3 Hydrodynamics: when water starts to flow

3.1 Introduction

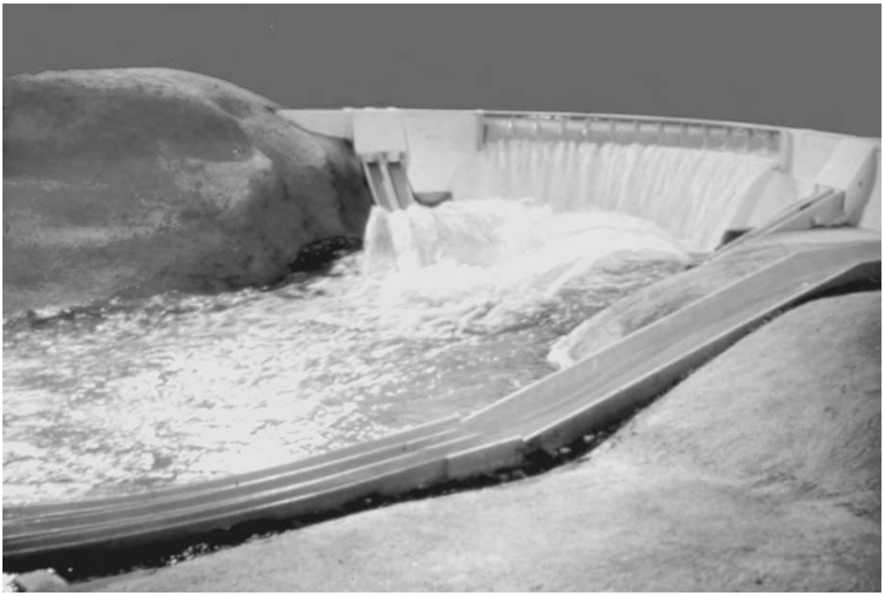
Hydrodynamics is the study of water flow. It helps us to understand how water behaves when it flows in pipes and channels and to answer such questions as – what diameter of pipe is needed to supply a village or a town with water? How wide and deep must a channel be to carry water from a dam to an irrigation scheme? What kind of pumps may be required and how big must they be? These are the practical problems of hydrodynamics.

Hydrodynamics is more complex than hydrostatics because it must take account of more factors, particularly the direction and velocity in which the water is flowing and the influence of viscosity. In early times hydrodynamics, like many other developments, moved forward on a trial and error basis. If the flow was not enough then a larger diameter pipe was used, if a pipe burst under the water pressure then a stronger one was put in its place. But during the past 250 years or so scientists have found new ways of answering the questions about size, shape and strength. They experimented in laboratories and came up with mathematical theories that have now replaced trial and error methods for the most common hydraulic problems.

3.2 Experimentation and theory

Experimentation was a logical next step from trial and error. Scientists built physical models of hydraulic systems in the laboratory and tested them before building the real thing. Much of our current knowledge of water flow in pipes and open channels has come from this kind of experimentation; empirical formulae were derived from the data collected to link water flow with the size of pipes and channels. Today we use formulae for most design problems, but there are still some problems which are not easily solved in this way. Practical laboratory experiments are still used to find solutions for the design of complex works such as harbours, tidal power stations, river flood control schemes and dam spillways. Small-scale models are built to test new designs and to investigate the impact of new engineering works both locally and in the surrounding area (Figure 3.1).

Formulae that link water flow with pipe and channel sizes have also been developed analytically from our understanding of the basic principles of physics – the properties of water and Newton's laws of motion. The rules of hydrostatics were developed analytically and have proved to work very well. But when water starts to move it is difficult to take account of all the new factors involved, in particular viscosity. The engineering approach, rather than the scientific one,



3.1 Laboratory model of a dam spillway.

is to try and simplify a problem by ignoring those aspects which do not have a great bearing on the outcome. In the case of water, viscosity is usually ignored because its effects are very small. This greatly simplifies problems. For example, ignoring the forces of viscosity makes pipeline design much simpler and it makes no difference to the final choice of pipe size. Other more important factors dominate the design process such as velocity, pressure and the forces of friction. These do have significant influence on the choice of pipe size and so it is important to focus attention on them. This is why engineering is often regarded as much an art as a science. The science is about knowing what physical factors must be taken into account but the art of engineering is knowing which of the factors can be safely ignored in order to simplify a problem without it seriously affecting the accuracy of the outcome.

Remember that engineers are not always looking for high levels of accuracy. There are inherent errors in all data and so there is little point in calculating the diameter of a pipe to several decimal places when the data being used have not been recorded with the same precision. Electronic calculators and computers have created much of this problem and many students still continue to quote answers to many decimal places simply because the computer says so. The answer is only as good as the data going into the calculation and so another skill of the engineer is to know how accurate an answer needs to be. Unfortunately this is a skill which can only be learned through practice and experience. This is the reason why a vital part of training young engineers always involves working with older, more experienced engineers to acquire this skill. Just knowing the right formula is just not enough.

The practical issues of cost and availability also impose limitations on hydraulic designs. For example, commercially available pipes come in a limited range of sizes, for example, 50 mm, 75 mm, 100 mm diameter. If an engineer calculates that a 78 mm diameter pipe is needed he is likely to choose the next size of pipe to make sure it will do the job properly, that is, 100 mm. So there is nothing to be gained in spending a lot of time refining the design process in such circumstances.

Simplifying problems so that they can be solved more easily, without loss of accuracy, is at the heart of hydrodynamics – the study of water movement.

3.3 Hydraulic toolbox

The development of hydraulic theory has produced *three* important basic tools (equations) which are fundamental to solving most hydrodynamic problems:

- discharge and continuity
- energy
- momentum.

They are not difficult to master and you will need to understand them well.

3.4 Discharge and continuity

Discharge refers to the volume of water flowing along a pipe or channel each second. Volume is measured in cubic metres (m^3) and so discharge is measured in cubic metres per second (m^3/s). Alternative units are litres per second (l/s) and cubic metres per hour (m^3/h).

There are two ways of determining discharge. The first involves measuring the volume of water flowing in a system over a given time period. For example, water flowing from a pipe can be caught in a bucket of known volume (Figure 3.2a). If the time to fill the bucket is recorded then the discharge from the pipe can be determined using the following formula:

$$\text{discharge (m}^3\text{/s)} = \frac{\text{volume (m}^3\text{)}}{\text{time (s)}}$$

Discharge can also be determined by multiplying the velocity of the water by the area of the flow. To understand this, imagine water flowing along a pipeline (Figure 3.2b). In one second the volume of water flowing past ×–× will be the shaded volume. This volume can be calculated by multiplying the area of the pipe by the length of the shaded portion. But the shaded length is numerically equal to the velocity v and so the volume flowing each second (i.e. the discharge) is equal to the pipe area multiplied by the velocity. Writing this as an equation:

$$\text{discharge } (Q) = \text{velocity } (v) \times \text{area } (a)$$

$$Q = va$$

The *continuity equation* builds on the discharge equation and simply means that the amount of water flowing into a system must be equal to the amount of water flowing out of it (Figure 3.2c).

inflow = outflow

And so:

$$Q_1 = Q_2$$

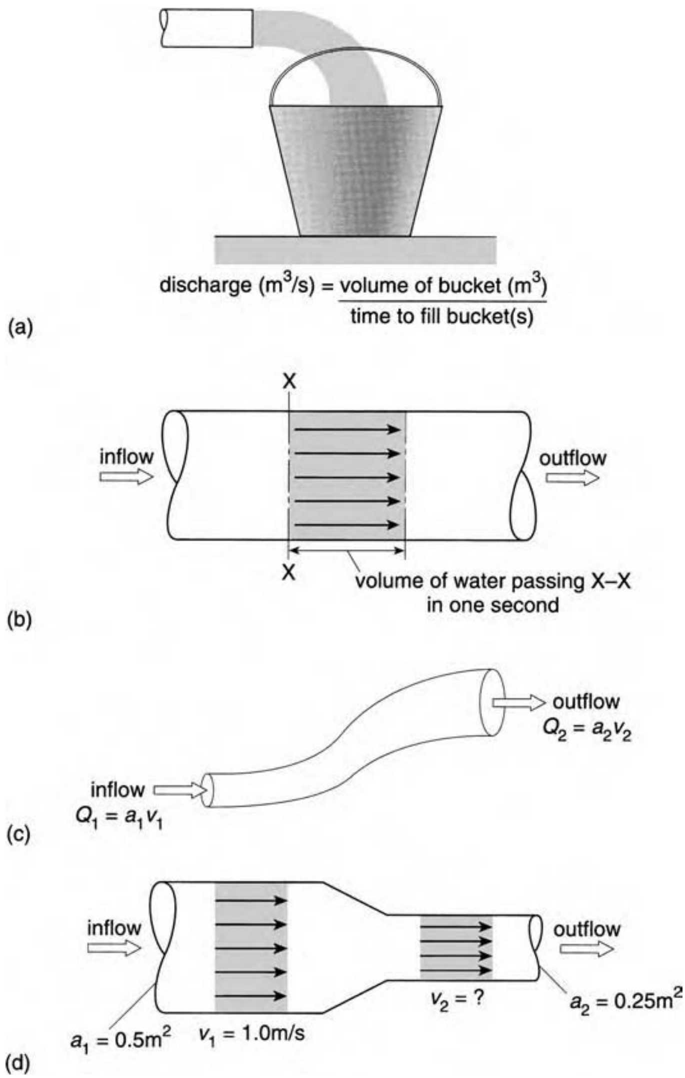
But from the discharge equation:

$$Q = va$$

And so:

$$v_1 a_1 = v_2 a_2$$

So the continuity equation not only links discharges it also links areas and velocities as well. This is a very simple but powerful equation and is fundamental to solving many hydraulic problems. An example in the box shows how this works in practice for a pipeline which changes diameter.



3.2 Discharge and continuity.

EXAMPLE: CALCULATING VELOCITY USING THE CONTINUITY EQUATION

A pipeline changes area from 0.5 to 0.25 m^2 (Figure 3.2d). If the velocity in the larger pipe is 1.0 m/s calculate the velocity in the smaller pipe.

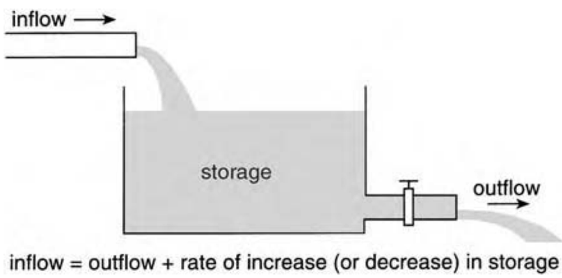
Use the continuity equation:

inflow = outflow

And so:

$$v_1a_1 = v_2a_2$$
$$1 \times 0.5 = v_2 \times 0.25$$
$$v_2 = 2 \text{ m/s}$$

Note how water moves much faster in the smaller pipe.



3.3 Continuity when there is water storage.

The simple equation of inflow equals outflow is only true when the flow is steady. This means the flow remains the same over time. But there are cases when inflow does not equal outflow. An example of this is a domestic storage tank found in most houses (Figure 3.3). The release of water from the tank may be quite different from the inflow. Dams are built on rivers to perform a similar function so that water supply can be more easily matched with water demand. In this case an additional term is added to the continuity equation to allow for the change in storage in the reservoir and so the continuity equation becomes:

inflow = outflow + rate of increase (or decrease) in storage

Hydrologists use this equation when studying rainfall and runoff from catchments and refer to it as the *water balance equation*.

