PX264: Physics of Fluids

Written by Matthew Bates
January 2010

Contents

1 Introduction **2**
1.1 Basic Terms 2

2 Equations of Motion **4**

3 Streamlined Flow **7**

4 Hydrodynamics of Viscous Flow **8**

5 Turbulence **9**

6 Irrotational Flow **10**

7 Real Flows **12**

8 Examples **12**

Disclaimer

This revision guide has been put together using the sweat, blood and tears of MathsPhys Society members. It is intended as a revision tool and as such does not contain the entirity of the module, only the key elements. **It was not compiled, nor ratified by the University and most certainly not collated by any past, current or future lecturer of the afore mentioned course. As such it may be incomplete, may contain slight inconsistencies and although the utmost care has gone into ensuring they are kept minimal, slight inaccuracies.** Should you find any of the above mentioned please accept our apologies and find the time to drop us an email at the following address so we can change it in future editions, exec@warwickmathsphys.co.uk.

First edition written by Matthew Bates in January 2010. Typeset by Bianca Tayler

1 Introduction

Fluid mechanics is the study of how fluids move and the forces on then - or so says Wikipedia. The "Physics of Fluids" is a second year module that covers elementary fluid mechanics without any difficult mathematics or confusing concepts. It is widely considered as a tough exam but hopefully this revision guide will help you with that. Obviously a revision guide is only supposed to be used for *revision* and you should not expect to sail through the exam without having attended the lectures, this should go hand in hand with trying the past exams. Any problems you come up with, this guide should be able to give you an understanding of what a question means or why such an answer is applicable or not. Best of luck!

1.1 Basic Terms

To model a fluid, we imagine that it is made up of many minute fluid elements which can then be manipulated mathematically to model how the fluid moves as a whole. To do this we need to consider how realistic a model we can make of these small "cubes". Clearly they need to be larger than the diameter of a molecule in the fluid (d_0), how can you sensibly define the macroscopic force on the inside of a molecule, or what is it's temperature? If the box is smaller than the mean free path within the fluid:

$$\lambda = \frac{1}{n}\left(\frac{1}{\pi R^2}\right) \tag{1}$$

Where n is the density of the particles, R is the radius of one particle.

If the box is smaller than λ, then the definition of temperature and density of the fluid element is still difficult to define, so this defines a lower limit on the size of cube we can model with. If the volume element is too large then macroscopic effects can affect the properties of the fluid element (if the length of one side were several kilometres the density of fluid at the bottom would be much greater than that at the top).

This is all defined in the **CONTINUUM HYPOTHESIS** : "In simple fluid mechanics we can extrapolate the properties of the system as L (the length of one side of the cube of fluid we are modelling) tends to 0 and ignore molecular effect." I personally find it easier to visualise this (as in Figure 1) than to learning that sentence by rote.

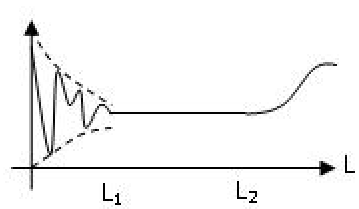

Figure 1: The Continuum Hypothesis

In this module we model **IDEAL FLUIDS** or **SIMPLE FLUIDS** (condensed matter, visco-elastic systems) which must be incompressible and have no changes of density by definition. We also say that there are two different kinds of forces to consider, **LONG RANGE FORCES** (also know as **BODY FORCES** or **VOLUME FORCES**) which are forces across all of the fluid elements (for example gravity), we also say that the interactions between different volume elements cause **SURFACE FORCES**. Surface forces are important in a fluid, if we compare what happens to a solid when we apply a tangential force to it's surface to that in a fluid we see why. A solid will retain it's shape and slide (or roll) in the direction of a force, not so with a fluid! The fluid will **CONTINUOUSLY DEFORM**, that is to say that it will stretch but not tear. A mathematical way of expressing this is by using the equations below:

$$\tau_{xy} = \mu \frac{de}{dt} = \mu \frac{du_x}{dy} \tag{2}$$

$$e = \frac{dX(y)}{dy} \sim \theta \tag{3}$$

Where τ is the shear stress, μ is the fluid viscosity and e is the deformation. This is the definition of a **NEWTONIAN FLUID**! In plain English, this says that the fluid continues to flow regardless of the forces acting on it.

