Chapter 9

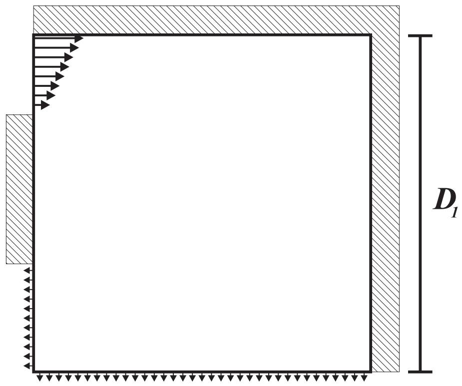
Dimensional analysis and scaling

- The Philosopher's approach
- The Mathematicians's approach
- The Engineer's approach
- Example - an orifice plate
- Example - an aeroplane
- Example - the drag force on a ship
- Further worked examples

9.1 The philosopher's approach

The French adopted *Systéme Internationale* units (metres, seconds etc.) soon after the revolution in 1789. Most of mankind has followed suit but Nature, and some Americans, still have no idea what a metre is. We have become so used to SI units that we sometimes forget that, when we express things dimensionally, we add man-made notions of length, time etc. It is much more natural to take them out. This is the basis of *dimensional analysis* and the Buckingham Pi law:

For example, if we want to work out the velocity field inside a box of a given shape, we would need to know its size, the entry velocities, and the fluid's density and viscosity:



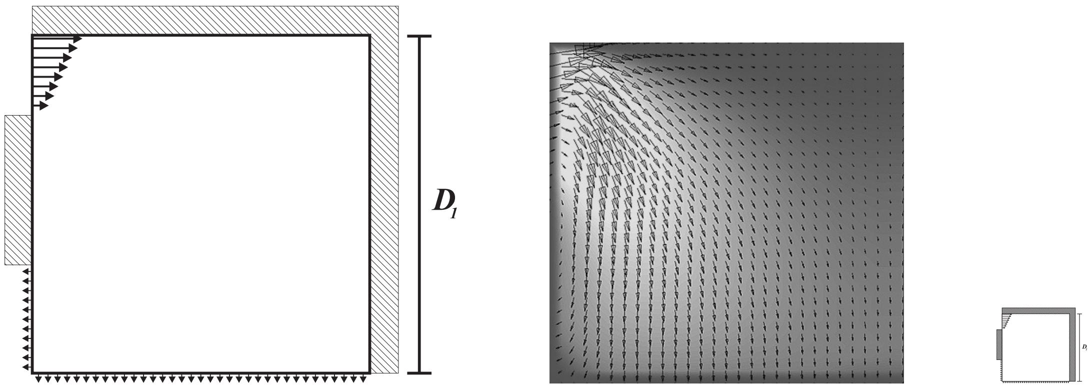
There are four independent variables. Hidden within these four variables, however, there are three man-made concepts: metres, seconds, and kilograms. The philosopher would say that, to be true to Nature, we have to remove these man-made concepts:

The control parameter is the Reynolds number. It is *dimensionless* so we are safe from French intervention. If the president of France decides to change the definition of a metre, our control parameter remains unchanged because $\rho V D$ and μ both change by exactly the same amount.

If the flow is compressible there is another independent variable: the speed of sound. There are then *two* control parameters: the Reynolds number and the Mach number.

