to derive one from the other by spatial integration/differentiation:

$$\int_{Vol} (2)\, dV \Leftrightarrow (1)$$

(Divergence theorem)

When gravity, $-\rho g\hat{\mathbf{e}}_{\mathbf{z}}$, acts on the fluid (and, in fact, for any other conservative force), its expression as a potential $-\nabla\rho g z$ allows us to embed it in the pressure gradient term. This will be very useful in Lecture 7.

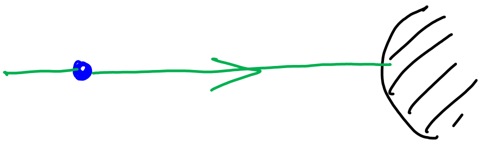
$$\begin{aligned}\frac{\partial\rho\mathbf{v}}{\partial t} + \rho(\mathbf{v}\cdot\nabla)\mathbf{v} &= -\rho g\hat{\mathbf{e}}_{\mathbf{z}} - \nabla p \\ &= -\nabla\rho g z - \nabla p \\ &= -\nabla\left(p + \rho g z\right)\end{aligned}$$

gravity force per unit volume:

$$-\rho g e_z = -\frac{\partial(\rho g z)}{\partial z}\hat{e}_z$$

3.3 Euler's equations applied along a straight streamline

We will look at the central streamline of the steady flow around a cylinder. The fluid on this streamline flows at velocity V in the x-direction. This streamline is easy to examine because the streamline coordinate system is aligned with the cartesian coordinate system. The more general case is considered in the next section. The cartesian unit vectors are $(\mathbf{e_x}, \mathbf{e_y})$ and ∇ is defined as $\nabla \equiv \mathbf{e_x}\partial/\partial x + \mathbf{e_y}\partial/\partial y$.



$\underset{\sim}{v} = V\underset{\sim}{e}_x$; $(\underset{\sim}{v}\cdot\nabla) = \begin{bmatrix} V \\ 0 \end{bmatrix} \cdot \begin{bmatrix} \partial/\partial x \\ \partial/\partial y \end{bmatrix}$

Euler's equations (for an incompressible fluid) are:

$$\cancel{\rho\frac{\partial \mathbf{v}}{\partial t}} + \rho(\mathbf{v}\cdot\nabla)\mathbf{v} = -\nabla p$$

steady

but the y-velocity is zero and the flow is steady, so it becomes:

- Along $\hat{\mathbf{e}}_\mathbf{x}$: $\rho\left(V\frac{\partial}{\partial x}\right)V = -\frac{\partial p}{\partial x}$
- Along $\hat{\mathbf{e}}_\mathbf{y}$: $\rho\left(V\frac{\partial}{\partial x}\right)0 = -\frac{\partial p}{\partial y}$

Re-arranging the $\mathbf{e_x}$ component gives:

$$\frac{\partial}{\partial x}\left(\frac{1}{2}\rho V^2\right) + \frac{\partial p}{\partial x} = 0$$

$$\frac{\partial}{\partial x}\left(p + \frac{1}{2}\rho V^2\right) = 0 \longrightarrow \frac{\partial}{\partial x}(\text{Bernoulli}) = 0$$

and this can be integrated along the streamline to give:

$p + \frac{1}{2}\rho V^2 = p_0 =$ constant (stagnation pressure)

[with gravity, $p + \rho g h + \frac{1}{2}\rho V^2 =$ constant (total pressure)]

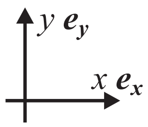
This is *Bernoulli's equation*, obtained from $\mathbf{f} = m\mathbf{a}$ in the absence of viscous forces. When integrated along the streamline, the force terms become energy terms in exactly the same way that the change in potential energy of a mass in a gravitational field is equal to the force integrated over the distance it moves. Indeed, the calculation can easily be repeated with gravity, which introduces an extra $\rho g z$ term. So each term can be thought of as an *energy per unit volume*. In an inviscid flow, Bernoulli's equation applies along any streamline, as we will see next. Furthermore, *if there is no vorticity in the flow then the total pressure is uniform and we can apply Bernoulli's equation across streamlines.* You will prove this when solving question 5 in Examples Paper 3.

Answer Q5 in Examples Paper 3

3.4 Bernoulli and streamline curvature

We can do the same analysis for flows in which the streamlines are not aligned with the cartesian coordinate system. The maths here is non-examinable but it is important to understand the result. The great advantage of vector notation is that it applies in any coordinate system:

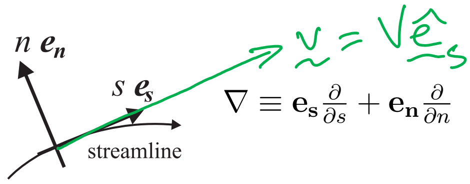
Cartesian coordinates



$$\nabla \equiv \mathbf{e_x}\frac{\partial}{\partial x} + \mathbf{e_y}\frac{\partial}{\partial y}$$

The advantage is that the unit vectors $\mathbf{e_x}$ and $\mathbf{e_y}$ do not change.

Instrinsic (streamline) coordinates



$$\nabla \equiv \mathbf{e_s}\frac{\partial}{\partial s} + \mathbf{e_n}\frac{\partial}{\partial n}$$

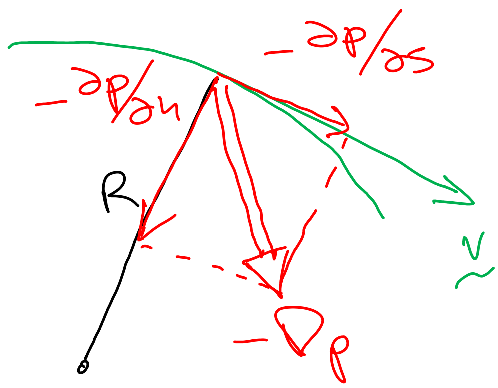
The advantage is that the velocity vector is simply $\mathbf{v} = V\mathbf{e_s}$. The disadvantage is that the unit vectors change as you go along a streamline. For example, $\partial \mathbf{e_s}/\partial s = -\mathbf{e_n}/R$, where R is the radius of curvature at that point on the streamline.

The Euler equations in steady flow can be re-arranged to:

$$\mathbf{v}\cdot\nabla\mathbf{v} = -\frac{1}{\rho}\nabla p$$

Applying the intrinsic definition of ∇ and noting that $\mathbf{e_s}\cdot\mathbf{e_s} = 1$ and $\mathbf{e_s}\cdot\mathbf{e_n} = 0$:

$$\begin{aligned}
\left[V\mathbf{e_s}\cdot\left(\mathbf{e_s}\frac{\partial}{\partial s} + \mathbf{e_n}\frac{\partial}{\partial n}\right)\right](V\mathbf{e_s}) &= \\
V\frac{\partial(V\mathbf{e_s})}{\partial s} &= \\
V\left(\frac{\partial V}{\partial s}\mathbf{e_s} + V\frac{\partial \mathbf{e_s}}{\partial s}\right) &= \\
V\frac{\partial V}{\partial s}\mathbf{e_s} - V^2\frac{1}{R}\mathbf{e_n} &= -\frac{1}{\rho}\left(\frac{\partial p}{\partial s}\mathbf{e_s} + \frac{\partial p}{\partial n}\mathbf{e_n}\right)
\end{aligned}$$



Resolving separately in the $\mathbf{e_s}$ and $\mathbf{e_n}$ directions gives:

$\hat{\underset{\sim}{e}}_s$: $$V\frac{\partial V}{\partial s} = -\frac{1}{\rho}\frac{\partial p}{\partial s}$$ along a streamline (as in 3.3) $\Rightarrow$ tangencial acceleration

$\frac{\partial}{\partial s}\left(\frac{1}{2}V^2 + p/\rho\right) = 0 \Rightarrow$ Bernoulli applies along streamlines

$\hat{\underset{\sim}{e}}_n$: $$\frac{V^2}{R} = \frac{1}{\rho}\frac{\partial p}{\partial n}$$ across a streamline (new) $\Rightarrow$ centripetal acceleration

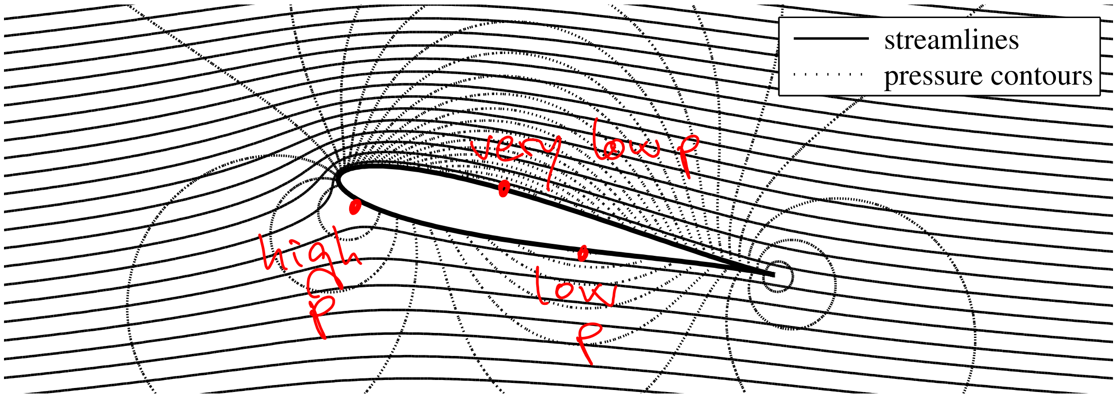
$\Rightarrow$ normal pressure gradients bend streamlines

The first of these integrates to give Bernoulli's equation, as before. The second describes how streamlines are bent by pressure gradients. Both equations are given on p9 of the databook.

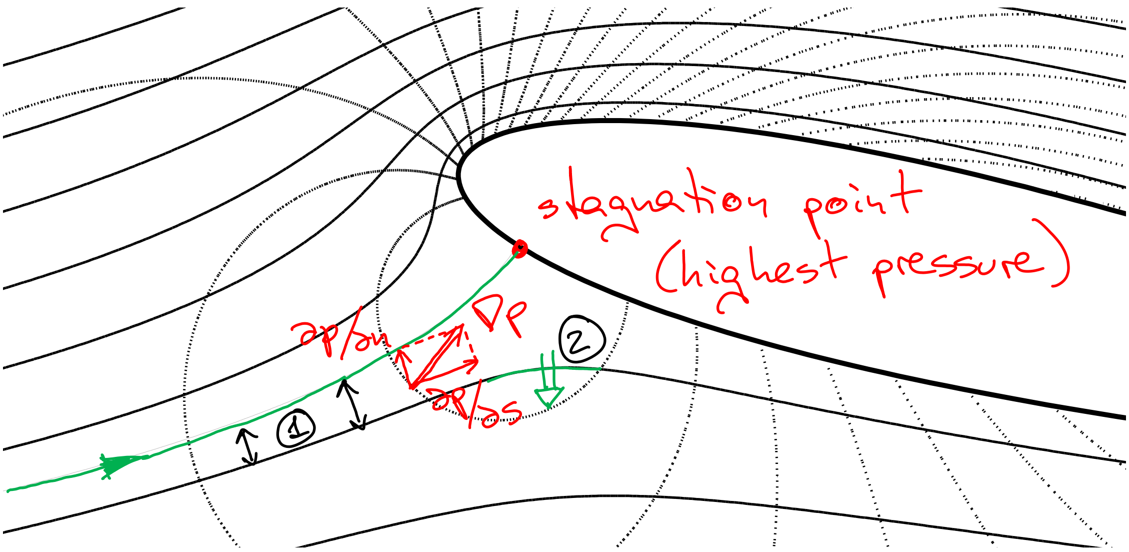
3.5 Determining the pressure field from a knowledge of the streamlines

Now we have the tools to determine the pressure field in a flow from a knowledge of the streamline shape. Along streamlines we can use Bernoulli's equation. Across streamlines we need to look at the streamline curvature.

Aerofoil at positive angle of attack:



Close-up of leading edge:



1 Along streamlines

- streamlines spread out

(continuity) ⇓

- velocity decreases

(Bernoulli) ⇓

- pressure increases

2 Across streamlines

- streamlines bend $(R \neq \infty)$

⇓ $(\partial p/\partial n = \rho V^2/R)$

- there must be a transverse pressure gradient

(higher p on outside of bend)

Now try Q2-6 in Examples Paper 3

3.6 Worked example - curved streamlines

A uniform water two-dimensional jet of density ρ, velocity V_0 and width h enters at A a semi-cylindrical surface, of radius r_2, supported at its mid-point B. The flow may be assumed steady and inviscid, and to have semi-circular streamlines between A and C.

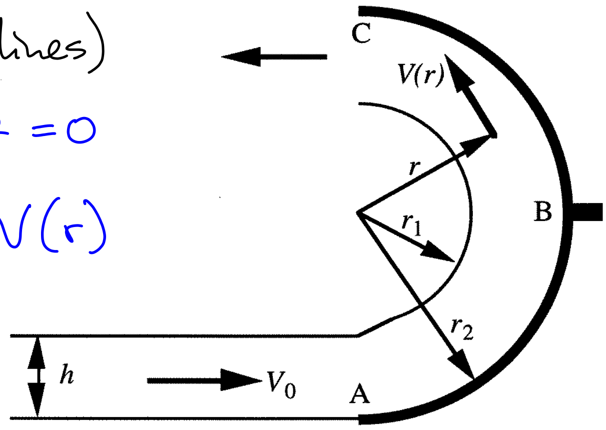
$\underset{\sim}{v} = \cancel{v_r \underline{\hat{e}}_r} + v_\theta \underline{\hat{e}}_\theta$

(circular streamlines)

$\nabla \cdot \underset{\sim}{v} = \cancel{\frac{\partial v_r}{\partial r}} + \cancel{\frac{v_r}{r}} + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} = 0$

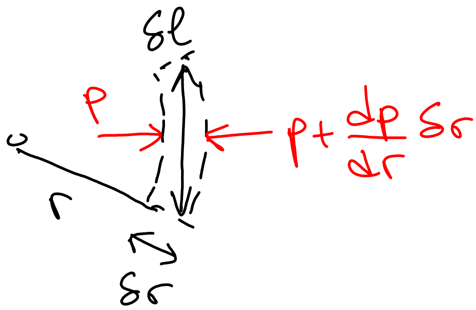
$\Rightarrow \partial v_\theta / \partial \theta = 0, \quad v_\theta = V(r)$

$\left\{ \begin{array}{l} \partial/\partial\theta = 0 \\ \partial/\partial r = d/dr \end{array} \right.$



- *Show, from first principles, that the pressure p at radius r in the jet satisfies the relation $dp/dr = \rho V^2/r$, where $V(r)$ is the jet velocity.*

We can write the force balance on a curved fluid element:



$$(f = ma)$$

$$p\,\delta\ell - \left(p + \frac{\mathrm{d}p}{\mathrm{d}r}\delta r\right)\delta\ell = -\rho\,(\delta\ell\delta r)\,\frac{V^2}{r}$$

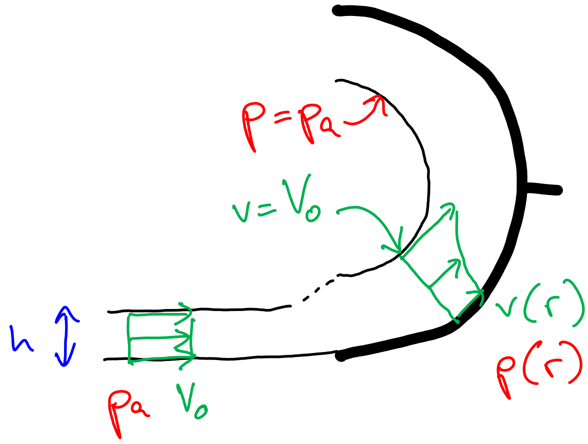
$$\frac{\mathrm{d}p}{\mathrm{d}r} = \rho\frac{V^2}{r}$$

(note that this is analogous to the expression in Section 3.4 for streamline curvature, which would however hold even when the streamlines are not circular)

- *The pressure in the uniform part of the jet is equal to the ambient pressure p_a. Write down an equation linking the pressure and velocity in the semi-circular region to p_a and V_0. Hence find $V(r)$ in terms of V_0, r and r_1. You may assume that the pressure at $r = r_1$ is also p_a.* (like in Q6, Ex.P.3)

Applying Bernoulli along streamlines, we have:

$$p_a + \frac{1}{2}\rho V_0^2 = p(r) + \frac{1}{2}\rho V(r)^2$$



Differentiating:

$$0 = \frac{\mathrm{d}p}{\mathrm{d}r} + \rho V \frac{\mathrm{d}V}{\mathrm{d}r}$$

And using the result from the previous question:

$$0 = \rho \frac{V^2}{r} + \rho V \frac{\mathrm{d}V}{\mathrm{d}r}$$

$$-\frac{\mathrm{d}r}{r} = \frac{\mathrm{d}V}{V} \quad \Rightarrow \quad V(r) = \frac{C}{r}$$

At $r = r_1$, $p = p_a$, and therefore –using Bernoulli– $V = V_0$, so

$$\boxed{V(r) = V_0 \frac{r_1}{r}}$$

- *Use the expression obtained for $V(r)$ to obtain the relationship between r_1, r_2, and h.*

  Applying the integral form of the continuity equation:

$$\int_{r_1}^{r_2} V \mathrm{d}r = \text{constant} \Rightarrow V_0 h = \int_{r_1}^{r_2} V_0 \frac{r_1}{r} \mathrm{d}r = V_0 r_1 \int_{r_1}^{r_2} \frac{\mathrm{d}r}{r} = V_0 r_1 \log \frac{r_2}{r_1}$$

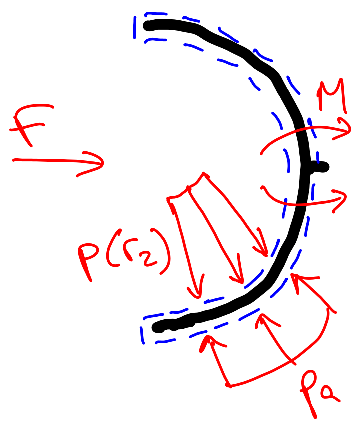
(note that in the limit $r_2 - r_1 \ll r_2$, $\log \frac{r_2}{r_1} = \log\left(1 + \frac{r_2 - r_1}{r_1}\right) \approx \frac{r_2 - r_1}{r_1}$, and therefore $r_2 - r_1 \approx h$. In that case, the jet maintains its thickness)

- *If the pressure on the outside of the semi-cylinder is p_a, find the pressure difference across it, and the bending moment at B per unit of cylinder length into the page.*

  Using Bernoulli, the pressure on the inside of the cylinder is

$$p(r_2) + \frac{1}{2}\rho \left(V_0 \frac{r_1}{r_2}\right)^2 = p_a + \frac{1}{2}\rho V_0^2,$$

$$p(r_2) = p_a + \frac{1}{2}\rho V_0^2 \left[1 - \left(\frac{r_1}{r_2}\right)^2\right],$$



and the pressure jump across the cylinder is

$$\Delta p = \frac{1}{2}\rho V_0^2 \left[1 - \left(\frac{r_1}{r_2}\right)^2\right].$$

From here, the force on each element of the cylinder can be easily obtained, as well as its contribution to the bending moment at B.

$$\begin{cases} F = 2 r_2 \Delta p \\ M = r_2^2 \Delta p \end{cases}$$

Engineering Tripos 1B

Paper 4

Fluid Mechanics

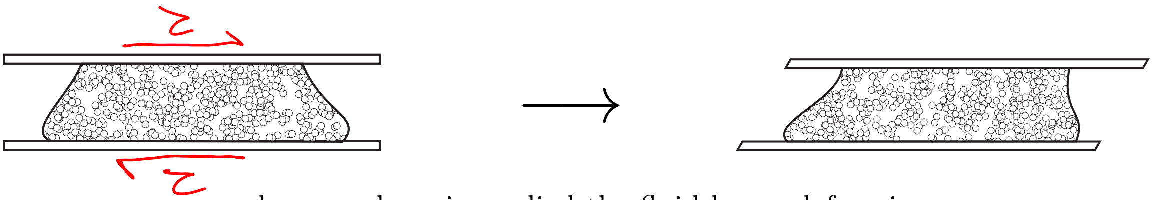
Lecture 4 - Viscous flow

- Viscosity and shear – recap
- The no-slip condition
- Viscous flow between parallel plates
- Couette flow
- Poiseuille flow
- The Navier-Stokes equations

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

4.1 Viscosity and shear – recap

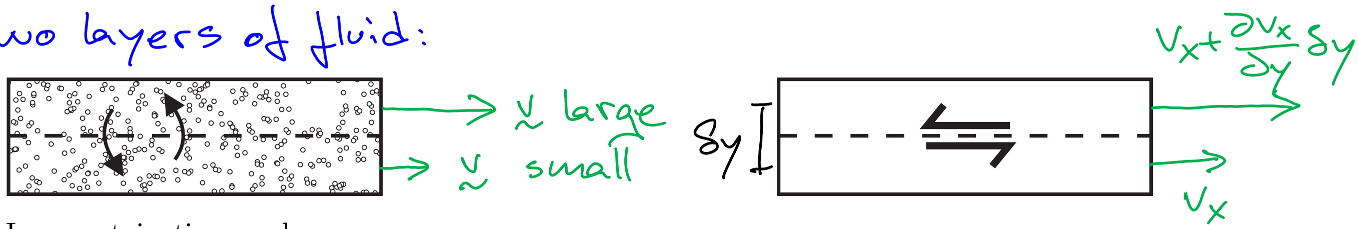
In a viscous flow, like a treacle, the molecules in the fluid do not have defined positions. When one layer is displaced, the molecules flow over each other to accommodate the displacement. There is a shear stress during that displacement, but once the motion stops and static equilibrium is reached, the shear stress disappears.



-so long as shear is applied the fluid keeps deforming
-when shearing ends the fluid stays in new equilibrium

Experimentally, it is observed that adjacent layers of conventional fluids exchange momentum at a rate that is proportional to the velocity gradient. By Newton's second law, $\mathbf{f} = m\mathbf{a}$, the rate of change of momentum across a certain area is simply a force. When divided by the area, this is the shear stress τ:

two layers of fluid:

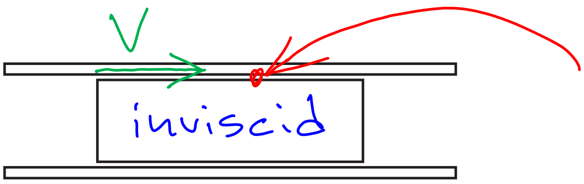


In a certain time and over a certain area, some molecules swap places ⇒ momentum exchange ⇒ shear force

The coefficient of proportionality is the viscosity, μ:

$$\boxed{\tau = \mu \frac{\mathrm{d}v_x}{\mathrm{d}y}}$$

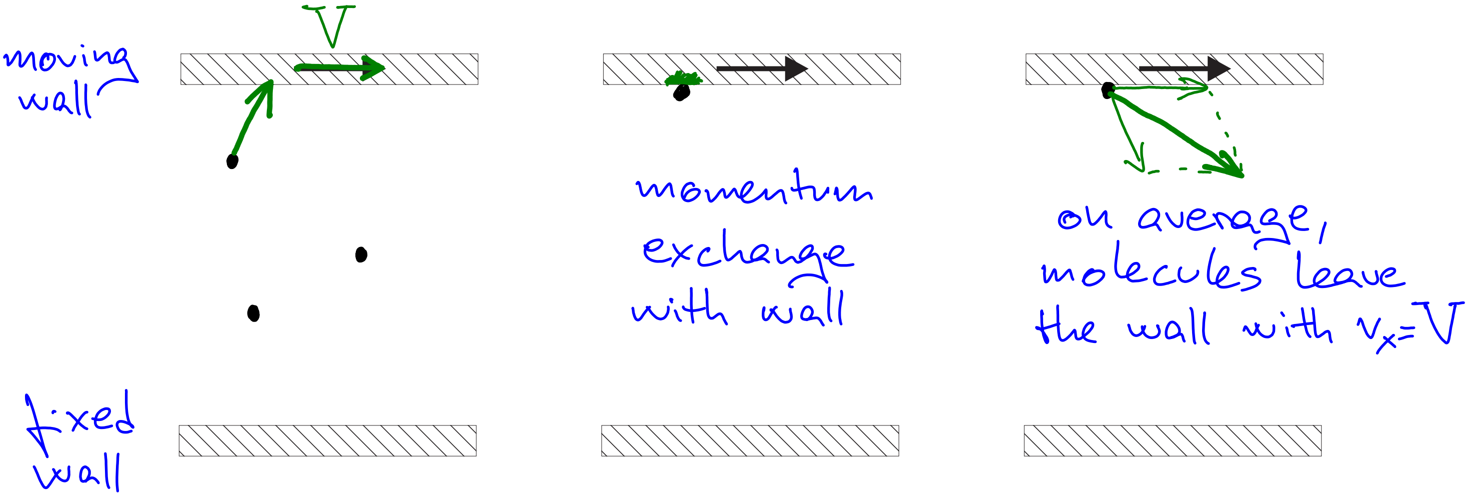
If the flow is inviscid, $\mu = 0$ and the flow is free to slip. It cannot support any shear stress at all, and the tangential velocity of the fluid in contact with the surface is completely independent from the velocity of the surface itself.



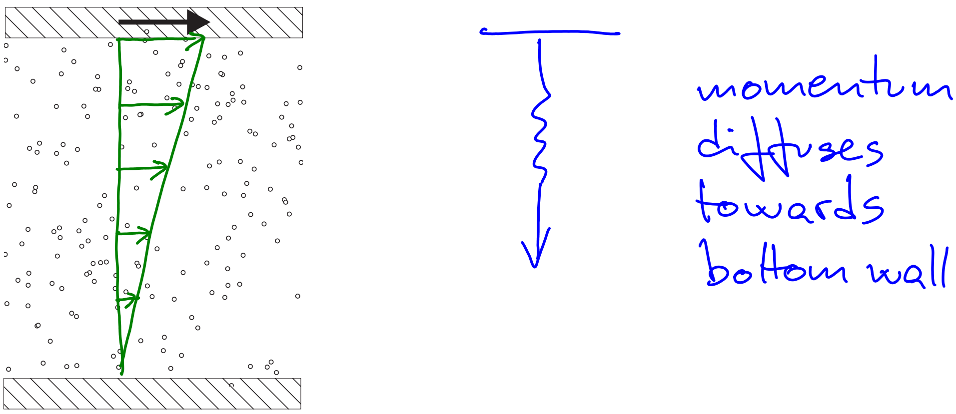
- no shear stress $\tau = 0$, free slip
- the flow sleeps freely over the surface

4.2 The no-slip condition and momentum transfer

Let us consider the molecules in a gas between two plates, where the top plate moves from left to right. From experiments we find that the molecules stick to surfaces for long enough to reach thermal equilibrium before they jump back into the gas. Consequently when they leave the surface they have, on average, the same x-velocity and the same temperature as the surface. This is the *no slip condition.*



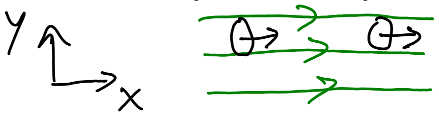
The molecules that have just left the surface collide into molecules nearby. After several collisions the extra x-momentum of the molecules coming from the top surface has been diffused into adjacent layers of fluid. These in turn jostle with the molecules adjacent to them, transferring x-momentum deeper into the fluid. Eventually x-momentum diffuses right down to the bottom plate and, averaging over all the molecules' velocities, one obtains a linear velocity profile. This is just like diffusion of heat, in which one obtains a linear temperature profile.



Watch the video on the no-slip condition in Moodle

4.3 Viscous flow between parallel plates

Let us consider steady, incompressible, viscous flows between two parallel plates. In these flows, the streamlines are straight and aligned with the plates, so from continuity the velocity is uniform along the flow direction.

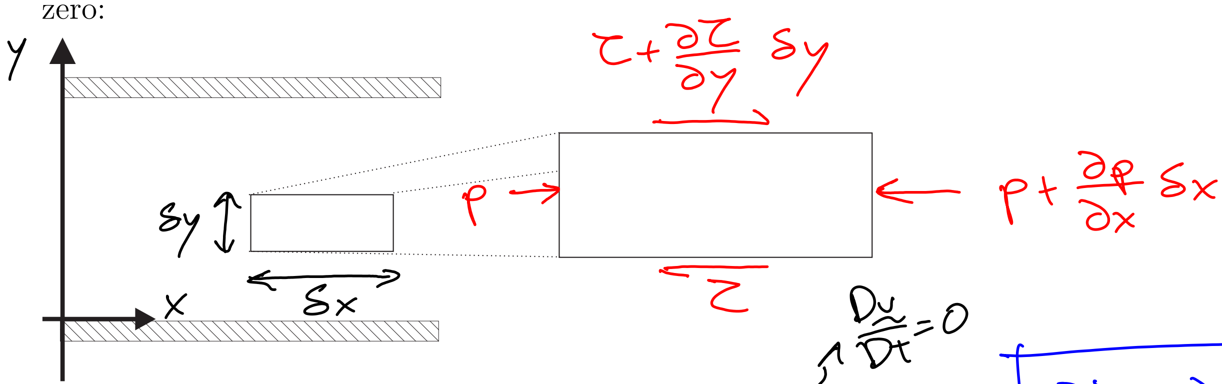


v_x constant along x, $v_y = 0$ $\quad (\nabla \cdot \underline{v} = 0)$

The flow acceleration is then zero everywhere:

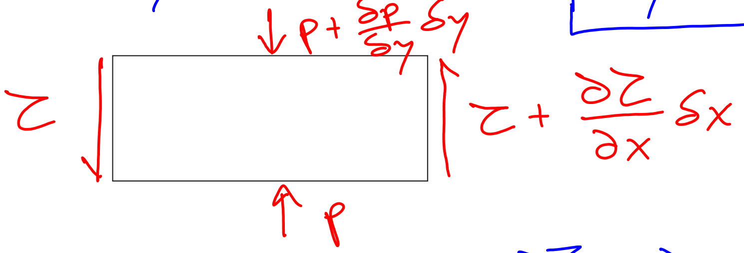
$$\underbrace{\frac{\mathrm{d}\mathbf{v}}{\mathrm{d}t}}_{\text{acceleration for fluid parcel}} = \underbrace{\frac{D\mathbf{v}}{Dt}}_{\text{material derivative for field}} = \left(\frac{\partial}{\partial t} + \mathbf{v}\cdot\nabla\right)\mathbf{v} = \underbrace{\cancel{\frac{\partial \mathbf{v}}{\partial t}}}_{\text{steady flow}} + \left(v_x \underbrace{\cancel{\frac{\partial}{\partial x}}}_{\text{velocity uniform along } x} + \underbrace{\cancel{v_y}}_{v_y = 0} \frac{\partial}{\partial y} \right) \mathbf{v} = 0$$

As there is no acceleration and $\mathbf{f} = m\mathbf{a}$, the forces on a control volume must sum zero:



net force along x $= -\frac{\partial p}{\partial x}\delta x \delta y + \frac{\partial \tau}{\partial y}\delta y \delta x = 0 \Longrightarrow \boxed{\frac{\partial \tau}{\partial y} = \frac{\partial p}{\partial x}}$

$\frac{D\underline{v}}{Dt} = 0$



net force along y $= -\frac{\partial p}{\partial y}\delta y \delta x + \frac{\partial \tau}{\partial x}\delta x \delta y = 0 \Longrightarrow \frac{\partial \tau}{\partial x} = \frac{\partial p}{\partial y}$

As we have seen, τ is a function of the velocity (of its gradient, actually), and since the velocity is uniform along x, from the force balance along y we have

$$\tau = \tau(y) \Longrightarrow \frac{\partial p}{\partial y} = \frac{\partial \tau}{\partial x} = 0 \Rightarrow \boxed{p = p(x)}$$

Looking now at the force balance along x, from our molecular argument we know that we can model the shear stress in terms of the viscosity of the fluid and the velocity gradient as $\tau = \mu \mathrm{d}v_x/\mathrm{d}y$, so we have

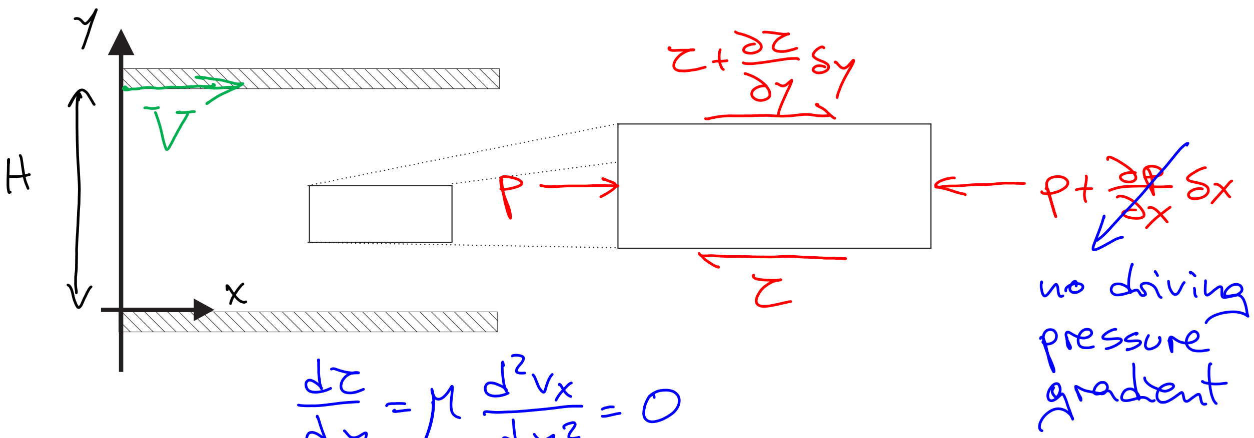
$$\frac{d\tau}{dy} = \frac{d}{dy}\left(\mu \frac{dv_x}{dy}\right) = \boxed{\mu \frac{d^2 v_x}{dy^2} = \frac{dp}{dx}} = \text{constant}$$

μ = constant

As v_x depends only on y, then so must $\mu \mathrm{d}^2 v_x/\mathrm{d}y^2$, and therefore $\mathrm{d}p/\mathrm{d}x$. But we also had, $p = p(x)$, so the only compatible possibility is that $\mathrm{d}p/\mathrm{d}x$ must be a constant.