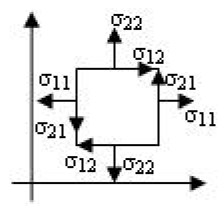

Figure 2: Surface forces on a fluid element

Figure 2 shows the surface forces acting on a fluid element, note that for each surface there is a tangential and perpendicular component of the force, these can be summarised in the surface force tensor.

$$\Sigma : \underline{\Sigma}(\underline{\hat{n}}) := \Sigma_i = \sigma_{ij} n_j = (\Sigma_i(\underline{\hat{a}})\hat{a}_j + \Sigma_i(\underline{\hat{b}})\hat{b}_j + \Sigma_i(\underline{\hat{c}})\hat{c}_j) n_j \tag{4}$$

Where σ_{ij} is the ith component of the force acting in the direction j and n is the unit vector normal to the surface. The tensor Σ is symmetric, i.e $\sigma_{ij} = \sigma_{ji}$. Note $\Sigma(-n) = -\Sigma(n)$.

There are several other terms to define and be learnt:

- **PATH LINE** - the name for the trajectory of fluid particles.
- **STREAK LINE** - the locus connecting all the points that have moved from the same origin.
- **STREAM LINE** - the locus of points where the line is always tangential to velocity. We get a characteristic equation for a streamline which can be written as:

$$\frac{dx}{u_x} = \frac{dy}{u_y} = \frac{dz}{u_z} \tag{5}$$

- **FLUX TUBE** (or **STREAM TUBE**) - defined from considering all streamlines from an initial volume moving to a second, any fluid element that starts within the flux tube must end within it (as its stram line cannot exit it).
- **INVISCIVE** - no viscosity, i.e $\mu = 0$
- **ISOPOTENTIAL** and **ISOCHOR** - constant potential (ϕ) and constant pressure (p) respectively.

- **VISCOSITY** - two types, static (μ) and dymanic (ν), are connected using density (ρ) by:

$$\mu = \rho\nu \tag{6}$$

You will need to know these terms and understand the concepts behind them as each has possible synonyms which may turn up in exam questions and if you dont know what the question is asking you cant answer it! The figure 3 below gives a visual aid for some of these definitions.

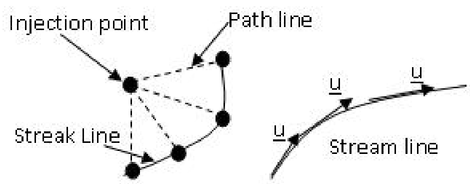

Figure 3: Pictorial descriptions of some of the concepts

2 Equations of Motion

The continuity equation for an incompressible fluid (i.e. density, $\rho = 0$) is a simple rule which is very easily derived it states:

$$\underline{\nabla} \cdot \underline{u} = 0 \tag{7}$$

All this means is that the divergence of the velocity field is 0, i.e the fluid cannot be packed into one point or be drawn from one point. The derivation follows the simple argument that: if we imagine a fixed, closed surface S with outward normal $\underline{n}$, then the amount of fluid flowing through S must be $\underline{u} \cdot \underline{n}$ where $\underline{u}$ is the velocity of the fluid and so the amount of fluid leaving a small area of δS is $\underline{u} \cdot \underline{\delta} Sn$. Thus the total fluid moving into the volume V enclosed by S is:

$$\int_S \underline{u} \cdot \underline{n} \; dS \tag{8}$$

Which by the divergence theorem, is the same as:

$$\int_V \underline{\nabla} \cdot \underline{u} \; dV \tag{9}$$

But since this is an incompressible fluid, this must be equal to 0. Since this condition holds throughout the fluid, let us assume that at some point $\underline{\nabla} \cdot \underline{u} < 0$, if this is true, and $\underline{u}$ is a continuous function, then since our surface S and the enclosed volume V are completely arbitrary, without loss of generality we can assume that V is simply a small sphere just enclosing this point and a small area around it. But this would make out intergral > 0 and since, by the same argument $\underline{\nabla} \cdot \underline{u}$ cannot be smaller than 0 anywhere it must mean that the integrand is 0, i.e $\underline{\nabla} \cdot \underline{u} = 0$ at all points in the fluid.

It is also important to define the convective derivative:

$$\frac{DA}{Dt} = lim_{dt \to 0} \frac{[(A_x dx + A_y dy + A_z dz)\underline{A} + \partial_t \underline{A}]dt}{dt} = (\partial_t + \underline{A} \cdot \underline{\nabla})\underline{A} \tag{10}$$

Where A is some property of the fluid. This concept is important in fluid mechanics as stream lines are not necessarily time independent and so rather than simply differentiating the velocity of a particle, the entire **VELOCITY FIELD** associated with the fluid element must be differentiated.

Flow is another important concept in fluid mechanics:

$$Q = \frac{dM}{dt} = \rho A u \tag{11}$$

Which in words is the rate of change of mass with respect to time, or the density multiplied by the cross surface area multiplied by velocity.

