# Notes on Fluid Dynamics
# and a basic understanding of Flight

Fawad Hassan, Physics Department, Stockholm University

Our motivation for discussing some aspects of fluid mechanics is two fold:

- To understand the basic principles of the mechanics of liquids and gases (fluids). We encounter fluid motion in almost every aspect of our daily lives. However, understanding fluid behaviour and motion is far less intuitive in comparison with solid objects. Hence, often phenomena invloving fluids require careful analysis to determine what physical principles are involved and how they conspire to lead to the observed effect.
- It can help us understand the application of vector calculus in physics with easy to visualize examples. Also, to study fluids, we need to understand some aspects of the mechanics of continuous media which is useful for understanding the more general framework of field theory (both classical and quantum).
- These notes are part of the "avancerad problemlösning I" course. Not everything here is covered in the class, and the level of detail may vary

## 1 General description of Fluids in motion

### 1.1 A fluid as a mechanical system

Superficially, the behaviour of a **fluid** (that is, liquid or gas) is very different from that of a rigid mechanical system, but at a fundamental level both obey the laws of Newtonian mechanics. In Newtonian mechanics one essentially describes the motion of particles under the action of forces. In order to apply Newtonian mechanics to fluids at the macroscopic level (that is, by considering the action of forces on macroscopic fluid elements, and not on the individual fluid molecules) we introduce some notions in terms of which the flow of a fluid can be described:

Consider a small **volume element** $\Delta V$ of the fluid moving along with the fluid. $\Delta V$ is defined such that the molecules it contains remain within it as the volume moves along with the fluid, at least during some time interval. The volume element is taken to be infinitesimally small from macroscopic point of view, but still large enough to contain a very large number of fluid molecules. This will enable us to reliably average over molecular properties (like molecular velocities) giving rise to well defined macroscopic quantities associated with the volume element. The location of $\Delta V$ can be identified as the location of a point $\vec{x}(t)$ within it. The arbitrariness in the choice of $\vec{x}$ within $\Delta V$

becomes negligiable in the limit of $\Delta V \to 0$. Also in this limit, the fluid contained in $\Delta V$ is characterized by its **velocity field** $\vec{v}(\vec{x},t)$, the local **mass density** $\rho(\vec{x},t)$ and the **static pressure** $p(\vec{x},t)$. For an **incompressible fluid**, the density $\rho$ is constant both in space and in time. Furthermore, in **steady state**, $\partial\vec{v}/\partial t = 0$ and $\partial p/\partial t = 0$, *i.e.*, at a given point in space both are constant in time (note the use of partial time derivatives). The fluid within the volume element $\Delta V$ is also referred to as a fluid **particle** (which in this sense, of course, is not the same as a fluid molecule).

In general, a force $\Delta\vec{F}$ acting on the fluid within $\Delta V$ of mass $\Delta m = \rho\Delta V$ produces an acceleration given by Newton's second law of motion,

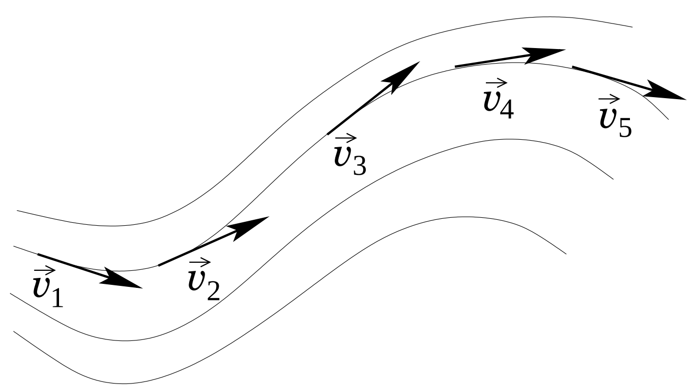
$$\Delta\vec{F} = \Delta m\,\frac{d\vec{v}}{dt} \tag{1}$$

This looks very similar to the corresponding equation for particle mechanics. However the distinctive properties of fluids are encoded in the detailed structure of $\vec{v}$ and $\Delta\vec{F}$: First, note that for a particle $\vec{v} = \vec{v}(t)$, while for a fluid element, $\vec{v} = \vec{v}(\vec{x}(t),t)$ and hence the distinction between $d/dt$ and $\partial/\partial t$ is very important (we will come back to this below). Second, the force $\Delta\vec{F}$ on the fluid element is a sum of external forces (for example, gravity) and internal forces like **pressure** and **viscosity** that act within fluids,

$$\Delta\vec{F} = \Delta\vec{F}_p + \Delta\vec{F}_{visc} + \Delta\vec{F}_g + \cdots\,. \tag{2}$$

These will also be discussed in more detail later.

The motion of fluid can be pictured in terms of **streamlines**. A streamline is a line drawn through the fluid such that a tangent to it at any point is in the direction of the fluid velocity at that point. A collection of neighboring streamlines forms a **streamtube**. In Steady state, the actual physical paths followed by fluid "particles" coincide with streamlines. This is not the case when the flow changes with time.




## 1.2 Pressure in Fluids: Static Pressure

An important component of the force $\Delta\vec{F}$ acting within fluids can be described in terms of **pressure**, or more precisely, in terms of **pressure gradients**. We now make the notion of pressure for a moving fluid more precise. First, recall that to a force $\vec{F}_p$ we can associate a pressure $\vec{p}$ as force per unit area,

$$\vec{p} = \lim_{\Delta A_\perp \to 0} \frac{\vec{F}_p}{\Delta A_\perp}$$

where $\Delta A_\perp$ is a small area element normal to the direction of the force and on which the force acts. For the time being, we regard pressure as a vector pointing naturally in the same direction as $\vec{F}_p$. Below we argue that for fluids, this can be replaced by a scalar quantity $p$.

Let us now consider a point $\vec{x}$ inside a fluid and a small area element $A$ passing through $\vec{x}$. The orientation of the area $A$ can be described by a unit vector $\hat{n}$ normal to the area element. This allows us to represent the small area element as a vector $\vec{A} = \Delta A\hat{n}$. In this way, the opposite faces of the same physical area $A$ can be denoted as $\vec{A}$ and $-\vec{A}$ and can be regarded as representing two different area elements oriented opposite to each other. For example, if we consider a small sheet of paper of scalar area $A$ passing through the point $\vec{x}$, then $\vec{A}$ and $-\vec{A}$ correspond to the two different faces of paper sheet. Although in practice a force will act on the scalar area $A$ (of the paper sheet, for example), theoretically we can make a distiction between forces acting on different faces of $A$ as follows: A force $\vec{F}$ is said to act on the face $\vec{A}$ provided its value remains unchanged in the hypothetical event that the fluid on the $-\vec{A}$ side of the area $A$ is removed (this will of course alter a force acting on the face $-\vec{A}$). From now on, we will be dealing with forces that act on an oriented area element in the above sense.

Now, for any elemental area $\vec{A}$ passing through a point $\vec{x}$ in a fluid, one can determine the force perpendicular to it and determine the associated pressure $\vec{p}$ at $\vec{x}$. In general, for each orientation of the area $A$ through $\vec{x}$ one could obtain a different value of pressure. However, in a fluid at rest, there cannot exist a net unballanced force acting on any point in the fluid (otherwise, the fluid will not remain at rest). This has two implications:

1) A force acting on the side $\vec{A}$ is balanced by an equal and opposite force acting on the opposite side $-\vec{A}$ and this is true for every orientation of the area element. For example, through a point $\vec{x}$ in a drum of liquid, one can consider a small horizontal area $A_h$ a distance $z$ below the surface. The downward pressure on $A_h$ due to the liquid column above it is $p_{down} = \rho g z$, where $\rho$ is the liquid density. At equilibrium this should be balanced by an upward pressure on $A_h$, so that $p_{up} = -p_{down}$. Similarly, one can consider a small vertical area $A_v$ through the same point $\vec{x}$ and argue that at equilibrium the left and right pressures on it must balance, $p_{right} = -p_{left}$.

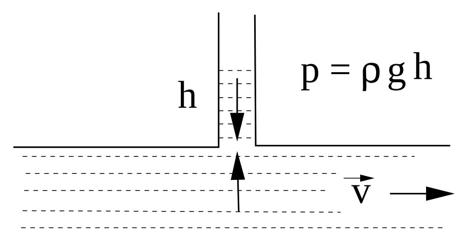
2) The arguments so far (which apply equally well to rigid bodies) do not tell us anything about the relation between pressures in different directions, say, $p_{down}$ and $p_{right}$. For a rigid body these would be unrelated, while for a fluid they turn out to be the same and the reason is not difficult to see: Consider a layer of fluid compressed vertically between two pistons. Since the fluid inbetween can easily change its shape, it will start flowing sideways, escaping from the region that is compressed by the pistons. The only way to counter this is to exert horizontal forces on the fluid (say, through the side walls of the container) to balance the vertical force of the pistons. Thus equilibrim in a fluid cannot be established unless at any point $\vec{x}$ within it pressure is the same in all directions and this is due to the fact that the shape of a fluid changes easily in response to applied forces. The technical statement is that, unlike solids, fluids in equilibrium do not support

shearing forces. The consequence is that **at equilibrium, pressure at a point $\vec{x}$ in the fluid is the same in all directions**.

The isotropy of fluid pressure implies that its vector nature has become irrelevant and hence it can be treated as a scalar $p(\vec{x}, t)$. Note that even in equilibrium, pressure can change from point to point, although at a given point it is the same in all directions (an example being the variation of pressure with depth in liquids). Pressure defined in this way for a liquid at rest is called **static pressure**.

In a moving fluid things are more complicated since the motion could be associated with unbalanced forces. However, the notion of pressure can still be defined unambiguously for an observer flowing along with the fluid. From the point of view of this observer, the fluid surrounding him/her is at rest and therefore the notion of pressure is the same as in the static case. The pressure in a moving fluid defined in this way is also called the **static pressure** (of the moving fluid).

How to measure static pressure without really flowing along with the liquid? To answer this, recall that fluid pressure is due to random motions of its molecules that impinge on a surface and is therefore related to the random molecular velocities. If the fluid as a whole flows with velocity $\vec{v}$, the components of molecular velocities in directions transverse to $\vec{v}$ are not affected by the flow. Hence the pressure measured in a direction transverse to the flow (*e.g.*, through the rise of liquid in a side tube) is the static pressure.