4.4 Couette flow

When one of the plates of the previous section moves parallel to the other with velocity V, the flow is known as 'Couette flow'. In Couette flow, the pressure gradient is zero so the balance of forces reduces to:

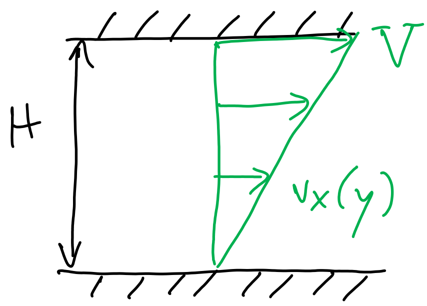


$$\frac{d\tau}{dy} = \mu \frac{d^2 v_x}{dy^2} = 0$$

This has a solution of the form $v_x = By + C$. The constants B and C are given by the boundary conditions:

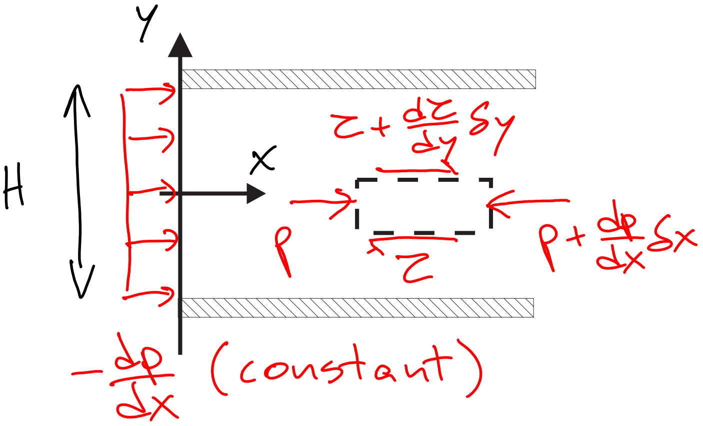
$$v_x = By + C \qquad \Longrightarrow \qquad v_x = V\, y/H$$

$$\text{boundary conditions} \begin{cases} v_x(y=0) = 0 \\ v_x(y=H) = V \end{cases}$$



4.5 Poiseuille flow

Let us now consider the flow between stationary plates driven by a constant pressure gradient. The velocity profile is obtained from a force balance on the same control volume:

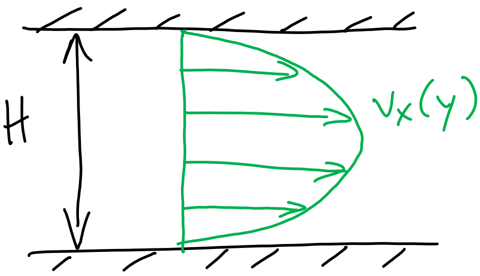


$$\frac{d\tau}{dy}\delta y \delta x - \frac{dp}{dx}\delta x \delta y = 0$$

$$\Rightarrow \frac{d\tau}{dy} = \frac{dp}{dx}$$

$$\Rightarrow \mu \frac{d^2 v_x}{dy^2} = \frac{dp}{dx} \; (= \text{constant})$$

$$\Rightarrow v_x = \left(\frac{1}{2\mu}\frac{dp}{dx}\right) y^2 + By + C$$

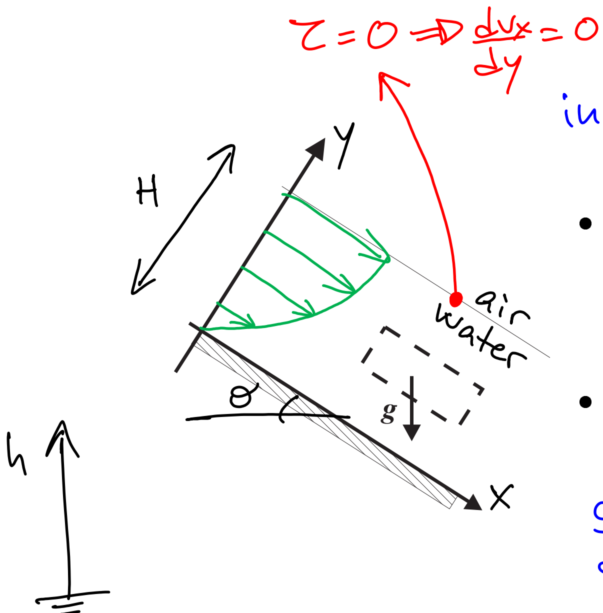


boundary conditions
$$\begin{cases} v_x(y = H/2) = 0 \\ v_x(y = -H/2) = 0 \end{cases}$$

$$\Rightarrow \boxed{v_x = -\left(\frac{1}{2\mu}\frac{dp}{dx}\right)\left(\frac{H^2}{4} - y^2\right)}$$

4.6 Viscous flow down a slope

A similar force balance can be performed on a steady viscous flow down a slope.



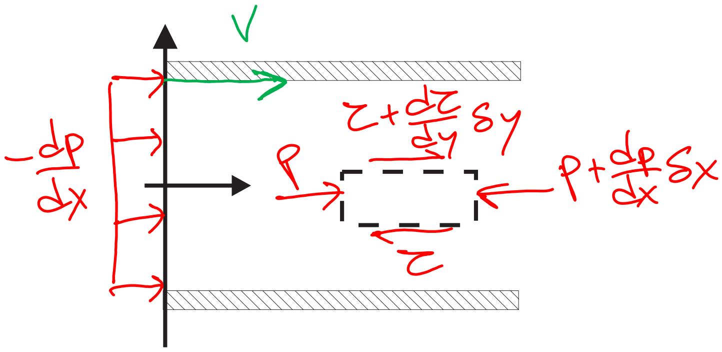
include gravity : $\nabla(p) \rightarrow \nabla(p + \rho g h)$

- along x: $\frac{d\tau}{dy} = \frac{d(p+\rho g h)}{dx}$ = constant
- along y: $0 = \frac{d\overbrace{(p+\rho g h)}}{dy}$ take for instance the value at $y = H$, with $p = p_a$, $h = h(x)$

$$\frac{d\tau}{dy} = \cancel{\frac{dp_a}{dx}} + \frac{d(\rho g h)}{dx} = -\rho g \sin\theta = \text{constant}$$

4.7 Combined Couette and Poiseuille flow

What happens when we combine Couette and Poiseuille flow? The force balance and the equation of motion are the same but the boundary conditions are different.



$$\frac{d\tau}{dy} = \frac{dp}{dx} \Rightarrow \mu \frac{d^2 v_x}{dy^2} = \frac{dp}{dx}$$

$$\Rightarrow v_x = \left(\frac{1}{2\mu}\frac{dp}{dx}\right) y^2 + By + C$$

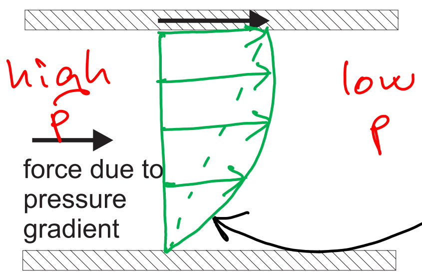
+ boundary conditions

$\Rightarrow$ Couette $\oplus$ Poiseuille

Depending on the sign of $\mathrm{d}p/\mathrm{d}x$, there are two types of solutions:

(a) Pressure pushes in the direction of the top plate's motion (*favourable* pressure gradient):

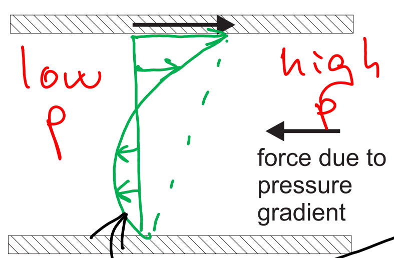
$dp/dx < 0$



- $v_x = v_x(y)$ is parabolic
- steep velocity gradient $\mathrm{d}v_x/\mathrm{d}y$ at bottom plate
- Couette $+$ Poiseuille

(b) Pressure pushes in the opposite direction to the top plate's motion (*adverse* pressure gradient):

$dp/dx > 0$



- $v_x = v_x(y)$ is parabolic
- the velocity profile may reverse near the bottom wall
- Couette $-$ Poiseuille

When the top plate moves one way and the pressure gradient pushes the other, there is a competition between the diffusion of momentum downwards from the top plate and the adverse pressure depleting that momentum, and the flow can reverse direction. A similar mechanism plays a critical role in boundary layers, as we will see in Lecture 8. A worked example of this flow can be found on the Moodle website.

4.8 The Navier-Stokes equations

The Navier-Stokes equations are the most important equations in fluid mechanics. They are introduced briefly here, and this is non-examinable. We derive them formally in the third year.

In Lecture 3 we derived the Euler equations, which are $\mathbf{f} = m\mathbf{a}$ for an inviscid fluid:

$$\underset{\sim}{f} = m\underset{\sim}{a}$$

$$-\nabla p = \rho \frac{D\underset{\sim}{v}}{Dt}$$

$$-\nabla p = \rho \frac{\partial \mathbf{v}}{\partial t} + \rho \mathbf{v} \cdot \nabla \mathbf{v}$$

inviscid $\Rightarrow$ EULER EQS.

When the shear stresses are included, there are extra force terms on the left hand side. These are shown here for a Newtonian, incompressible fluid such as water or air:

$$\frac{\partial}{\partial y}\left(\mu \frac{\partial \underset{\sim}{v}}{\partial y}\right)$$

$$\underbrace{\mu \frac{\partial^2 \mathbf{v}}{\partial x^2} + \mu \frac{\partial^2 \mathbf{v}}{\partial y^2} + \mu \frac{\partial^2 \mathbf{v}}{\partial z^2}} - \nabla p = \rho \frac{\partial \mathbf{v}}{\partial t} + \rho \mathbf{v} \cdot \nabla \mathbf{v}$$

$$\mu\left(\frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}\right)\underset{\sim}{v} = \mu\,(\underbrace{\nabla \cdot \nabla})\,\underset{\sim}{v}$$

$$\nabla \cdot \nabla = \nabla^2$$

In vector notation, this can be written as:

$$\mu(\nabla \cdot \nabla)\mathbf{v} - \nabla p = \rho \frac{\partial \mathbf{v}}{\partial t} + \rho \mathbf{v} \cdot \nabla \mathbf{v}$$

'divergence of gradient'
'Laplacian operator'

which is abbreviated to:

$$\mu \nabla^2 \mathbf{v} - \nabla p = \rho \frac{\partial \mathbf{v}}{\partial t} + \rho \mathbf{v} \cdot \nabla \mathbf{v}$$

These are the Navier-Stokes equations. They are $\mathbf{f} = m\mathbf{a}$ for a viscous fluid. They are normally written the other way round and divided by ρ:

$$\boxed{\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = -\frac{1}{\rho}\nabla p + \frac{\mu}{\rho}\nabla^2 \mathbf{v}}$$

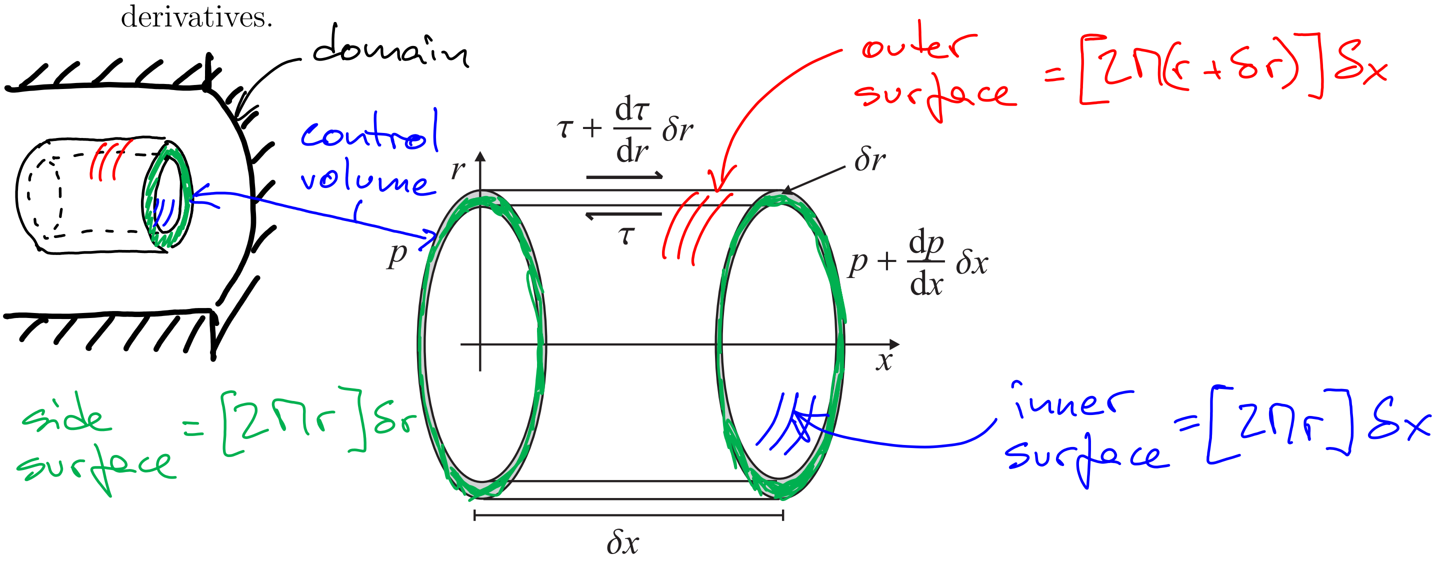
NAVIER-STOKES EQS.

The Navier-Stokes equations are very hard to solve in general, and not even a proof of the existence and uniqueness of their solution exists. There are only solutions for certain simple problems, with special symmetries or for which several of the terms in the equations are negligible. It is often the case that different terms can be neglected in different regions of a flow. Identifying those regions, which simplified equations hold in them, and combining the different local solutions is an ubiquitous challenge in fluid mechanics.

Answer Q7, Q8, Q9 and Q11 in Examples Paper 1

4.9 Worked example - viscous pipe flow

We would like to work out the velocity profile $v_x(r)$ in a pipe of radius R with a pressure gradient dp/dx. The control volume is a cylindrical shell of thickness δr and length δx. Note that the outer surface of this shell has a larger area than the inner surface. We assume that v_x and τ only vary in the r-direction and that p only varies in the x-direction, which means that our partial derivatives become ordinary derivatives.



If the flow is steady and R is constant, the forces on the fluid element must sum to zero:

$$(2\pi r\delta r)p - (2\pi r\delta r)\left(p + \frac{\mathrm{d}p}{\mathrm{d}x}\delta x\right) + 2\pi(r+\delta r)\left(\tau + \frac{\mathrm{d}\tau}{\mathrm{d}r}\delta r\right)\delta x - (2\pi r\tau)\delta x = 0$$

Multiplying out the brackets, cancelling terms and dropping the very small $(\delta r)^2$ term gives

$$-(2\pi r\delta r)\frac{\mathrm{d}p}{\mathrm{d}x}\delta x + 2\pi r\frac{\mathrm{d}\tau}{\mathrm{d}r}\delta r\delta x + 2\pi\delta r\tau\delta x = 0$$

which reduces to

$$\boxed{-\frac{\mathrm{d}p}{\mathrm{d}x} + \frac{\mathrm{d}\tau}{\mathrm{d}r} + \frac{\tau}{r} = 0}$$

note the additional term in curvilinear coordinates!

Now we substitute in $\tau = \mu \mathrm{d}v_x/\mathrm{d}r$:

$$-\frac{\mathrm{d}p}{\mathrm{d}x} + \mu\frac{\mathrm{d}^2 v_x}{\mathrm{d}r^2} + \frac{\mu}{r}\frac{\mathrm{d}v_x}{\mathrm{d}r} = 0$$

As in the 2D case, we need to integrate this twice in order to find $v_x(r)$. However, this is difficult when there are three terms. The clever trick is to notice that the second two terms can be re-written as a single term,

$$-\frac{\mathrm{d}p}{\mathrm{d}x} + \frac{\mu}{r}\frac{\mathrm{d}}{\mathrm{d}r}\left(r\frac{\mathrm{d}v_x}{\mathrm{d}r}\right) = 0$$

Now we integrate this, leaving in constants (A and B) instead of specifying the bounds of the integration:

$$\frac{\mu}{r}\frac{\mathrm{d}}{\mathrm{d}r}\left(r\frac{\mathrm{d}v_x}{\mathrm{d}r}\right) = \frac{\mathrm{d}p}{\mathrm{d}x} \qquad \Rightarrow \int (\quad) \mathrm{d}r$$

$$\Rightarrow \quad r\frac{\mathrm{d}v_x}{\mathrm{d}r} = \int \frac{\mathrm{d}p}{\mathrm{d}x}\frac{r}{\mu}\mathrm{d}r = \frac{\mathrm{d}p}{\mathrm{d}x}\frac{r^2}{2\mu} + B$$

$$\Rightarrow \quad \frac{\mathrm{d}v_x}{\mathrm{d}r} = \frac{\mathrm{d}p}{\mathrm{d}x}\frac{r}{2\mu} + \frac{B}{r}$$

$$\Rightarrow \quad \int \mathrm{d}v_x = \int \frac{\mathrm{d}p}{\mathrm{d}x}\frac{r}{2\mu}\mathrm{d}r + \int \frac{B}{r}\mathrm{d}r$$

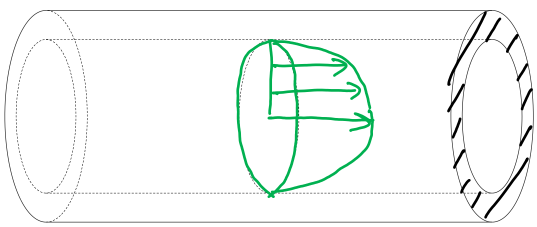
$$\Rightarrow \quad v_x = \frac{\mathrm{d}p}{\mathrm{d}x}\frac{r^2}{4\mu} + B\ln r + A$$

singular at $r=0$

For the present configuration, the constants A and B can be found by imposing the no slip condition at the pipe wall, $v_x(r = R) = 0$ and that the solution is not singular at $r = 0$, which forces $B = 0$. From these we finally obtain the solution,

$$\boxed{v_x = -\frac{\mathrm{d}p}{\mathrm{d}x}\left(\frac{R^2 - r^2}{4\mu}\right)}$$

We can recognise the parabolic profile, analogous to that of Poiseuille flow between two plates.



Answer Q10 in Examples Paper 1

Engineering Tripos 1B

Paper 4

Fluid Mechanics

Lecture 5 - Dimensional analysis and scaling

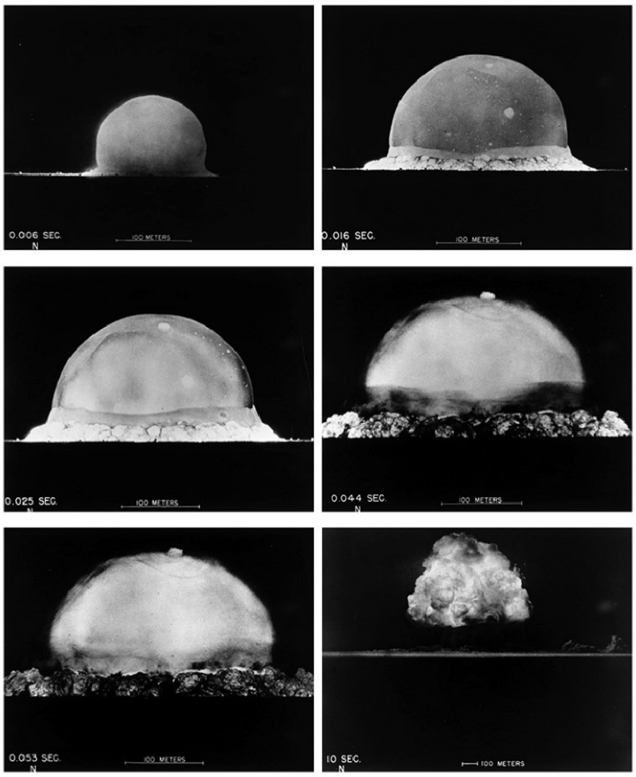
- Introduction
- Example - Drag on a sphere
- The π theorem
- The dimensionless form of Navier-Stokes equations
- Order-of-magnitude analysis
- Example - An orifice plate
- Example - An aeroplane
- Example - Drag force on a ship

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

5.1 The power of dimensional analysis - A historical note

Dimensional analysis is a very powerful tool, because it allows us to say a lot of things about problems for which we have very little information. It is an essential tool in fluid mechanics, as it often allows us to circumvent the impossibility to solve Navier-Stokes equations. It also allows us to minimise the number of experiments required to characterize a given phenomenon, and to obtain results applicable to a problem too big or too expensive from a down-scaled model.

A famous historical example of the power of dimensional analysis is the study of an atomic blast by G.I. Taylor. In 1941, he was tasked by the Ministry of Home Security to estimate the amount of energy that would be released by the explosion of an atomic bomb. The subsequent work was kept classified for some years but eventually was made available to the public in 1950. The US Government kept the data from their nuclear tests secret for even longer, but had released some photographs of the first detonation of a nuclear weapon, the Trinity test, believing that these did not contain any sensitive information.



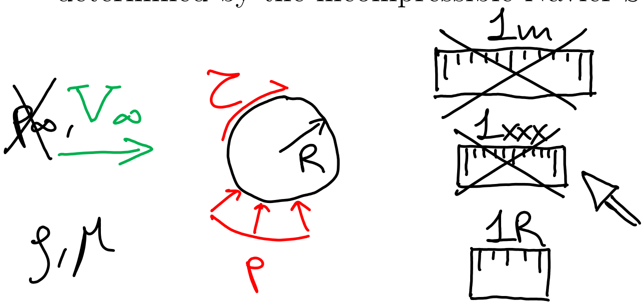
In his 1950 paper, however, Taylor applied dimensional analysis to those photographs and estimated the power of the US blast to be between 17 and 24 kilotons. After he published his work, Taylor was "mildly admonished by the US Army for publishing his deductions." When the data from the Trinity test was eventually declassified, the actual power turned out to be about 20 kilotons. Taylor had anticipated the result from very little information –so little that the US had released it thinking that it was not relevant.

5.2 Dimensional analysis through an example - Drag on a sphere

To illustrate how dimensional analysis works, let us use the example of the drag force on a sphere. In the case of viscous, incompressible flow, the drag would be made up of the pressure and shear forces on the sphere,

$$D = \int_\Sigma \left[-p\mathbf{e_n} + \tau \mathbf{e_t} \right]_x \, \mathrm{d}\sigma.$$

To obtain the value of D we would then need to solve the flow around the sphere, determined by the incompressible Navier-Stokes equations,



$$\begin{cases} \rho \dfrac{D\underset{\sim}{v}}{Dt} = -\nabla p + \mu \nabla^2 \underset{\sim}{v} \\ \nabla \underset{\sim}{v} = 0 \end{cases}$$

The scale chosen <u>cannot</u> influence the physical laws!

with boundary conditions $\mathbf{v} = 0$ on the sphere and $\mathbf{v} = V_\infty \mathbf{e_x}$ and $p = P_\infty$ far away from it. In incompressible flows, only the derivatives of the pressure appear in the equations, so we can subtract P_∞ everywhere without changing the solution, thereby eliminating P_∞ as a parameter. The solution for D will then be of the form

$$D = f(R, \rho, V_\infty, \mu, \cancel{P_\infty})$$

We can now start our dimensional analysis. The first thing to note is that, as you saw in IA Dimensional Analysis, the laws of nature are *dimensionally consistent*. For instance $\mathbf{f} = m\mathbf{a}$ will hold no matter what scale we decide to use to measure $\mathbf{f}$, m and $\mathbf{a}$. If we had for instance $6\mathrm{N} = 3\mathrm{kg} \times 2\mathrm{m/s}^2$, and decided for some reason to measure distances in centimetres instead of meters, but keeping the set of units self-consistent, we would need to carry this into any derived magnitude, like the force or the acceleration. We would then have $600\mathrm{cN} = 3\mathrm{kg} \times 200\mathrm{cm/s}^2$, so the equality –and Newton's law– would still hold. The idea of a metre, or of a centimetre, are human conventions completely extraneous to nature itself, and cannot therefore play any role in the laws of physics.

In the problem of the sphere, this tells us that the solution for D cannot depend on the scale that we choose to measure, say, the radius R or any other distance in the problem. We may choose to measure the radius as 50cm, 0.5m, or in fact, *1 radius*, $1R$. This would actually be a rather natural choice as a scale for lengths in our problem. Following the choice of R as the length scale, we would need to carry it into any derived magnitude. Let us look at which magnitudes are fundamental and which derived:

$[D] = MLT^{-2}$; $[R] = L$; $[\rho] = ML^{-3}$; $[V_\infty] = LT^{-1}$; $[\mu] = ML^{-1}T^{-1}$

We would then need to rescale not only R, which is now 1 in 'radius units,' $R' = 1$, but also D, ρ, V_∞ and μ, taking into account the power of L for each of them. We then have

$$D' = \frac{D}{R} \quad ; \quad R' = 1 \quad ; \quad \rho' = \rho R^3 \quad ; \quad V'_\infty = \frac{V_\infty}{R} \quad ; \quad \mu' = \mu R \, .$$

Remember that we had the physical relation $D = f(R, \rho, V_\infty, \mu)$, and that it must be invariant. Therefore, we can express it in our new set of scales, for which it will be

$$D' = f(R', \rho', V'_\infty, \mu')$$

$$\frac{D}{R} = f\left(1, \rho R^3, \frac{V_\infty}{R}, \mu R\right)$$

so, simply by dimensional considerations, we have removed the dependence of the problem on R as an independent entity!

But we've only just begun. We can rescale the mass with $\rho' = \rho R^3$, so that $\rho'' = 1$. We would need to carry this rescaling into any magnitude whose dimensions include M, that is D', ρ', and μ'. We would then have

$$D'' = f(R'', \rho'', V''_\infty, \mu'')$$

$$\frac{D}{\rho R^4} = f\left(1, 1, \frac{V_\infty}{R}, \frac{\mu}{\rho R^2}\right)$$

Finally, we can rescale the time with R/V_∞, so that $V'''_\infty = 1$. Proceeding as before, we obtain,

$$\frac{D}{\rho V_\infty^2 R^2} = f\left(1, 1, 1, \frac{\mu}{\rho V_\infty R}\right)$$

or

$$\frac{D}{\rho V_\infty^2 R^2} = F\left(\frac{\mu}{\rho V_\infty R}\right) .$$

"dimensionless"
"non-dimensional"
"adimensional"

We have derived this in a step-by-step procedure, but with a bit of practice you will soon get used to doing this in a single step.

"the dimensionless drag depends only on the dimensionless viscosity"

Notice the implications of the above expression. We have found that *the dimensionless drag depends only on the dimensionless viscosity.* If, for example, we had to find the drag on a sphere of size 1 m when immersed in a free stream of air with velocity 10 m/s, we could do a reduced-scale model of size 10 cm and measure the drag in the lab, under a free stream with velocity 1 m/s, if we guaranteed that

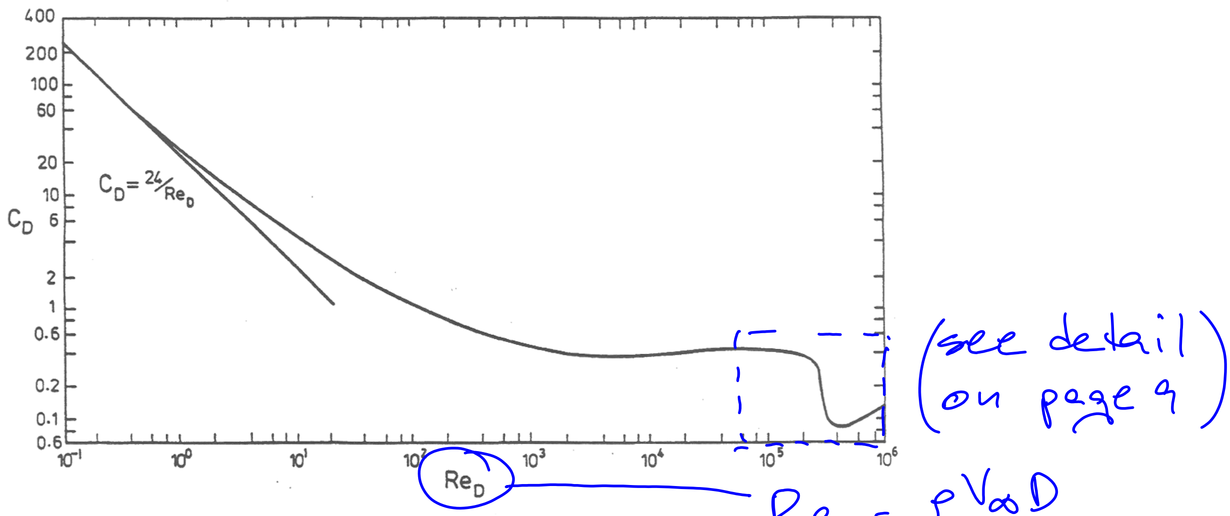
$$\left[\frac{\mu}{\rho\, V_\infty\, R}\right]_{\text{lab}} = \left[\frac{\mu}{\rho\, V_\infty\, R}\right]_{\text{real}} \quad \Rightarrow \quad \left[\frac{\mu}{\rho}\right]_{\text{lab}} = \frac{1}{100}\left[\frac{\mu}{\rho}\right]_{\text{real}}.$$

We would need to choose the fluid for the experiment accordingly. Once we measure the drag in the lab, the real value would be obtained by rescaling the obtained dimensionless value,

$$D_{\text{real}} = \left[\rho V_\infty^2 R^2\right]_{\text{real}} \left[\frac{D}{\rho V_\infty^2 R^2}\right]_{\text{lab}}.$$

Another important implication is that, if we wanted to compile the drag for a wide range of sphere sizes, flow velocities and fluid densities and viscosities, we know that we can reduce the number of measurements substantially, because *the solution, when expressed in dimensionless form, depends only on the dimensionless viscosity.*

There is no end to the applications of this technique in engineering in general, and in fluid problems in particular. It is very frequent to find graphs like the one below, which compiles values of the drag coefficient (the dimensionless drag based on the diameter, $C_D = D/[\rho V_\infty^2 (2R)^2]$) as a function of the Reynolds number (the inverse of the dimensionless viscosity based also on the diameter, $Re_D = \rho\, V_\infty\, (2R)/\mu$).



Drag coefficient of spheres as a function of Reynolds number.

