

Lecture 18: Experimentation and Dimensional Analysis

The largely theoretical approach taken so far is limited in that real fluid behaviour frequently violates most if not all of the simplifying assumptions needed for a tractable analytic solution.

An *ideal fluid*, defined as inviscid, irrotational and incompressible, can accurately reproduce reversible processes such as sink and source flows, but breaks down in the representation of irreversible and asymmetric processes such as friction losses and turbulence. Non-ideal processes are far more common in the natural and built environment and present substantial practical problems.

The difficulty is evident in the previous section's development of Bernoulli's equation, in which we suggested a range of loss factors in general terms but were unable to specify their values. It is not in fact possible to derive these loss factors theoretically.

Real fluids differ from ideal fluids mainly due to the effects of viscosity and compressibility, which produce a range of dynamic effects including drag, turbulence and heating or cooling. Gases have a wider range of thermodynamic pathways that exchange energy between mechanical variables (velocity, altitude) and state variables (temperature, pressure, density) with associated dynamic effects.

The modern understanding of the full range of fluid behaviour, both reversible and irreversible, is built on a collaboration of experiment and theory over many centuries. This process has generally succeeded because even when no theory is available to "explain" observation, valid data can be accepted and modelled numerically or by curve fitting. The resulting parameters, equations, charts etc are applied even when we don't fully "understand" why they work.

Often experimental results are later (sometimes many years later) derived theoretically, with the experimentally derived model as a guide. Experiment supports theory and vice versa.

This section examines the linked experimental-theoretical development of fluid dynamics, leading to important applications in pipe and channel flow, compressible flow, streamlining, aircraft design, propulsion, and other general areas.

Dimensional Analysis

In addition to the usual tenets of scientific method, the experimental-theoretical process relies on two ideas relating to the *scale* and *dimensions* of fluid phenomena:

- We can *scale* from scale model tests to full-scale prototype tests, to estimate forces and velocities that can be expected in the real world on the basis of small-scale trials.
- We can *dimension* fluid phenomena in ways that can then be analysed and calibrated theoretically, as a guide to the theoretical form of unknown equations.

Dimensioning is a complex theoretical and mathematical tool that is not covered in this course. If you are interested, Granger (Reading 9) provides an introduction.

Scaling however is essential to an understanding of fluid behaviour across the full range of scales. In many cases it is not possible to theoretically predict a force or velocity other than by scaling from a laboratory model test. This requires an understanding of how small and large scales are dynamically related.

In particular we are interested in quantities that remain constant from the small scale to the large. These quantities are known as *dimensionless numbers*, for two reasons:

- Dimensionless numbers are constant in similar dynamic situations across a wide range of scales, hence they are unrelated to the dimensions (size) of the situation.
- The units of dimensionless numbers all cancel out, since they are based on ratios of force or velocity at different model and prototype scales. A number with the dimensions of $[\text{force}_m/\text{force}_p] = [1]$ is dimensionless. Similarly $[\text{velocity}_m/\text{velocity}_p] = [1]$ is dimensionless.

The subscripts m and p refer to model and prototype respectively. We'll stick to this convention.

Knowing the value of relevant dimensionless numbers from model tests, it is possible to predict the performance of a full-scale prototype – its velocity, power, pressure loss, efficiency, etc.

Dimensional analysis reveals that similar systems across a range of scales have certain things in common, with important differences. This can lead to useful insights as to how they work.



Scale models of landscapes, infrastructure, machinery and vessels are an essential part of the formulation of consistent theory and its application to prototype design.

Dimensionless Numbers

The most useful dimensionless numbers are ratios of forces such as inertia and viscous shear. These numbers are dimensionless because in the ratio of F:F all units cancel out: [m.a]/[m.a] = [1]

Force ratios provide an indication of the relative importance of different physical effects in the behaviour of a system, and often indicate ranges of different types of behaviour:

Reynolds #	$Re = \frac{\rho DV}{\mu}$	Ratio of inertia to viscous shear (laminar/turbulent flows)
Froude #	$Fr = \frac{V}{\sqrt{gy}}$	Ratio of inertia to weight (surface waves and critical flow)
Euler #	$Eu = \frac{P}{\rho V^2}$	Ratio of pressure to inertia (induced pressure effects)
Mach #	$M = \frac{V}{c}$	.. inertia to pressure (submerged waves and compression)
Weber #	$W = \frac{\sigma}{\rho V^2 L}$	.. surface tension to inertia (capillarity effects)