Using the concept of mass conservation it is easy to see that given two different imaginary surfaces (as in figure 4) within the fluid through which fluid is flowing, the flow through each must be the same, as the mass of fluid must be conserved between the two (it cannot be destroyed and no new fluid can be created) and so the following condition, known as the **STEADY FLOW CONDITION**, holds:

$$\rho_1 A_1 u_1 = \rho_2 A_2 u_2 \tag{12}$$

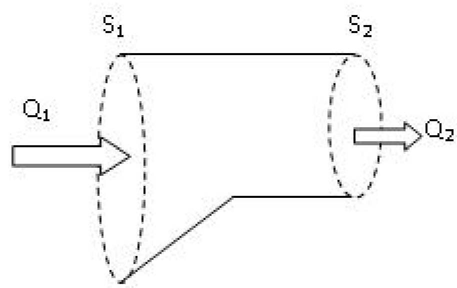

Figure 4: The pictorial continuity equation

If we make a stricter condition on the continuity equation and say that the fluid is also incompressible then the density throughout the fluid can be taken as constant (we assume that this is an ideal fluid with no temperature variations etc.) and we get the relation:

$$u_2 = \frac{A_1}{A_2} u_1 \tag{13}$$

As was mentioned in the introduction there are two kinds of forces, long range and surface forces. A force acting across a surface element δS is defined to be

$$F = p\underline{n}\delta S \tag{14}$$

Where p is more commonly known as the **PRESSURE** and is a funtion of position and time [i.e $p = p(a, y, z, t)$], as this is the force acting ON the surface and not the force FROM the surface the unit normal here is actually pointing into the fluid element from the one applying the force. Using this relation and a corollary of the divergence theorem we can get the relation:

$$-\int_S p\underline{n}dS = -\int_V \nabla \cdot p dV \tag{15}$$

Where the negative sign arises due to the definition of the directionality of p. We also define pressure as:

$$\frac{dE}{dt} = F_1 u_1 - F_2 u_2 = p \tag{16}$$

Where E is energy and is defined thus:

$$E = \int dV \left[\rho \frac{u^2}{2} + \rho g z + W \right] \tag{17}$$

Where the first term on the right hand side corresponds to kinetic energy, the second to the gravitational potential energy and the W term takes account of any other thermodynamic energy that may be present.

As pressure clearly changes with density (the deeper into the fluid you go, the more mass there is above you and so the greater the force) it is necessary to know the relation between pressure and depth, which is:

$$\Delta p = \rho g h \tag{18}$$

With ρ being the (constant) density of the fluid, g the acceleration due to gravity and h the difference in height between the two points for which you wish to know Δp

Before it is possible to derive the important equations of fluid dynamics it is necessary to first know how to define a general force:

$$\underline{F} = -\underline{\nabla}\phi \tag{19}$$

Where ϕ is the potential of the force. (e.g. for gravity $\phi = \rho g z$). Now that we are equipped with the definition of both pressure and force, using equation 2.9 we can say that the total force on a fluid element is:

$$(-\nabla p + \rho \underline{g})\delta V \tag{20}$$

Which is the sum of the pressure (surface) and gravity (body) forces. Now using Newton's second law ($F = ma$) we get **EULER'S EQUATION**:

$$\rho(\partial_t + \underline{u} \cdot \underline{\nabla})\underline{u} = -\underline{\nabla} p + \rho \underline{g} \tag{21}$$

Note that the acceleration is not the conventional $\frac{du}{dt}$ but the convective derivative: $\frac{Du}{Dt}$. This is Euler's equation for an inviscive fluid with no thermal conduction, it is a vector equation and so can be written, after dividing through by ρ, in components as:

$$\frac{\partial u}{\partial t} + \underline{u}\frac{\partial u}{\partial x} + \underline{v}\frac{\partial u}{\partial y} + \underline{w}\frac{\partial u}{\partial z} = -\frac{1}{p}\frac{\partial p}{\partial x} \tag{22}$$

$$\frac{\partial v}{\partial t} + \underline{u}\frac{\partial v}{\partial x} + \underline{v}\frac{\partial v}{\partial y} + \underline{w}\frac{\partial v}{\partial z} = -\frac{1}{p}\frac{\partial p}{\partial y} \tag{23}$$

$$\frac{\partial w}{\partial t} + \underline{u}\frac{\partial w}{\partial x} + \underline{v}\frac{\partial w}{\partial y} + \underline{w}\frac{\partial w}{\partial z} = -\frac{1}{p}\frac{\partial p}{\partial z} + g \tag{24}$$

