Pearson BTEC Level 4 Higher Nationals in Engineering (RQF)

## Unit 11: Fluid Mechanics

# Unit Workbook 3

in a series of 4 for this unit

Learning Outcome 3

# Dynamic Fluid Flow

# Contents

# INTRODUCTION

**Investigate dynamic fluid parameters of real fluid flow**

- *Fluid flow theory:*
  - Energy present within a flowing fluid and the formulation of Bernoulli's equation.
  - Classification of fluid flow using Reynolds numbers.
  - Calculations of flow within pipelines.
  - Head losses that occur within a fluid flowing in a pipeline.
  - Viscous drag resulting from fluid flow and the formulation of the drag equation.
- *Aerodynamics:*
  - Application of prior theory of fluid flow to aerodynamics.
  - Principles of aerofoils and how drag induces lift.
  - Flow measuring devices and their operating principles.

# GUIDANCE

This document is prepared to break the unit material down into bite size chunks. You will see the learning outcomes above treated in their own sections. Therein you will encounter the following structures;

**Purpose**

Explains *why* you need to study the current section of material. Quite often learners are put off by material which does not initially seem to be relevant to a topic or profession. Once you understand the importance of new learning or theory you will embrace the concepts more readily.

**Theory**

Conveys new material to you in a straightforward fashion. To support the treatments in this section you are strongly advised to follow the given hyperlinks, which may be useful documents or applications on the web.

**Example**

The examples/worked examples are presented in a knowledge-building order. Make sure you follow them all through. If you are feeling confident then you might like to treat an example as a question, in which case cover it up and have a go yourself. Many of the examples given resemble assignment questions which will come your way, so follow them through diligently.

**Question**

Questions should not be avoided if you are determined to learn. Please do take the time to tackle each of the given questions, in the order in which they are presented. The order is important, as further knowledge and confidence is built upon previous knowledge and confidence. As an Online Learner it is important that the answers to questions are immediately available to you. Contact your Unit Tutor if you need help.

**Challenge**

You can really cement your new knowledge by undertaking the challenges. A challenge could be to download software and perform an exercise. An alternative challenge might involve a practical activity or other form of research.

**Video**

Videos on the web can be very useful supplements to your distance learning efforts. Wherever an online video(s) will help you then it will be hyperlinked at the appropriate point.

### 4.1.3 The Reynold's Number

They Reynold's number (Re) is a dimensionless constant that is used to describe fluid flow. The Reynold's number is calculated using Eq.3.1, and is a ratio of inertia forces to viscous forces.

$$\mathrm{Re} = \frac{\rho u L}{\mu} = \frac{uL}{v} \quad (3.1)$$

Where:

- ρ is the density of the fluid $(\mathrm{kg/m^3})$
- u is the velocity of the fluid $(\mathrm{m/s})$
- L is the length that you are measuring over $(\mathrm{m})$
- μ is the dynamic viscosity $(\mathrm{Pa \cdot s})$
- v is the kinematic viscosity $(\mathrm{m^2/s})$

The corresponding value for the Reynold's number defines the flow as:

- $\mathrm{Re} < 2000$: Flow is laminar
- $\mathrm{Re} = 2000$: Known as the critical Reynold's number, flow is no longer laminar and will start to transition towards turbulent flow
- $2000 < \mathrm{Re} < 4000$: Flow is considered transitional, or unstable, it is not laminar, but it is not fully turbulent yet either.
- $4000 < \mathrm{Re}$: Flow is turbulent

**Example 1**

Give the flow characteristic of:

a) Honey ($1450\ \mathrm{kg/m^3}$, $14.095\ \mathrm{Pa \cdot s}$) flowing through a $3\ \mathrm{m}$ length of pipe at $0.3\ \mathrm{m/s}$.
b) Castor oil ($961\ \mathrm{kg/m^3}$, $950\ \mathrm{cP}$) flowing through $1\ \mathrm{m}$ length of pipe at $20\ \mathrm{m/s}$
c) Water ($1000\ kg/m^3$, $1\ cP$) flowing through a $1\ m$ length of pipe at $0.3\ m/s$.