Exercise: Confirm that the units of these numbers cancel out. Check at least the first three.

eg
$$Re = \frac{\rho VD}{\mu} = \left[\frac{kg}{m^3} \right] \left[\frac{m}{s} \right] \left[\frac{m}{-} \right] \left[\frac{Pa.s}{-} \right]^{-1} = \left[\frac{kg}{m^3} \right] \left[\frac{m}{s} \right] \left[\frac{m}{-} \right] \left[\frac{kg}{s^2} \frac{m}{m^2} \frac{1}{-} \frac{s}{-} \right]^{-1} = [1]$$

Reynolds Number: Pipe and Submerged Flow

The Reynolds number applies to submerged flows, that is flow in closed pipes or at some depth from the surface in an open channel or reservoir. The Reynolds number distinguishes situations where viscous forces predominate (small Re) from situations where momentum forces dominate (large Re).

To model the performance of a submerged system such as inside a flooded pipe or on the hull of a submarine, we require the Reynolds numbers for a model and the prototype to be the same at comparable states of flow, such as where drag characteristics on model and prototype are the same:

$$Re = \frac{\rho DV}{\mu} \Bigg]_m = \frac{\rho DV}{\mu} \Bigg]_p$$

ie the Reynolds number for the model and prototype are the same at a kinematically similar velocity.

D is any relevant length, such as the diameter of the pipe or submarine hull, to be used for length scaling. By measuring model variables (subscript m) we can calculate those for the prototype (subscript p) for a similar dynamic situation. For example a laboratory wind tunnel may be calibrated to an equivalent full-scale gale. This is often expressed as a unit *Reynolds ratio* = 1:

$$Re_r = \frac{Re_m}{Re_p} = \frac{\rho_r D_r V_r}{\mu_r} = 1$$

Here all the relevant variables in the formula are replaced by their ratios from m to p (always in that order: eg $D_r = D_m/D_p$, the ratio of model diameter to prototype diameter). This process gives a dimensionless number of unit value and provides a conversion for unknown variables of interest:

eg $V_r = \frac{\mu_r}{\rho_r D_r}$ Reynolds Velocity Conversion (V_r is the ratio of velocities)

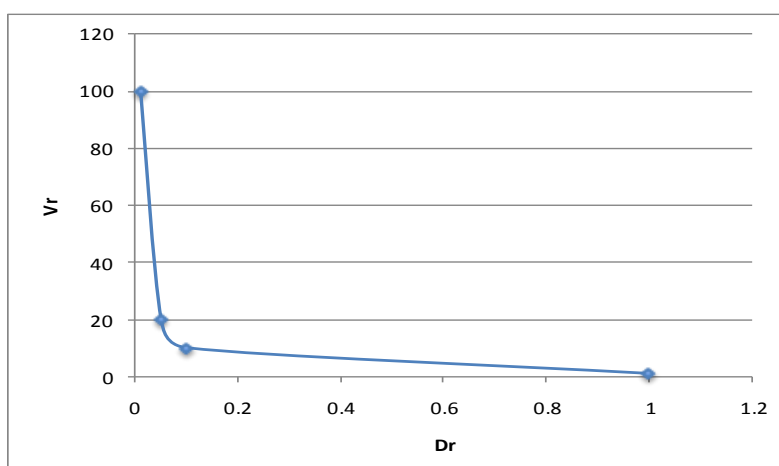
The fluid is often the same in lab and real settings, hence $\nu_r = \rho_r = 1$:

so $V_r = \frac{1}{D_r}$ kinematically similar velocity (same fluid) is inverse to length scale

To calculate the prototype velocity V_p , convert ratios back to model and prototype:

$$V_r = \frac{V_m}{V_p} = \frac{1}{D_r} = \frac{D_p}{D_m} \quad \text{so} \quad V_p = V_m \frac{D_m}{D_p} \quad \text{Reynolds Velocity Conversion (same fluid)}$$

Submerged models at high speeds are kinematically similar to prototypes at low speeds. For example a narrow pipe flows faster than a large-diameter pipe at the same level of turbulence.