9.2 The mathematician's approach

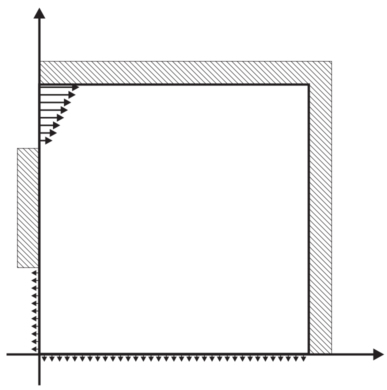
The Navier-Stokes equation is $\mathbf{f} = m\mathbf{a}$ written for a viscous fluid. It is a partial differential equation that must be satisfied at every point in the fluid. Given certain boundary conditions and the physical properties of the fluid, the equation has a unique solution[*1]:



If we have two situations that are geometrically similar (i.e. one is a scaled-up version of the other) we can define equivalent reference lengths and reference velocities in both situations and measure all distances in these units. In these new units, the boundary conditions are identical but the physical properties, ρ and μ, are not.

$$\begin{aligned} x_1 &= x^* D_1 \\ y_1 &= y^* D_1 \\ \mathbf{v_1} &= \mathbf{v}^* V_1 \\ t_1 &= t^* \frac{D_1}{V_1} \\ p_1 &= p^* \frac{1}{2}\rho_1 V_1^2 \end{aligned}$$

physical props: $\rho_1,\ \mu_1$



$$\begin{aligned} x_2 &= x^* D_2 \\ y_2 &= y^* D_2 \\ \mathbf{v_2} &= \mathbf{v}^* V_2 \\ t_2 &= t^* \frac{D_2}{V_2} \\ p_2 &= p^* \frac{1}{2}\rho_2 V_2^2 \end{aligned}$$

physical props: $\rho_2,\ \mu_2$

Note also that:

$$\nabla_1 = \begin{pmatrix} \frac{\partial}{\partial x_1} \\ \frac{\partial}{\partial y_1} \end{pmatrix} = \begin{pmatrix} \frac{\partial}{\partial (x^* D_1)} \\ \frac{\partial}{\partial (y^* D_1)} \end{pmatrix} = \frac{1}{D_1}\begin{pmatrix} \frac{\partial}{\partial x^*} \\ \frac{\partial}{\partial y^*} \end{pmatrix} = \frac{\nabla^*}{D_1} \qquad \text{and} \qquad \nabla_2 = \frac{\nabla^*}{D_2}$$

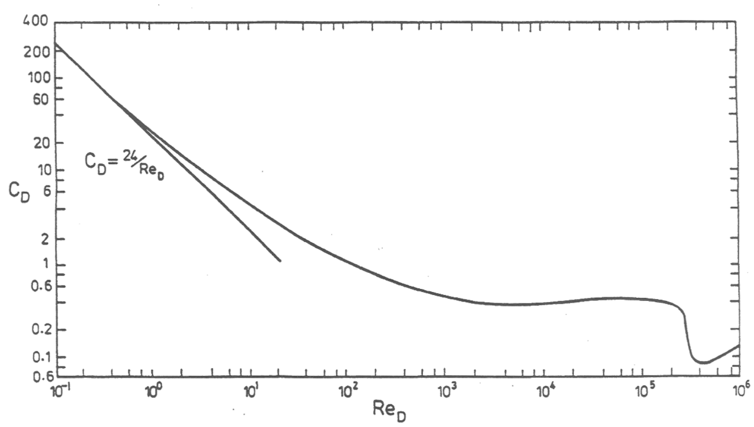
We want to know the condition for which the two situations have the same solution to the Navier-Stokes equation.

[1]*Actually, nobody has yet proved that the solution is *unique and smooth*, but we expect it to be. There is a $1m prize for the first person to do so: http://www.claymath.org/millennium/

We write the *dimensional* Navier-Stokes equations for both situations side by side. Then we substitute in x^*, $\mathbf{v}^*$ etc. from the previous page and then re-arrange to get the *non-dimensional* Navier-Stokes equations for both situations:

Big square	Little square
$\rho_1 \frac{D\mathbf{v_1}}{Dt} = -\nabla_1 p_1 + \mu_1 (\nabla_1)^2 \mathbf{v_1}$	$\rho_2 \frac{D\mathbf{v_2}}{Dt} = -\nabla_2 p_2 + \mu_2 (\nabla_2)^2 \mathbf{v_2}$
$\rho_1 \frac{D(\mathbf{v}^* V_1)}{D(t^* D_1/V_1)} = -\frac{\nabla^*}{D_1}\left(p^* \frac{1}{2}\rho_1 V_1^2\right) + \mu_1 \frac{(\nabla^*)^2}{D_1^2}(\mathbf{v}^* V_1)$	$\rho_2 \frac{D(\mathbf{v}^* V_2)}{D(t^* D_2/V_2)} = -\frac{\nabla^*}{D_2}\left(p^* \frac{1}{2}\rho_2 V_2^2\right) + \mu_2 \frac{(\nabla^*)^2}{D_2^2}(\mathbf{v}^* V_2)$
$\Rightarrow \quad \frac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \frac{\mu_1}{\rho_1 V_1 D_1}(\nabla^*)^2 \mathbf{v}^*$	$\Rightarrow \quad \frac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \frac{\mu_2}{\rho_2 V_2 D_2}(\nabla^*)^2 \mathbf{v}^*$
$\frac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \frac{1}{Re_1}(\nabla^*)^2 \mathbf{v}^*$	$\frac{D\mathbf{v}^*}{Dt^*} = -\nabla^* p^* + \frac{1}{Re_2}(\nabla^*)^2 \mathbf{v}^*$

We already know that the boundary conditions are the same. Therefore, if the Reynolds numbers are also the same, *the equations must have exactly the same solution*. In other words, the Reynolds number is the *only* control parameter for geometrically-similar objects in incompressible flows (in compressible flows, the non-dimensional Navier-Stokes equation also contains the Mach number).



Drag coefficient of spheres as a function of Reynolds number.

This is why we can plot C_D for a sphere as a function of the Reynolds number alone. Each point on the line corresponds to a solution of the Navier-Stokes equation at a particular Reynolds number. It will be valid for *all* perfectly smooth spheres.

9.3 The engineer's approach

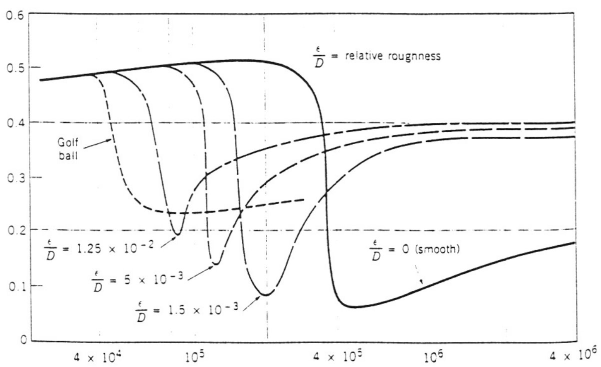
The engineer's approach is firmly rooted in the philosopher's and mathematician's approach. We will consider the example of a rough sphere, which introduces a new lengthscale: the height of the bumps, ϵ.

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

Step 2 - Count up the number of dimensions (mass, length, time etc.) and subtract this from the number of variables to obtain the number of dimensionless numbers in the problem. This is the philosopher's approach: it must be possible to describe the problem without any man-made constructs.

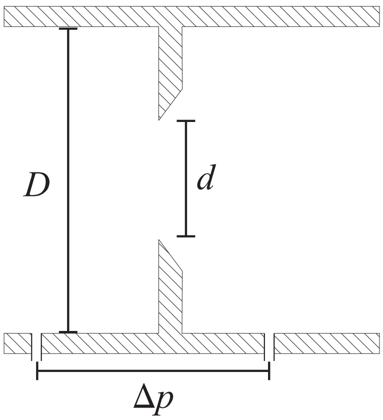
Step 3 - Create the dimensionless numbers. There are often several ways to do this but it is best to use standard dimensionless numbers, such as the Reynolds number, Mach number etc. These can be found in the Thermofluids data book. This is the mathematician's approach: the standard dimensionless numbers appear as control parameters if you work out the dimensionless Navier-Stokes equations.