3.5 Energy

The second of the basic tools uses energy to make the link between pressure and velocity in pipes and channels. Energy is described in some detail in Section 1.10 and in Chapter 8 on pumping. Suffice here to say that energy is the capacity of water to do useful work and water can possess energy in three ways:

- pressure energy
- kinetic energy
- potential energy.

Energy for solid objects has the dimensions of Nm. For fluids the dimensions are a little different. It is common practice to measure energy in terms of *energy per unit weight* and so energy for fluids has dimensions of Nm/N. The Newton terms cancel each other out and we are left with metres (m). This make energy look similar to pressure head as both are measured in metres. Indeed we shall see that the terms energy and pressure head are in fact interchangeable.

So let's explore these three types of energy.

3.5.1 Pressure energy

When water is under pressure it can do useful work for us. Water released from a tank could be used to drive a small turbine which in turn drives a generator to produce electrical energy (Figure 3.4a). So the pressure available in the tank is a measure of the energy available to do that work. It is calculated as follows:

$$\text{pressure energy} = \frac{p}{\rho g}$$

where p is pressure (kN/m^2); ρ is mass density (kg/m^3); g *is* gravity constant (9.81 m/s^2).

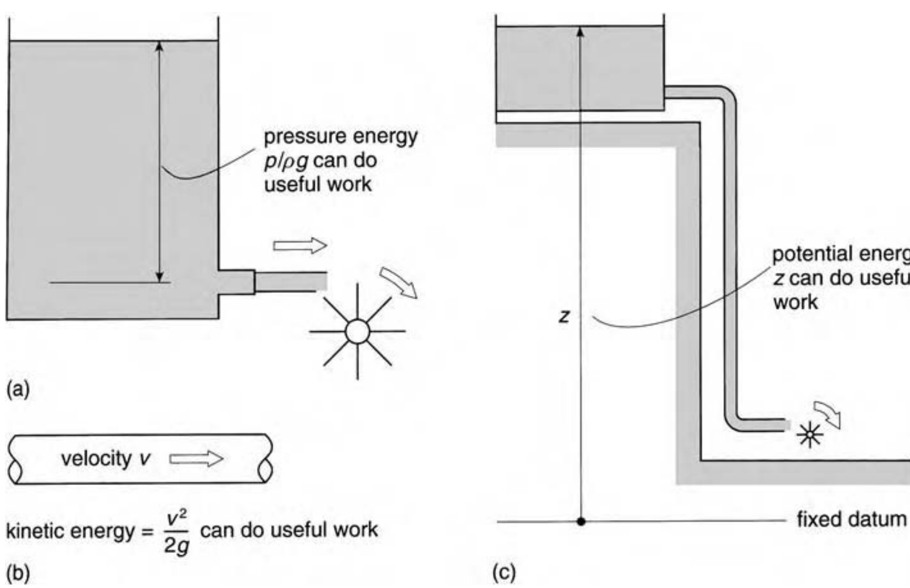
Notice that the equation for pressure energy is actually the same as the familiar pressure-head equation (remember). It is just presented in a different way. So pressure energy is in fact the same as the pressure head and is measured in metres (m).

3.5.2 Kinetic energy

When water flows it possesses energy because of this movement; this is known as *kinetic energy* – or sometimes velocity energy. The faster water flows the greater is its kinetic energy (Figure 3.4b). It is calculated as follows:

$$\text{kinetic energy} = \frac{v^2}{2g}$$

where v is velocity (m/s); g is gravity constant (9.81 m/s^2).



3.4 Pressure, kinetic and potential energy.

Kinetic energy is also measured in metres (m) and for this reason it is sometimes referred to as *velocity head*. An example of how to calculate kinetic energy is shown in the box.

EXAMPLE: CALCULATING KINETIC ENERGY

Calculate the kinetic energy in a pipeline when the flow velocity is 3.7 m/s.

$$\text{kinetic energy} = \frac{v^2}{2g}$$

$$= \frac{3.7^2}{2 \times 9.81} = 0.7 \text{ m}$$

This can also be thought of as a velocity head so calculate the equivalent pressure in kN/m^2 that would produce this kinetic energy.

To calculate velocity head as a pressure in kN/m^2 use:

$$\text{pressure} = \rho g h$$

$$= 1000 \times 9.81 \times 0.7$$

$$= 6867 \text{ N/m}^2 = 6.87 \text{ kN/m}^2$$

3.5.3 Potential energy

Water also has energy because of its location. Water stored in the mountains can do useful work by generating hydro-power whereas water stored on a flood plain has little or no potential for work (Figure 3.4c). So the higher the water source the more energy water has. This is called *potential energy*. It is determined by the height of the water in metres above some fixed datum point:

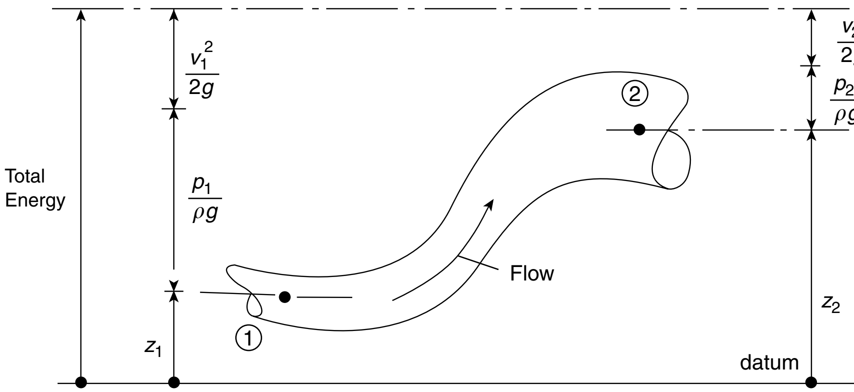
$$\text{potential energy} = z$$

where z is the height of the water in metres (m) above a fixed datum.

When measuring potential energy it is important to relate it to a fixed datum. It is similar to using sea level as the fixed datum for measuring changes in land elevation.

3.5.4 Total energy

The really interesting point of all this is that all the different forms of energy interchangeable (pressure energy can be changed to velocity energy and so on) and they can be added together to help us solve a whole range of hydraulic problems. The Swiss mathematician Daniel Bernoulli (1700–1782) made this most important discovery. Indeed it was Bernoulli who is said to have put forward the name of hydrodynamics to describe water flow. It led to one of the best known equations in hydraulics – *total energy equation*. It is often referred to as the *Bernoulli equation* in recognition of his contribution to the study of fluid behaviour.



3.5 Total energy is the same throughout the system.

The total energy in a system is the sum of all the different energies:

$$\text{total energy} = \frac{p}{\rho g} + \frac{v^2}{2g} + z$$

On its own, simply knowing the total energy in a system is of limited value. But the fact that the total energy will be the same throughout a system, even though the various components of energy may be different, makes it much more useful.

Take, for example, water flowing in a pipe from point 1 to point 2 (Figure 3.5). The total energy at point 1 will be the same as the total energy at point 2. So we can rewrite the total energy equation in a different and more useful way:

total energy at point 1 = total energy at point 2

And so:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

The velocity, pressure and height at 1 are all different to those at point 2 but when they are added together at each point the total is the same. This means that if we know some of the values at say point 1 we can now predict values at point 2. There are examples of this in the next section.

Note that the energy equation only works for flows where there is little or no energy loss. However, it is a reasonable assumption to make in many situations although not so reasonable for long pipelines where energy losses can be significant and so cannot be ignored. But for now, assume that water is an ideal fluid and that no energy is lost. Later, in Chapter 5, we will see how to incorporate energy losses into the equation.

3.6 Some useful applications of the energy equation

The usefulness of the energy equation is well demonstrated in the following examples.

3.6.1 Pressure and elevation changes

Pipelines tend to follow the natural ground contours up and down the hills. As a result, pressure changes simply because of differences in ground levels. For example, a pipeline running up the side of a hill will experience a drop in pressure of 10 m head for every 10 m rise in ground level. Similarly the pressure in a pipe running downhill will increase by 10 m for every 10 m fall in ground level. The energy equation explains why this is so.

Consider total energy at two points 1 and 2 along a pipeline some distance apart and at different elevations (Figure 3.6).

Assuming no energy losses between these two points, the total energy in the pipeline at point 1 is equal to the total energy at point 2.

total energy at 1 = total energy at 2

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

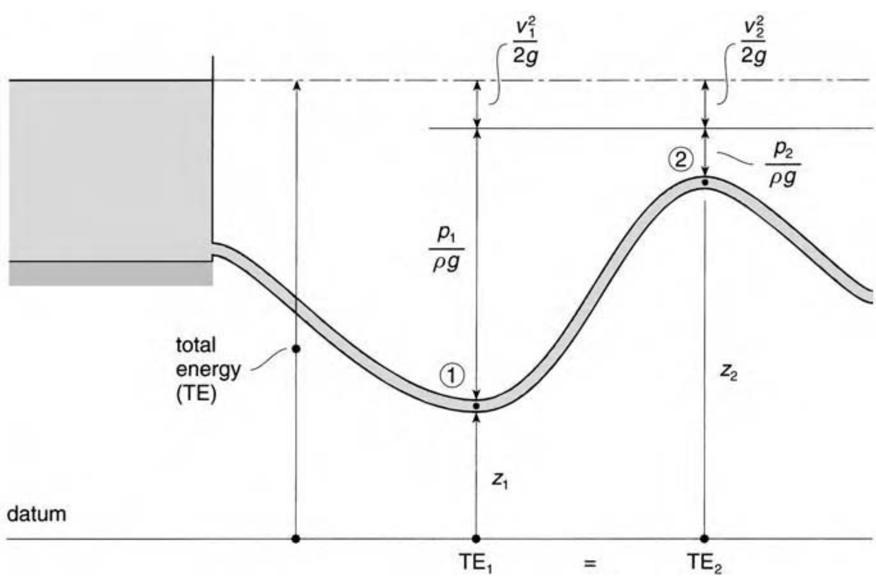
z_1 and z_2 are measured from some chosen horizontal datum.

Normally pipelines would have the same diameter and so the velocity at point 1 is the same as the velocity at point 2. This means that the kinetic energy at points 1 and 2 are also the same. The above equation then simplifies to:

$$\frac{p_1}{\rho g} + z_1 = \frac{p_2}{\rho g} + z_2$$

Rearranging this to bring the pressure terms and the potential terms together:

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = z_2 - z_1$$



3.6 Pressure changes with elevation.

Putting this into words:

changes in pressure (m) = changes in ground level (m)

Here p_1 and p_2 represent a pressure change between points 1 and 2 (measured in metres) which is a direct result of the change in ground level from z_1 to z_2. Note that this has nothing to do with pressure loss due to friction as is often thought – just ground elevation changes.

A numerical example of how to calculate changes in pressure due to changes in ground elevation is shown in the box.