# 2 Bernoulli's Principle (A Simplified Treatment)

Even before further developing the fluid equation of motion, we are already in a position to give a simple derivation of **Bernoulli's principle** as a consequence of energy conservation in fluid flows (However, to get better understanding of this principle, later we will derive it more rigorously after we have discussed the fluid equation of motion in the next section).

Besides being an important principle of fluid dynamics, this will also help us develop a better understanding of fluid pressure as experienced in daily life. We will then discuss some applications and common misapplications of the principle.

## 2.1 Statement of the Principle

Let us consider the fluid contained in a small volume element $\Delta V$ which is flowing along a streamline with velocity $\vec{v}$ in steady state. The total mechanical energy of this fluid elements consists of three parts:

1. A kinetic energy $\frac{1}{2}\rho v^2 \Delta V$,

2. A potential energy due to pressure, $p\Delta V$. This it the work done to push the fluid into the volume $\Delta V = \Delta A \Delta x$ against a force $p\Delta A$ (more explicitly, $work = F dx = p\Delta V$).

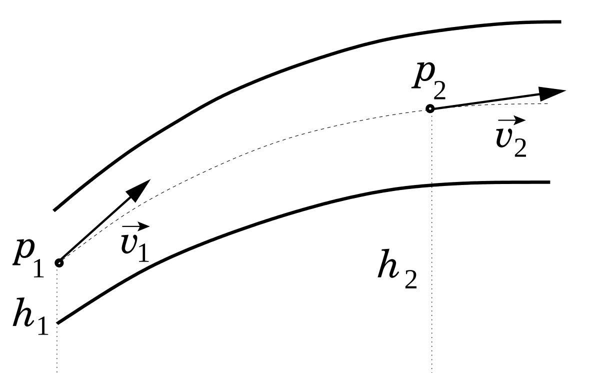
3. A potential energy due to gravity $\rho g h \Delta V$, where $h$ is the hight of $\Delta V$ above some given reference level.

Combining these together, the total mechnical energy of the fluid element per unit volume is obtained as $\frac{1}{2}\rho v^2 + p + \rho g h$. This energy density can change only if a) there is external work done on/by the fluid, b) there are internal dissipative forces (viscosity) against which the fluid has to do work as it flows. If there is no external work, and if the effect of dissipative forces can be ignored, then the energy conservation law states that as the fluid element in $\Delta V$ flows along a streamline,

$$\frac{1}{2}\rho v^2 + p + \rho g h = constant$$

Note, however, that the constant on the right-hand-side can have different values for different streamlines. This is the Bernoulli principle. Each term in the left-hand-side is a function of the position $\vec{x}$ within the fluid. The theorem states that the sum, however, is independent of $\vec{x}$ and is the same everywhere along a given streamline. This holds for inviscid (non-viscous) flows in steady state.

Under certain conditions the constant on the right-hand-side is the same for all streamlines and so remains the same throughout the fluid volume. This happens when the flow is *irrotational* (these concepts will be discussed later using the fluid equation of motion).




Consider a regular flow, say, in a tube. If we consider points 1 and 2 along a tube in which the fluid flows, then it is customary to express the Bernoulli's principle as

$$\frac{1}{2}\rho v_1^2 + p_1 + \rho g h_1 = \frac{1}{2}\rho v_2^2 + p_2 + \rho g h_2$$

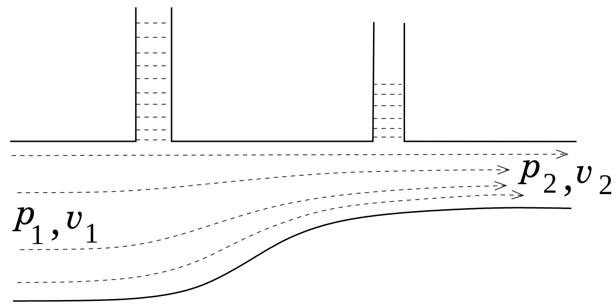
## 2.2 Some Applications of Bernoulli's Principle

The theorem can be applied to flows in tubes with varying cross sections: Consider a horizontal tube the cross section of which is $A_1$ at point 1 and $A_2$ at point 2 with $A_1 > A_2$.

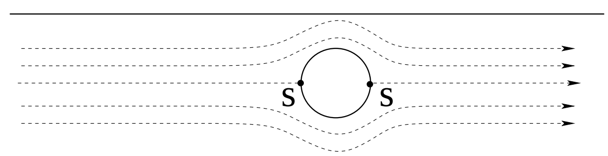
Since the flow is horizontal, $h_1 = h_2$, and the theorem takes the form,

$$\frac{1}{2}\rho v_1^2 + p_1 = \frac{1}{2}\rho v_2^2 + p_2$$

As a consequence of conservation of mass and incompressibility of the fluid, $\rho v_1 A_1 = \rho v_2 A_2$ so that $v_2 = v_1 A_1/A_2$. Thus, as the tube cross section reduces $A_1 > A_2$, the flow velocity increases ($v_1^2 < v_2^2$). Then the above equation then tells us that static pressure is lower in regions of higher velocity ($p_1 > p_2$). Measuring pressures $p_1$ and $p_2$ (for example, by measuring the rise of liquid in narrow side-tubes inserted at points 1 and 2) one can get information about the change in fluid velocity.




The notion that regions of higher fluid velocity have lower pressure may sound somewhat counterintuitive. After all, if you place your hand in a fluid flow, you will find that pressure on your hand grows with fluid velocity and not the other way around. The resolution of course is that the pressure felt by your hand is not the static pressure of the moving fluid (it is the pressure felt by you, not by the fluid). This can be clarified further by introducing the notion of a **stagnation point**: This is a point in the steamline where the flow velocity reduces to zero, $v_s = 0$ (due to the presence of an obstacle). At this point the static pressure has the maximum value $p_s$ (since $\vec{v}_s = 0$):




$$\frac{1}{2}\rho v^2 + p = p_s$$

A hand placed in a fluid flow feels the stagnation pressure $p_s$ (and *not* $p$) which increases with $v^2$. Measuring $p_s$ and $p$, one can determine the flow velocity $v$. (For an example see problem set 1).

## 2.3 Erroneous applications of Bernoulli's principle

Bernoulli's principle easily lends itself to erroneous applications to a range of everyday phenomena. The misapplications include the explantion of flight (air flow and forces on an airfoil), the working of a vaporizer, the behaviour of a light ball in an air jet, etc. We begin with a misapplication that characterizes the situation very well.

a) Hangning paper strip: Take a strip $AB$ of paper around $1cm$ wide and some $7-8cm$ long. Hold the end $A$ in front of your mouth with the end $B$ free. The strip will then take a convex shape with the end $B$ hanging downwards. Now if you start blowing mildly along the surface, end $B$ of the paper strip will rise untill the strip is horizontal and no longer hangning down. One may be tempted to explain this in terms of Bernoulli's principle as follows: the air below the paper strip is stationary and at a pressure of 1 atmosphere. As you blow, the air above the strip acquires a velocity and hence its pressure drops below 1 atmosphere making the paper strip rise. Here is the flaw in this argument: As you blow, you impart kinetic energy to the air above the strip by doing external work on it. This is no longer the isolated system to which the Bernoulli theorem applied. Also the air above and below belong to different streamlines. Because of these facts, the air above the strip moves faster without its pressure droping (subsequently a pressure difference will develop but for other reasons as will be explained below). To confirm this as a misapplication of the Bernoulli principle, repeat the above experiment after pushing the end $B$ of the strip up with your finger so that now the strip has a concave shape with its middle part sagging down. If you now start blowing along it, the strip will be pushed further down, rather than raised up while our naive application of the Bernoulli principle would predict the opposite.

This experiment shows that the behaviour of paper strip depends on its curvature (convex or concave). We will show later that the effect is actually caused by the curvilinear flow of the fluid. All misapplications of the Bernoulli's principle fall in this category: the real cause of the phenomenon is fluid motion along a curve.

b) Flight: A major misapplication of Bernoulli's theorem is to the problem of flight and the explanation of the origin of forces that act on the airfoil. In this, one looks at the nature of the air-flow around the wing of an aircraft. It is argued that air on the upper side of the wing travels a longer distance than on the lower side during the same time interval and hence it has a higher velocity. This gives rise to a lower pressure region above the wing that sucks the wing up. The problem here is not that Bernoulli's theorem does not apply to the air flow near the wing; it certainly does and turns out to be very useful. The problem is that of cause and effect and of the other important effects that are crucial to establishing the right flow pattern, but are overlooked in this simplified explanation. Below, we give a slightly more detailed description of air-flow based on the fluid equation of motion and then use the results to gain a better understanding of the above phenomena. In short, we will follow the following reasoning: Air flow above the wing follows a curved path due to Coanda effect. Viscosity effects near the wing further shape the flow. Finally, curved flow generates a lower pressure region above the wing as can be seen from the fluid equation of motion (and interestingly, the force component responsible for this is orthogonal to the component responsible the Bernoulli principle!). This creats the lift. Now, the pressure difference also creats a velocity difference above and below the wing by Bernoulli principle. Then the velocity difference contains information about the lift and can be used to compute the upward force.

# 3 Equations of Fluid Dynamics

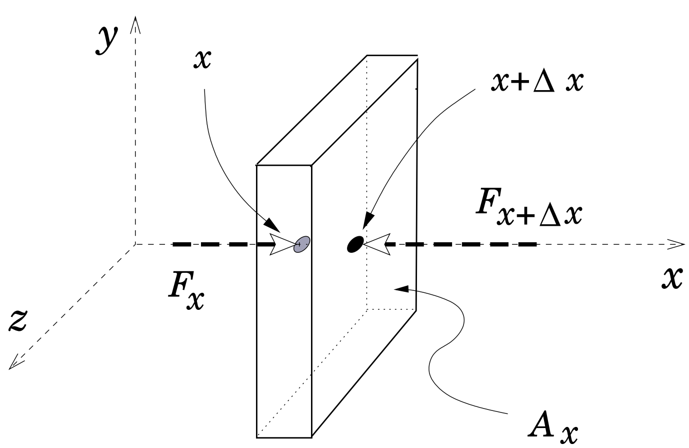
The fluid equation of motion $\Delta\vec{F} = \Delta m(d\vec{v}/dt)$ describes the motion of a small fluid element of volume $\Delta V$ under the influence of force $\Delta\vec{F}$. The main contributions to the force are written in the form $\Delta\vec{F} = \Delta\vec{F}_p + \Delta\vec{F}_{visc} + \Delta\vec{F}_g$ and each will be discussed below. We will then describe the acceleration $d\vec{v}/dt$ of the fluid element. The continuity equation is described at the end.

