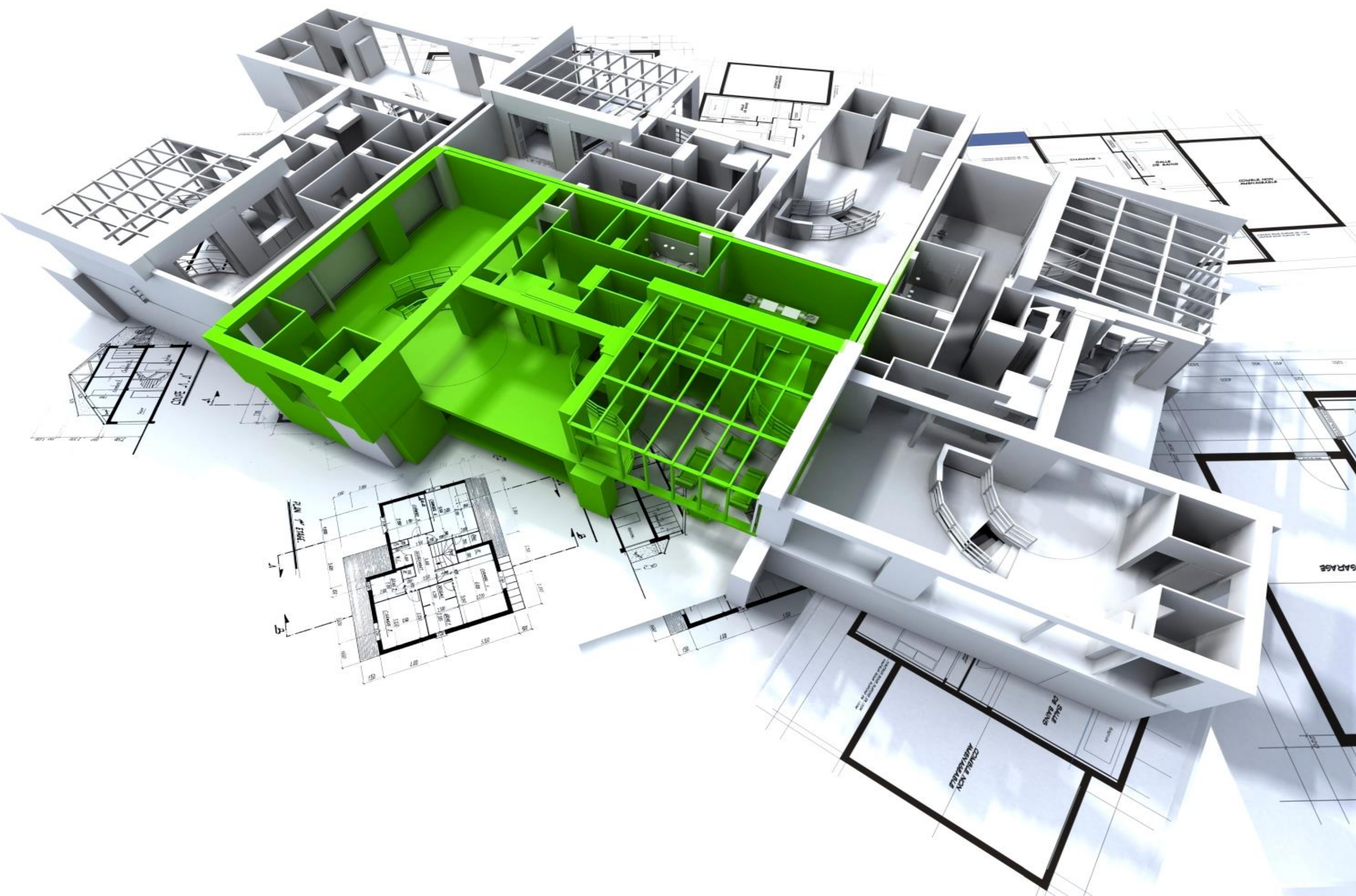


FLUIDS 3

ANALYSIS AND MODELLING METHODS

In this section we will cover the topic of dimensional analysis which is useful in showing that equations are correct, similarity which we can use to show that scale models will behave like the real thing, flow numbers which show use which type of flow is present and an introduction to computation fluid dynamics – using computers to simulate flow.