EXAMPLE: CALCULATING PRESSURE CHANGES DUE TO ELEVATION CHANGES

A pipeline is constructed across undulating ground (Figure 3.6). Calculate the pressure at point 2 when the pressure at point 1 is 150 kN/m^2 and point 2 is 7.5 m above point 1. Assuming no energy loss along the pipeline this problem can be solved using the energy equation:

total energy at 1 = total energy at 2

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

As the pipe diameter is the same throughout, the velocity will also be the same as will the kinetic energy. So the kinetic energy terms on each side of the equation cancel each other out.
The equation simplifies to:

$$\frac{p_1}{\rho g} + z_1 = \frac{p_2}{\rho g} + z_2$$

Rearranging this gives:

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = z_2 - z_1$$

All elevation measurements are made from the same datum level and so:

$$z_2 - z_1 = 7.5 \text{ m}$$

This means that:

$$\frac{p_1 - p_2}{\rho g} = 7.5 \text{ m}$$

And so:

$$p_1 - p_2 = 1000 \times 9.81 \times 7.5 = 73\,575 \text{ N/m}^2 = 73.6 \text{ kN/m}^2$$

known pressure at point 1 = 150 kN/m^2

And so:

pressure at point 2 = 150 − 73.6 = 76.4 kN/m^2

So there is a drop in pressure at point 2 which is directly attributed to the elevation rise in the pipeline.

3.6.2 Measuring velocity

Another very useful application of the energy equation is for measuring velocity. This is done by stopping a small part of the flow and measuring the pressure change that results from this. Airline pilots use this principle to measure their air speed.

When water (or air) flows around an object (Figure 3.7a) most of it is deflected around it but there is one small part of the flow which hits the object head-on and stops. Stopping the water in this way is called *stagnation* and the point at which this occurs is the *stagnation point*. Applying the energy equation to the main stream and the stagnation point:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_s}{\rho g} + \frac{v_s^2}{2g} + z_s$$

Assuming the flow is horizontal:

$$z_1 = z_s$$

As the water stops:

$$v_s = 0$$

And so:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} = \frac{p_s}{\rho g}$$

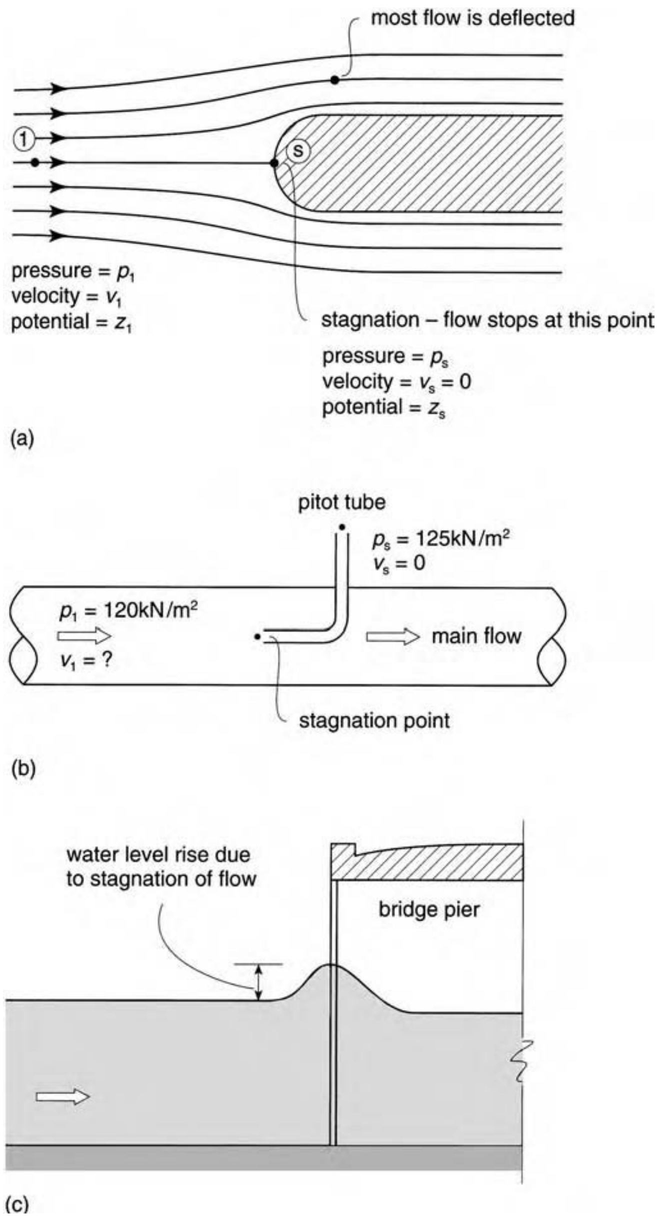
Rearranging this equation to bring all the velocity and pressure terms together:

$$\frac{v_1^2}{2g} = \frac{p_s}{\rho g} - \frac{p_1}{\rho g}$$

Rearranging it again for an equation for velocity v_1:

$$v_1 = \sqrt{2\left(\frac{p_s - p_1}{\rho}\right)}$$

So it is possible to calculate the main stream velocity by creating a stagnation point and measuring p_1 and p_s. This idea is used extensively for measuring water velocity in pipes using a



3.7 Measuring velocity using stagnation points.

device known as a pitot tube (Figure 3.7b). The stagnation pressure p_s on the end of the tube is measured together with the general pressure in the pipe p_1. The velocity is then calculated using the energy equation. One disadvantage of this device is that it does not measure the average velocity in a pipe but only the velocity at the particular point where the pitot tube is located. However, this can be very useful for experimental work that explores the changes in velocity across the diameter of a pipe to produce velocity profiles. Pitot tubes are also used on aircraft to measure their velocity. Usually the air is moving as well as the aircraft and so the pilot will adjust the velocity reading to take account of this.

Stagnation points also occur in channels. One example occurs at a bridge pier (Figure 3.7c). Notice how the water level rises a little just in front of the pier as the kinetic energy in the river changes to pressure energy as the flow stops. In this case the pressure rise is seen as a rise in water level. Although this change in water level could be used to determine the velocity of the river, it is rather small and difficult to measure accurately. So it is not a very reliable way of measuring velocity in channels.

EXAMPLE: CALCULATING THE VELOCITY IN A PIPE USING A PITOT TUBE

Calculate the velocity in a pipe using a pitot tube when the normal pipe operating pressure is 120 kN/m^2 and the pitot pressure is 125 kN/m^2 (Figure 3.7b).
Although there is an equation for velocity given in the text it is a good idea at first to work from basic principles to build up your confidence in its use. The problem is solved using the energy equation. Point 1 describes the main flow and point *s* describes the stagnation point on the end of the pitot tube:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_s}{\rho g} + \frac{v_s^2}{2g} + z_s$$

At the stagnation point:

$$v_s = 0$$

And as the system is horizontal:

$$z_1 = z_s = 0$$

This reduces the energy equation to:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} = \frac{p_s}{\rho g}$$

All the values in the equation are known except for v_1 so calculate v_1:

$$\frac{120\,000}{1000 \times 9.81} + \frac{v_1^2}{2 \times 9.81} = \frac{125\,000}{1000 \times 9.81}$$

$$12.23 + \frac{v_1^2}{2g} = 12.74$$

$$\frac{v_1^2}{2 \times 9.81} = 12.74 - 12.23 = 0.51 \text{ m}$$

$$v_1 = \sqrt{2 \times 9.81 \times 0.51}$$

$$v_1 = 3.16 \text{ m/s}$$

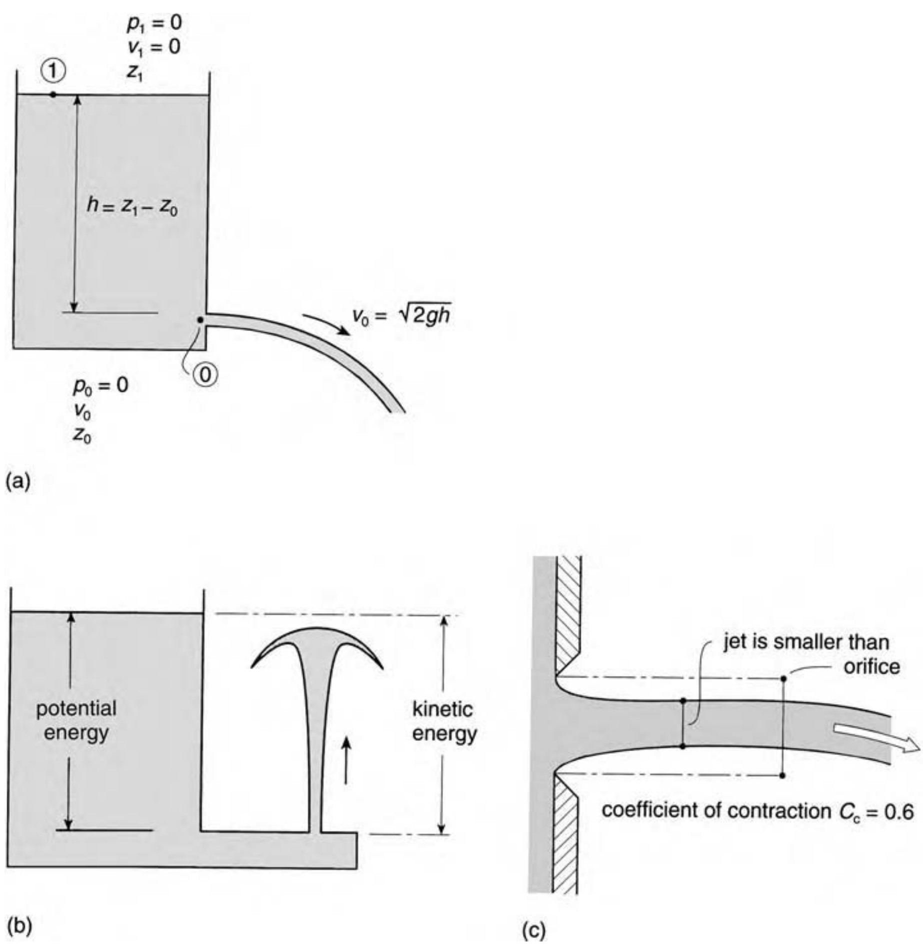
3.6.3 Orifices

Orifices are usually gated openings at the bottom of tanks and reservoirs used to control the release of water flow into a channel or some other collecting basin (Figure 3.8a). They are mostly rectangular or circular openings. The energy equation makes it possible to calculate the discharge released through an orifice by first calculating the flow velocity from the orifice and then multiplying it by the area of the opening. One important proviso at this stage is that the orifice must discharge freely and unhindered into the atmosphere, otherwise this approach will not work. Some orifices do operate in submerged conditions and this does affect the flow. But this is described later in Section 7.2.

The energy equation for a tank with an orifice (Figure 3.8a) is written as:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_0}{\rho g} + \frac{v_0^2}{2g} + z_0$$

Note the careful choice of the points for writing the energy terms. Point 1 is chosen at the water surface in the tank and point 0 is at the centre of the orifice.



3.8 Flow through orifices.

At the water surface the pressure is atmospheric and so is assumed to be zero (remember all pressures are measured relative to atmospheric pressure which is taken as the zero point). Also the downward velocity in the tank is very small and so the kinetic energy is also zero. All the initial energy is potential. At the orifice the jet comes out into the atmosphere; as the jet does not burst open it is assumed that the pressure in and around the jet is atmospheric pressure, that is, zero. So the equation reduces to:

$$z_1 = \frac{v_0^2}{2g} + z_0$$

Rearranging this equation:

$$\frac{v_0^2}{2g} = z_1 - z_0$$

Put:

$$z_1 - z_0 = h$$

Now rearrange again to obtain an equation for v_0:

$$v_0 = \sqrt{2gh}$$

Evangelista Torricelli (1608–1647) first made this connection between the pressure head available in the tank and the velocity of the emerging jet some considerable time before Bernoulli developed his energy equation. As a pupil of Galileo he was greatly influenced by him and applied his concepts of mechanics to water falling under the influence of gravity. Although the above equation is now referred to as Torricelli's law he did not include the *2 g* term. This was introduced much later by other investigators.