## 3.1 Force due to Pressure Gradient

A component $\Delta\vec{F}_p$ of the force acting on small fluid elements is caused by **pressure gradient** (or unbalanced pressure) in the fluid. To see how this arises, it should be emphasized that $\Delta\vec{F}$ is the net force acting on the fluid in $\Delta V$, that is the sum of all forces acting on it from different directions. Even when the net force at a point $\vec{x}$ within the fluid vanishes, the net force on $\Delta V$ may not vanish:

Let us first compute the component $(\Delta F_p)_x$ of $\Delta\vec{F}_p$ in the $x$-direction. Consider a small rectangular slab of thickness $\Delta x$ (along the $x$-axis) and area $A_x$ (in the $y - z$ plane) within the fluid. The volume of the slab is $\Delta V = A_x \Delta x$. The two faces intersect the $x$-axis at $x$ and $x + \Delta x$ where the pressures are $p$ and $p + \Delta p$, respectively, corresponding to a pressure gradient $\Delta p/\Delta x$ across the slab. The forces on the faces are $(F_p)_x = p\, A_x$ and the oppositly directed $(F_p)_{x+\Delta x} = -(p + \Delta p)\, A_x$. Thus the net force acting on the slab in the $x$-direction is

$$(\Delta F_p)_x = (F_p)_x + (F_p)_{x+\Delta x} = [p\, A_x - (p + \Delta p)\, A_x] = -\Delta p\, A_x = -\frac{\Delta p}{\Delta x}\Delta V$$




In the limit of zero thickness, one gets a force per unit volume,

$$\lim_{\Delta V \to 0} \frac{(\Delta F_p)_x}{\Delta V} = -\frac{\partial p}{\partial x}$$

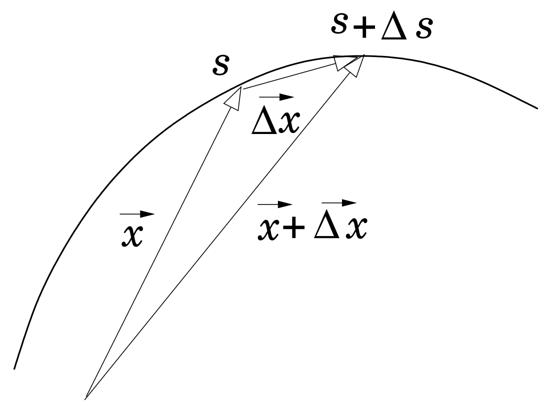
The sign indicates that the force is directed from region of higher pressure to region of lower perssure which is the direction in which fluid particles move as a result of a pressure difference.

Clearly, one can derive similar expressions for the $y$ and $z$ components of the force. At the end, taking the small volume limit, we get,

$$\Delta\vec{F}_p = -\left(\hat{x}\,\frac{\partial p}{\partial x} + \hat{y}\,\frac{\partial p}{\partial y} + \hat{z}\,\frac{\partial p}{\partial z}\right)\,\Delta V = -\vec{\nabla}p\,\Delta V$$

where, $\hat{x}$, *etc.* denote unit vectors in the corresponding directions and the small volume limit is implicitly assumed.

A streamline is a one dimensional curve in space. At any point, we can draw a tangent vector to this curve. By the definition of a streamline, this tangent vector has the same direction as the fluid velocity at that point. Hence, if $\hat{s}$ is a unit tangent vector to the streamline at a point ($\hat{s} \cdot \hat{s} = 1$), then in terms of the fluid velocity at that point, $\hat{s} = \vec{v}/|\vec{v}|$. We can now ask the question: What is the component of the force $\Delta\vec{F}_p$ in the direction of a given streamline? More explicitly, we want to evaluate $\Delta\vec{F}_p \cdot \hat{s}$. To this end, it is convenient to first give a different description of $\hat{s}$: Any point along the streamline can be parametrized in terms of its distance, $s$, from a reference point on the streamline. For example, a point $A$ will be at some distance $s_A$ from the reference point and its distance from point $B$ will be $s_A - s_B$ (note that $s$ is measured along the curve and not along a straight line). Consider two adjacent points $s$ and $s + \Delta s$ with vector positions $\vec{x}$ and $\vec{x} + \Delta\vec{x}$, respectively, on the streamline. For very close separation, the distance between the two in terms of the length parameter is $\Delta s = |\Delta\vec{x}|$. In the limit that the two points approach each other (that is, $\Delta s \to 0$) $\Delta\vec{x}$ becomes tangent to the curve at the point $\vec{x}$ (or $s$). Then $\hat{s} = \Delta\vec{x}/\Delta s$ is a unit tangent vector. The component of $\Delta\vec{F}_p$ tangent to the streamline at $s$ is then clearly




$$\Delta F_p^s \equiv \Delta\vec{F}_p \cdot \hat{s} = (-\vec{\nabla}p \cdot \hat{s})\,\Delta V = (-\frac{\partial p}{\partial s})\,\Delta V$$

To understand the last step, note that

$$\Delta p = p(\vec{x} + \Delta\vec{x}) - p(\vec{x}) = \frac{\partial p}{\partial x}\Delta x + \frac{\partial p}{\partial y}\Delta y + \frac{\partial p}{\partial z}\Delta z = \vec{\nabla}p \cdot \Delta\vec{x}$$

Then, dividing by $\Delta s$ one obtains the **directional derivative** of $p$ along the streamline as

$$\frac{\partial p}{\partial s} \equiv \lim_{\Delta s \to 0} \frac{\Delta p}{\Delta s} = \vec{\nabla}p \cdot \hat{s}$$

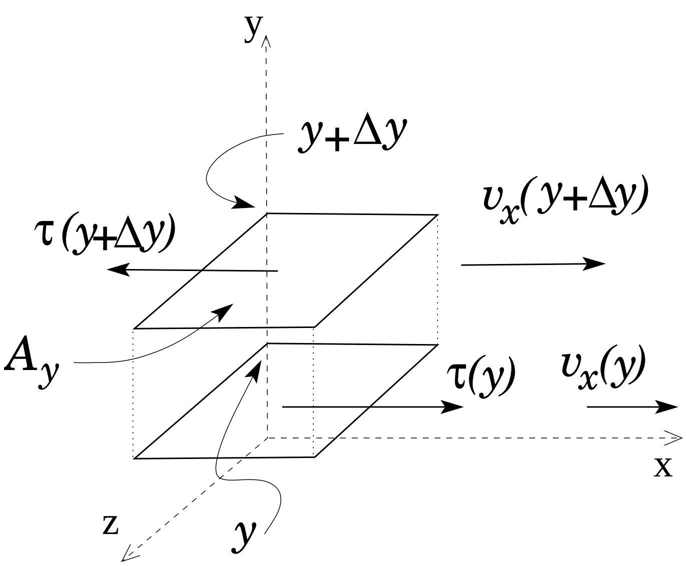
To summarize, the component of force along the streamline is

$$\Delta F_p^s = (-\frac{\partial p}{\partial s})\,\Delta V$$

## 3.2 Force due to Viscosity

Viscosity in fluids is the analogue of friction in solids. It is the property that makes adjacent layers of moving fluid tend to stick together. So if two adjacent layers have different velocities, the faster layer tends to slowdown and the slower one tends to speed up due to the pull of the other layer. For fluids flowing near a boundary, this gives rise to the following profile: the fluid layer immediately next to the boundary is almost stationary due to friction between fluid and the boundary. This stationary level pulls on the moving fluid next to it and tends to slow in down. So as we move away from the boundary, the flow speed get faster. This behaviour can be quantified in a simple way:

Consider a horizontal flow of fluid with velocity in the $x$ direction. The flow is parallel to the $x-z$ plane and the $y$ coordinate measures the fluid height. Consider 2 flow planes parallel to the $x-z$ plane at heights $y$ and $y+\Delta y$ with flow velocities $v_x$ and $v_x+\Delta v_x$, respectively (note that the lower layer, being closer to the bottom boundary, has a lower velocity than the upper one, due to viscosity). The faster flow at height $y+\Delta y$ tries to speed up the flow at height $y$ through viscous interaction by exerting a force on it in the $x$ direction (that is, the flow direction). Such a force per unit horizontal area is called the **stress**, $\tau(y)$ (Note the differece: pressure is force per unit area acting normal to a surface, while stress is force per unit area acting parallel to the surface). Similarly, the slower flow at height $y$ tries to retard the faster flow at height $y+\Delta y$ by exerting on it a force per unit area $\tau(y+\Delta y)$ in the $-x$ direction. Such forces are the cause of the velocity difference between the layers. For simple fluids (the so called Newtonian fluids), the resulting velocity gradient is proportional to $\tau$,




$$\tau = \mu \frac{\partial v_x}{\partial y}$$

where $\mu$ is called the **viscosity coefficient**.

Let us now consider a small volume element of height $\Delta y$ and cross section area $\Delta A_y$ between the flow plans at heights $y$ and $y+\Delta y$. The oppositely directed stresses $\tau(y)$ and $\tau(y+\Delta y)$ acting on the lower and upper faces of this volume give rise to a net force on $\Delta V = \Delta y \Delta A_y$ in the $x$-direction,

$$(\Delta F_v)_x = [\tau(y+\Delta y) - \tau(y)]\,\Delta A_y = \mu\left[\left(\frac{\partial v_x}{\partial y}\right)_{y+\Delta y} - \left(\frac{\partial v_x}{\partial y}\right)_y\right]\Delta A_y = \mu\frac{\partial^2 v_x}{\partial y^2}\Delta y \Delta A_y$$

Velocity gradients in the $z$ and $x$ directions also contribute to the force, and the total $(\Delta F_v)_x$ (which we do not derive here [1]) is given by $(\Delta F_v)_x = (\mu\nabla^2 v_x)\Delta V$. Similar equations hold for other components of the force, hence,

$$\Delta \vec{F}_v = (\mu \nabla^2 \vec{v})\,\Delta V$$

[1]The appearance of the term $\mu\frac{\partial^2 v_x}{\partial x^2}$ is not obvious for our simple definition of $\mu$ and the above derivation based on it. A more rigorous treatment is based on the analysis of stresses and strains in fluids. For Newtonian fluids, the two are linearly related through the viscosity coefficients. For incompressible fluids, the expression coincides with the one given here.