Where $\underline{u} = \underline{u}(u, v, w)$ and $g = g(0, 0, g)$. In the **HYDROSTATIC** case (i.e $\underline{u} = 0$) Euler's equation reduces to the relationship:

$$\underline{\nabla}(p + \phi) = 0 \tag{25}$$

This is using the relationship between a force and it's potential field from equation 2.13. Equation 2.17 now tells us the relationship between isochors and isopotentials:

$$p = -\phi + const. \tag{26}$$

Isochors are horizontal planes in the fluid (assuming that our body force, gravity, is acting vertically). While it is not necessary to see the mathematical expression for Newton's second law for a fluid it may be of interest:

$$\int_V dV \rho \underline{a} = \int_V dV \rho \underline{g} + \int_S dS \underline{\Sigma \hat{n}} = F_{Volume} + F_{Surface} \tag{27}$$

If we again consider the conservation of mass passing through a surface S we get the relation:

$$\frac{dM}{dt} = -\int_S \rho \underline{u} d\underline{S} = -\int_V \underline{\nabla}(\rho \underline{u}) dV \tag{28}$$

Where we get the second equality by recognising the $\underline{dS}$ is simply $\underline{n}dS$ and using Stoke's theorem. We can also say that:

$$M = \int_V \rho dV \Rightarrow \partial_t M = \int_V \partial_t \rho dV \tag{29}$$

Where ∂_t means $\frac{\partial}{\partial t}$. We can now equate equations 2.20 and 2.21 to reach the conclusion that:

$$\int_V dV[\partial_t \rho + \underline{\nabla} \cdot (\rho \underline{u}) = 0 \tag{30}$$

And using the same arguments as applied when deriving the incompressibility condition, we get the **MASS CONSERVATION** or **CONTINUITY EQUATION**:

$$\partial_t \rho + \underline{\nabla} \cdot (\rho \underline{u}) = 0 \tag{31}$$

As density and pressure are important concepts in thermodynamics there is another relation that needs deriving. In thermodynamics entropy is a state function depending on pressure and density and so,

$$DS(p, \rho) = \frac{DQ}{T} \tag{32}$$

However, along an adiabat there is no heat flow and so $dQ = 0$, this means that we get the relation:

$$(\partial_t + \underline{u} \cdot \underline{\nabla})S = 0 \tag{33}$$

Known as the **ADIABATIC EQUATION**. It should be realised that in most cases this equation will not need to be used and only the Euler and Continuity equations will be necessary, but as its in the notes it cant hurt to know it!

3 Streamlined Flow

We can define a streamline by the function $f = f(\underline{r}(s))$, this function varies as a function of the vector $\underline{r}$ which itself changes with the variables s. Differentiating with respect to s and using the chain rule we show that:

$$\frac{df(\underline{r}(s)}{ds} = \frac{df}{dx}\frac{dx}{dt}\frac{dt}{ds} + \frac{df}{dy}\frac{dy}{dt}\frac{dt}{ds} + \frac{df}{dz}\frac{dz}{dt}\frac{dt}{ds} \tag{34}$$

Clearly $\frac{dx}{dt} = u_x$, the velocity in the x-direction; if we also define that;

$$\frac{dt}{ds} = \alpha(t) \tag{35}$$

For α being some function of t, then equations 3.1 and 3.2 combine to show that along a streamline:

$$\frac{df(\underline{r}(s))}{ds} = \alpha(\underline{u} \cdot \underline{\nabla})f \tag{36}$$

For Euler's equation (2.15) in a steady flow (i.e velocity is constant so $\partial_t \underline{u} = 0$) we get that:

$$\rho(\underline{u} \cdot \underline{\nabla})\underline{u} = -\underline{\nabla}(p + \phi) \tag{37}$$

Remembering that the force (g) is simply the derivative of a potential (ϕ). Now if we divide through by ρ and use the following identity $(\underline{u} \cdot \underline{\nabla})\underline{u} = (\nabla \times \underline{u}) \times \underline{u} + \underline{\nabla}(\frac{1}{2}\underline{u^2})$, we get the result:

$$(\nabla \times \underline{u}) \times \underline{u} = -\underline{\nabla}\left(\frac{p}{\rho} + \frac{\phi}{\rho} + \frac{u^2}{2}\right) \tag{38}$$

If each side is now dotted with u we see:

$$0 = -\underline{u} \cdot \underline{\nabla} \left(\frac{p}{\rho} + \frac{\phi}{\rho} + \frac{u^2}{2} \right) \tag{39}$$

i.e

$$\frac{p}{\rho} + \frac{\phi}{\rho} + \frac{u^2}{2} = const. \tag{40}$$

This is **BERNOULLI'S THEOREM** and can be used to derive a very important equation for fluid dynamics which we shall do in the next section.