Torricelli sought to verify this law by directing a water jet from an orifice vertically upwards (Figure 3.8b). He showed that the jet rose to almost the same height as the free water surface in the tank showing that the potential energy in the tank and the velocity energy at the orifice were equal. So knowing the pressure head available in a pipe, it is possible to calculate the height to which a water jet would rise if a nozzle was attached to it – very useful for designing fountains!

The velocity of a jet can also be used to calculate the jet discharge using the discharge equation:

$$Q = a\,v$$

So:

$$Q = a\sqrt{2gh}$$

The area of the orifice *a* is used in the equation because it is easy to measure, but this means the end result is not so accurate because the area of the jet of water is not the same as the area of the orifice. As the jet emerges and flows around the edge of the orifice it follows a curved path and so the jet ends up smaller in diameter than the orifice (Figure 3.8c). The contraction of the jet is taken into account by introducing a *coefficient of contraction* C_c. This has a value of approximately 0.6. So the discharge formula now becomes:

$$Q = C_c a\sqrt{2gh}$$

Although it might be interesting to work out the discharge from holes in tanks, a more useful application of Torricelli's law is the design of underflow gates for both measuring and controlling discharges in open channels (Section 7.8).

3.6.4 *Pressure and velocity changes in a pipe*

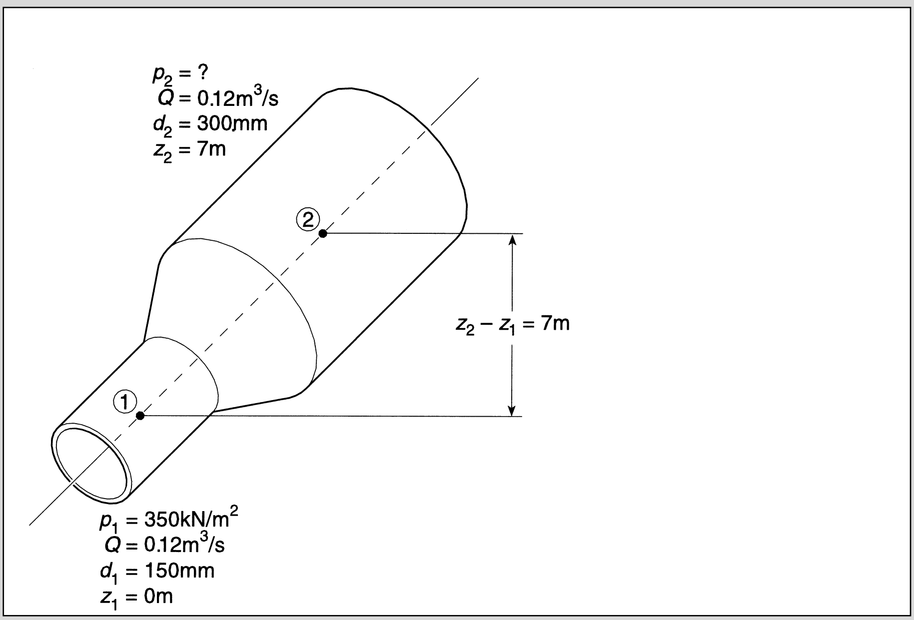
A more general and very practical application of the energy equation is to predict pressures and velocities in pipelines as a result of changes in ground elevation and pipe sizes. An example in the box shows just how versatile this equation can be.

EXAMPLE: CALCULATING PRESSURE CHANGES USING THE ENERGY EQUATION

A pipeline carrying a discharge of 0.12 m^3/s changes from 150 mm diameter to 300 mm diameter and rises through 7 m. Calculate the pressure in the 300 mm pipe when the pressure in the 150 mm pipe is 350 kN/m^2.

This problem involves changes in pressure, kinetic and potential energy and its solution requires both the energy and continuity equations. The first step is to write down the energy equation for the two points in the systems 1 and 2:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$



3.9 Calculating changes in pressure in a pipeline.

The next step is to put all the known values into the equation, identify the unknowns, and then determine their values. Here p_1, z_1 and z_2 are known values but p_2 is unknown and so are v_1 and v_2. First determine v_1 and v_2, use the continuity equation:

$$Q = va$$

Rearranging this to calculate *v:*

$$v = \frac{Q}{a}$$

And so:

$$v_1 = \frac{Q}{a_1} \quad \text{and} \quad v_2 = \frac{Q}{a_2}$$

The pipe areas are not known but their diameters are known, so next calculate their cross-sectional areas:

$$a_1 = \frac{\pi d_1^2}{4} = \frac{\pi 0.15^2}{4} = 0.018 \text{ m}^2$$

$$a_2 = \frac{\pi d_2^2}{4} = \frac{\pi 0.3^2}{4} = 0.07 \text{ m}^2$$

Now calculate the velocities:

$$v_1 = \frac{Q}{a} = \frac{0.120}{0.018} = 6.67 \text{ m/s}$$

$$v_2 = \frac{Q}{a} = \frac{0.120}{0.07} = 1.71 \text{ m/s}$$

Putting all the known values into the energy equation:

$$\frac{350\,000}{1000 \times 9.81} + \frac{6.67^2}{2 \times 9.81} + 0 = \frac{p_2}{\rho g} + \frac{1.71^2}{2 \times 9.81} + 7$$

Note although pressures are quoted in kN/m^2 it is less confusing to work all calculations in N/m^2 and then convert back to kN/m^2. The equation simplifies to:

$$35.68 + 2.26 = \frac{p_2}{\rho g} + 0.15 + 7$$

Rearranging this equation for p_2:

$$\frac{p_2}{\rho g} = 35.68 + 2.26 - 0.15 - 7$$
$$= 30.8 \text{ m head of water}$$

To determine this head as a pressure in kN/m^2 use the pressure-head equation:

$$\text{pressure} = \rho g h$$
$$p_2 = 1000 \times 9.81 \times 30.8$$
$$p_2 = 302\ 000 \text{ N/m}^2 = 302 \text{ kN/m}^2$$

3.7 Some more energy applications

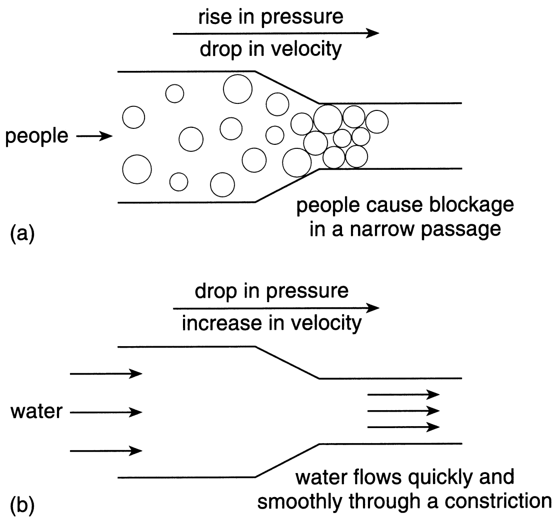
3.7.1 Flow through narrow openings

When water flows through narrow openings in pipes and channels such as valves or gates there is a tendency to assume they are constricting the flow. But this is not always the case. The reason for this misunderstanding is that we live in a solid world and so we logically apply what we see to water. Cars and people jam and cause chaos when too many try to get through a narrow opening at the same time. So surely water must behave in a similar way. Well this is where water surprises everyone – it behaves quite differently.

When water flows along a pipe and meets a constriction continuity and energy control what happens. As the pipe becomes narrower the water, rather than slowing down, actually increases in velocity. The continuity equation tells us that when the area is smaller the velocity must be greater. But surely the constriction must slow the whole discharge and hence the velocity? Well no – the discharge is governed by the total energy available to drive the flow and as there is no change in the total energy between the main pipe and the constriction the discharge does not change. So the flow passes quickly and smoothly through the constriction without fuss. It would seem that water behaves much more sensibly than people!

What does happen, of course, is that the pressure in the system changes at the constriction. It drops as the increase in kinetic energy is gained at the expense of pressure energy. So a narrow pipe, or indeed any other constriction such as a partly open valve, does not throttle the flow, it just speeds it up so that it goes through much faster. You can see this when you open and close a tap at home. The discharge through a partially open tap is almost the same as that through a fully open one. The total energy available is the same but the flow area is smaller when it is partially opened and so the water just flows through with a greater velocity. Of course, the velocity is eventually slowed when the tap is almost shut and at this point energy losses at the tap dominate the flow.

This same principle also applies to flow in open channels. When flow is constricted it speeds up (kinetic energy increases) and the water level drops (pressure energy decreases). You can see this effect as water flows through channel constrictions at bridges and weirs (this is discussed more fully in Chapters 4 Pipes and 5 Channels).



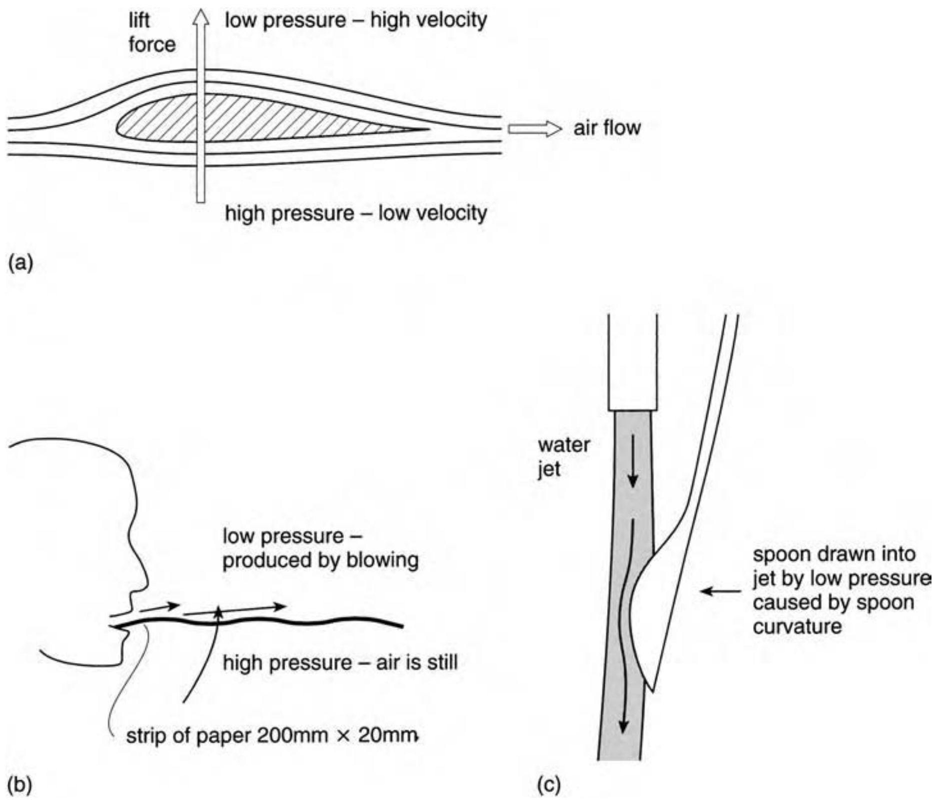
3.10 People and water flow differently through narrow passages.