Sometimes fluid viscosity does not play an important role in a phenomenon and can be ignored for the sake of that phenomenon. Such a flow for which, effectively, $\mu = 0$, is called **inviscid flow**.

## 3.3 Force due to Gravity

$\Delta\vec{F}_g$ is the gravitational force, $(\Delta m)\vec{g}$, acting on the mass element $\Delta m = \rho\Delta V$, where $\vec{g}$ is the acceleration due to gravity. This can also be written in terms of the gradient of a gravitational potential $\Phi$, using $\vec{g} = -\vec{\nabla}\Phi$,

$$\Delta\vec{F}_g = \rho\,\vec{g}\,\Delta V = -\rho\,\vec{\nabla}\Phi\,\Delta V$$

## 3.4 Fluid Acceleration

Newton's second law applied to fluids contains the acceleration $\vec{a} = d\vec{v}/dt$ of a fluid element. To elucidate the meaning of the total derivative $d/dt$ (in contrast with the partial time derivative $\partial/\partial t$), we start with a generic function $f(\vec{x}, t)$ which varies both in space and time. For example, $f$ could represent the local fluid density, local temperature or any component of fluid velocity. Often we are interested in the value of such a quantity for a fluid element as it flows along a streamline. The fluid element follows a path $\vec{x}(t)$ in space. Then the values of the function along this path are given by $f(\vec{x}(t), t)$. Note the two sources of time dependence: Even for a function $f(\vec{x})$ which does not explicitly depend on time, the corresponding function $f(\vec{x}(t))$ evaluated on the path $\vec{x}(t)$ still has a time dependence because of the motion of the fluid element.

The total time derivative of $f$ takes both types of time variations into account. To evaluate it, note that a small time interval $\delta t$ the fluid point $\vec{x}(t)$ moves to $\vec{x}(t + \delta t) = \vec{x}(t) + \vec{v}\delta t$. The total time derivative of $f(\vec{x}(t), t)$ is then defined as

$$\frac{df}{dt} = \lim_{\delta t \to 0} \frac{f(\vec{x} + \vec{v}\delta t, t + \delta t) - f(\vec{x}, t)}{\delta t}$$

On Taylor expanding the numerator and taking the limit one obtains,

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \sum_{i=1}^{3} v^i \frac{\partial f}{\partial x^i} \equiv \frac{\partial f}{\partial t} + \vec{v} \cdot \vec{\nabla} f$$

Now taking $f$ to be the components $v^i(\vec{x}, t)$ of the velocity of a fluid element, the components $a^i$ of acceleration can be easily computed as,

$$a^i = \frac{dv^i}{dt} = \frac{\partial v^i}{\partial t} + (\vec{v} \cdot \vec{\nabla}) v^i \,,$$

or, in vector notation,

$$\vec{a} = \frac{d\vec{v}}{dt} = \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \vec{\nabla})\vec{v} \,.$$

We have now deciphered all quantities that appear in Newton's second law as applied to Fluids.

## 3.5 The Navier-Stokes Equation

The **Navier-Stokes equation** is simply a rewriting of Newton's second law,

$$\Delta \vec{F} = \Delta m \, \vec{a} \,,$$

applied to the motion of fluid elements. $\Delta \vec{F}$ is the force acting on a fluid mass $\Delta m = \rho \Delta V$ that moves with velocity $\vec{v}(\vec{x}(t), t)$ and imparts to it an acceleration $\vec{a} = d\vec{v}/dt$. Collecting the expressions from the previous sections one obtains the equation,

$$\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \vec{\nabla})\vec{v} = -\frac{1}{\rho}\vec{\nabla} p + \eta \nabla^2 \vec{v} - \vec{\nabla}\Phi$$

where $\eta = \mu/\rho$ is the viscosity coefficient. This is the Navier-Stokes equation for incompressible flow:

In order to solve this equation, in general, one has to specify boundary conditions. The boundary conditions often depend on the shape of the objects that constrict or divert the flow, like the geometry of air ducts or the shape of aircraft wings (airfoil). In most cases it suffices to impose as two boundary conditions the requirements that, at the boundary, both the normal $\vec{v}_\perp$ and tangential $\vec{v}_{||}$ components of $\vec{v}$ vanish. Physically $\vec{v}_\perp = 0$ means that the fluid does not penetrate the boundary and $\vec{v}_{||} = 0$ means that the fluid layer just adjacent to the boundary is at rest as supported by empirical evidence.

## 3.6 The Continuity Equation

This equation is a statement of conservation of mass in fluid dynamics. Within the flow, let us consider a **fixed** volume $V$ bounded by a surface $S$ (this is a fixed volume in space through which fluid flows). At a given time $t$, the total fluid mass contained in $V$ is given by $M_V = \int_V \rho(t) dV$. During some time interval $\Delta t$, some fluid may enter or leave the volume $V$ by passing through the surface $S$. Let us denote the quantity of fluid passing through $S$ per unit time by $I_S$. To compute this fluid flux through the surface, consider a small surface element $d\vec{S}$ on the surface $S$. The magnitude of this vector gives the area of the surface element while its direction is normal of to the surface element and fixes its orientation. Then the amount of fluid crossing $d\vec{S}$ per unit time is given by the scalar product $\rho\vec{v} \cdot d\vec{S}$ and the total mass of fluid leaving/entering through the surface $S$, per unit time, is

$$I_S = \oint_S \rho\vec{v} \cdot d\vec{S} = \int_V \vec{\nabla} \cdot (\rho\vec{v}) dV$$

To write the expression in terms of a volume integral we have used the **divergence theorem**.

If no mass is created or destroyed in $V$, then this in/out-flow of mass across the boundary of $V$ should result in an increase/decrease (per unit time) of mass within $V$. Hence the total change of mass per unit time is given by

$$\frac{\partial M_V}{\partial t} = -I_A$$

The negative sign signifies that if the flow is directed outwards (so that $\vec{v}$ and $\vec{dS}$ are roughly in the same direction and hence $I_S = \oint_S \rho\vec{v}\cdot\vec{dS}$ is positive), then $\partial M_V/\partial t$ is negative, consistent with the fact that mass inside $V$ reduces as a result of the outward flow. Since the integration region $V$ is time independent, one gets,

$$\int_V \frac{\partial\rho}{\partial t} dV = -\int_V \vec{\nabla}\cdot(\rho\vec{v})dV$$

This holds for any arbitrary volume $V$, which implies that

$$\frac{\partial\rho}{\partial t} + \vec{\nabla}\cdot(\rho\vec{v}) = 0$$

This is the continuity equation which is a mathematical statement of the principle of conservation of mass. It holds for both viscous and invicid fluids.

If the fluid is incompressible, then its density $\rho$ is constant *i.e.*, $\partial\rho/\partial t = 0$ and $\vec{\nabla}\rho = 0$. The continuity equation then reduces to

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

# 4 Simple Applications of Fluid Equation of Motion

Having derived the Navier-Stokes equation, now we consider some of it's simple implications. This includes a derivation of the Bernoulli principle and study of forces that result for the curvature of the flow (the later is useful in understanding the physics of flight, and shape of fluid rotating in a drum).

## 4.1 A Closer Look at Fluid Acceleration

In steady state, the fluid acceleration reduces to

$$\vec{a} = \frac{d\vec{v}}{dt} = (\vec{v}\cdot\vec{\nabla})\vec{v}\,.$$

Obviously, at any poing along the streamline, $\vec{a}$ can be resolved into components parallel and perpendicular to the velocity at that point,

$$\vec{a} = \vec{a}_{||} + \vec{a}_{\perp}$$

such that $\vec{a}_{||} = (\vec{a}\cdot\hat{s})\hat{s}$, and $\vec{a}_{\perp}\cdot\hat{s} = 0$. As discussed above, $\hat{s}$ is a unit vector along the velocity and is also a unit tangent vector to the streamline at a point $s$. Now we want to obtain the expressions for $\vec{a}_{||}$ and $\vec{a}_{\perp}$.

Denoting the magnitude of the velocity by $v = |\vec{v}|$, we have $\vec{v} = v\hat{s}$. Also from the discussion above, we know that $\hat{s}\cdot\vec{\nabla} = \partial/\partial s$. Substituting these in $\vec{a}$ we get

$$\vec{a} = v(\hat{s}\cdot\vec{\nabla})(v\hat{s}) = v\frac{\partial}{\partial s}(v\hat{s}) = v\frac{\partial v}{\partial s}\hat{s} + v^2\frac{\partial\hat{s}}{\partial s}$$

The first term is clearly parallel to $\hat{s}$. As for the second term, note that $\hat{s} \cdot \hat{s} = 1$ implies that $\hat{s} \cdot \frac{\partial \hat{s}}{\partial s} = 0$ (this is easy to understand: a unit vector cannot change parallel to itself since that would change its magnitude, so it can only vary normal to itself subject to its norm remaining unchanged). Hence the second term is normal to $\hat{s}$. Now we have the sought after decomposition of acceleration,

$$\vec{a}_{||} = \frac{1}{2} \frac{\partial v^2}{\partial s} \hat{s}, \qquad \vec{a}_{\perp} = v^2 \frac{\partial \hat{s}}{\partial s}$$

(Note that $\partial \hat{s}/\partial s$ is in general not a unit vector).

It is useful to illustrate this with an example: Consider circular motion of the fluid along a circle of radius $r$. For the circular streamline, we can choose $s = r\theta$ and $\hat{s} = \hat{\theta}$. Since $r$ is fixed,

$$\frac{\partial}{\partial s} = \frac{1}{r}\frac{\partial}{\partial \theta}, \qquad \frac{\partial \hat{s}}{\partial s} = \frac{1}{r}\frac{\partial \hat{\theta}}{\partial \theta} = -\frac{\hat{r}}{r}$$

Then $\vec{a}_{\perp} = -(v^2/r)\hat{r}$ which is the familiar centripetal acceleration of the fluid volume element. Moreover, if the flow is uniform, then $\vec{a}_{||} = 0$.