$Re_D = \dfrac{\rho V_\infty D}{\mu}$

We can plot C_D for a sphere as a function of Re_D alone. Each point on the line corresponds to the solution for a particular combination $\rho\, V_\infty\, R/\mu$, and it is valid for *all* perfectly smooth spheres. We will revisit this problem in Lecture 10, and find out why the curve has the irregular shape that it has.

5.3 The π theorem

The procedure followed in the previous section can be generalised in what is known as the π theorem:

If a given physical phenomenon can be represented by a mathematical equation that involves a certain number N of physical variables, and the number of fundamental dimensions involved is K, the original equation can be rewritten merely in terms of a set of $P = N - K$ dimensionless parameters.

number of variables − number of fundamental dimensions = number of truly independent (dimensionless) variables

$$N \quad - \quad K \quad = \quad P$$

For instance, in the example of Section 5.2, we would have:

$$(D, R, \rho, V_\infty, \mu) - [M, L, T] = \left(\frac{D}{\rho V_\infty^2 R^2}, \frac{\rho V_\infty R}{\mu}\right)$$

$$5 \quad - \quad 3 \quad = \quad 2$$

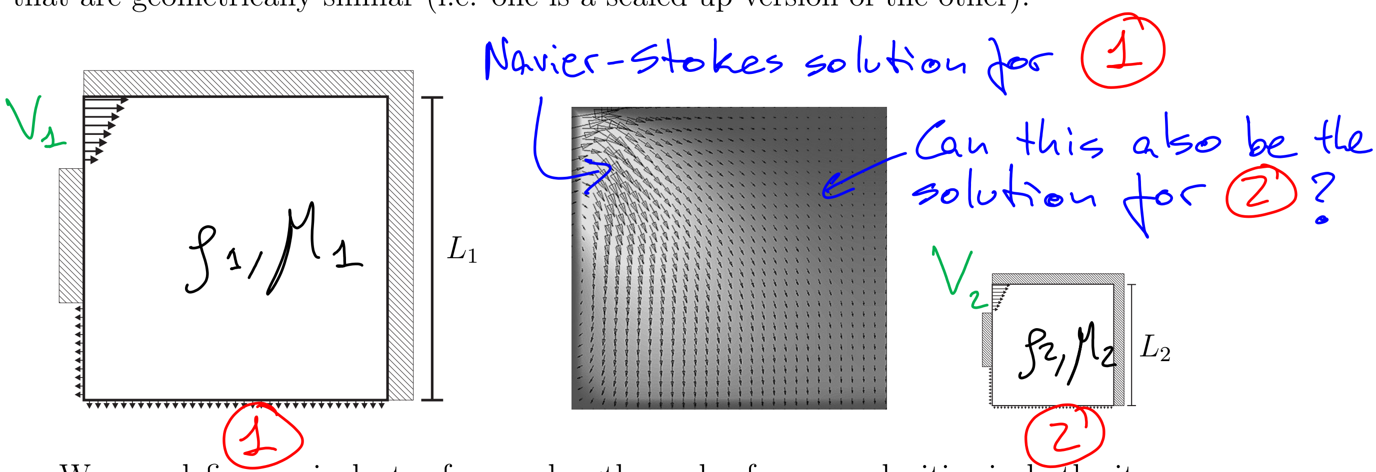
(rarely more than these in IB Fluids)

The π theorem is also known as the Buckingham theorem (mostly in English-speaking countries), and also as the Vaschy-Buckingham theorem (elsewhere). It was given by Vaschy in 1892, and by Buckingham in 1914.

During this course, we will rarely encounter problems that involve more than three fundamental dimensions, $K = 3$. These will be those related to dynamics, namely L, T, and M. Problems with richer physics, for instance involving electrical forces, would need additional fundamental dimensions, for instance the electrical charge Q.

5.4 The dimensionless form of Navier-Stokes equations

So far we have considered dimensional analysis in scenarios where we do not necessarily know the equations that govern the motion of the fluid, but the analysis is equally powerful when we know those equations, if not more. We have seen in Lecture 4 that the Navier-Stokes equations are $\mathbf{f} = m\mathbf{a}$ for a viscous fluid. They are partial differential equations that must be satisfied at every point in the flow. Given a set of boundary and initial conditions and the physical properties of the fluid, the equations have a given solution. Let us imagine that we have two flows that are geometrically similar (i.e. one is a scaled-up version of the other):

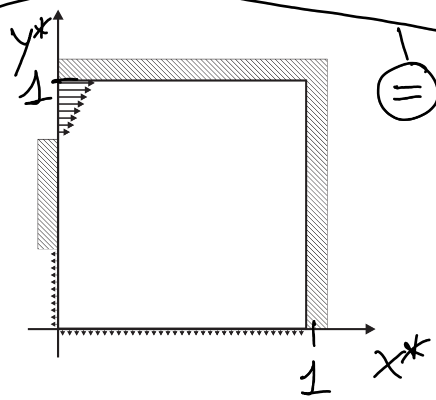


We can define equivalent reference lengths and reference velocities in both situations, and measure all distances in these units. In the new units, the boundary conditions for both problems are identical, but the fluid properties ρ and μ are not.

1:

$$\begin{aligned} x_1 &= x^*\, L_1 \\ y_1 &= y^*\, L_1 \\ \mathbf{v_1} &= \mathbf{v}^*\, V_1 \\ t_1 &= t^*\, L_1/V_1 \\ p_1 &= p^*\, \rho_1 V_1^2 \end{aligned}$$

physical props: $\rho_1,\ \mu_1$



=

2:

$$\begin{aligned} x_2 &= x^*\, L_2 \\ y_2 &= y^*\, L_2 \\ \mathbf{v_2} &= \mathbf{v}^*\, V_2 \\ t_2 &= t^*\, L_2/V_2 \\ p_2 &= p^*\, \rho_2 V_2^2 \end{aligned}$$

physical props: $\rho_2,\ \mu_2$

Note also that: $\dfrac{\partial}{\partial x_1} = \dfrac{\partial}{\partial (x^* L_1)} = \dfrac{1}{L_1}\dfrac{\partial}{\partial x^*}$

$$\nabla_1 = \begin{pmatrix} \frac{\partial}{\partial x_1} \\ \frac{\partial}{\partial y_1} \end{pmatrix} = \begin{pmatrix} \frac{\partial}{\partial (x^* L_1)} \\ \frac{\partial}{\partial (y^* L_1)} \end{pmatrix} = \frac{1}{L_1}\begin{pmatrix} \frac{\partial}{\partial x^*} \\ \frac{\partial}{\partial y^*} \end{pmatrix} = \frac{\nabla^*}{L_1} \qquad \text{and} \qquad \nabla_2 = \frac{\nabla^*}{L_2}$$

We may then ask: under what conditions would the solutions for the two problems be identical, other than the obvious answer of equal L, V, ρ and μ? Let us write the *dimensional* Navier-Stokes equations for both problems side by side, and substitute in x^*, $\mathbf{v}^*$ etc. from the previous page. We can rearrange the resulting expressions so that they depend only on a single dimensionless parameter. We then obtain the *non-dimensional* form of Navier-Stokes equations for both situations:

Big square	Small square
$\rho_1 \dfrac{D\mathbf{v_1}}{Dt} = -\nabla_1 p_1 + \mu_1 (\nabla_1)^2 \mathbf{v_1}$	$\rho_2 \dfrac{D\mathbf{v_2}}{Dt} = -\nabla_2 p_2 + \mu_2 (\nabla_2)^2 \mathbf{v_2}$
$\rho_1 \dfrac{D(\mathbf{v}^* V_1)}{D(t^* L_1/V_1)} = -\dfrac{\nabla^*}{L_1}(p^* \rho_1 V_1^2) + \mu_1 \left(\dfrac{\nabla^*}{L_1}\right)^2 (\mathbf{v}^* V_1)$	$\rho_2 \dfrac{D(\mathbf{v}^* V_2)}{D(t^* L_2/V_2)} = -\dfrac{\nabla^*}{L_2}(p^* \rho_2 V_2^2) + \mu_2 \left(\dfrac{\nabla^*}{L_2}\right)^2 (\mathbf{v}^* V_2)$
$\dfrac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \dfrac{\mu_1}{\rho_1 V_1 L_1} (\nabla^*)^2 \mathbf{v}^*$	$\dfrac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \dfrac{\mu_2}{\rho_2 V_2 L_2} (\nabla^*)^2 \mathbf{v}^*$
$\Rightarrow \boxed{\dfrac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \dfrac{1}{Re_1} (\nabla^*)^2 \mathbf{v}^*}$	$\Rightarrow \boxed{\dfrac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \dfrac{1}{Re_2} (\nabla^*)^2 \mathbf{v}^*}$

identical if and only if $Re_1 = Re_2$

If the boundary and initial conditions are identical in '*' variables, *for the equations to have exactly the same solution the Reynolds numbers must also be the same*. In addition to *geometrical similarity*, we would then also have *dynamical similarity*. If there were more physical processes involved in the problem, we could similarly seek complete *physical similarity*. In our case, the Reynolds number is the *only* control parameter for geometrically-similar objects. For compressible flows, which are beyond the scope of this course, there would be an additional control parameter in the dimensionless Navier-Stokes equations, the Mach number.

5.5 Order-of-magnitude analysis

When we choose typical values of the variables in a problem to write it in dimensionless form, as we have done in the previous section by choosing ρ_i, V_i, L_i, μ_i, the corresponding dimensionless variables are of order one. The resulting dimensionless numbers give us an estimate of the relative importance of the different terms in the equations. This procedure can also be done directly on the equations in dimensional form, and is known as *order-of-magnitude analysis*.

$$\rho \frac{\partial \mathbf{v}}{\partial t} \quad + \quad \rho\, \mathbf{v} \cdot \nabla \mathbf{v} \quad = \quad -\nabla p \quad + \quad \mu \nabla^2 \mathbf{v}$$

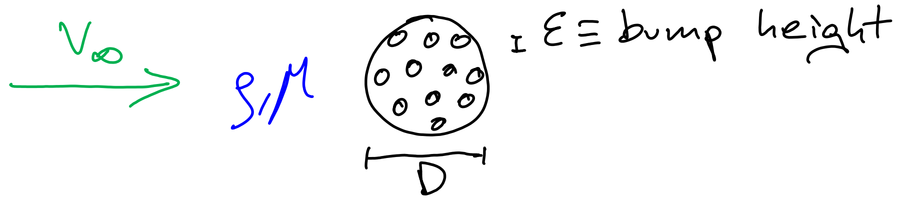
$$\left(\sim \rho \frac{V_\infty}{T_c}\right) \qquad \sim \rho V_\infty \frac{V_\infty}{L} \qquad \sim \frac{\Delta p}{L} \qquad \sim \mu \frac{V_\infty}{L^2}$$

$$\left(\sim \frac{L}{V_\infty T_c}\right) \qquad \sim 1 \qquad \sim 1 \qquad \sim \frac{\mu}{\rho V_\infty L}$$

Re measures the relative importance of viscous effects

5.6 Dimensional analysis - an engineers approach

Let us consider the problem of the drag of the sphere, when its surface is rough. A new parameter would then be introduced: the height of the bumps, ϵ. An engineer would seek to reduce the problem to its simplest expression, and could follow the following procedure:



Step 1 - Decide which variables you want to measure (the dependent variables) and the variables which have influence on the problem (the independent variables).

dependent vars	independent variables
drag	$\rho,\ V_\infty,\ D,\ \mu,\ \epsilon$

(total = 6)

Step 2 - Count up the number of fundamental dimensions (mass, length, time, etc.) and subtract this from the number of variables to obtain the number of dimensionless numbers in the problem.

6 variables − 3 fundamental dimensions = 3 dimensionless variables

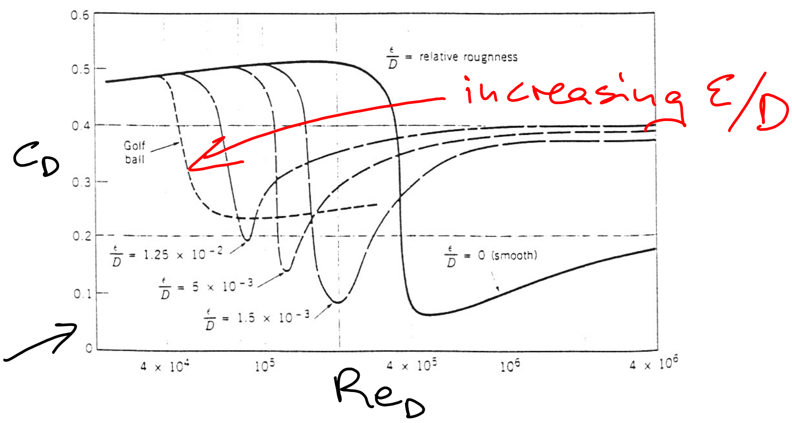
Step 3 - Create the dimensionless numbers. There are often several ways to do this but it is usually convenient to use standard dimensionless numbers, such as Reynolds number, Mach number etc. These can be found in the Thermofluids data book.

dependent	independent
$C_D = \dfrac{\text{drag}}{\rho V_\infty^2 D^2}$	$\underbrace{\dfrac{\rho V_\infty D}{\mu}}_{Re_D},\ \dfrac{\epsilon}{D}$

$$\Rightarrow C_D = f(Re_D, \epsilon/D)$$

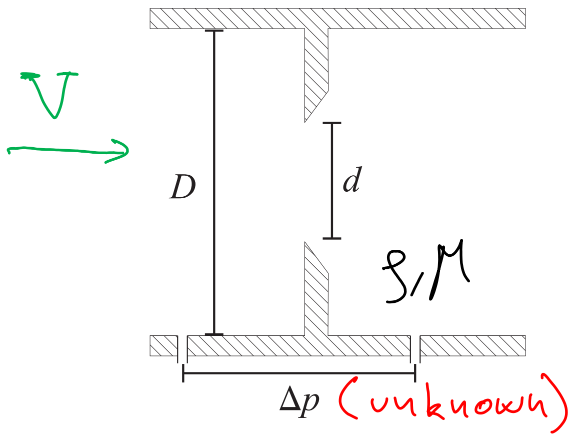
Step 4 - Create an experiment or a numerical simulation to measure the dependent dimensionless number as a function of the independent control parameters:

(detail from figure on page 5)



5.7 Example - An orifice plate

In Lecture 7 we will work out the pressure drop across an orifice plate, Δp, in terms of the average velocity upstream, V, using a simple model of the flow. However, the real flow is more complicated than that assumed by the simple model. In real life, we need to do experiments (or numerical simulations) to obtain more accurate values of Δp as a function of V. How do we express these in a way that is easily scalable to geometrically-similar orifice plates?



Step 1	Dependent variable	Independent variables
	Δp	ρ, V, μ, D, d $\rightarrow$ 6 total

Step 2 - count up the dimensions:

$$6 - 3 = 3$$

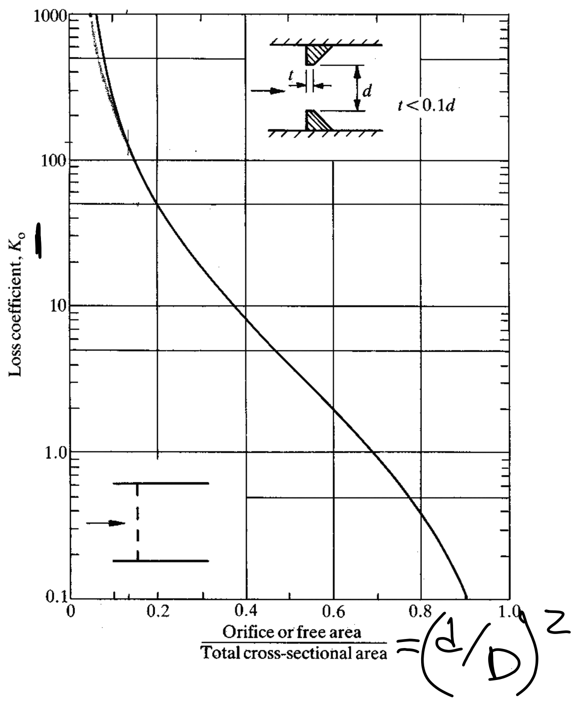
Step 3 - create the dimensionless numbers:

dep	indep
$K_o = \dfrac{\Delta p}{\frac{1}{2}\rho V^2}$	$Re_D = \dfrac{\rho V D}{\mu}$, d/D

Step 4 - carry out the experiment:

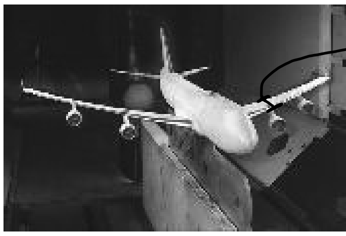
$$K_o = f(Re_D, d/D)$$

(but K_o becomes independent of Re_D at sufficiently high Re_D)



5.8 Example - An aeroplane

We want to evaluate the lift and drag coefficients of a Boeing 747 by testing a geometrically-similar model in a wind tunnel. What conditions are required in the wind tunnel for complete similarity? We need to keep in mind that commercial aeroplanes fly at transonic speeds, so the density of the fluid cannot be taken as a constant.



cord L
wing surface area $A \sim L^2$

Step 1: identify dependent and independent variables

Dependent	Independent
drag, lift	ρ, V, L, μ, α, a

$\rightarrow$ (total = 8)

angle of attack

speed of sound (accounts for compressible effects when $V \gtrsim a$)

Step 2: count up the dimensions

$8 - 3 = 5$ dimensionless variables

Step 3: create the dimensionless numbers

Dependent	Independent
$C_D = \dfrac{\text{drag}}{\frac{1}{2}\rho V^2 A}$, $C_L = \dfrac{\text{lift}}{\frac{1}{2}\rho V^2 A}$	Re, α, M

Mach number, $\dfrac{V}{a}$

$$\boxed{\begin{aligned} C_L &= f_1(Re, \alpha, M) \\ C_D &= f_2(Re, \alpha, M) \end{aligned}}$$

Step 4 - It is easy to match the angles of attack. However, for complete similarity we require $M_m = M_f$ and $Re_m = Re_f$:

(model) (full-scale)

$$M_m = M_f \qquad \text{and} \qquad Re_m = Re_f$$

that is:

$$\frac{V_m}{a_m} = \frac{V_f}{a_f} \qquad \text{and} \qquad \frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_f V_f L_f}{\mu_f}$$

re-arranging gives:

$$\frac{V_m}{V_f} = \frac{a_m}{a_f} \qquad \text{and} \qquad \frac{V_m}{V_f} = \frac{\rho_f \mu_m}{\rho_m \mu_f} \frac{L_f}{L_m}$$

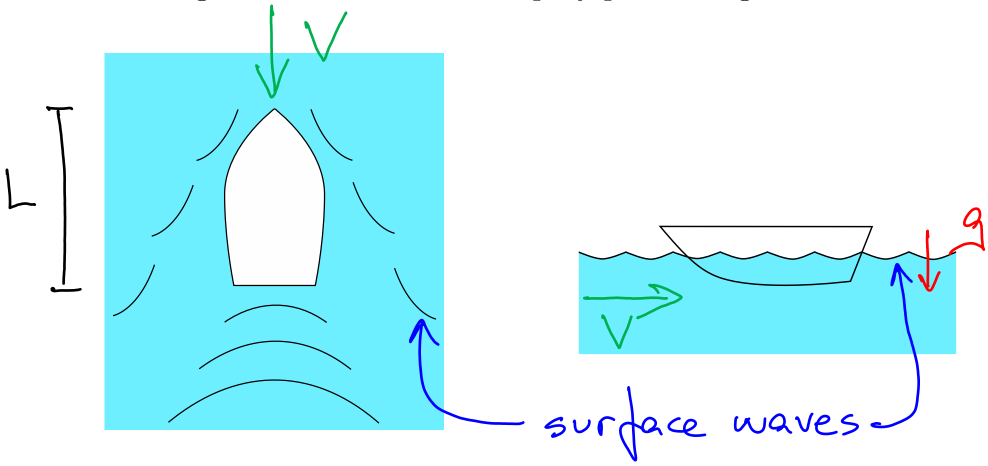
ratio of length-scales (!)

can play with these to match M and Re (cryogenic wind tunnels)

This is an over-constrained problem: it will not be possible to match both M and Re without manipulating the density and/or the viscosity of the fluid. This can be done for instance by pressurising the wind tunnel, which increases the density of air, or by using cryogenic temperatures, which both increases the density and decreases the viscosity of air, but such solutions are expensive. However, we know from experience that in many cases the Reynolds number has little influence once it is $Re \gtrsim 10^6$. For this reason, wind-tunnel tests typically match the Mach number and let the Reynolds number float, ensuring only that it does not drop into the region where it could have influence. This is called *partial* or *incomplete similarity*.

5.9 Example - Drag force on a ship

Behind a ship there is a wave pattern that propagates energy away from the ship as well as the conventional wake associated with a body. These are *surface* waves. The restoring force is gravity, so it must be included in the problem. How would we work out the drag force on a full-scale ship by performing model tests?



Step 1 - Dependent and independent variables

Dependent	Independent
drag	ρ, V, L, μ, g $\longrightarrow$ (6 total)

Step 2 - Number of dimensions

6 − 3 = 3 dimensionless variables

Step 3 - Create the dimensionless numbers (look in Thermofluids data book)

Dependent	Independent
$C_D = \dfrac{\text{drag}}{\frac{1}{2}\rho V^2 L^2}$	$Re = \dfrac{\rho V L}{\mu}$, $\dfrac{gL}{V^2}$ — $Fr = \dfrac{V}{\sqrt{gL}}$

Step 4 - Work out conditions for complete similarity, assuming that V_m is unrestricted:

$$\frac{V_m^2}{gL_m} = \frac{V_f^2}{gL_f} \qquad \text{and} \qquad \frac{\rho_m V_m L_m}{\mu_m} = \frac{\rho_f V_f L_f}{\mu_f} \qquad \Rightarrow \frac{\mu_m \rho_f}{\mu_f \rho_m} = \left(\frac{L_m}{L_f}\right)^{3/2}$$

Thus if the model is 1:20 scale, the ratios of the kinematic viscosities, μ/ρ must be 1:89. There are no safe, cheap fluids with such a small viscosity so we cannot force complete similarity.

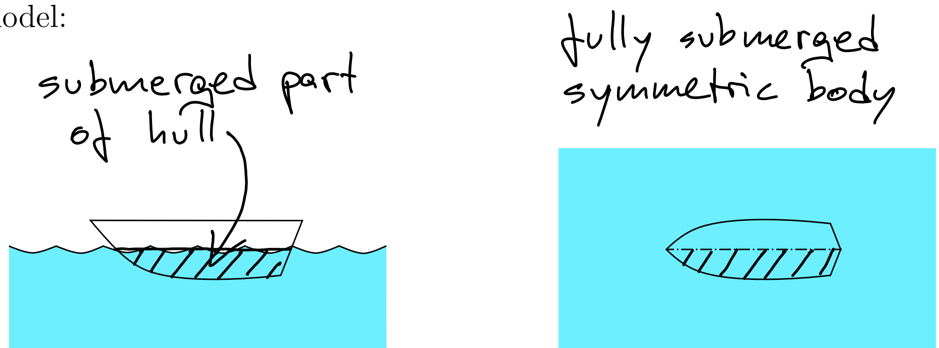
However, we know that C_D is some function of Re and Fr only. What can we say about the nature of this function, using physical reasoning? Are the Re effects likely to be independent of the Fr effects or not?

$$C_D = f(Re, Fr)$$

It takes the sea around a day to become calm after a storm -through the action of viscous forces- while the typical period of a wave is around few seconds. Therefore, viscosity can only have a very weak effect on wave motion. Furthermore, gravity can have little effect on the usual wake drag associated with the body, that is, the form and friction drag. Therefore, we can treat the wave drag and the form/friction drag as independent and additive:

$$C_{D_{Total}} = C_{D_{wave}}(Fr) + C_{D_{form/fric}}(Re)$$

We can then perform two separate experiments. First, we test the model at the correct Froude number, and measure the total drag. Then, we measure the form and friction drag at that Froude number by testing a completely submerged reflected model:



$$\left(C_{D_{wave}} + C_{D_{form/fric}}\right) - C_{D_{form/fric}} = C_{D_{wave}}$$

We subtract one from the other to obtain the Froude number drag coefficient, $C_{D,\text{wave}}(Fr)$. For large ships it can be hard to test at the correct Reynolds number, because very large velocities are required. However, we will see in Lecture 10 (External Flow) how to estimate $C_{D,\text{form+fric}}$ at large Re.

Engineering Tripos 1B

Paper 4

Fluid Mechanics

Lecture 6 - Pipe Flow

Watch the video on this experiment in Moodle (6.3 Video 1)

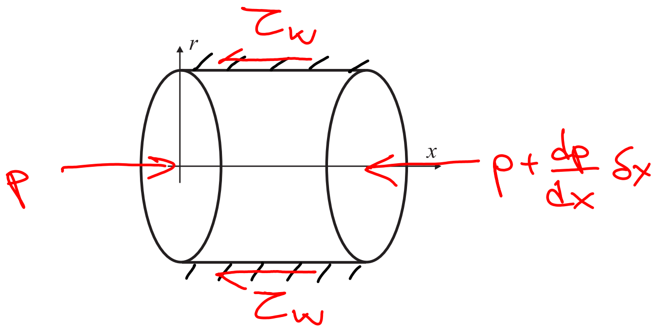
- Friction and pressure loss - Dimensional analysis
- Laminar flow in a circular pipe
- Turbulent flow in a circular pipe
- Turbulence, mixing and friction
- Roughness

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

6.1 Friction and pressure loss - Dimensional analysis

In Lecture 5 we have seen how we can use dimensional analysis to reduce the number of variables that any physical problem depends on to the minimum set of (dimensionless) parameters. Let us now apply this to the flow inside a pipe. We are mainly interested in the pressure drop –or loss– along the pipe, $\mathrm{d}p/\mathrm{d}x$. This is because, as we will see in Lecture 7, the pressure loss determines how much flow we can move through the pipe per unit time, and how much power will be required to do so. When the bulk velocity V, defined as the volumetric flowrate divided by the cross-sectional area, is constant in time, the flow experiences zero mean acceleration, and the pressure loss can be directly related to the friction force done by the walls:

$$V = \text{bulk velocity} = \frac{\text{flow rate}}{\text{cross-sectional area}} = \frac{Q}{\pi R^2} = \frac{\int_0^R v 2\pi r \, dr}{\pi R^2} = \text{constant in } x$$



$$\Rightarrow -2\pi R \tau_w - \pi R^2 \frac{dp}{dx} = 0$$

$$\Rightarrow \tau_w = -\frac{\pi R^2}{2\pi R}\frac{\mathrm{d}p}{\mathrm{d}x}$$

$$\Rightarrow \tau_w = -\frac{R}{2}\frac{\mathrm{d}p}{\mathrm{d}x}$$

The wall shear stress τ_w is a force per unit area and has dimensions Nm^{-2}, just like the pressure.

Let us then apply dimensional analysis to the problem of the friction stress at the pipe walls:

Dependent variable	Independent variables
τ_w	ρ, V, D, μ

[M, L, T]

$5 - 3 = 2$

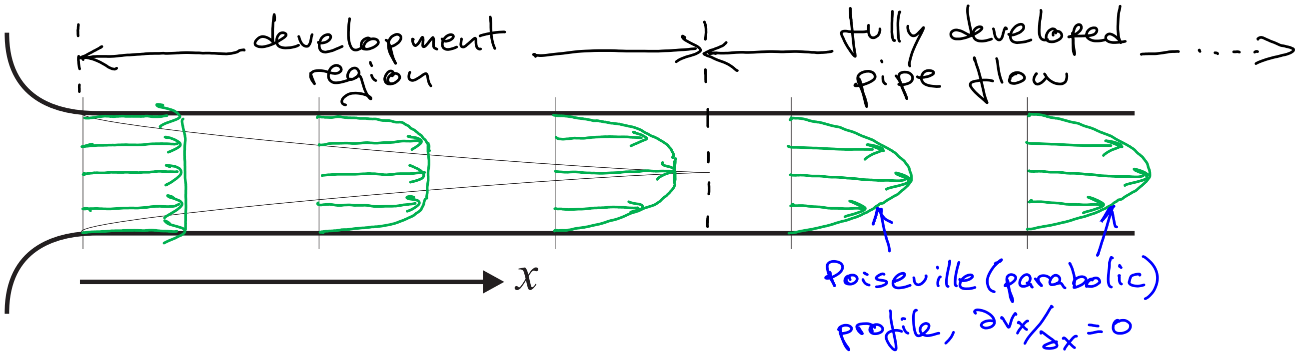
dimensionless variables

We can therefore construct a dimensionless friction that will depend only on a single dimensionless parameter (for which the Reynolds number is a good choice). We can nondimensionalise τ with ρV^2, but it is typical to use the dynamic pressure $\rho V^2/2$ instead. The resulting dimensionless number is the friction coefficient, c_f,

$$\underbrace{c_f}_{\text{dimensionless}} = \frac{\tau_w}{\frac{1}{2}\rho V^2} = -\frac{R}{\rho V^2}\frac{dp}{dx} = f\Big(\underbrace{\frac{\rho V D}{\mu}}_{\text{dimensionless, Re}}\Big)$$

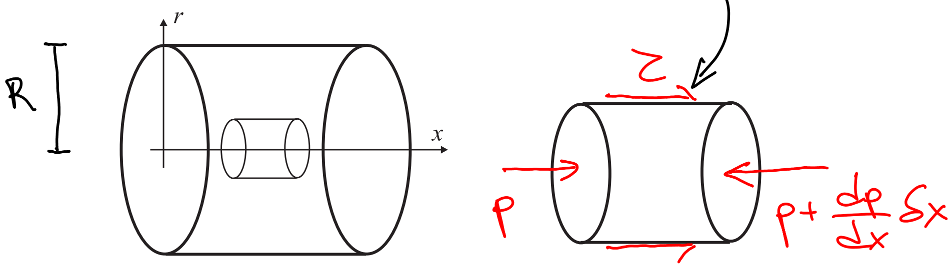
6.2 Laminar flow in a circular pipe

The friction coefficient can be calculated exactly when a viscous fluid is forced slowly down a pipe. The flow takes some length from the entrance to develop fully (we will see in Lecture 8, when we discuss boundary layers, that the development length is proportional to R^2), but from there on it becomes independent of x and adopts the familiar Poiseuille velocity profile.



