PRACTICAL HYDRAULICS

SECOND EDITION

SPON TEXT

MELVYN KAY

Practical Hydraulics

Also available from Taylor & Francis

Hydraulic Structures 4th edition
P. Novak *et al.*
Hb: ISBN 9780415386258
Pb: ISBN 9780415386265

Hydraulics in Civil and Environmental Engineering 4th edition
A. Chadwick *et al*.
Hb: ISBN 9780415306089
Pb: ISBN 9780415306096

Mechanics of Fluids 8th edition
B. Massey and J. Ward Smith
Hb: ISBN 9780415362054
Pb: ISBN 9780415362061

Hydraulic Canals
J. Liria
Hb: ISBN 9780415362115

Practical Hydraulics

Second edition

Melvyn Kay

Taylor & Francis
Taylor & Francis Group
LONDON AND NEW YORK

First edition published 1998 by E&FN Spon, an imprint of Routledge

This edition published 2008
by Taylor & Francis
2 Park Square, Milton Park, Abingdon, Oxon OX14 4RN

Simultaneously published in the USA and Canada
by Taylor & Francis
270 Madison Ave, New York, NY 10016

This edition published in the Taylor & Francis e-Library, 2007.

"To purchase your own copy of this or any of Taylor & Francis or Routledge's collection of thousands of eBooks please go to www.eBookstore.tandf.co.uk."

Taylor & Francis is an imprint of the Taylor & Francis Group, an informa business

British Library Cataloguing in Publication Data
A catalogue record for this book is available from the British Library

Library of Congress Cataloging in Publication Data
Kay, Melvyn.
Practical hydraulics / Melvyn Kay. – 2nd ed.
p. cm.
Includes bibliographical references and index.
1. Hydraulics. I. Title.

TC160.K38 2007
620.1'06–dc22 2007012472

ISBN 0-203-96077-7 Master e-book ISBN

ISBN10: 0–415–35114–6 (hbk)
ISBN10: 0–415–35115–4 (pbk)
ISBN10: 0–203–96077–7 (ebk)

ISBN13: 978–0–415–35114–0 (hbk)
ISBN13: 978–0–415–35115–7 (pbk)
ISBN13: 978–0–203–96077–6 (ebk)

Contents

Preface

Who wants to know about hydraulics? Well, my six-year-old daughter for a start. She wants to know why water swirls as it goes down the plug hole when she has a bath and why it always seems to go in the same direction. Many people in various walks of life have to deal with water – engineers who design and build our domestic water supply systems and hydro-electric dams, environmental scientists concerned about our natural rivers and wet lands, farmers who irrigate their crops and fire crews using pumps and high pressure hoses to put out fires. They want to store it, pump it, spray it or just move it from one place to another in pipes or channels. Whatever their requirements, they all need an understanding of how water behaves and how to deal with it. This is the study of hydraulics.

But hydraulics is not just about water. Many other fluids behave like water and affect a wide range of people. Doctors need to understand about the heart as a pump and how blood flows in arteries and veins that are just like small pipelines. Aircraft designers must understand how air flowing around an aircraft wing can create lift. Car designers want to know how air flows around cars in order to improve road holding and reduce wind drag to save fuel. Sportsmen too soon learn that a ball can be made to move in a curved path by changing its velocity and the air flow around it and so confuse an opponent.

There are many misconceptions and misunderstandings about water and few people have any real idea about how it behaves. We all live in a 'solid' world and so we naturally think that water behaves in much the same way as everything else around us. But this assumption can lead to all kinds of problems, some of them amusing, but some more serious and some even fatal. The fact that water does not always do what people expect it to do is what makes hydraulics such a fascinating subject – it has kept me busy all my working life.

As a lecturer I found that many students were afraid of hydraulics because of its reputation for being too mathematical or too complicated. Most hydraulics text books do little to allay such fears as they are usually written by engineers for engineers and assume that the reader has a degree in mathematics. So in writing this book I have attempted to overcome these misconceptions and to show that hydraulics is really easy to understand and a subject to enjoy rather than fear. You do not have to be an engineer or a mathematician to understand hydraulics. Water is all around us and is an important part of our everyday lives. Just go straight to Chapter 9 to see how much you can learn about water simply by having a bath!

But bathtubs apart, hydraulics can explain many other everyday things – how aeroplanes fly, why the wind rushes in the gaps between buildings and doors start banging, why some tall chimneys and bridges collapse when the wind blows around them, why it takes two firemen to hold down a small hose-pipe when fighting a fire, why there is a violent banging noise in water pipes when you turn a tap off quickly, how competition swimmers can increase their speed in the water by changing their swim suit and why tea leaves always go to the centre of the cup when you stir your tea!

But there is a more serious side to hydraulics. It can be about building a storage reservoir, selecting the right size of pipes and pumps to supply domestic water to a town, controlling water levels in wetland habitats or choosing the right size channel to supply farms with irrigation water or solve a drainage problem.

It would have been easier to write a 'simple', descriptive book on hydraulics by omitting the more complex ideas of water flow but this would have been simplicity at the expense of reality. It would be like writing a cookbook with recipes rather than examining why certain things happen when ingredients are mixed together. So I have tried to cater for a range of tastes. At one level this book is descriptive and provides a qualitative understanding of hydraulics. At another level it is more rigorous and quantitative. These are more mathematical bits for those who wish to go that extra step. It was the physicist Lord Kelvin (1824–1907) who said that it is essential to put numbers on things if we are really going to understand them. So if you are curious about solving problems I have included a number of worked examples, as well as some of the more interesting formula derivations and put them into boxes in the text so that you can spot them easily, and avoid them if you wish.

Be aware that understanding hydraulics and solving problems mathematically are two different skills. Many people achieve a good understanding of water behaviour but then get frustrated because they cannot easily apply the maths. This is a common problem and in my experience as a teacher it is a skill that can only be acquired through lots of practice – hence the reason why I have included many worked examples in the text. I have also included a list of problems at the end of each chapter for you to try out your new skills. It does help to have *some* mathematical skills – basic algebra should be enough to get you started.

This is the second edition of *Practical Hydraulics*. In response to those who have read and used the first edition I have added in many new 'stories' to help readers to better understand hydraulics and more worked examples, particularly on pumps and pipelines. I have also included an additional chapter on 'bathtub' hydraulics which I hope you will find both enjoyable and useful – bath-time will never be the same again.

So enjoy learning about hydraulics!

Melvyn Kay
October 2007

Acknowledgements

I would like to make special mention of two books which have greatly influenced my writing of this text. The first is *Water in the Service of Man* by H.R. Vallentine, published by Pelican Books Ltd in 1967. The second is *Fluid Mechanics for Civil Engineers* by N.B. Webber first published in 1965 by E & FN Spon Ltd. Unfortunately both are now out of print but copies can still be found via Amazon.

I would like to acknowledge my use of the method described in *Handbook of Hydraulics for the Solution of Hydrostatic and Fluid Flow Problems* by H.W. King and E.F. Brater published in 1963 for the design of channels using Manning's equation (Section 5.8.4).

I am also grateful for ideas I obtained from *The Economist* on the use of boundary drag on swim suits (Section 3.10) and from *New Scientist* on momentum transfer (Section 1.12) and the hydrodynamics of cricket balls (Section 3.12).

I would like to thank the following people and organisations for permission to use photographs and diagrams:

Chadwick, A. and Morfett, J. (1998) *Hydraulics in Civil and Environmental Engineering*. 3rd edition E & FN Spon, London for Figure 5.24.

FC Concrete Ltd, Derby UK for Figure 7.14.

Fox, J. (1977) *An Introduction to Engineering Fluid Mechanics*. The MacMillan Press Ltd London for Figure 5.28.

Fraenkel, P.L. (1986) *Water Lifting Devices.* Irrigation and Drainage Paper No.43 Food and Agriculture Organisation, Rome for Figures 8.5a and b, 8.11 and 8.21.

Hydraulics Research Wallingford (1983) *Charts for the Hydraulic Design of Channels and Pipes*. 5th edition for Figure 4.8.

IPTRID-FAO (2000) *Treadle pumps for irrigation in Africa*. Knowledge synthesis paper No. 1 for Figure 8.3b.

ITT Lowara Pumps Ltd for Figure 8.19.

Marine Current Turbines TM Ltd for use of Figure 6.8.

Open University Oceanography COURIS Team (1995) *Waves, Tides and Shallow Water Processes*. Butterworth and Heineman 1995, for Figures 6.2 and 6.6.

Pdphoto for the use of Figure 6.1c.

Photographer Rene Kragelund for Figure 6.7.

Photographer Tom Brabben for Figure 8.3b.

The Environment Agency, UK for Figure 6.3.
Vallentine, H.R. (1967) *Water in the Service of Man.* Penguin Books Ltd, Harmondsworth, UK for Figures 2.7, 8.2a,b and c.
US Navy photo by Ensign John Gay for Figure 5.15c.
Webber, N.B. (1971) *Fluid Mechanics for Civil Engineers*. E & FN Spon Ltd, London for Figures 8.10b and 8.21c.

1 Some basic mechanics

1.1 Introduction

This is a reference chapter rather than one for general reading. It is useful as a reminder about the physical properties of water and for those who want to re-visit some basic physics which is directly relevant to the behaviour of water.

1.2 Units and dimensions

To understand hydraulics properly it is essential to be able to put numerical values on such things as pressure, velocity and discharge in order for them to have meaning. It is not enough to say the pressure is high or the discharge is large; some specific value needs to be given to quantify it. Also, just providing a number is quite meaningless. To say a pipeline is 6 long is not enough. It might be 6 centimetres, 6 metres or 6 kilometres. So the numbers must have dimensions to give them some useful meaning.

Different units of measurement are used in different parts of the world. The foot, pounds and second system (known as fps) is still used extensively in the USA and to some extent in the UK. The metric system, which relies on centimetres, grammes and seconds (known as cgs), is widely used in continental Europe. But in engineering and hydraulics the most common units are those in the SI system and it is this system which is used throughout this book.

1.2.1 SI units

The Systeme International d'Unites, usually abbreviated to SI, is not difficult to grasp and it has many advantages over the other systems. It is based on metric measurement and is slowly replacing the old fps system and the European cgs system. All length measurements are in metres, mass is in kilograms and time is in seconds (Table 1.1). SI units are simple to use and their big advantage is they can help to avoid much of the confusion which surrounds the use of other units. For example, it is quite easy to confuse mass and weight in both fps and cgs units as they are both measured in pounds in fps and in kilograms in cgs. Any mix-up between them can have serious consequences for the design of engineering works. In the SI system the difference is clear because they have different dimensions – mass is in kilograms whereas weight is in Newtons. This is discussed later in Section 1.7.

Table 1.1 Basic SI units of measurement.

Measurement	*Unit*	*Symbol*
Length	Metre	m
Mass	Kilogram	kg
Time	Second	s

Table 1.2 Some useful derived units.

Measurement	*Dimension*	*Measurement*	*Dimension*
Area	m^2	Force	N
Volume	m^3	Mass density	kg/m^3
Velocity	m/s	Specific weight	N/m^3
Acceleration	m/s^2	Pressure	N/m^2
Viscosity	kg/ms	Momentum	kgm/s
Kinematic viscosity	m^2/s	Energy for solids	Nm/N
		Energy for fluids	Nm/N

Note there is no mention of centimetres in Table 1.1. Centimetres are part of the cgs units and not SI and so play no part in hydraulics or in this text. Millimetres are acceptable for very small measurements and kilometres for long lengths – but *not* centimetres.

1.2.2 Dimensions

Every measurement must have a dimension so that it has meaning. The units chosen for measurement do not affect the quantities measured and so, for example, 1.0 metre is exactly the same as 3.28 feet. However, when solving problems, all the measurements used must be in the same system of units. If they are mixed up (e.g. centimetres or inches instead of metres, or minutes instead of seconds) and added together, the answer will be meaningless. Some useful dimensions which come from the SI system of units in Table 1.1 are included in Table 1.2.

1.3 Velocity and acceleration

In everyday language *velocity* is often used in place of *speed*. But they are different. Speed is the rate at which some object is travelling and is measured in metres/second (m/s) but there is no indication of the direction of travel. Velocity is speed plus direction. It defines movement in a particular direction and is also measured in metres/second (m/s). In hydraulics, it is useful to know which direction water is moving and so the term velocity is used instead of speed. When an object travels a known distance and the time taken to do this is also known, then the velocity can be calculated as follows:

$$\text{velocity (m/s)} = \frac{\text{distance (m)}}{\text{time (s)}}$$

Acceleration describes change in velocity. When an object's velocity is increasing then it is *accelerating*; when it is slowing down it is *decelerating.* Acceleration is measured in metres/second/

second (m/s^2). If the initial and final velocities are known as well as the time taken for the velocity to change then the acceleration can be calculated as follows:

$$\text{acceleration (m/s}^2\text{)} = \frac{\text{change in velocity (m/s)}}{\text{time (s)}}$$

EXAMPLE: CALCULATING VELOCITY AND ACCELERATION

An object is moving along at a steady velocity and it takes 150 s to travel 100 m. Calculate the velocity.

$$\text{velocity} = \frac{\text{distance (m)}}{\text{time (s)}} = \frac{100}{150} = 0.67 \text{ m/s}$$

If the object starts from rest, calculate the acceleration if its final velocity of 1.5 m/s is reached in 50 s:

$$\text{acceleration} = \frac{\text{change in velocity (m/s)}}{\text{time (s)}} = \frac{1.5-0}{50} = 0.03 \text{ m/s}^2$$

1.4 Forces

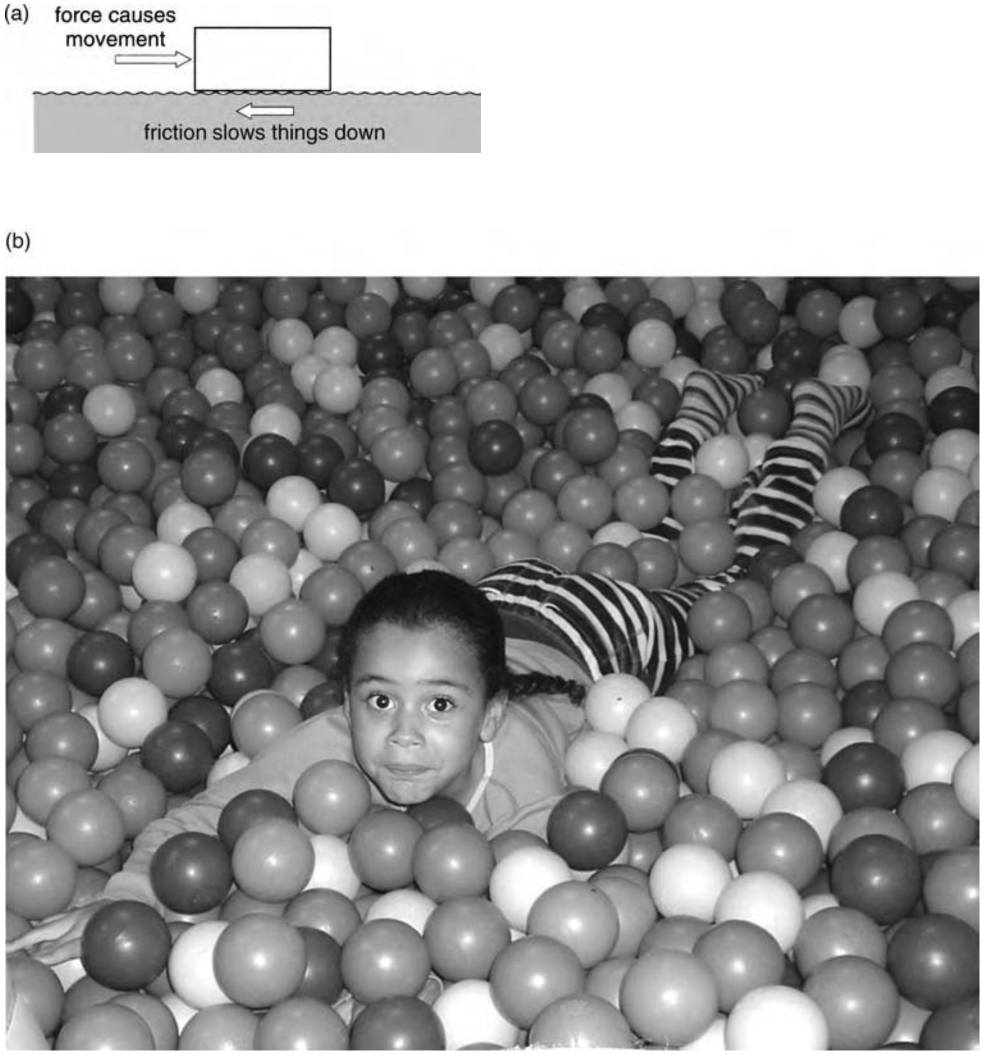
Force is not a word that can be easily described in the same way as some material object. It is commonly used and understood to mean a pushing or a pulling action and so it is only possible to say what a force will do and not what it is. Using this idea, if a force is applied to some stationary object then, if the force is large enough, the object will begin to move. If the force is applied for long enough then the object will begin to move faster, that is, it will accelerate. The same applies to water and to other fluids as well. It may be difficult to think of pushing water, but, if it is to flow along a pipeline or a channel, a force will be needed to move it. So one way of describing force is to say that *a force causes movement.*

1.5 Friction

Friction is the name given to the force which resists movement and so causes objects to slow down (Figure 1.1a). It is an important aspect of all our daily lives. Without friction between our feet and the ground surface it would be difficult to walk and we are reminded of this each time we step onto ice or some smooth oily surface. We would not be able to swim if water was frictionless. Our arms would just slide through the water and we would not make any headway – just like children trying to 'swim' in a sea of plastic balls in the playground (Figure 1.1b).

Friction is an essential part of our existence but sometimes it can be a nuisance. In car engines, for example, friction between the moving parts would cause them to quickly heat up and the engine would seize up. But oil lubricates the surfaces and reduces the friction.

Friction also occurs in pipes and channels between flowing water and the internal surface of a pipe or the bed and sides of a channel. Indeed, much of pipe and channel hydraulics is concerned with predicting this friction force so that the right size of pipe or channel can be chosen to carry a given flow (see Chapter 4 Pipes and Chapter 5 Channels).



1.1 (a) Friction resists movement and (b) Trying to 'swim in a frictionless fluid'.

Friction is not only confined to boundaries, there is also friction inside fluids (internal friction) which makes some fluids flow more easily than others. The term *viscosity* is used to describe this internal friction (see Section 1.13.3).

1.6 Newton's laws of motion

Sir Isaac Newton (1642–1728) was one of the first to begin the study of forces and how they cause movement. His work is now enshrined in three basic rules known as *Newton's laws of motion.* They are very simple laws and at first sight they appear so obvious, they seem hardly worth writing down. But they form the basis of all our understanding of hydraulics (and movement of solid objects as well) and it took the genius of Newton to recognise their importance.

Law 1: forces cause movement

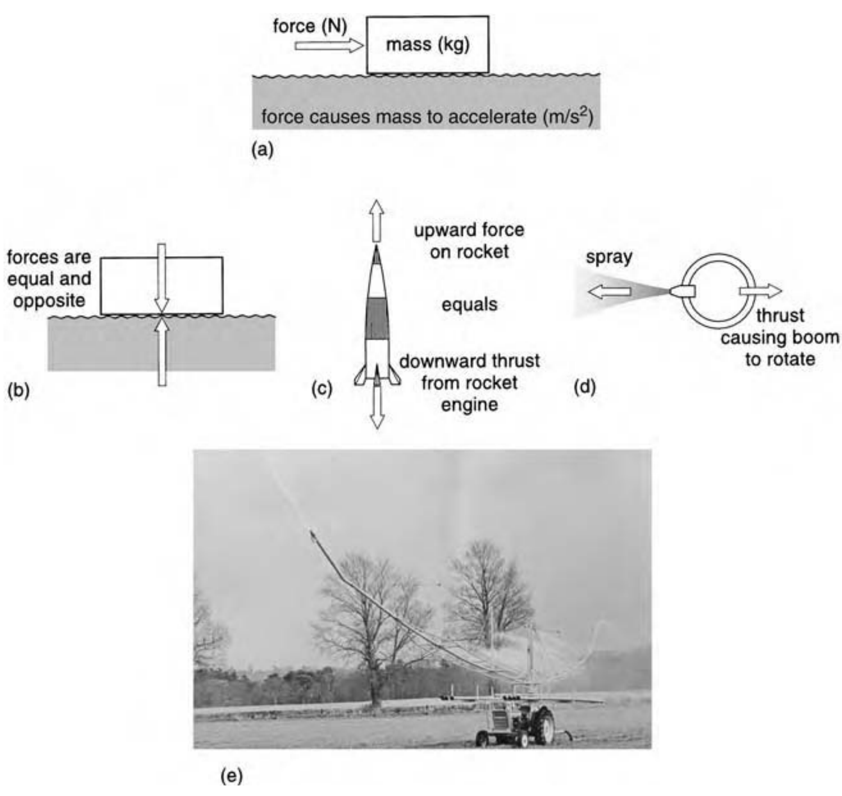
First imagine this in terms of solid objects. A block of wood placed on a table will stay there unless it is pushed (i.e. a force is applied to it). Equally, if it is moving, it will continue to move unless some force (e.g. friction) causes it to slow down or to change direction. So forces are needed to make objects move or to stop them. This same law also applies to water.

Law 2: forces cause objects to accelerate

This law builds on the first and provides the link between force, mass and acceleration (Figure 1.2a). Again think in solid material terms first. If the block of wood is to move it will need a force to do it. The size of this force depends on the size of the block (its mass) and how fast it needs to go (its acceleration). The larger the block and the faster it must go, the larger must be the force. Water behaves in the same way. If water is to be moved along a pipeline then some force will be needed to do it. Newton linked these three together in mathematical terms to calculate the force required:

force (N) = mass (kg) $\times$ acceleration (m/s^2)

The dimension of force can be derived from multiplying mass and acceleration, that is, kgm/s^2. But this is a complicated dimension and so in the SI system it is simplified and called the



1.2 Newton's laws of motion.

Newton (N) in recognition of Sir Isaac Newton's contribution to our understanding of mechanics. A force of 1 Newton is defined as the force needed to cause a mass of 1 kg to accelerate at 1 m/s^2. This is not a large force. An apple held in the palm of your hand weighs approximately 1 Newton – an interesting point, since it was supposed to have been an apple falling on Newton's head, which set off his thoughts on forces, gravity and motion.

Using Newtons in hydraulics will produce some very large numbers and so to overcome this, forces are measured in kilo Newtons (kN).

1 kN = 1000 N

EXAMPLE: CALCULATING FORCE USING NEWTON'S SECOND LAW

A mass of 3 kg is to be moved from rest to reach a speed of 6 m/s and this must be done in 10 s. Calculate the force needed.
First calculate acceleration:

$$\text{acceleration (m/s}^2) = \frac{\text{change in velocity (m/s)}}{\text{time (s)}}$$

$$\text{acceleration} = \frac{6}{10} = 0.6 \text{ m/s}^2$$

Using Newton's second law:

$$\text{force} = \text{mass} \times \text{acceleration}$$
$$\text{force} = 3 \times 0.6 = 1.8 \text{ N}$$

So a force of 1.8 N is needed to move a mass of 3 kg to a velocity of 6 m/s in 10 s.

Law 3: for every force there is always an equal and opposite force

To understand this simple but vitally important law, again think of the block of wood sitting on a table (Figure 1.2b). The block exerts a force (its weight) downwards on the table; but the table also exerts an equal but opposite upward force on the block. If it did not, the block would drop down through the table under the influence of gravity. So there are two forces, exactly equal in magnitude but in opposite directions and so the block does not move.

The same idea can be applied to moving objects as well. In earlier times it was thought that objects were propelled forward by the air rushing in behind them. This idea was put forward by the Greeks but it failed to show how an object could be propelled in a vacuum as is the case when a rocket travels into space. What in fact happens is the downward thrust of the burning fuel creates an equal and opposite thrust which pushes the rocket upwards (Figure 1.2c). Newton helped to discredit the Greek idea by setting up an experiment which showed that rather than encourage an object to move faster, the air (or water) flow around it slowed it down because of the friction between the object and the air.

Another example of Newton's third law occurs in irrigation where rotating booms spray water over crops (Figure 1.2d). The booms are not powered by a motor but by the reaction of the water jets. As water is forced out of the nozzles along the boom it creates an equal and opposite force on the boom itself which causes it to rotate. The same principle is used to drive the water distrubutors on the circular water-cleaning filters at the sewage works.

1.7 Mass and weight

There is often confusion between mass and weight and this has not been helped by the system of units used in the past. It is also not helped by our common use of the terms in everyday language. Mass and weight have very specific scientific meanings and for any study of water it is essential to have a clear understanding of the difference between them.