4 Hydrodynamics of Viscous Flow

If we imagine that there is a fluid element with forces acting in the y-direction (as in figure 5), then the cross-sectional area A must be $dxdz$, the force on the uppermost of the planes parallel to the $x - y$ plane must be (by eqn. 1.4):

$$F = A\tau = A\mu\partial_y u_x(y + dy) \tag{41}$$

This is due to the flow being one dimensional and in the x-direction.

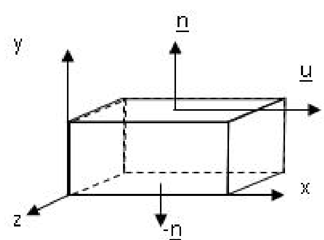

Figure 5: A fluid element with forces acting in the y-direction

Clearly for the lowermost plane the force must be the negative of that for the uppermost and with $u_x(y)$ (see the explanation around eqn. 1.4). If we now take the total force on the fluid particle, using the knowledge that $u_x(y) + \partial_y u_x dy$ we find that in 1D:

$$F = A\mu\partial_y^2 u_x dy = \mu\partial_y^2 u_x dV \tag{42}$$

Which in 3D translates to:

$$\underline{F} = \mu\underline{\nabla}^2\underline{u} + (\xi + \frac{\mu}{3}\underline{\nabla}(\underline{\nabla} \cdot \underline{u}) \tag{43}$$

Where ξ is a second viscosity coefficient and the second term on the right hand side comes from the expression for the **STRESS TENSOR**:

$$\tau_{ij} = \mu[(\partial_i u_j + \partial_j u_i) - \frac{2}{3}\delta_{ij}\partial_k u_k] + \xi\delta_{ij}\partial_k u_k \tag{44}$$

The subscripted i, j and k's represent "Einstein Notation", a concept that you will need to be familiar with for a few 2^{nd} year modules. The idea is that you sum over repeated indices, so $\partial_i u_j$ would represent the sum for i=1,2,3 and $j \neq i$ over $\partial_j u_j$. The derivation for the stress tensor is not necessary for the exam and so shall be omitted. If we combine equation 4.4 with previous descriptions of what a force is, then we achieve the result:

$$\rho(\partial_t + \underline{u} \cdot \underline{\nabla})\underline{u} = \rho\underline{g} - \underline{\nabla} p + \mu\underline{\nabla}^2\underline{u} + \left(\frac{\mu}{3} + \xi\right)\underline{\nabla}(\underline{\nabla} \cdot \underline{u}) \tag{45}$$

Which is called the **NAVIER STOKES EQUATION** and is the sum of the volume, pressere and viscous forces. Note, first term is an acceleration, second term is force due to gravity, third term is pressure, the fourth and fifth terms are viscous forces. Note also that the fifth term does not apply to an incompressible fluid as $\xi = 0$ in this case.

Another important concept in viscous flow is **LAMINAR FLOW**. This is the term for when a fluid flows in parallel layers with no disruption between the layers as opposed to **TURBULENT FLOW** when there are chaotic property changes. We normally expect that a fluid will have laminar flow until it interacts with an object at which point the flow will become turbulent and gradually return to a laminar flow as the distance to the object tends to infinity.

There are some important boundary conditions when dealing with fluids interacting with objects, the most important is that $u_n = 0$ i.e the velocity of the fluid in the perpendicular direction to the surface must be 0 at the surface, this comes about from the continuity of the velocity field and as the fluid cannot travel into the object it must have 0 velocity within and due to continuity, the velocity of the fluid at the surface must be 0 also. This even applies in the case of immiscible fluids, which makes sense as they will not mix so one fluid cannot flow into the other. For immiscible fluids we have the slightly more long winded:

$$u_n^{(1)} = u_n^{(2)}; \sigma_{ij}^{(1)} n_j^{(1)} = \sigma_{ij}^{(2)} n_j^{(2)} \rightarrow [\sigma_{ij}^{(1)} + \sigma_{ij}^{(2)}]n_j = 0 \tag{46}$$

Where the σ's are components of the stress tensor.

We also have the **NO SLIP BOUNDARY CONDITION**: $u_{//} = 0$. That is, for any fluid with $\mu \neq 0$ the velocity parallel with the object must be 0 (in the rest frame of the object). This is obvious as the viscosity of the liquid will cause it to move with the object and so be at rest with the object. If the fluid has no viscosity then we have the **FREE SLIP BOUNDARY CONDITION**, where the velocity at the boundary can be non-zero. At any point on a fluid boundary it pays to stop and think about the boundary conditions, the velocity field of the fluid must be continuous and if the fluid cannot move into a region equation 4.7 holds.

5 Turbulence

To define different types of flow we use what is called the **REYNOLDS NUMBER**, and to derive this important feature of fluids we must use dimensional analysis.