FLUIDS 3

MODELLING AND ANALYSIS

OVERVIEW

In this section you'll learn about:

- Dimensional analysis and how it may be used to check or even generate equations
- Flow numbers and types of flow
- Similarity and making physical models
- What CFD is and how it is used

ASSUMED KNOWLEDGE FOR THIS MODULE

In this subject it is assumed that you already have a knowledge of the following topics:

- *Basic fluid Mechanics – The Continuity Equation, Bernoulli's Equation and Forces in Fluids*
- *Fluid parameters – Density, Pressure and Viscosity.*
- *Fluid Statics – Hydrostaic pressure*

OBJECTIVE

To understand the different analytical tools available to us in the study of fluids.

TOPIC 1 – DIMENSIONAL ANALYSIS

i) Using DA to show an equation is consistent.

Dimensional Analysis uses units present in a formula (for example, Time labelled T, Distance labelled L and Mass labelled M) to analyze whether the formula is likely to be correct or even to find an unknown equation. As an example, we say that the dimensions of velocity (measured in ms^{-1}) are LT^{-1}.

One of the simplest things to do with Dimensional Analysis is to prove a formula is consistent and to also to shed light on its operation. We can examine this by considering Bernoulli's equation.

$$\rho gh + \rho\frac{v^2}{2} + p = const$$

Let's take the first term ρgh.

$$\rho = \frac{M}{L^3}, g = \frac{L}{T^2}, h = L$$
$$\rho gh = \frac{M}{L^3}\frac{L}{T^2}L$$
$$= \frac{M}{LT^2}$$

and the second term $\rho\frac{v^2}{2}$.

$$\rho = \frac{M}{L^3}, v = \frac{L}{T}, v^2 = \frac{L^2}{T^2}$$
$$\rho\frac{v^2}{2} = \frac{M}{L^3}\frac{L^2}{T^2}$$
$$= \frac{M}{LT^2}$$

and the third term *p*.

$$p = \frac{f}{a} and (f = ma = M \frac{L}{T^2})$$
$$so, f = M \frac{L}{T^2} and, a = L^2$$
$$p = \frac{ML}{T^2} \frac{1}{L^2}$$
$$= \frac{M}{T^2 L}$$

So, we can see clearly from this example that all three elements of Bernoulli's equation are consistent, having dimensions of $\frac{M}{T^2 L}$. Now, it's often said that Bernoulli's Equation represents an energy balance in that all the energies in the system must add up to a constant.

Pressure energy + Kinetic energy + Gravitational potential energy = Const

So, let's see if this is true. Let's consider the dimensions of energy itself (we'll take Kinetic as an example).

$$\frac{1}{2} mv^2$$
$$m = M, v^2 = \frac{L^2}{T^2}$$
$$= M \frac{L^2}{T^2}$$
$$= \frac{ML^2}{T^2}$$

So it's not true! Energy has dimensions $\frac{ML^2}{T^2}$ but Bernoulli's equation has dimensions $\frac{M}{T^2 L}$ - how can this be reconciled? Well if we divide the energy dimensions by a volume L^3 we get $\frac{ML^2}{T^2(L^3)} = \frac{M}{T^2 L}$. In other words, Bernoulli's equation is giving us not the energy, but the *energy density* of the flow. This makes perfect sense if you think about it - it's the energy at any one point on a streamline we want (not the energy of the whole flow). We can see from this example that DA is quite useful in shedding light on the operation and "meaning" of a formula.

Before we move on, let's consider one more simple example, which illustrates how we can prove a formula is consistent using DA.

Consider the formula for force in a fluid, which you encountered in second year.

$$F = \dot{m}v$$

From the previous example, we know that force is $\frac{ML}{T^2}$. We also know that $v = \frac{L}{T}$. Now $\dot{m} = \frac{dm}{dt} = \frac{M}{T}$ so.

$$F = \dot{m}v$$
$$\frac{ML}{T^2} = \frac{M}{T}\frac{L}{T}$$

We can see clearly that the right and left parts of the equation are the same units - and so we know that the equation is consistent.