Some people have suggested that the design of sports stadiums, which can easily become congested with people, could benefit from linking the flow of people to the flow of water. Some years ago there was a major accident at a football stadium in Belgium in which many people were crushed to death when those at the rear of the stadium suddenly surged forward in a narrow tunnel pushing those in front onto fixed barriers and crushing them. At the time it was suggested that stadiums should, in future, be designed with hydraulics in mind so the layout, size and shape of tunnels and barriers would allow people to 'flow' smoothly onto the terraces in a more orderly and safe manner. This is a dangerous analogy because people do not 'flow' like water. They tend to get stuck in narrow passages and against solid barriers whereas water behaves much more sensibly, flowing around barriers and speeding up and slowing down when needed to get through the tight spots.

3.7.2 How aeroplanes fly

Although some people think that aircrafts are lifted much in the same way as a flat stone is lifted as it skips across water (Section 3.13), this is not the way it works. Aircrafts rely on energy changes around their wings to fly. These changes the forces necessary to lift it into the air. An aircraft wing is specially shaped so that the air flow path is longer over the wing than under it (Figure 3.11a). So when an aircraft is taking off the air moves faster over the wing than under it. This is necessary to maintain continuity of air flow around the wing. The result is an increase in kinetic energy over the wing. But the total energy around the wing does not change and so there is a corresponding reduction in the pressure energy above the wing. This means that the pressure above the wing is less than that below it and so the wing experiences a lift force. This can be a significant force that can lift hundreds of tons of aeroplane into the air. It never ceases to amaze people and it works every time.

Have you noticed that aeroplanes usually take off into the wind. This is because the extra velocity of the wind provides a larger change in pressure and so provides extra lift. This is particularly important at take-off when an aeroplane it at its heaviest and carrying it full fuel load.



3.11 Aircraft rely on energy changes around their wings to fly.

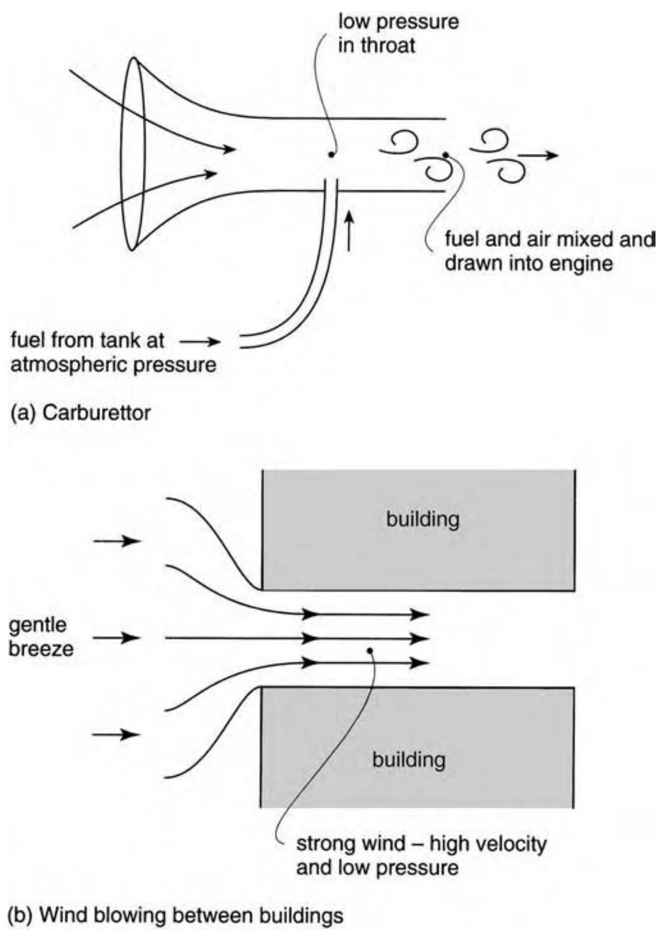
The same principle is used in reverse on racing cars. In this case the wing is upside down and located on the back of the car. The velocity of the air flowing over it, due to the forward movement of the car, produces a downward thrust which holds the car firmly on the road. The faster the car the greater is the down thrust which improves road holding and helps drivers to maintain high speeds even when cornering.

You can simply experience this lift force yourself (Figure 3.11b). Tear off a strip of paper approx. 20 mm wide and 200 mm long. Grip the paper firmly in your teeth and blow gently across the top of the paper. You will see the paper rise to a horizontal position. This is because the blowing action increases the velocity of the air and so reduces the pressure. The pressure below the paper is higher than above it and so the paper lifts – just like the aeroplane.

One way to feel the substantial force involved is to hold a spoon with its convex side close to water running from a tap (Figure 3.11c). Surprisingly the water does not push the spoon away; rather it draws it into the water. This is because the water velocity increases as it flows around the spoon causing a drop in pressure. This draws the spoon into the jet with surprising force.

3.7.3 Carburettors

Carburettors use changes in energy to put fuel into petrol engines (Figure 3.12a). The movement of the pistons in the engine draws air into the carburettor. The air velocity increases as it



3.12 Applying the energy equation.

moves into the narrow section (or throat) but as the total energy of the air remains the same the pressure energy in the throat drops below that of the surrounding atmosphere. The petrol in the carburettor is stored at atmospheric pressure and so fuel begins to flow from the higher (atmospheric) pressure in the tank to the lower pressure in the throat. It then mixes with the air and is drawn into the engine.

3.7.4 Fluid injectors

A device similar to a carburettor is used to inject one fluid into another such as the injection of fertiliser into irrigation water. A narrow section of pipe is located in the main irrigation pipeline which causes the velocity to increase and the pressure to drop. Some of the flow passes from the main pipe upstream of the throat (where the pressure is high) through the fertiliser tank and back into the pipe via the throat (where the pressure is low) taking some fertiliser with it. The turbulence just downstream of the throat, where the pipe expands again to its normal size, ensures that the fertiliser is well mixed in the flow.

3.7.5 Strong winds

Most people have noticed how suddenly the wind becomes much stronger in the gaps between buildings (Figure 3.12b). This is another example of the effect of changing energy. A narrow gap causes an increase in wind velocity and a corresponding drop in air pressure. The pressure drop can cause doors to bang because the pressure between the buildings is lower than the pressure inside them (remember the air inside is still and at normal atmospheric pressure).

3.7.6 Measuring discharge

These are just some examples of how continuity and energy can explain many interesting phenomena. But one very useful application for water is for measuring discharge. Changing the energy in pipes and channels produces changes in pressure which can be more easily measured than velocity. Using the energy and continuity equations, the pressure change is used to calculate velocities and hence the discharge (see Sections 4.10 and 7.7).

3.8 Momentum

The momentum equation is the third tool in the box. Momentum is about movement and the forces which cause it (see Section 1.11). It is the link between force, mass and velocity and is used to determine the forces created by water as it moves through pipes and hydraulic structures.

The momentum equation is normally written as:

$$\text{force (N)} = \text{mass flow (kg/s)} \times \text{change in velocity (m/s)}$$

But:

$$\text{mass flow (kg/s)} = \text{mass density (kg/m}^3) \times \text{discharge (m}^3\text{/s)}$$
$$= \rho Q$$

And:

$$\text{velocity change} = v_2 - v_1$$

where v_1 and v_2 represent two velocities in a system, and so:

$$\text{force} = \rho Q(v_2 - v_1)$$

This is now in a form that is useful for calculating forces in hydraulics. An example of the use of this equation is shown in the box. Other more practical applications in pipes and hydraulic structures follow in Sections 4.11 and 5.7.

EXAMPLE: CALCULATING THE FORCE ON A PLATE FROM A JET OF WATER

A jet of water with a diameter of 60 mm and a velocity of 5 m/s hits a vertical plate. Calculate the force of impact of the jet on the plate (Figure 3.13).

It is important to remember two points when dealing with momentum:

- forces and velocities are vectors and so their direction is important as well as their magnitude;
- the force of the water jet on the plate is equal to the force of the plate on the water. They are the same magnitude but in opposite directions (remember Newton's third law).

When forces are involved in a problem use the momentum equation

$$-F = \rho Q\,(v_2 - v_1)$$

Notice that flow and forces from left to right are all positive and those from right to left are negative. *F* is the force of the plate on the water and is in the opposite direction to the flow and so it is negative. (Working out the right direction can be rather tricky sometimes and so working with the momentum equation does take some practice.) Reversing all the signs in the above equation makes *F* positive:

$$F = \rho\, Q\,(v_1 - v_2)$$

The next step is to calculate the discharge *Q*:

$$Q = va = v \times \frac{\pi d^2}{4}$$

$$= 5 \times \frac{\pi 0.06^2}{4} = 0.014 \text{ m}^3\text{/s}$$

For this problem $v_2 = 0$ because the velocity of the jet after impact *in the direction of the flow* is zero. So putting in the known values into the momentum equation:

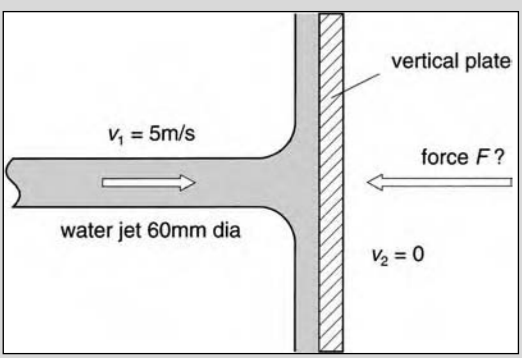


Figure 3.13 Applying momentum.

3.9 Real fluids

The assumption made so far in this chapter is that water is an ideal fluid. This means it has no viscosity and there is no friction between the flow and the boundaries. Real fluids have internal friction (viscosity) and also friction forces that exist between the fluid and the flow boundary such as the inside of a pipe. Water is a real fluid but its viscosity is low and so ignoring this has little or no effect on the design of pipes and channels. However the friction between the flow and the boundary is important and cannot be ignored for design purposes. We use a modified version of the energy equation to take account of this.

3.9.1 Taking account of energy losses

When water flows along pipes and channels energy is lost from friction between the water and its boundaries; we can account for this in the energy equation. Writing the energy equation for points 1 and 2 along a pipeline carrying a real fluid needs an additional term h_f to describe the energy loss between them:

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 + h_f$$

h_f is the most important element in this equation for determining the size of pipe or channel needed to carry a given flow. The question is how to measure or calculate it and what factors influence its value. This was the challenge faced by 19th century scientists investigating fluid flow and the results of their work now form the basis of all pipe and channel design procedures. But more about this in Chapters 4 and 5.