If we take a fluid flowing through a "pipe" with velocity u, a characteristic length scale (see below) R, the pipe having length C, the density of water being ρ, the pressure difference across the pipe being Δp and the fluid having dynamic viscosity ν, then we know the following dimensions:

$$[u] = \frac{L}{T}; [R] = L; [C] = L; [\rho] = \frac{M}{L^3}; [\Delta p] = \frac{M}{LT^2}; \nu = \frac{L^2}{T} \tag{47}$$

Where L=length, T=time and M=mass. We then say that the dimensionless quantity π is:

$$\pi = [u]^a[R]^b[L]^{-c}[\rho]^d[\Delta p]^e[\nu]^f \tag{48}$$

Where the index of L is "$-c$" because we know where the derivation is going and it stops any trouble with minus signs later on. This relation tells us that:

$$[\pi] = L^{a+b-c-3d-e+2f}T^{-a-2e-f}M^{d+e} \tag{49}$$

i.e we have the three simultaneous equations:

$$a + b - c - 3d - e + 2f = 0; -a - 2e - f = 0; d + e = 0 \tag{50}$$

As is always the case with dimensional analysis, we have control over one of the variables and so choose to investigate the case where $u = 1$. We can also choose to neglect Δp as this will make solving the equations easier. i.e $e = 0$ also. Now we get:

$$f = -1; b - c = 1 \tag{51}$$

From the third and first line of eqn. 5.3 respectively. This gives us two subcases: Case 1 $c = 0$ and $b = 1$ or Case 2, $b = 0$ and $c = -1$. For Case 1 we have that $a = 1$, $b = 1$ and $f = -1$ i.e:

$$Re = \frac{uR}{\nu} \tag{52}$$

Where Re is called the **REYNOLDS NUMBER**. For Case 2, we have $a = 1$, $c = -1$, $f = -1$, i.e

$$\frac{1}{\tilde{R}} = \frac{uL}{\nu} \tag{53}$$

Which says that $\tilde{R} \to 0$ as $L \to \infty$, so this equation is not useful (in a long pipe).

The Reynolds number is sometimes seen written with an "L" instead of "R" which is confusing, but actually makes sense when thinking that in this case, "L" would represent the **CHARACTERISTIC LENGTH** of the situation to which the formula is applied. The length is specific to each problem, so a person swimming would have a characteristic length of 1.5m, say i.e their height; stirring a cup of tea would give a characteristic length of the radius of the tea cup etc.

The Reynolds number is used to characterise how the fluid will behave, for $Re << 1$ we have **LINEAR VISCOUS** flow, for $Re \sim 1$ we have the full Navier Stokes Equations and laminar flow, with $Re > 1$ we egt laminar flow characterised by the Euler equation. For $Re \leq 2 \times 10^3$ we get turbulent flow.

As well as deriving Re from this dimensional analysis, if we now consider the case where $u = 0$ and we can set the exponent of Δp i.e $a = 0$, $e = 1$. This gives us that $d = -1$, $f = -2$ and $b - c = 2$ so we set $c = 1$ as we want to have $\frac{\Delta p}{L}$ and this gives us $b = 3$ and the relation:

$$N_p = R^3 \frac{\Delta p}{L} \frac{1}{\rho \nu^2} \tag{54}$$

The Navier Stokes equation tells us that this is a funtion of the Reynolds number and so we can apply a Taylor expansion to it, to get:

$$N_p = f(Re) = a_0 + \alpha Re + \beta Re^2 + ... \tag{55}$$

But it can be shown that $a_0 = 0$ and as the higher order terms tend to zero we can simply say that:

$$N_p = \alpha Re \tag{56}$$

If we take equations, 1.6, 5.5, 5.6 and 5.8 we can get the relation that:

$$u = \frac{1}{\alpha} \frac{\Delta p}{\rho} \frac{R^2}{\mu} \tag{57}$$

6 Irrotational Flow

We define the concept of **VORTICITY** to be such that:

$$\underline{\omega} = \underline{\nabla} \times \underline{u} \tag{58}$$

Which is simply the curl of the velocity field. In 2D we get that $\underline{u} = (u_x(x, y), u_y(x, y), 0)$ and so:

$$\underline{\omega} = (0, 0, \partial_x u_y - \partial_y u_x) \tag{59}$$

For a 2D solid body rotation anticlockwise about a point **VORTEX** at the origin, we define that the velocity field:

$$\underline{u} = \Omega \underline{r} \underline{\hat{e}}_y \tag{60}$$

Where Ω is our amplitude of velocity, $\underline{r}$ is the position vector of the fluid element, $\underline{r} = (x, y, 0)$ as we're in 2D. We also define the **CIRCULATION** as:

$$K = \oint_\Gamma \underline{u} \cdot d\underline{l} = \int_S \underline{\omega} \cdot d\underline{S} \tag{61}$$