TASK 1

Show that these two equations from the last section (on pipes and pumps) are consistent:

$$Q = \frac{\pi D^2}{4}v$$

$$\Delta p = -\rho g\left[\Delta z + f\frac{L}{D}\frac{v^2}{2g}\right]$$

In this equation, the friction factor (f) is a simple number (no dimensions). You can use the result for pressure from page 3 as a starting point

ii) Using DA to develop a formula

Dimensional analysis can also be used to work out formulae. This is quite a large area of interest with several methods and theorems, so we'll only just show the operation of a simple example here. Refer to your textbooks for more detail.

As an example, consider that we are trying to develop a formula to express the flow-rate of a viscous fluid in a round pipe (a useful formula in itself).

The first thing to establish is what parameters might be involved. Thinking about this, we'd expect the viscosity of the fluid (μ) to be important. Also, the radius of the pipe (r) and the pressure drop (which I'll express as the pressure drop per unit length - dp/dx). So volumetric flow rate Q is a function of all these, or:

$$Q = f\left(r, \mu, \frac{dp}{dx}\right)$$

Now, let's consider the dimensions involved in each of these parameters.

Q	r	μ	dp/dx
L^3T^{-1}	L	$ML^{-1}T^{-1}$	$ML^{-2}T^{-2}$

Now, we don't know exactly the form of the equation of Q. But it's probably of a form like this.

$$Q = \Pi r^{\alpha} \mu^{\beta} \left(\frac{dp}{dx}\right)^{\delta}$$

Where Π is a constant and α, β and δ are powers which these variables are raised to. Of course it's also possible that there could be sum terms as well - but this example has been chosen to work it out without them - refer to the textbooks if you'd like to know how to handle these.

What we do now is rewrite this in terms of Π.

$$\Pi = r^{\alpha} \mu^{\beta} \left(\frac{dp}{dx}\right)^{\delta} Q$$

Actually, I've cheated slightly here because the signs of α, β and δ are going be different in this version as I've transposed the formula. Now, we can write this in terms of the dimensions of each parameter.

$$\Pi = (L)^{\alpha}(ML^{-1}T^{-1})^{\beta}(ML^{-2}T^{2})^{\delta}(L^{3}T^{-1})$$

As a check you can see if this multiplies out to $M^0L^0T^0$ as it has to since Π is a dimensionless constant.

Now let's take each dimension in turn and write down the powers to which it's raised (their all equal to zero because they multiple out to the power of zero).

Mass (M): $\beta + \delta = 0$

Length (L): $\alpha - \beta - 2\delta + 3 = 0$

Time (T): $-\beta - 2\delta - 1 = 0$

These three equations can be solved simultaneously and we get $\alpha = -4$, $\beta = 1$ and $\delta = -1$. So:

$$\Pi = r^{-4}\mu\left(\frac{dp}{dx}\right)^{-1}Q$$

$$\Rightarrow \Pi = \frac{Q\mu}{r^4\left(\frac{dp}{dx}\right)}$$

This is really as far as we can go. We need to conduct some experiments or use some other theory to determine the constant Π. However, in reality it turns out to be $\frac{\pi}{8}$ and so the formula is often written like this:

$$Q = \frac{\pi r^4 \Delta p}{8\mu L}$$

Where *L* is the length of pipe and *Δp* is the pressure drop in the pipe.

TASK 2

Use the technique explained above to derive the equation shown in task one (not including the exact constants, of course) for volumetric flow rate in a frictionless pipe. You can assume the Q = f(D, v)

TOPIC 2 – SIMILARITY AND FLOW NUMBERS

Similarity is often included in book chapters with Dimensional Analysis, but in reality it's only distantly related to the above and it's been included here, along with DA, because it's convenient. It has to do with the number describing types of flow and their effect on modelling in fluids.

There are a number of parameters, related to flow which tell us about the type of flow present and its physical attributes. We've come across some of these before, others are new. When we a solving a problem involving fluid mechanics we often start by calculating these numbers to tell us which flow regime we are in and therefore which equations to use.

- Reynold's number

$$\mathrm{Re} = \frac{\rho V l}{\mu}$$

We have already met Reynold's number several times. It tells us the ratio between inertial and viscous forces in the fluid. This in turn tell us when to expect the onset on turbulence. Re < 2000 - Laminar flow, Re > 2000 turbulent flow.