The worked example of Lecture 4 (section 4.9) shows one way to derive the velocity profile. An easier way to derive it is to balance the forces on cylindrical elements centred on the centreline:

($\tau = \mu \, dv_x/dr$ would be rightwards if $dv_x/dr > 0$)



$$2\pi r \tau - \pi r^2 \frac{dp}{dx} = 0$$

$$\Rightarrow \tau = \frac{r}{2}\frac{dp}{dx}$$

$$\Rightarrow \mu \frac{dv_x}{dr} = \frac{r}{2}\frac{dp}{dx}$$

This gives us the velocity gradient in terms of the pressure gradient. We need to work out the velocity profile by integrating the expression and applying the no slip boundary condition:

$$\int_0^{v_x} dv_x = \frac{1}{2\mu}\frac{dp}{dx}\int_R^r r\,dr$$

$$v_x = \frac{\mathrm{d}p}{\mathrm{d}x}\left[\frac{r^2}{4\mu}\right]_R^r$$

$$v_x = \underbrace{-\frac{\mathrm{d}p}{\mathrm{d}x}}_{\frac{dp}{dx} < 0 \text{ is favourable}}\underbrace{\left(\frac{R^2 - r^2}{4\mu}\right)}_{\geq 0}$$

$\Rightarrow v_x \geq 0$

Now we need to evaluate the average velocity in the pipe, V, in terms of the pressure drop. The total flowrate divided by the cross-sectional area is

$$
\begin{aligned}
V &= \overbrace{\frac{1}{\pi R^2}}^{\text{cross-section area}} \overbrace{\int_0^R v_x(r) 2\pi r dr}^{\text{total flow rate, } Q} = \frac{1}{\pi R^2} \int_0^R -\frac{\mathrm{d}p}{\mathrm{d}x} \left(\frac{R^2 - r^2}{4\mu} \right) 2\pi r dr \\
\Rightarrow V &= -\frac{1}{2\mu R^2} \frac{\mathrm{d}p}{\mathrm{d}x} \left[\frac{R^2 r^2}{2} - \frac{r^4}{4} \right]_0^R \\
\Rightarrow V &= -\frac{R^2}{8\mu} \frac{\mathrm{d}p}{\mathrm{d}x} \\
\Rightarrow \frac{dp}{dx} &= -\frac{8\mu}{R^2} V
\end{aligned}
$$

We have already determined an expression for the friction coefficient, c_f, in terms of the pressure drop. Now we can express it in terms of the average flowrate by substituting in the above expression,

$$
\begin{aligned}
c_f &= -\frac{R}{\rho V^2} \frac{\mathrm{d}p}{\mathrm{d}x} = \frac{R}{\rho V^2} \frac{8\mu V}{R^2} \\
&= \frac{8\mu}{\rho V R}
\end{aligned}
\qquad \Rightarrow c_f = c_f = f(Re)
$$

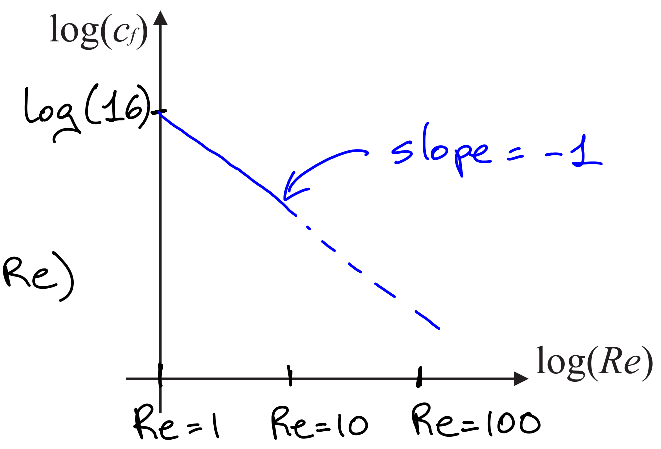
and if we define the Reynolds number using the diameter D, as is most frequent in hydraulics, we have

$$
Re = \frac{\rho V \overbrace{D}^{2R}}{\mu} \Rightarrow \boxed{c_f = \frac{16}{Re}}
$$

So for a laminar pipe flow the friction coefficient only depends on the Reynolds number, as shown by dimensional analysis. The results from the lab experiment actually match the theoretical result quite well in the laminar flow region.

$$
\log \left\{ c_f = \frac{16}{Re} \right\}
$$

$$
\log(c_f) = \log(16) - \log(Re)
$$



On a side note, let us recall the order-of-magnitude analysis that we did in Lecture 5, section 5.5:

$$\text{(steady)} \quad \cancel{\rho\frac{\partial \mathbf{v}}{\partial t}} + \rho\,\mathbf{v}\cdot\nabla\mathbf{v} = -\nabla p + \mu\nabla^2\mathbf{v}$$

$$\sim \rho\frac{V^2}{D} \qquad \sim \frac{dp}{dx} \qquad \sim \mu\frac{V}{D^2}$$

advective vs. viscous: $\sim \rho V^2$ $\qquad \sim \mu \frac{V}{D}$

~ dynamic pressure $\qquad$ ~ viscous stress

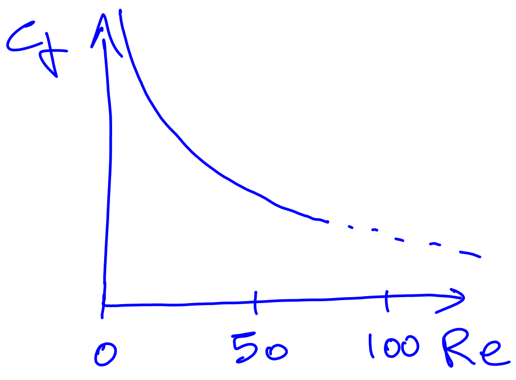
If the advective and the viscous terms are of very different order of magnitude, $\mathrm{d}p/\mathrm{d}x$ will necessarily be of the order of magnitude of whichever is larger. This is what happens at very large and very small Reynolds number –remember that the Reynolds number gives a measure of the relative importance of advective vs. viscous effects.

$$\frac{\text{"advective"}}{\text{"viscous"}} = \frac{\rho V^2}{\mu V/D} = \frac{\rho V D}{\mu} = Re$$

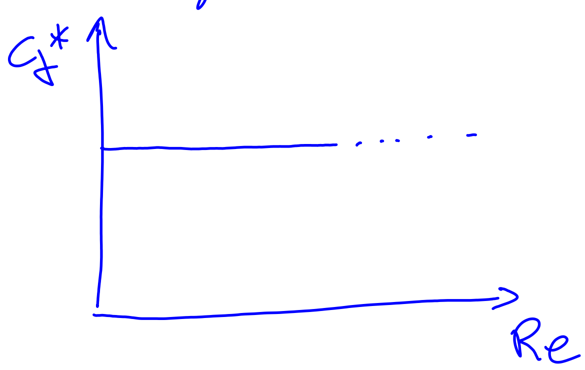
When we defined c_f, we did so by forming a dimensionless group for the wall shear stress. For that, we used the dynamic pressure, $1/2\rho V^2$, but we might as well have used the order-of-magnitude value of the viscous stress, $\mu V/D$. This would have been a more convenient choice at low Reynolds number, when viscous terms dominate. Nevertheless, we are interested in always using the same definition of c_f, and we will be more interested in what happens at large Re.

$$c_f = \frac{\tau_w}{\frac{1}{2}\rho V^2} = \frac{8\mu}{\rho V R} = \; \ldots \; = \frac{16}{Re}$$

$$c_f^* = \frac{\tau_w}{\mu \frac{V}{D}} = \frac{\tau_w}{\frac{1}{2}\rho V^2}\,\frac{\frac{1}{2}\rho V^2}{\mu V/D} = c_f\,\frac{\rho V D}{2\mu} = \frac{c_f}{2}Re = 8\,!$$



(varies with Re)

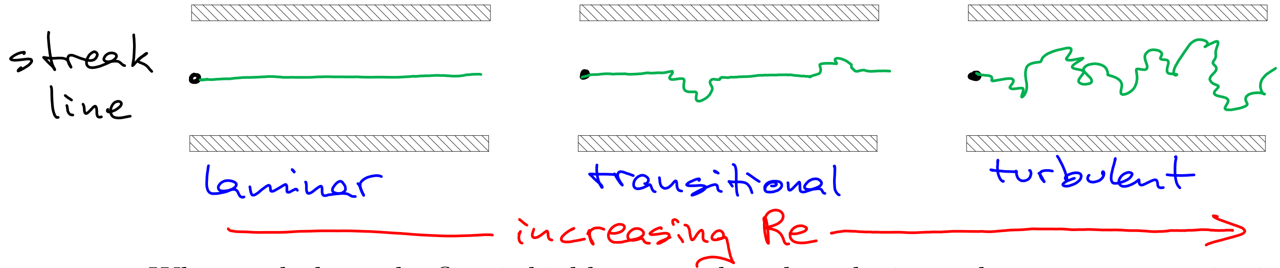


(constant with Re)

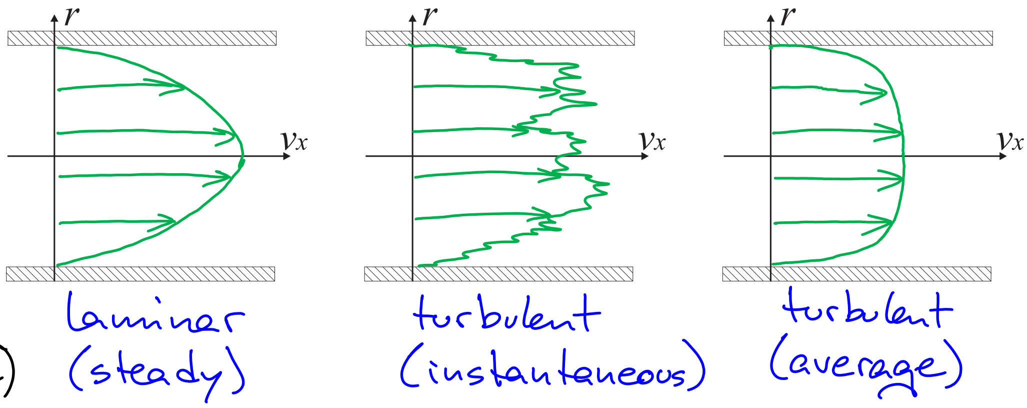
Now revisit Q1 in Examples Paper 4, and compare scaling F with $\frac{1}{2}\rho V_1^2 h_1$ (using the dynamic pressure) vs. $\rho g h_1^2$ (using the hydrostatic pressure)

6.3 Turbulence, mixing and friction

If the flow were to remain laminar, the friction coefficient would always decrease with increasing Reynolds number. However, as the Reynolds number increases (in a given pipe and for a given fluid, by increasing V), the flow is not viscous enough to dissipate all the energy put into it merely by viscous friction, and breaks down into smaller and smaller eddies, down to sizes where viscosity can eventually act. The flow is then highly fluctuating and chaotic, and we say that it has become turbulent.



When turbulent, the flow is highly unsteady - the velocity and pressure at a point in space vary with time. However, the *time-averaged* quantities do have steady values in pipe flow, so we can use these quantities to define the Reynolds number and the friction coefficient.



(Watch 6.3 Video 6 on Moodle)

v_x profile is:

- flatter near centre
- steeper near walls

⇓

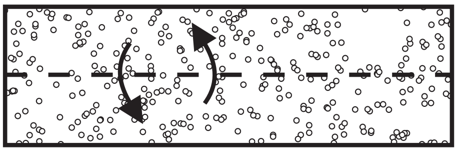
higher $\tau_w = \mu \left|\frac{dv_x}{dr}\right|_R$

In a laminar flow, molecular diffusion is the only transport process between layers of fluid. Laminar flows are very hard to mix - they need to be *folded* rather than *stirred.* In a turbulent flow, packets of fluid move between layers of fluid in turbulent eddies. This occurs on a much larger scale than molecular diffusion so the mixing rate is greatly increased.

(Watch 6.3 Video 2 and 6.3 Video 3 on Moodle)

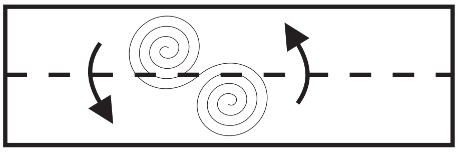
Consequently, the rate of momentum transport in a turbulent flow is much greater than that in a laminar flow. There is a higher rate of momentum transfer from the fluid to the pipe walls, hence a higher shear stress and a greater pressure drop.

In Lecture 1 we saw that the transport of momentum due to molecular diffusion is modelled by viscosity. In a turbulent flow however, whole eddies move between layers, greatly increasing the rate of transport of momentum. We could *model* the effect of turbulence by increasing the viscosity by some amount. This added viscosity is called the *eddy viscosity*.



laminar: molecular mixing only

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



turbulent: eddy diffusion, much larger scale

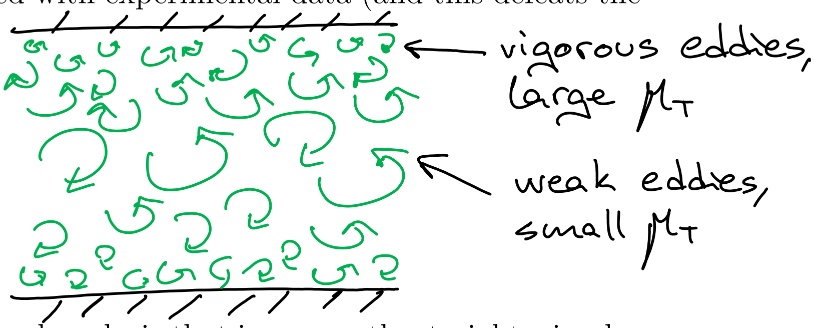
$$\tau = \mu_T \frac{dv_x}{dy}$$

$\mu_T = \mu +$ 'eddy viscosity' $(\gg \mu)$

The 'eddy viscosity' model illustrates reasonably well the enhanced transport in turbulent flows. Unfortunately, it is a rather crude model, and has only limited usefulness. The value of the eddy viscosity depends on eddy size and intensity, and it is difficult to find a universal model. The existing models typically have several free constants, which need to be fitted with experimental data (and this defeats the purpose of developing a model!).

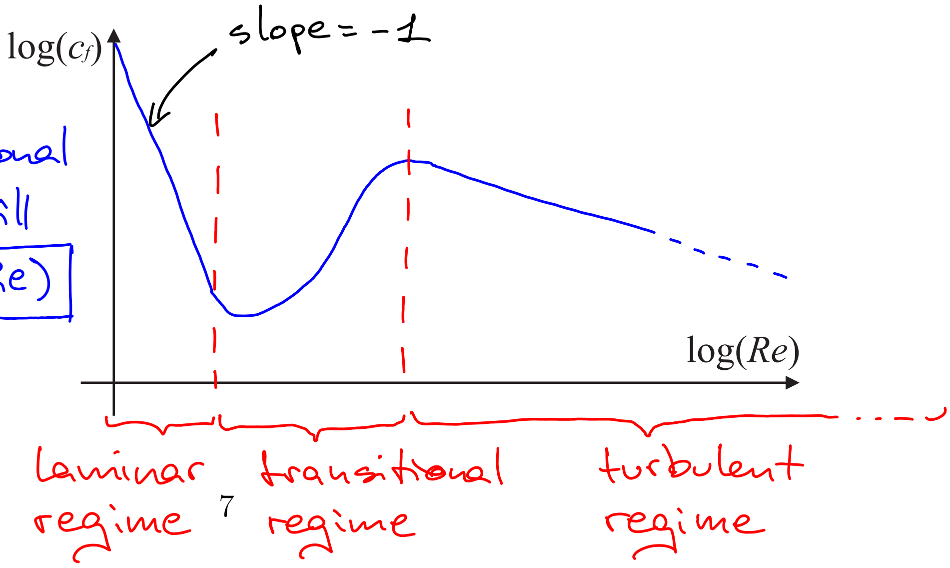
(Watch 6.3 Video 5 on Moodle)

- Eddy size and intensity vary throughout the flow
- No universal model for μ_T !



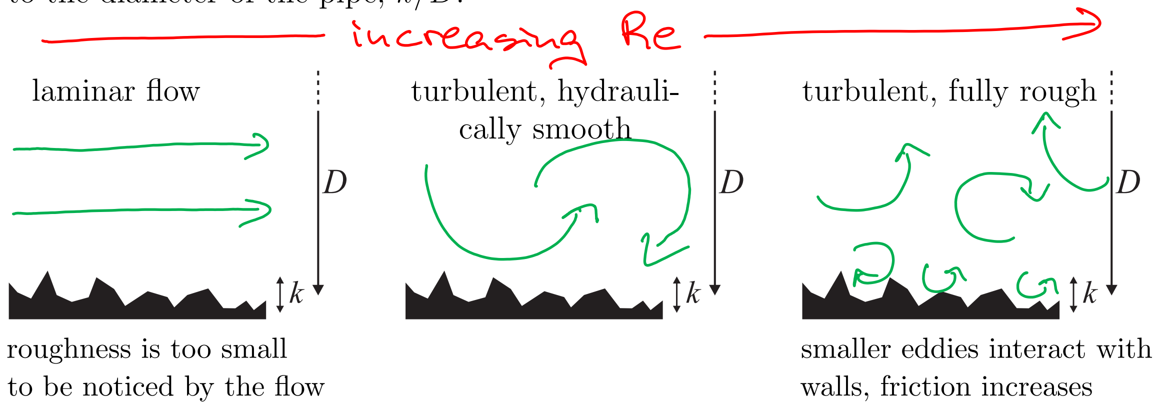
However, we know from dimensional analysis that in a smooth, straight, circular pipe the friction coefficient *only* depends on the Reynolds number. We can therefore measure that dependence experimentally, resting assured that our measures will be valid for any smooth, straight, circular pipe.

From dimensional analysis, we still have $\boxed{c_f = f(Re)}$



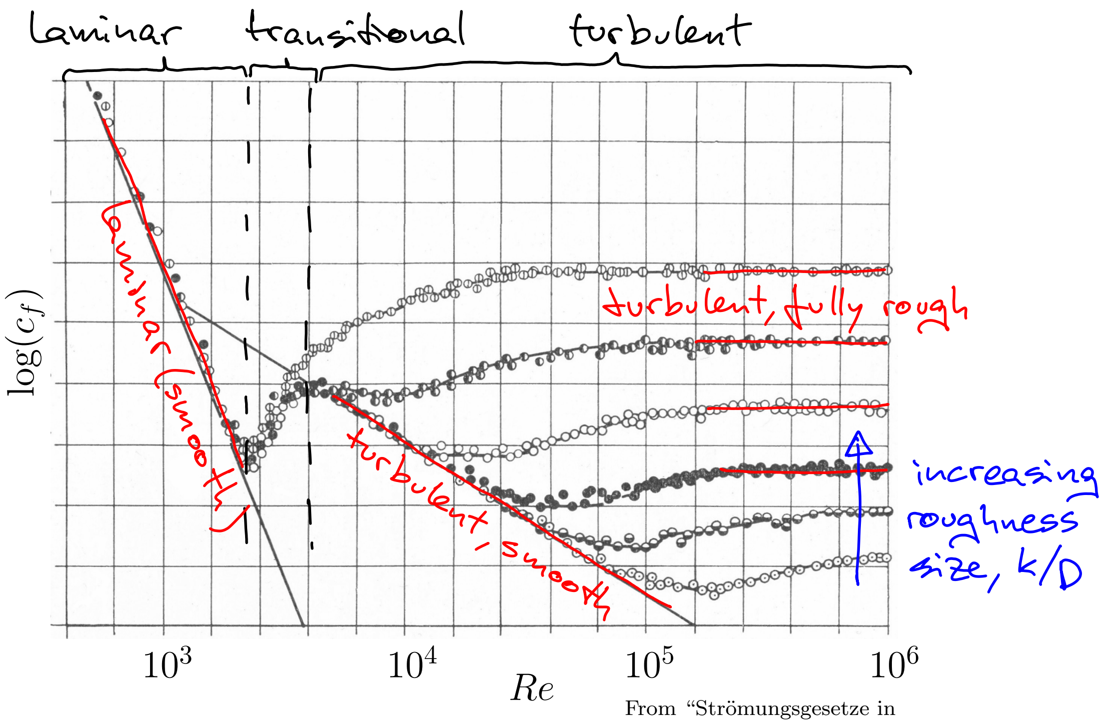
6.4 Roughness

Roughness increases friction, so we typically try to eliminate it from pipes. However, no matter how smooth a pipe is, it will always have a certain roughness. The roughness of a pipe can be characterized through the ratio of the average bump size to the diameter of the pipe, k/D.



When $k/D \ll 1$, a laminar flow does not notice that the pipe walls are rough. Even turbulent flows at moderately low Reynolds numbers do not notice it. This occurs because the flow breaks down into eddies which are still too large to notice the detail of the surface roughness. The flow is then called *hydraulically smooth*. However, as the Reynolds number increases, eventually the flow breaks down into eddies small enough to interact with the rough bumps. The friction then increases, compared to a hydraulically smooth flow. Eventually, the smallest eddies, for which viscosity is important, are so small that viscosity becomes irrelevant at the length scale of the bumps k. The friction becomes then independent of the viscosity, and thus of the Reynolds number, as it is caused entirely by the pressure drag from the roughness bumps. The turbulent flow is then called *fully rough*.

From dimensional analysis, $c_f = f(Re, k/D)$



From "Strömungsgesetze in rauhen Rohren" J. Nikuradse (1933)

Engineering Tripos 1B

Paper 4

Fluid Mechanics

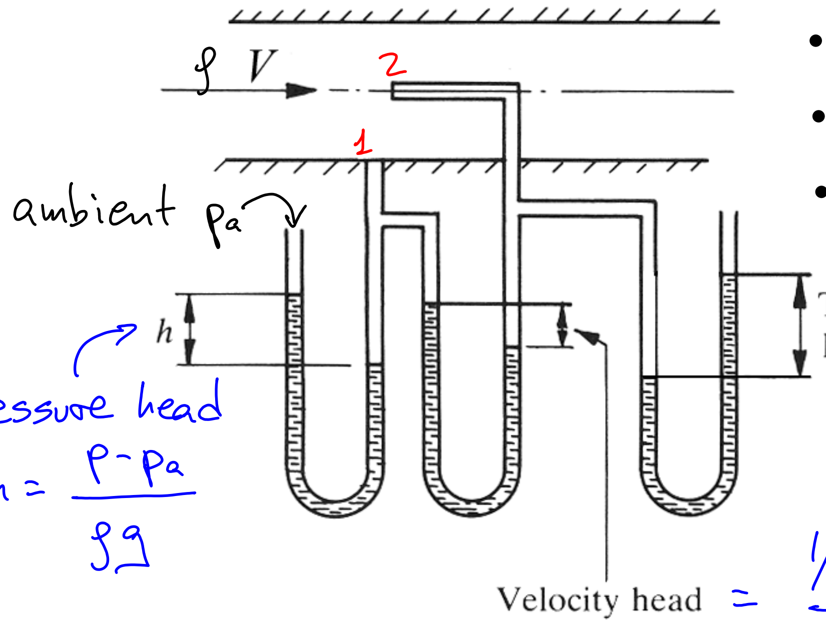
Lecture 7 - Network analysis

- Static, dynamic, stagnation, and total pressure
- Loss of total pressure along a pipe
- Loss of total pressure across different network components
- Pumps, turbines and mechanical work
- Worked example

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

7.1 Static pressure, stagnation pressure, and total pressure

In Lecture 3 we briefly visited the concepts of stagnation pressure and total pressure when discussing Bernoulli's equation. We saw how, when pressure is the only source of forces acting on the fluid, the quantity $p + \rho V^2/2$ is conserved along streamlines. If gravity is also relevant, then it is the quantity $p + \rho g h + \rho V^2/2$ that remains constant. We can therefore see that both the pressure and the gravitational potential energy have the capacity to be transformed into flow kinetic energy, and can thus be viewed as forms of stored energy for the flow. In this lecture, we will analyse the loss (or gain) of energy in hydraulics networks, to find out the power required by the system or the power that can be extracted from it.



In the duct:

- Static pressure $p = p_1$
- Dynamic pressure $\frac{1}{2}\rho V^2$
- Stagnation pressure $p + \frac{1}{2}\rho V^2 = p_2$

Total head $= \dfrac{p + \frac{1}{2}\rho V^2 - p_a}{\rho g}$

Pressure head $h = \dfrac{p - p_a}{\rho g}$

Velocity head $= \dfrac{\frac{1}{2}\rho V^2}{\rho g}$

Let us start by defining static, stagnation and total pressure. The *static pressure*, p, is the actual pressure at any given point in the flow. It is the pressure that we would measure with a sensor that did not alter the flow velocity, e.g. at point A on the diagram below. We define the *stagnation pressure*, p_0, as the quantity that would be conserved for an inviscid flow in the absence of gravity effects,

$$p_0 = p + \frac{1}{2}\rho V^2.$$

It is the pressure we would measure if we managed to bring the flow to rest without mechanical losses, e.g. at point B on the pitot tube, which is a stagnation point. We will refer to the quantity $\rho V^2/2$ as the *dynamic pressure*, which is the difference between the stagnation and static ones. Finally, we define the *total pressure*,

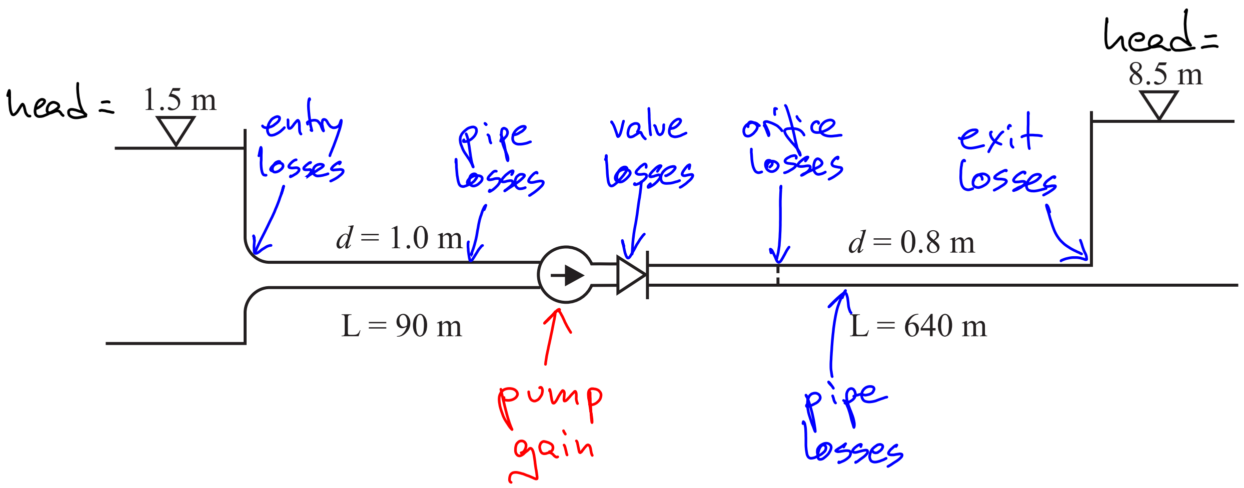
total pressure: $$p_T = p + \rho g h + \frac{1}{2}\rho V^2,$$

as the quantity that would be conserved when the work done by gravity is relevant, that is, when changes in height in the fluid are important.

In real life, however, all fluids have certain viscosity, so losses are bound to occur and the total pressure is not conserved. We will devote most of Lecture 7 to estimating the losses in different parts of a network, so that we can find how much energy per unit volume, or rather, total pressure, the fluid has at the end of its course.

Hydraulics engineers often talk about *heads* instead of pressures, as any pressure difference can be transformed into a height difference in a static column of liquid, as is done in the diagram on page 2. The head is simply the corresponding 'pressure' magnitude divided by ρg,

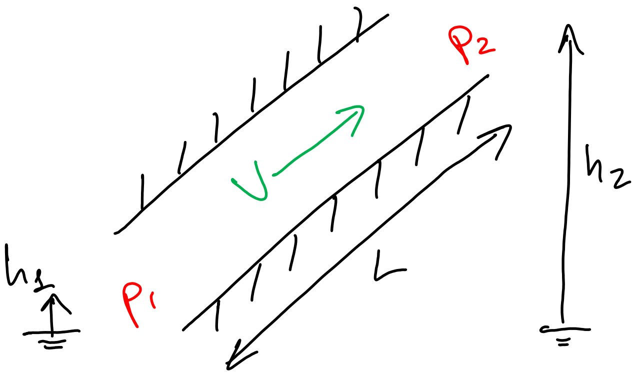
$$\text{total head} = \frac{p_T}{\rho g} = \frac{p + \rho g h + \frac{1}{2}\rho V^2}{\rho g}$$



Consider for instance the network sketched above. The flow starts the circuit with a head of 1.5 m. As it passes through the different network elements, it will experience different mechanical energy losses that we will measure in terms of losses in total pressure. The flow will also undergo a gain of total pressure through the pump, so that there is a net gain that results in a higher final head of 8.5m.

In long pipes, h may change substantially from one end to the other, so the factor $\rho g h$ in p_T needs to be taken into account. Along short components, like bends, valves, etc, the change in height can often be neglected and the change in total pressure is then just the change in stagnation pressure, $p + \rho V^2/2$. We will neglect height variations in Sections 7.3 to 7.5, but in any event they could be included in the analysis by simply substituting static pressures, p, for the corresponding values of $p + \rho g h$, as we saw in Lectures 3 and 4.

7.2 Loss of total pressure along a pipe

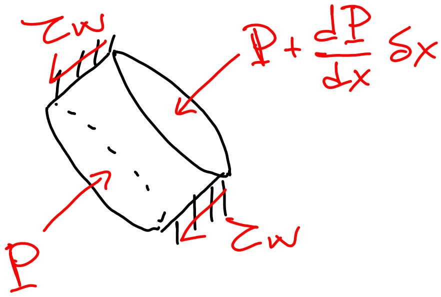


$$p_{1T} = p_1 + \rho g h_1 + \frac{1}{2}\rho V^2$$

$$p_{2T} = \underbrace{p_2 + \rho g h_2}_{P = p + \rho g h} + \frac{1}{2}\rho V^2$$

In Lecture 6, section 1, we derived an expression for the pressure drop along a pipe in terms of the friction coefficient $c_f = f(Re)$,

$$\frac{dP}{dx} = -\frac{\rho V^2}{R} c_f,$$



or, in terms of the pipe diameter, D,

$$\frac{dP}{dx} = -\frac{4}{D} c_f \left(\frac{1}{2}\rho V^2\right).$$

There is an alternative definition of the friction coefficient, which we will denote f, and which is simply $4c_f$. In this course we call f the *friction factor* although in some books it too is called the friction coefficient.

$$f = 4c_f$$

If the cross-sectional area is uniform along the pipe, R, V and $Re = \rho V D/\mu$ are constant, so we have

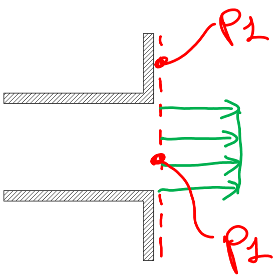
$$\frac{P_2 - P_1}{L} = \frac{(p + \rho g h)_2 - (p + \rho g h)_1}{L} = -\frac{4}{D} c_f \frac{1}{2}\rho V^2$$

Knowing that the bulk velocity is uniform, we can use this difference in static pressures to obtain the change in total pressure,

$$\boxed{p_{T2} - p_{T1}} = \left(p_2 + \rho g h_2 + \cancel{\frac{1}{2}\rho V_2^2}\right) - \left(p_1 + \rho g h_1 + \cancel{\frac{1}{2}\rho V_1^2}\right)$$
$$= p_2 - p_1 + \rho g\,(h_2 - h_1) =$$
$$= \boxed{-\frac{1}{2}\rho V^2 \frac{4L}{D} c_f}$$

7.3 Loss of total pressure at a pipe discharge

When a pipe of constant section ends in a sudden expansion, for instance in a reservoir or directly into open air, the out-coming flow forms a jet. It is generally safe to assume that the jet streamlines are straight, which implies that there are no normal pressure gradients, so the *static* pressure in the jet and in the quiescent fluid surrounding it is the same. Furthermore, if the surrounding fluid is air and the discharging one is much denser, like water, the friction between the two, and the associated energy loss, can typically be neglected.