**Answers:**

a) $Re$ is given as:

$$\mathrm{Re} = \frac{\rho u L}{\mu} = \frac{1450(0.3)(3)}{14.095} = \mathbf{92.6}$$

The flow is laminar

b) $950\ \mathrm{cP} = 9.5\ \mathrm{Pa \cdot s}$, so Re is:

$$\mathrm{Re} = \frac{961(20)(1)}{9.5} = \mathbf{2023}$$

The flow is transitional

$$\text{Relative Roughness} = \frac{\text{k}}{\text{d}} \quad (3.2)$$

Fig.3.5 gives a graphical demonstration of the dimensions.

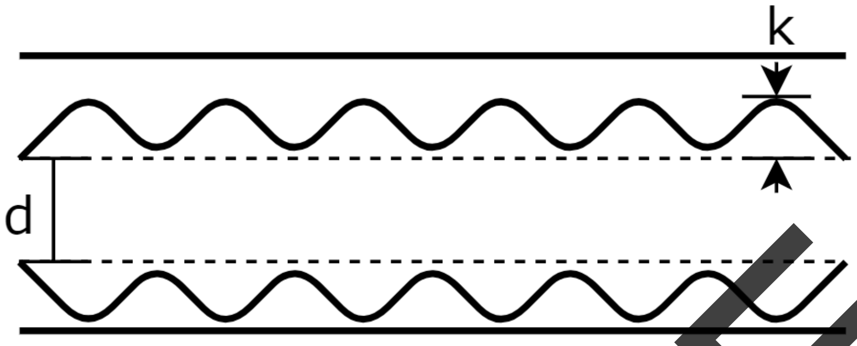


*Figure 3.5: The measurement of the diameter and height of irregularities in a pipe.*

## 4.2.2 Moody Diagrams

A Moody diagram is a diagram to measure the coefficient of friction in pipes (noted in this workbook as f, although some sources will use λ). Calculating the coefficient of friction (sometimes referred to as the friction factor) depends on both the flow in the pipe and its relative roughness.

The equations used for calculating f are:

Laminar: $$\text{f} = \frac{16}{\text{Re}} \quad (3.3)$$

Turbulent Smooth pipes: $$\text{f} = 0.079Re^{-0.25} \quad (3.4)$$

Turbulent Rough pipes: $$\frac{1}{\sqrt{f}} = -3.6\, log_{10}\left[\frac{6.9}{Re} + \left(\frac{k}{3.71d}\right)^{1.11}\right] \quad (3.5)$$

These equations are quite long to calculate, with the exception of the laminar equation (Eq.3.3). So, alternatively, the Moody chart is available for reference. The Moody chart is a graph that has already plotted the values for the friction factor across a range of Reynold's numbers, and relative roughness, to give a quick (and fairly accurate) estimate. Fig.3.6 shows a Moody chart, and the lines show the variation of friction factor at a given relative roughness, but a varying Reynold's number. Most Moody diagrams will also include an absolute roughness value for some materials. The absolute roughness value is a typical estimate for ϵ for certain materials.