- Mach number

$$M = \frac{V}{c}$$

This tells us whether the flow is faster or slower than sound (and so if we can expect shockwaves to dominate the flow). In the formula above, c is the speed of sound. The formula actually indicates the square root of the ratio of the inertial force to the compression force in the fluid. M < 1 subsonic flow, M > 1 supersonic flow.

- Froude number

$$Fe = \frac{V}{\sqrt{gl}}$$

Here's one we haven't come across before. It's the square root of the ratio of the inertia force to the gravity force. It tells us whether a disturbance in the flow (for example a wave) travels faster than the flow itself (and so can propagate upstream) - this is called *Subcritical Flow*. The alternative is that the flow is going faster than the wave (in which case, the wave gets swept downstream) - this is *Supercritical Flow*. Fe < 1 is Subcritical, Fe > 1 is Supercritical. You can see from this that Froude number is similar in some ways to Mach number.

- Weber Number

$$We = \frac{\rho V^2 l}{\sigma}$$

This is the ratio between inertial force and surface tension. It tells us when surface tension effects become important in a fluid (if We < 1). Surface tension is important in flow through thin tubes (capillary flow) and of very shallow flows and those of small droplets and bubbles. The topic of fluids in which surface tension is important is called "microfluidics."

These are some of the most important numbers representing flows – but there are several others sometimes used, which you can look up in your textbooks. These include: The *Strouhal number* which tells us when flows cause structures to vibrate or self-oscillate - important in the design of bridges, tall chimneys, etc). The *Euler Number* tells us about the pressure force in relation to inertial force and is important in the design of aerofoils and in cavitation calculations. *Drag Coefficient* and *Lift Coefficient* tell us about the ratio of drag and lift to inertial force. Finally *Friction Factor* we've already met and is the ratio of pressure drop to inertial force $\frac{\Delta p}{0.5\rho V^2}$ As an exercise have a look at the list of flow description numbers in the Wikipedia entry "Dimensionless numbers in fluid mechanics".

TOPIC 3 – SIMILARITY AND SCALE MODELS

The numbers described in the section above tell us about the nature of various flows. They also play an important part in other calculations – particularly those which involve making and using scale models. If we are designing a new aircraft or a new wind turbine, we can't build a full-scale model to test it - that would be expensive and dangerous. We'd like to build scale models and test them instead. But how do we know that the models are going to behave in the same way as the real full-sized thing. The answer is that they generally don't.

You can see this if you compare a small model of an aircraft wing to the full sized thing. The small model will have a low Reynold's number $\frac{\rho V l}{\mu}$ because *l* is small (the breadth of the wing) and probably so is *V*. If the Reynold's number is small, then the airflow over the model wing will be laminar. However, that over the full sized wing probably won't be. This means that the model doesn't behave like a cut-down version of the big wing - it behaves quite differently.

The way to overcome this problem and to make the model behave like the full sized version is to arrange for the numbers above to be equal in both cases. Actually, in many cases, not all the numbers are important and often it's the Reynold's number that has the major effect (particularly in incompressible gas flow). However, you must be cautious when considering this and think carefully about the flow regime – for example, in small-scale flows the Weber Number may be the most important.

In our example above, we could make the Reynold's number the same by increasing the density, the speed or both. A decrease in l by a factor of five, for example, could be compensated for by an increase in speed or density by a similar factor of five.

This why high speed, pressurized wind tunnels are sometimes used. Another solution which is sometimes adopted is to test in water (which has a higher viscosity) or another gas.

TASK 3

The mid-wing chord-length of a Boeing 787 "dreamliner" aircraft is about 6 meters (if you don't know what this means, look up "wing chord-length"). The operating speed is 900km/h at an altitude of 10.7km. We wish to build a 1/6th model of this part of the wing for wind-tunnel testing.

Discuss which of the flow-numbers mentioned in topic 2 are important in this application.

Suggest how a tunnel could be set up for this testing – in particular discuss the operating fluids, pressure and velocity of tunnel.

TOPIC 4 – COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics or CFD is the most important computational method in our modern fluids toolkit. It involves using a computer to solve the complex fundamental equations of fluid mechanics.