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



9.4 Example - an orifice plate

In chapter 6 we worked out the pressure drop across an orifice plate, Δp, in terms of the average velocity upstream, V, using a simple model of the flow. However, the real flow is more complicated than that assumed by the simple model. We will need to do experiments (or a numerical simulation) to obtain a more accurate evaluation of Δp as a function of V. How do we express these in a way that is easily scalable to geometrically-similar orifice plates?

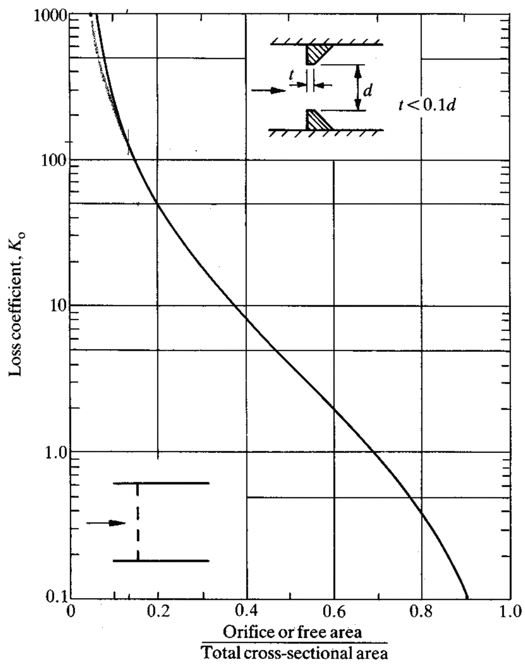


Step 1	Dependent variable	Independent variables

Step 2 - count up the dimensions:

Step 3 - create the dimensionless numbers:

Step 4 - carry out the experiment:



9.5 Example - an aeroplane

We want to evaluate the lift and drag coefficients of a Boeing 747 by testing a geometrically-similar model in a wind tunnel. What conditions are required in the wind tunnel for complete similarity? Remember that an aeroplane travels near Mach 1, so the density of the fluid cannot be taken as a constant.

Step 1

Dependent	Independent

Step 2: count up the dimensions

Step 3: create the dimensionless numbers

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

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

re-arranging gives:

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

This is an over-constrained problem. It will not be possible to match both M and Re without pressurising the wind tunnel to change the density. This is expensive. However, we know from experience that the Reynolds number has little influence once it is above around 10^6. Therefore we match the Mach number and let the Reynolds number float, making sure that it does not drop into the region where it could have influence.

9.6 Example - the drag force on a ship

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

Step 1 - Dependent and independent variables

Step 2 - Number of dimensions

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

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

$$\frac{U_m^2}{gD_m} = \frac{U_f^2}{gD_f} \qquad \text{and} \qquad \frac{\rho_m U_m D_m}{\mu_m} = \frac{\rho_f U_f D_f}{\mu_f} \qquad \Rightarrow \frac{\mu_m \rho_f}{\mu_f \rho_m} = \left(\frac{D_m}{D_f}\right)^{3/2}$$

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

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

We know that the sea takes around a day to become calm after a storm (through the action of viscous forces) and that a wave period is a few seconds. Therefore viscosity can only have a very weak effect on wave motion. Furthermore, gravity can have little effect on the normal fluid-mechanical wake associated with the body. Therefore we treat the wave terms and the fluid-mechanical terms as independent and additive:

So we perform two separate experiments. First we test the model around the correct Froude numbers and measure $C_{D total}$. Then we measure the fluid-mechanical drag at those Froude numbers by testing a completely submerged reflected model:

We subtract one from the other to obtain the Froude number dependence: $C_{D wave}(Fr)$. For large ships it can be hard to test at the correct Reynolds number because very large velocities are required. However, we can use our knowledge from chapter 8 (external flow) to estimate $C_{D fluid-mech}(Re)$ at large Re.