3.9.2 Cavitation

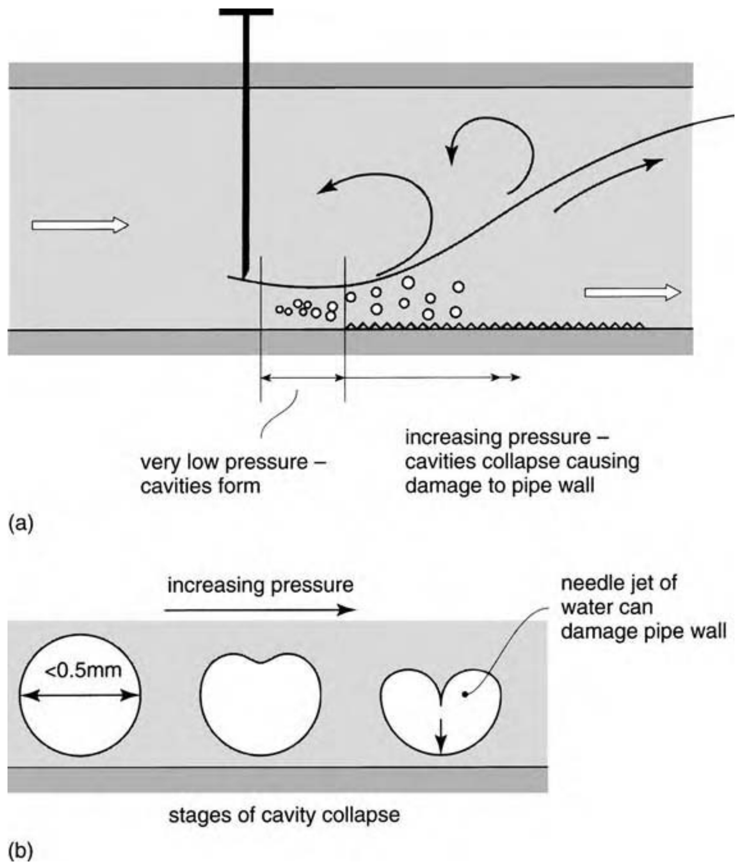
Real fluids suffer from cavitation and it can cause lots of problems, particularly in pumps and control valves. It occurs when a fluid is moving very fast; as a consequence, the pressure can drop to very low values approaching zero (vacuum pressure).

The control valve on a pipeline provides a good example (Figure 3.14a). When the valve is almost closed the water velocity under the gate can be very high. This also means high kinetic energy and this is gained at the expense of the pressure energy. If the pressure drops below the vapour pressure of water (this is approximately 0.3 m absolute) bubbles, called *cavities*, start to form in the water. They are very small (less than 0.5 mm in diameter) but there are many thousands of them and give the water a milky appearance. The bubbles are filled with water vapour and the pressure inside them is very low. But as the bubbles move under the gate and into the pipe downstream, the velocity slows, the pressure rises and the bubbles begin to collapse. It is at this point that the danger arises. If the bubbles collapse in the main flow they do no harm, but if they are close to the pipe wall they can do a great deal of damage. Notice the way in which the bubbles collapse (Figure 3.14b). As the bubble becomes unstable a tiny needle jet of water rushes across the cavity and it is this which can do great damage even to steel and concrete because the pressure under the jet can be as high as 4000 bar! See Section 8.4.4 for more details of cavitation in pumps.

Some people confuse cavitation with air entrainment, but it is a very different phenomenon. Air entrainment occurs when there is turbulence at hydraulic structures and air bubbles are drawn into the flow. The milky appearance of the water is similar but the bubbles are air filled and will do no harm to pumps and valves.

3.9.3 Boundary layers

Friction between water flow and its boundaries and the internal friction (viscosity) within the water gives rise to an effect known as the *boundary layer*. Water flowing in a pipe moves faster in the middle of the pipe than near the pipe wall. This is because friction between the water and

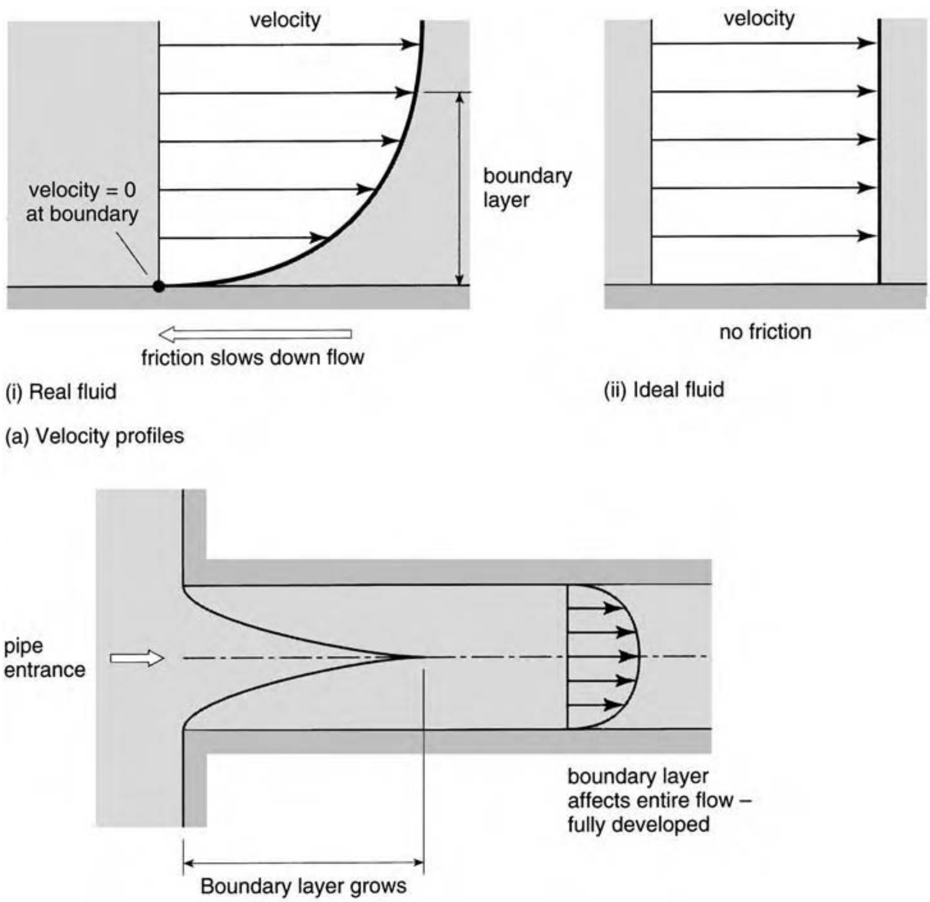


3.14 Dangers of cavitation.

the pipe wall slows down the flow. Very near to the pipe wall water actually sticks to it and the velocity is zero, although it is not possible to see this with the naked eye. Gradually the velocity increases further away from the wall until it reaches its maximum velocity in the centre of the pipe. To understand how this happens. imagine the flow is like a set of thin plates that can slide over each other. The plate nearest the wall is not moving and so it tries to slow down the plate next to it – the friction between the plates comes from the viscosity of the water (see Section 1.12.3 for more on viscosity). Plates further away from the wall are less affected by the boundary wall and so they move faster until the ones in the middle of the flow are moving fastest. All the flow affected by the pipe wall in this way is called the *boundary layer*. The use of the word *layer* can be misleading here as it is often confused with the layer of water closest to the pipe wall. This is not the case. It refers to all the flow which is slowed down as a result of the friction from the boundary. In the case of a pipe it can affect the entire flow across the pipe.

A graphical representation of the changes in velocity near a boundary is called the *velocity profile* (Figure 3.15a). The velocity varies from zero near the boundary to a maximum in the centre of a pipe or channel where the boundary has least effect. Compare this with the velocity profile for an ideal fluid. Here there is no viscosity and no friction from the boundary and so the velocity is the same across the entire flow.

Boundary layers grow as water enters a pipeline (Figure 3.15b). It quickly develops over the first few metres until it meets in the middle. From this point onwards the pipe boundary



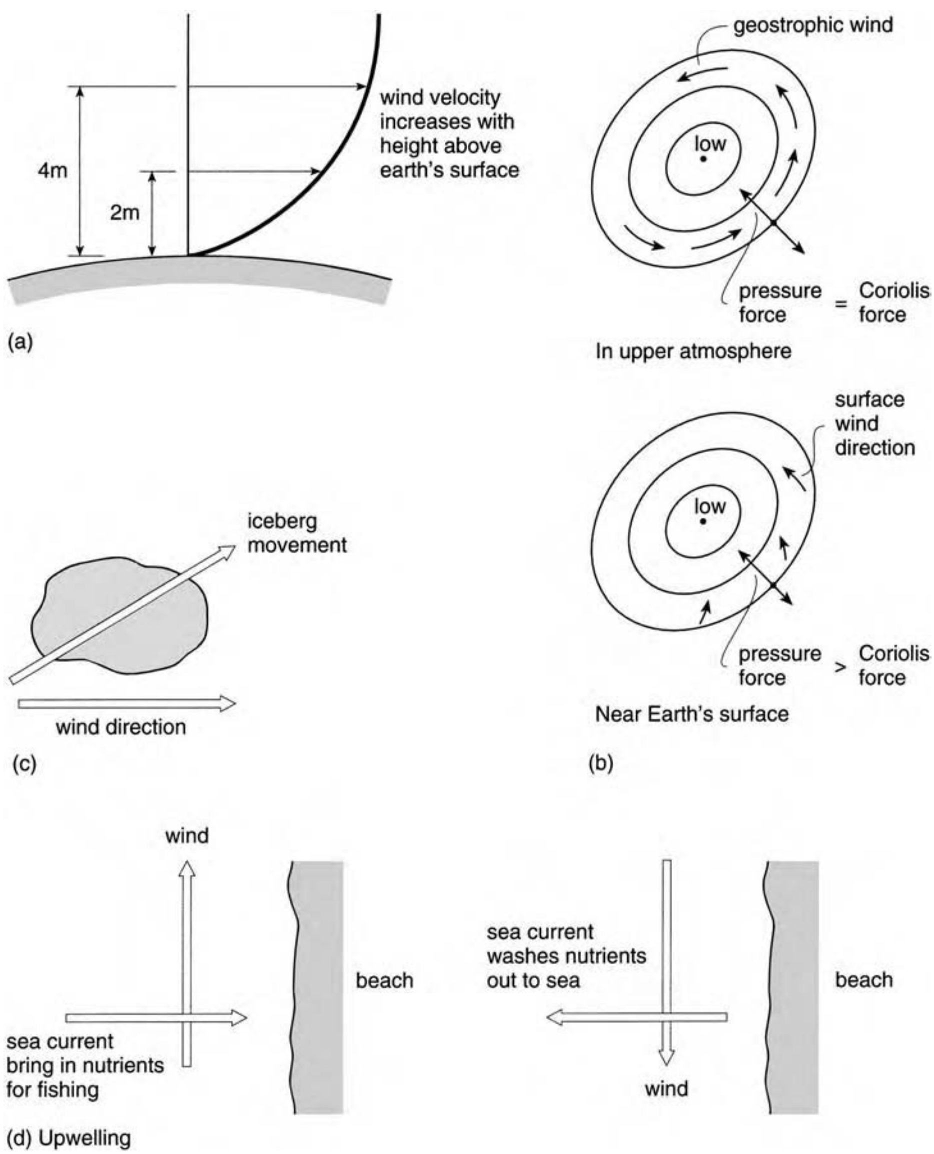
3.15 Boundary effects.

influences the entire flow in the pipe. In channels the boundary effects of the bed and sides similarly grow over a few metres of channel and soon influence the entire flow. When the boundary layer fills the entire flow it is said to be *fully developed*. This fully developed state is the basis on which all the pipe and channel formulae are based in Chapters 4 and 5.