Mass refers to an amount of matter or material. It is a constant value and is measured in kilograms (kg). A specific quantity of matter is often referred to as an *object*. Hence the use of this term in the earlier description of Newton's laws.

Weight is a force. Weight is a measure of the force of gravity on an object and this will be different from place to place depending on the gravity. On the earth there are only slight variations in gravity, but the gravity on the moon is much less than it is on the earth. So the mass of an object on the moon would be the same as it is on the earth but its weight would be much less. As weight is a force, it is measured in Newtons. This clearly distinguishes it from mass which is measured in kilograms.

Newton's second law also links mass and weight and in this case the acceleration term is the acceleration resulting from gravity. This is the acceleration that any object experiences when dropped and allowed to fall to the earth's surface. Objects dropped in the atmosphere do, in fact, experience different rates of acceleration because of the resistance of the air – hence the reason why a feather falls more slowly than a coin. But if both were dropped at the same time in a vacuum they would fall (accelerate) at the same rate. There are also minor variations over the earth's surface and this is the reason why athletes can sometimes run faster or throw the javelin further in some parts of the world. However, for engineering purposes, acceleration due to gravity is assumed to have a constant value of 9.81 m/s^2 – usually called the *gravity constant* and denoted by the letter *g*. The following equation based on Newton's second law provides the link between weight and mass:

$$\text{weight (N)} = \text{mass (kg)} \times \text{gravity constant (m/s}^2\text{)}$$

EXAMPLE: CALCULATING THE WEIGHT OF AN OBJECT

Calculate the weight of an object when its mass is 5 kg.
Using Newton's second law:

$$\text{weight} = \text{mass} \times \text{gravity constant}$$
$$\text{weight} = 5 \times 9.81 = 49.05 \text{ N}$$

Sometimes engineers assume that the gravity constant is 10 m/s^2 because it is easier to multiply by 10 and the error involved in this is not significant in engineering terms.
In this case:

$$\text{weight} = 5 \times 10 = 50 \text{ N}$$

Confusion between mass and weight occurs in our everyday lives. When visiting a shop and asking for 5 kg of potatoes these are duly weighed out on a weigh balance. To be strictly correct we should ask for 50 N of potatoes, as the balance is measuring the *weight* of the potatoes (i.e. the force of gravity) and not their mass. But because gravity acceleration is constant all over the world (or nearly so for most engineering purposes) the conversion factor between mass and weight is a constant value. So the shopkeeper's balance will most likely show kilograms and not Newtons. If shopkeepers were to change their balances to read in Newtons to resolve a scientific confusion, engineers and scientists might be happy but no doubt a lot of shoppers would not be so happy!

1.8 Scalar and vector quantities

Measurements in hydraulics are either called *scalar* or *vector* quantities. Scalar measurements only indicate magnitude. Examples of this are mass, volume, area and length. So if there are 120 boxes in a room and they each have a volume of 2 m^3 both the number of boxes and the volume of each are scalar quantities.

Vectors have direction as well as magnitude. Examples of vectors include force and velocity. It is just as important to know which direction forces are pushing and water is moving as well as their magnitude.

1.9 Dealing with vectors

Scalar quantities can be added together by following the rules of arithmetic. Thus, 5 boxes and 4 boxes can be added to make 9 boxes and 3 m and 7 m can be added to make 10 m.

Vectors can also be added together provided their direction is taken into account. The addition (or subtraction) of two or more vectors results in another single vector called the *resultant* and the vectors that make up the resultant are called the *components*. If two forces, 25 N and 15 N, are pushing in the same direction then their resultant is found simply by adding the two together, that is, 40 N (Figure 1.3a). If they are pushing in opposite directions then their resultant is found by subtracting them, that is, 10 N. So one direction is considered positive and the opposite direction negative for the purposes of combining vectors.

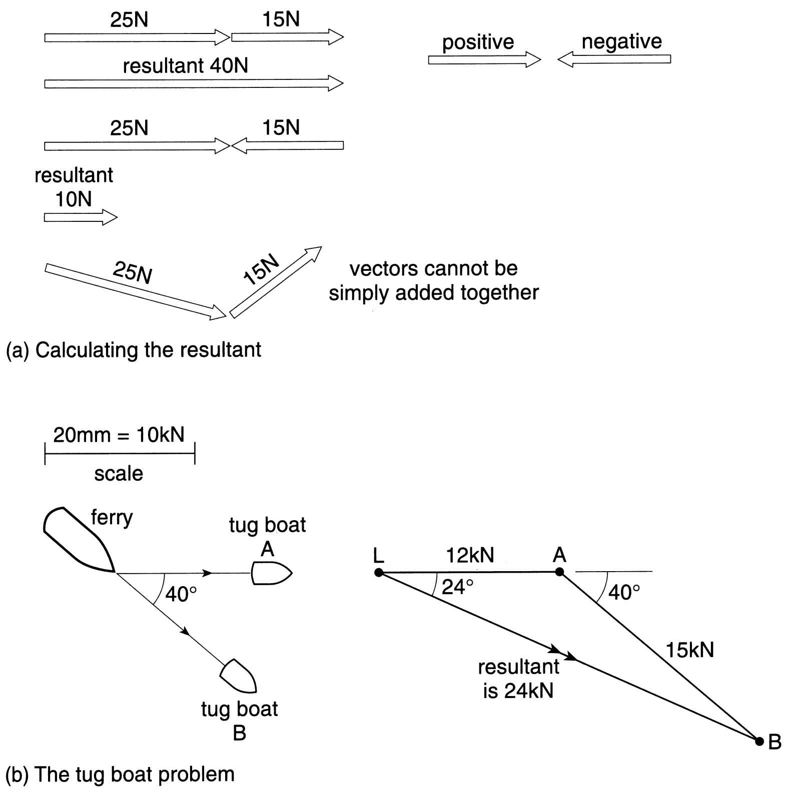
But forces can also be at an angle to each other and in such cases a different way of adding or subtracting them is needed – a *vector diagram* is used for this purpose. This is a diagram drawn to a chosen scale to show both the magnitude and the direction of the vectors and hence the magnitude of the resultant vector. An example of how this is done is shown in the box.

Vectors can also be added and subtracted mathematically but a knowledge of trigonometry is needed. For those interested in this approach, it is described in most basic books on maths and mechanics.

EXAMPLE: CALCULATING THE RESULTANT FORCE USING A VECTOR DIAGRAM

Two tug boats A and B are pulling a large ferry boat into a harbour. Tug A is pulling with a force of 12 kN, tug B with a force of 15 kN and the angle between the two tow ropes is 40° (Figure 1.3b). Calculate the resultant force and show the direction in which the ferry boat will move.

First draw a diagram of the ferry and the two tugs. Then, assuming a scale of 40 mm equals 10 kN (this is chosen so that the diagram fits conveniently onto a sheet of paper)



1.3 Adding and subtracting vectors.

draw the 12 kN force to scale, that is, the line LA. Next, draw the second force, 15 kN, to the same scale but starting the line at A and drawing it at an angle of 40° to the first line. This 'adds' the second force to the first one. The resultant force is found by joining the points L and B, measuring this in mm and converting this to a value in kN using the scale. Its value is 24 kN. The line of the resultant is shown by the positioning of the line LB in the diagram.

To summarise, the ferry boat will move in a direction LB as a result of the pull exerted by the two tugs and the resultant force pulling on the ferry in that direction is 24 kN. The triangle drawn in Figure 1.3b is the *vector diagram* and shows how two forces can be added. As there are three forces in this problem it is sometimes called a *triangle of forces.* It is possible to add together many forces using the same technique. In such cases the diagram is referred to as a *polygon of forces.*

1.10 Work, energy and power

Work, energy and power are all words commonly used in everyday language, but in engineering and hydraulics they have very specific meanings and so it is important to clarify what each means.

1.10.1 Work

Work refers to almost any kind of physical activity but in engineering it has a very specific meaning. Work is done when a force produces movement. A crane does work when it lifts a load against the force of gravity and a train does work when it pulls trucks. But if you hold a large weight for a long period of time you will undoubtedly get very tired and feel that you have done a lot of work but you will not have done any work at all in an engineering sense because nothing moved.

Work done on an object can be calculated as follows:

work done (Nm) = force (N) × distance moved by the object (m)

Work done is the product of force (N) and distance (m) so it is measured in Newton-metres (Nm).

1.10.2 Energy

Energy enables useful work to be done. People and animals require energy to do work. They get this by eating food and converting it into useful energy for work through the muscles of the body. Energy is also needed to make water flow and this is why reservoirs are built in mountainous areas so that the natural energy of water can be used to make it flow downhill to a town or to a hydro-electric power station. In many cases energy must be added to water to lift it from a well or a river. This can be supplied by a pumping device driven by a motor using energy from fossil fuels such as diesel or petrol. Solar and wind energy are alternatives and so is energy provided by human hands or animals.

The amount of energy needed to do a job is determined by the amount of work to be done. So that:

energy required = work done

so

energy required (Nm) = force (N) × distance (m)

Energy, like work, is measured in Newton-metres (Nm) but the more conventional measurement of energy is *watt-seconds* (Ws) where:

1 Ws = 1 Nm

But this is a very small quantity for engineers to use and so rather than calculate energy in large numbers of Newton-metres or watt-seconds they prefer to use *watt-hours* (Wh) or *kilowatt-hours* (kWh). So multiply both sides of this equation by 3600 to change seconds to hours:

1 Wh = 3600 Nm

Now multiply both sides by 1000 to change watts-hours to kilowatt-hours (Wh to kWh):

1 kWh = 3 600 000 Nm
= 3600 kNm

Just to add to the confusion some scientists measure energy in *joules* (J). This is in recognition of the contribution made by the English physicist, James Joule (1818–1889) to our understanding of energy, in particular, the conversion of mechanical energy to heat energy (see next section).

So for the record:

1 joule = 1 Nm

To avoid confusion the term joule is not used in this text. Some everyday examples of energy use include:

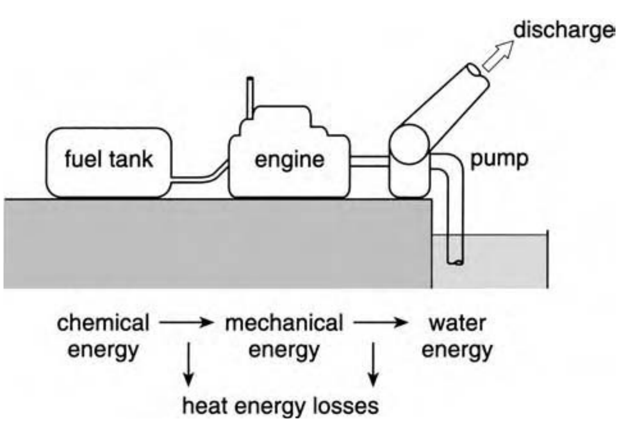
- A farmer working in the field uses 0.2–0.3 kWh every day.
- An electric desk fan uses 0.3 kWh every hour.
- An air-conditioner uses 1 kWh every hour.

Notice how it is important to specify the time period (e.g. every hour, every day) over which the energy is used. Energy used for pumping water is discussed more fully in Chapter 8.

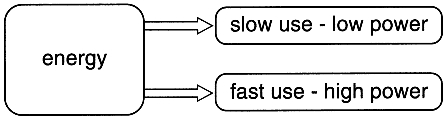
1.10.2.1 Changing energy

An important aspect of energy is that it can be changed from one form to another. People and animals are able to convert food into useful energy to drive their muscles. The farmer using 0.2 kWh every day, for example, must eat enough food each day to supply this energy need otherwise the farmer would not be able to work properly. In a typical diesel engine pumping system, the energy is changed several times before it gets to the water. Chemical energy contained within the fuel (e.g. diesel oil) is burnt in a diesel engine to produce mechanical energy. This is converted to useful water energy via the drive shaft and pump (Figure 1.4). So a pumping unit is both an energy converter as well as a device for adding energy into a water system.

The system of energy transfer is not perfect and energy losses occur through friction between the moving parts and is usually lost as heat energy. These losses can be significant and costly in terms of fuel use. For this reason it is important to match a pump and its power unit with the job to be done to maximise the efficiency of energy use (see Chapter 8).



1.4 Changing energy from one form to another.



1.5 Power is the rate of energy use.

1.10.3 Power

Power is often confused with the term energy. They are related but they have different meanings. Whilst energy is the capacity to do useful work, power is the rate at which the energy is used (Figure 1.5).
And so:

$$\text{power (kW)} = \frac{\text{energy (kWh)}}{\text{time (h)}}$$

Examples of power requirements, a typical room air-conditioner has a power rating of 3 kW. This means that it consumes 3 kWh of energy every hour it is working. A small electric radiator has a rating of 1–2 kW and the average person walking up and down stairs has a power requirement of about 70 W.

Energy requirements are sometimes calculated from knowing the equipment power rating and the time over which it is used rather than trying to calculate it from the work done. In this case:

$$\text{energy (kWh)} = \text{power (kW)} \times \text{time (h)}$$

Horse Power (HP) is still a very commonly used measure of power but it is not used in this book, as it is not an SI unit. However, for the record:

$$1 \text{ kW} = 1.36 \text{ HP}$$

Power used for pumping water is discussed more fully in Chapter 8.

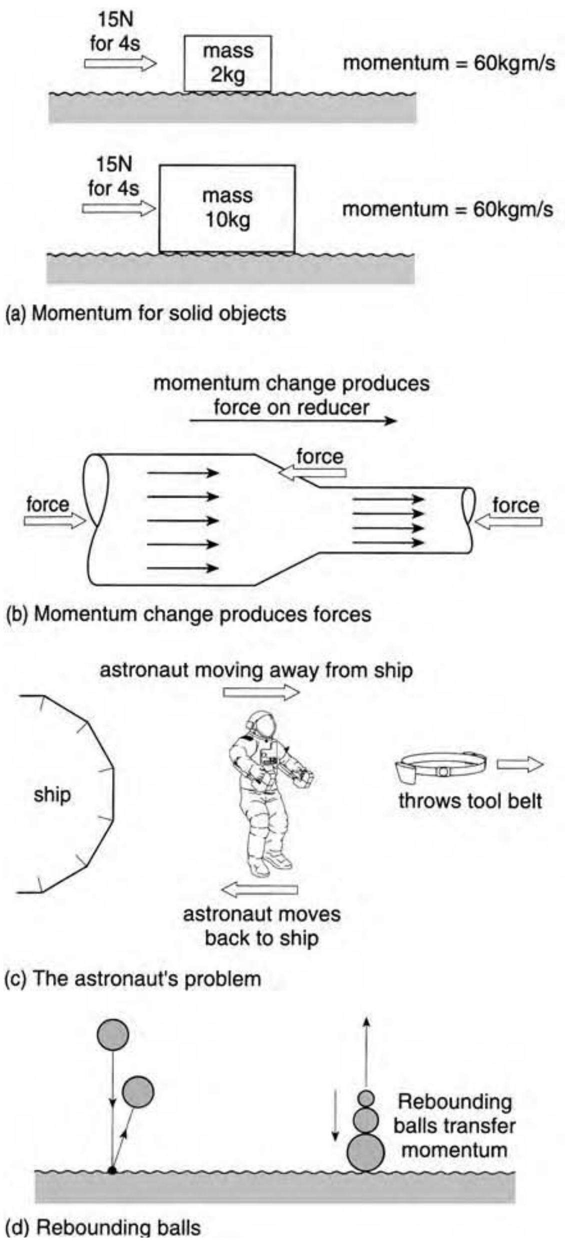
1.11 Momentum

Applying a force to a mass causes it to accelerate (Newton's second law) and the effect of this is to cause a change in velocity. This means there is a link between mass and velocity and this is called *momentum*. Momentum is another scientific term that is used in everyday language to describe something that is moving – we say that some object or a football game has momentum if it is moving along and making good progress. In engineering terms it has a specific meaning and it can be calculated by multiplying the mass and the velocity together:

$$\text{momentum (kgm/s)} = \text{mass (kg)} \times \text{velocity (m/s)}$$

Note the dimensions of momentum are a combination of those of velocity and mass.

The following example demonstrates the links between force, mass and velocity. Figure 1.6 shows two blocks that are to be pushed along by applying a force to them. Imagine that the sliding surface is very smooth and so there is no friction.



1.6 Understanding momentum.

The first block of mass 2 kg is pushed by a force of 15 N for 4 s. Using Newton's second law the acceleration and the resulting velocity after a period of 4 s can be calculated:

$$\text{force} = \text{mass} \times \text{acceleration}$$
$$15 = 2 \times f$$
$$f = 7.5 \text{ m/s}^2$$

So for every second the force is applied the block will move faster by 7.5 m/s. After 4 s it will have reached a velocity of:

$$4 \times 7.5 = 30 \text{ m/s}$$

Calculate the momentum of the block:

$$\begin{aligned} \text{momentum} &= \text{mass} \times \text{velocity} \\ &= 2 \times 30 \\ &= 60 \text{ kgm/s} \end{aligned}$$

Now hold this information for a moment. Suppose a larger block of mass 10 kg is pushed by the same force of 15 N for the same time of 4 s. Use the same calculations as before to calculate the acceleration and the velocity of the block after 4 s:

$$15 = 10 \times f$$

And so:

$$f = 1.5 \text{ m/s}^2$$

So when the same force is applied to this larger block it accelerates more slowly at 1.5 m/s for every second the force is applied. After 4 s it will have a velocity of:

$$4 \times 1.5 = 6 \text{ m/s}$$

Now calculate momentum of this block:

$$\begin{aligned} \text{momentum} &= 10 \times 6 \\ &= 60 \text{ kgm/s} \end{aligned}$$

Although the masses and the resulting accelerations are very different the momentum produced in each case when the same force is applied for the same time period is the same.

Now multiply the force by the time:

$$\begin{aligned} \text{force} \times \text{time} &= 15 \times 4 \\ &= 60 \text{ Ns} \end{aligned}$$

But the dimension for Newtons can also be written as kgm/s^2. And so:

$$\text{force} \times \text{time} = 60 \text{ kgm/s}$$

This is equal to the momentum and has the same dimensions. It is called the *impulse* and it is equal to the momentum it creates. So:

$$\text{impulse} = \text{momentum}$$

And:

$$\text{force} \times \text{time} = \text{mass} \times \text{velocity}$$

This is more commonly written as:

impulse = change of momentum

Writing 'change in momentum' is more appropriate because an object need not be starting from rest – it may already be moving. In such cases the object will have some momentum and an impulse would be increasing (changing) it. A momentum change need not be just a change in velocity but also a change in mass. If a lorry loses some of its load when travelling at speed its mass will change. In this case the lorry would gain speed as a result of being smaller in mass, the momentum before being equal to the momentum after the loss of load.

The equation for momentum change becomes:

force × time = mass × change in velocity

This equation works well for solid blocks which are forced to move but it is not easily applied to flowing water in its present form. For water it is better to look at the rate at which the water mass is flowing rather than thinking of the flow as a series of discrete solid blocks of water. This is done by dividing both sides of the equation by time:

$$\text{force} = \frac{\text{mass}}{\text{time}} \times \text{change in velocity}$$

Mass divided by time is the mass flow in kg/s and so the equation becomes:

force (N) = mass flow (kg/s) × change in velocity (m/s)

So when flowing water undergoes a change of momentum either by a change in velocity or a change in mass flow (e.g. water flowing around a pipe bend or through a reducer) then a force is produced by that change (Figure 1.6b). Equally if a force is applied to water (e.g. in a pump or turbine) then the water will experience a change in momentum.

As momentum is about forces and velocities the direction in which momentum changes is also important. In the simple force example, the forces are pushing from left to right and so the movement is from left to right. This is assumed to be the positive direction. Any force or movement from right to left would be considered negative. So if several forces are involved they can be added or subtracted to find a single resultant force. Another important point to note is that Newton's third law also applies to momentum. The force on the reducer (Figure 1.6b) could be drawn in either direction. In the diagram the force is shown in the negative direction (right to left) and this is the force that the reducer exerts on the water. Equally it could be drawn in the opposite direction, that is, the positive direction (left to right) when it would be the force of the water on the reducer. Either way the two forces are equal and opposite as Newton’s third law states.

The application of this idea to water flow is developed further in Section 4.1.3.

Those not so familiar with Newton's laws might find momentum more difficult to deal with than other aspects of hydraulics. To help understand the concept here are two interesting examples of momentum change which may help.

1.11.1 The astronaut's problem

An astronaut has just completed a repair job on his space ship and secures his tools on his belt. He then pushes off from the ship to drift in space only to find that his life-line has come undone and he is drifting further and further away from his ship (Figure 1.6c). How can he get back? He could radio for help, but another solution would be to take off his tool belt and throw it as hard as he can in the direction he is travelling. The reaction from this will be to propel him in the opposite direction and back to his space ship. The momentum created by throwing the tool belt in one direction (i.e. mass of tool belt multiplied by velocity of tool belt) will be matched by momentum in the opposite direction (i.e. mass of spaceman multiplied by velocity of spaceman). His mass will be much larger than the tool belt and so his velocity will be smaller but at least it will be in the right direction!

1.11.2 Rebounding balls

Another interesting example of momentum change occurs when several balls are dropped onto the ground together (Figure 1.6d). If dropped individually they rebound to a modest height – less than the height from which they were dropped. If several balls, each one slightly smaller than the previous one, are now dropped together, one on top of the other, the top one will shoot upwards at an alarming velocity to a height far greater than any of the individual balls. The reason for this is the first ball rebounds on impact with the ground and hits the second ball and the second ball hits the third and so on. Each ball transfers its momentum to the next one. If it was possible to drop eight balls onto each other in this way the top ball would reach a velocity of 10 000 m/s. This would be fast enough to put it into orbit if it did not vaporise from the heat created by friction as it went through in the earth's atmosphere! Eight balls may be difficult to manage but even with two or three the effect is quite dramatic. Try it with just two and see for yourself.

1.12 Properties of water

The following are some of the physical properties of water. This will be a useful reference for work in later chapters.

1.12.1 Density

When dealing with solid objects their mass and weight are important, but when dealing with fluids it is much more useful to know about their *density.* There are two ways of expressing density; *mass density* and *weight density*. Mass density of any material is the mass of one cubic metre of the material and is a fixed value for the material concerned. For example, the mass density of air is 1.29 kg/m^3, steel is 7800 kg/m^3 and gold is 19 300 kg/m^3.

Mass density is determined by dividing the mass of some object by its volume:

$$\text{density (kg/m}^3) = \frac{\text{mass (kg)}}{\text{volume (m}^3)}$$

Mass density is usually denoted by the Greek letter ρ (rho).

For water the mass of one cubic metre of water is 1000 kg and so:

$$\rho = 1000 \text{ kg/m}^3$$

Density can also be written in terms of weight as well as mass. This is referred to as *weight density* but engineers often use the term *specific weight* (w). This is the weight of one cubic metre of water.

Newton's second law is used to link mass and weight:

weight density (kN/m^3) = mass density (kg/m^3) × gravity constant (m/s^2)

For water:

$$\begin{aligned}\text{weight density} &= 1000 \times 9.81 \\ &= 9810\ \text{N/m}^3\ (\text{or } 9.81\ \text{kN/m}^3) \\ &= 10\ \text{kN/m}^3\ (\text{approximately})\end{aligned}$$

Sometimes weight density for water is rounded off by engineers to 10 kN/m^3. Usually this makes very little difference to the design of most hydraulic works. Note the equation for weight density is applicable to all fluids and not just water. It can be used to find the weight density of any fluid provided the mass density is known.

Engineers generally use the term specific weight in their calculations whereas scientists tend to use the term ρg to describe the weight density. They are in effect the same but for clarity, ρg is used throughout this book.

1.12.2 Relative density or specific gravity

Sometimes it is more convenient to use *relative density* rather than just density. It is more commonly referred to as *specific gravity* and is the ratio of the density of a material or fluid to that of some standard density – usually water. It can be written both in terms of the mass density and the weight density.

$$\text{specific gravity (SG)} = \frac{\text{density of an object (kg/m}^3)}{\text{density of water (kg/m}^3)}$$

Note that specific gravity has no dimensions. As the volume is the same for both the object and the water, another way of writing this formula is in terms of weight:

$$\text{specific gravity} = \frac{\text{weight of an object}}{\text{weight of an equal volume of water}}$$

Some useful specific gravity values are included in Table 1.3.

The density of any other fluid (or any solid object) can be calculated by knowing the specific gravity. The mass density of mercury, for example, can be calculated from its specific gravity:

$$\text{specific gravity of mercury (SG)} = \frac{\text{mass density of mercury (kg/m}^3)}{\text{mass density of water (kg/m}^3)}$$

Table 1.3 Some values of specific gravity.