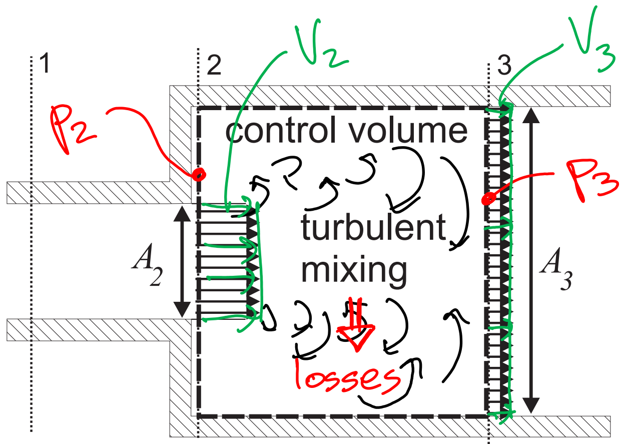
- Parallel straight streamlines

$\Rightarrow \partial p / \partial n = 0$

$\Rightarrow$ pressure is uniform in the section

- If jet is water into air $\Rightarrow$ losses negligible

When the discharge is into a the same fluid, viscous friction will not be negligible and soon turbulent mixing sets in, causing a drop in total pressure. When the discharge is simply a sudden expansion into a new, wider section of the pipe, it is difficult to know the details of the flow in the mixing region. Because of the turbulent dissipation, we cannot apply Bernoulli's equation, but we can establish a balance between the discharge section and another section further downstream, once the flow is fully developed again:



$\dot{m} = \rho A_2 V_2$

Continuity equation:

$$V_2 A_2 = V_3 A_3 \;\Rightarrow\; \frac{V_3}{V_2} = \frac{A_2}{A_3}$$

Steady flow momentum equation:

$$p_2 A_3 + \dot{m} V_2 = p_3 A_3 + \dot{m} V_3$$

Re-arranging the momentum equation we have:

$$p_2 - p_3 = \frac{\dot{m}}{A_3}(V_3 - V_2) = \frac{\rho A_2 V_2}{A_3}(V_3 - V_2) = \rho V_2^2 \left(\frac{A_2 V_3}{A_3 V_2} - \frac{A_2}{A_3} \right)$$

Substituting the continuity equation into this expression gives:

$$p_2 - p_3 = \rho V_2^2 \left(\left[\frac{A_2}{A_3} \right]^2 - \frac{A_2}{A_3} \right)$$

If changes in $\rho g h$ are negligible, the drop in total pressure is

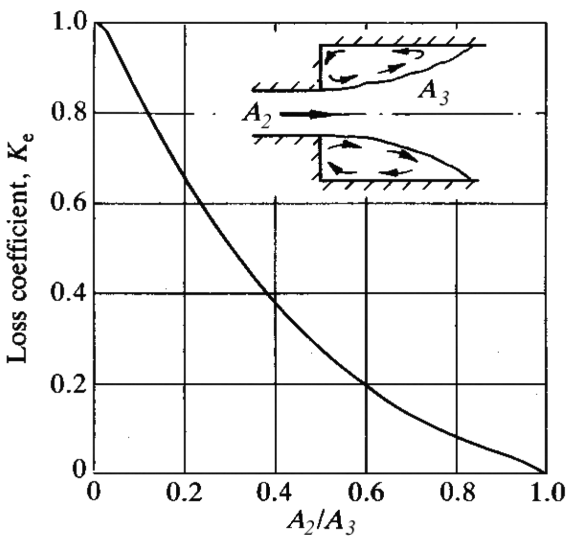
$$\begin{aligned} p_{T2} - p_{T3} =& p_2 + \frac{1}{2}\rho V_2^2 - p_3 - \frac{1}{2}\rho V_3^2 \\ =& p_2 - p_3 + \frac{1}{2}\rho (V_2^2 - V_3^2) \\ =& p_2 - p_3 + \frac{1}{2}\rho V_2^2 \left(1 - \left[\frac{V_3}{V_2} \right]^2 \right) \end{aligned}$$

$= A_2/A_3$

Substituting we finally have

$$\begin{aligned} p_{T2} - p_{T3} =& \rho V_2^2 \left(\left[\frac{A_2}{A_3} \right]^2 - \frac{A_2}{A_3} \right) + \frac{1}{2}\rho V_2^2 \left(1 - \left[\frac{A_2}{A_3} \right]^2 \right) \\ =& \frac{1}{2}\rho V_2^2 \left(1 - 2\frac{A_2}{A_3} + \left[\frac{A_2}{A_3} \right]^2 \right) \\ =& \frac{1}{2}\rho V_2^2 \left(1 - \frac{A_2}{A_3} \right)^2 = \frac{1}{2}\rho V_2^2 K, \end{aligned}$$

where K is the *loss coefficient* for the sudden expansion. We can compare this value of K with the experimental values compiled below:



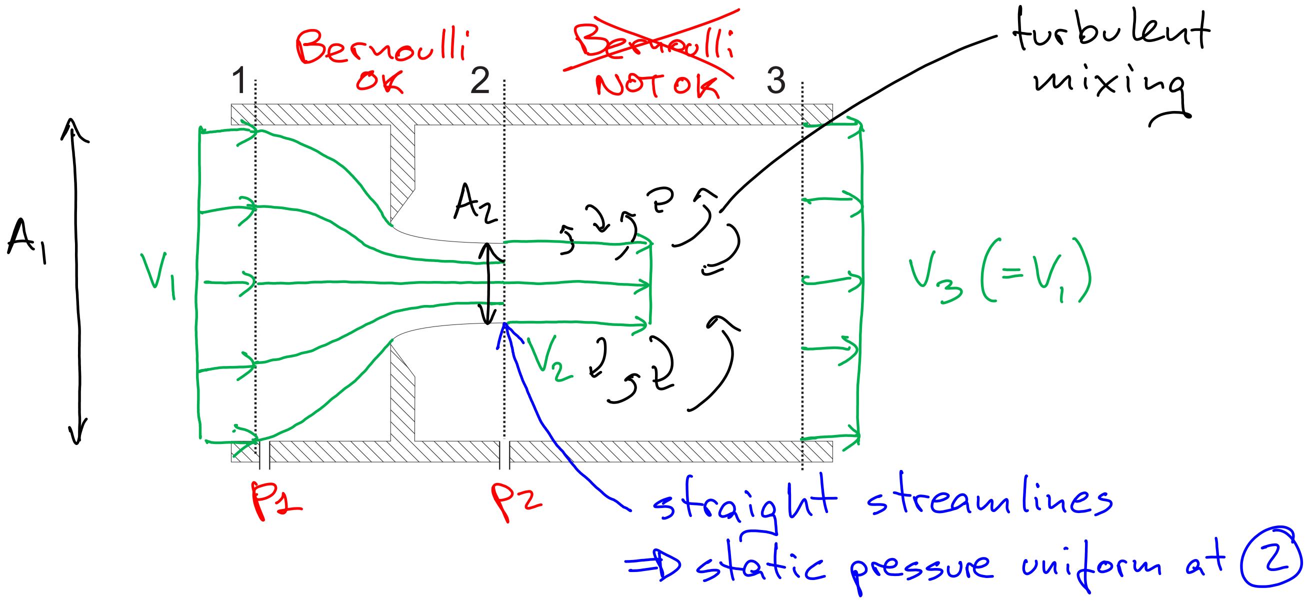
When a pipe discharges into a reservoir the above model is still valid, but then A_3 tends to infinity (or $V_3 \approx 0$) and the loss coefficient is $K = 1$.

If from A_2 to A_3 the variation in height was not negligible, we would need to repeat the analysis accounting for it. This can be done by simply replacing p_2 with $p_2 + \rho g h_2$ and p_3 with $p_3 + \rho g h_3$, and the resulting loss of total pressure would still be

$$\boxed{p_{T2} - p_{T3} = \frac{1}{2}\rho V_2^2 K.}$$

7.4 Loss of total pressure across an orifice plate

Orifice plates are used to measure the flowrate through a pipe. The flowrate is calculated from the static pressure drop across the plate. The pressure drop depends on the size of orifice, the sharpness of the edges and where the pressure tappings are placed relative to the plate. Orifice plates are calibrated experimentally but here we use a simple model to estimate the pressure drop. We assume that the velocity is, on average, uniform and steady across section 1, section 3 and the central jet in section 2.



Section 2 is chosen so that the jet streamlines are locally straight. This implies that there are no normal pressure gradients, so the *static* pressure is the same across all of section 2. If we assume that there are no viscous losses between section 1 and section 2, then Bernoulli can be applied along each streamline in this region.

$$p_1 + \frac{1}{2}\rho V_1^2 = p_2 + \frac{1}{2}\rho V_2^2$$

$$\Rightarrow \qquad p_1 - p_2 = \frac{1}{2}\rho V_1^2 \left(\frac{V_2^2}{V_1^2} - 1\right)$$

If we knew A_2/A_1 we could calculate V_2/V_1 from conservation of mass between sections 1 and 2:

$$\rho A_1 V_1 = \rho A_2 V_2$$

$$\Rightarrow \qquad p_1 - p_2 = \frac{1}{2}\rho V_1^2 \left(\left(\frac{A_1}{A_2}\right)^2 - 1\right)$$

Measure p_1, p_2

$\Downarrow$

Obtain V_1

Let as now calculate the loss of total pressure across the whole component. Between sections 2 and 3 the jet mixes turbulently and it behaves just like the flow between sections 2 and 3 of Section 7.3, but in addition now $V_3 = V_1$ because of continuity. The resulting overall drop in static pressure is

Losses:

$$p_1 - p_3 = (p_1 - p_2) + (p_2 - p_3) = \frac{1}{2}\rho V_1^2 \left(\frac{A_1}{A_2} - 1\right)^2 = \frac{1}{2}\rho V_1^2 K,$$

and the total pressure drop (neglecting height variations) is the same, since $V_3 = V_1$,

$$p_{T1} - p_{T3} = p_1 + \cancel{\frac{1}{2}\rho V_1^2} - p_3 - \cancel{\frac{1}{2}\rho V_3^2} = p_1 - p_3 = \frac{1}{2}\rho V_1^2 K$$

.

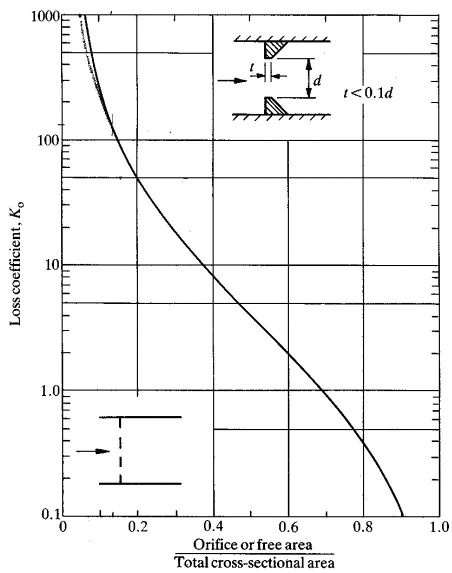
The difficulty to determine the loss coefficient, K, lies in determining the position and value of A_2, which is very difficult experimentally. However, we can apply dimensional analysis as in Lecture 5, section 6:

Dependent variable	Independent variables
Δp	ρ, V, D, μ, d
$K = \dfrac{\Delta p}{\frac{1}{2}\rho V^2}$	$Re = \dfrac{\rho V D}{\mu}, \dfrac{d}{D}$

$[M, L, T]$

$6 - 3 = 3$

At the very large Reynolds numbers of pipes, typically larger than $Re \approx 10^6$, the problem becomes independent of the actual value of Re, and K is then only a function of the geometry of the orifice. Its values are often tabulated or plotted for different orifice shapes:



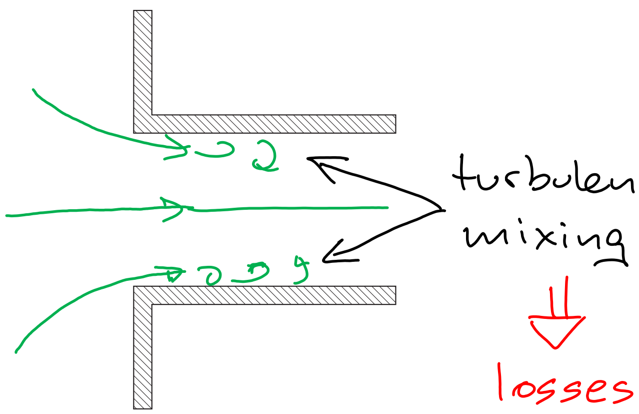
in principle, $K = f(Re, d/D)$, but for $Re \gtrsim 10^5$ K becomes Re-independent

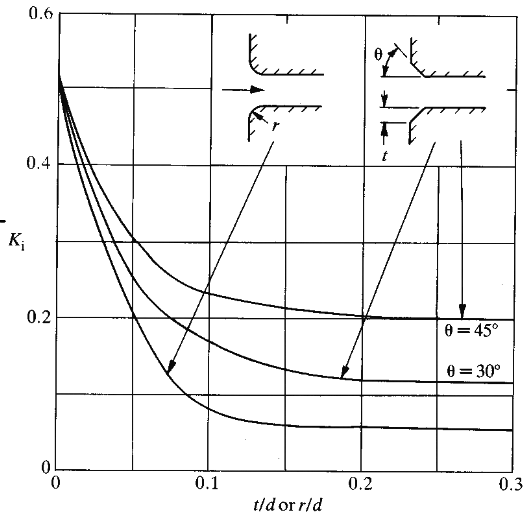
Once more, if the variation in height was not negligible, it could be accounted for by simply replacing p_1 with $p_1 + \rho g h_1$, etc., and the resulting loss of total pressure would still be

$$\boxed{p_{T1} - p_{T3} = \frac{1}{2}\rho V_1^2 K.}$$

7.5 Changes in total pressure across other network components

There are similar losses of total pressure across other network elements, for instance at the entrance to a pipe:





Other components such as valves, junctions and bends cause a similar total pressure loss. Dimensional analysis always shows that the loss is of the form

$$\boxed{\Delta p_T = \frac{1}{2}\rho V_1^2 K,}$$

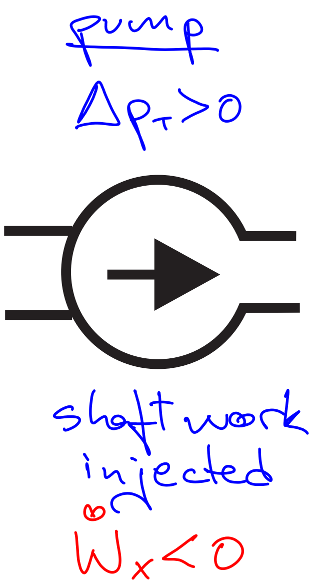
where K is a function of Re and the geometry. However, at high Reynolds numbers the flow is highly turbulent and K becomes Re-independent, leaving only the geometry dependence. The values of K for a wide variety of components has been measured experimentally and is tabulated in books such as *Internal Flow Systems* by D. S. Miller:

Table 12.4. Loss coefficients for composite diffusers with thick inlet boundary layers (values in parentheses are for thin inlet boundary layers)

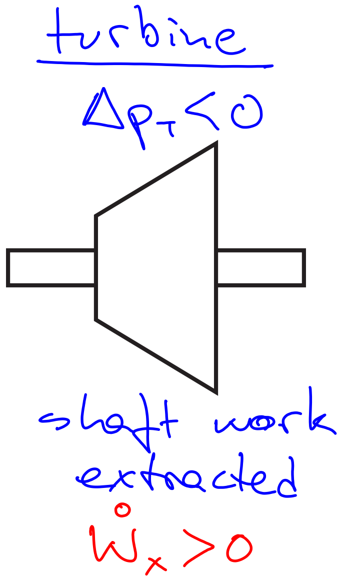
No.	K_{td}	Arrangement
1	0.74 (0.72)	1
2	0.72 0.93	1 b
3	0.64 (0.57)	1 b p
11	0.75 (0.64)	3 b p
12	0.75	3 p b
13	0.52	3 p

7.6 Mechanical work, pumps and turbines

Pumps do mechanical work on a fluid to produce a rise in total pressure. Turbines, on the other hand, extract mechanical work from a fluid and cause a total pressure loss. The exact mechanisms of this are described in the third year (3A3 Compressible Flow) and the fourth year (Turbomachinery).



$$\Delta p_T = -\frac{\dot{W}_x}{Q}$$



The losses in total pressure that we have seen in Sections 7.2 to 7.5 correspond to losses in the mechanical energy of the fluid. In all those cases, the losses were due to viscous and turbulent dissipation. It is also possible to transfer part of the fluid's energy via an energy-extracting device. For instance, a flow can do shaft work on a turbine, exerting a shaft power, $\dot{W}_x$, on it. These processes are never perfectly reversible, so some energy is lost to irreversibilities. The total pressure change across a control volume is related to the shaft power extracted from the fluid by

$Q \equiv$ volumetric flow rate $[L^3/T]$

$$\frac{\dot{m}}{\rho}(p_{T_{out}} - p_{T_{in}}) + (\text{irreversibility losses}) = -\dot{W}_x$$

examples

pump: $(400) + (100) = -(-500)$

turbine: $(-250) + (50) = -(200)$

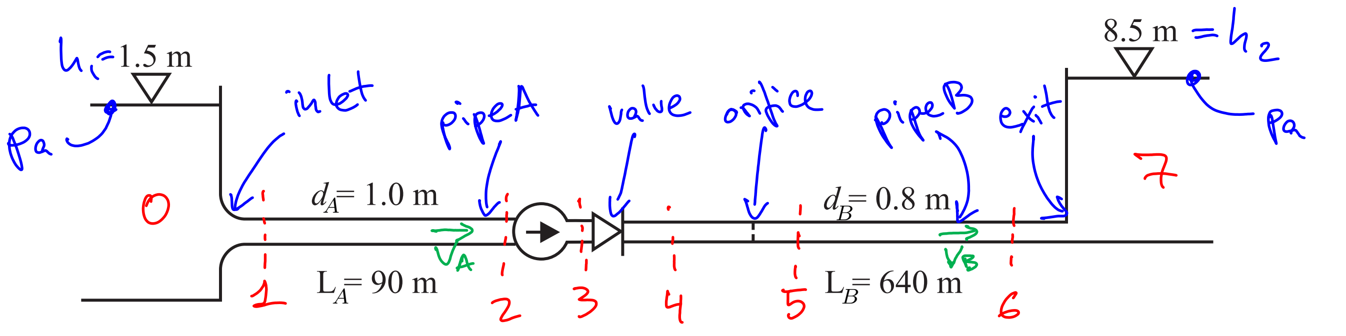
Similarly, if mechanical shaft power is exerted on the fluid then the total pressure rises. Again, the power exerted is equal to the volumetric flowrate multiplied by the total pressure change.

7.7 Worked example

The network of the figure has the following friction and loss coefficients (assuming that Re is large enough):

$$c_{f,pipe} = 3.75\times10^{-3},\quad K_{inlet} = 0.1,\quad K_{valve} = 0.5,\quad K_{orifice} = 0.5,\quad K_{outlet} = 1.0$$

Find the power required for the pump to drive $Q = 2.75\mathrm{m^3s^{-1}}$ of water from the left tank to the right tank. Assume that both tanks are at the atmospheric pressure, p_a.



Let us start by writing the changes in total pressure across the different network elements:

-Left tank: $\quad p_{T0} = p_a + \rho g h_1$

-Inlet: $\quad \Delta p_{T0\to1} = -\frac{1}{2}\rho V_A^2 K_{inlet}$

-Pipe A: $\quad \Delta p_{T1\to2} = -\frac{1}{2}\rho V_A^2\left(\frac{4L_A}{d_A}c_{f,pipe}\right)$

-Pump: $\quad \Delta p_{T2\to3} = \boxed{-\frac{\dot{W}_x}{Q}}$ this increases energy (p_T), so should end up being >0

-Valve: $\quad \Delta p_{T3\to4} = -\frac{1}{2}\rho V_B^2 K_{valve}$

-Orifice plate: $\quad \Delta p_{T4\to5} = -\frac{1}{2}\rho V_B^2 K_{orifice}$

-Pipe B: $\quad \Delta p_{T5\to6} = -\frac{1}{2}\rho V_B^2\left(\frac{4L_B}{d_B}c_{f,pipe}\right)$

-Outlet: $\quad \Delta p_{T6\to7} = -\frac{1}{2}\rho V_B^2 K_{outlet}$

-Right tank: $\quad p_{T7} = p_a + \rho g h_2$

$$p_{T7} = p_{T0} + \Delta p_{T0\to1} + \Delta p_{T1\to2} + \ldots + \Delta p_{T6\to7}$$

The volumetric flow rate gives us the values of V_A and V_B through mass conservation:

$$Q = \int_{\text{(section)}} v_x \, \mathrm{d}A = \frac{\pi \, d_A^2}{4} \, V_A = \frac{\pi \, d_B^2}{4} \, V_B$$

Substituting we have:

$$\begin{aligned}
p_{T7} = & \quad p_{T0} \quad + \Delta p_{T0\to 1} + \Delta p_{T1\to 2} + \ldots + \Delta p_{T6\to 7} \\
p_a + \rho g h_2 = p_a + \rho g h_1 & - \frac{1}{2} \rho V_A^2 \left(K_{inlet} + \frac{4\, L_A}{d_A} \, c_{f,pipe} \right) \\
& - \frac{\dot{W}_x}{Q} \\
& - \frac{1}{2} \rho V_B^2 \left(K_{valve} + K_{orifice} + \frac{4\, L_B}{d_B} \, c_{f,pipe} + K_{outlet} \right)
\end{aligned}$$

The pumping power is finally:

$$\begin{aligned}
-\dot{W}_x = \rho Q \Bigg\{ g\,(h_2 - h_1) + \frac{1}{2} \Bigg[& \frac{16 Q^2}{\pi^2 d_A^4} \left(\underbrace{K_{inlet}}_{0.1} + \underbrace{\frac{4\, L_A}{d_A} \, c_{f,pipe}}_{1.35} \right) \\
& + \frac{16 Q^2}{\pi^2 d_B^4} \left(\underbrace{K_{valve}}_{0.5} + \underbrace{K_{orifice}}_{0.5} + \underbrace{\frac{4\, L_B}{d_B} \, c_{f,pipe}}_{12.0} + \underbrace{K_{outlet}}_{1.0} \right) \Bigg] \Bigg\}
\end{aligned}$$

5th power!!

$-\dot{W}_x = 789\text{Kw}$

(if there were irrebersibilities, we would need <u>more</u> power into the pump:

$-\dot{W}_x = 789\,\text{kw} + \text{irreversibilities}$)

Engineering Tripos 1B

Paper 4

Fluid Mechanics

Lecture 8 - Laminar boundary layers

- Boundary layers
- Boundary layer growth
- Bernoulli and streamline curvature in boundary layers
- Pressure gradients in boundary layers
- Boundary layer separation
- Delaying separation

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

8.1 Inviscid vs. high-Re flows - Boundary layers

In Lectures 3 and 4 we studied viscous and inviscid flows. Inviscid flows are governed by Euler equations,

$$\rho\frac{\partial \mathbf{v}}{\partial t} \quad + \quad \rho\left(\mathbf{v}\cdot\nabla\mathbf{v}\right) \quad = \quad -\nabla p\,,$$

while viscous flows are governed by Navier-Stokes equations. Comparing both equations, it would be natural to assume that, when the viscous effects are negligible, Navier-Stokes' equations become Euler's. Let us look at the order of magnitude of the different terms to see when the viscous one would indeed be negligible:

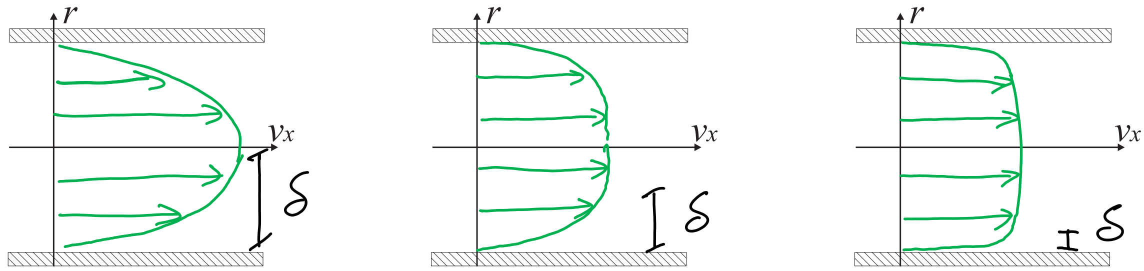
$$\rho\frac{\partial \mathbf{v}}{\partial t} \quad + \quad \rho\left(\mathbf{v}\cdot\nabla\mathbf{v}\right) \quad = \quad -\nabla p \quad + \quad \mu\nabla^2\mathbf{v}.$$

$$\left(\sim\rho\frac{V}{T}\right) \qquad \underbrace{\sim\rho V\frac{V}{D}}_{} \qquad \sim\frac{\Delta p}{D} \qquad \underbrace{\sim\mu\frac{V}{D^2}}_{}$$

or $\emptyset$, steady $\qquad \gg ?$

$$\frac{\rho V^2/D}{\mu V/D^2} = \frac{\rho V D}{\mu} = Re \gg 1?$$

This shows that the Reynolds number is nothing but a measure of the relative importance of advective versus viscous effects. As the Reynolds number tends to infinity, we would expect the flow to become inviscid.



Re increases $\longrightarrow$

However, this is not entirely true. An inviscid flow would slip over a solid surface, but one with viscosity –small as it may be– will always satisfy the no-slip condition. This is so because *viscous effects are never negligible near a wall.* In our previous order-of-magnitude analysis, we assumed that variations in $\mathbf{v}$ for the viscous term occurred over distances D, but in high-Re flows they occur in very thin layers near the walls, of thickness δ. We call these regions *boundary layers*, and we can define a Reynolds number for them,

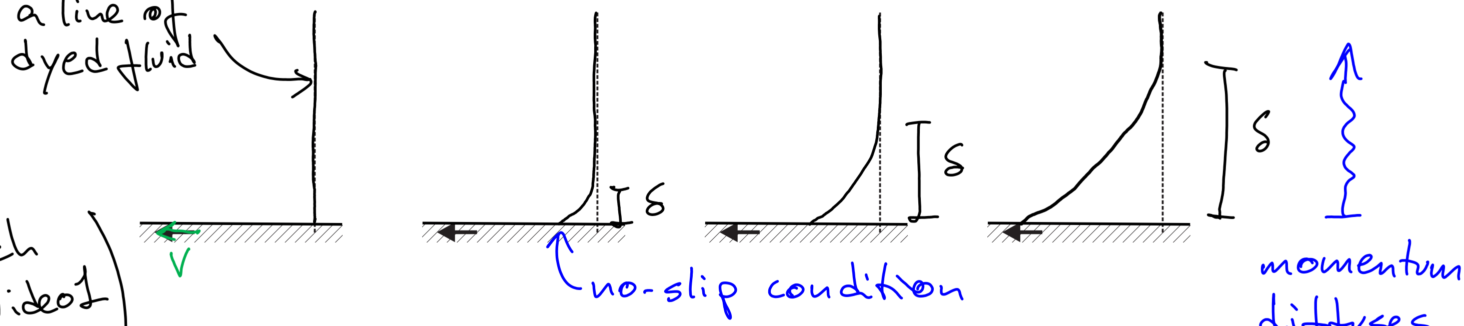
$$Re_\delta = \frac{\rho V\delta}{\mu} \quad \cancel{\gg 1}$$

In laminar flows, Re_δ turns out to be small enough that viscous effects are important –but only up to a distance $\sim\delta$ from the wall.

8.2 Boundary layer growth

The previous section offers an intuitive idea of why boundary layers exist, but the concept of boundary layer only attains its full significance when the boundary layer is allowed to *evolve* in the streamwise direction (this does not occur in the fully developed flow in a pipe or between plates, which as we saw in Lectures 4 and 6 is streamwise-uniform). To illustrate this idea, let us consider what happens when a free stream encounters a plate aligned with the flow, or conversely, when a plate advances through a fluid initially at rest.

Time evolution for a plate moving through a viscous fluid:



(Watch 8.2 Video 1 on Moodle)

Due to the no-slip condition, the flow in immediate contact with the plate must have the same velocity as the plate. Through shear, the fluid layer near the plate will exchange momentum with the layer just above, which will begin to move. This will in turn generate some shear with the next layer above, and so forth.

In the frame of reference of the moving plate, if we focus on a fixed x in space, we would observe a y-diffusion of momentum away from the plate over time. Conversely, if we think of the plate as stationary, and the flow moving with respect to it, we would observe that the effect of the no-slip condition propagates farther in y as we move downstream along x.

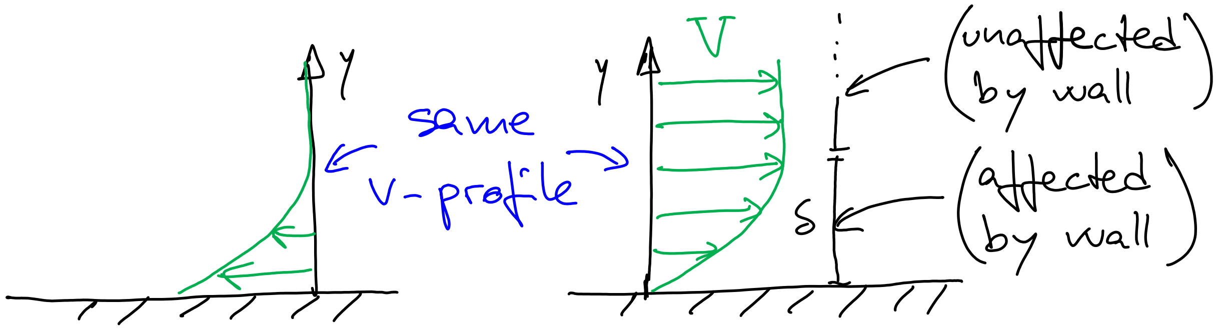


Plate moving, fluid stationary. Plate stationary, fluid moving.

The velocity profile is not parabolic, or linear, as it was for the flow between flat plates. In fact, there is no top boundary condition –or it is at infinity– so there is no physical bound limiting the diffusion away from the plate of the boundary layer. This gives rise to an entirely different solution for the velocity profile, although its curvature can be reminiscent of Poiseuille flow.

(Watch 8.2 Videos 2-3 on Moodle)

To obtain an estimate for the growth of the boundary layer thickness, we can revisit the order of magnitude argument of Section 8.1. When viewing the flow as a free stream over a fixed plate, the boundary layer is steady and two-dimensional, and there are no pressure gradients. We then have

- continuity:

$$\nabla \cdot \mathbf{v} = 0$$

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} = 0$$

$$\sim \frac{V}{x} \qquad \sim \frac{V_y}{\delta} \qquad \Longrightarrow \boxed{V_y \sim \frac{\delta}{x} V}$$

- momentum in x:

$$\cancel{\rho \frac{\partial \mathbf{v}}{\partial t}} + \rho\left(\mathbf{v} \cdot \nabla \mathbf{v}\right) = -\nabla p + \mu \nabla^2 \mathbf{v}$$

$$\rho\left(v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y}\right) = \cancel{\frac{\partial p}{\partial x}} + \mu\left(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2}\right)$$

$$\sim \rho\left(V\frac{V}{x} + V_y \frac{V}{\delta}\right) \qquad \sim \mu\left(\cancel{\frac{V}{x^2}} + \frac{V}{\delta^2}\right)$$

$$x \gg \delta$$

$$\sim \rho \frac{V^2}{x} \qquad \sim \mu \frac{V}{\delta^2} \qquad \Longrightarrow \boxed{\delta^2 \sim \frac{\mu\, x}{\rho\, V}}$$

- momentum in y:

$$x \gg \delta$$

$$\rho\left(v_x \frac{\partial v_y}{\partial x} + v_y \frac{\partial v_y}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\left(\cancel{\frac{\partial^2 v_y}{\partial x^2}} + \frac{\partial^2 v_y}{\partial y^2}\right)$$

$$\delta \times \left(\sim \rho\, \delta^2 \frac{V^2}{x^2} \qquad \sim \frac{\Delta_y p}{\cancel{\delta}} \qquad \sim \mu \frac{V}{\cancel{\delta} x}\right)$$

$$\underbrace{\sim \mu \frac{V}{x} \qquad \sim \Delta p \qquad \sim \mu \frac{V}{x}}_{\text{much smaller than anything in x-equation}} \qquad \Longrightarrow \boxed{\frac{\partial p}{\partial y} \approx 0}$$

We obtain that the wall-normal velocity is much smaller than V, the thickness of the boundary layer increases in proportion to $\sqrt{x}$, where x is the distance from the front of the plate, and the wall-normal pressure gradient is negligible,

$$\boxed{\delta \sim \sqrt{\frac{\mu\, x}{\rho\, V}}, \quad V_y \sim \frac{\delta}{x} V \ll V, \quad \frac{\partial p}{\partial y} \approx 0.}$$

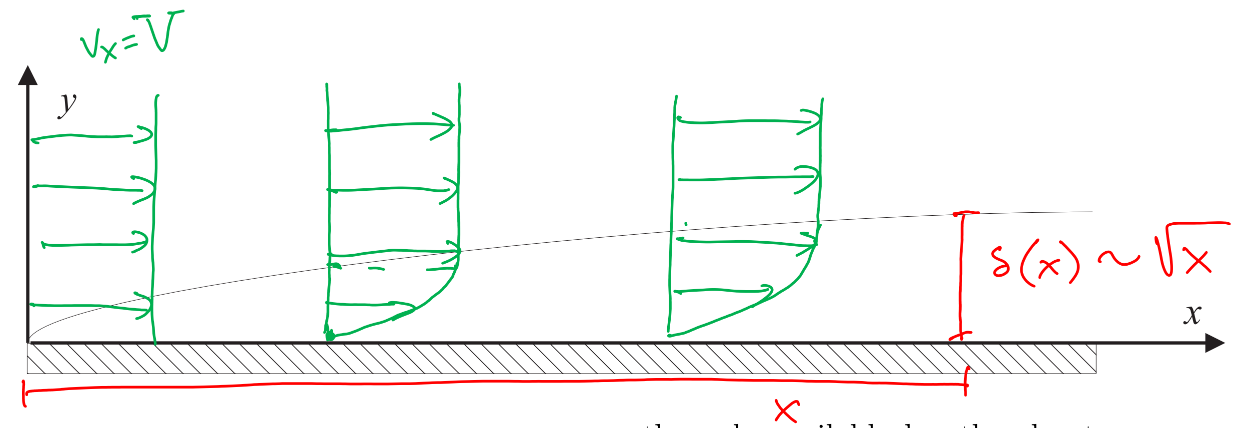
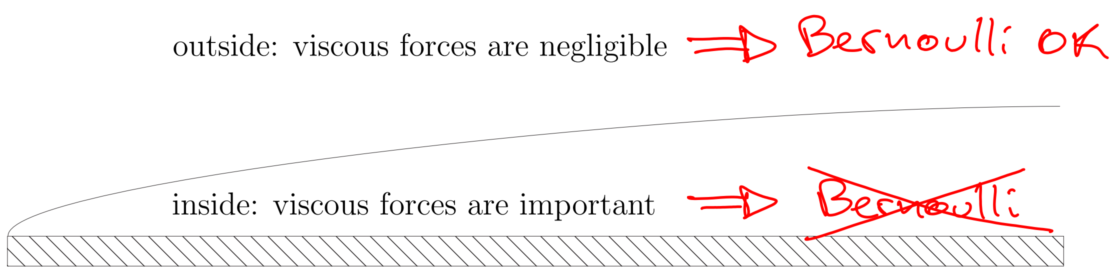


plate has infinite length along x $\Rightarrow$ the only available lengthscale at each point is the local x

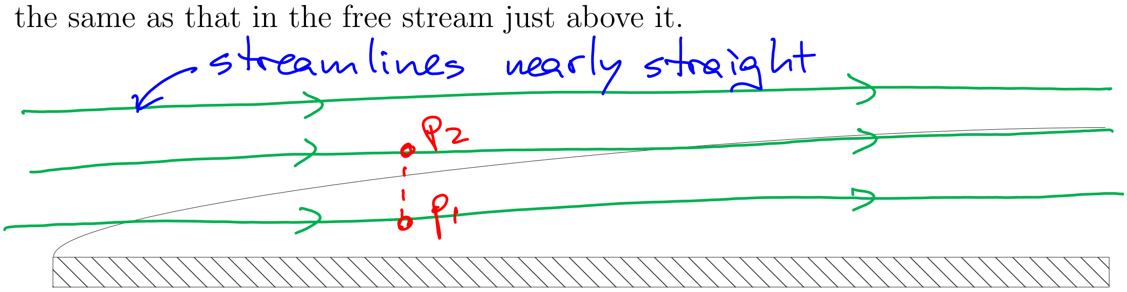
8.3 Bernoulli and boundary layers

In Lecture 3, Bernoulli's equation was derived for inviscid flows, where only pressure and inertial forces are present. Within a boundary layer, *viscous forces* are important, which implies that *Bernoulli does not hold*. We can however still use Bernoulli outside the boundary layer where, by definition, viscous effects are negligible. This has a very important consequence: for flows with relatively low viscosity, we can solve the flow field assuming that it is inviscid –using Euler– if we neglect the the thin boundary layer regions. We could then use that *external* solution as the boundary condition to solve the boundary layer region.