3.9.3.1 The earth's boundary layer

The earth's surface produces a boundary layer when the wind blows (Figure 3.16a). The wind is much slower near the ground where it is affected by friction between the air and the earth's surface and its influence extends many metres above the earth's surface. For this reason it is important to specify the height at which wind speed is measured in meteorological stations. At 2 m above the ground the wind is much slower than at 4 m.

An interesting feature of the earth's boundary layer is that not only does the wind slow down near the earth's surface but it also gradually changes direction (Figure 3.16b). In the upper atmosphere, well beyond the boundary layer, the isobars (the lines of equal pressure) in a depression circle around the point of lowest pressure and the direction of the wind is always parallel to the isobars. This is because there is a balance between two important forces; the



3.16 Earth's boundary layer.

Coriolis force, which is a small but significant force that comes from the earth's rotation, and the force trying to pull the air into the centre of the depression because of the difference in pressure. So the wind circulates around the centre of the depression and is known as the *geostrophic wind*. The Coriolis force does not affect us as individuals as we are too small but it does affect the movement of large masses such as the air and the sea. Nearer the earth's surface, in the boundary layer, the wind slows down and this reduces the effect of the Coriolis force. The two forces are now out of balance and so the wind direction gradually changes as it is pulled in towards the centre of the depression. This is why the ground surface wind direction on weather maps is always at an angle to the isobars and points inwards towards the centre of the depression. This gradual twisting of the wind direction produces a spiral which is called the *Eckman Spiral*.

Eckman first observed this spiral at sea. He noticed that, in a strong wind, icebergs do not drift in the same direction as the wind but at an angle to it (Figure 3.16b). Surface winds can cause strong sea water currents and although the surface current may be in the direction of the wind those currents below the surface are influenced both by the boundary resistance from the sea bed and the Coriolis force from the earth's rotation. The effect is similar to that in the atmosphere. The lower currents slow down because of friction and gradually turn under the influence of the Coriolis force. So at the sea surface the water is moving in the same direction as the wind, but close to the sea bed it is moving at an angle to the wind. As icebergs float over 90% submerged their movement follows the water current rather than the wind direction and so they move at an angle to the wind.

This spiral effect is vital to several fishing communities around the world and is referred to as up-welling and is associated with the more well known El Niño effect (Figure 3.16d). In Peru when the surface wind blows along the coast line the boundary layer and the Coriolis force conspire to induce a current along the sea bed at right angles to the wind direction. This brings all the vegetative debris and plankton on which fish like to feed into the shallow waters of the shoreline and so the fishing is very good. However, when the wind blows in the opposite direction the current is reversed and all the food is washed out to sea leaving the shallow coastal fishing grounds bare and the fishing industry devastated.

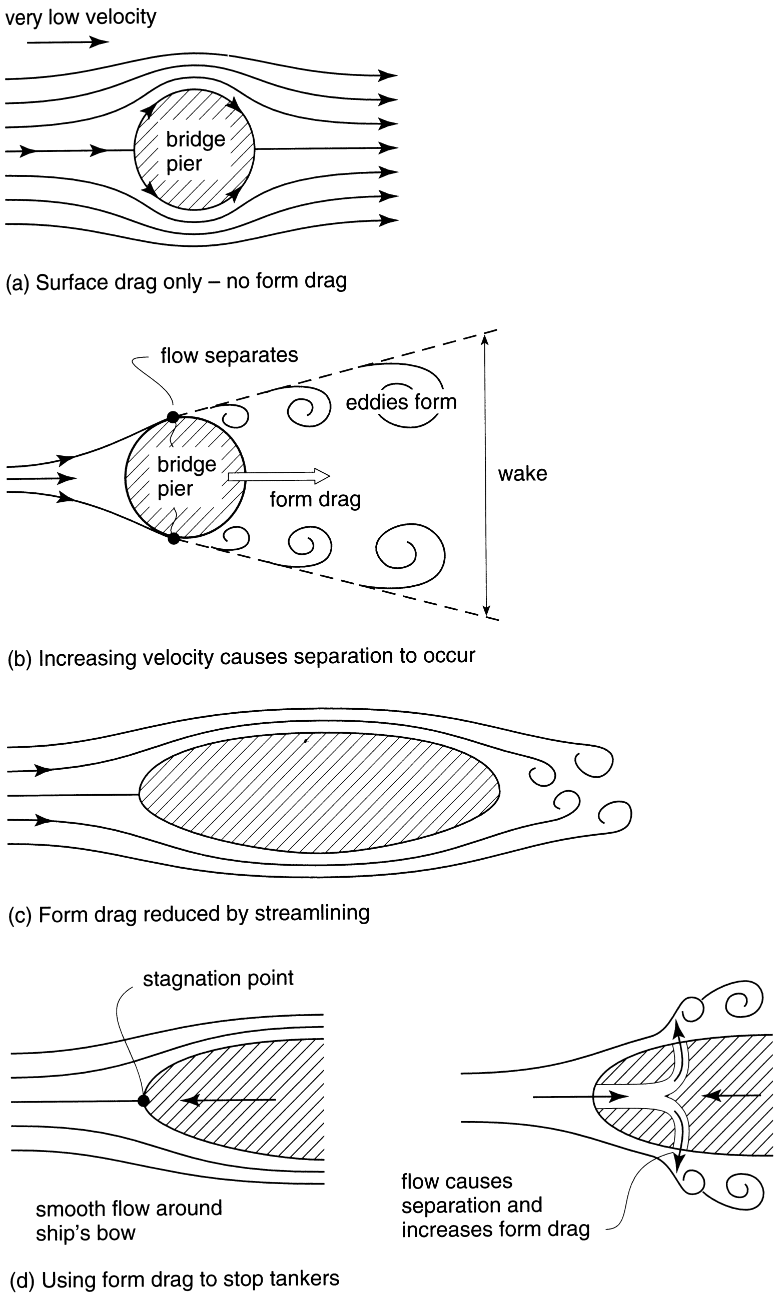
3.10 Drag forces

Boundary layers occur around all kinds of objects, for example, water flow around ships and submarines, air flow around aircraft and balls thrown through the air. Friction between the object and the fluid slows them down and it is referred to as a *drag force*. You can feel this force by putting your hand through the window of a moving car or in a stream of flowing water.

Sir George Stokes (1819–1903), an eminent physicist in his day, was one of the first people to investigate drag by examining the forces on spheres falling through different fluids. He noticed that the spheres fell at different rates, not just because of the viscosity of the fluids but also because of the size of the spheres. He also found that the falling spheres eventually reach a constant velocity which he called the *terminal velocity*. This occurred when the force of gravity causing the balls to accelerate was balanced by the resistance resulting from the fluid viscosity and the size of the balls.

Stokes also demonstrated that for any object dropped in a fluid (or a stationary object placed in a flowing fluid which is essentially the same) there were two types of drag: *surface drag* or skin friction which resulted from friction between the fluid and the object, and *form drag* which resulted from the shape and size of the object.

Water flowing around a bridge pier in a river provides a good example of the two types of drag. When the velocity is very low, the flow moves around the pier as shown in Figure 3.17a. The water clings to the pier and in this situation there is only surface drag and the shape of the pier has no effect. The flow pattern behind the pier is the same as the pattern upstream. But as the velocity increases, the boundary layer grows and the flow can no longer cling to the pier and so it separates (Figure 3.17b). It behaves like a car that is travelling too fast to get around a tight bend. It spins away from the pier and creates several small whirlpools which are swept downstream. These are called *vortices* or *eddies* and together they form what is known as the *wake* (Figure 3.17b). The flow pattern behind the pier is now quite different from that in front and in the wake the pressure is much lower than in front. It is this difference in pressure that results in the *form drag*. It is additional to the surface drag and its magnitude depends on the shape of the pier. Going back to your hand through the car window. Notice how the force changes when



3.17 Boundaries and drag.

you place the back or side of your hand in the direction of the flow. The shape of your hand in the flow determines the form drag.

Form drag is usually more important than surface drag and it can be reduced by shaping a bridge pier so that the water flows around it more easily and separation is delayed or avoided. Indeed, if separation could be avoided completely then form drag would be eliminated and the only concern would be surface drag. Shaping piers to produce a narrow wake and reduce form drag is often called *streamlining* (Figure 3.17c). This is the basis of design not just for bridge piers but also for aircraft, ships and cars to reduce drag and so increase speed or reduce energy requirements.

Swimmers too can benefit from reducing drag. This is particularly important at competitive levels when a few hundredths of a second can mean the difference between a gold and a silver medal. Approximately 90% of the drag on a swimmer is form drag and only 10% is surface drag. Some female swimmers try to reduce form drag by squeezing into a swim suit two or three sizes too small for them in order to improve their shape in the water.

Although women swimmers may seem to have an advantage in having a more streamline shape than bulky males, their shape does present some hydraulic problems. A woman's breasts cause early flow separation which increases turbulence and form drag. One swimwear manufacturer has found a solution to this by using a technique used by the aircraft industry to solve a similar problem. Aircraft wings often have small vertical spikes on their upper surface and these stop the flow from separating too early by creating small vortices, that is, zones of low pressure, close to the wing surface. This not only reduces form drag significantly but helps to avoid stalling (very early separation) which can be disastrous for an aircraft. The new swimsuit has tiny vortex generators located just below the breasts, which cause the boundary layer to cling to the swimmer and not separate, thus reducing form drag. The same manufacturer has also developed a ribbed swimsuit which creates similar vortices along the swimmer's body to try and stop the flow from separating. The manufacturer claims a 9% reduction in drag for the average female swimmer over a conventional swim suit.

Dolphins probably have the best known natural shape and skin for swimming. Both their form and surface drag are very low and this enables them to move through the water with incredible ease and speed – something that human beings have been trying to emulate for many years!

There is a way of calculating drag force:

$$\text{drag force} = \frac{1}{2} C \rho a v^2$$

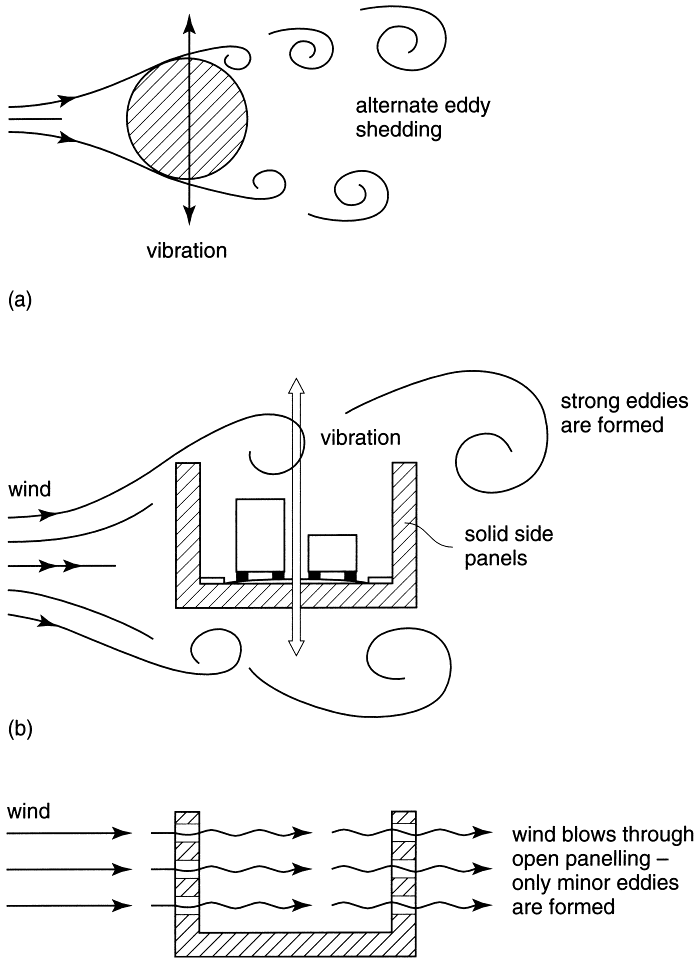
where ρ is fluid density (kg/m^3), a is the cross-sectional area (m^2), v is velocity (m/s) and C is drag coefficient. The coefficient C is dependent on the shape of the body, the velocity of the flow and the density of the fluid.