Material/fluid	*Specific gravity*	*Comments*
Water	1	All other specific gravity measurements are made relative to that of water
Oil	0.9	Less than 1.0 and so it floats on water
Sand/silt	2.65	Important in sediment transport problems
Mercury	13.6	Fluid used in manometers for measuring pressure

So:

$$\begin{aligned}\text{mass density of mercury} &= \text{SG of mercury} \times \text{mass density of water}\\ &= 13.6 \times 1000\\ &= 13\,600\ \text{kg/m}^3\end{aligned}$$

The mass density of mercury is 13.6 times greater than that of water.

Archimedes used this concept of specific gravity in his famous principle (Table 1.3), which is discussed in Section 2.12.

1.12.3 Viscosity

This is the friction force which exists inside a fluid as it flows. It is sometimes referred to as the *dynamic viscosity*. To understand the influence of viscosity imagine a fluid flowing along a pipe as a set of thin layers (Figure 1.7a). Although it cannot be seen and it is not very obvious, the layer nearest to the boundary actually sticks to it and does not slide along as the other layers do. The next layer away from the boundary is moving but is slowed down by friction between it and the first layer. The third layer moves faster but is slowed by the second. This effect continues until the entire flow is affected. It is similar to the sliding effect of a pack of playing cards (Figure 1.7b). This internal friction between the layers of fluid is known as the *viscosity*. Some fluids, such as water, have a low viscosity and this means the friction between the layers of fluid is low and its influence is not so evident when water is flowing. In contrast engine oils have a much higher viscosity and they seem to flow more slowly. This is because the internal friction is much greater.

One way to see viscosity at work is to try and pull out a spoon from a jar of honey. Some of the honey sticks to the spoon and some sticks to the jar, demonstrating that fluid sticks to the boundaries as referred to above. There is also a resistance to pulling out the spoon and this is the influence of viscosity. This effect is the same for all fluids including water but it cannot be so clearly demonstrated as in the honey jar. In fact, viscous resistance in water is ignored in many hydraulic designs. To take account of it not only complicates the problem but also has little or no effect on the outcome because the forces of viscosity are usually very small relative to other forces involved. When forces of viscosity are ignored the fluid is described as an *ideal fluid*.

Another interesting feature of the honey jar is that the resistance changes depending on how quickly the spoon is pulled out. The faster it is pulled the more resistance there is to the pulling. Newton related this rate of movement (the velocity) to the resistance and found they were proportional. This means the resistance increases directly as the velocity of the fluid increases. In other words the faster you try to pull the spoon out of the honey jar the greater will be the force required to do it. Most common fluids conform to this relationship and are still known today as *Newtonian fluids*.

(a) Flow in a pipe as a set of thin layers

(b) Flow is similar to a pack of cards

1.7 Understanding viscosity.

Some modern fluids however, have different viscous properties and are called *non-Newtonian fluids*. One good example is tomato ketchup. When left on the shelf it is a highly viscous fluid which does not flow easily from the bottle. Sometimes you can turn a full bottle upside down and nothing comes out. But shake it vigorously (in scientific terms this means applying a shear force) its viscosity suddenly changes and the ketchup flows easily from the bottle. In other words, applying a force to a fluid can change its viscous properties often to our advantage.

Although viscosity is often ignored in hydraulics, life would be difficult without it. The spoon in the honey jar would come out clean and it would be difficult to get the honey out of the jar. Rivers rely on viscosity to slow down flows otherwise they would continue to accelerate to very high speeds. The Mississippi river would reach a speed of over 300 km/h as its flow gradually descends 450 m towards the sea if water had no viscosity. Pumps would not work because impellers would not be able to grip the water and swimmers would not be able to propel themselves through the water for the same reason.

Viscosity is usually denoted by the Greek letter (mu).
For water:

$$\mu = 0.00114 \text{ kg/ms at a temperature of } 15°\text{C}$$
$$= 1.14 \times 10^{3} \text{ kg/ms}$$

The viscosity of all fluids is influenced by temperature. Viscosity decreases with increasing temperature.

1.12.4 Kinematic viscosity

In many hydraulic calculations viscosity and mass density go together and so they are often combined into a term known as the *kinematic viscosity*. It is denoted by the Greek letter (nu) and is calculated as follows:

$$\text{kinematic viscosity } (\nu) = \frac{\text{viscosity } (\mu)}{\text{density } (\rho)}$$

For water:

$$\nu = 1.14 \times 10^{-2} \text{ m}^2\text{/s at a temperature of } 15°\text{C}$$

Sometimes kinematic viscosity is measured in Stokes in recognition of the work of Sir George Stokes who helped to develop a fuller understanding of the role of viscosity in fluids.

$$10^{4} \text{ Stokes} = 1 \text{ m}^2\text{/s}$$

For water:

$$\nu = 1.14 \times 10^{-2} \text{ Stokes}$$

1.12.5 Surface tension

An ordinary steel sewing needle can be made to float on water if it is placed there very carefully. A close examination of the water surface around the needle shows that it appears to be sitting in a slight depression and the water behaves as if it is covered with an elastic skin. This property is known as *surface tension.* The force of surface tension is very small and is normally expressed in terms of force per unit length.

For water:

surface tension = 0.51 N/m at a temperature of 20°C

This force is ignored in most hydraulic calculations but in hydraulic modelling, where small-scale models are constructed in a laboratory to try and work out forces and flows in large, complex problems, surface tension may influence the outcome because of the small water depths and flows involved.

1.12.6 Compressibility

It is easy to imagine a gas being compressible and to some extent some solid materials such as rubber. In fact all materials are compressible to some degree including water which is 100 times more compressible than steel! The compressibility of water is important in many aspects of hydraulics. Take for example the task of closing a sluice valve to stop water flowing along a pipeline. If the water was incompressible it would be like trying to stop a solid 40 ton truck. The water column would be a solid mass running into the valve and the force of impact could be significant. Fortunately water is compressible and as it impacts on the valve it compresses like a spring and this absorbs the energy of the impact. Returning to the road analogy, it is similar to what happens when cars crash on the road because of some sudden stoppage. Each car collapses on impact and this absorbs much of the energy of the collision. However, this is not the end of the story. As the water compresses the energy that is absorbed causes the water pressure to suddenly rise and this leads to another problem known as water hammer. This is discussed more fully in Section 4.16.

2 Hydrostatics: water at rest

2.1 Introduction

Hydrostatics is the study of water which is not moving, that is, it is at rest. It is important to civil engineers for the design of water storage tanks and dams. What are the forces created by water and how strong must a tank or a dam be to resist them? It is also important to naval architects who design ships and submarines. How deep can a submarine go before the pressures become too great and damage it? The answers to these questions can be found from studying hydrostatics. The theory is quite simple both in concept and in use. It is also a well-established theory that was set down by Archimedes (287–212BC) over 2000 years ago and is still used in much the same way today.

2.2 Pressure

The term *pressure* is used to describe the force exerted by water on each square metre of some object submerged in water, that is, force per unit area. It may be the bottom of a tank, the side of a dam, a ship or a submerged submarine. It is calculated as follows:

$$\text{pressure} = \frac{\text{force}}{\text{area}}$$

Introducing the units of measurement:

$$\text{pressure (kN/m}^2\text{)} = \frac{\text{force (kN)}}{\text{area (m}^2\text{)}}$$

Force is in kilo-Newtons (kN), area is in square metres (m^2) and so pressure is measured in kN/m^2. Sometimes pressure is measured in *Pascals* (Pa) in recognition of Blaise Pascal (1620–1662) who clarified much of modern-day thinking about pressure and barometers for measuring atmospheric pressure.

$$1 \text{ Pa} = 1 \text{ N/m}^2$$

One Pascal is a very small quantity and so kilo-Pascals are often used so that:

1 kPa = 1 kN/m^2

Although it is in order to use Pascals, kilo-Newtons per square metre is used throughout this text for the dimensions of pressure.

EXAMPLE: CALCULATING PRESSURE IN A TANK OF WATER

Calculate the pressure on a flat plate 3 m by 2 m when a mass of 50 kg rests on it. Calculate the pressure when the plate is reduced to 1.5 m by 2 m (Figure 2.1).

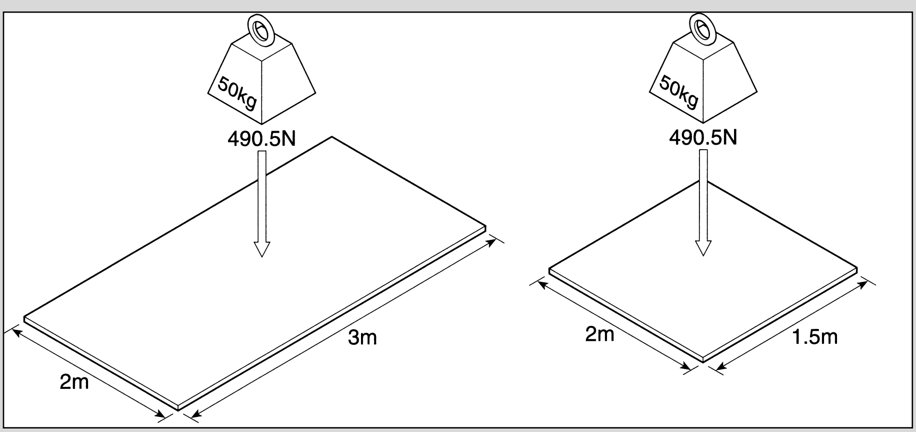
First calculate the weight on the plate. Remember weight is a force.

$$\begin{aligned} \text{mass on plate} &= 50 \text{ kg} \\ \text{weight on the plate} &= \text{mass} \times \text{gravity constant} \\ &= 50 \times 9.81 = 490.5 \text{ N} \\ \text{plate area} &= 3 \times 2 = 6 \text{ m}^2 \\ \text{pressure on plate} &= \frac{\text{force}}{\text{area}} = \frac{490.5}{6} \\ &= 81.75 \text{ N/m}^2 \end{aligned}$$

When the plate is reduced to 1.5 m by 2 m:

$$\begin{aligned} \text{plate area} &= 1.5 \times 2 = 3 \text{ m}^2 \\ \text{pressure on plate} &= \frac{490.5}{3} \\ &= 163.5 \text{ N/m}^2 \end{aligned}$$

Note that the mass and the weight remain the same in each case. But the areas of the plate are different and so the pressures are also different.



2.1 Different areas produce different pressures for the same force.

2.3 Force and pressure are different

Force and pressure are terms that are often confused. The difference between them is best illustrated by an example. If you had to choose between an elephant standing on your foot or a woman in a high-heel (stiletto) shoe, which would you choose? The sensible answer would be the elephant, as it is less likely to do damage to your foot than the high-heel shoe. To understand this is to appreciate the important difference between force and pressure.

The weight of the elephant is obviously greater than that of the woman but the pressure under the elephant's foot is much less than that under the high-heel shoe (see calculations in the box). The woman's weight (force) is small in comparison to that of the elephant, but the area of the shoe heel is very small and so the pressure is extremely high. So the high-heel shoe is likely to cause you more pain than the elephant. This is why high-heel shoes, particularly those with a very fine heel, are sometimes banned indoors as they can so easily punch holes in flooring and furniture!

There are many other examples which highlight the difference. Agricultural tractors often use wide (floatation) tyres to spread their load and reduce soil compaction. Military tanks use caterpillar tracks to spread the load to avoid getting bogged down in muddy conditions. Eskimos use shoes like tennis rackets to avoid sinking into the soft snow.

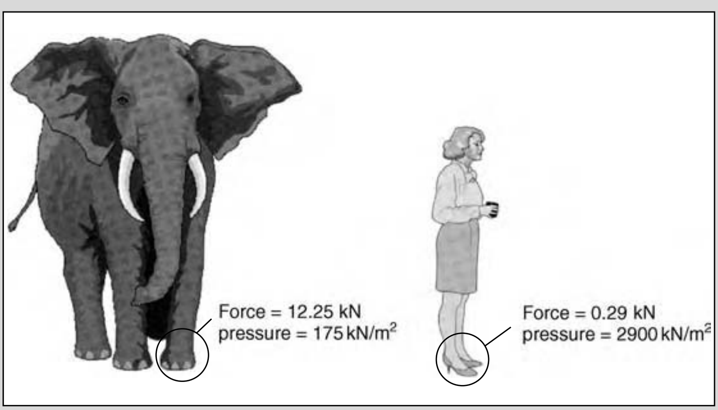
EXAMPLE: THE ELEPHANT'S FOOT AND THE WOMAN'S SHOE

An elephant has a mass of 5000 kg and its feet are 0.3 m in diameter. A woman has a mass of 60 kg and her shoe heel has a diameter of 0.01 m. Which produces the greater pressure – the elephant's foot or woman's shoe heel (Figure 2.2)?

First calculate the pressure under the elephant's foot:

$$\text{elephant's mass} = 5000 \text{ kg}$$
$$\text{elephant's weight} = 5000 \times 9.81$$
$$= 49\,050 \text{ N} = 49 \text{ kN}$$

$$\text{weight on each foot} = \frac{49}{4} = 12.25 \text{ kN}$$



2.2 Which produces the greater pressure?

$$\text{foot area} = \frac{\pi d^2}{4} = \frac{\pi 0.3^2}{4}$$

$$= 0.07 \text{ m}^2$$

$$\text{pressure under foot} = \frac{\text{force}}{\text{area}} = \frac{12.25}{0.07}$$

$$= 175 \text{ kN/m}^2$$

Now calculate the pressure under the woman's shoe heel:

$$\text{woman's mass} = 60 \text{ kg}$$

$$\text{woman's weight} = 589 \text{ N} = 0.59 \text{ kN}$$

$$\text{weight on each foot} = \frac{0.59}{2} = 0.29 \text{ kN}$$

$$\text{area of shoe heel} = \frac{\pi d^2}{4} = \frac{\pi 0.01^2}{4}$$

$$= 0.0001 \text{ m}^2$$

$$\text{pressure under heel} = \frac{\text{force}}{\text{area}} = \frac{0.29}{0.0001}$$

$$= 2900 \text{ kN/m}^2$$

The pressure under the woman's heel is 16 times greater than under the elephant's foot. So which would you rather have standing on your foot?

2.4 Pressure and depth

The pressure on some object under water is determined by the depth of water above it. So the deeper the object is below the surface, the higher will be the pressure. The pressure can be calculated using the pressure-head equation:

$$p = \rho g h$$

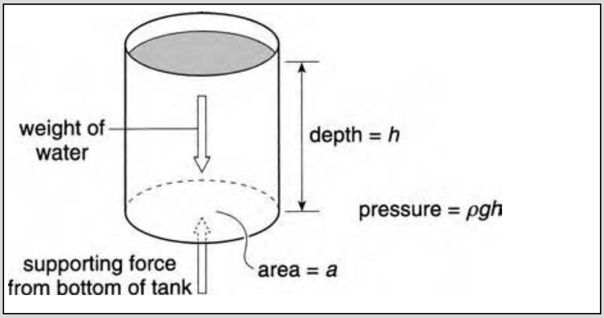
where p is pressure (kN/m^2); ρ is mass density of water (kN/m^3); g is gravity constant (m/s^2); h is depth of water (m).

This equation works for all fluids and not just water, provided of course that the correct value of density is used for the fluid concerned.

To see how the pressure-head equation is derived look in the box below.

DERIVATION: PRESSURE-HEAD EQUATION

Imagine a tank of water of depth h and a cross-sectional area of a. The weight of water on the bottom of the tank (remember that weight is a force and is acting downwards) is balanced by an upward force from the bottom of the tank supporting the water (Newton's third law). The pressure-head equation is derived by calculating these two forces and putting them equal to each other (Figure 2.3).



2.3 Calculating the forces on the bottom of a tank.

First calculate the downward force. This is the weight of water. To do this first calculate the volume and then the weight using the density:

$$\text{volume of water} = \text{cross-sectional area} \times \text{depth}$$
$$= a \times h$$

And so:

$$\text{weight of water in tank} = \text{volume} \times \text{density} \times \text{gravity constant}$$
$$= a \times h \times \rho \times g$$

This is the downward force of the water ↓. Next calculate the supporting (upward) force from the base:

$$\text{supporting force} = \text{pressure} \times \text{area}$$
$$= p \times a$$

Now put these two forces equal to each other:

$$p \times a = a \times h \times \rho \times g$$

The area *a* cancels out from both sides of the equation and so:

$$p = \rho g h$$

pressure = mass density × gravity constant × depth of water

This is the pressure-head equation and it links pressure with the depth of water. It shows that pressure increases directly as the depth increases. Note that it is completely independent of the shape of the tank or the area of base.

EXAMPLE: CALCULATING PRESSURE AND FORCE ON THE BASE OF A WATER TANK

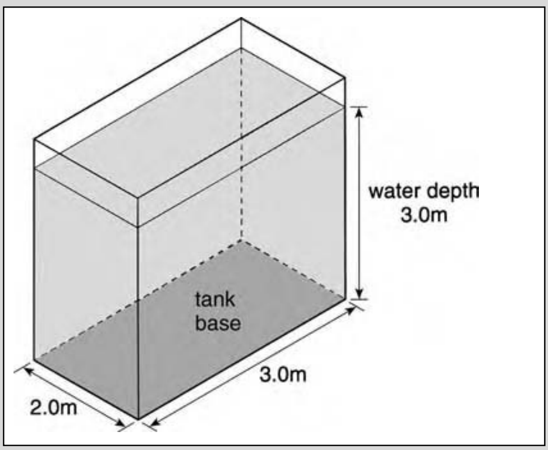
A rectangular tank of water is 3 m deep. If the base measures 3 m by 2 m, calculate the pressure and force on the base of the tank (Figure 2.4).

Use the pressure-head equation:

$$\begin{aligned} p &= \rho g h \\ &= 1000 \times 9.81 \times 3.0 \\ &= 29\,430 \text{ N/m}^2 \\ &= 29.43 \text{ kN/m}^2 \end{aligned}$$

Calculate the force on the tank base using the pressure and the area:

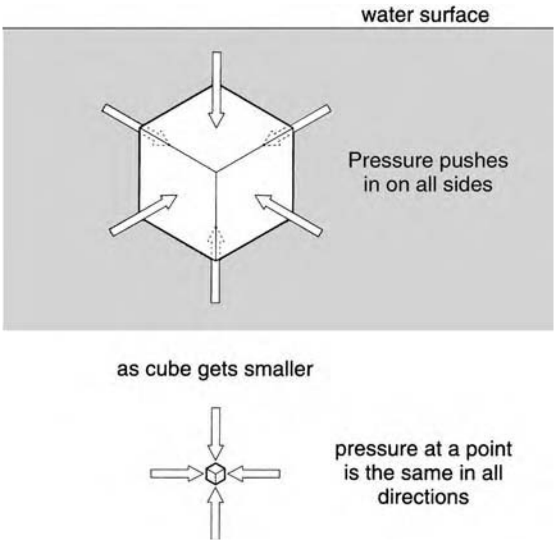
$$\begin{aligned} \text{force} &= \text{pressure} \times \text{area} \\ \text{base area} &= 3 \times 2 = 6 \text{ m}^2 \\ \text{force} &= 29.43 \times 6 \\ &= 176.6 \text{ kN} \end{aligned}$$



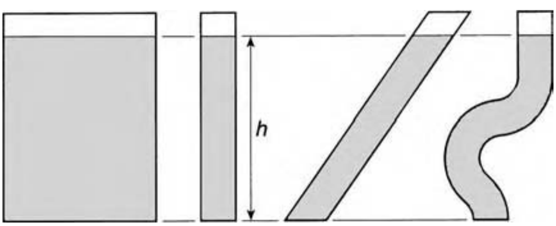
2.4 Calculating force and pressure on tank base.

2.5 Pressure is same in all directions

Although in the box example the pressure is used to calculate the downward force on the tank base, pressure does not in fact have a specific direction – it pushes in all directions. To understand this, imagine a cube immersed in water (Figure 2.5). The water pressure pushes on all sides of the cube and not just on the top. If the cube was very small then the pressure on all six faces would be almost the same. If the cube gets smaller and smaller until it almost disappears, it becomes clear that *the pressure at any point in the water is the same in all directions*. So the pressure pushes in all directions and not just vertically. This idea is important for designing dams because it is the horizontal action of pressure which pushes on a dam and which must



2.5 Pressure is the same in all directions.



2.6 Pressure is the same at the base of all the containers.

be resisted if the dam is not to fail. Note also that the 'pressures' in Figure 2.5 are drawn pushing inwards. But they could equally have been drawn pushing outwards to make the same argument – remember Newton's third law.

2.6 The hydrostatic paradox

It is often assumed that the size of a water tank or its shape influences pressure but this is not the case (Figure 2.6). It does not matter if the water is in a large tank or in a narrow tube. The pressure-head equation tells us that water depth is the only variable that determines the pressure. So the base area has no effect on the pressure nor does the amount of water in the tank. What is different of course is the force on the base of different containers. The *force* on the base of each tank is equal to weight of water in each of the containers. But if the depth of water in each is the same then the pressure will also be the same.

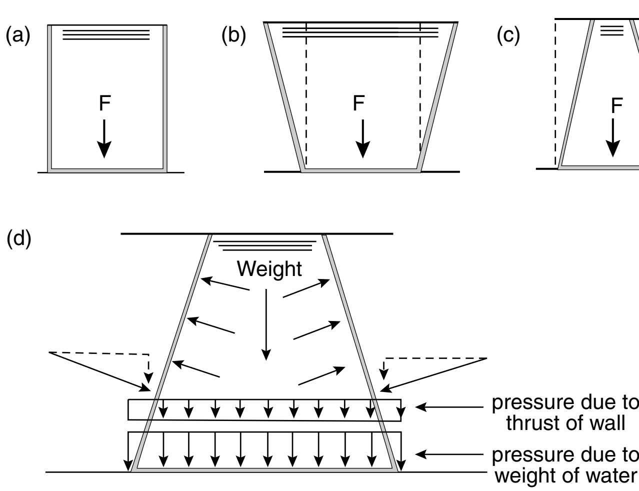
2.6.1 The bucket problem

The Dutch mathematician Simon Stevin (1548–1620) made a similar point by showing that the *force* on the base of a tank depended only on the area of the base and the vertical depth of water – and not the weight of water it contained. This is well demonstrated using three different-shaped buckets each with the same base area and the same depth of water in them (Figure 2.7).

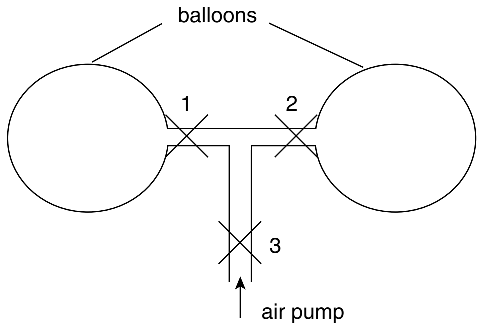
The weight of water in each bucket is clearly different and a casual observer might assume from this that the force on the base of each bucket will also be different: the force on the base of bucket *b* being greater than bucket *a* and the force on bucket *c* being less than bucket *a*. But thinking about this in hydraulic terms, the pressure-head equation tells us otherwise. In fact it predicts that the force on the base is the *same* in each case. It is equal to the area of the base multiplied by the pressure on the base, which is only a function of the water depth. So the force on the base is the same regardless of whether the sides are vertical or inclined inwards or outwards. In bucket *a*, with vertical sides, the force does in fact equal the weight of water in the container. But when the sides slope outwards, as in bucket *b*, the force is less than the weight of water and when the sides slope inwards, as in bucket *c*, the force on the base is greater than the weight of water. All this seems rather absurd but it is true.

The key to the paradox lies with the fact that the pressure at any point in the water is the same in all directions. The water not only pushes down onto the base but also pushes on the sides of the container as well. So when the sides slope inwards (bucket *c*) the water pushes outwards and also upwards. Newton's second law says that this produces a corresponding downward force on the water and this is transmitted to the base adding to the force due to the weight of the water (Figure 2.7d). In fact, the total vertical force on the walls and base (the force on the base less the upward force on the walls) is exactly equal to the weight of water in the bucket! The same argument can be applied to bucket *b*. The water pushes on the sides of the tank and in this case push outwards and downwards. Newton's second law says that this produces a corresponding upward force on the water and this is transmitted to the base reducing the force due to the weight of the water. So in this case the force on the base is less than the weight of water in the bucket.

Clear? If so then you are well on your way not only to understanding the important difference between force and pressure but also appreciating the significance of Newton's contribution to our understanding of the way in which our world works.



2.7 The bucket problem.



2.8 The balloon problem.

2.6.2 The balloon problem

One more 'absurdity' to test your understanding. Two identical balloons are connected to a manifold and blown up independently so that one is larger than the other (Figure 2.8). When valve 3 is closed and valves 1 and 2 are opened the air can flow between the balloons to equalise the air pressure. The question is – What happens to the balloons?