## 4.2 Derivatin of Bernoulli's Principle

Bernoulli's principle is valid for the steady state flow of inviscid fluids for which the equation of motion takes the form,

$$(\vec{v} \cdot \vec{\nabla})\vec{v} = -\frac{1}{\rho}\vec{\nabla}p - \vec{\nabla}\Phi \tag{3}$$

The scalar product of this equation with $\hat{s}$ correcponds to the effect of forces acting parallel to the streamline, $\Delta F_{||} = \Delta m a_{||}$. Based on our previous results, this gives,

$$\frac{\partial}{\partial s}\left(\frac{v^2}{2} + \frac{p}{\rho} + \Phi\right) = 0$$

Hence the quantity within the brackets is a constant along the streamline,

$$\frac{1}{2}\rho v^2(s) + p(s) + \rho\Phi(s) = c \text{ (independent of } s)$$

From this derivation it is clear that $c$ is constant only along a given streamline but could have different values for other streamlines.

There is however a special type of flow for which $c$ is constant throughout the fluid. To see this, we start from the full equation of motion (3). In components, the first term on the LHS is $\sum_{i=1}^{3} v^i \partial v^j/\partial x^i$. Note that if we make the assumption (to be discussed later) that the velocity field satisfies

$$\frac{\partial v^j}{\partial x^i} = \frac{\partial v^i}{\partial x^j}, \tag{4}$$

then this term can be written as $\sum_{i=1}^3 v^i \partial v^i/\partial x^j = \frac{1}{2}\partial(\sum_{i=1}^3 v^i v^i)/\partial x^j$ (this assumption is non-trivial only for $i \neq j$). The equation then reduces to

$$\vec{\nabla}(\frac{1}{2}\rho\vec{v}^2 + \rho\,\Phi + p) = 0$$

The solution is then,

$$\frac{1}{2}\rho\vec{v}^2 + \rho\,\Phi + p = constant \;\; (independent\ of\ \vec{x})$$

which is the more restrictive form of Bernoulli theorem.

Recal that for the above solution to be valid, it is important that the velocity field satisfies the assumption (4) made above. In vector form, this assumption is simply,

$$\vec{\nabla} \times \vec{v} = 0$$

A flow satisfying this condition is said to be **irrotational**. Equivalently, the condition states that the line-integral of $\vec{v}$ along any closed curved $C$ in the flow is zero,

$$\oint_C \vec{v}\cdot d\vec{l} = \int_S (\vec{\nabla} \times \vec{v}) \cdot d\vec{S} = 0$$

where use has been made of **Stokes's theorm** in vector calculus. To make the discussion more formal, note that one has the vector identity,

$$(\vec{v}\cdot\vec{\nabla})\vec{v} = \vec{\nabla}(\frac{1}{2}\vec{v}^2) - \vec{v} \times (\vec{\nabla} \times \vec{v})$$

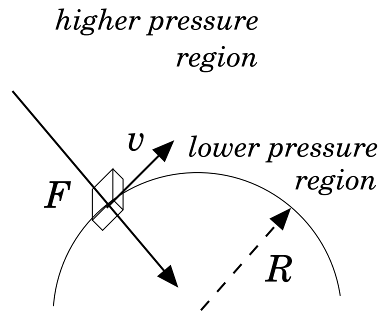
and that for irrotational flow, the last term on the RHS vanishes leading to the desired result.

## 4.3 Uniform fluid flow along a circular path

The study of fluid acceleration parallel to the streamline, $a_{||}$, gave us information about the variation of pressure along the streamline. In a similar way, the study of the component of fluid acceleration perpendicular to the streamline, $a_\perp$, will tell us how pressure varies normal to streamlines. In particular, we will see that the variation of pressure normal to the streamline is correleted with the curvature of the streamline. This understanding is very useful and can provide explanations for diverse phenomena including the basic dynamics of **flight**. For the sake of simplicity, in this section we consider uniform flow along a circular path since it already exhibits the main features of curved flow. In a later section we will consider general curved flows.

In order to understand, for example, the mechanism of flight, we have to know something about fluid motion along a curved path. The basic feature of such flows can be understood by considering the simple case of **uniform** fluid flow along a circular arc so that the streamlines form arcs of concentric circles. The volume element $\Delta V$ then moves along a circular arc of radius $R$ at constant speed. Since the magnitude of $\vec{v}$ remains constant, $d\vec{v}/dt$ is simply the centripetal acceleration directed towards the centre of the circle,

$$\frac{d\vec{v}}{dt} = -\frac{v^2}{R}\hat{R}$$




where $\hat{R}$ is a unit vector in the radial direction. The fluid equation of motion (1) applied to this situation implies that there is a force

$$\vec{F}_{\perp} = -(\rho\Delta V)\frac{v^2}{R}\hat{R}$$

directed toward the centre of the circle. The relationship between force and pressure gradient discussed earlier (that the force vector points from a region of high pressure to a region of low pressure) then implies that **pressure is higher for larger values of the radius**. Later we will apply this formula to a drum of rotating liquid. For now, two points should be emphasized:

1. Often in problems that involve flow of air in an open environment, the large radius region maybe contiuously connected to the atmosphere. The high pressure at large radius is then the atmospheric pressure (say, 1 $atm$). This means that the low pressure at small radius should be below 1 $atm$.

2. The second point to emphasize is that although we have described how a curved flow generates pressure gradients, we have not explained how the flow is curved in the first place. This is usually due to the so called **Coanda effect** that will be described later. The combination of Coanda effect and pressure gradient arising from curved flow can explain a number of interesting phenomena including the paper strip experiment.

## 4.4 Fluid flow along a general curved path

For the sake of completeness, we now consider curved flows not necessarily along arcs of circles (but it is convenient to still restrict to 2-dimensional flows). The general formula for $a_{\perp}$ gives the force transverse to the streamline,

$$\vec{F}_{\perp} = (\rho\Delta V)v^2\,\frac{\partial\hat{s}}{\partial s}$$

So everything depends on the behavior of $\partial\hat{s}/\partial s$. If $\hat{s}$ lies in the $x-y$ plane, we can parametrize it in terms of its $x$ and $y$ components, $\hat{s} = \hat{\imath}\cos\theta + \hat{\jmath}\sin\theta$, where $\theta$ is the angle between $\hat{s}$ and the $x-axis$ and it's value depends on the value of the paremeter $s$ along the curve. Hence $\theta(s)$ encodes the information about the curvature of the flow. Since $\hat{\imath}$ and $\hat{\jmath}$ are fixed vectors in space, we have

$$\frac{\partial\hat{s}}{\partial s} = (-\hat{\imath}\sin\theta + \hat{\jmath}\cos\theta)\frac{\partial\theta}{\partial s} \equiv \hat{n}\,\frac{\partial\theta}{\partial s}$$

$\hat{n}$ is a unit vector normal to $\hat{s}$ as can be easily verified. If we ignore the contributions of viscosity and gravity to the force, then $\vec{F}_{\perp}$ is given in terms of the component of the pressure gradient $-\vec{\nabla} p$ normal to the flow,

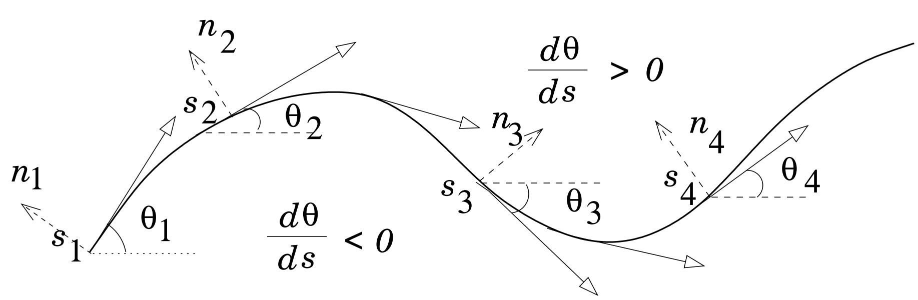
$$\hat{n} \cdot \vec{F}_{\perp} = -\hat{n} \cdot (\vec{\nabla} p)\, \Delta V = -\frac{\partial p}{\partial n}\, \Delta V$$

where $\partial/\partial n$ denotes a derivative normal to the streamline and can be taken to simply stand for $\hat{n} \cdot \vec{\nabla}$. Putting all this together in the equation of motion, one gets

$$\frac{\partial p}{\partial n} = -\rho v^2\, \frac{\partial \theta}{\partial s}$$

This gives the variation of pressure normal to the streamline as a function of its curvature.

To understand the implication of this equation, let's consider a flow that is **convex upward**. We want to study pressure variation in the region above a certain streamline in the flow. Then as we move along the flow, say, from left to right, $\theta$ starts from a positive value, decreases to zero and then becomes negative; hence, $\partial\theta/\partial s < 0$ and therefore,




$\partial p/\partial n > 0$. In other words, pressure increases in the normal direction as we go away from the streamline (just as in the circular flow case considered above). This is to be contrasted with a flow pattern that is **concave upward**. Now, as $s$ increases, $\theta$ goes from a negative value to zero and then becomes positive, hence $\partial\theta/\partial s > 0$ and therefore, $\partial p/\partial n < 0$. This states that in the region above the streamline, as we go away form the streamline in the transverse direction, pressure falls.

To summarize, at any point on the streamline, the normal force $\vec{F}_{\perp}$ is directed toward the **center of curvature** at that point. This implies that pressure decreases in the direction of center of curvature, but increases in the opposite direction.

## 4.5 Drum of rotating (inviscid) liquid

As an application of the result of the preceding section we consider a rotating cylidrical drum filled with liquid. The liquid rotates together with the drum and any small volume element $\Delta V$ in it moves around a circular path. At a qualitative level, the discussion in the previous section tell us that pressure in the liquid increases as we go away from the centre of the drum. However, in this specific case, we can also get a quantitative expression for the pressure.