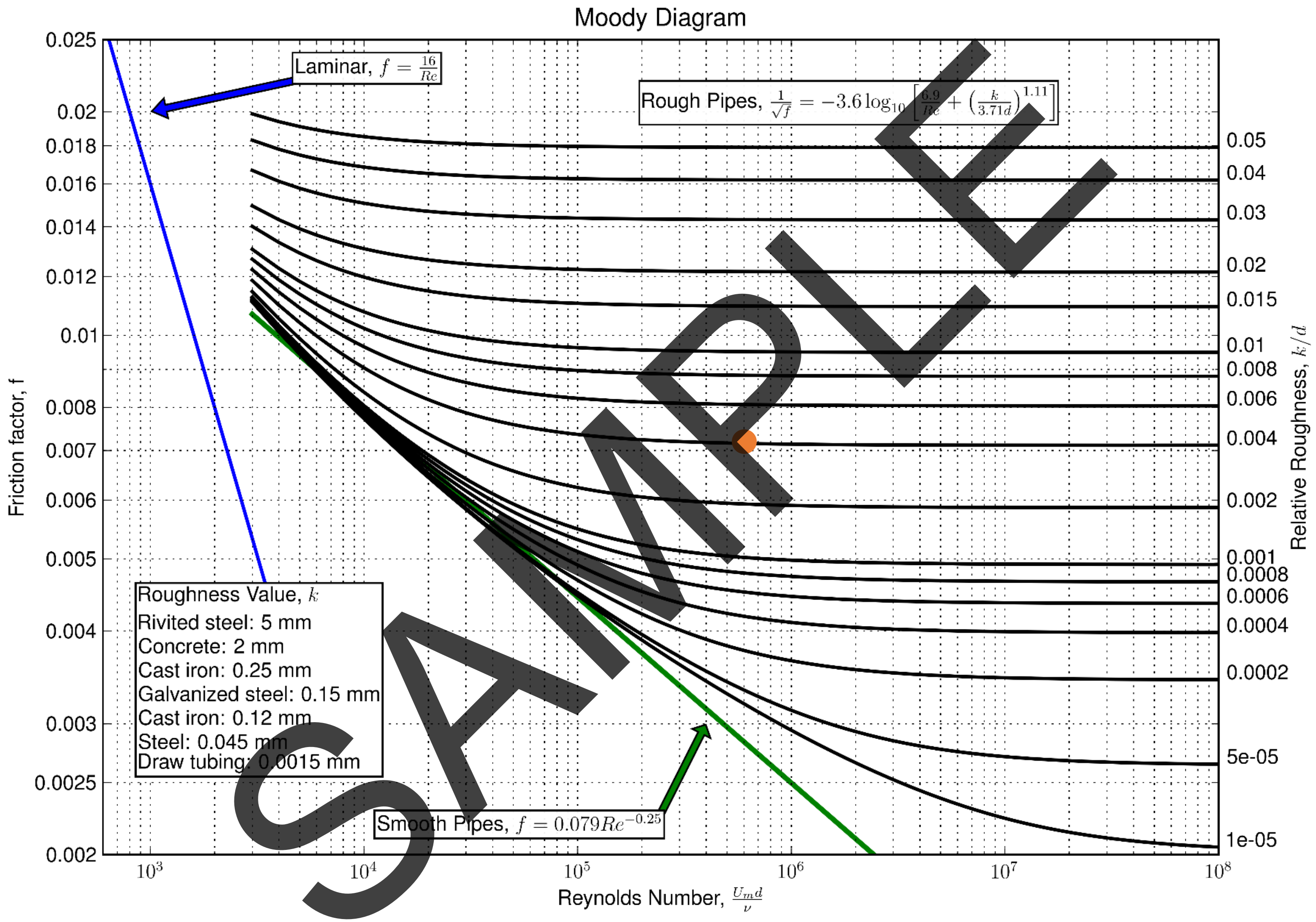


*Figure 3.6: Moody Diagram*

### 3.2.3 Bernoulli's Equation

Bernoulli's equation is a conservation of energy equation used in fluid mechanics. This can draw similarities to the thermodynamic Steady Flow Energy Equation (SFEE).

$$Q - W = \left(U_2 + \frac{1}{2}mc_2^2 + mgz_2\right) - \left(U_1 + \frac{1}{2}mc_1^2 + mgz_1\right)$$

Where:

- $Q$ is the heat transferred through the system
- $W$ is the work done in the system
- $U_i$ is the internal energy of the system at point $i$
- $\frac{1}{2}mc_i^2$ is the kinetic energy at point $i$
- $mgz_i$ is the potential energy at point $i$

The same principle can be applied to flow through a pipe, since no heat is transferred in the flow of the pipe. Bernoulli's equation is given as Eq.3.7:

$$\mathrm{P_1} + \frac{1}{2}\rho \mathrm{u}_1^2 + \rho \mathrm{g} \mathrm{h}_1 = \mathrm{P_2} + \frac{1}{2}\rho \mathrm{u}_2^2 + \rho \mathrm{g} \mathrm{h}_2 \quad (3.7)$$

Where:

- $P$ is the pressure at a given point ($Pa$)
- $\rho$ is the density ($kg/m^3$)
- $u$ is the velocity at a given point ($m/s$)
- $g$ is acceleration due to gravity ($m/s^2$)
- $h$ is the height at a given point ($m$)

When using Bernoulli's equation, it's always important to know the equation for volumetric flow rate $\dot{\mathrm{Q}}$, which is Eq.3.8, where $\mathrm{A}$ is the area of the pipe.

$$\dot{\mathrm{Q}} = \mathrm{u} \cdot \mathrm{A} \quad (3.8)$$

Where:

- u is the velocity of the fluid (m/s)
- A is the cross-sectional area ($m^2$)

**Example 4**

Fig.3.7 shows water $\rho = 1000\ \mathrm{kg/m^3}$ travelling through a converging-diverging pipe. The figure also gives the known values for each point in the pipe. Calculate

a) The volumetric flow rate
b) The velocity at point 2
c) The pressure at point 2

# 3.3 Drag

## 3.3.1 Drag Around a Sphere

Consider Fig.3.8, which shows fluid streams passing around a sphere. From the Figure, it can be seen that the fluid will move around the sphere and slowly move back to its original state, there is also a generation of turbulent eddy currents that get caught behind the sphere.

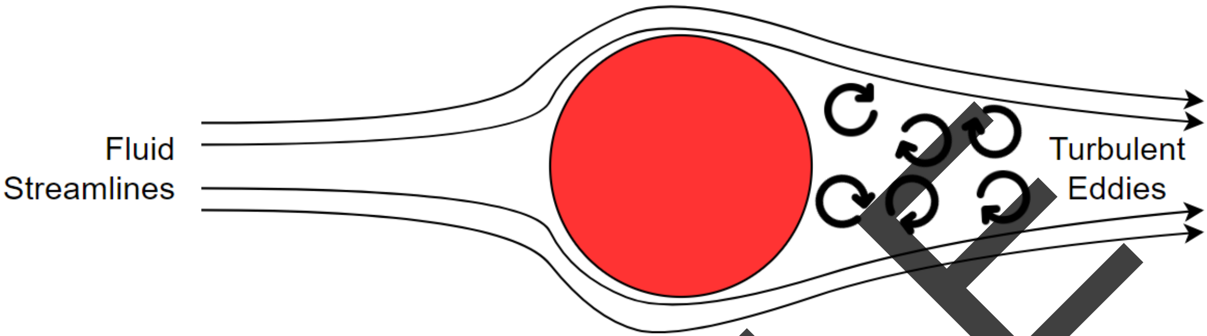


*Figure 3.8: Streamlines against a sphere*

There are two types of drag that can exist in a system; viscous drag and pressure drag.

## 3.3.2 Viscous Drag

Viscous Drag, or friction drag, occurs on the contact point of the surface of the object in the streamline. This type of drag is so-called because it is caused by the viscosity of the fluid itself and involves the transition from laminar drag to turbulent drag. It therefore stands that if the Reynold's number is higher, then there is less friction drag. Fig.3.9 shows the development of the "boundary layer" on a flat plate, which is the layer of fluid affected by viscous drag.