The normal expectation is that air moves from the larger balloon to the smaller one so they both become the same size, but this is not what happens. The larger balloon in fact gets larger and the smaller balloon gets smaller. The reason for this is again explained by the difference between pressure and force. The larger balloon has a much greater surface area than the small one and the force on the skin of the balloon will be greater as it approaches bursting point. But because of the large surface area the pressure inside is much smaller than it is in the smaller balloon. So when the two balloons are connected the higher air pressure in the small balloon flows into the larger balloon thus making the large balloon even larger and the small balloon smaller. So do not confuse size with pressure. If you are not convinced or you are still confused, try the balloon experiment by making up a small manifold using some plastic pipes and laboratory taps and see for yourself.

2.7 Pressure head

Engineers often refer to pressure in terms of metres of water rather than as a pressure in kN/m^2. So, referring to the pressure calculation in the box, instead of saying the pressure is 29.43 kN/m^2 they will say the pressure is 3 m head of water. They can do this because of the unique relationship between pressure and water depth ($p = \rho gh$). It is called the *pressure head* or just *head* and is measured in metres. It is the water depth h referred to in the pressure-head equation. Both ways of stating the pressure are correct and one can easily be converted to the other using the pressure-head equation.

Engineers prefer to use head measurements because, as will be seen later, differences in ground level can affect the pressure in a pipeline. It is then an easy matter to add (or subtract) changes in ground level to pressure values because they both have the same dimensions.

A word of warning though. When head is measured in metres it is important to say what the liquid is – 3 m head of water will be a very different pressure from 3 m head of mercury. This is because the density term ρ is different. So the rule is – say what liquid is being measured, for example, 3 metres head of water or 3 metres head of mercury etc. See the worked example in the box.

EXAMPLE: CALCULATING PRESSURE HEAD IN MERCURY

Building on the previous example, calculate the depth of mercury needed in the tank to produce the same pressure as 3 m depth of water (29.43 kN/m^2). Specific gravity (SG) of mercury is 13.6.

First calculate the density of mercury:

$$\begin{aligned}\rho\text{ (mercury)} &= \rho\text{ (water)} \times \text{SG (mercury)}\\ &= 1000 \times 13.6\\ &= 13\,600\text{ kg/m}^3\end{aligned}$$

Use the pressure-head equation to calculate the head of mercury:

$$p = \rho g h$$

Where ρ is now the density and h is the depth of mercury:

$$\begin{aligned}29\,430 &= 13\,600 \times 9.81 \times h\\ h &= 0.22\text{ m of mercury}\end{aligned}$$

So the depth of mercury required to create the same pressure as 3 m of water is only 0.22 m. This is because mercury is much denser than water.

2.8 Atmospheric pressure

The pressure of the atmosphere is all around us pressing on our bodies. Although we often talk about things being ‘as light as air’ when there is a large depth of air, as on the earth’s surface, it creates a very high pressure of approximately 100 kN/m^2. The average person has a skin area of 2 m^2 so the force acting on each of us from the air around us is approximately 200 kN (the equivalent of 200 000 apples or approximately 20 tons). A very large force indeed! Fortunately there is an equal and opposite pressure from within our bodies that balances the air pressure and so we feel no effect (Newton’s third law).

At high altitudes where atmospheric pressure is less than at the earth’s surface, some people suffer from nose bleeds due to their blood pressure being much higher than that of the surrounding atmosphere. We also notice slight, sudden changes in air pressure. For instance, when we fly in an aeroplane, even though the cabin is pressurised, our ears pop as our bodies adjust to changes in the cabin pressure. But if for some reason the cabin pressure system failed suddenly removing one side of this pressure balance then the result could be catastrophic. Inert gases such as nitrogen, which are normally dissolved in our body fluids and tissues, would rapidly start to form gas bubbles which can result in sensory failure, paralysis and death. Deep sea divers are well aware of this rapid pressure change problem and so make sure that they return to the surface slowly so that their bodies have enough time to adjust to the changing pressure. It is known as ‘the bends’. A good practical demonstration of what happens can be seen when you open a fizzy drink bottle. When the cap is removed from the bottle, gas is heard escaping, and bubbles can be seen forming in the drink. This is carbon dioxide gas coming out of solution as a result of the sudden pressure drop inside the bottle as it equalises with the pressure of the atmosphere.

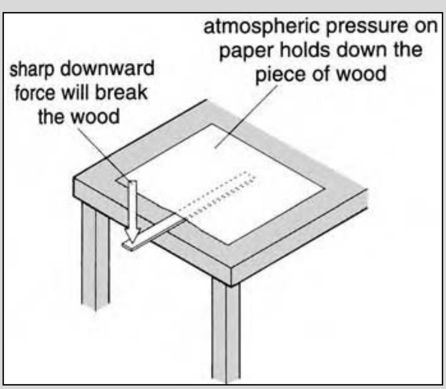
It was in the 17th century that scientists such as Evangelista Torricelli (1608–1647), a pupil of Galileo Galilei (1564–1642), began to understand about atmospheric pressure and to study the importance of vacuums – the empty space when all the air is removed. Scientists previously explained atmospheric effects by saying *that nature abhors a vacuum*. By this they meant that if the air is sucked out of a bottle it will immediately fill by sucking air back in again when it is opened to the atmosphere. But Galileo commented that a suction pump could not lift water more than 10 m so there appeared to be a limit to this abhorrence. Today we know that it is not the vacuum in the bottle that sucks in the air but the outside air pressure that pushes the air in. The end result is the same (i.e. the bottle is filled with air), but the mechanism is quite different.

Galileo realised that this had important consequences for suction pumps. Suction pumps do not 'suck' up water as was commonly thought. It is atmospheric pressure on the surface of the water that pushes water into the pump and to do this the air must first be removed from the pump to create a vacuum – a process known as 'priming'. The implication of this is that atmospheric pressure (10 m of water) puts an absolute limit on how high a pump can be located above the water surface and still work. In practice the limit is a lot lower than this but more about this in Section 8.4. Siphons too rely on atmospheric pressure in a similar way (Section 7.11).

Atmospheric pressure does vary over the surface of the earth. It is lower in mountainous regions and also varies as a result of the earth's rotation and temperature changes in the atmosphere which both cause large air movements. They create high and low pressure areas that create winds as air flows from high pressure to low pressure areas in an attempt to try and equalise the air pressure. This may be important in meteorology but in hydraulics such differences are relatively small and have little effect on solving problems – except of course if you happen to be building a pumping station for a community in the Andes or the Alps. So for all intents and purposes atmospheric pressure close to sea level can be assumed constant at 100 kN/m^2 – or approximately 10 m head of water.

EXAMPLE: EXPERIENCING ATMOSPHERIC PRESSURE

One way of experiencing atmospheric pressure is to place a large sheet of paper on a table over a thin piece of wood. If you hit the wood sharply it is possible to strike a considerable blow without disturbing the paper. You may even break the wood. This is



2.9 Experiencing atmospheric pressure.

because the paper is being held down by the pressure of the atmosphere.

If the paper is 1.0 m^2 then the force holding down the paper can be calculated as follows:

force = pressure × area

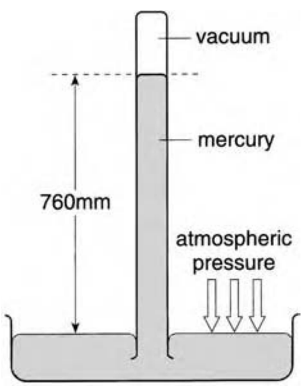
In this case:

pressure = atmospheric pressure
= 100 kN/m^2

And so:

force = 100 × 1 = 100 kN

In terms of apples this is about 100 000, which is a large force. It is little wonder that the wood breaks before the paper lifts.



2.10 Measuring atmospheric pressure.

2.8.1 Mercury barometer

One of the instruments used to measure atmospheric pressure is the mercury barometer. It was developed by Evangelista Torricelli in 1643, and has largely remained unchanged since except for the introduction of a vernier measuring scale to measure accurately the small changes in atmospheric pressure. This was done by Fortin in 1810 and so the instrument is now referred to as the *Fortin barometer*.

Torricelli's barometer consists of a vertical glass tube closed at one end, filled with mercury and inverted with the open end immersed in a cistern of mercury (Figure 2.10). The cistern surface is exposed to atmospheric pressure and this supports the mercury column, the height of which is a measure of atmospheric pressure. It is normally measured in mm and the long-term average value at sea level is 760 mm.

Torricelli could have used water for the barometer instead of mercury, but he would have needed a tube over 10 m high to do it – not a very practical proposition for the laboratory or for taking measurements.

EXAMPLE: MEASURING ATMOSPHERIC PRESSURE USING A MERCURY BAROMETER

Calculate atmospheric pressure when the reading on a mercury barometer is 760 mm of mercury. What would be the height of the column if the same air pressure was measured using water instead of mercury?

The pressure-head equation links together atmospheric pressure and the height of the mercury column, but remember the fluid is now mercury and not water:

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

h is 760 mm and ρ for mercury is 13 600 kg/m^3 (13.6 times denser than water)

So:

$$\begin{aligned}\text{atmospheric pressure} &= 13\,600 \times 9.81 \times 0.76 \\ &= 101\,400 \text{ N/m}^2 \text{ or } 101.4 \text{ kN/m}^2\end{aligned}$$

Calculate the height of the water column to measure atmospheric pressure using the pressure-head equation again:

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

This time the fluid is water and so:

$$\begin{aligned}101\,400 &= 1000 \times 9.81 \times h \\ h &= 10.32 \text{ m}\end{aligned}$$

This is a very tall water column and there would be practical difficulties if it was used for routine measurement of atmospheric pressure. Hence the reason why a very dense liquid like mercury is used to make measurement more manageable.

Atmospheric pressure is also used as a unit of measurement for pressure both for meteorological purposes and in hydraulics. This unit is known as the *bar*. For convenience I bar pressure is rounded off to 100 kN/m^2.

A more commonly used term in meteorology is the *millibar*.

So:

$$1 \text{ millibar} = 0.1 \text{ kN/m}^2 = 100 \text{ N/m}^2$$

To summarise – there are several ways of expressing atmospheric pressure:

$$\begin{aligned}\text{atmospheric pressure} &= 1 \text{ bar} \\ \text{or} &= 100 \text{ kN/m}^2 \\ \text{or} &= 10 \text{ m of water} \\ \text{or} &= 760 \text{ mm of mercury}\end{aligned}$$

EXAMPLE: CALCULATING PRESSURE HEAD

A pipeline is operating at a pressure of 3.5 bar. Calculate the pressure in metres head of water.

$$1 \text{ bar} = 100 \text{ kN/m}^2 = 100\,000 \text{ N/m}^2$$

And so:

$$3.5 \text{ bar} = 350 \text{ kN/m}^2 = 350\,000 \text{ N/m}^2$$

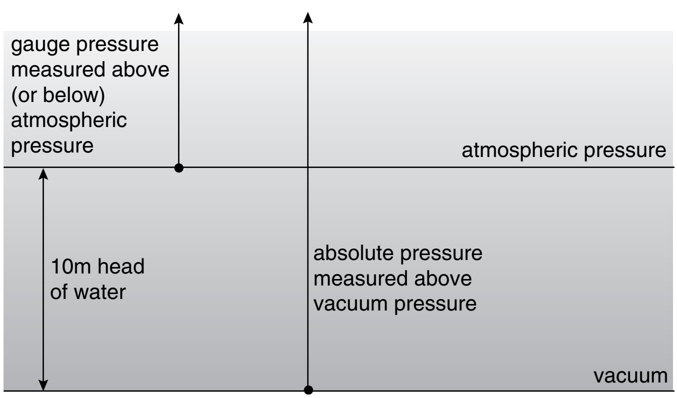
Use the pressure-head equation:

$$p = \rho g h$$
$$350\,000 = 1000 \times 9.81 \times h$$

Calculate head h:

$$h = 35.67 \text{ m}$$

Round this off: 3.5 bar = 36 m of water (approximately)



2.11 Gauge and absolute pressures.

2.9 Measuring pressure

2.9.1 Gauge and absolute pressures

Pressure measuring devices work in the atmosphere with normal atmospheric pressure all around them. Rather than add atmospheric pressure each time a measurement is made it is common practice to assume that atmospheric pressure is equal to zero and so it becomes the base line (or zero point) from which all pressure measurements are made. It is rather like setting sea level as the zero from which all ground elevations are measured (Figure 2.11). Pressures measured in this way are called *gauge pressures*. They can either be positive (above atmospheric pressure) or negative (below atmospheric pressure).

Most pressure measurements in hydraulics are gauge pressures but some mechanical engineers, working with gas systems occasionally measure pressure using a vacuum as the datum. In such cases the pressures are referred to as *absolute pressures*. It is not possible to have a pressure lower than vacuum pressure and so all absolute pressures have positive values.

To summarise:

Gauge pressures are pressures measured above or below atmospheric pressure. Absolute pressures are pressures measured above a vacuum.

To change from one to the other:

absolute pressure = gauge pressure + atmospheric pressure

Note, if only the word pressure is used, it is reasonable to assume that this means gauge pressure.

2.9.2 Bourdon gauges

Pressure can be measured in several ways. The most common instrument used is the *Bourdon Gauge* (Figure 2.12a). This is located at some convenient point on a pipeline or pump to record pressure, usually in kN/m^2 or bar. It is a simple device and works on the same principle as a party toy. When you blow into it, the coil of paper unfolds and the feather rotates. Inside a Bourdon gauge there is a similar curved tube which tries to straighten out under pressure and causes a pointer to move through a gearing system across a scale of pressure values.

2.9.3 Piezometers

This is another device for measuring pressure. A vertical tube is connected to a pipe so that water can rise up the tube because of the pressure in the pipe (Figure 2.12a). This is called a *piezometer* or *standpipe.* The height of the water column in the tube is a measure of the pressure in the pipe, that is, the pressure head. The pressure in kN/m^2 can be calculated using the pressure-head equation.

EXAMPLE: MEASURING PRESSURE USING A STANDPIPE

Calculate the height of a standpipe needed to measure a pressure of 200 kN/m^2 in a water pipe.

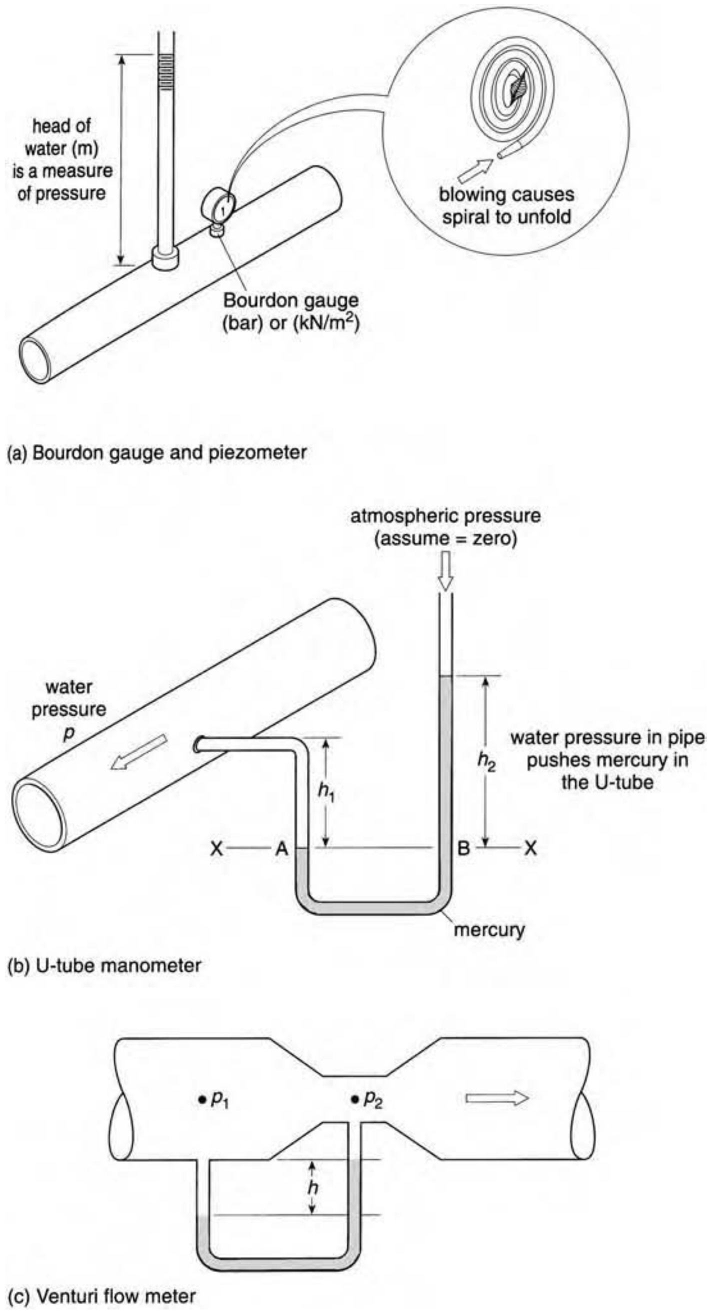
Using the pressure-head equation:

$$p = \rho g h$$
$$200\,000 = 1000 \times 9.81 \times h$$

Note in the equation pressure and density are both in N – not kN.

$$h = 20.4 \text{ m}$$

A very high tube would be needed to measure this pressure and it would be a rather impracticable measuring device! For this reason high pressures are normally measured using a Bourdon gauge or a manometer.



2.12 Measuring pressure.

2.9.4 Manometers

Vertical standpipes are not very practical for measuring high pressures (see example in box). An alternative is to use a *U-tube manometer* (Figure 2.12b).

The bottom of the U-tube is filled with a different liquid which does not mix with that in the pipe. When measuring pressures in a water system, oil or mercury is used. Mercury is very useful because high pressures can be measured with a relatively small tube (see atmospheric pressure).

To measure pressure, a manometer is connected to a pipeline and mercury is placed in the bottom of the U-bend. The basic assumption is that as the mercury in the manometer is not moving the pressures in the two limbs must be the same. If a horizontal line X–X is drawn through the mercury surface in the first limb and extended to the second limb then it can be assumed that:

pressure at point A = pressure at point B

This is the fundamental assumption on which all manometer calculations are based. It is then a matter of adding up all the components which make up the pressures at A and B to work out a value for the pressure in the pipe.

First calculate the pressure at A:

pressure at A = water pressure at centre of pipe (p)
+ pressure due to water column h_1

$$= p + \rho_{(water)} g\, h_1$$
$$= p + (1000 \times 9.81 \times h_1)$$
$$= p + 9810 \times h_1$$

Now calculate the pressure at B:

pressure at B = pressure due to mercury column h_2
+ atmospheric pressure

Normally atmospheric pressure is assumed to be zero. So:

pressure at B = $\rho_{(mercury)}\, g\, h_2 + 0$

$$= 1000 \times 13.6 \times 9.81 \times h_2$$
$$= 133\,430\, h_2$$

Putting the pressure at A equal to the pressure at B:

$$p + 9810\, h_1 = 133\,430\, h_2$$

Rearrange this to determine the pressure in the pipe p:

$$p = 133\,430\, h_2 - 9810\, h_1$$

Note that p is in N/m^2.

So the pressure in this pipeline can be calculated by measuring h_1 and h_2 and using the above equation.

Some manometers are used to measure pressure differences rather than actual values of pressure. One example of this is the measurement of the pressure difference in a venturi meter used for measuring water flow in pipes (Figure 2.12c). In this case it is the drop (difference) in pressure as water passes through a narrow section of pipe that is important. By connecting one limb of the manometer to the main pipe and the other limb to the narrow section, the difference in pressure can be determined. Note that the pressure difference is not just the difference in the mercury readings on the two columns as is often thought. The pressure difference must be calculated using the principle described above for the simple manometer. More about venture meters and using manometers in Section 4.10.

The best way to deal with manometer measurements is to remember the principle on which all manometer calculations are based and not the formula for p. There are many different ways of arranging manometers with different fluids in them and so there will be too many formulae to remember. So just remember and apply the principle – pressure on each side of the manometer is the same across a horizontal line AB – then the pressure can be easily determined. See the worked example in the box.

EXAMPLE: MEASURING PRESSURE USING A MANOMETER

A mercury manometer is used to measure the pressure in a water pipe (Figure 2.12c). Calculate the pressure in the pipe when $h_1 = 1.5$ m and $h_2 = 0.8$ m.

To solve this problem start with the principle on which all manometers are based:

pressure at A = pressure at B

Calculate the pressures at A and B:

pressure at A = water pressure in pipe (p)
+ pressure due to water column h_1

$$= p + \rho_{(water)} g h_1$$
$$= p + 1000 \times 9.81 \times 1.5$$

pressure at B = pressure due to mercury column h_2
+ atmospheric pressure

$$= \rho_{(mercury)} g h_2 + 0$$
$$= 1000 \times 13.6 \times 9.81 \times 0.8$$

Note that as all the pressures are gauge pressures, atmospheric pressure is assumed to be zero.

Putting the pressure at A equal to the pressure at B:

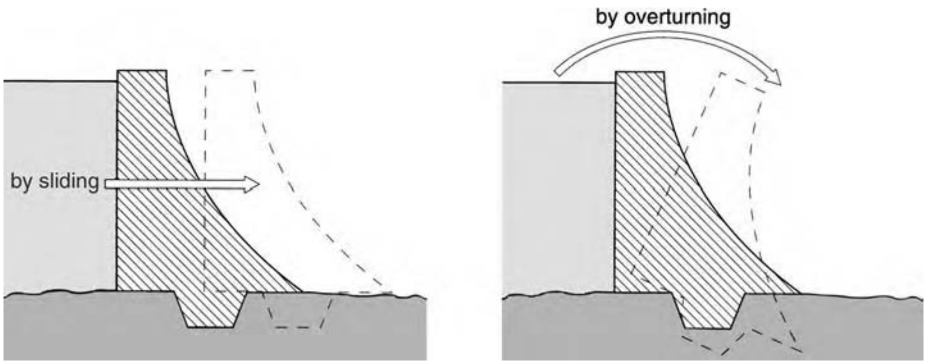
$$p + 1000 \times 9.81 \times 1.5 = 1000 \times 13.6 \times 9.81 \times 0.8$$

Rearrange this to determine p:

$$p = (1000 \times 13.6 \times 9.81 \times 0.8) - (1000 \times 9.81 \times 1.5)$$
$$= 106\,732 - 14\,715$$
$$= 92\,017 \text{ N/m}^2$$
$$p = 92 \text{ kN/m}^2$$

2.10 Designing dams

Engineers are always interested in the ways in which things fall down or collapse so they can devise design and construction procedures that produce safe reliable structures. Dams in



2.13 Dams can fail by sliding and overturning.

particular are critical structures because failure can cause a great deal of damage and loss of life. Hydraulically a dam structure can fail in two ways – the pressure of water can cause the dam to slide forward and it can also cause it to overturn (Figure 2.13). The engineer must design a structure that is strong enough to resist both these possible modes of failure. This is where the principles of hydrostatics play a key role – the same principles apply to small dams only a few metres high as they do for major dams 40 m or more in height.

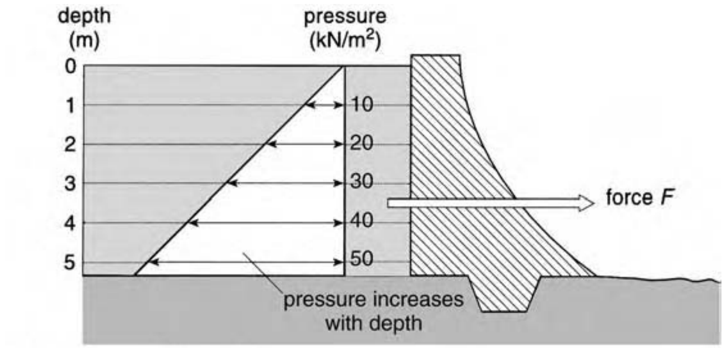
The pressure of water stored behind a dam produces a horizontal force which could cause it to slide forward if the dam was not strong enough to resist. So the total force resulting from the water pressure must first be calculated. The location of this force is also important. If it is near the top of the dam then it may cause the dam to overturn. If it is near the base then it may fail by sliding.

The force on a dam is calculated from the water pressure (Figure 2.14a). Remember that pressure pushes in all directions; in this case it is the horizontal push on the dam which is important. At the water surface the pressure is zero, but 1.0 m below the surface the pressure rises to 10 kN/m^2 (approximately), at 2.0 m it reaches 20 kN/m^2 and so on (remember the pressure-head equation $p = \rho gh$). A graph of the changes in pressure with depth is a straight line; together with the axes of the graph it forms a triangle. The pressure at the top of the triangle (the water surface) is zero and increases uniformly with depth. This triangle is called the pressure diagram and shows how the pressure varies with depth on the upstream face of a dam.