To describe the rotation of the liquid in the drum, let us choose a cylindrical polar coordinate system $(z, r, \theta)$ where the $z$-axis coincides with the axis of the drum around which it rotates. $r$ is the radial distance from the axis of rotation and $\theta$ is the angle of rotation so that the angular velocity of the drum is given by $\omega = d\theta/dt$. In this coordinate system it is convenient to choose a volume element $dV$ such that its three sides are given by the height $dz$, width $rd\theta$ and depth $dr$ so that $dV = rdrd\theta dz$. Hence in the fluid equation (1), we set $\Delta m = \rho rdrd\theta dz$ and $d\vec{v}/dt = -\omega^2 r\hat{r}$, so that

$$\vec{F} = -(\rho\,\omega^2\,r^2\,drd\theta dz)\,\hat{r}$$

$\vec{F}$ is the net radial force that acts on the volume element $dV$. If we study the motion of $dV$ at a fixed height, then we can ignore the effect of gravity. To a good approximation, we may also ignore viscocity and concentrate on inviscid flow. Then, recalling our earlier discussion, $\vec{F}$ can be related to the pressure gradient $dp/dr$ across the volume $dV$ as follows:

The face of this volume element that is perpendicular to the radius vector has area $rd\theta dz$. The force on the face at radius $r$ is $\vec{F}_r = [p(r)\,r\,d\theta dz]\,\hat{r}$ and that on the face at radius $r+dr$ is $\vec{F}_{r+dr} = -[(p(r)+dp)\,r\,d\theta dz]\,\hat{r}$. The choice of sign signifies that the forces are always directed into $dV$. Now, the total radial force on $dV$ becomes,

$$\vec{F} = \vec{F}_r + \vec{F}_{r+dr} = -[dp\,r\,d\theta dz]\,\hat{r}$$

Combining this with the last equation, one gets an expression for the pressure gradient,

$$\frac{dp}{dr} = \omega^2\,r\,\rho$$

This gives the variation of pressure as a function of $r$ at a fixed height $z$ from the bottom of the drum. Integrating this expression from $r = 0$ to any radius $r$ gives (for any constant $z$),

$$p_r = p_0 + \frac{\omega^2\,r^2\rho}{2}$$

where $p_0$ is the value of pressure at $r = 0$ and at a given height $z$. Changing the height $z$ affects $p_r$ through $p_0$ as the second term is independent of height.

Using the above expression one can also obtain the height profile of the surface of the rotating liquid as function of the radius (See Problem set 1). It is an interesting exercise to generalize the above cosiderations to the case of viscous fluids.

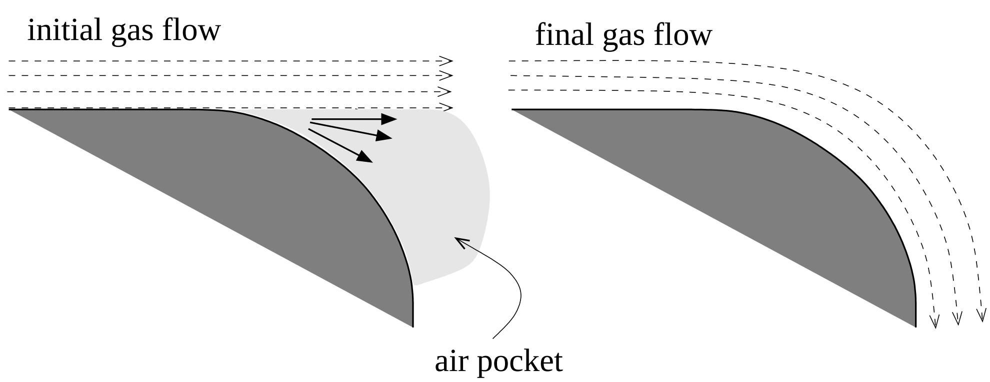
# 5 Understanding the Aerodynamics of Flight

Flight is a consequence of the pattern of air flow around the wings of an aeroplane. Air flows around the wing such that pressure above the wing is less than the pressure below it, generating an upward force on the wing. The purpose is to understand how such a flow pattern is created. This information is in principle contained in the fluid equations of motion. But we are more interested in the physical origins of the dominant effects. Thus in this section, we start with a very qualitative discussion of the flow pattern in terms of the property of the gas to follow the shape of the wing (usually called the Coanda effect). But this effect, by itself, is not enough to understand an important feature of the flow, the circulation around the wing, which is caused by viscous effects at the rear end of the wing. We address this when discussing the Kutta condition.

## 5.1 Coanda effect

Coanda effect is the tendency of a fluid flow along a curved surface to follow the shape of the surface. The effect is observed in liquid as well as in gas flows very easily. The trivial case is when a surface adjacent to a flow bends toward the flow, partially obstructing it. Since the flow cannot penetrate the surface, it has no option but to bend and continue flowing along the surface. The non-trivial case arises when a surface adjacent to a flow bends away with respect to the initial flow direction. Even in this case, the flow tends to change direction as well so that it remains in contact with the surface.

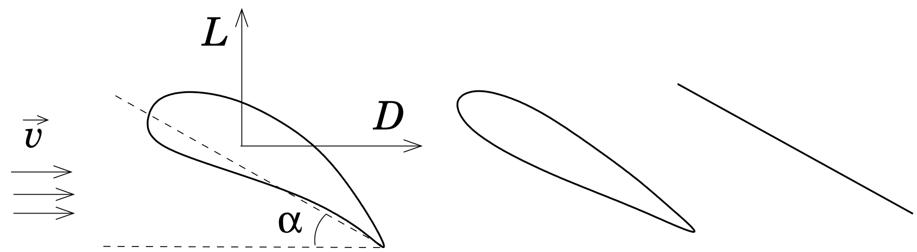
The Coanda effect has different causes depending on the situation. In the case of liquids, this happens mostly due to surface tension and the adhesive forces acting between the liquid and the boundary material it is in contact with. For gases, there is a different cause: If a surface curves away from a gas flow, the flow tends to blow away air molecules trapped in the small wedge formed between the initial flow direction and bent surface, creating a low pressure region. The gas flow then bends to fill up this low pressure void.




Instances of the Coanda effect are water falling on a vertically held curved surface or the bending down the air flow along the upper surface of an airfoil.

## 5.2 Qualitative explanation of flight

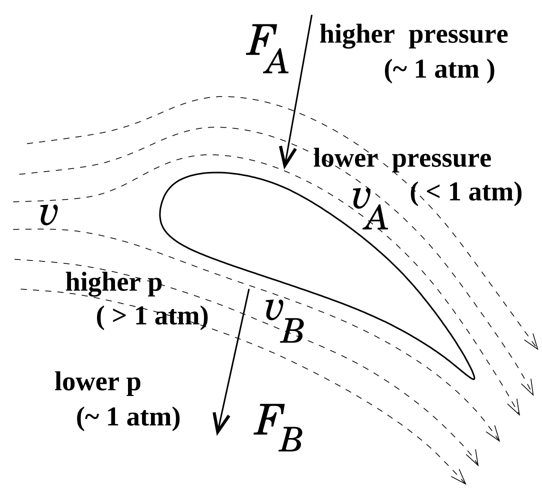
The vertical force that lifts an aeroplane is created by the air flow pattern around the wings. This pattern is essentially created by the shape of the cross section of the wing and the angle it makes with the direction of motion. The cross section of the wing is usually designed to have the shape of an **aerofile**, with a rounded front region and a sharp **trailing edge** (But in the simplest case, the wing can have a rectangular shape with a cross section which is simply a line). For simplicity one assumes that the wing has a uniform cross section throughout its length and consider forces per unit lenght of the wing. This enables one to concentrate on the two dimensional air flow pattern in the cross sectional plane of the wing.




In practice, it is the motion of the wing with a velocity $-\vec{v}$ through the air that creates the flow pattern around it. But often it is useful to consider a reference frame in which the wing is stationary and air rushes toward it with velocity $\vec{v}$ (this is the air velocity well before reaching the wing; the **upstream velocity**). In wind tunnel experiments it is covenient to use a set up with a fixed wing and moving air. In the following, we picture the wing as moving from right to left, or the air as moving from left to right.

The vertical force per unit length of the the wing is called the **lift** $L$. Among other things, $L$ depends on the angle that the aerofile makes with the direction of air flow, called the **angle of attack**, $\alpha$. This angle is defined such that $\alpha = 0$ corresponds to the aerofile orientation that generates zero lift ($L = 0$). The airflow around the aerofile also generates a horizontal force that opposes the motion of the wing through the air. Such a force, per unit length of the wing, is called the **Drag** $D$ (in our conventions, $D$ has the same direction as $\vec{v}$). External work is needed to overcome the drag.

We are now in a position to describe how $L$ and $D$ are generated by the air flow pattern around the wing. In a frame in which the wing is stationary, consider a uniform horizontal flow of air with speed $v$ toward the aerofile. As the flow reaches the aerofile, it splits into a part going above the wing with velocity a profile $\vec{v}_A$, and a part going below the wing with velocity profile $\vec{v}_B$. The Coanda effect now requires that the flow above the wing must "generally" follow the shape of the upper surface of the wing [2] and, hence, the flow




[2]It should be emphasized that although this may explain the general shape of the flow, it does not as

streamlines above the aerofile first rise up
and then bend down to follow the shape of the aerofile, generating a convex upward pattern. The flow below the wing bends down without rising up first, hence the streamlines are again convex upward, but with a smaller curvature as compared to the streamlines above the wing, which thus play a dominant role. The curvatures of the streamlines above the wing indicate a (generally) downward "centripetal" force that is exerted on the airflow by the wing. By Newton's third law, the airflow must exert and equal and oppositely directed reaction force on the wing pulling it upward. The vertical component of this force is the Lift $L$ and its horizontal component is the Drag $D$. The flow below the wing enhances this effect but to a lesser extent.

It is instructive to understand the lift in terms of pressure gradients (this is the explicit mechanism through which the reaction force is generated). Remember that in fluids, force vectors point from regions of higher pressure to regions of lower pressure. Then the downward "centripetal" force acting on the air above the wing implies a pressure gradient with a low pressure region just above the wing, and increasing pressure as one goes up. But the maximum pressure far above the wing is simply the atmospheric pressure. Hence the pressure just above the wing must be less than the atmospheric value. By a similar argument, the pressure just below the wing is somewhat higher than the atmospheric value (at least having the atmospheric value, if we ignore the flow curvature below the wing), approaching the atmospheric value far below the wing. The lower pressure region above the wing thus sucks the wing upward, giving rise to the lift. This will also generate a drag force as will be explained later.