8.4 Streamline curvature and normal pressure gradients

In Section 8.2 we have seen that the wall-normal velocity in the boundary layer is very small. The streamlines are therefore almost wall-parallel and have negligible curvature. We have also seen that the wall-normal pressure gradient is also negligible. For this reason, the pressure at a given point in the boundary layer is essentially the same as that in the free stream just above it.

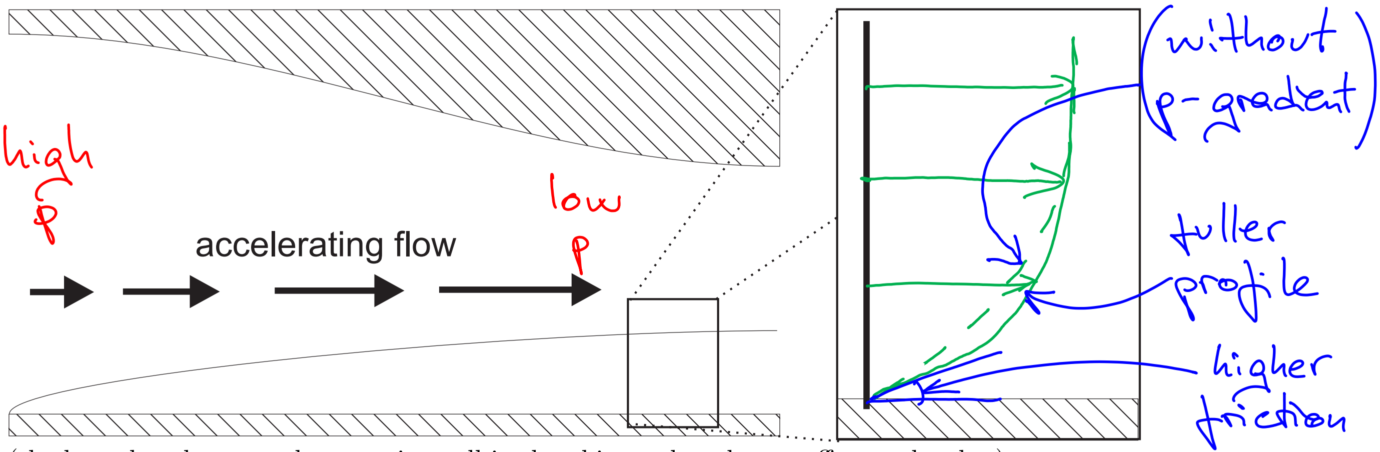


$\boxed{p_2 = p_1}$

8.5 Boundary layers in flows with a pressure gradient

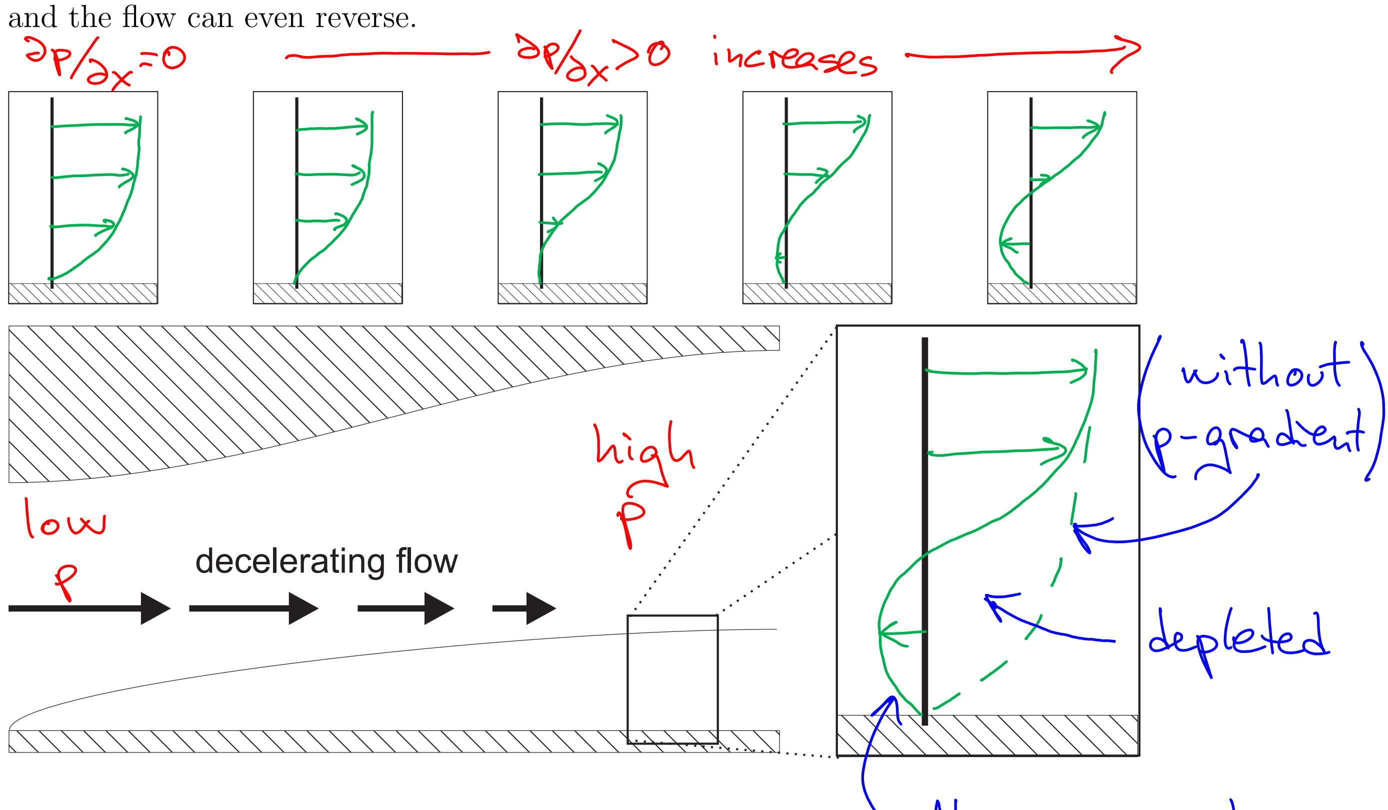
The discussion on boundary layer growth in Section 8.2 was based on the absence of pressure gradients. If these are present the boundary layer is a bit more complicated but, qualitatively, the effect is similar to that of pressure gradients on a Couette flow, which we saw in Lecture 4, section 7.

- In the presence of a *favourable* pressure gradient, as the flow moves towards low pressures it accelerates, and the velocity profile within the boundary layer becomes fuller. The velocity gradient at the wall becomes steeper and friction increases:



(the boundary layer on the opposite wall is also thin so they do not affect each other)

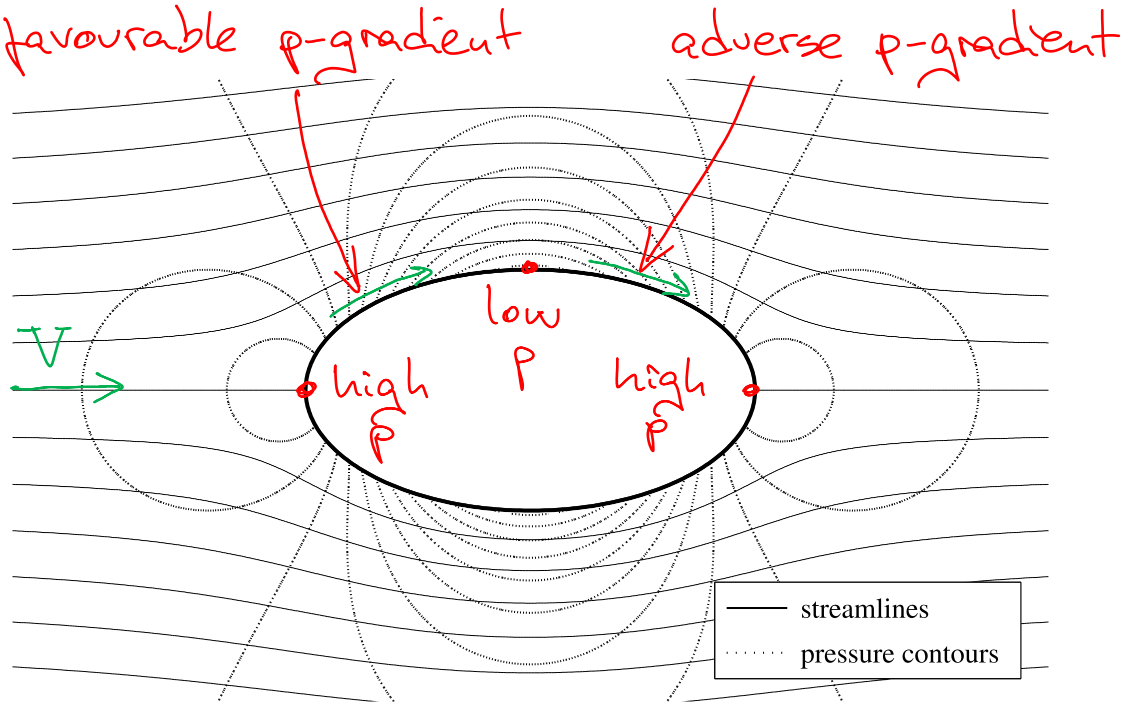
- If, conversely, the pressure gradient is *adverse*, the flow decelerates, the boundary layer velocity profile is depleted, the velocity gradient at the wall becomes less steep and the flow can even reverse.



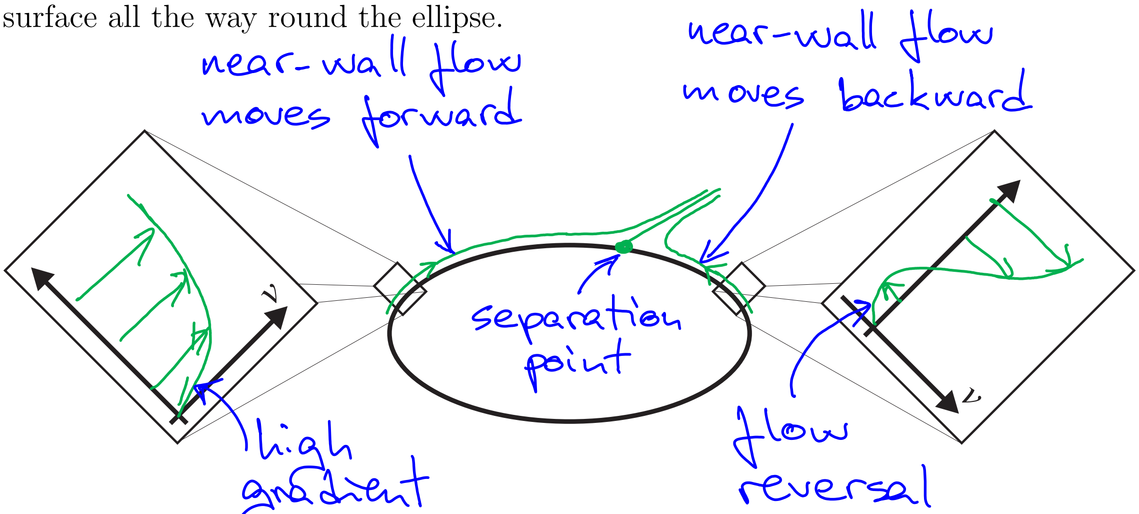
Attempt questions S2 and S3 in Examples Paper 2

8.6 Boundary layer separation

The streamlines and pressure field of *inviscid* flow around an ellipse are shown below. There are high pressure regions around the front and rear stagnation points.



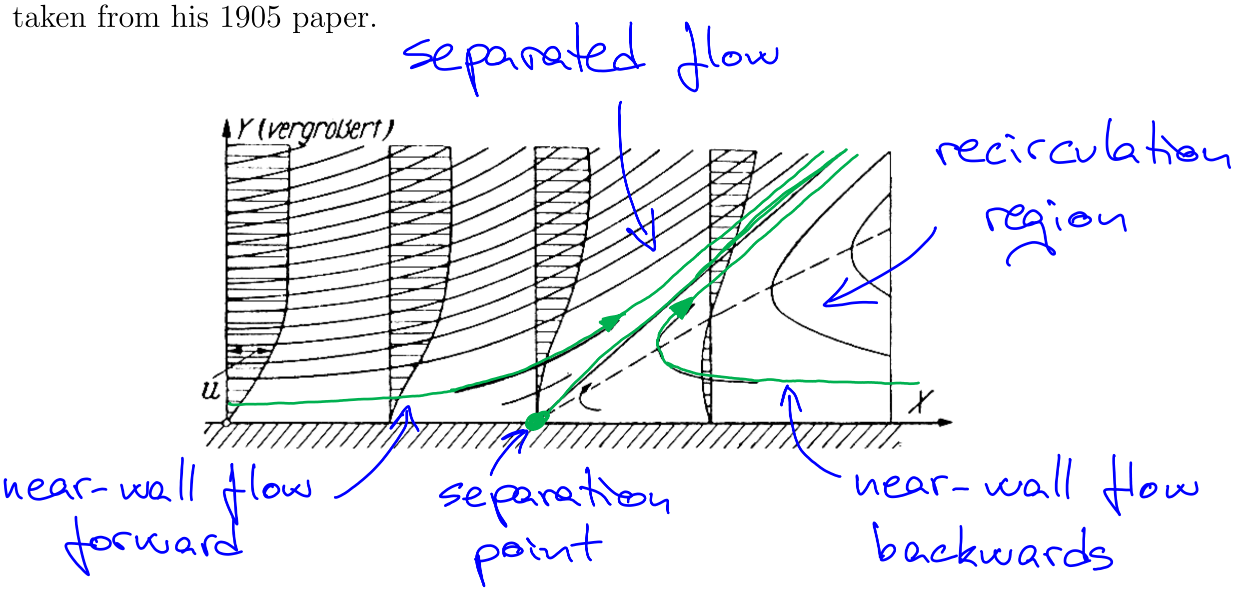
All real fluids are viscous and obey the no slip condition. This means that a thin boundary layer forms around the surface of the ellipse, growing as the fluid moves downstream. Let us imagine for a moment that the boundary layer stuck to the surface all the way round the ellipse.



Around the front of the ellipse there is a *favourable* pressure gradient - i.e. the pressure is pushing in the same direction as the bulk fluid motion. The favourable pressure gradient makes the velocity gradient at the wall steeper. Around the back of the ellipse there is an *adverse* pressure gradient - i.e. the pressure is pushing in the opposite direction to the bulk fluid motion. The adverse pressure gradient makes the velocity gradient at the wall less steep and *eventually will cause flow reversal.*

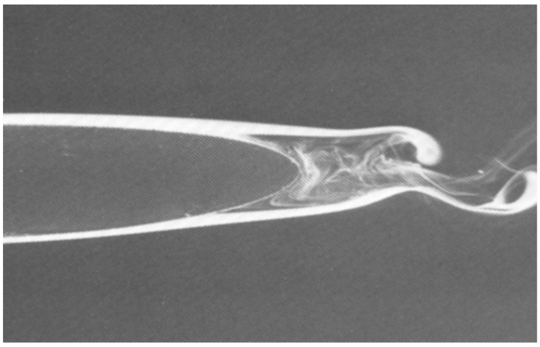
Watch 8.6 Video 1 on Moodle

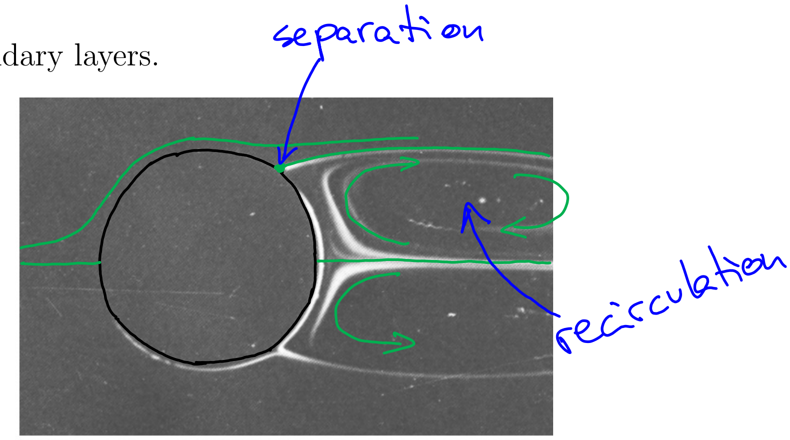
This flow reversal *completely* changes the flow! The reversing fluid has to go somewhere. It cannot reverse all the way to the front of the ellipse because there is a favourable pressure gradient there and all the fluid is moving forwards. Instead it separates from the body at a mini-stagnation point, which is called the point of separation. Ludwig Prandtl was the first person to realize this. The figure below is taken from his 1905 paper.

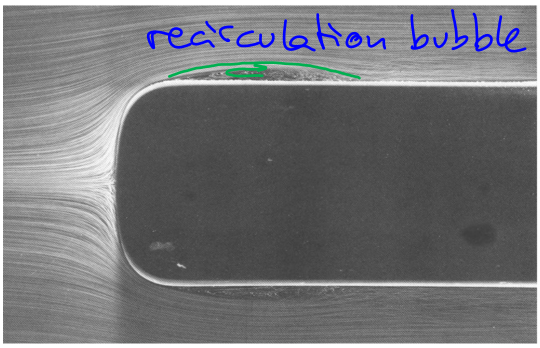


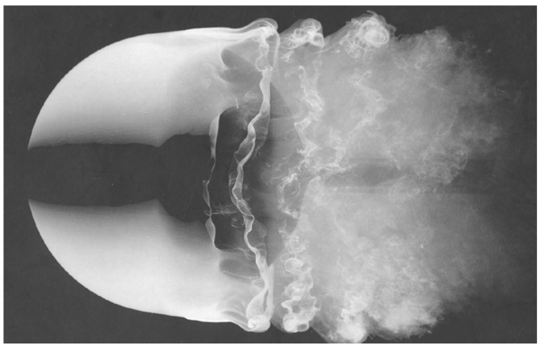
"A fluid layer which is set into rotation by friction at the wall pushes itself out into the free fluid where it causes a complete transformation of the motion."

Here are some examples of separated boundary layers.





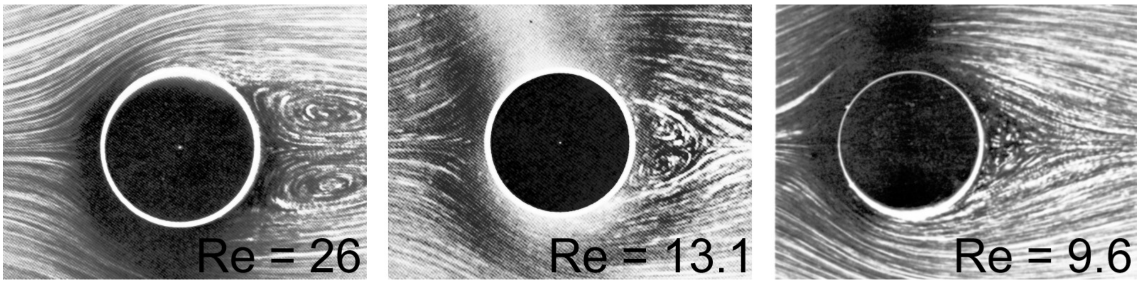




(Watch 8.6 Video 2 on Moodle)

8.7 Delaying boundary layer separation

Under adverse pressure gradients, whether the flow reverses near the wall is determined by the competing effects of the pressure gradient and the diffusion of momentum from the external flow towards the wall. That momentum diffusion is more efficient when the flow is more viscous, so increasing the viscosity can make the flow more robust against separation.

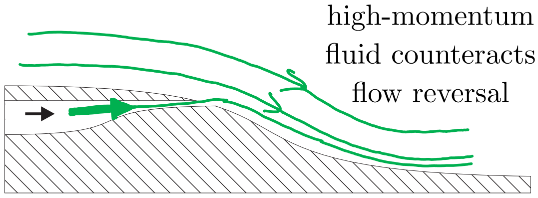


'viscosity increases'

momentum diffuses from the outer flow more efficiently into the boundary layer, which can then withstand the adverse pressure gradient longer

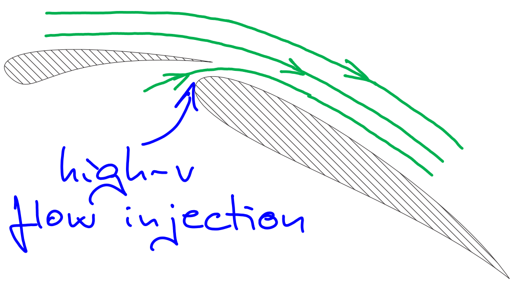
In aerodynamics, separation is usually detrimental because it increases drag. However, increasing viscosity (if it were possible) would also increase drag by increasing the wall shear stress. It is possible to delay separation in other ways, for example by injecting momentum directly into the boundary layer:

(Watch 8.7 Video 1 on Moodle)



injection of high-momentum fluid counteracts flow reversal

This is *part* of the principle behind leading edge slats on wings:[1]

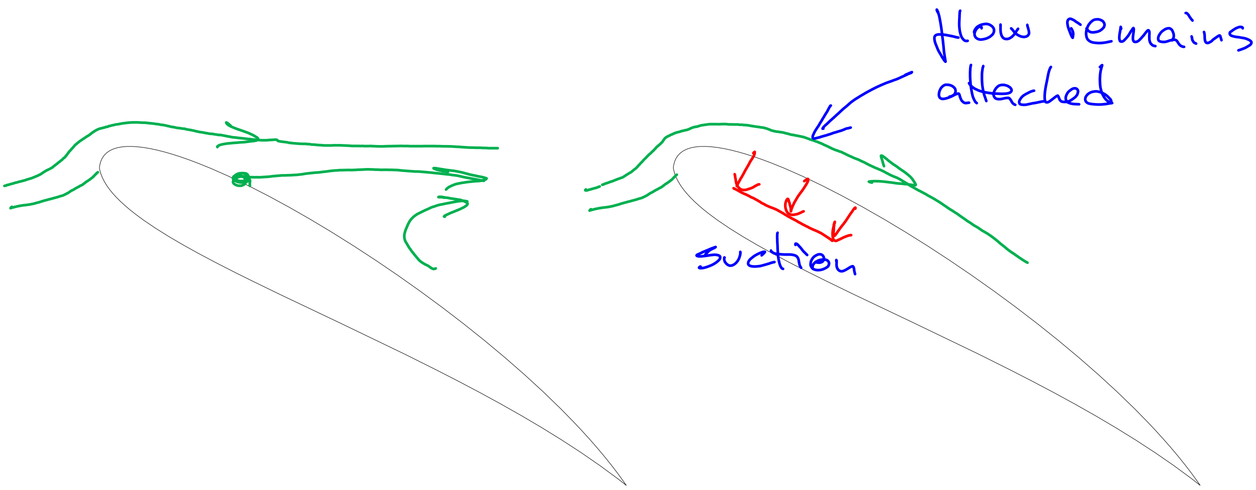


boundary layer remains attached at higher angles of attack

[1]Actually, this is not the main reason that leading edge slats are used. The main benefit of slats is that they change the mean flow around the wing. We look at this further in the 3rd year.

(Watch 8.7 Videos 2-3 on Moodle)

Another technique to avoid separation is to suck the low momentum part of the boundary layer into the wing:



This has been achieved on aeroplanes in flight but requires a great deal of power. In Lecture 9 we will look at a more common way to increase momentum transfer into the boundary layer and hence avoid separation.

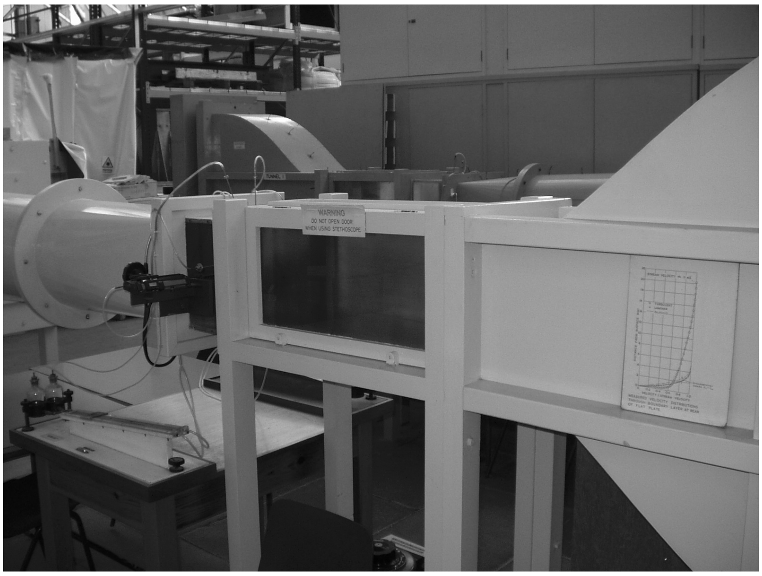
(Watch 8.7 Video 4 on Moodle)

Engineering Tripos 1B

Paper 4

Fluid Mechanics

Lecture 9 - Turbulent boundary layers

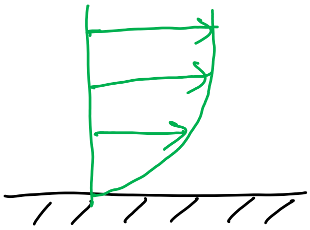


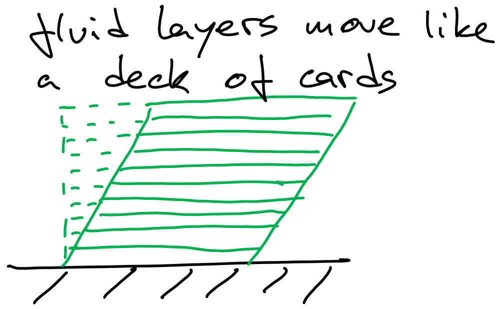
- Reynolds numbers in a boundary layer
- Momentum loss
- Transition to turbulence
- The effect of turbulence
- Transition vs. separation
- Boundary layer re-attachment

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

9.1 Recap on laminar boundary layers

In Lecture 8 we have studied *laminar* boundary layers. In these, wall-parallel layers of fluid slide and shear smoothly over each other, like a pack of cards.



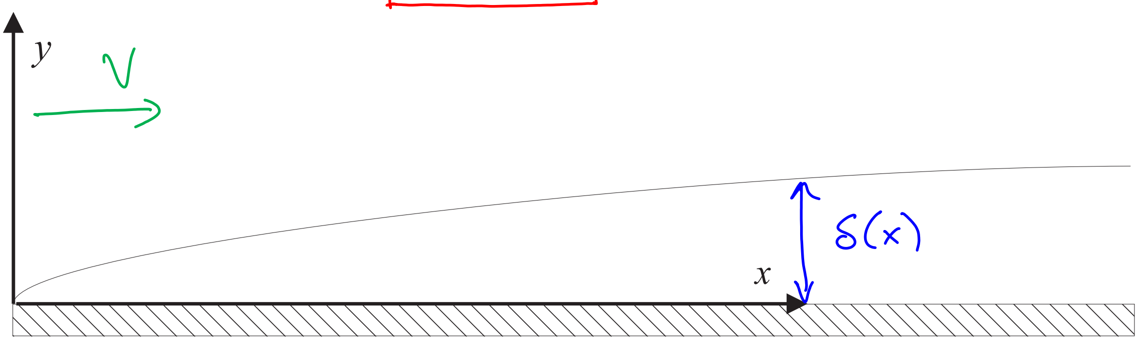


In this lecture, we will focus on turbulent boundary layers, where viscous dissipation is not sufficient to absorb all the energy fed into the flow, which subsequently breaks up in a disorganised way. In a sense, turbulent boundary layers are to their laminar counterparts what turbulent pipe flow is to Poiseuille flow, as we saw in Lecture 6.