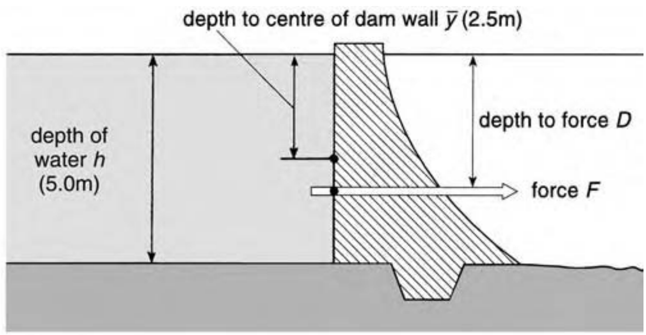
The force on the dam can be calculated from the pressure and the area of the dam face using the equation $F = pa$. But the pressure is not constant – it varies down the face of the dam and so the question is: which value of pressure should be used? One approach is to divide the dam face into lots of small areas and use the average pressure for each area. The force on each area is then calculated using the equation $F = pa$. But this results in lots of small forces to deal with. A much simpler method is to use a formula derived from combining all the small forces mathematically into a single larger force (F) (Figure 2.14a). This single force has the same effect as the sum of all the smaller forces and is much easier to deal with.

$$\text{force } (F) = \rho g a \bar{y}$$

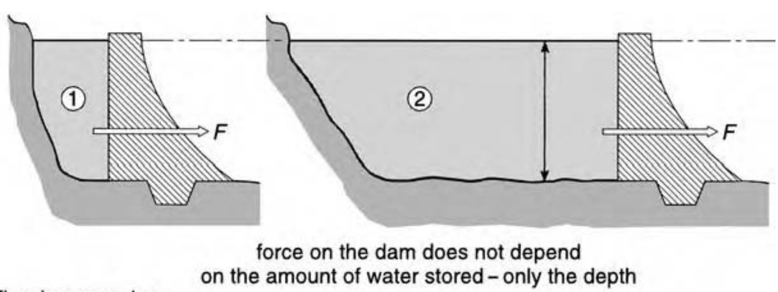
where ρ is density of water (kg/m^3)
g is gravity constant (9.81 m/s^2)
a is area of the face of the dam (m^2)
$\bar{y}$ is the depth from the water surface to the centre of the area of the dam (m).



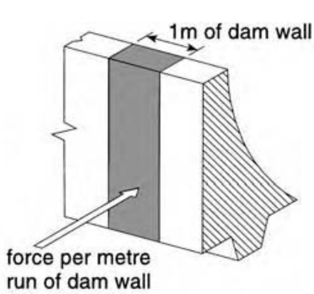
(a) The pressure diagram



(b) Location of force



(c) The dam paradox



(d) Assume dam is 1.0m long.

(e) Typical concrete dam

2.14 Designing a dam.

For the mathematically minded, a derivation of this formula can be found in most engineering hydraulics text books.

Returning to the pressure diagram, this can also be used to determine the resultant force. It is in fact equal to the area of the diagram, that is, the area of the triangle. To see how this, and the formula for force, works look at the example of how to calculate the force on a dam in the box.

The position of this force is also important. To determine the depth D from the water surface to the resultant force F (Figure 2.14b) on the dam the following formula can be used:

$$D = \frac{h^2}{12\bar{y}} + \bar{y}$$

where h is height of the dam face in contact with the water (m) and $\bar{y}$ is the depth from the water surface to the centre of the area of the dam (m).

As with the force formula, this one can also be derived from the principles of hydrostatics. The pressure diagram can also be used to determine D. The force is located at the centre of the diagram; as this is a triangle it is located two-thirds down from the apex (i.e. from the water surface).

Note that these formulae only work for simple vertical dams. When more complex shapes are involved, such as earth dams with sloping sides, then the formulae do not work. But solving the problem is not so difficult – it relies on applying the same hydrostatic principles. Most standard civil engineering texts will show you how.

EXAMPLE: CALCULATING THE FORCE ON A DAM

A farm dam is to be constructed to contain water up to 5 m deep. Calculate the force on the dam and the position of the force in relation to the water surface (Figure 2.14b).
Calculate the force F:

$$F = \rho g a \bar{y}$$

where $\rho = 1000$ kg/m^3, $g = 9.81$ m/s^2 and $a = b \times h = 1 \times 5 = 5$ m.
Note: when the length of the dam is not given assume that $b = 1$ m. The force is then the force per metre length of the dam (Figure 2.14d).

$$\bar{y} = \frac{h}{2} = 2.5 \text{ m}$$

Put all these values into the formula for F:

$$F = 1000 \times 9.81 \times 5 \times 2.5$$
$$= 122\,625 \text{ N}$$
$$F = 122.6 \text{ kNm length of the dam}$$

Using the alternative method of calculating the area of the pressure diagram:

$$\text{area of pressure diagram (triangle)} = \frac{1}{2} \times \text{base} \times \text{height}$$

$$= \frac{1}{2} \times \rho g h \times h$$
$$= \frac{1}{2} \times 1000 \times 9.81 \times 5 \times 5$$
$$F = 122.6 \text{ kNm length of the dam}$$

This produces the same answer as the formula.
To locate the force use:

$$D = \frac{h^2}{12\bar{y}} + \bar{y}$$
$$= \frac{5^2}{12 \times 2.5} + 2.5$$
$$D = 3.33 \text{ m below the water surface}$$

Using the pressure diagram method, the force is located at the centre of the triangle, which is two-thirds the depth from the apex (or the water surface):

$$D = \frac{2}{3} \times h = \frac{2}{3} \times 5$$
$$D = 3.33 \text{ m below the water surface}$$

This produces the same answer as the formula.

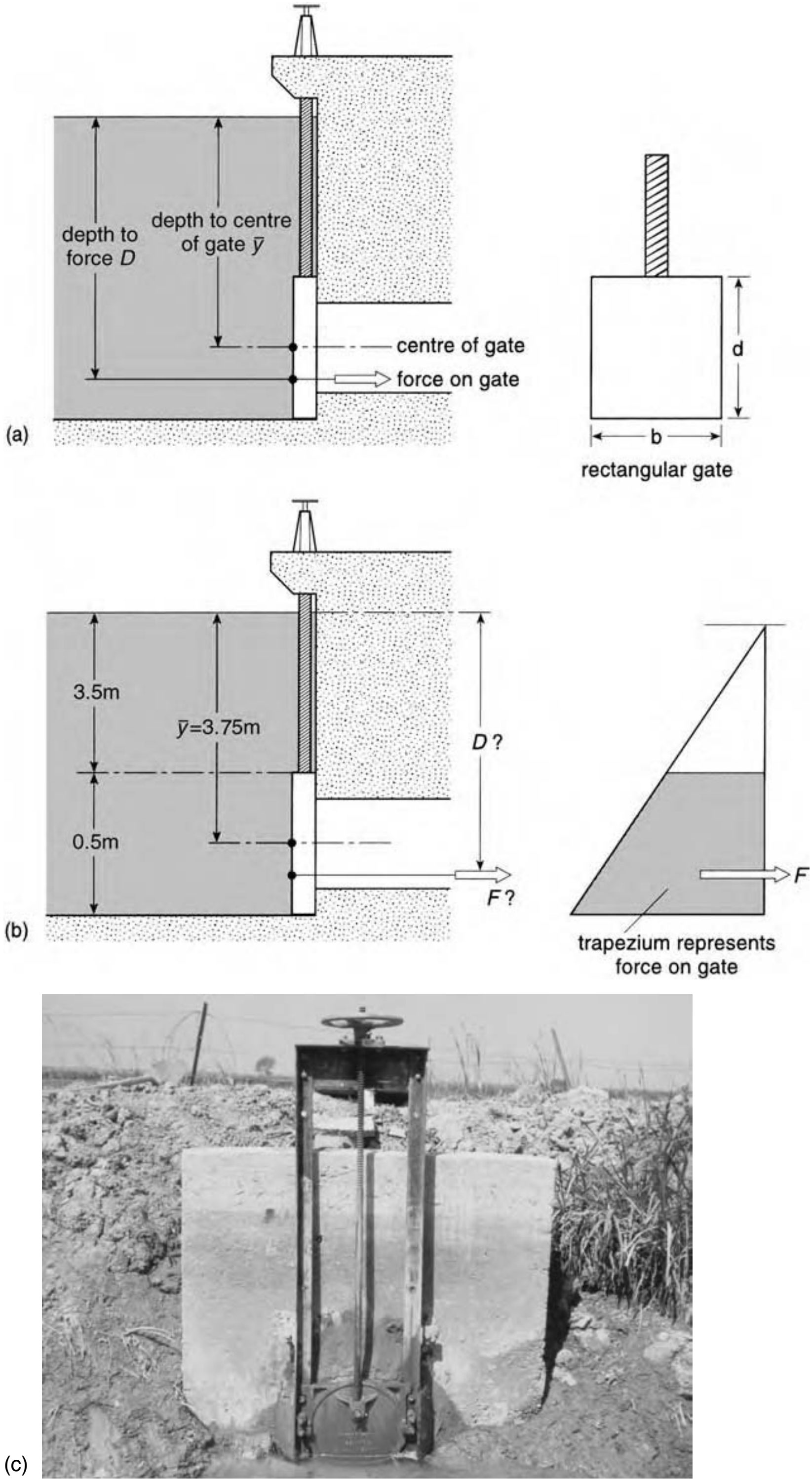
2.10.1 The dam paradox

Dams raise an interesting paradox. If two dams are built and are the same height but hold back very different amounts of water how will they differ in their hydraulic design (Figure 2.14c)? Many people would say that dam 2 would need to be much stronger than dam 1 because it is holding more water. But this is not the case. Hydraulically the design of the two dams will be the same. The force on dam 1 is the same as on dam 2 because the force depends only on the depth of water and not the amount stored. The effects of failure would obviously be more serious with dam 2 as the potential for damage and loss of life from all that extra water could be immense. The designer may introduce extra factors of safety against failure. So if you thought the forces would be different, place your trust in the well-established principles of hydrostatics and not your intuition.

2.11 Forces on sluice gates

Sluice gates are used to control the flow of water from dams into pipes and channels (see Section 7.2). They may be circular or rectangular in shape and are raised and lowered by turning a wheel on a threaded shaft (Figure 2.15a).

Gates must be made strong enough to withstand the forces created by hydrostatic pressure. The pressure also forces the gate against the face of the dam which can make it difficult to lift easily because of the friction it creates. So the greater the pressure the greater will be the force required to lift the gate. This is the reason why some gates have gears and hand-wheels fitted to make lifting easier.



2.15 Forces on sluice gates.

The force on a gate and its location can be calculated in the same way as for a dam. The force on any gate can be calculated using the same formula as was used for the dam:

$$F = \rho g a \bar{y}$$

In this case a is the area of the gate and $\bar{y}$ is the depth from the water surface to the centre of the gate. The formula for calculating D, the depth to the force, depends on the shape of the gate.
For rectangular gates:

$$D = \frac{d^2}{12\bar{y}} + \bar{y}$$

where d is depth of gate (m), $\bar{y}$ is depth from water surface to centre of the gate (m). Note: in this case d is the depth of the gate (m) and *not* the depth of water behind the dam.
For circular gates:

$$D = \frac{r^2}{4\bar{y}} + \bar{y}$$

where r is radius of the gate (m).

The depth D from the water surface to the force F must not be confused with $\bar{y}$. D is the depth to the point where the force acts on the gate. It is always greater than $\bar{y}$.

The force and its location can also be obtained from the pressure diagram but in this case it is only that part of the diagram in line with the gate that is of interest. The force on the gate can be calculated from the area of the trapezium and its location is at the centre of the trapezium. This can be found by using the principle of moments. But if you are not so familiar with moments, the centre can be found by cutting out a paper shape of the trapezium and freely suspending it from each corner in turn and drawing a vertical line across the shape. The point where all the lines cross is the centre. A common mistake is to assume that depth D is two-thirds of the depth from the water surface. It is true for a simple dam but not for a sluice gate.

The above equations cover most hydraulic sluice gate problems, but occasionally gates of different shapes may be encountered and they may also be at an angle rather than vertical. It is possible to work out the forces on such gates, but more difficult. Other hydraulic text books will show you how if you are curious enough. An example of calculating the force and its location on a hydraulic gate is shown in the box.

EXAMPLE: CALCULATING THE FORCE ON A SLUICE GATE

A rectangular sluice gate controls the release of water from a reservoir. If the gate is 0.5 m × 0.5 m and located 3.5 m below the water surface calculate the force on the gate and its location below the water surface (Figure 2.15b).

First calculate the force F on the gate

$$F = \rho g a \bar{y}$$

where:

a = area of the gate = 0.5 × 0.5 = 0.25 m^2

$\bar{y}$ = depth from water surface to the centre of the gate
= 3.5 + 0.25 = 3.75 m

$$F = 1000 \times 9.81 \times 0.25 \times 3.75$$
$$F = 8580 \text{ N or } 8.58 \text{ kN}$$

Next calculate depth from water surface to where force F is acting:

$$D = \frac{d^2}{12\bar{y}} + \bar{y}$$
$$= \frac{0.25}{12 \times 3.75} + 3.75$$
$$D = 3.76 \text{ m}$$

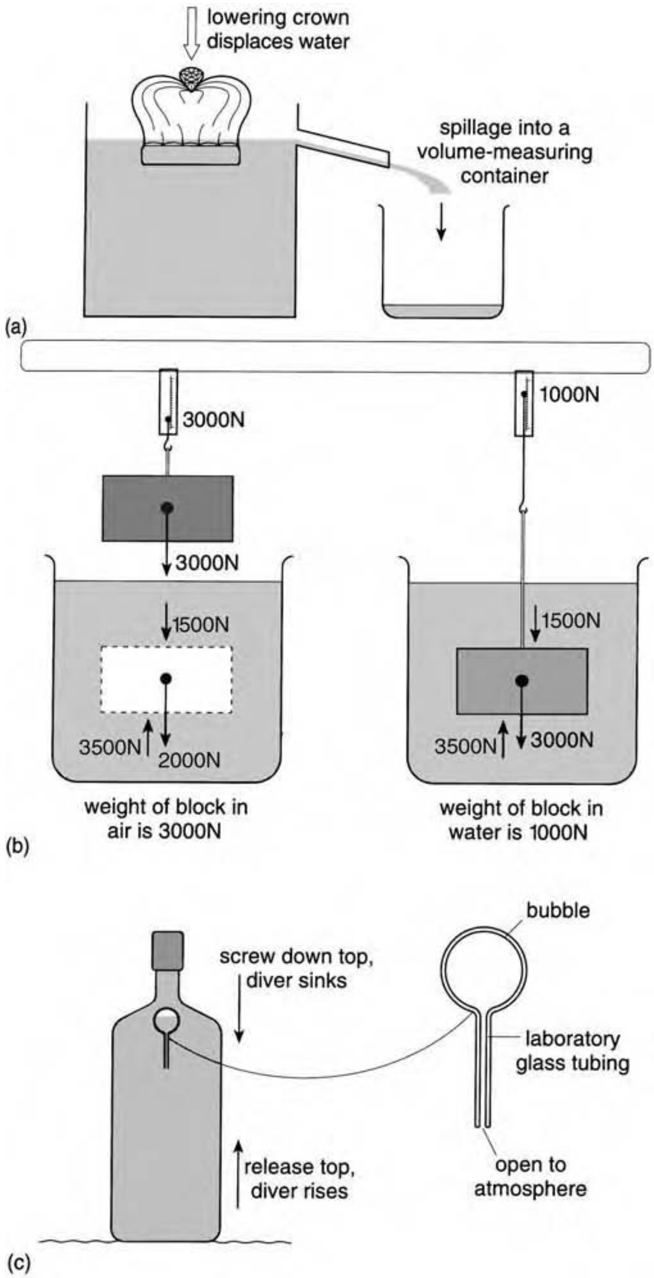
2.12 Archimedes' principle

Returning now to Archimedes who first set down the basic rules of hydrostatics. His most famous venture seems to have been in the public baths in Greece around the year 250BC. He allegedly ran naked into the street shouting 'Eureka!' – he had discovered an experimental method of detecting the gold content of the crown of the King of Syracuse. He realised that when he got into his bath, the water level rose around him because his body was displacing the water and that this was linked to the feeling of weight loss – that uplifting feeling everyone experiences in the bath. As the baths were usually public places he probably noticed as well that smaller people displaced less water than larger ones. It is at this point that many people draw the wrong conclusion. They assume that this has something to do with a person's weight. This is quite wrong – it is all about their volume. To explain this, let us return to the king's crown.

Perhaps the king had two crowns that looked the same in every way but one was made of gold and he suspected that someone had short-changed him by making the other of a mixture of gold and some cheaper metal. The problem that he set Archimedes was to tell him which was the gold one. Weighing them on a normal balance in air would not have provided the answer because a clever forger would make sure that both crowns were the same weight. If, however, he could measure their densities he would then know which was gold because the density of gold has a fixed value (19 300 kg/m^3) and this would be different from that of the crown of mixed metals. But to determine their densities their volumes must first be known. If the crowns were simple shapes such as cubes then it would be easy to calculate their volume. But crowns are not simple shapes and it would have been almost impossible to measure them accurately enough for calculation purposes. This is where immersing them in water helps.

The crowns may have weighed the same in air but when Archimedes weighed the crowns immersed in water he observed that they had different weights. Putting this another way, each crown experienced a different loss in weight due to the buoyancy effect of the water. It is this *loss in weight* that was the key to solving the mystery. By measuring the loss in weight of the crowns, Archimedes was indirectly measuring their volumes.

To understand this, imagine a crown is immersed in a container full of water up to the overflow pipe (Figure 2.16a). The crown displaces the water, spilling it down the overflow where it is caught in another container. The volume of the spillage water can easily be measured and it has exactly the same volume as the crown. But the most interesting point is that the weight of the spillage water (water displaced) is equal to the loss in weight of the crown. So by measuring the loss in



2.16 Archimedes' principle.

weight Archimedes was in fact measuring the weight of displaced water, that is, the weight of an equal volume of water. As the density of water is a fixed value (9810 N/m^3) it is a simple matter to convert this weight of water into a volume and so determine the density of the crown.

This is the principle that Archimedes discovered: when an object is immersed in water it experiences a loss in weight and this is equal to the weight of water it displaces.

What Archimedes measured was not actually the density of gold but its relative density or specific gravity as it is more commonly known. This is the density of gold relative to that of water and he calculated this using the formula:

$$\text{specific gravity} = \frac{\text{weight of crown}}{\text{weight loss when immersed in water}}$$

This may not look like the formula for specific gravity in Section 1.13.2 but it is the same. From Section 1.13.2:

$$\text{specific gravity} = \frac{\text{weight of an object}}{\text{weight of an equal volume of water}}$$

But Archimedes' principle states that:

weight loss when immersed in water = weight of an equal volume of water

So the two formulae are in fact identical and Archimedes was able to tell whether the crown was made of gold or not by some ingenious thinking and some simple calculations. The method works for all materials and not just gold, also for all fluids and not just water. Indeed, this immersion technique is now a standard laboratory method for measuring the volume of irregular-shaped objects and for determining their specific gravity.

Still not convinced? Try this example with numbers. A block of material has a volume of 0.2 m^3 and is suspended on a spring balance (Figure 2.16b) and weighs 3000 N. When the block is lowered into the water it displaces 0.2 m^3 of water. As water weighs 10 000 N/m^3 (approximately) the displaced water weighs 2000 N (i.e. 0.2 $m^3 \times$ 10 000 N/m^3). Now according to Archimedes the weight of this water should be equal to the weight loss by the block and so the spring balance should now be reading only 1000 N (i.e. 3000 N−2000 N).

To explain this, think about the space that the block (0.2 m^3) will occupy when it is lowered into the water (Figure 2.16b). The 'space' is currently occupied by 0.2 m^3 of water weighing 2000 N. Suppose that the water directly above the block weighs 1500 N (note that any number will do for this argument). These two weights of water added together are 3500 N and this is supported by the underlying water and so there is an upward balancing force of 3500 N. The block is now lowered into the water and it displaces 0.2 m^3 of water. The water under the block takes no account of this change and continues to push upwards with a force of 3500 N and the downward force of the water above it continues to exert a downward force of 1500 N. The block thus experiences a net upward force or a loss in weight of 2000 N (i.e. 3500 N−1500 N). This is exactly the same value as the weight of water that was displaced by the block. The reading on the spring balance is reduced by this amount from 3000 N down to 1000 N.

A simple but striking example of this apparent weight loss is to tie a length of cotton thread around a brick and try to suspend it first in air and then in water. If you try to lift the brick in air the thread will very likely break. But the uplift force when the brick is in water means that the brick can now be lifted easily. It is this same apparent loss in weight that enables rivers to move great boulders during floods and the sea to move shingle along the beach.

2.12.1 Floating objects

When an object such as a cork floats on water it appears that the object has *lost* all of its weight. If the cork was held below the water surface and then released it rises to the surface. This is because the weight of the water displaced by the cork is greater than the weight of the cork itself and so the cork rises under the unbalanced force. Once at the surface the weight of the cork is balanced by the lifting effect of the water. In this case the water displaced by the cork *is not a measure of its volume but a measure of its weight*.

Another way of determining if an object will float is to measure its density. When the density is less than that of water it will float. When it is greater it will sink. A block of wood, for example, is half the density of water and so it floats half submerged. Icebergs, which have a density close to that of water, float with only one tenth of their mass above the surface. The same principle also applies to other fluids. Hydrogen balloons for example, rise in air because hydrogen is 14 times less dense than air.

Steel is six times denser than water and so it will sink. People laughed when it was first proposed that ships could be made of steel and would float. But today we just take such things for granted. Ships float because even when loaded, much of their volume is filled with relatively light cargo and a lot of air space and so their average density is less than that of sea water.

Buoyancy is also affected by the density of seawater, which varies considerably around the world. In Bombay the sea is more salty than it is near Britain so ships ride higher in the water in Bombay. If a ship is loaded with cargo in Bombay and is bound for London, as it nears the UK it will lie much lower in the water and this could be dangerous if it is overloaded.

The Cartesian diver is an interesting example of an object, which can either sink or float by slightly varying its density a little above or below that of water (Figure 2.16c). The diver is really a small length of glass tubing, sealed and blown into a bubble at one end and open at the other. You can make one easily in a laboratory using a bunsen burner and a short length of glass tube. Next find a bottle with a screw top, fill it to the top with water and put the diver into the water. The diver will float because the air bubble ensures the average density is less than that of water. Now screw down the top and the diver will sink. This is because this action increases the water pressure, which compresses the air in the diver and increases its average density above that of water. Releasing the screw top allows the diver to rise to the surface again. This same principle is used to control submarines. When a submarine dives, its tanks are allowed to fill with water so that its average density is greater than that of water. The depth of submergence is determined by the extent to which its tanks are flooded. To make the submarine rise, water is blown out of its tanks using compressed air.

To summarise:

An object floats when it is less dense than water but sinks when it is denser than water. When an object floats it displaces water equal to its weight.

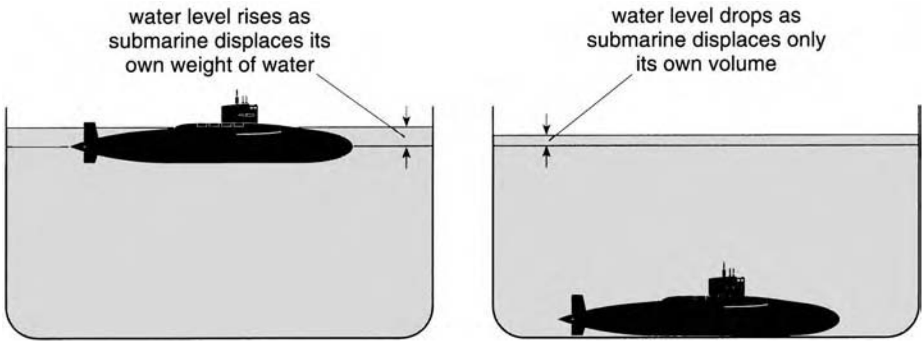
2.12.2 Applying the principle

Here are two problems to test your understanding of Archimedes' principle.

A submarine is floating in a lock (Figure 2.17). *It then submerges and sinks to the bottom. What happens to the water level in the lock? Does it rise or fall?*

Archimedes' principle says that when an object floats it displaces its own weight of water and when it sinks it displaces its own volume.

Applying this to the submarine – when it is floating on the water surface, the submarine displaces its own weight of water which will be substantial because submarines are heavy. But when the submarine sinks to the bottom it only displaces water equivalent to its volume. So the amount of water displaced by the floating submarine will be much greater than the volume of water displaced when it is submerged. This means that when the submarine dives the water level in the lock will drop (very slightly!).



2.17 Applying Archimedes' principle.

When ice is added to a tank of water, the water level rises (Figure 2.7b). When the ice melts what happens to the water level? Does it rise, fall or stay the same?