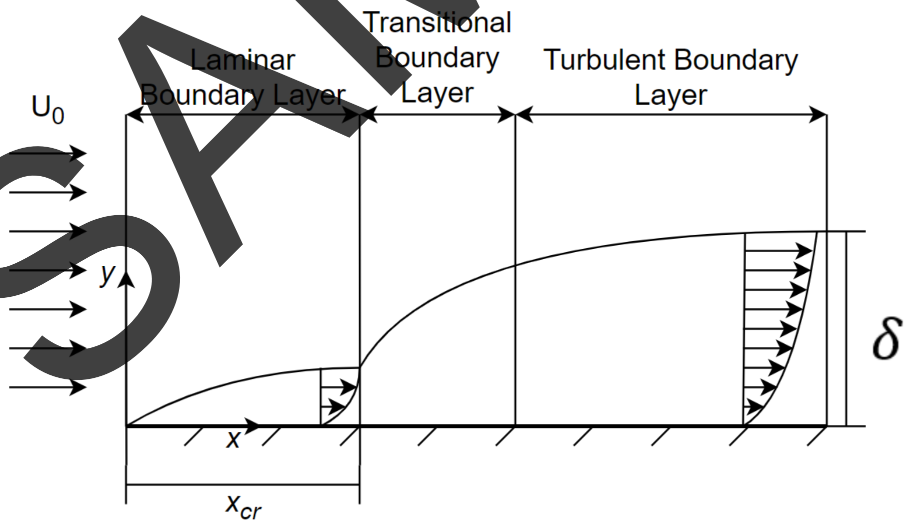


*Figure 3.9: The development of a boundary layer along a flat plate.*

The boundary layer is the stream that is travelling slower than the free stream speed, $U_0$. The further along the flat plate travelled shows an increase in the boundary layer thickness $\delta$. There is also a given distance along the plate $x_{cr}$ when the laminar boundary layer's Reynold's number moves higher than 2000, the fluid in the boundary layer transitional, before moving to turbulent.

### 3.3.3 Pressure Drag

Pressure drag is caused by a differential in pressure due to the shape of the plate. Consider a plate that is placed perpendicular to the air flow of the system, shown in Fig.3.10; as the fluid moves around the plate, there will a tail behind it as it tries to occupy the space. In this separation, turbulent eddy currents form in the gap, causing a pressure drop behind the surface. The pressure difference means that there is a force imbalance, consequently the plate is pushed backwards.

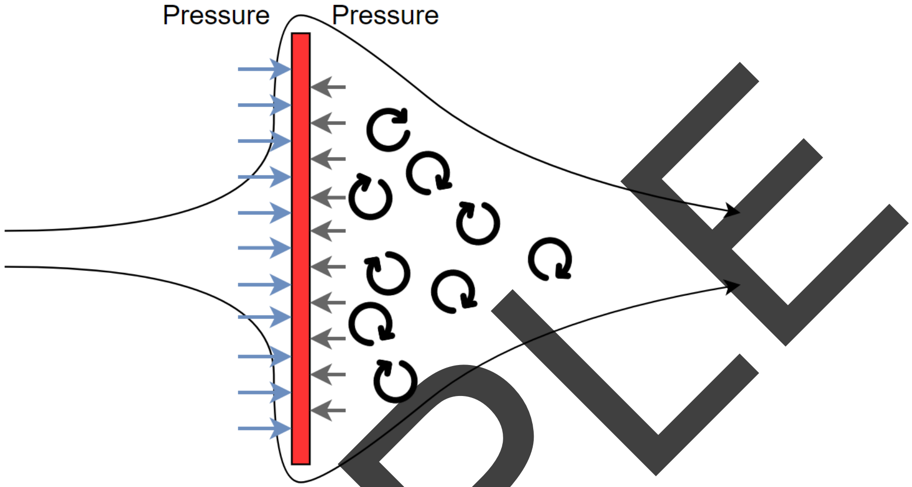


*Figure 3.10: Streamlines against a flat plate*

This type of drag is dominated by the formation of a wake (the turbulent eddies in the separation of the flow), which is governed by the shape of the object, rather than the viscosity of the fluid.

### 3.3.4 The Drag Equation

The amount of drag force that a body experiences against a moving fluid is given as Eq.3.9:

$$F_d = C_d \times \frac{1}{2}(\rho u^2)DL \quad (3.9)$$

Where:

- $F_d$ is the force of drag.
- $C_d$ is the coefficient of drag, a dimensionless constant that will vary between shapes.
- $\rho$ is the fluid density.
- $u$ is the fluid speed.
- $D$ is the diameter of the body's face.
- $L$ is the length of the body.

The drag coefficient is usually determined experimentally, and will vary depending on the amount of friction and pressure drag that is acting on the body. The form of drag that dominates the overall force can be seen as a ratio of the body's diameter and its length, and is shown in Fig.3.11. From the figure, it can be seen that there is a minimal point for total drag, which is an important piece of information.

# 3.4 Aerodynamics

Aerodynamics is how a body behaves when moving through a fluid (or if a fluid is moving through the body, depending on your frame of reference). By understanding the effects of aerodynamics, it can be applied to improve a number of systems.