The description above, of the flow patterns around the wing (with the extra assumption of flow seperation at the rear edge to be justified later) and of the associated forces and pressures, already provides a qualitative explanation of flight. This can be sharpened further by studying the velocity profile of the flow. While the incoming flow is horizontal, the convex nature of streamlines above the aerofile means that by the time the flow leaves the upper surface of the wing, it has acquired a downward velocity component. Thus the air stream leaving the wing has a downward momentum. By momentum conservation, this is balanced by the upward momentum of the plane (since, momentum change per unit time is equal to force, this is equivalent to Newton's third law).

## 5.3 The Kutta-Zhukovsky theorem

One can now apply Bernoulli's principle to the streamlines around the aerofile. First, note that there is a thin layer of air in direct contact with the wing for which viscosity effects cannot be neglected. Outside this region, the flow can be safely approximated as inviscid. This is so because at the high flow velocities we are concerned with, fluid particles get to

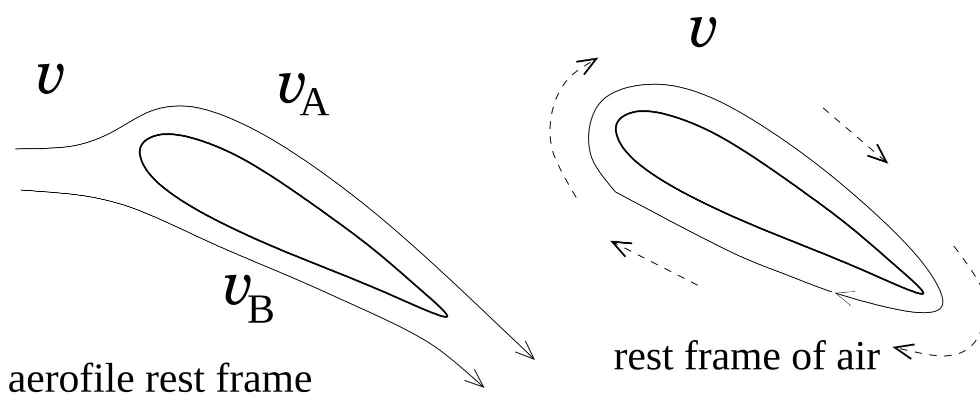
such specify the special flow features that generate lift. In particular, it does not specify where on the aerofile the rear stagnation point should be situated and where the flow should separate from the aerofile. We will discuss this in the subsection on the Kutta condition. Here we simply assume that the flow has a pattern as shown in the figure interact with the wing for a very short time. During this time, energy transfers due to viscosity are very small as compared to the inviscid energy exchanges between the fluid and the wing. After treating the flow as inviscid, an important effect due to viscosity in the boundary layer can be incorporated into the theory in terms of the Kutta condition to be discusses in the later subsections.

Consider two streamlines just outside the thin viscous region, one just above the wing and one just below it, along which the flow can be treated as inviscid. Well before reaching the wing, these two streamlines flow parallel to each other with very similar properties (that is, speed $v$ and pressure $p$). If flow speeds and pressures on the streamlines above and below the wing are denoted by $v_A, p_A$ and $v_B, p_B$, respectively, then,

$$\begin{aligned} \tfrac{1}{2}\rho v^2 + p &= \tfrac{1}{2}\rho v_A^2 + p_A \\ \tfrac{1}{2}\rho v^2 + p &= \tfrac{1}{2}\rho v_B^2 + p_B \end{aligned} \tag{5}$$

Here, $p$ is the atmospheric pressure, and from our discussion above it follows that $p_A < p$ and $p_B > p$ (this must be the case for lift to get generated). Bernoulli's principle then implies that $v_A > v > v_B$. Thus the air above the wing moves faster than the air below it.

To see what this velocity profile entails, let us consider this problem in a frame in which the air is stationary and the wing is moving against it with velocity $-\vec{v}$, say, from right to the left. Then the upstream air velocity in front of the aerofile is zero, above the aerofile velocity is positive (that is, directed from left to right) and below the aerofile velocity is negative (directed from right to left). Thus if we take a snap shot of the velocities of fluid



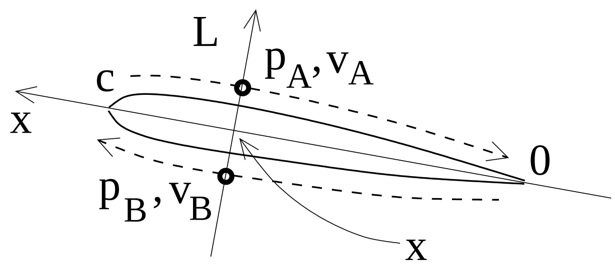

particles on the two streamlines, the velocity vectors give rise to a **circulation** around the aerofile in the clockwise direction [3]. Mathematically, the circulation is quantified by

$$\Gamma = \oint_C \vec{v} \cdot \vec{dl}$$

where $C$ is a path around the aerofile which, by convention, is taken to run counter-clockwise. Although it was easy to visualize the circulation $\Gamma$ in a given frame, its actual

[3] The term "circulation" here refers only to a configuration of velocity vectors, it does not mean that fluid particles circulate around the aerofile.

value does not depend on the choice of the inertial frame (since the frames are related by a constant shift of velocity and a constant vector field has zero loop integral). Note that the existence of circulation was inferred from the requirement that pressure above the wing is lower than that below it. Conversely, given a circulation, one infers a pressure difference based on the velocity differences. We now show that the circulation $\Gamma$ determines the lift $L$, a result known as the Kutta-Zhukovsky theorem.




Let $c$ denote the width (**chord**) of the wing and consider a coordinate axis parallel to this width (to fix ideas, one may consider a very thin wing, or simply a rectangular wing the cross section of which is a line). At a point $x$ on the coordinate axis, the pressures and velocities on the two streamlines just above and below the wing are $p_A(x), p_B(x)$ and $\vec{v}_A(x), \vec{v}_B(x)$, respectively. Bernoulli's principle as in equations (5), applied to points corresponding to $x$ on the two streamlines then gives

$$p_A(x) - p_B(x) = \frac{1}{2}\rho(v_A + v_B)(v_B - v_A) = \frac{1}{2}\rho(2v + \delta v_A + \delta v_B)(v_B(x) - v_A(x))$$

In practice, $|\delta v_{A,B}| = |v_{A,B} - v| << v$ and since $(v_B - v_A)$ is of the same order as $\delta v_{A,B}$, we can write

$$p_A(x) - p_B(x) \approx -\rho\, v\ (v_A(x) - v_B(x))$$

This gives the upward force per unit length *i.e.*, lift, as

$$L = -\int_0^c dx\,(p_A(x) - p_B(x)) = \rho v \int_0^c dx(v_A(x) - v_B(x))$$

On the other hand, for the flow near the thin wing,

$$\Gamma \approx \int_0^c v_B dx + \int_c^0 v_A dx = \int_0^c v_B dx - \int_0^c v_A dx$$

Hence we have,

$$L = -\rho\, v\, \Gamma$$

This is the **Kutta-Zhukovsky** theorem. In our setup $\Gamma$ is negative (since $\vec{v}$ around the wing has a clockwise flow while the curve $C$ runs counter-clockwise) and $v$ is positive. Hence $L$ is positive corresponding to an upward direction. Although here the theorem is derived in the context of an aerofile, it applies more generally to 2-dimensional flows around objects.

Note that, stricktly speaking, $L$ is perpendicular to axis that we introduced along the wing, essentially at an angle $\alpha$ with the horizontal direction. Hence lift consists only of the component of $L$ perpendicular on the horizontal direction. The component in the horizontal direction gives drag.

To summarize, lift is due to pressure difference across the wing caused by the curvature of air flow around it. Bernoulli's principle allows us to relate pressures to velocities and hence to express this lift in terms of the circulation around the wing.

## 5.4 A topological issue

When deriving the Bernoulli principle, we defined an irrotational flow as one characterized by the condition $\vec{\nabla} \times \vec{v} = 0$. Ordinarily, through Stockes's theorem,

$$\oint_{C=\partial S} \vec{v} \cdot d\vec{l} = \int_S (\vec{\nabla} \times \vec{v}) \cdot d\vec{S} \,,$$

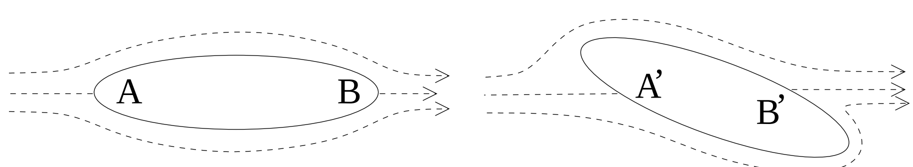
this implies the vanishing of circulation, $\Gamma = 0$. There is, however, a caveat here: one must be able to identidy the curve $C$ as the boundary of some surface $S$ through the fluid so that the $\vec{v}(\vec{x})$ is defined everywhere over it. For our contour $C$ around an aerofile, this is not possible in a purely 2-dimensional setup because the aerofile acts as a **defect** in space preventing us from filling the region within the curve with a surface $S$. For the same reason, the curve $C$ cannot be contracted to a point however it is deformed and the space is not **simply connected** (rather, it is multiply connected). Such a two dimensional setup is equivalent to a wing of infinite length in 3 dimensions. Since Stockes's theorem does not apply to such cases, it is possible to have non-zero circulation around non-contractible loops even in an irrotational flow [4]. Moreover, the shape of $C$ is not relevant to the value of the circulation (since in the region where $\vec{\nabla} \times \vec{v} = 0$ we can deform contours to each other without affecting the value of circulation).

However, if the wing has a finite length (as is really the case), then it is always possible to construct a surface that goes over the open end of the wing with $C$ as its boundary. In such a case Stockes's theorem is applicable and therefore $\Gamma \neq 0$ implies $\vec{\nabla} \times \vec{v} \neq 0$.

## 5.5 The Kutta condition

As we have seen, the existence of a non-zero circulation around the wing is crucial for the generation of lift. The **Kutta condition** describes the origin of this circulation and how it generates the correct flow pattern around the wing; something that we have so far assumed without a proper explanation (it was emphasized that this aspect of the flow cannot be explained in terms of the Coanda effect).