These equations are generally referred to as the *Navier-Stokes Equations* (in their full form) or the *Euler Equations* (in their inviscid form). The names are somewhat informal, as other equations are also required to provide the full solution – in particular the full set requires equations which describe the conservation of mass, momentum and energy in the fluid.

We have only been able to approximate these equations since computers became widely available (they are complex and in most cases they cannot be solved explicitly) - so CFD has only recently become very widely used. The full set of equations is very complex and so for simpler flows – like inviscid or incompressible flows, a more basic set is used and often a 2D (or even a 1D) simulation is performed, rather than the full 3-dimensional treatment.

The idea works like this: All the different conservation equations which we have written down for solving fluid problems can also be written in a differential (or integral) form. For example, take conservation of mass (the continuity equation, $\rho vA = constant$):

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \hat{v}) = 0$$

or, to expand the *grad* symbol out:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial y}(\rho v_y) + \frac{\partial}{\partial z}(\rho v_z) = 0$$

This looks horrendously complex, but actually it's just saying exactly the same thing as the simple version (that the mass in a confined volume is contant):

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \hat{v}) = 0$$

A change in density + a change in mass (due to stuff flowing in or out) = 0

It can also be written as an integral equation.

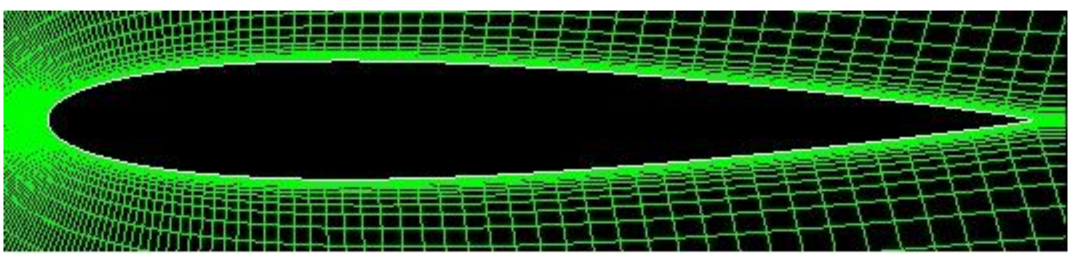
$$\frac{\partial}{\partial t}\oiiint_V \rho dV + \oiint_S \rho\hat{v} \bullet d\hat{s} = 0$$

Change of density + Mass flowing through = 0

But why write them is such a complex way? Well it turns out that, although this makes them very difficult for a human to solve, they are easy for a computer to solve. The computer code replaces all the derivatives in the equations by “difference equations”, which approximate them and, which we can add actual values into – for example:

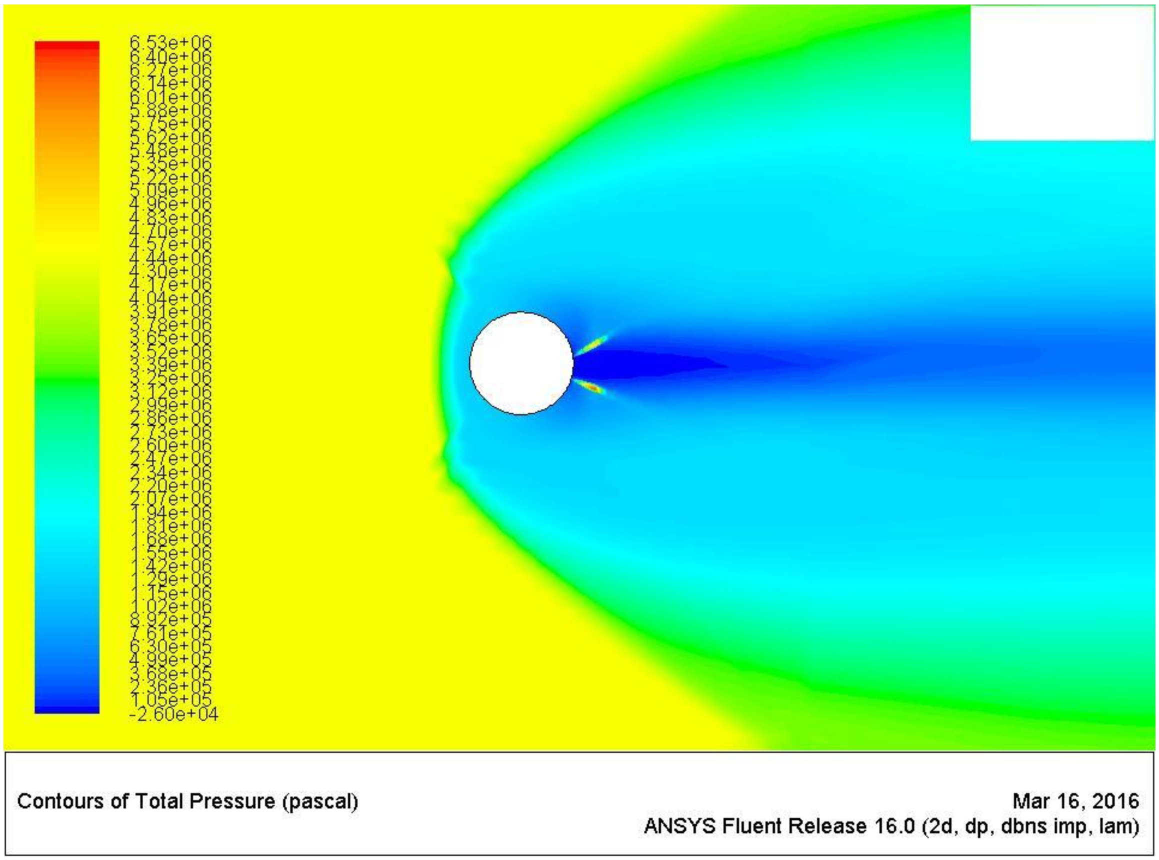
$$\frac{d\theta}{dt} \approx \frac{\Delta\theta}{\Delta t} = \frac{\theta_{T+1} - \theta_T}{\Delta t}$$

This replaces the derivative on the left by an algebraic equation on the right. Doing this is called *discretizing* the equation. If we now set up initial values for the parameters at a particular time (θ_T) then we can use this approximation to calculate *an approximate value* for the parameter at some short time (Δt) later (θ_{T+1}). A computer can do this for thousands or millions of closely spaced points and so accurately approximate the real function. It typically does this by setting up a mesh of points through the space to be simulated, in the form of a grid as shown below (around a wing):



The placing and form of this mesh or grid is absolutely critical to the success of the simulation and whether the results are accurate. The grid needs to be fine around places where the flow is complex and course around where it is not (making it fine everywhere doesn’t work, as the simulation takes too long). If the grid is not correctly used, not only will the flow around the complex parts be incorrect, but because these results are used in further down-stream calculations, these will also be wrong.

This enables all the important parameters to be calculated (typically pressures, density, temperature and velocity) at each point and produce a “map” of them as shown (pressure around a sphere in this case):



CFD results must be used with great care and “normal” paper calculations should be used to check that they are giving expected results. Things which can go wrong include:

- Miss-structuring the grid or mesh as explained above.
- Because the simulation is an approximation, the accuracy degenerates and diverges from reality as further calculations are performed.
- The simulation is only as good as the input data.
- Because it is an iterative process, sometimes the simulation becomes unstable (like instability in a control system) and the values become wildly inaccurate.
- Turbulence is a chaotic phenomenon and inherently unpredictable – so simulation of it is at best a “good guess”.

The upshot of this is that you must treat CFD results with suspicion and remember their limitations.

Three main types of simulation are used commonly used in CFD (shown overleaf).

- Finite difference method – uses the differential equations as a starting point and calculates the values at points on the grid.
- Finite volume method – uses integral equations and calculates the values in small volumes of space. This is now the most common type.
- Finite elements – uses the FEA techniques discussed in solid mechanics.

SUMMARY

- Dimensional analysis can be used to check that equations are consistent and also to provide information about their structure.
- DA may also be used to find an approximate formula for a parameter (but lacks the ability to find the constants involved).
- Flow numbers allow us to identify the flow regime a system is operating in and also which flow parameters are important within that regime.
- When building scale models for wind-tunnel or water testing we much identify which flow numbers are important to our system and take step to ensure that the model results are valid through similarity.
- CFD is one of the most important modern tools for fluid mechanics.
- Results from CFD models must be treated with care as they may be misleading.