## 3.4.1 Using the Drag Equation

The drag equation discussed in Section 3.3 has only discussed a body perpendicular to the fluid flow. But what if an "angle of attack" (commonly noted in mathematics as α) was introduced to the system. The angle of attack is altering the position of the body in order to direct the force, rather than the force just acting in the same direction as the fluid.

Consider the aerofoil in the Fig.3.12. In this scenario, the centreline of the aerofoil (otherwise known as the chord), is not in line with the flight path; as α increases, and the angle of attack becomes greater, there will be a greater pressure drag acting on the aerofoil, and the flow will begin to apply a force that will push the aerofoil upwards, giving "lift".

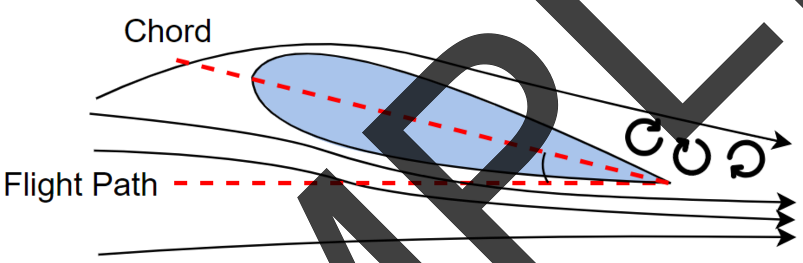


*Figure 3.12: An aerofoil with an angle of attack against the flight path.*

## 3.4.2 Coefficient of Lift

The coefficient of lift $C_L$ is a term used to describe the lifting force $F_L$ that acts on an aerofoil, as shown by Eq.3.10, and is essentially similar to Eq.3.9, since the lift relies on the drag force.

$$C_L = F_L \times \frac{1}{2}(\rho u^2)DL \quad (3.10)$$

The highest value for $C_L$ occurs at the critical angle of attack $\alpha_{cr}$. Once the angle passes this, $C_L$ begins to drop, and $C_d$ increases substantially, thus the system begins to "stall".

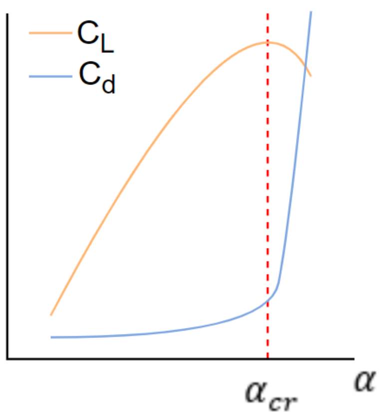


*Figure 3.13: $C_d$ and $C_L$ relative to $\alpha$*

#### 3.4.2.3 Cycling

There is a common misconception about why cyclists shave their legs, with people usually answering that "it makes them more aerodynamic". The real reason is that it is much easier to clean a wound in the event of a crash; the hair on their body makes a negligible impact on aerodynamics in the air.

Over the past decade, the UK has endured a golden age of cycling, with huge successes during the Olympics and cycling endurance competitions, such as the Tour De France. While this is largely part of the training facilities available, it is also largely due to the research and development the cycling teams have completed, to the extent where competitors insinuate that there is foul play.

The reality of the situation is that British cycling focuses on Olympics, and while they use standard equipment during the European and World championships, the teams use specialist equipment that has been heavily developed during the four-year gap that gives them an incredible competitive edge during the games. The helmets, bicycles and suits are constantly tested in wind tunnels and developed further.

#### 3.4.2.4 Swimming

In the case of water, hair does have a significant impact on aerodynamics. Hence why all swimmers are hairless, with the exception of their head, which is covered by a smooth swimming cap.

#### 3.4.2.5 Flight

Commercial planes (passenger flights, cargo etc.) do not have a high thrust to weight ratio, meaning that they need a high drag coefficient to produce the lift required to begin the climb. Fig.3.15 shows a CFD analysis on an aircraft's surface. The measurements recorded are the pressure coefficients at each point on the aircraft. Notice that the top of the aircraft wings are exhibiting a large negative coefficient, this means that there will be a large force acting upwards on the wings, thus providing the lift that is required to get them off the ground.