The Reynolds number is often stated with the kinematic viscosity $\nu = \mu/\rho$:

$$Re = \frac{DV}{\nu} \quad \text{Kinematic Form of Reynolds Number}$$

Example: A laboratory valve fitting of diameter $D = 0.1$ m causes a 0.3 m head loss at a flow velocity of $V = 1.3$ m/s. At what velocity can we expect a comparable head loss in our industrial plant, where the valve diameter is 0.5 m? Use $\nu = 1 \times 10^{-6}$

We expect the Reynolds number to be the same in the prototype as in the model:

Model: $Re = VD/\nu = 1.3(0.1)/1 \times 10^{-6} = 130 \times 10^3$

Prototype: $Re = 130 \times 10^3 = V_p(0.5)/1 \times 10^{-6}$:

Hence the *kinematically similar* prototype velocity $V_p = 130 \times 10^3(1 \times 10^{-6})/0.5 = \underline{0.26 \text{ m/s}}$.

Use the V_p formula as a check:

$$V_p = V_m D_m / D_p = 1.3(0.1)/0.5 = \underline{0.26 \text{ m/s}}$$

The comparable head loss in the full scale industrial plant is scaled by the length scale, since head has the units of length and scales accordingly: the expected headloss at that speed is $0.3 \times 0.5/0.1 = 1.5$ m.

Another way of stating this question might have been “Our 0.5 m industrial valve loses too much head at this speed. What speed do we need in the lab model to study the problem?” In such cases you simply invert the subscripts to solve V_m or other variables of interest.

Example: A 20-m silk parachute design is proposed. Laboratory tests will use a 1-m parafilm (a high tech nanofibre) model in water, $\nu = 1 \times 10^{-6} \text{ m}^2/\text{s}$. The kinematic viscosity of air at 10°C is $8 \times 10^{-6} \text{ m}^2/\text{s}$. What vertical flow speed do we need in the lab to simulate a safe real-world fall speed of 6 m/s?

We expect the Reynolds numbers to be the same. As a ratio:

$$Re_r = \frac{D_r V_r}{\nu_r} = 1 \quad : \quad V_r = \frac{\nu_r}{D_r}$$

The length ratio is $D_m:D_p = 1:20 = 0.05$

The kinematic velocity ratio is $\nu_m:\nu_p = 1 \times 10^{-6}/8 \times 10^{-6} = 0.125$

So the velocity ratio $V_r = \nu_r / D_r = 0.125/0.05 = 2.5 = V_m/V_p$

$$\text{So } V_m = 2.5 V_p = 2.5 \times 6 = \underline{15 \text{ m/s}}$$

Scaling problems will tie you in knots with the correct direction of ratios. It pays to write it all out in full, down to all the numbers on the RHS, *before* reaching for your calculator.

Reynolds numbers are used to solve a wide range of common pipe and submersible problems, including

- Head loss in pipes, fittings and surfaces
- Dynamic forces such as wind loading
- Lift and drag coefficients on aircraft
- Submarine testing and design
- Any subsonic external flow, from parachute design to conventional jet propulsion.

Froude Number: Surface Flow

The Froude number applies to gravitationally-bound surface flows, such as rivers, lakes and oceans, and distinguishes *supercritical flow*, where flow velocity exceeds surface wave speed ($V > C$; $Fr > 1$), from *subcritical flow* where velocity is less than surface wave speed ($V < C$; $Fr < 1$).

To model supercritical (fast, shallow) streamflow ($Fr > 1$) or subcritical (slow, deep) streamflow ($Fr < 1$) we require the Froude numbers for model and prototype to be the same at comparable states of flow:

$$Fr = \frac{V}{\sqrt{gy}} \Bigg|_m = \frac{V}{\sqrt{gy}} \Bigg|_p$$

y is a characteristic length, often a flow depth or hull length, for comparison of model to prototype length scales. Again, this equivalence is often expressed as a unit *Froude ratio*:

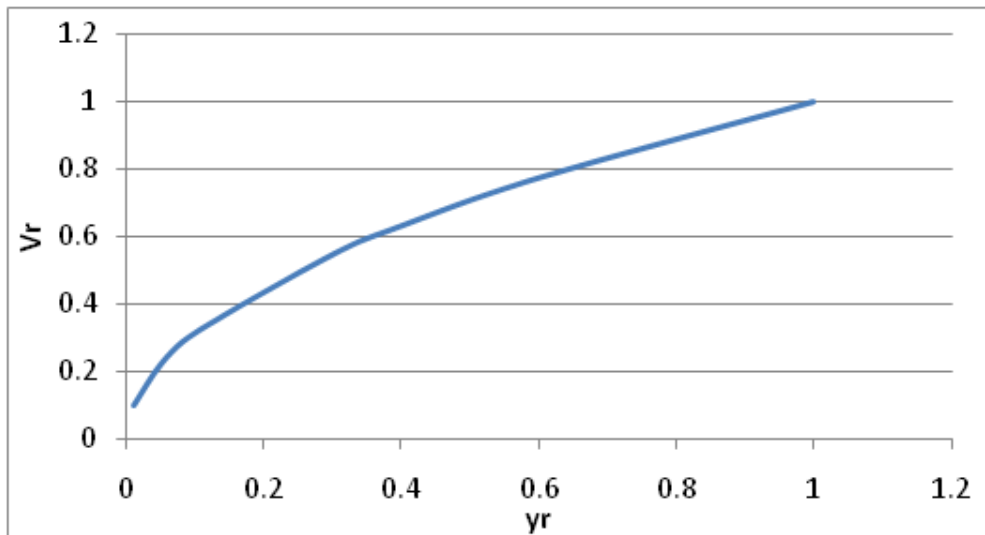
$$Fr_r = \frac{V}{\sqrt{gy}} \Bigg|_{m/p} = \frac{V}{\sqrt{gy}} \Bigg|_r = \frac{V_r}{\sqrt{g_r y_r}} = 1$$

We can use this to calculate unknown prototype variables from model measurements or vice versa. Since model and prototype tests are usually in the same gravitational field, $g_r = 1$:

eg $V_r = \sqrt{y_r}$

or $\frac{V_m}{V_p} = \sqrt{\frac{y_m}{y_p}} : V_p = V_m \sqrt{\frac{y_p}{y_m}}$ Froude Velocity Conversion

Surface models at low speeds are kinematically similar to prototypes at high speeds, ie a large wave travels faster than a small one, and waves in deep water are faster than those in shallow. Similarly a large ship travels faster than a small one for kinematically similar propulsion forces.



The Froude number is used to solve problems involving a free surface:

- Open channel hydraulics
- Ship and hydrofoil hull design and wave drag
- Waves and wave shear phenomena
- Pipes flowing partly full

Example: A 2-m model ship moves at 0.7 m/s under a given towing force. Assuming a kinematically similar full-scale propulsion force, at what velocity will the 120-m prototype travel?

We expect the Froude numbers to be the same for the two situations:

$$Fr_m = V_m / \sqrt{y_m} = 0.7 / \sqrt{2} = 0.495$$

$$Fr_p = 0.495 = V_p / \sqrt{120} = V_p / 10.95$$

$$\text{So } V_p = 0.495 \times 10.95 = \underline{5.4 \text{ m/s}}$$

Example: a model spillway at 1:100 scale discharges 2 m³/s at a head of 0.04 m. What will be the full-scale discharge at a kinematically similar head?

The velocity ratio is as per Froude number, $\sqrt{y_r} = \sqrt{1:100} = 0.1$. We are not told the model velocity, but the discharge.

We can work out a *discharge ratio* as $Q_r = A_r V_r = y_r^2 \sqrt{y_r} = y_r^{5/2}$

Here the area ratio A_r is based on the square of the length ratio (just as areas are squares of lengths), and the velocity ratio is taken from the Froude number $V_r = \sqrt{y_r}$

$$\text{So } Q_r = Q_m / Q_p = (1:100)^{5/2} = 10 \times 10^{-6}$$

$$\text{And } Q_p = Q_m / 10 \times 10^{-6} = 2 / 10 \times 10^{-6} = \underline{200 \times 10^3 \text{ m}^3/\text{s}}$$

The kinematically similar head is just scaled by the length ratio, ie the prototype dam is at a head of $0.04 \times 100 = 4 \text{ m}$ (a big dam).

Reading

Granger 1985 (Reading 9) Section 7.1 and 7.2, 7.5.3 and 7.5.4