9.2 Reynolds numbers in a boundary layer

In Lecture 8 we also defined a Reynolds number based on the boundary layer thickness, $\boxed{Re_\delta = \rho V \delta / \mu.}$ However, it is often difficult to determine δ, so alternatively we use the Reynolds number based on the streamwise distance from the beginning of the boundary layer x,

$$\boxed{Re_x = \frac{\rho V x}{\mu}}$$

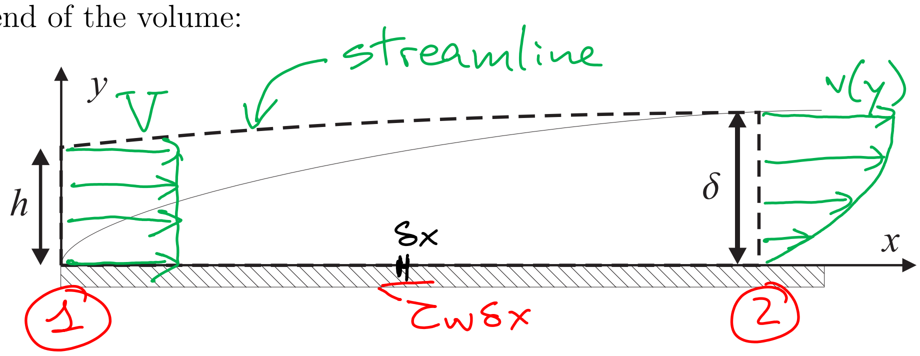


Note that Re_δ and Re_x have a one-to-one correspondence, as the relationship between δ and x is univocal,

$$\delta \sim \sqrt{\frac{\mu x}{\rho V}} \quad \Rightarrow \boxed{\frac{\delta}{x} \sim \sqrt{\frac{\mu}{\rho V x}}} = \boxed{\frac{1}{\sqrt{Re_x}}}$$

9.3 Momentum loss

As the boundary layer grows, the flow loses momentum because of the friction shear stress exerted by the wall. The momentum loss can be related to the shear stress by considering a control volume delimited by the streamline that passes through $y = \delta$ at the end of the volume:



The balance of x-momentum (per unit length into page) gives that the force on the control volume equals the net flux of momentum out,

$$\underbrace{-\int_0^x \tau_w \,dx}_{\text{force on control volume}} = \underbrace{\int_0^\delta \rho v v \,dy}_{2} - \underbrace{\rho V V h}_{1}$$

The height h is unknown, but by conservation of mass (per unit length into the page), $\dot{m}_{out} = \dot{m}_{in}$, we have

$$V \times \left(\int_0^\delta \rho v \,dy = \rho V h \right)$$

We can use this to eliminate h from the momentum equation, obtaining

$$\underbrace{\int_0^x \tau_w \mathrm{d}x}_{\substack{(a)\\ \text{net force on} \\ \text{the plate}}} = \underbrace{V \int_0^\delta \rho v \mathrm{d}y}_{\substack{(b)\\ \text{momentum flux} \\ \text{in through ①}}} - \underbrace{\int_0^\delta \rho v^2 \mathrm{d}y.}_{\substack{(c)\\ \text{momentum flux} \\ \text{out through ②}}}$$

The integral on the left gives the total force that the plate exerts on the fluid between 0 and x. On the right hand side, the second term is the flux of momentum when the flow leaves the control volume, and the first term is equal to the momentum flux when the flow entered the control volume –except we have managed to express it in 'exit' variables. Using this equation, we can work out the friction at the wall if we know the velocity profile $v(y)$. Note that we have made no assumptions on the type of flow, so this works both for laminar and turbulent boundary layers, and could easily be extended to cases with a pressure gradient.

Let us use the previous relationship to calculate the growth rate of a boundary layer. For example, the velocity profile of a laminar boundary layer is given approximately by

$$v(y) = V\left[\frac{3}{2}\frac{y}{\delta} - \frac{1}{2}\left(\frac{y}{\delta}\right)^3\right].$$

(approximate laminar-boundary-layer profile)

The wall shear stress is then

(a) $$\tau_w = \mu \frac{\mathrm{d}v}{\mathrm{d}y}\bigg|_{y=0} = \frac{3\mu V}{2\delta}$$

The momentum flux on entry is equal to

(b) $$V\int_0^\delta \rho v \mathrm{d}y = \ldots\ldots = \frac{5}{8}\rho V^2\delta$$

The momentum on leaving the control volume is

(c) $$\int_0^\delta \rho v^2 \mathrm{d}y = \ldots\ldots = \frac{17}{35}\rho V^2\delta$$

Combining all the above we have

(a) stress = (b) momentum flux in − (c) momentum flux out

$$\frac{3\mu V}{2}\int_0^x \frac{\mathrm{d}x}{\delta} = \frac{5}{8}\rho V^2\delta - \frac{17}{35}\rho V^2\delta = \frac{39}{280}\rho V^2\delta$$

$$\Rightarrow \int_0^x \frac{\mathrm{d}x}{\delta} = \frac{13}{140}\frac{\rho V}{\mu}\delta,$$

and differentiating with respect to x we finally obtain

$$\frac{1}{\delta} = \frac{13}{140}\frac{\rho V}{\mu}\frac{\mathrm{d}\delta}{\mathrm{d}x}$$

$$\Rightarrow \delta\frac{\mathrm{d}\delta}{\mathrm{d}x} = \frac{\mathrm{d}}{\mathrm{d}x}\left(\frac{\delta^2}{2}\right) = \frac{140}{13}\frac{\mu}{\rho V}$$

$$\Rightarrow \delta^2 = \frac{280}{13}\frac{\mu}{\rho V}x$$

$(\Rightarrow \delta \sim \sqrt{x})$

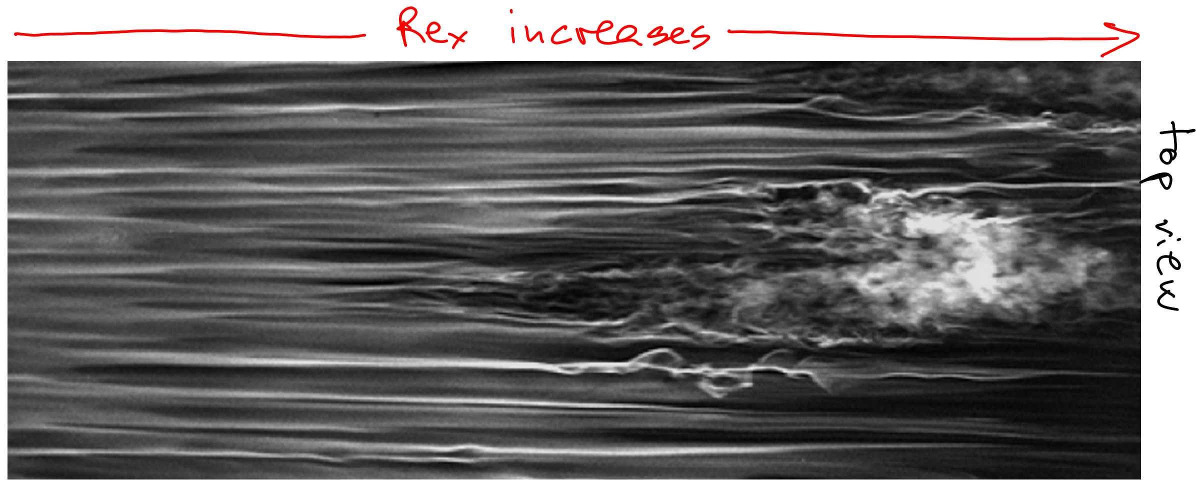
$$\Rightarrow \frac{\delta}{x} = \frac{4.64}{\sqrt{Re_x}}.$$

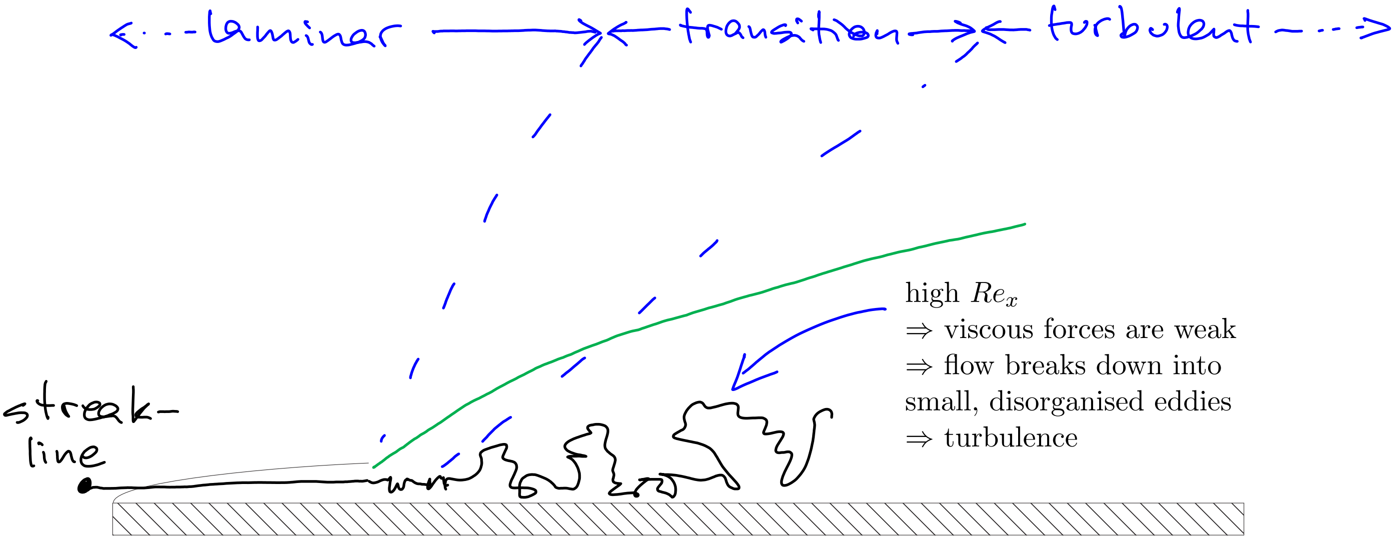
We can see that, by combining the equation for the momentum loss in a boundary layer with the velocity profile $v(y)$, we can calculate the boundary layer growth δ along x. Note however that this derivation is very sensitive to the velocity gradient at the wall, $\mathrm{d}v/\mathrm{d}y|_{y=0}$.

Do Q9 in Examples Paper 2

9.4 Transition to turbulence in boundary layers

We have seen that a Reynolds number compares the relative importance of advective and viscous terms. In a boundary layer, the Reynolds number Re_x increases as the flow moves downstream, and the viscous terms become comparatively less and less important. As in pipe flow, at high Re_x the viscosity is not able to support the flow shear, and the flow breaks down into smaller and smaller, disorganised eddies, until a scale is reached at which viscosity can act. In the early stages of a boundary layer, Re_x is relatively low, the viscous forces are strong enough to dissipate all the energy fed into the flow, and the boundary layer remains laminar. Once Re_x is large enough, the boundary layer transitions to turbulence, and remains turbulent thereafter.



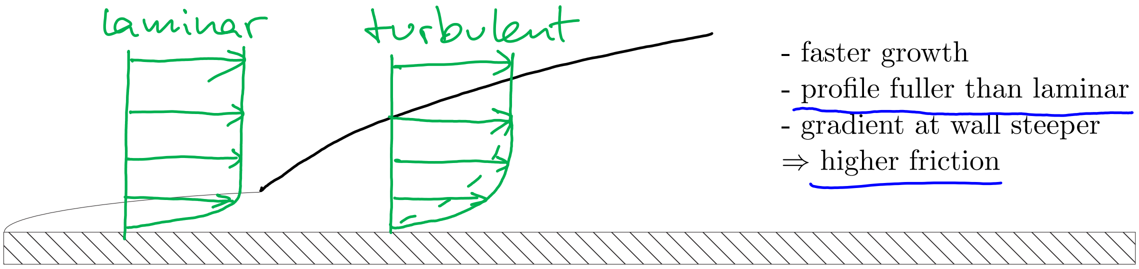


In a very controlled environment, like a wind tunnel with extremely low noise, the transition to turbulence occurs suddenly for a given value of Re_x. In most real life applications, however, the transition is less organised, and turbulent spots form irregularly, as in the top view above. Nevertheless, the region over which the flow transitions from laminar to fully turbulent is usually small compared to the length of the boundary layer.

(See resources on Moodle on section 9.4)

9.5 Effect of turbulence on the boundary layer

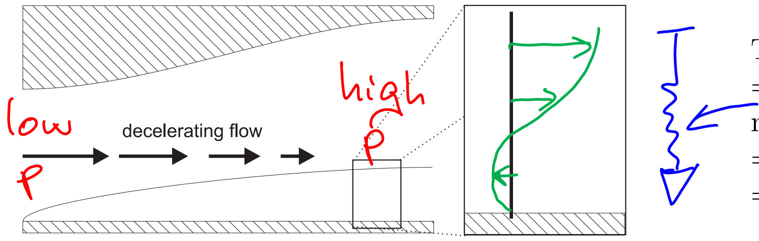
Turbulence increases the rate of momentum transfer between the surface and the free stream, as we discussed in Lecture 6 when we introduced the concept of eddy viscosity. This has three direct effects: the boundary layer grows more quickly, the velocity profile is fuller and the skin friction increases.



- faster growth
- profile fuller than laminar
- gradient at wall steeper

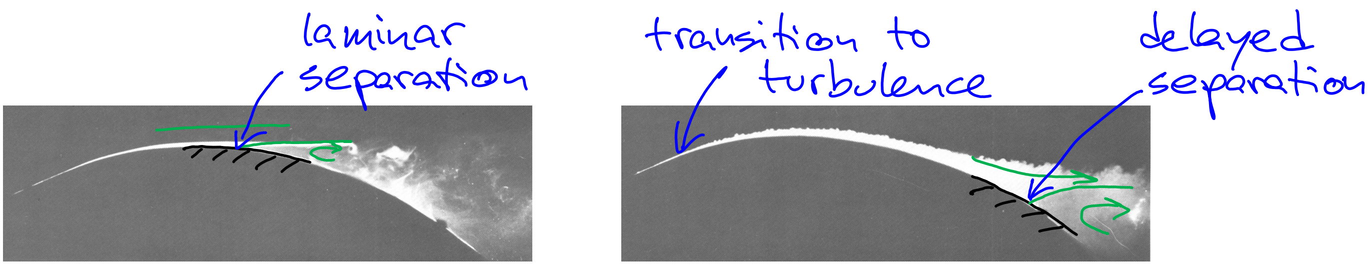
⇒ higher friction

The increase in momentum transfer has another very important consequence. In Lecture 8 we saw that the flow inside a boundary layer can reverse direction in the presence of a strong adverse pressure gradient, and that this causes the *separation* of the boundary layer from the body.



Turbulent boundary layer:
⇒ momentum eddy diffusion makes profile fuller
⇒ more robust vs. flow reversal
⇒ separation delayed

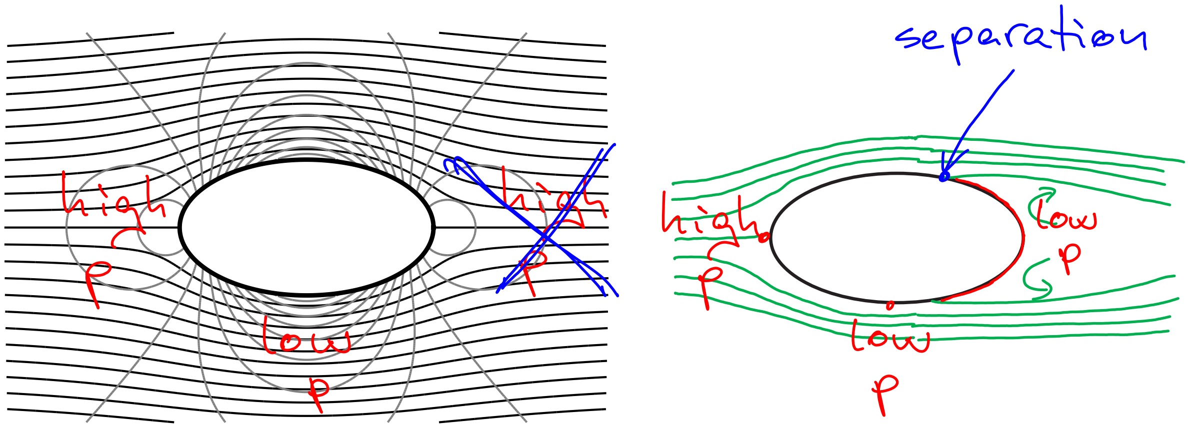
There is a competition between momentum transfer from the free stream, which resists flow reversal, and the adverse pressure gradient, which enhances flow reversal. Turbulence *increases* momentum transfer from the free stream, and therefore it makes the boundary layer *more resistant* to adverse pressure gradients.



laminar boundary layer　　　　turbulent boundary layer

9.6 Effect of separation on pressure distribution

Downstream of separation points, the flow in recirculation regions is usually much quieter than upstream, and can comparatively be considered stationary. The pressure distribution on the wall in these regions is then roughly constant, and it's value is approximately that at the separation point, because the pressure gradient across the separated streamline is finite. This results in the pressure in the back of obstacles not recovering fully to its corresponding high value for the idealised inviscid solution. As a result, a suction force is generated in the wake, giving rise to *pressure drag* or *form drag*.



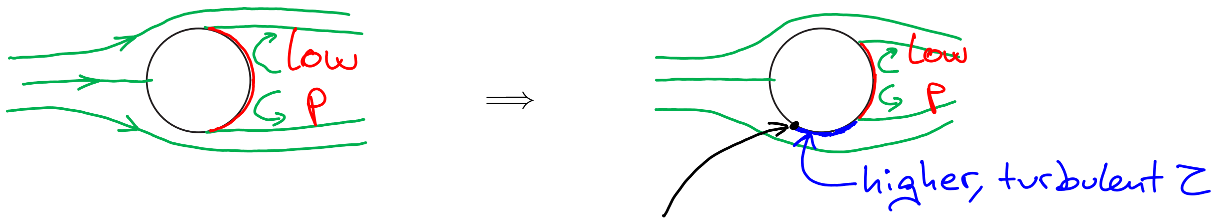
inviscid flow:
- no boundary layers

real viscous flow:
- boundary layers separate

Given that turbulent boundary layers can delay separation, and thus produce smaller separated regions and wakes behind obstacles, they can mitigate the above lack of pressure recovery. Therefore, an obstacle with turbulent boundary layers will have less form drag than the same obstacle with laminar boundary layers, and triggering turbulence can *reduce* form drag. This is why golf balls are dimpled. We will examine this further in Lecture 10.

laminar boundary layer

turbulent boundary layer

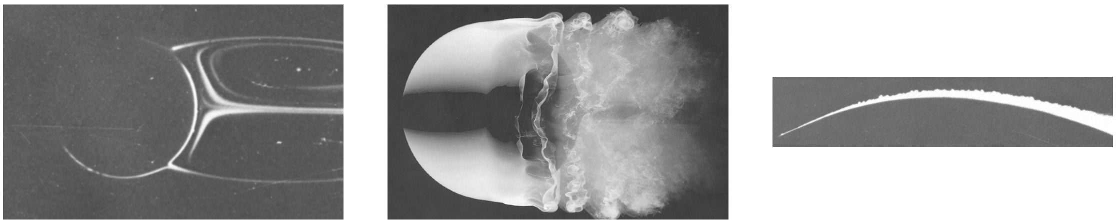


(Watch 9.5 Videos 1 and 2 on Moodle)

early transition, low-p back region is reduced

9.7 Comparison of separation and transition to turbulence

Boundary layer separation and boundary layer transition to turbulence are entirely different phenomena. Nevertheless, transition to turbulence often occurs in boundary layers that have just separated, and this can cause some confusion between the two.



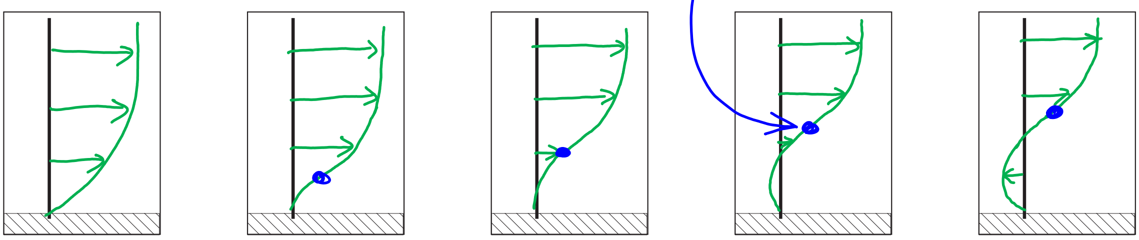
(a) separation without transition

(b) separation immediately followed by transition

(c) transition without separation

Transition to turbulence rapidly occurs in laminar boundary layers that have just separated, because their velocity profiles contain an inflexion point (i.e. at which $\partial^2 v_x/\partial y^2 = 0$) which makes them inherently unstable. In the presence of adverse pressure gradients, laminar boundary layers are likely to separate, develop an inflexional profile and immediately transition to turbulence.

inflexion points are prone to become unstable ⇒ turbulence

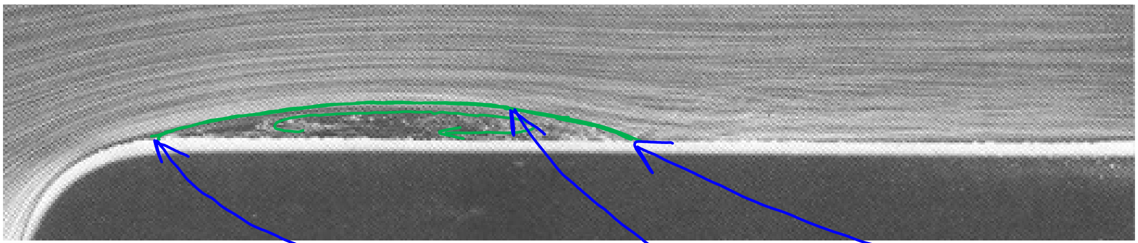


— effect of adverse $\partial p/\partial x$ increases →

In the majority of flows, separation causes almost instantaneous transition to turbulence, as in figure (b) above. However, in very viscous separated flow the viscosity is high enough to damp down any perturbations, and the flow remains laminar. This is the case of figure (a) above. If there is little or no adverse pressure gradient, the boundary layer will become turbulent without undergoing separation, as in figure (c).

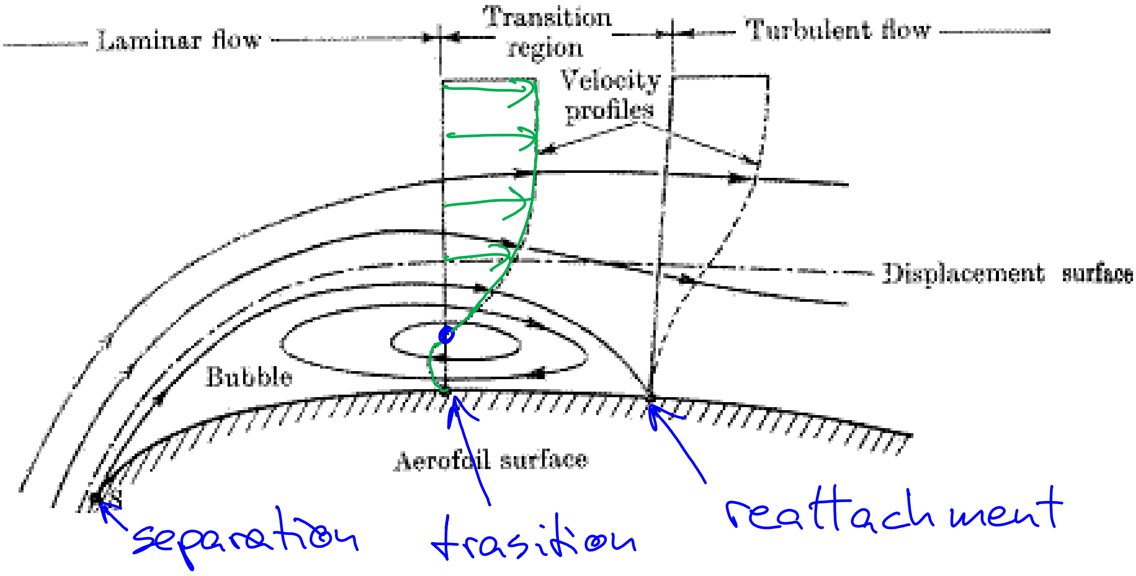
9.8 Boundary layer re-attachment

When a boundary layer separates and subsequently becomes turbulent, the velocity profile becomes fuller, as corresponds to the enhanced momentum transfer of the new, turbulent state. This new profile may be robust enough to overcome the adverse pressure gradient that had originally produced the separation, so it will not present flow reversal. As a result, the flow will *re-attach*. The whole process results in a small separation bubble



laminar boundary layer
+
adverse pressure gradient ⇒ separation ⇒ transition ⇒ reattachment

the boundary layer becomes turbulent, i.e. more robust



Separation bubbles can often be observed at the leading edge of wings at high angle of attack. Aerofoil shapes are often designed to insure that, once separated, the flow transitions to turbulence quickly, so that the separation bubble is short. If the transition occurs too far downstream, the bubble will cover a large area of the wing, or the flow may not even be able to re-attach at all. This can be a very dangerous in-flight situation, because the wing could suddenly stall and lose most of its lift force.

Engineering Tripos 1B

Paper 4

Fluid Mechanics

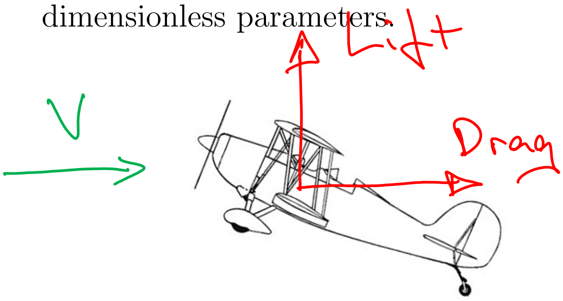
Lecture 10 - External flows and drag

- Forces on solid bodies – lift and drag
- External flows at different Reynolds numbers
- Use and limitations of the inviscid model
- Boundary layer separation and drag reduction
- Unsteady effects

Ricardo Garcia-Mayoral r.gmayoral@eng.cam.ac.uk

10.1 Lift and drag

When a fluid flows around an object (or the object moves through the fluid) the flow exerts a force on the object. We call these external flows, as opposed to the internal, confined flows which are typical for instance of pipes, and which we studied in Lectures 6 and 7. We are usually interested in the components of the force exerted in the direction of the free-stream velocity –the drag– and that perpendicular to it –the lift. In Lecture 5, sections 6 and 8, we saw that these two forces can be expressed in dimensionless form, so that they depend only on a minimum set of dimensionless parameters.

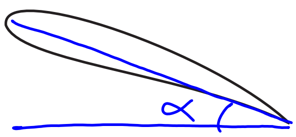


$$C_L = \frac{\text{Lift}}{1/2\,\rho V^2 A}$$

$$C_D = \frac{\text{Drag}}{1/2\,\rho V^2 A}$$

$\left(A \equiv \text{wing surface area} \right)$

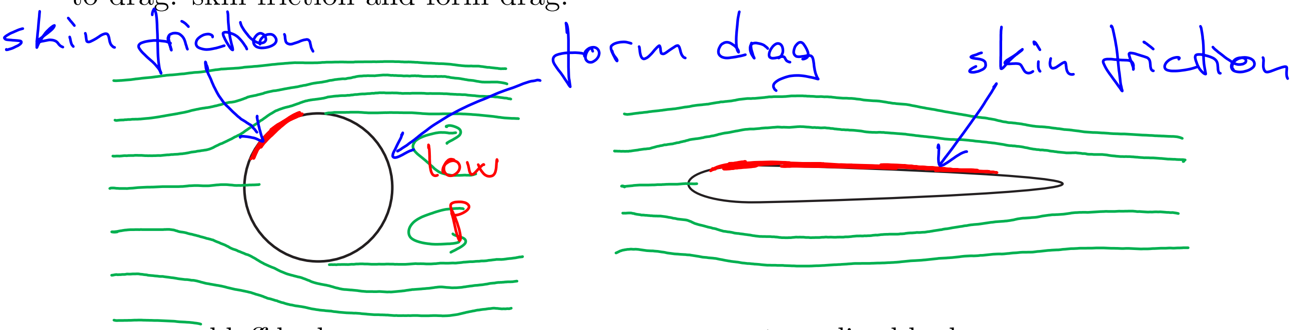
If the flow is incompressible (i.e. at low Mach number) the lift and drag coefficients, C_L and C_D are functions of the shape of the object, its surface roughness, the angle of attack and the Reynolds number.



$$C_L, C_D = f(Re, \cancel{M}, \alpha, \text{geometry}, ...)$$

$M << 1$ (incompressible)

In Lectures 8 and 9 we have seen that real fluids with non-zero viscosity produce boundary layers near solid surfaces, and that these generate frictional shear stress. We have also seen that when a flow separates, a low pressure region forms behind the separation point. These two phenomena give rise to two different contributions to drag: skin friction and form drag.



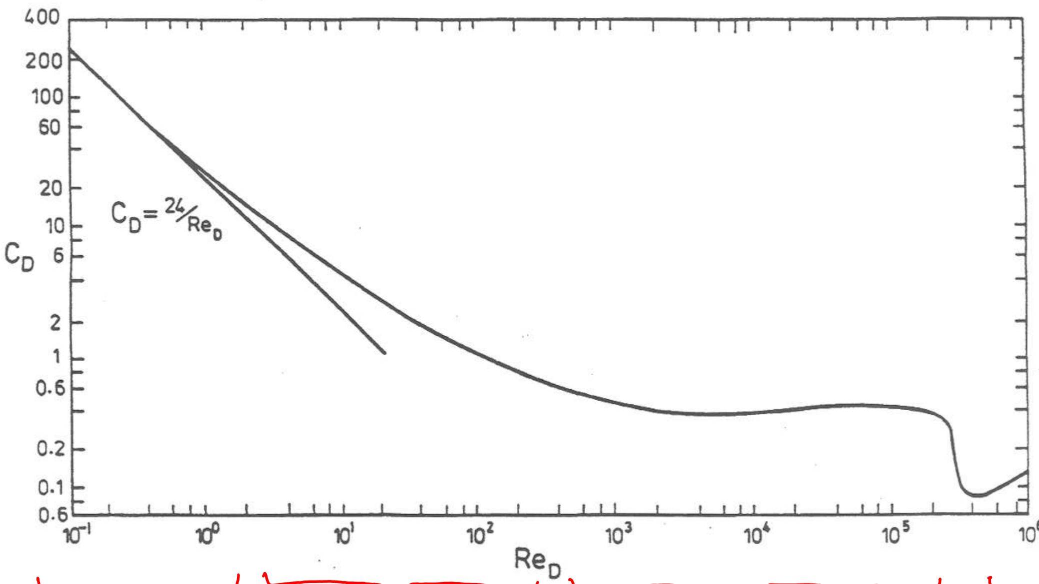
bluff body:
- lower skin friction
- higher form drag

streamlined body:
- higher skin friction
- lower form drag

When the Reynolds number is sufficiently large, form drag usually dominates in bluff bodies, while in streamlined bodies form drag is small and most of the drag is due to viscous friction.

(Watch 10.1 Videos 1-2 on Moodle)

In the following sections, we will illustrate how friction and pressure drag evolve with the Reynolds number, using the example of the flow around a sphere. In Lecture 5, sections 1 and 5, we introduced the corresponding $C_D = f(Re)$ curve, and we will now analyse it in detail. Note that a sphere, because of its symmetry, does not produce a net lift, but this would need to be considered for an asymmetric body.



very low Re
Sec 10.2

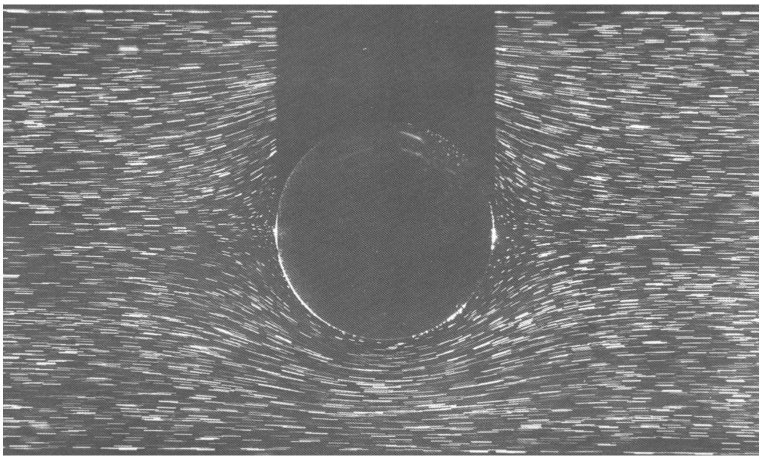
low Re
Sec 10.3

high Re
Sec 10.4

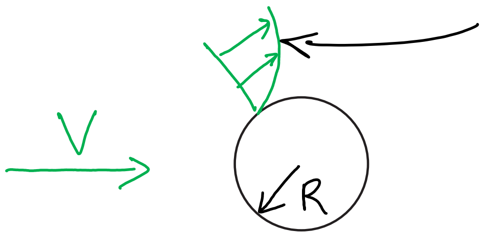
very high Re
Sec 10.5

10.2 Flows at very low Reynolds number ($Re \ll 1$)

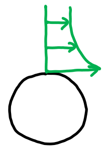
When the Reynolds number is vanishingly small, the flow is perfectly attached and does not separate. This implies that there is no form drag, as that caused by flow separation. The velocity increases with the distance from the surface, and the whole flow resembles a very thick boundary layer.



$Re \ll 1$
$\Downarrow$
viscous forces dominate



very different from inviscid flow, where $V_{shoulder} = 2V$!