3.10.1 Stopping super tankers

Super tankers are large ships which are designed for low drag so that they can travel the seas with only modest energy requirements to drive them. The problem comes when they want to stop. When the engines stop they can travel for several kilometres before drag forces finally stop them. How then do you put the brakes on a super tanker? One way is to increase the form drag by taking advantage of the stagnation point at the bow of the ship to push water through an inlet pipe in the bow and out at the sides of the ship (Figure 3.17d). This flow at right angles to the movement of the ship, causes the boundary layer to separate and greatly increase the form drag. It is as if the ship is suddenly made much wider and this upsets its streamline shape.

3.11 Eddy shedding

Eddies which form in the wake around bridge piers can also cause other problems besides drag. Eddies are not shed from each side of the pier at the same time but alternately, first from one side, then from the other. Under the right flow conditions large eddies can form and the alternate



3.18 Eddy shedding problems.

eddy-shedding can induce a lateral force which can push an object back and forth causing a slow rhythmic vibration (Figure 3.18a). This problem is not just confined to bridge piers. It can happen to tall chimneys and to bridge decks in windy conditions. The vibration can become so bad that structures collapse.

A famous suspension bridge, the Tacoma Narrows Bridge in the USA, was destroyed in the 1930s because of this problem (Figure 3.18b). In order to protect traffic from high winds blowing down the river channel, the sides of the bridge were boarded up. Unfortunately the boarding deflected the wind around the bridge deck, the air flow separated forming large eddies and this set the bridge deck oscillating violently up and down. The bridge deck was quite flexible as it was a suspension bridge and could in fact tolerate quite a lot of movement but this was so violent that eventually it destroyed the bridge.

The solution to the problem was quite simple, but it was not appreciated at the time. If the side panels had been removed this would have stopped the large eddies from forming and there would have been no vibration. So next time you are on a suspension bridge and a strong wind is blowing and you are feeling uncomfortable be thankful that the engineers have decided not to protect you from the wind by boarding up the sides.

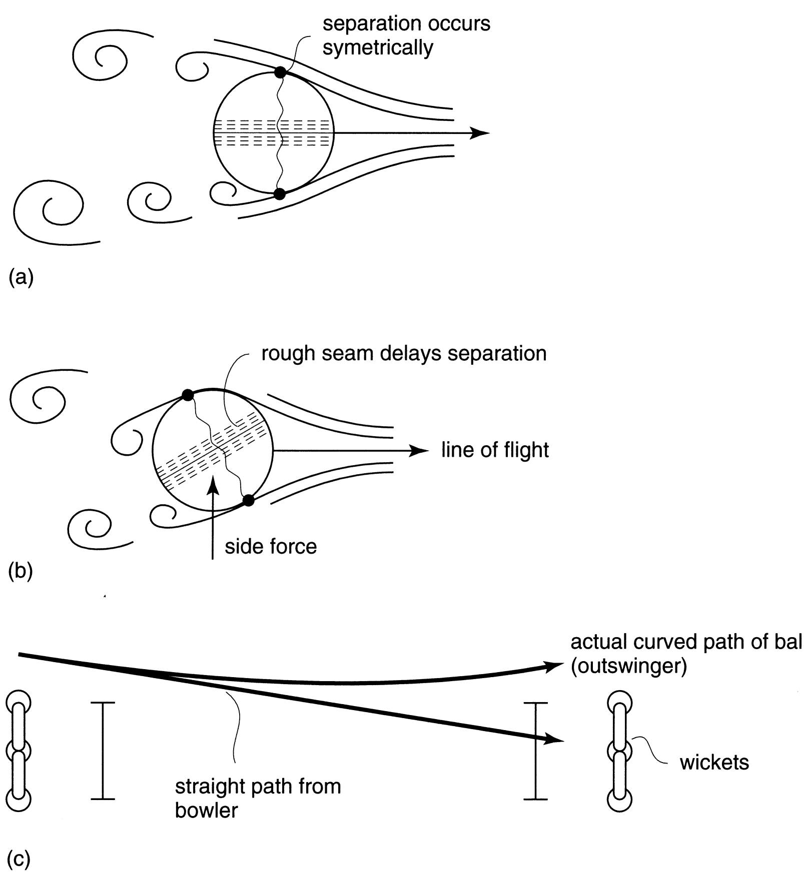
A similar problem can occur around tall chimneys when eddies are shed in windy conditions. To avoid large eddies forming a perforated sleeve or a spiral collar is placed around the top of the chimney. This breaks up the flow into lots of small eddies which are usually quite harmless.

3.12 Making balls swing

Sports players soon learn how useful boundary layers can be when they realise that balls can be made to move in a curved path and so confuse their opponents. A good example of this is the way some bowlers are able to make a ball 'swing' (move in curved path) in cricket.

When a ball is thrown (for cricket enthusiasts this means bowled), the air flows around it and at some point it separates (Figure 3.19). When the separation occurs at the same point all around the ball then it moves along a straight path. However, when it occurs asymmetrically there is a larger pressure on one side of the ball and so it starts to move in a curved path (i.e. it swings). The bowler's task is to work out how to do this.

Laboratory experiments have shown that as air flows around a ball it can be either laminar or turbulent (these are two different kinds of flow that are described in Section 4.3.1). When it is turbulent the air clings to the ball more easily than when it is laminar. So the bowler tries to make the air flow turbulent on one side of the ball and laminar on the other. This is done by making one side very smooth and the other side very rough. In cricket, this situation is helped by a special



3.19 Making balls swing.

stitched seam around the middle of the ball which ensures that the ball is rough enough to create turbulent conditions. The ball is bowled so that the polished side of the ball is facing the batsman and the seam is at an angle to the main direction of travel. The air flow on the smooth side separates earlier than on the rough side and so the ball swings towards the turbulent side. The cross force can be up to 0.8 N depending on how fast the ball is travelling and may cause a swing of up to 0.6 m in 20 m – the length of a cricket pitch. This can be more than enough to seriously upset a batsman who may be expecting a straight ball. This is why bowlers seem to spend so much of their time polishing the ball on their trousers prior to their run up in order to get it as smooth as possible to get the maximum swing. The swing may be in or out depending on how the bowler holds the ball. However, not all the surprise is with the bowler. An observant batsman may know what is coming by looking to see how the bowler is holding the ball and so anticipate the swing.

Sometimes strange things happen which even puzzle those who understand hydraulics. Just occasionally bowlers have noticed that a ball that was meant to swing in towards the batsman swings away from him instead – an *outswinger*. What happens is that when a ball is bowled fast enough the entire air flow around the ball turns turbulent and so the separation occurs much earlier. The stitched seam around the ball now acts as a ramp causing the air to be pushed away, creating a side force in the opposite direction to what was expected. This causes great delight for the bowler but it can give the batsman quite a fright. But most batsmen can relax as this special swing only occurs when the ball reaches 130 to 150 km/h and only a few bowlers can actually reach this velocity. However, some unscrupulous bowlers have discovered a way of doing this at much lower velocities. By deliberately roughening the ball on one side (which is not allowed) and polishing it on the other (which is allowed) they can bowl an outswinger at much lower velocities. This caused a major row in cricket in the early 1990s and again in 2006 when a Pakistani bowler was accused of deliberately roughening the ball. Imran Khan though was famous for his high speed bowling and could produce outswingers without resorting to such tactics. It is of course allowed for the ball to scuff or become rough naturally through play but this can take some time.

Causing a ball to spin at the same time as driving it forward can also add to the complexities of air flow and also to the excitement of ball sports. Some famous ball swings in recent years resulted in the goals scored by the Brazilian footballer, Roberto Carlos, in 1997 and by David Beckham in the 2006 World Cup games. In each case the goal area was completely blocked by the opposing team players. As each player kicked the ball it seemed to be heading for the corner flag but instead it followed a curved path around the defending players and into the goal. They achieved this amazing feat by striking the ball on its edge causing it to spin, which induced a sideways force. This, together with the boundary layer effect and a great deal of skill (and a little luck) produced some of the best goals ever scored.

3.13 Successful stone-skipping

Skipping stones across water has been a popular pastime for many thousands of years. Apparently the Greeks started it and according to the Guiness Book of Records the world record is held by Kurt Steiner. It was set in 2003 at 40 rebounds.

Various parameters affect the number of skips such as the weight and shape of the stone, ideally this should be flat plate shape, and the velocity and spin. This was very much an art until research undertaken at the Institutut de Researche sur les Phenomenones Hors Equilibre in Marseille and published in *Nature* in 2004 investigated the hydraulics of this past time. Among the many parameters investigated the most important one was the angle at which the stone hits the water. The 'magic' angle, as the researchers described it, was 20°and at this angle the energy dissipated by the stone impact with the water is minimised. So at this angle you will achieve the

3.20 Preparing for the record stone skip.

maximum number of skips. Spinning the stone also helps because this stabilises the stone owing to the gyroscopic effect. You may not reach 40 skips but at this angle you have the best chance.

Interestingly, stone skipping is often thought to provide a theory for lift on an aircraft wing. Instead of water hitting the underside of a stone and lifting it the idea of air hitting the underside of an aerofoil has some appeal. But it is quite wrong – the lift comes from the energy changes which take place around the wing as described in Section 3.7.2.

3.14 Some examples to test your understanding

1 A 10 litre bucket is used to catch and measure the flow discharging from a pipeline. If it takes 3.5 s to fill the bucket, calculate the discharge in m^3/s. Calculate the velocity in the pipe when the diameter is 100 mm (0.0028 m^3/s; 0.35 m/s).

2 A main pipeline 300 mm in diameter carries a discharge of 0.16 m^3/s and a smaller pipe of diameter 100 mm is joined to it to form a tee junction and takes 0.04 m^3/s. Calculate the velocity in the 100 mm pipe and the discharge and velocity in the main pipe downstream of the junction (5.09 m/s; 0.12 m^3/s; 1.7 m/s).

3 A fountain is to be designed for a local park. A nozzle diameter of 50 mm is chosen and the water velocity at the nozzle will be 8.5 m/s. Calculate the height to which the water will rise. The jet passes through a circular opening 2 m above the nozzle. Calculate the diameter of the opening so that the jet just passes through without interference (3.68 m; opening greater than 61 mm).

4 A pipeline 500 mm diameter carrying a discharge of 0.5 m^3/s at a pressure of 55 kN/m^2 reduces to 300 mm diameter. Calculate the velocity and pressure in the 300 mm pipe (7.14 m/s; 33 kN/m^2).