This equation is known as **KELVIN'S CIRCULATION THEOREM**, where Γ is a closed contour within the fluid. We also say that:

$$\dot{K} = v \oint \underline{\nabla}^2 \underline{u} d\underline{l} \tag{62}$$

Which holds the obvious condition that K is constant is $v = 0$. If we have the case where $K = 0$ if and only if:

$$\underline{u} = \underline{\nabla} \phi \tag{63}$$

Which happens if and only if:

$$\underline{\omega} = \underline{\nabla} \times \underline{u} = 0 \tag{64}$$

If this is the case then the fluid is called **IRROTATIONAL**. Equation 6.6 combined with the continuity equation gives u the **LAPLACE EQUATION**:

$$\nabla^2 \phi = 0 \tag{65}$$

In the case of irrotational flow we must first solve Laplaces equation, then select the correct solution by imposing boundary conditions, we then use eqn. 6.6 and can derive the pressure from **BERNOULLI'S EQUATION FOR IRROTATIONAL FLOW**:

$$\partial_t \phi + \frac{1}{2} \underline{u}^2 + \frac{p}{\rho} + gz = 0 \tag{66}$$

Solutions of the Laplace equation tell us that the potential ϕ (written in cylindrical polar coordinates) of a flow of velocity w going past a cylinder of radius R must be of the form:

$$\phi(r, \theta) = wr \cos(\theta) or, \phi = wr \cos(\theta) \left(r + \frac{R^2}{r} \right) \tag{67}$$

If we neglect viscosity then the shear tensor is equal to zero whihc means that there is no net force acting on a surface element of the cylinder, which gives us the relation:

$$\underline{F} = \int p d\underline{s} = 0 \tag{68}$$

If we consider a steady flow, with velocity w, past a cylinder of radius R then we already know that the velocity is the gradient of the potential and from eqn. 6.10 we know what the potential is. However if the cylinder is also rotating then we have a second potential of $\phi = \frac{K}{2\pi}\theta$ and so the velocity is the gradient of the superposition of these two components, i.e:

$$\underline{u} = \partial_r \phi \underline{\hat{e}}_r + \frac{1}{r} \partial_\theta \phi \underline{\hat{e}}_\theta = \frac{1}{r} \left[-w \left(r + \frac{R^2}{r} \right) \sin(\theta) + \frac{k}{2\pi} \right] \tag{69}$$

If we then evaluate the pressure from the Bernoulli equation at the point $r = R$ then we find that:

$$p(r = R) = const + \frac{\rho w K \sin(\theta)}{\pi R} \tag{70}$$

If we then use eqn. 6.11 with the viscosity included (i.e the integral is non-zero) and realise that $d\underline{s} = LRd\theta\underline{\hat{e}}_r$ we can integrate to get:

$$\frac{F}{L} = -\rho w K \underline{\hat{e}}_y \tag{71}$$

Which is the **KUTTA ZHUKOWSKI THEOREM**

7 Real Flows

When we consider real flows we talk about the concept of a boundary layer which is simply the layer of fluid in the immediate vicinity of a bounding surface. What we find is that on flowing past an object we start to get flow separation (where the boundary layer may not be moving at the same speed and in the same direction as other fluid near to it), which causes drag on an object as the non-ideal flow behind the object does not create the same force as the ideal flow in front of the object.

This is where the concept of aerodynamics becomes important, an aerodynamical shape will cause minimal/no separation, which minimises the amount of drag felt by the body.

Strangely, turbulent flow has less drag than might be expected, we find (as in figure 6) that the drag value decreases slowly with increasing Reynolds number and then suddenly between for an Re between 10^5 and 10^6 we find the **DRAG CRISIS** where the value of the drag suddenly decreases very rapidly. This is due to the chaotic behaviour of the fluid facilitating transport across the boundary layer, we find that this allows for more ideal flow behind the object producing more force on that side and so decreasing the drag.