Let us consider a wing the cross section of which is an ellipse instead of an aerofile. This is placed in an airstream at zero angle of attack, $\alpha = 0$, meaning that the major axis of ellipse is parallel to the upstream velocity of the flow. The flow now has a front

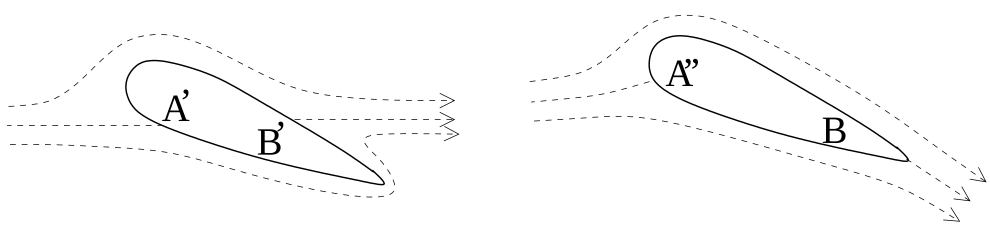



stagnation point $A$ and a rear stagnation point $B$, these being the points where the major

[4] Exactly the same mathematics is required for understanding a very important phenomenon in quantum physics; the Aharonov-Bohm effect. In this case, $\vec{v}$ is replaced by the magnetic vector potential $\vec{A}$ and the wing is replaced by a long current carrying solenoid

axis intersects the ellipse. Let us now increase the angle of attack slowly to some value $\alpha$. The front stagnation point moves to some point $A'$ below $A$ while the rear stagnation point moves to some point $B'$ above $B$ on the upper side of the ellipse. This means that at the rear end, the flow streamlines bend around at $B$ and continue following the contour of the ellipse upwards until they detach in the neighbourhood of the new stagnation point $B'$. The location of $B'$ is determined by the interaction between the flows below and above the ellipse. Even though the angle of attack is non-zero, the flow has zero circulation and hence no lift is generated (The flow curvature is such that the upward force generated near the front region is balance by a downward force around the rear end). Such a wing cross section is obviously not appropriate for flight.

Let us now replace the rounded rear end of the ellipse around $B$ by a sharp edge so that the wing cross section becomes an aerofile with a sharp trailing edge at $B$. As in the case of ellipse, the flow below the wing tends to turn around the sharp edge at $B$ and go up the aerofile until $B'$. This indeed is the flow pattern for a short time interval after the start of the flow. However, since $B$ is now a sharp edge, the flow must take a very sharp turn if it is to follow the shape of the rear end of the aerofile. This implies a large change in velocity over a short time interval and hance a very large force. But fluids cannot sustain large (unballanced) forces and hence the flow tends to detach from the aerofile at $B$ rather than move up to $B'$. This also eliminates the resistance that the flow above the wing encountered at $B'$, which then continues following the shape of the aerofile until it too detaches at $B$. As a result of this the rear stagnation point moves down from $B'$ back to $B$ where it stays.



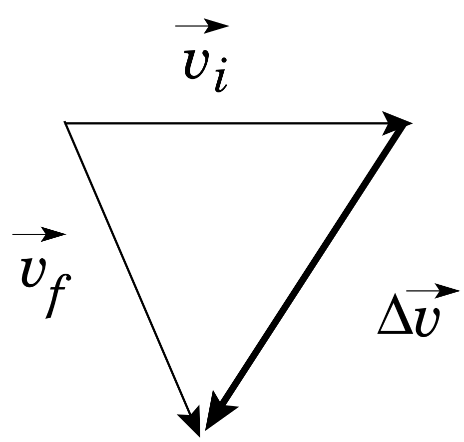

The downward shift of the rear stagnation point is accompanied by the generation of a non-zero circulation around the wing as follows: Soon after the start of the flow, when the stagnation point is still at $B'$, let us denote the velocity field around the aerofile by $\vec{v}_1$. In particular, this is such that at $B$ the velocity is non-zero, between $B$ and $B'$ air adjacent to the aerofile moves upward and $\oint \vec{v}_1 \cdot \vec{dl} = 0$. The interaction of air with the aerofile changes the flow adjacent to the wing by generating a another velocity field $\vec{v}_2$ with a circulation around the wing $\oint \vec{v}_2 \cdot \vec{dl} = \Gamma \neq 0$ such that $\vec{v} = \vec{v}_1 + \vec{v}_2$ is zero at $B$ and not at $B'$. Hence, to keep the stagnation point at $B$ we need a specific circulation $\Gamma = \oint \vec{v} \cdot \vec{dl}$ around the wing. Thus the **Kutta condition** states that, under stable flow conditions, an aerofile creats a circulation in the velocity field around itself such that the rear stagnation point is maintained at the sharp trailing edge.

Note that the Kutta condition does not intend to explain the physical origin of circulation (which we have qualitatively explained in terms of velocity differences above and below the wing related, by the Bernoulli principle, to pressure differences). Rather, it gives a criterion for determining the circulation $\Gamma$ using the requirement of keeping the rear stagnation point at the trailing edge (instead of doing complicated calculations using the Navier-Stokes equation in the presence of the viscous forces).

For the sake of completeness we should also address the issue of conservation of circulation in inviscid flows. Consider a loop in a fluid moving along with it such that it contains the same set of particles at all times. At a given instant of time, we can evaluate the circulation around this loop. It can be shown (although not here) that in an inviscid flow, the value of this circulation remains constant in time. This result if known as **Kelvin's circulation theorem**. In particular, if the circulation was zero, it will retain that value during flow. This seems to contradict the essence of the Kutta condition where a circulation $\Gamma$ is created around an aerofile by increasing its angle of attack from zero to some non-zero value. The resolution is that at the same time that the circulation $\Gamma$ forms around the aerofile, the downward motion of the stagnation point from $B'$ to $B$ also creates a vortex with circulation $-\Gamma$ (where, in our conventions, the velocity field circulates in the counter-clockwise direction) at the trailing edge. This vortex is not bound to the wing and is left behind as the wing moves on. It is called the **starting vortex**. The creation of the starting vortex insures that the total circulation remains zero in agreement with Kelvin's circulation theorem. With zero viscosity, this vortex left behind would continue forever, but in the real world it dissipates because of air viscosity (which cannot be ignored over the longer time scales relevant to the decay of starting vortex).

## 5.6 The origin of drag and the "level flight" condition

We have stated that, beside the lift, air flow around the wing also created a **drag**, a force parallel to the upstream flow direction that opposes the horizontal motion of the wing in air. Drag is caused mainly by two effects (beside turbulence): Friction between air and the wing surface gives rise to **friction drag**, $D_{fric}$. Another component of $D$ is the **induced drag** $D_{ind}$ which has a non-viscous origin and simply follows from the curving of the air flow. Let us denote the initial upstream flow velocity by $\vec{v}_i$ and the final average velocity of the flow emerging from the trailing edge of the aerofile by $\vec{v}_f$. The magnitude of $\vec{v}_f$ is close to that of $\vec{v}_i$ but it has a downward component. The change $\Delta\vec{v} = \vec{v}_f - \vec{v}_i$ is related to the total generally downward force on the air flow by the wing. An equal and opposite force then acts on the wing. $\Delta\vec{v}$ has a purely downward component $\Delta\vec{v}_\perp$, normal to the initial flow direction $\vec{v}$ and a component along $\vec{v}$, $\Delta\vec{v}_{||}$. Since the magnitudes of $\vec{v}_i$ and $\vec{v}_f$ are not too different, $\Delta\vec{v}_{||}$ is directed opposite to $\vec{v}$. Hence the direction of the reaction force on the wing is along $\vec{v}$, that is, it opposes the motion of the wing. This is the



induced drag.

Let us now briefly look at the energetics of flight. As the plane is moving upward, the lift does work against gravity and also provides kinetic energy due to upward motion. One also needs a supply of energy to work against the drag and supply the planes kinetic energy due to its forward motion (if that velocity changes). After the plane levels up, the situation simplifies. There is no upward displacement and hence no work associated with it. The lift in only required to ballance the weight of the plane,

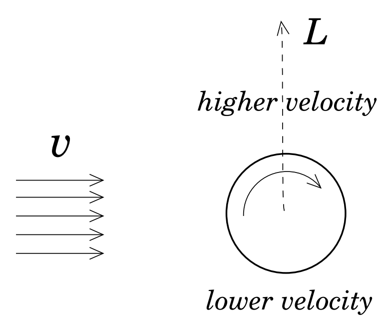
$$L \times (\text{wing span}) = M\,g$$

The power (work per unit time) supplied by the engine is then only needed to overcome the drag effects,

$$P_{engine} = D\,v$$

For a given aerofile and flow conditions, $L$ and $D$ can be computed in terms of the parameters of the theory.

### 5.7 Spinning bodies moving in fluids (Magnus effect)




Consider a rotating cylinder in a fluid flow normal. The rotation of the cylinder drags the adjacent fluid along, setting up a non-zero circulation. The 2 relevant directions are: the axis of spin and the direction of flow (for simplicity, one can take these to be perpendicular to each other). Then there is a force on the cylinder in the 3rd direction as implied by the Kutta-Zhukovsky theorem. The physical origin of the force can also be understood by studying the strealines and velocity profile around the cylinder, where the rotation causes fluid velocity on one side to be higher than the other side. Then the force direction can be inferred from the application of Bernoulli's principle, or from the curvatures of streamlines on both sides of the cylinder.

## 6 A Simple Model of Flight

See "Flight without Bernoulli" by Chris Waltham at

`http://www.physics.ubc.ca/~waltham/air/FwB.pdf`

## 7 Further Reading

- For detailed discussions of fluid mechanics see some of the many books on the subject, for example,

  - Mechanics of Fluids, by B.S. Massey
  - Physical Fluid Dynamics, D.J. Tritton

- A very good reference for technical as well as background information is the free web encyclopedia, **Wikipedia**, and the references therein.

- Some more resources on the web (with nice animationa and detailed explanations):

  `http://www.av8n.com/how/`
  `http://www.diam.unige.it/~irro/`
  `http://firstflight.open.ac.uk/aerodynamics/index.html`

  Popular and less technical descriptions of flight and misinterpretations of Bernoulli's principle:

  `http://user.uni-frankfurt.de/~weltner/Mis6/mis6.html`
  `http://user.uni-frankfurt.de/~weltner/Flight/PHYSIC4.htm`