Ice is a solid object that floats and so it should behave in the same way as any other solid object. When it melts, however, it becomes part of the water and, in effect, it sinks.

To see what happens let's take 1.0 litre of water with a mass of 1 kg and weight 10 N and freeze it. Water expands as it freezes and so as it turns into ice its volume will increase by approximately 8% to 1.08 litres of ice. But remember it is still only 1.0 litre of water and so its weight has not changed – just its volume. If the block of ice is now put into the tank it will float on the water because the density of the ice is slightly lower than that of the water. The water level in the tank will also rise as a result of the displacement by the ice – like any solid object that floats the ice displaces its own weight of water, which is still 10 N. Now 10 N of water has a volume of 1.0 litre and so 1.0 litre of water will be displaced. It has nothing to do with the volume of the ice – only its weight. However, when the ice melts it 'sinks' into the tank and like any other object that sinks it displaces its own volume of water. But the volume of the melted ice (now water again) is 1.0 litre. So the displacement in each case is the same – 1.0 litre – which means the water level in the tank remains unchanged when the ice melts.

There is a lot of discussion about what happens to sea levels when polar ice melts as a result of global warming. Clearly if the ice is floating, which is the case with a lot of ice at the north pole, then melting will not change the sea level. If the ice melts off the land then of course it will cause the sea level to rise.

It is the volume of the ice that can mislead your thinking because it changes significantly when the water freezes. Do not be misled by this, just follow the principle of Archimedes and everything will work out right.

2.12.3 Drowning in quicksand: myth or reality?

A common scene in many adventure films is of someone stumbling into a patch of quicksand and getting sucked under. Great drama but is this what really happens? Recent research at the Ecole Normal Supérieure in Paris, based partly on Archimedes' principle, suggests otherwise. Apparently the work was inspired by a holiday trip to the legendary quicksands at Daryacheh -ye Namak salt lake, near Qom in Iran, where local shepherds speak of complete camels disappearing without trace.

Quicksand is a mixture of fine sand, clay and salt water, in which the grains are delicately balanced and very unstable. This makes the mixture appear solid but once it is disturbed it starts to behave like a liquid and so if you stand on it you will very easily start to sink into it. But just like any liquid there is a buoyancy effect (Archimedes' principle) and so how far you will sink depends on your density. So if you float in water you will also float in quicksand.

But the problem is not just a hydraulic one. The research showed that when the mixture liquefies the sand and clay fall to the bottom and create thick sediment that also helps to prevent you sinking further. So the good news is that the two combined mean that you are unlikely to sink much beyond your waist – not such good news if you fell in head first. Struggling and kicking will not make you sink further – it just makes the mixture more unstable and so you will sink faster. The bad news comes when you try to get out because the mixture will hold you fast. It can take as much force to pull you out of quicksand as it does to lift a typical family car. So you are more likely to have your limbs pulled off than get out of the mess! So how do you get out? One suggestion by the researchers is to gently wriggle your feet to liquefy the mixture and then slowly pull yourself up a few millimetres at a time. The myth surrounding quicksand probably originates from people falling in head first and in such circumstances you are most likely to drown. Science also spoils a good story – it means that all those film dramas about quicksand such as *The Hound of the Baskervilles* are pure fantasy!

2.13 Some examples to test your understanding

1. Determine the pressure in kN/m^2 for a head of a) 14 m of water and b) 1.7 m of oil. Assume the mass density of water is 1000 kg/m^3 and oil is 785 kg/m^3 (137.34 kN/m^2, 13.09 kN/m^2).
2. A storage tank, 2.3 m long by 1.2 m wide and 0.8 m deep is full of water. Calculate (a) the mass of water in the tank, (b) the pressure on the bottom of the tank, (c) the force on the end of the tank and (d) the position of this force below the water surface (2210 kg, 7848 N/m^2, 3767 N, 0.53 m below the water surface).
3. Calculate atmospheric pressure in kN/m^2 when the barometer reading is 750 mm of mercury. Calculate the height of a water barometer needed to measure atmospheric pressure (100.06 kN/m^2, 10.2 m).
4. Calculate the pressure in kN/m^2 and in m head of water in a pipeline carrying water using a mercury manometer when $h_1 = 0.5$ m and $h_2 = 1.2$ m. Assume the specific gravity of mercury is 13.6 (155 kN/m^2, 15.82 m).
5. A vertical rectangular sluice gate 1.0 m high by 0.5 m wide is used to control the discharge from a storage reservoir. Calculate the horizontal force on the gate and it location in relation to the water surface when the top of the gate is located 2.3 m below the water surface (13.73 kN, 2.83 m).
6. Calculate the force and its location below the water surface on a 0.75 m diameter circular sluice gate located when the top of the gate is located 2.3 m below the water surface (11.57 kN, 2.69 m).

3 Hydrodynamics: when water starts to flow

3.1 Introduction

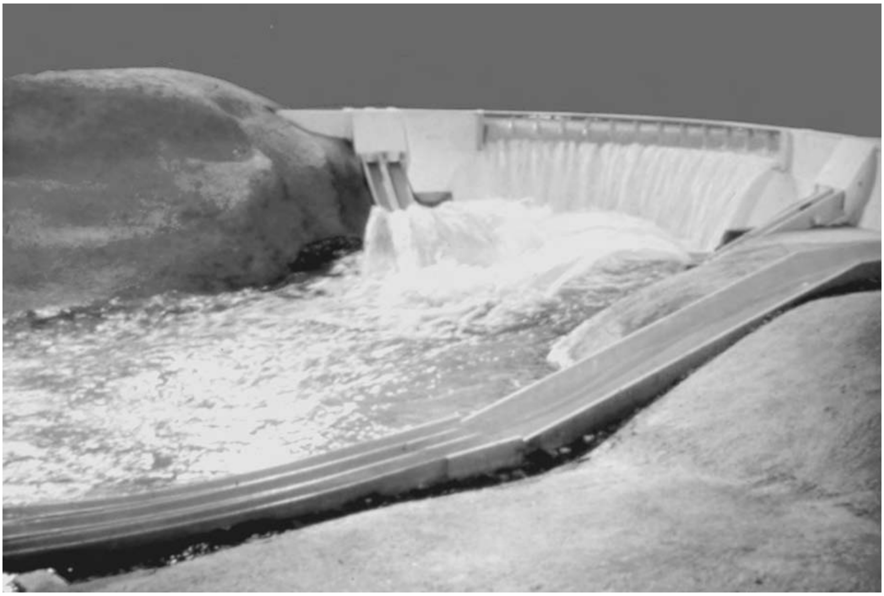
Hydrodynamics is the study of water flow. It helps us to understand how water behaves when it flows in pipes and channels and to answer such questions as – what diameter of pipe is needed to supply a village or a town with water? How wide and deep must a channel be to carry water from a dam to an irrigation 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:

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

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:

$$Q_1 = Q_2$$

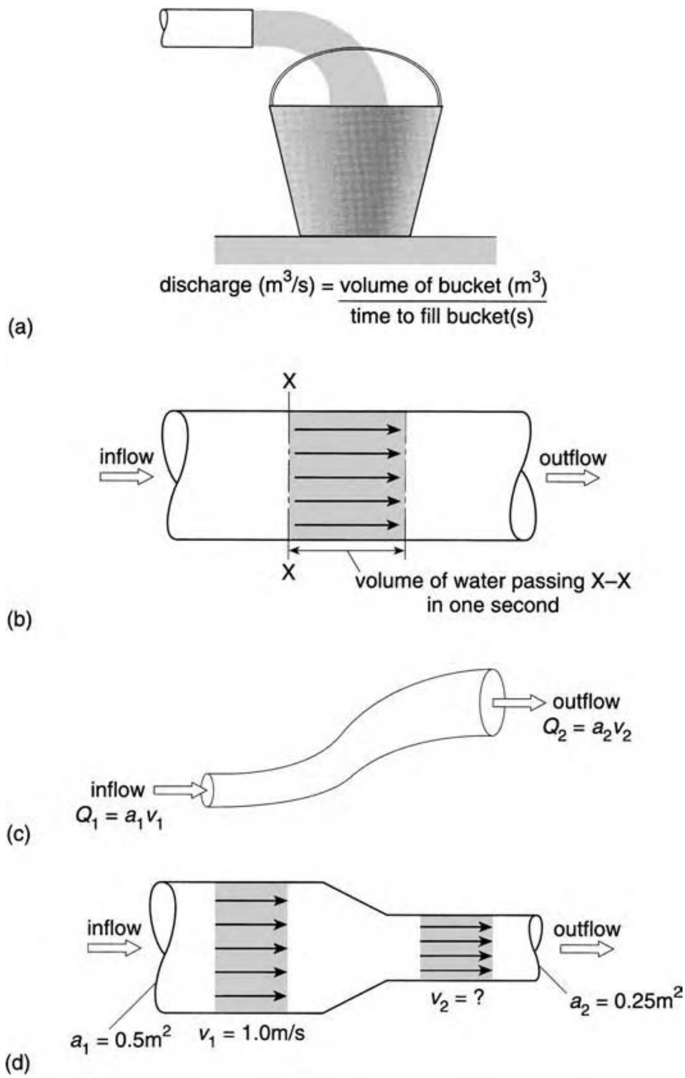
But from the discharge equation:

$$Q = va$$

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.

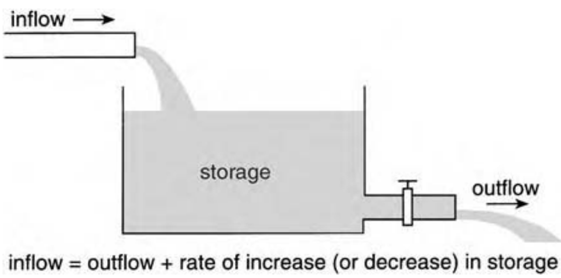
Use the continuity equation:

inflow = outflow

And so:

$$v_1 a_1 = v_2 a_2$$
$$1 \times 0.5 = v_2 \times 0.25$$
$$v_2 = 2 \text{ m/s}$$

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).

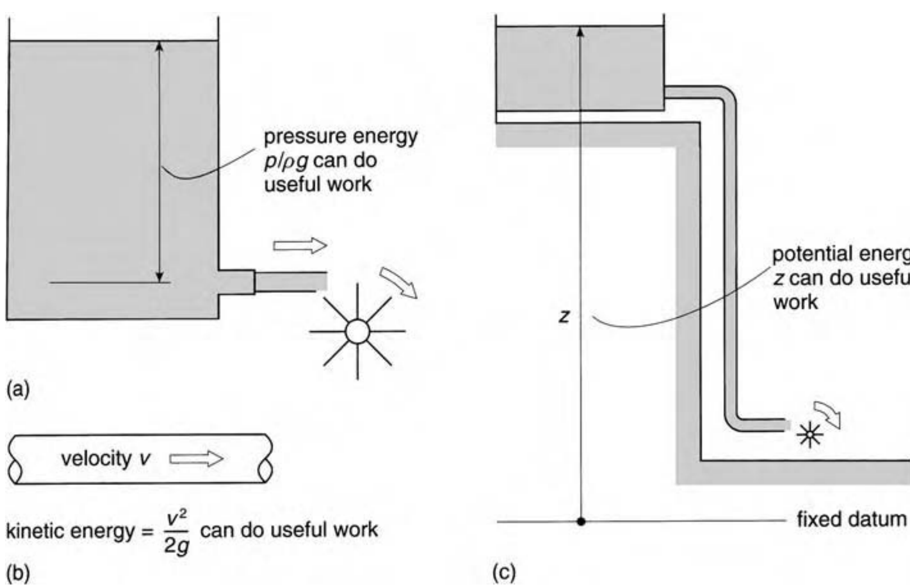
Notice that the equation for pressure energy is actually the same as the familiar pressure-head equation (remember). It is just presented in a different way. So pressure energy is in fact the same as the pressure head and is measured in metres (m).

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:

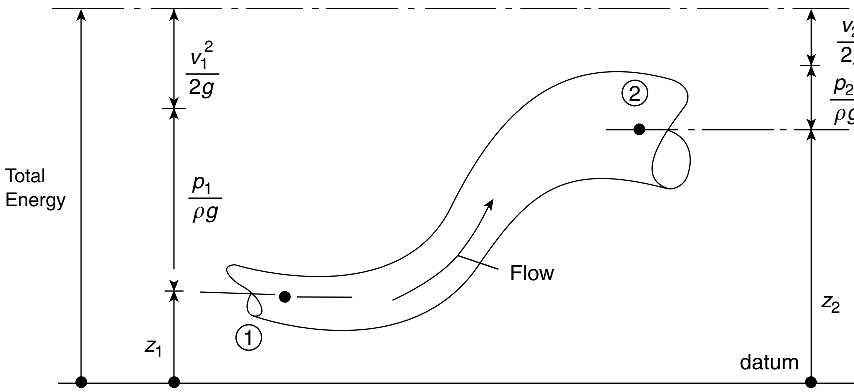
$$\text{potential energy} = z$$

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

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

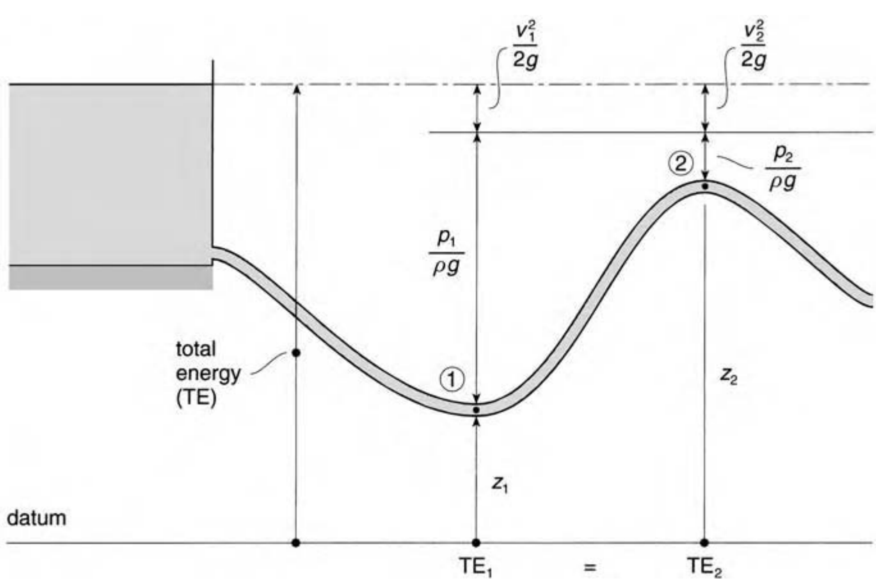
z_1 and z_2 are measured from some chosen horizontal datum.

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:

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

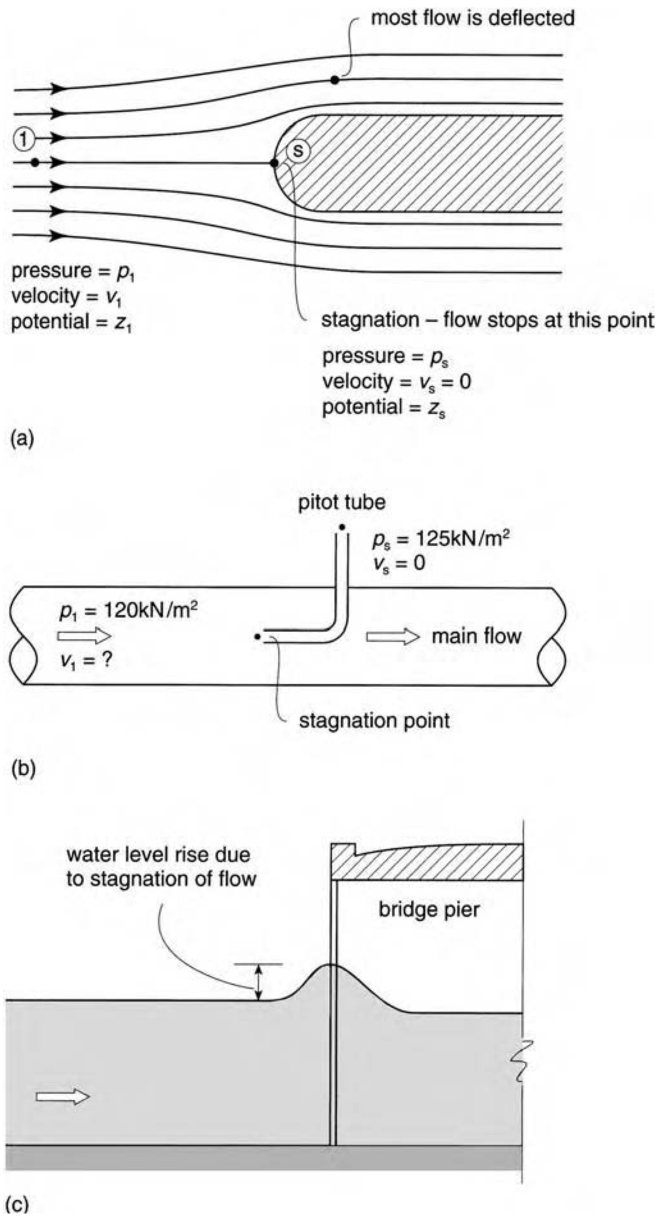
Rearranging this equation to bring all the velocity and pressure terms together:

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

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:

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

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

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

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

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

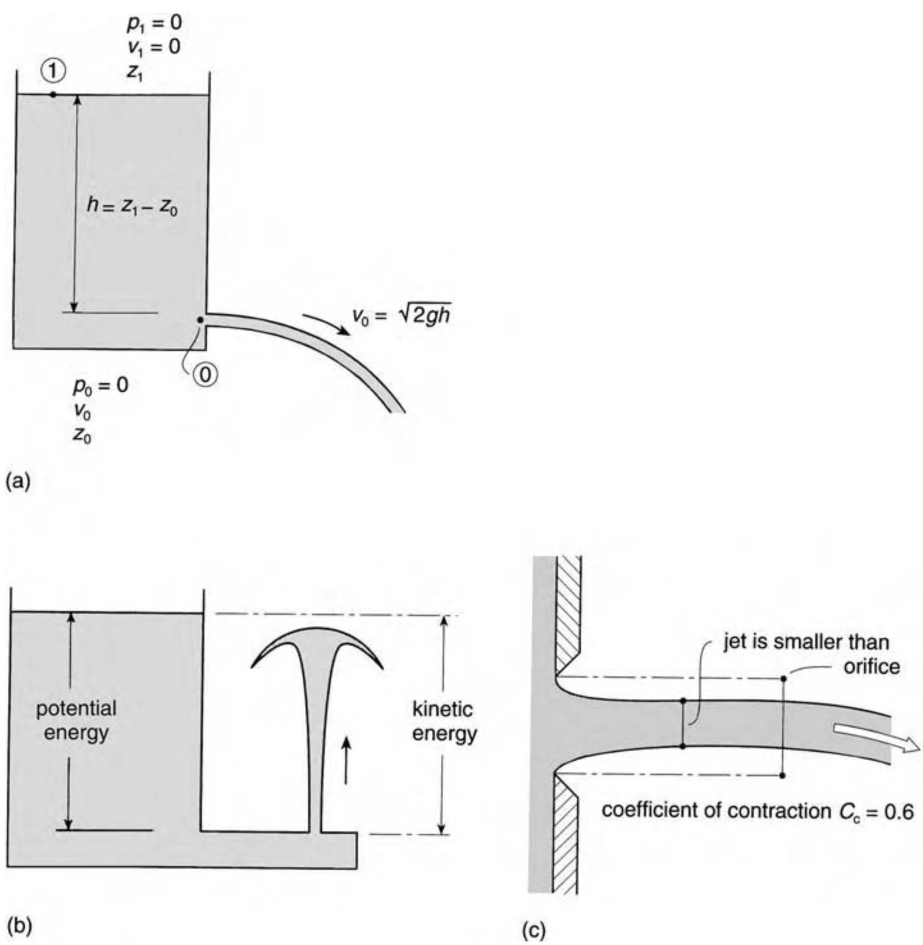
3.6.3 Orifices

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*

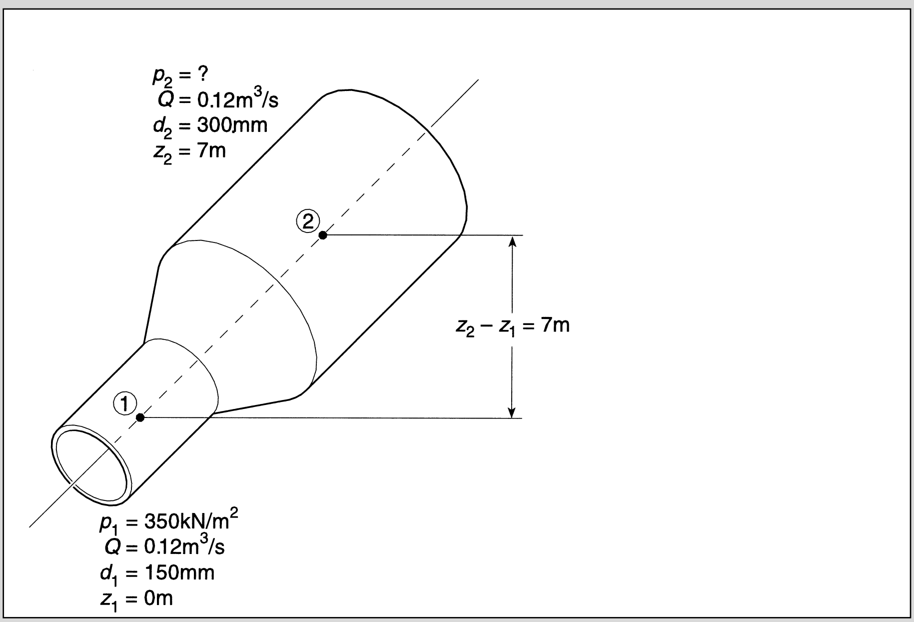
A more general and very practical application of the energy equation is to predict pressures and velocities in pipelines as a result of changes in ground elevation and pipe sizes. An example in the box shows just how versatile this equation can be.

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

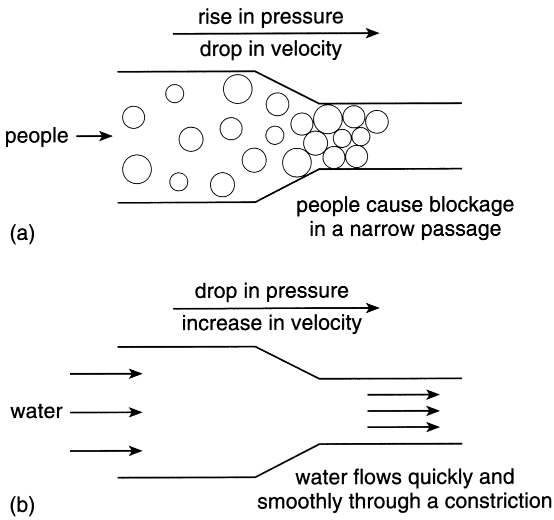
3.7.1 Flow through narrow openings

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).



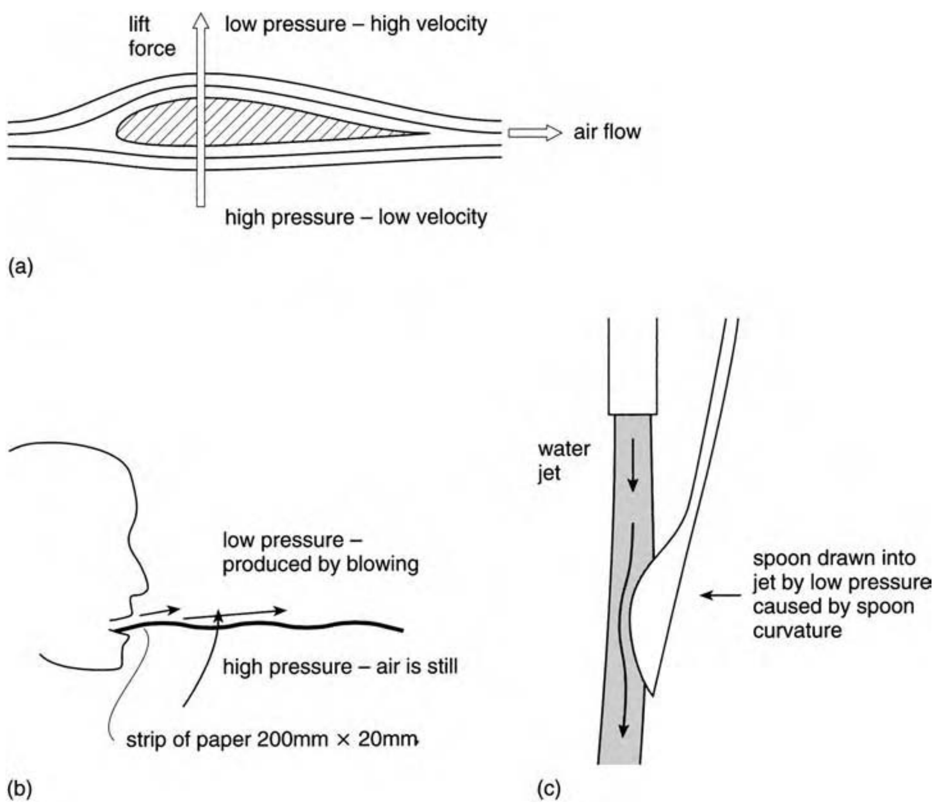
3.10 People and water flow differently through narrow passages.