The flow around a sphere is governed by Navier-Stokes equations, but at very low Reynolds number, $Re \ll 1$, viscous terms dominate over inertial ones, so we have

$$\rho \underset{\sim}{a} = \underset{\sim}{f}$$

$$\rho \frac{\partial \mathbf{v}}{\partial t} + \rho \mathbf{v} \cdot \nabla \mathbf{v} = -\nabla p + \mu \nabla^2 \mathbf{v}$$

$$0 = -\nabla p + \mu \nabla^2 \underset{\sim}{v}$$

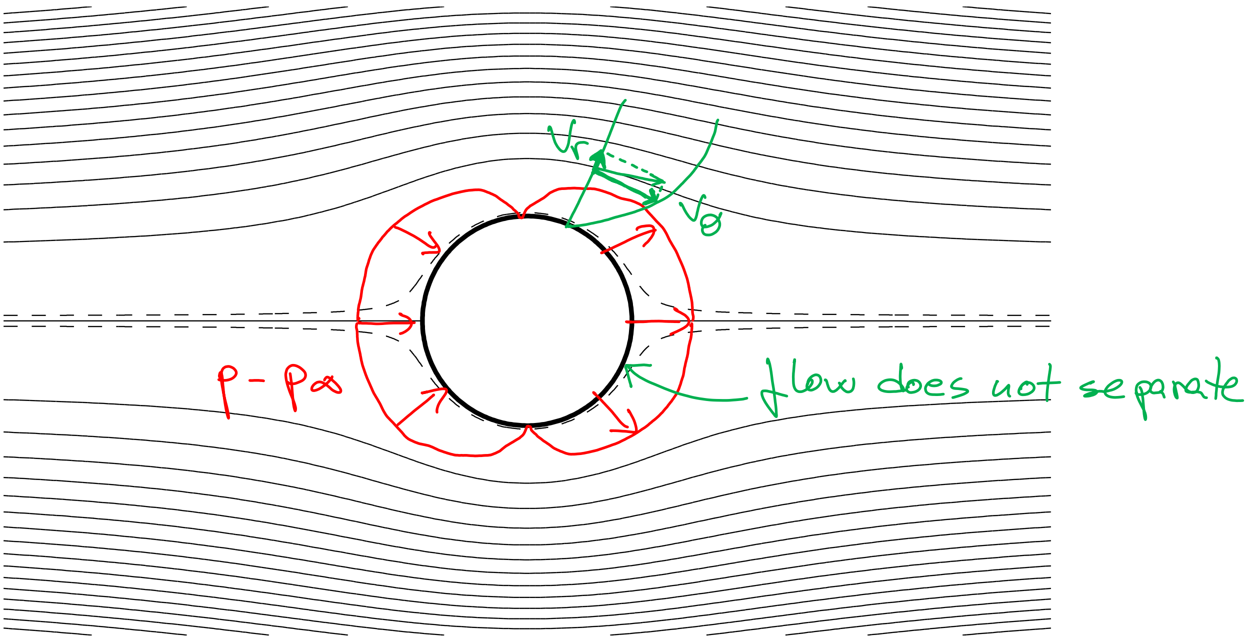
$$\boxed{\mu \nabla^2 \underset{\sim}{v} = \nabla p}$$ Stokes flow

This type of purely viscous flow is known as Stokes or creeping flow. For the flow around a sphere, the Stokes equations have an analytical solution –as they did for laminar flow between two flat plates in Lecture 4. The solution is

$$\frac{v_r}{V} = \left(1 - \frac{3}{2}\left[\frac{r}{R}\right]^{-1} + \frac{1}{2}\left[\frac{r}{R}\right]^{-3}\right)\cos\theta,$$

$$\frac{v_\theta}{V} = -\left(1 - \frac{3}{4}\left[\frac{r}{R}\right]^{-1} - \frac{1}{4}\left[\frac{r}{R}\right]^{-3}\right)\sin\theta,$$

$$\frac{p - p_\infty}{\mu V/R} = -\left(\frac{3}{2}\left[\frac{r}{R}\right]^{-2}\right)\cos\theta.$$



The differential contributions to the drag can be integrated along the sphere's surface, and the resulting total drag is $6\pi\mu RV$. The drag coefficient is therefore

$$C_D = \frac{\text{drag}}{\frac{1}{2}\rho V^2 \pi R^2} = \frac{6\pi\mu RV}{\frac{1}{2}\rho V^2 \pi R^2} = 12\frac{\mu}{\rho V R} = \frac{24}{Re_D}$$

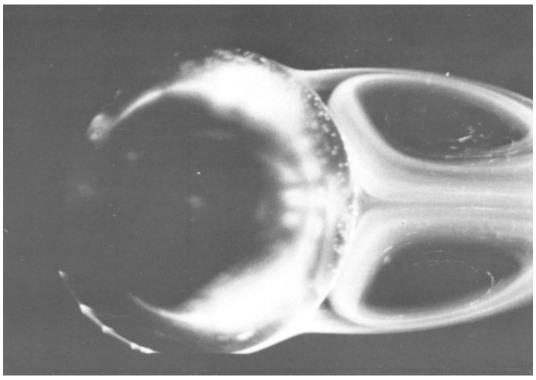
$R = D/2$

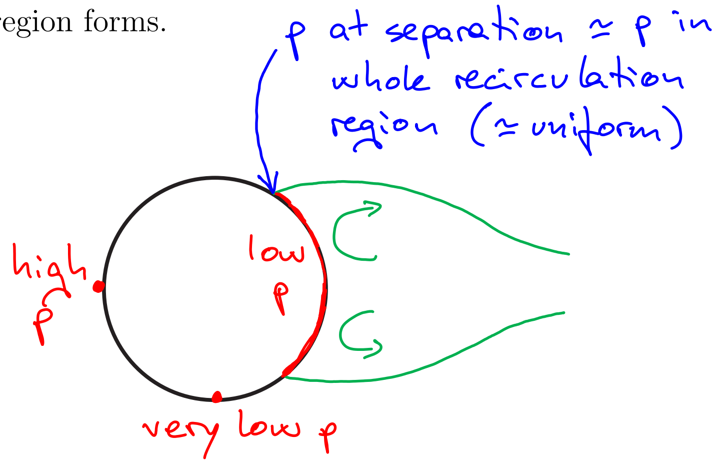
(as shown on figure in next page)

Stokes flows are qualitatively very different from inviscid flows –the most notable difference being that they satisfy no-slip at solid surfaces– even if the streamlines appear to be similar. Stokes flows only exist at very small length scales, or for very viscous flows. These conditions are rarely found in everyday life, so we do not have a good sense for such flows, and their behaviour can be somewhat counter-intuitive.

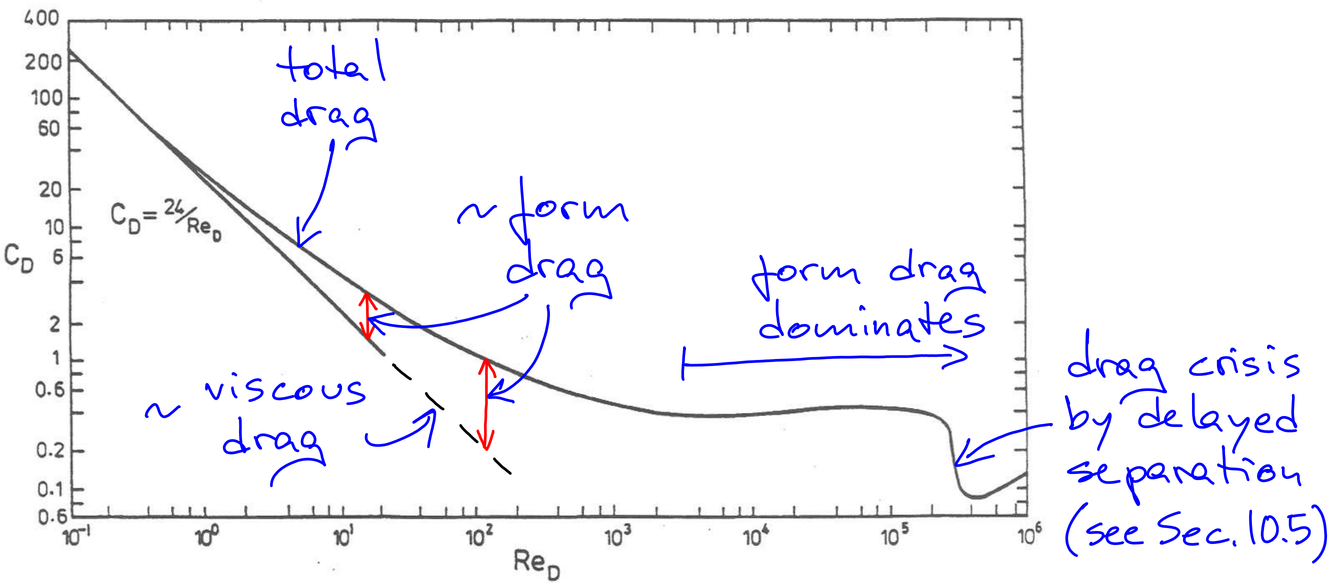
10.3 Flows at moderately low Reynolds number ($Re \approx$ 10-100)

In the flow around a sphere at $Re \approx 100$, the boundary layer separates just behind the shoulder and a toroidal recirculating region forms.





The pressure in the re-circulating region is approximately the same as that at the point of separation, and is lower than the pressure on the corresponding region at the front of the sphere. This is the origin of *form drag*. On a plot of C_D as a function of Re, the total C_D increases relative to $24/Re$ as Re increases. Note that the scale for C_D is logarithmic, so seemingly small deviations imply that the actual C_D may be several times larger than the Stokes-flow C_D.

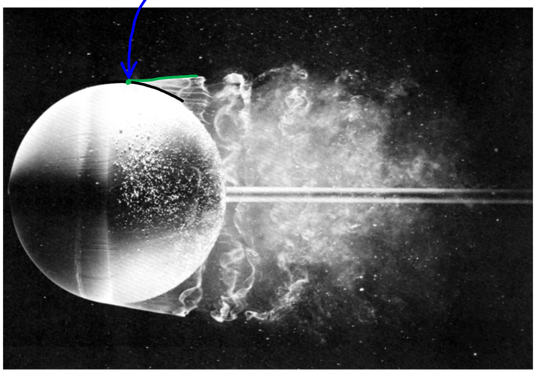


As discussed in Lectures 8 and 9, the point of separation is determined by the competition between the diffusion of momentum from the free stream due to viscosity, which delays separation, and the adverse pressure gradient at the back of the sphere, which promotes separation. As the Reynolds number increases, the relative importance of viscous effects decreases, and the point of separation moves upstream towards the shoulder.

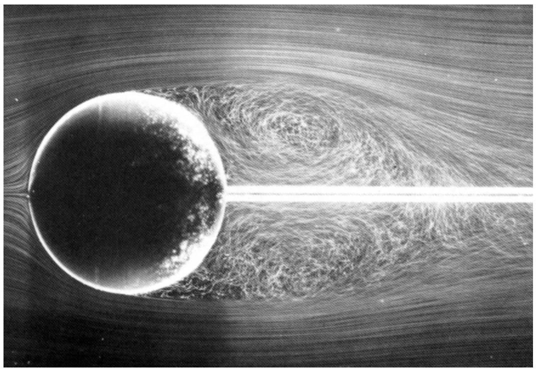
10.4 Flows at moderately high Reynolds number ($Re \approx 10^3$-10^5)

As the Reynolds number keeps increasing, the form drag becomes more and more dominant. Beyond a Reynolds number of $\approx$ 1000, the skin friction is negligible. Between $Re \approx 1000$ and $Re \approx 200000$, the point of separation remains very near the shoulder, and the drag coefficient remains approximately constant at $C_D \approx 0.4$.

separation (Watch 10.4 Video 1 on Moodle)



$Re = 15,000$, instantaneous image



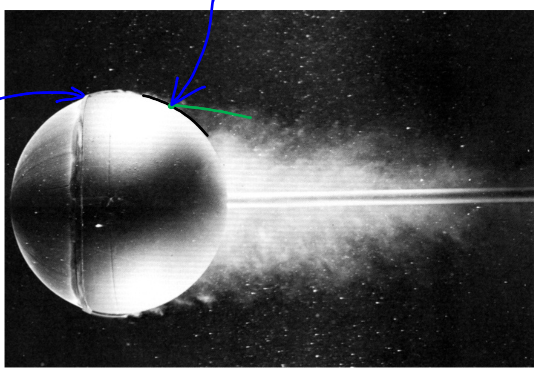
$Re = 15,000$, time-averaged image

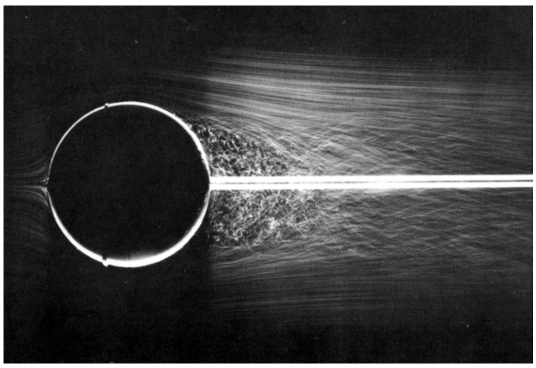
10.5 Flows at very high Reynolds number ($Re \gtrsim 10^6$)

At Reynolds numbers above 200000, the boundary layer becomes turbulent before it reaches the shoulder. Turbulence in the boundary layer increases the transport of momentum from the free stream and therefore delays separation. The separation point moves towards the back of the sphere and the size of the wake reduces. As this happens there is a sudden drop in form drag.

separation moves back ⇒ less form drag

"natural" transition to turbulence

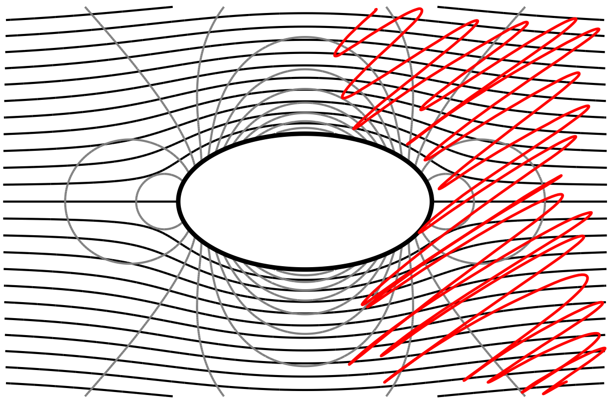




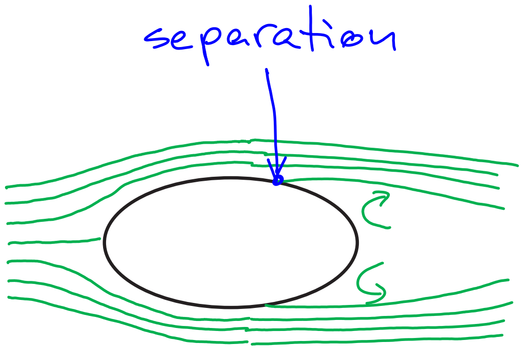
In the above photos, the wind tunnel could not reach a Reynolds number sufficiently high to trigger natural transition, so the boundary layer was *tripped* with a wire to become turbulent.

10.6 Use and limitations of inviscid flow models

By definition, inviscid fluids have no viscosity. Consequently, there is perfect slip with a solid boundary and there can be no boundary layers. If there are no boundary layers, there can be no boundary layer separation, even in the presence of very large adverse pressure gradients.



inviscid flow:
• no boundary layers

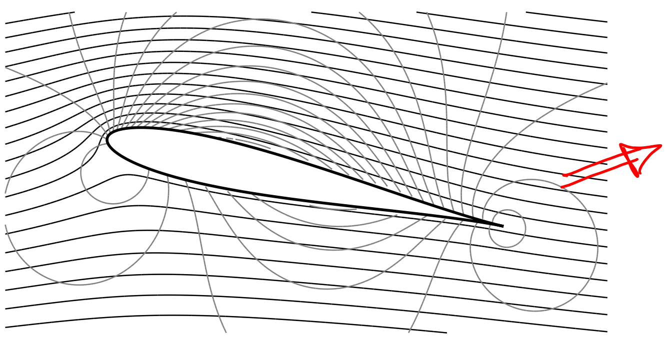


high-*Re* flow:
• boundary layers separate

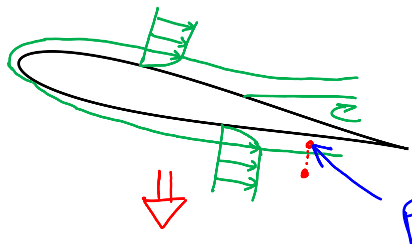
It is tempting to think of inviscid flow as being the solution when the Reynolds number tends to infinity *but this is not the case!* Assuming that a flow is inviscid is only a good approximation if boundary layers (the regions where viscosity counts) are thin and closely attached to the solid surfaces. As soon as a boundary layer separates, the morphology of the external, inviscid-like flow can change completely. Inviscid models therefore only work in regions where the pressure gradients are *favourable* or nearly zero.

However, obtaining the inviscid-flow solution is usually easy, or at least much easier than obtaining the full Navier-Stokes solution, and serves as a first approach to determining the real solution. The resulting pressure gradients can be used to estimate where the flow will separate, and in turn it may be possible to include those separated regions to obtain a new inviscid solution in a modified domain.

• inviscid flow solution:



• calculate boundary layer growth (and separation):



• recalculate inviscid flow

p known from external, inviscid solution

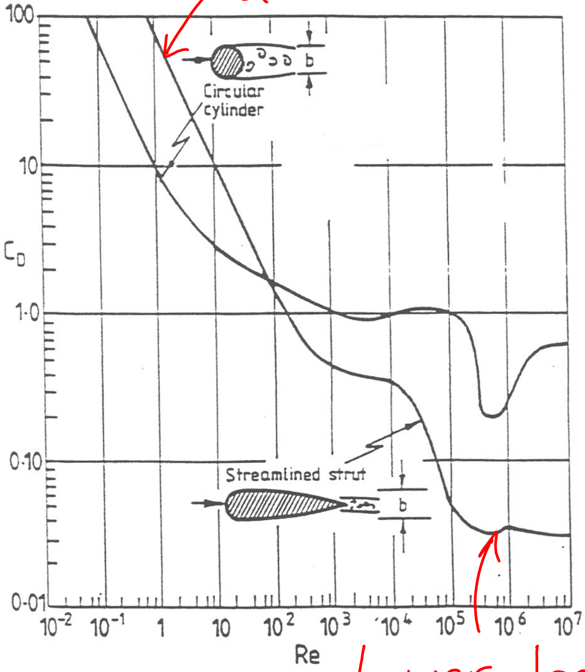
10.7 Drag reduction - streamlining

Form drag increases in proportion to the cross-sectional area of the separated region behind a body and ρV^2. On the other hand, skin friction increases in proportion to the surface area of the body and $\mu V/\delta$:

skin friction drag	form drag
$\sim$ surface area $\times \mu V/\delta$	$\sim$ wake area $\times \frac{1}{2}\rho V^2$

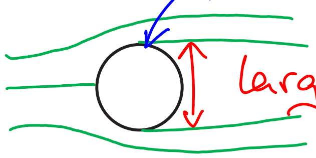
Many important applications, like cars, aeroplanes and ships, are at high Reynolds number, so form drag is much larger than skin friction drag. Therefore, the first priority is to reduce form drag. This is achieved by delaying separation of the boundary layers, which requires adverse pressure gradients to be as gentle as possible.

higher drag at low Re



lower drag at high Re

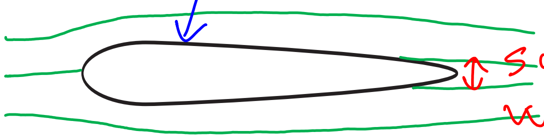
• bluff body: strong adverse pressure gradient



large wake

• streamlined body:

gentle pressure gradient



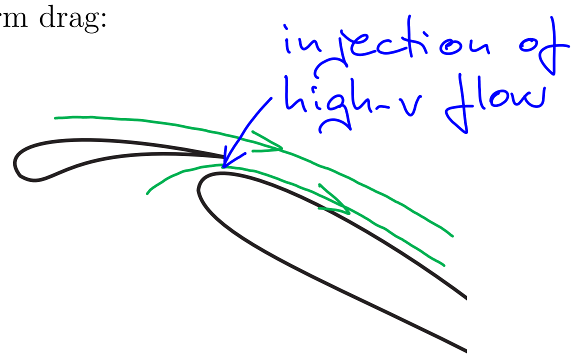
small wake

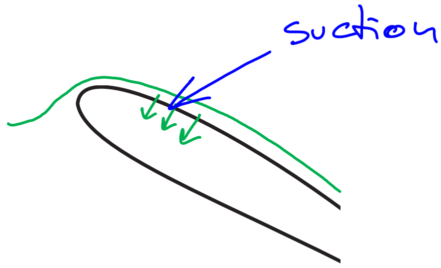
(but more surface with skin friction)

Streamlining often has the side-effect of increasing the skin friction drag. However, this is only influential at low Reynolds number. The drag coefficients for a cylinder and a streamlined strut are shown above (note that C_D for a cylinder is generally greater than for a sphere). The streamlined strut has higher C_D at low Re, when skin friction drag dominates, but lower C_D at high Re, when form drag dominates.

10.8 Drag reduction - other methods

In Lecture 8 we presented two ways to delay boundary layer separation and reduce form drag:

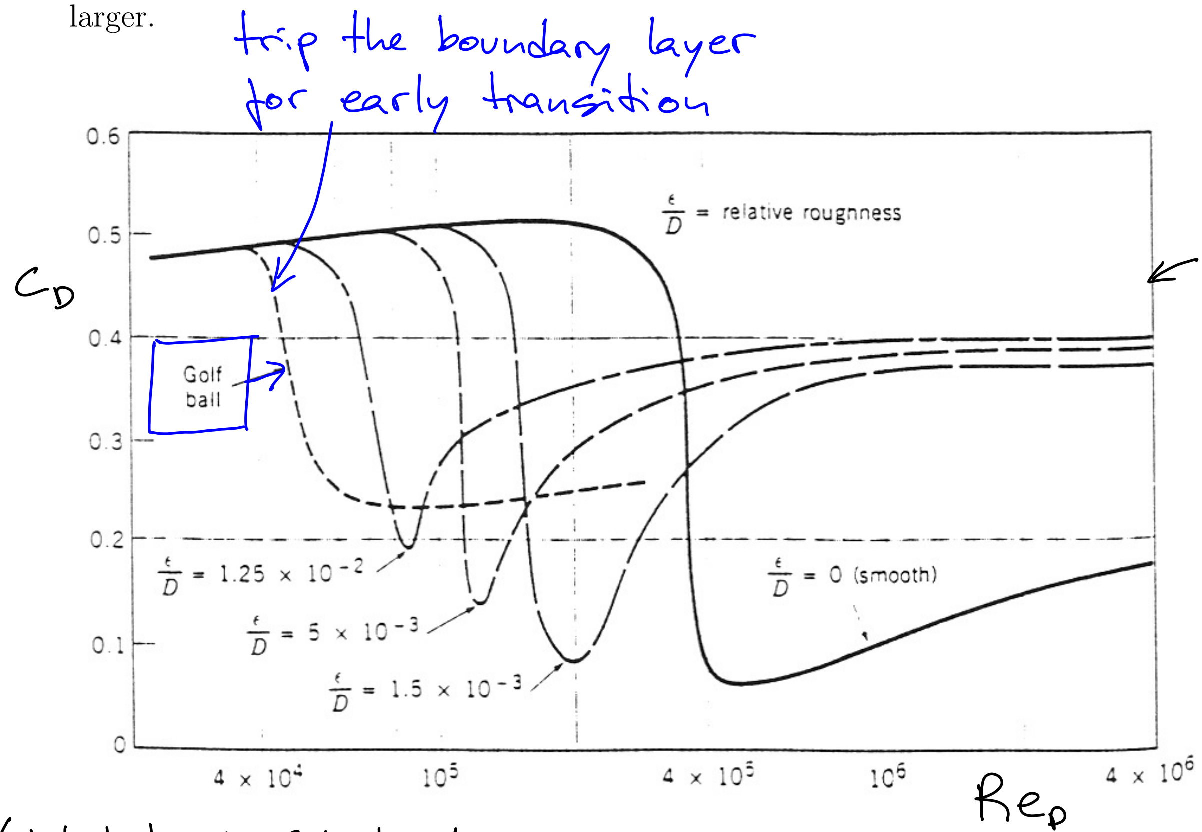




1. Inject high momentum air into the boundary layer. This works well and is the idea behind slats at the front of aircraft wings for high angle of attack operation.

2. Drain away the layer of slow-moving air at the bottom of the boundary layer by sucking it through small holes in the object. This works well but often requires more power than is saved in reducing form drag.

An easier way to reduce form drag is to trigger turbulence in the boundary layer by roughening the surface of the body. This comes of course at the cost of increasing the (now turbulent) skin friction, but the reduction in form drag is typically much larger.

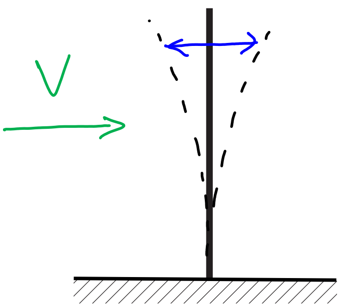


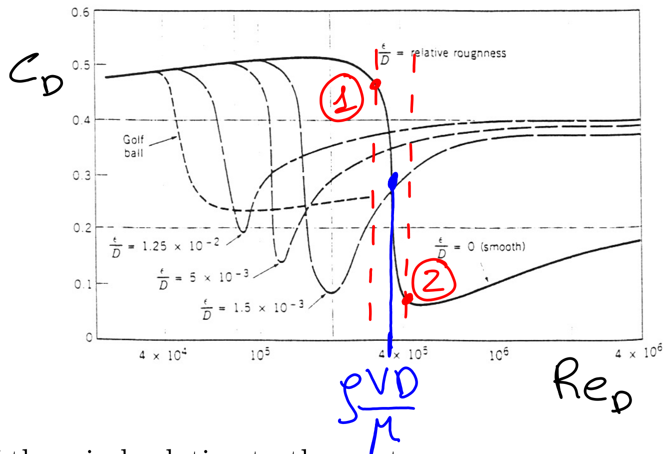
detail of drag crisis in figure on page 6

(Watch 10.8 Video 1 on Moodle)

10.9 Resonance due to laminar/turbulent transition

The sudden reduction of drag for slightly larger velocities due to laminar/turbulent transition in the boundary layer, and the corresponding separation delay, can give rise to a resonance phenomenon. This can sometimes be observed in poles, chimneys and other slender objects, and needs to be addressed to avoid structural failure.

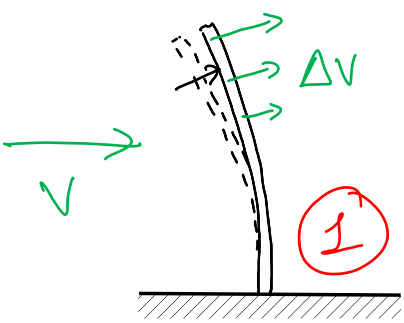


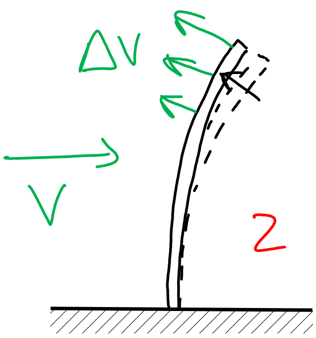


V' = velocity of the wind relative to the post

motion away from wind:

motion towards wind:





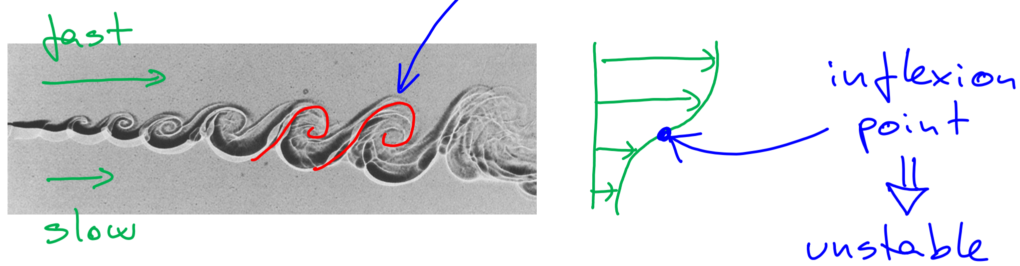
$V_1' = V - \Delta V < V$ $\qquad$ $V_2' = V + \Delta V > V$

lower Re ⇒ higher C_D $\qquad$ higher Re ⇒ lower C_D

If a pole under a uniform free stream is allowed to rock back and forth, its velocity relative to the flow may oscillate enough that the boundary layers are laminar or turbulent at different times of the oscillating cycle. When the pole advances against the wind, its velocity is higher, boundary layers are turbulent, the separation region is reduced and form drag is smaller. In turn, when the pole moves back with the wind, its velocity relative to the flow is smaller, boundary layers are laminar, the separation region is increased and form drag is larger. As a result, the rocking motion is enhanced, in a similar fashion as propelling oneself in a swing.

10.10 Flow instability and vortex shedding

When boundary layers separate, they create a *shear layer*. Shear layers are inherently unstable because they have inflexion points in their velocity profiles. They develop waves that roll up into *Kelvin-Helmholtz* vortices.

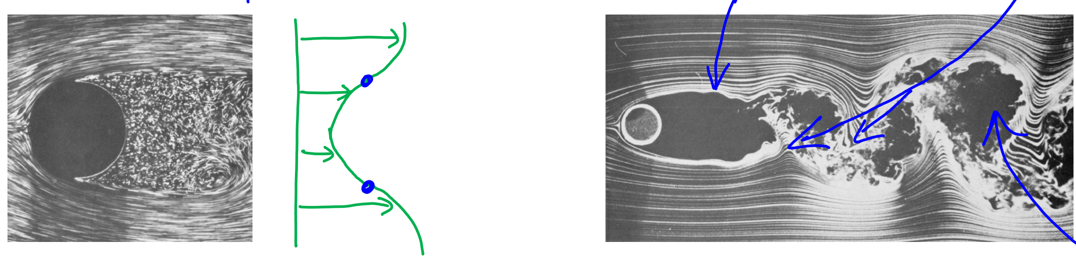


There are two approximately-parallel shear layers behind a bluff body such as a cylinder. They feed back on each other and resonate, and the resulting flow is even more unstable than a single shear layer. The shear layers start by snaking up and down together, and soon they roll up into vortices that are shed alternately from each side of the cylinder. This is known as *vortex shedding*.

two inflexions, very unstable

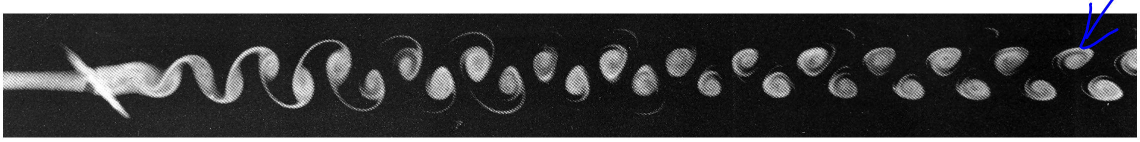
single mode, K-H

dual mode, lock in



shed vortices

The vortex shedding frequency, f, is a function of the velocity of the flow, V, and the distance between the shear layers, D. Experimentally one finds that the *Strouhal number*, fD/V, is approximately 0.2 at moderate and high Reynolds numbers. Vortex shedding also has important consequences for slender structures such as chimneys, particularly if the frequency of vortex shedding matches the resonant frequency of the structure.



constant frequency, $\frac{fD}{V} \simeq 0.2$

Answer S1, Q10 and Q11 in Examples Paper 2

IB-Fluids Lecture Feedback

R. Garcia-Mayoral

I would be very grateful if you could answer the following questions and return the form to me. Your answers will help me to keep improving the course. I am particularly interested in any suggestions you may have.

(please cross the most appropriate box)

	Strongly Disagree	Disagree	Indifferent	Agree	Strongly Agree
The subject matter is interesting					
The handouts cover the material adequately					
The lectures are useful					
The demos in the lectures are useful					
The online support material is useful					
Overall, the course was successful					

	Too fast	A bit fast	**OK**	A bit slow	Too slow
Pace					

	Too hard	A bit hard	**OK**	A bit easy	Too easy
Difficulty (Lectures)					
Difficulty (Examples papers)					

Comments / Suggestions:

Thanks! RGM