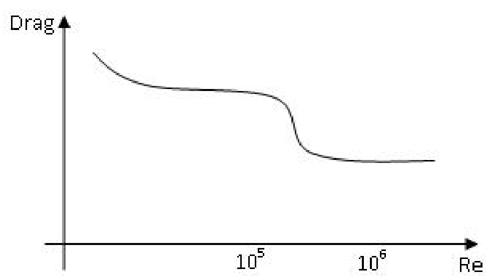

Figure 6: Graph of drag value against Reynolds number

8 Examples

Using the non-slip boundary condition, if we imagine that there are two infinite parallel planes, one stationary at $y = 0$ and one moving with a constant velocity u_x at a height of $y = h$ we can create a velocity profile through the fluid. If we imagine that the height difference between 0 and h is negligible then we can ignore the force due to gravity as it would have no actual effect on the result and just confuse the calculation. Figure 7 shows how the situation looks, due to the translational symmetry along the z axis we can say that $\underline{u}_x = \underline{u}(x, y)$ only. We can enforce a stricted condition by looking at the continuity equation and seeing that $0 = \underline{\nabla} \cdot \underline{u} = \partial_x u_x$ which means that u_x is a function of y only. So from the Navier Stokes equation we get that:

$$0 = -\underline{\nabla} p + \mu \nabla^2 \underline{u} \tag{72}$$

But there is no change in pressure in the y or z direction which maens that: $p = p(x) = p_0$. Which means that: $\mu\nabla^2\underline{u} = \nabla^2\underline{u} = 0$. As the solution is independent of μ, we can solve the differential

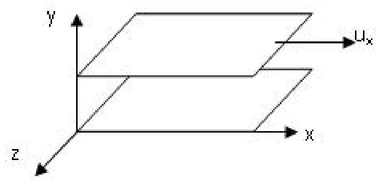

Figure 7: Two infinite parallel planes, one stationary the other moving

equation by inspection to note that $u_x(y) = ay + b$. If we apply the no-slip boundary condition then we see that $u_x(0) = 0$ (the bottom plane is not moving) and $u_x(h) = u_0 = ah$ do the solution is:

$$u_x(y) = \frac{u_0}{h} y \tag{73}$$

And we can see the graphical solution in figure 8.

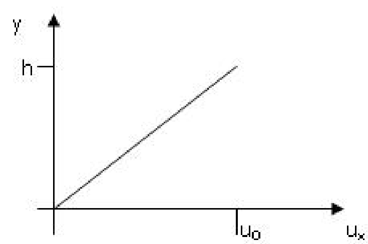

Figure 8: A graph of $u_x(y)$ against y.

Using a Pito Tube (figure 9A)we can measure the velocity of water, and using the Bernoulli equation with $W = 0$ due to no temperature gradients and with $z_1 = z_2$ we can derive: $\underline{u} = \sqrt{2gh}$. This also uses the relation for pressure changes.

A Venturi meter (figure 9B) is used to give the "flow" of fluid through the tube, with A_1 and A_2 being the widths of the tubes in the direction of flow. From this we derive the relation that $Q = \sqrt{\frac{2\rho^2 gh}{\frac{1}{A_1^2} - \frac{1}{A_2^2}}}$

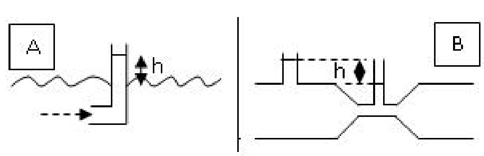

Figure 9: A) A Pito Tube, B) A Venturi Meter

To determine the rate of release of water from a bucket with a hole in it, we use the Bernoulli equation, assuming W is constant and that $\mu = 0$ (we are in steady state) also, assume that the diameter of the hole is significantly smaller than the height of the bucket. Working through all of this (using similar tricks to the Pito meter and the Venturi meter) we determine that the time to empty the bucket is: $t_0 = \frac{2}{\sqrt{2g}} \frac{A}{a} h_0^{\frac{1}{2}}$ where A is the cross-sectional area of the bucket, a is the area of the hole and h_0 is the starting height of the fluid.

We model an aerofoil (see figure 10) by having two different lengths for the wing sections, l and L, the air flowing across the longer section L must cover a greater distance in the same time, thus causing a lower pressure on the top of the aerofoil than is on the bottom, so causing a force pushing upwards on the aerofoil. It can be derived (again using the Bernoulli equation and by calculating the velocities of the two streams of air with respect to the lengths of the sides of the aerofoil) that the total force is: $A\Delta p - Mg = \frac{A\rho}{2}\left[\left(\frac{L}{l}\right)^2 - 1\right]u_0^2 - Mg$ and this can be rearranged to find the minimum velocity (u_0) with which the aerofoil will take off (as the velocity heads above u_0, the force turns from negative to positive meaning there is a net upwards force on the aerofoil).

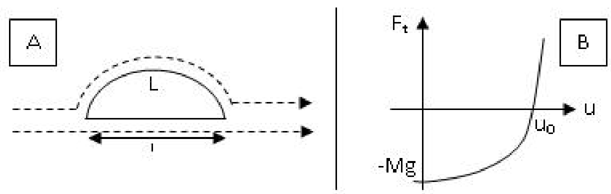

Figure 10: A) An aerofoil, B) Graph of force against velocity on an aerofoil