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.

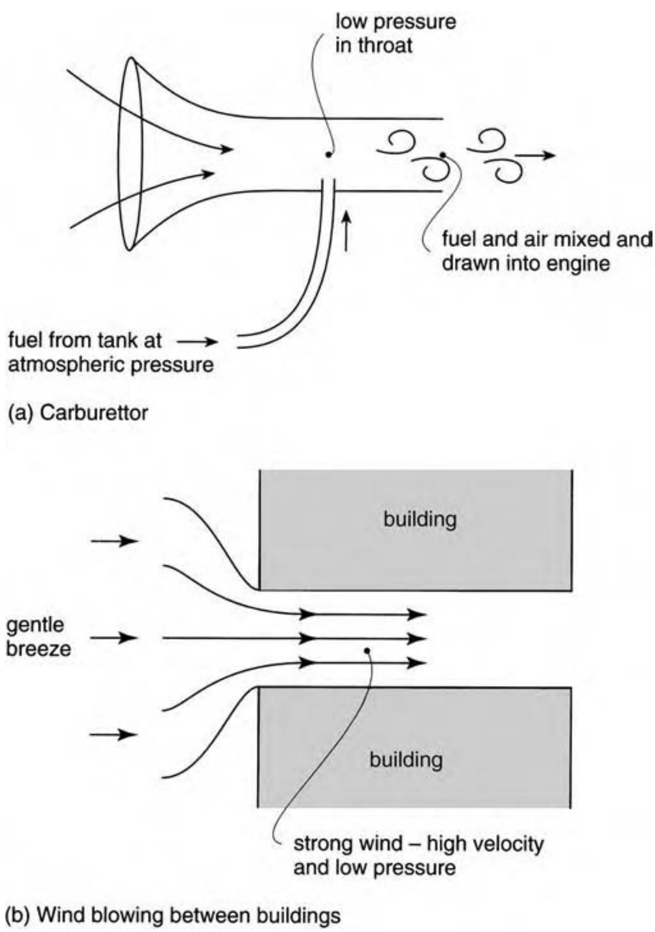
The same principle is used in reverse on racing cars. In this case the wing is upside down and located on the back of the car. The velocity of the air flowing over it, due to the forward movement of the car, produces a downward thrust which holds the car firmly on the road. The faster the car the greater 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:

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

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:

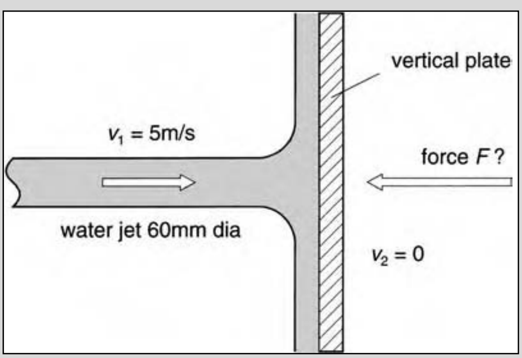


Figure 3.13 Applying momentum.

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

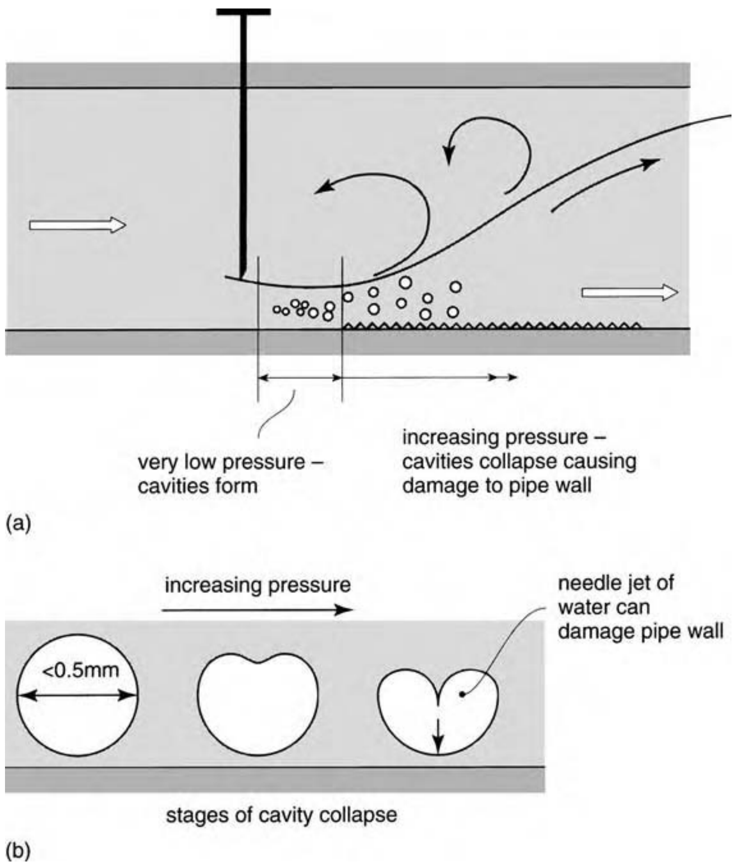
Real fluids suffer from cavitation and it can cause lots of problems, particularly in pumps and control valves. It occurs when a fluid is moving very fast; as a consequence, the pressure can drop to very low values approaching zero (vacuum pressure).

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

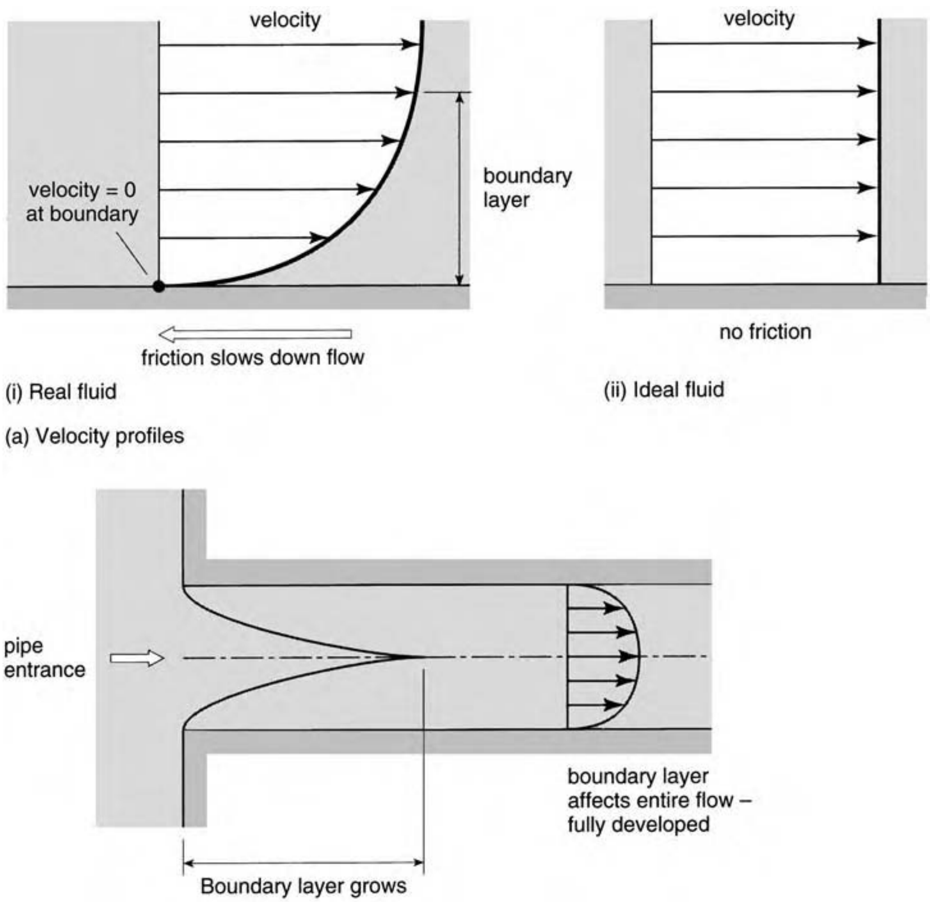


3.14 Dangers of cavitation.

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



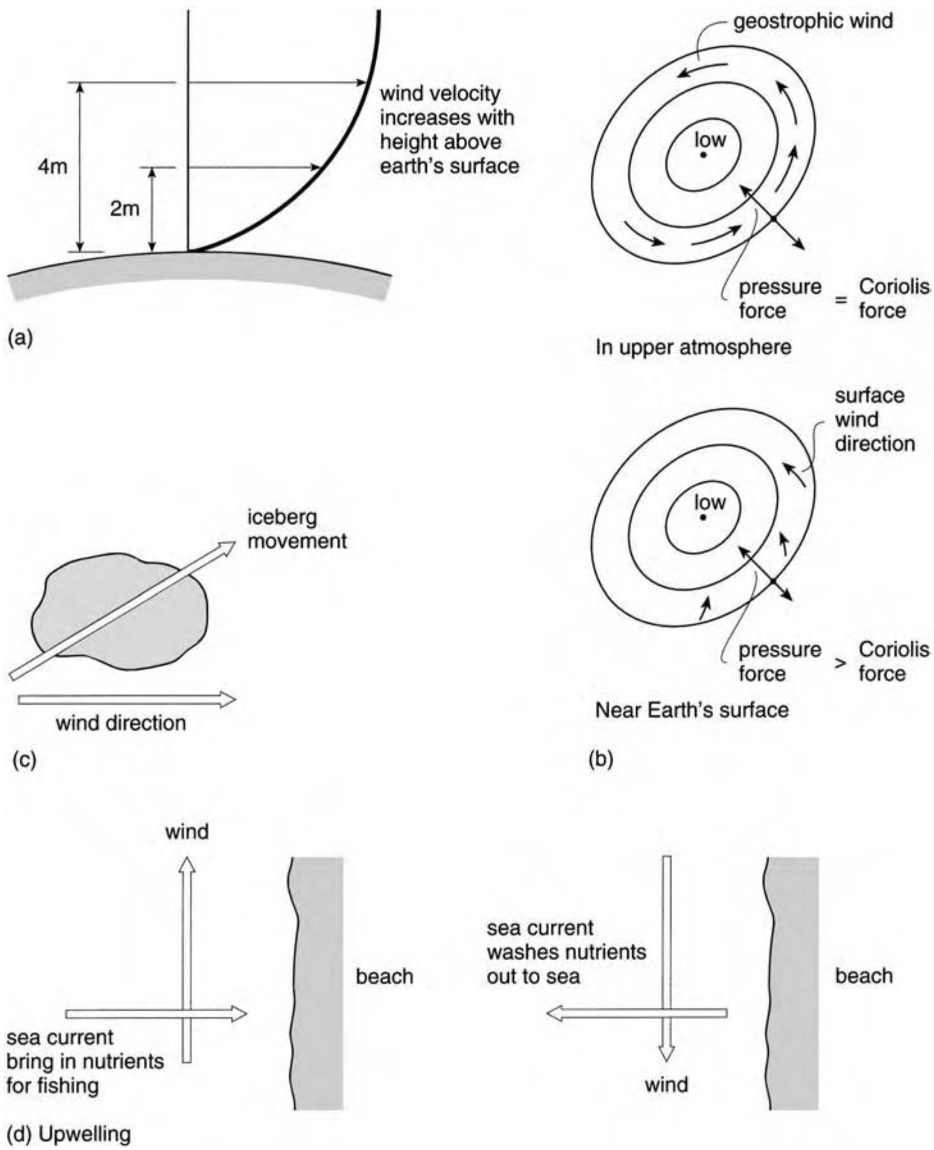
3.15 Boundary effects.

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.

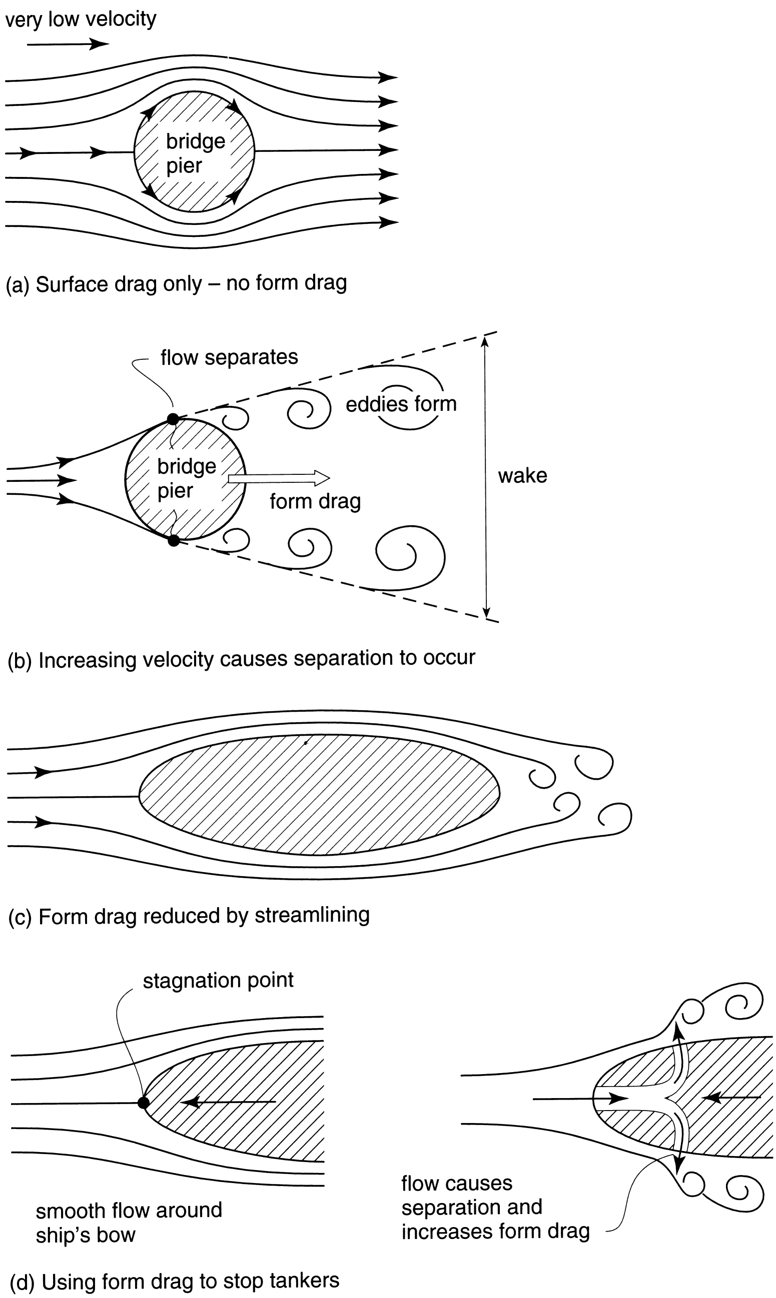
3.10 Drag forces

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:

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

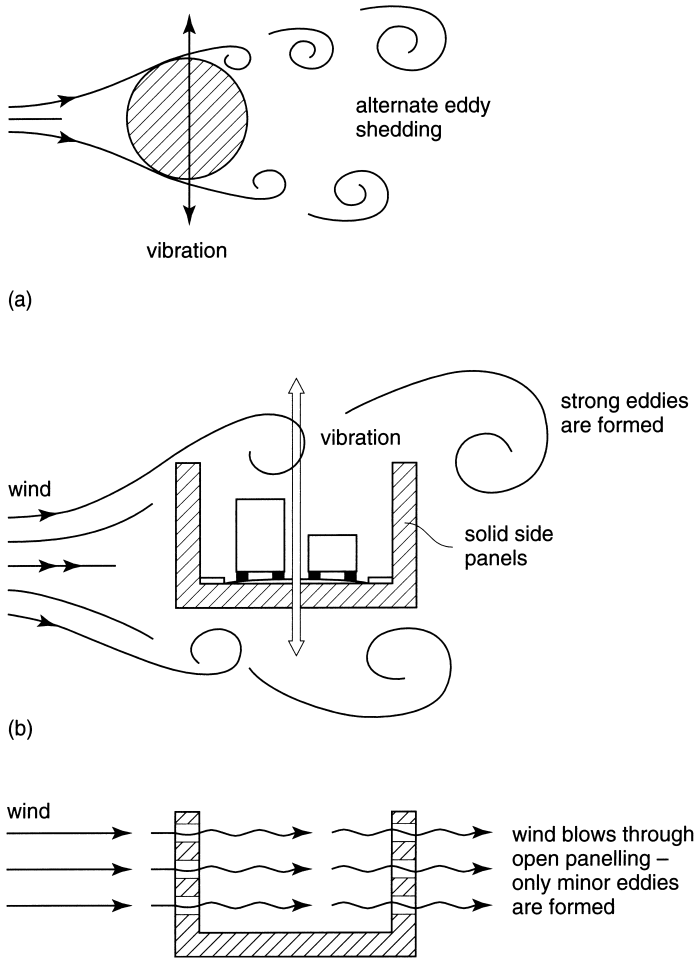
where ρ is fluid density (kg/m^3), a is the cross-sectional area (m^2), v is velocity (m/s) and C is drag coefficient. The coefficient C is dependent on the shape of the body, the velocity of the flow and the density of the fluid.

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.

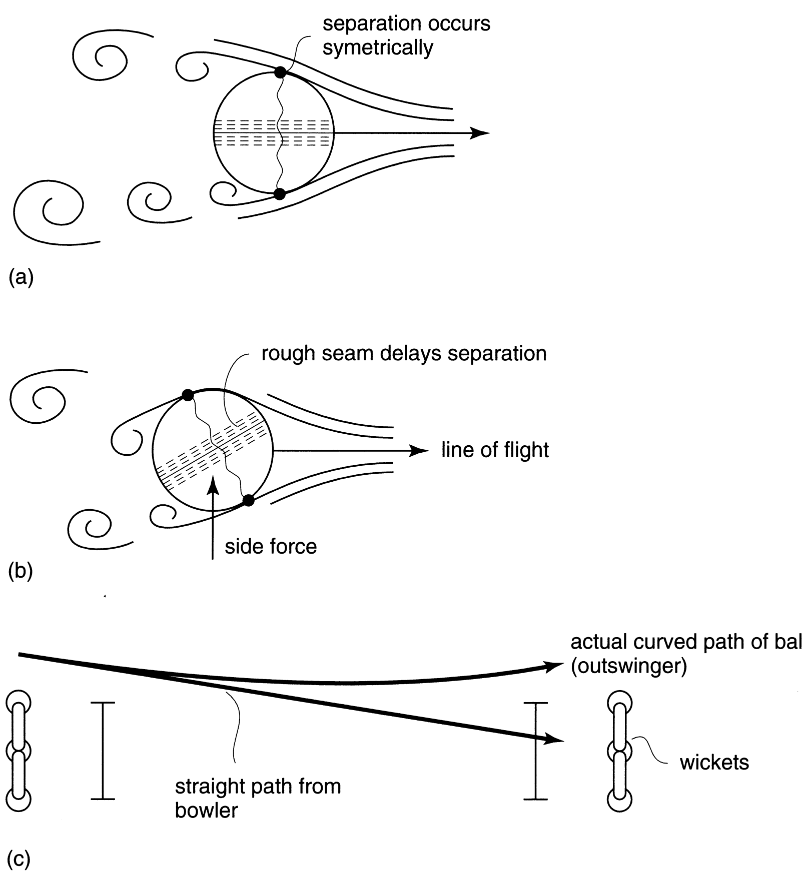
A similar problem can occur around tall chimneys when eddies are shed in windy conditions. To avoid large eddies forming a perforated sleeve or a spiral collar is placed around the top of the chimney. This breaks up the flow into lots of small eddies which are usually quite harmless.

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).

4 Pipes

4.1 Introduction

Pipes are a common feature of water supply systems and have many advantages over open channels. They are completely enclosed, usually circular in section and always flow full of water. This is in contrast to channels which are open to the atmosphere and can have many different shapes and sizes – but more about channels in Chapter 5. One big advantage of pipes is that water can flow uphill as well as downhill so land topography is not such a constraint when taking water from one location to another.

There are occasions when pipes do not flow full – one example is gravity flow sewers. They take sewage away from homes and factories and often only flow partially full under the force of gravity in order to avoid pumping. They look like pipes and are indeed pipes but hydraulically they behave like open channels. The reason pipes are used for this purpose is that sewers are usually buried below ground to avoid public health problems and it would be difficult to bury an open channel!

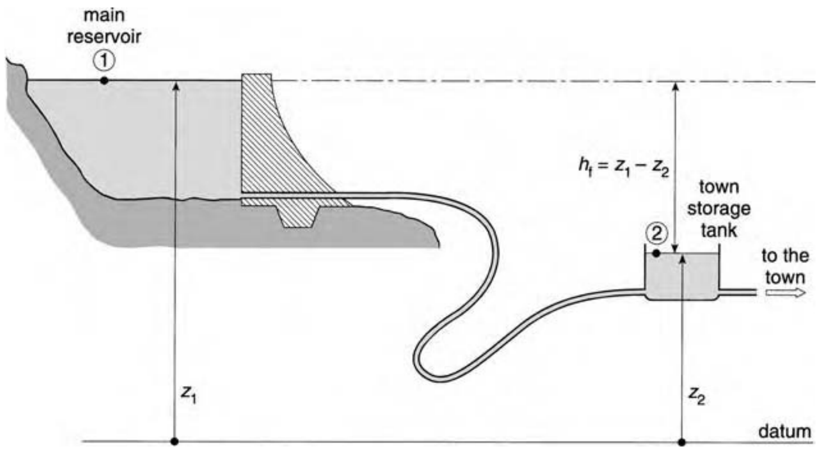
4.2 A typical pipe flow problem

Pipe flow problems usually involve calculating the right size of pipe to use for a given discharge. A typical example is a water supply to a village (Figure 4.1). A pipeline connects a main storage reservoir to a small service (storage) tank just outside the village which then supplies water to individual houses. The required discharge (Q m^3/s) for the village is determined by the water demand of each user and the number of users being supplied. We now need to determine the right size of pipe to use to ensure that this discharge is supplied from the main storage reservoir to the service tank.

A formula to calculate pipe size would be ideal. However, to get there we first need to look at the energy available to 'push' water through the system, so the place to start is the energy equation. But this is a real fluid problem and so energy losses due to friction must be taken into account. So writing the energy equation for two points in this system – point 1 is at the main reservoir and point 2 at the service tank – and allowing for the energy loss as water flows between the two:

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

Points 1 and 2 are carefully chosen in order to simplify the equation and also the solution. Point 1 is at the surface of the main reservoir where the pressure p_1 is atmospheric pressure and



4.1 A typical pipe flow problem.

so equal to zero (remember we are working in gauge pressures). Point 2 is also at the water surface in the service tank so p_2 is zero as well. The water velocities v_1 and v_2 in the reservoir and the tank are very small and so the kinetic energy terms are also very small and can be assumed to be zero. This leaves just the potential energy terms z_1 and z_2 and the energy loss term h_f. So the energy equation simplifies down to:

$$h_f = z_1 - z_2$$

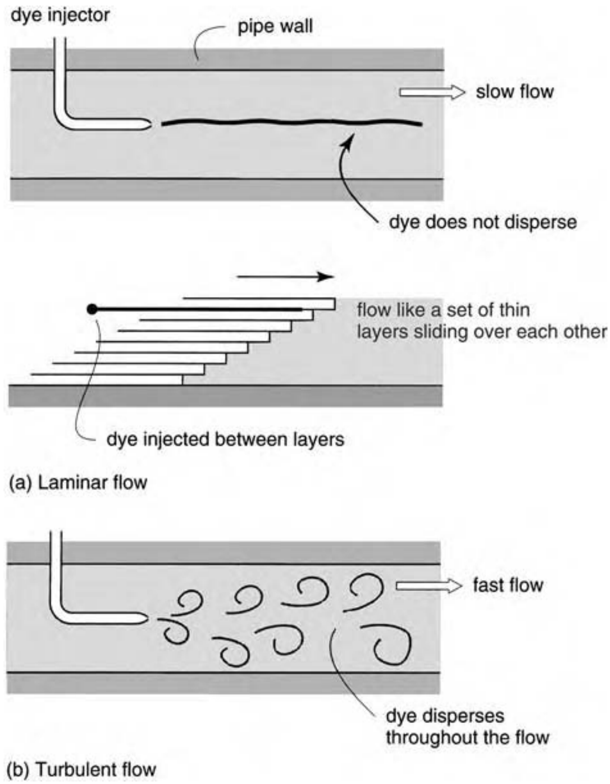
$z_1 - z_2$ is the difference in water levels between the reservoir and the storage tank and this represents the energy available to 'push' water through the system. h_f is the energy loss due to friction in the pipe. The energy available is usually known and so this means we also know the amount of energy that can be lost through friction. The question now is – is there a formula that links this energy loss h_f with the pipe diameter? The short answer is yes – but it has taken some 150 years of research to sort this out. So let us first step through this bit of history and see what it tells us about pipe flow.