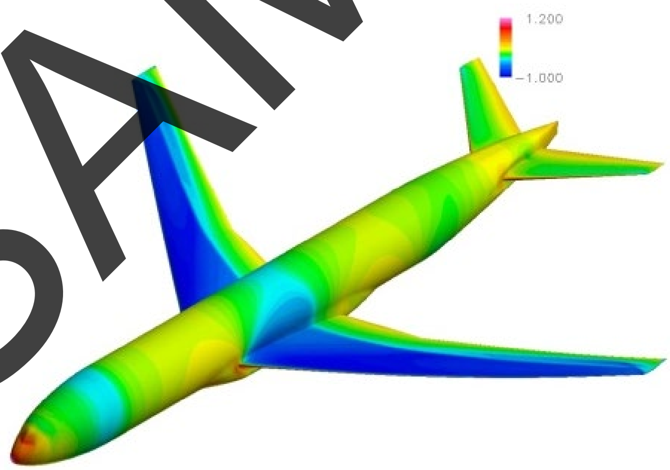


*Figure 3.15: CFD analysis of an aircraft*

The coefficient of drag (and therefore lift) is adjusted using the slats at the front of the wings, and flaps on the back. The slats and flaps can extend and retract to increase the area of the wings, which will increase lift. To increase lift even further, the slats and flaps can also rotate downwards. Slats and flaps are typically deployed during low speed operations (take off, landing, etc.). Having the flaps and slats deployed does have a significant impact on the drag of the plane, which will, in turn, reduce fuel efficiency. However, the high coefficient of lift is only required to reach the cruising altitude, after that, the coefficient of lift can be reduced to maintain the altitude of the plane, and so the slats and flaps can be adjusted hydraulically to reduce drag.

Drag is also an important concern during cruising, but by reducing the amount of air that causes drag on the plane, the plane can improve fuel efficiency. This is why cruising altitude for a commercial flight is $43{,}100$ feet ($8.1$ miles/$13$km). The low pressure at this altitude means less drag, but is a fair compromise for the engines that need the oxygen in the air to allow combustion.

Fighter jets need to be agile and quick to turn during combat, and so there needs to be as little drag force as possible. This does mean that there is almost no lift provided by the wings themselves, and relies on the incredible thrust provided by the jet. Considering the lack of lift provided by the wings, there needs to be a lot of micro-adjustments made constantly to keep the jet stable, this means that jets are "fly-by-wire" as the requirements for the adjustments are too complex for the pilot to complete (after all, the pilot most likely has more important things to worry about during combat). Computer failure would simply result in a complete loss of control in the jet.

Fighter jets do have a thrust to weight ratio higher than or equal to 1:1, meaning that these jets can fly vertically without much trouble, and hence why these jets can also perform such advanced manoeuvres.

# 3.5 Measuring Fluid Flow

Measuring fluid flow is important for a number of applications:

- Measuring air intake for combustion engines, and maintaining fuel efficient combustion by altering the fuel injection volume.
- Checking for potential blockages in pipework
- Testing heating, ventilation, air conditioning and cooling (HVAC) systems

There are five common systems to measure fluid flow.

## 3.5.1 Hot Wire Sensor

The hot wire sensor relies on the laws of convective heat transfer. A heated electrical wire is placed in the tube, connected to a current sensor. As the fluid passes the heated wire, convective heat transfer will cool the wire. As the wire cools, its resistance decreases, which, provided a constant voltage is passed through the wire, means the current will increase. The current reading in the system will relay this information back to give the flow rate.

## 3.5.2 Vane Meters

Vane meters work by changing the area of the pipe by installing a vane (a partial blockage) in the system. As the water works its way past the bluff vane, there is separation behind the vane, where there will be a small amount of turbulence. This will create a lot of pressure drag and forces the vane to rotate. The vane will be attached to a torsion spring, and the resistance this provides can be measured, and the flow can be calculated. Once the flow rate drops, the spring will return the vane to its original position. Fig.3.16 shows the principle of a vane flowmeter, and also shows the motion of fluid flow throughout the system.