4.3 A formula to link energy loss and pipe size

Some of the early research work on pipe friction was done by Osborne Reynolds (1842–1912), a mathematician and engineer working at the University in Manchester in UK. He measured the pressure loss in pipes of different lengths and diameters at different discharges with some interesting results. At low flows he found that the energy loss varied directly with the velocity. So when the velocity was doubled the energy loss also doubled. But at high flows the energy loss varied as the square of the velocity. So when the velocity was doubled the energy loss increased four-fold. Clearly, Reynolds was observing two quite different types of flow. This thinking led to Reynolds classic experiment that established the difference between what is now referred to as *laminar* and *turbulent* flow and formulae which would enable the energy loss to be calculated for each flow type from a knowledge of the pipes themselves.

4.3.1 Laminar and turbulent flow

Reynolds experiment involved setting up a glass tube through which he could pass water at different velocities (Figure 4.2). A thin jet of coloured dye was injected into the flow so that the flow patterns were visible.



4.2 Laminar and turbulent flow.

When the water moved slowly the dye remained in a thin line as it followed the flow path of the water down the pipe. This was described as *laminar flow*. It was as though the water was moving as a series of very thin layers – like a pack of cards – each one sliding over the other, and the dye had been injected between two of the layers. This type of flow rarely exists in nature and so is not of great practical concern in hydraulics. However, you can see it occasionally under very special conditions. Examples include smoke rising in a thin column from a chimney on a very still day or a slow flow of water from a tap that looks so much like a glass rod that you feel you could get hold of it. Blood flow in our bodies is usually laminar.

The second and more common type of flow he identified was *turbulent flow*. This occurred when water was moving faster. The dye was broken up as the water whirled around in a random manner and was dissipated throughout the flow. Turbulence was a word introduced by Lord Kelvin (1824–1907) to describe this kind of flow behaviour.

There are very clear visual differences between laminar and turbulent flow but what was not clear was how to predict which one would occur in any given set of circumstances. Velocity was obviously important. As velocity increased so the flow would change from laminar to turbulent flow. But it was obvious that from the experiments that velocity was not the only factor. It was Reynolds who first suggested that the type of flow depended not just on velocity (v) but also on mass density (ρ), viscosity (μ) and pipe diameter (d). He put these factors together in a way which is now called the *Reynolds Number* in recognition of his work.

$$\text{Reynolds No.} = \frac{\rho v d}{\mu}$$

Note that Reynolds Number has no dimensions. All the dimensions cancel out. Reynolds found that he could use this number to reliably predict when laminar and turbulent flow would occur.

$R < 2000$ flow would always be laminar
$R > 4000$ flow would always be turbulent

Between $R = 2000$ and 4000 he observed a very unstable zone as the flow seemed to jump from laminar to turbulent and back again as if the flow could not decide which of the two conditions it preferred. This is a zone to avoid as both the pressure and flow fluctuate widely in an uncontrolled manner.

Reynolds Number also shows just how important is viscosity in pipe flow. Low Reynolds Number ($R < 2000$) means that viscosity (μ) is large compared with the term ρvd. So viscosity is important in laminar flow and cannot be ignored. High Reynolds Number ($R > 4000$) means viscosity is small compared with the ρvd term and so it follows that viscosity is less important in turbulent flow. This is the reason why engineers ignore the viscosity of water when designing pipes and channels as it has no material effect on the solution. Ignoring viscosity also greatly simplifies pipeline design.

It has since been found that Reynolds Number is very useful in other ways besides telling us the difference between laminar and turbulent flow. It is used extensively in hydraulic modelling (physical models – not mathematical models) for solving complex hydraulic problems. When a problem cannot be solved using some formula, another approach is to construct a small-scale model in a laboratory and test it to see how it performs. The guideline for modelling pipe systems (or indeed any fully enclosed system) is to ensure that the Reynolds Number in the model is similar to the Reynolds Number in the real situation. This ensures that the forces and velocities are similar so that the model, as near as possible, produces similar results to those in the real pipe systems.

Although it is useful to know that laminar flow exists it is not important in practical hydraulics for designing pipes and channels and so only turbulent flow is considered in this text. Turbulent flow is very important to us in our daily lives. Indeed it would be difficult for us to live if it was not for the mixing that takes place in turbulent flow which dilutes fluids. When we breathe out, the carbon dioxide from our lungs is dissipated into the surrounding air through turbulent mixing. If it did not disperse in this way we would have to move our heads to avoid breathing in the same gases as we had just breathed out. Car exhaust fumes are dispersed in a similar way, otherwise we could be quickly poisoned by the intake of concentrated carbon monoxide.

4.3.2 A formula for turbulent flow

Several formulae link energy loss with pipe size for turbulent flow but one of the most commonly used today is that devised by Julius Weisbach (1806–1871) and Henry Darcy (1803–1858). It is often referred to as the *Darcy–Weisbach equation*:

$$h_f = \frac{\lambda l v^2}{2gd}$$

where λ is a friction factor; l is pipe length (m); v is velocity (m/s); g is gravity constant (9.81 m/s^2) and d is pipe diameter (m).

This formula shows that energy loss depends on pipe length, velocity and diameter but also on the friction between the pipe and the flow as represented by λ:

- *Length* has a direct influence on energy loss. The longer the pipeline the greater the energy loss.
- *Velocity* has a great influence on energy loss because it is the square of the velocity that counts. When the velocity is doubled (say by increasing the discharge), the energy loss increases four-fold. It is usual practice in water supply systems to keep the velocity below 1.6 m/s. This is done primarily to avoid excessive energy losses but it also helps to reduce water hammer problems (see Chapter 5, Section 5.9).
- *Pipe diameter* has the most dramatic effect on energy loss. As the pipe diameter is reduced so the energy losses increase, not only because of the direct effect of d in the formula but also because of its effect on the velocity v (remember the discharge equation $Q = va$). The overall effect of reducing the diameter by half (say from 300 to 150 mm) is to increase h_f by 32-times. See box below for an illustration of this.
- *Pipe friction* λ Unfortunately this is not just a simple measure of pipe roughness; it depends on several other factors which are discussed more fully in the next section.

Take care when using the Darcy–Weisbach formula as some text books, particularly American, use f as the friction factor and not and they are not the same. The link between them is $\lambda = 4f$.

EXAMPLE: HOW PIPE DIAMETER AFFECTS ENERGY LOSS

A pipeline 1000 m long carries a flow of 100 l/s. Calculate the energy loss when the pipe diameter is 0.3 m, 0.25 m, 0.2 m and 0.15 m and $\lambda = 0.04$.

The first step is to calculate the velocities for each pipe diameter using the discharge equation:

$$Q = va$$

And so:

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

Use this equation to calculate velocity v for each diameter and then use the Darcy–Weisbach equation to calculate h_f. The results are shown in Table 4.1. Notice the very large rise in head loss as the pipe diameter is reduced. Clearly the choice of pipe diameter is a critical issue in any pipeline system.

Table 4.1 Results of Darcy–Weisbach equation to calculate h_f.

Diameter (m)	*Pipe area (m^2)*	*Velocity (m/s)*	*Head loss h_f (m)*
0.30	0.07	1.43	13.6
0.25	0.049	2.04	33.3
0.20	0.031	3.22	103.7
0.15	0.018	5.55	418.6

4.4 The λ story

It would be convenient if λ was just a constant number for a given pipe that depended only on its roughness and hence its resistance to the flow. But few things are so simple and λ is no exception. Some of the earliest work on pipe friction was done by Paul Blazius in 1913. He carried out a wide range of experiments on different pipes and different flows and came to the conclusion that λ depended only on the Reynolds Number and surprisingly, the roughness of the pipe seemed to have no effect at all on friction.

From this he developed a formula for λ (note *R* is Reynolds Number):

$$\lambda = \frac{0.316}{R^{0.25}}$$

Another investigator was Johann Nikuradse who may well have been puzzled by the Blazius results. He set up a series of laboratory experiments in the 1930s with different pipe sizes and flows and he roughened the inside of the pipes with sand grains of a known size in order to create different but known roughness. His data showed that values of λ were independent of Reynolds Number and depended only on the roughness of the pipe. Clearly, either someone was wrong or they were both right and each was looking at something different.

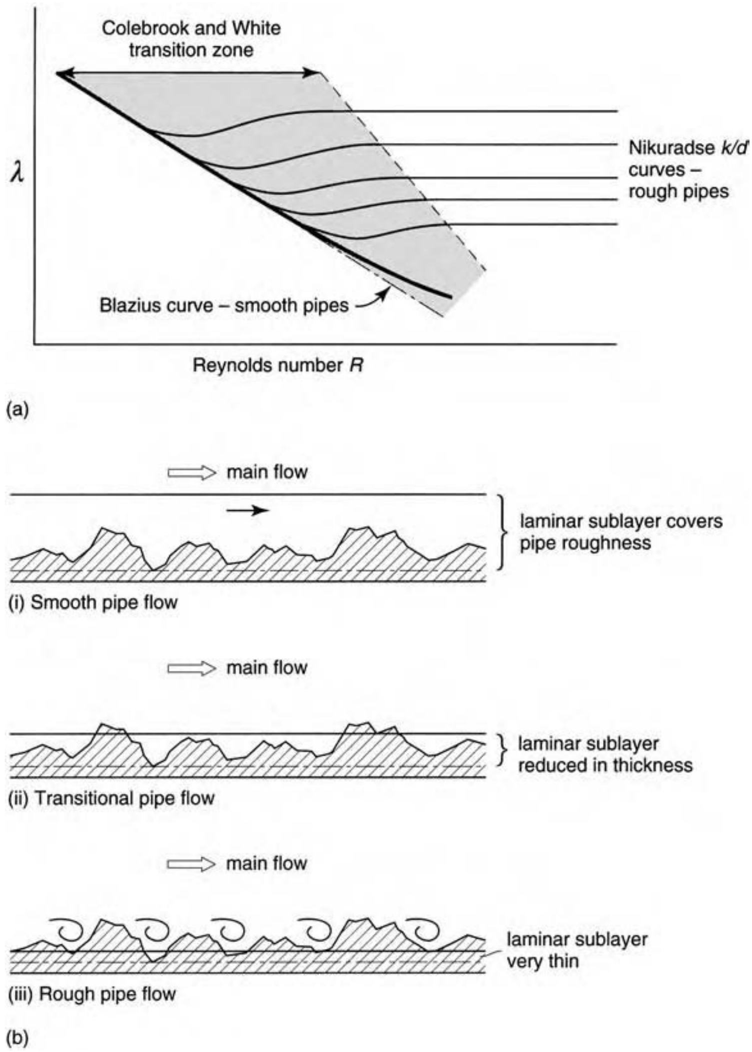
4.4.1 Smooth and rough pipes

We now know that both investigators were right but they were looking at different aspects of the same problem. Blazius was looking at flows with relatively low Reynolds Numbers (4000 to 100 000) and his results refer to what are now called *smooth pipes*. Nikuradse's experiments dealt with high Reynolds Number flows (greater than 100 000) and his results refer to what are now called *rough pipes*. Both Blazius and Nikuradse results are shown graphically in Figure 4.3a. This is a graph with a special logarithmic scale for Reynolds Number so that a wide range of values can be shown on the same graph. It shows how λ varies with both Reynolds Number and pipe roughness which is expressed as the height of the sand grains (*k*) divided by the pipe diameter (*d*). The Blazius formula produces a single line on this graph and is almost a straight line.

The terms rough and smooth refer as much to the flow conditions in pipes as to the pipes themselves and so, paradoxically, it is possible for the same pipe to be described as both rough and smooth. Roughness and smoothness are also relative terms. How the inside of a pipe feels to touch is not a good guide to its smoothness in hydraulic terms. Pipes which are smooth to the touch can still be quite rough hydraulically. However, a pipe that feels rough to touch will be very rough hydraulically and very high energy losses can be expected.

As there are two distinct types of flow it implies that there must be some point or zone where the flow changes from one to the other. This is indeed the case. It is not a specific point but a zone known as the *transition zone* when λ depends on both Reynolds Number and pipe roughness (Figure 4.3a). This zone was successfully investigated by C.F. Colebrook and C.M. White working at Imperial College in London in the 1930s and they developed a formula to cover this flow range. This is not quoted here as it is quite a complex formula and in practice there is no need to use it because it has now been simplified to design charts. These can be used to select pipe sizes for a wide range of hydraulic conditions. The use of typical pipe charts is described later in this chapter in Section 4.8.

The transition zone between smooth and rough pipe flow should *not* be confused with the transition zone from laminar to turbulent flow, as is often done. The flow is fully turbulent for all smooth and rough pipes and the transition is from smooth to rough pipe flow.



4.3 The λ story.

To summarise the different flows in pipes:

laminar flow
↓
transition from laminar to turbulent flow
(this zone is very unstable and should be avoided)
↓
turbulent flow
smooth pipe flow
↓
transition from smooth to rough pipe flow
↓
rough pipe flow

4.4.2 A physical explanation

Since those early experiments, modern scientific techniques have enabled investigators to look more closely at what happens close to a pipe wall. This has resulted in a physical explanation for smooth and rough pipe flow (Figure 4.3b). Investigators have found that even when the flow is turbulent there exists a very thin layer of fluid – less than 1 mm thick – close to the boundary that is laminar. This is called the laminar sub-layer. At low Reynolds Numbers the laminar sub-layer is at its thickest and completely covers the roughness of the pipe. The main flow is unaffected by the boundary roughness and is influenced mainly by viscosity in the laminar sub-layer. It seems that the layer covers the roughness like a blanket and protects the flow from the pipe wall. This is the smooth pipe flow that Blazius investigated. As Reynolds Number increases, the laminar sub-layer becomes thinner and roughness elements start to protrude into the main flow. The flow is now influenced both by viscosity and pipe roughness. This is the transition zone. As Reynolds Number is further increased the sub-layer all but disappears and the roughness of the pipe wall takes over and dominates the friction. This is rough pipe flow which Nikuradse investigated.

Commercially manufactured pipes are not artificially roughened with sand like experimental pipes, they are manufactured as smooth as possible to reduce energy losses. For this reason they tend to come within the transition zone where λ varies with both Reynolds Number and pipe roughness.

4.5 Hydraulic gradient

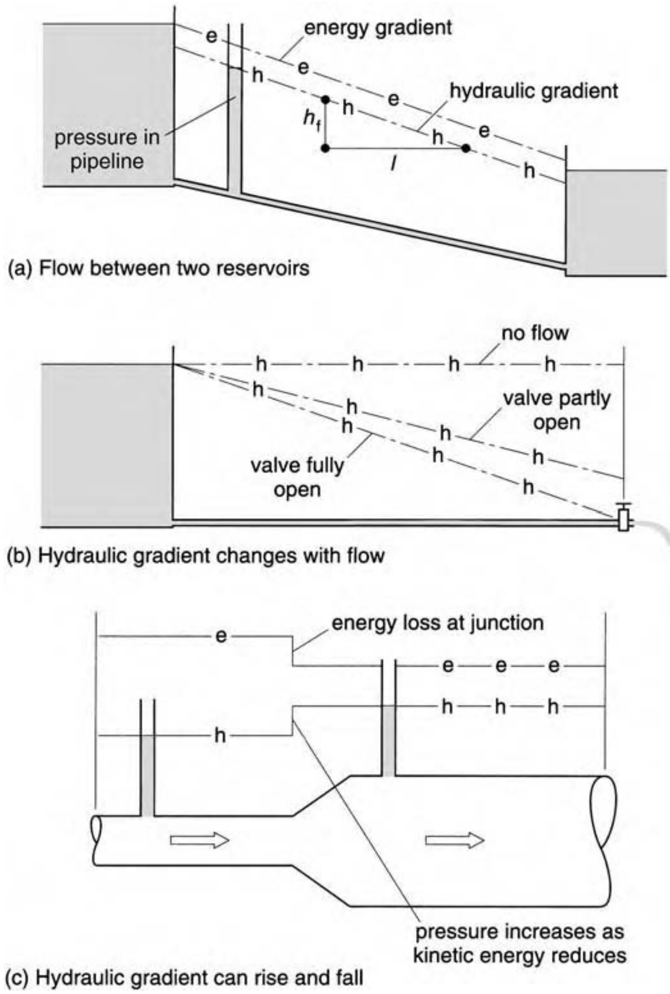
One way of showing energy losses in a pipeline is to use a diagram (Figure 4.4a). The total energy is shown as a line drawn along the pipe length and marked *e—e—e*. This line always slopes downwards in the direction of the flow and demonstrates that energy is continually being lost through friction. It connects the water surfaces in the two tanks. There is a small step at the downstream tank to represent the energy loss at the outlet from the pipeline into the tank. Note that the energy line is not necessarily parallel to the pipeline. The pipeline usually just follows the natural ground surface profile.

Although total energy is of interest, pressure is more important because this determines how strong the pipes must be to avoid bursts. For this reason a second line is drawn below the energy line, but parallel to it, to represent the pressure (pressure energy) and is marked *h—h—h*. This shows the pressure change along the pipeline. Imagine standpipes are attached to the pipe. Water would rise up to this line to represent the pressure head (Figure 4.4a). The difference between the two lines is the kinetic energy. Notice how both the energy line and the hydraulic gradient are straight lines. This shows that the rate of energy loss and the pressure loss are uniform (at the same rate). The slope of the pressure line is called the *hydraulic gradient*. It is calculated as follows:

$$\text{hydraulic gradient} = \frac{h_f}{l}$$

where h_f is change in pressure (m); l is the pipe length over which the pressure change takes place (m).

The hydraulic gradient has no dimensions as it comes from dividing a length in metres by a head difference in metres. However, it is often expressed in terms of metres head per metre length of pipeline. As an example a hydraulic gradient of 0.02 means for every one metre of pipeline there will be a pressure loss of 0.02 m. This may also be written as 0.02 m/m or as 2 m/100 m of pipeline. This reduces the number of decimal places that must be dealt with and means that for



4.4 Hydraulic gradient.

every 100 m of pipeline 2 m of head is lost through friction. So if a pipeline is 500 m long (there are five 100 m lengths) the pressure loss over 500 m will be 5 × 2 = 10 m head.

The hydraulic gradient is not a fixed line for a pipe; it depends on the flow (Figure 4.4b). When there is no flow the gradient is horizontal but when there is full flow the gradient is at its steepest. Adjusting the outlet valve will produce a range of gradients between these two extremes.

The energy gradient can only slope downwards in the direction of flow to show how energy is lost, but the hydraulic gradient can slope upwards as well as downwards. An example of this is a pipe junction when water flows from a smaller pipe into a larger one (Figure 4.4c). As water enters the larger pipe the velocity reduces and so does the kinetic energy. Although there is some energy loss when the flow expands (this causes the energy line to drop suddenly) most of the loss of kinetic energy is recovered as pressure energy and so the pressure rises slightly.

Two more points of detail about the energy and hydraulic gradients (Figure 4.4a). At the first reservoir, the energy gradient starts at the water surface but the hydraulic gradient starts just below it. This is because the kinetic energy increases as water enters the pipe so there is a corresponding drop in the pressure energy. As the flow enters the second reservoir the energy line is just above the water surface. This is because there is a small loss in energy as the flow expands from the pipe into the reservoir. The hydraulic gradient is located just below the water level because there is still some kinetic energy in the flow. When it enters the reservoir this changes back to pressure energy. The downstream water level represents the final energy condition in the system. These changes close to the reservoirs are really very small in comparison to the friction losses along the pipe and so they play little or no part in the design of the pipeline.

Normally pipelines are located well below the hydraulic gradient. This means that the pressure in the pipe is always positive – see the standpipes in Figure 4.4a. Even though it may rise and fall as it follows the natural ground profile, water will flow as long as it is always below the hydraulic gradient and provided the outlet is below the inlet. There are limits to how far below the hydraulic gradient a pipeline can be located. The further below the higher will be the pressure in the pipe and the risk of a burst if the pressure exceeds the limits set by the pipe manufacturer.

4.6 Energy loss at pipe fittings

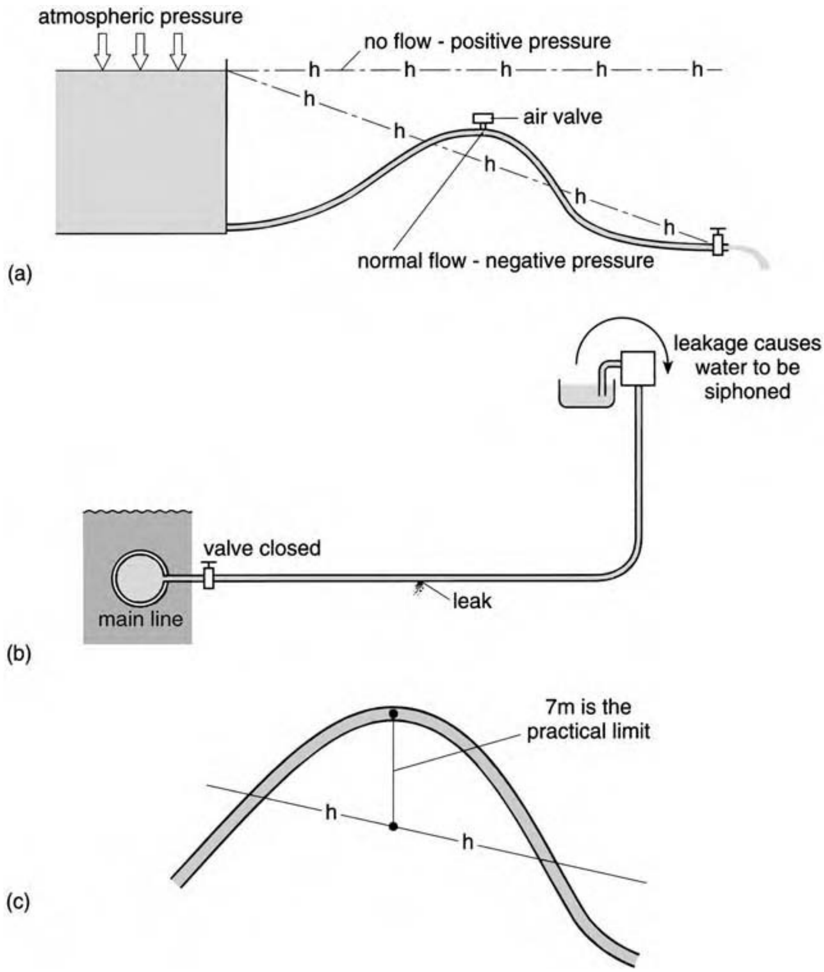
Although there is an energy loss at the pipe connection with the reservoir in Figure 4.4a this is usually very small in comparison with the loss in the main pipeline and so it is often ignored. Similar losses occur at pipe bends, reducers, pipe junctions and valves and although each one is small, together they can add up. They can all be calculated individually but normal design practice is to simply increase the energy loss in the main pipeline by 10% to allow for all these minor losses.

4.7 Siphons

Siphon is the name given to sections of pipe that rise above the hydraulic gradient. Normally pipes are located well below the hydraulic gradient and this ensures that the pressure is always positive and so well above atmospheric pressure. Under these conditions water flows freely under gravity provided the outlet is lower than the inlet (Figure 4.4a). But when part of a pipeline is located above the hydraulic gradient, even though the outlet is located below the inlet, water will not flow without some help (Figure 4.5a). This is because the pressure in the section of pipe above the hydraulic gradient is negative.

4.7.1 How they work

Before water will flow, all the air must be taken out of the pipe to create a vacuum. When this happens atmospheric pressure on the open water surface pushes water into the pipe to fill the vacuum and once it is full of water it will begin to flow. Under these conditions the pipe is working as a siphon. Taking the air out of a pipeline is known as *priming*. Sometimes a pump is needed to extract the air but if the pipeline can be temporarily brought below the hydraulic gradient the resulting positive pressure will push the air out and it will prime itself. This can be done by closing the main valve at the end of the pipeline so that the hydraulic gradient rises to a horizontal line at the same level as the reservoir surface. An air valve on top of the siphon then releases the air. Once the pipe is full of water, the main valve can then be opened and the pipeline flows normally.



4.5 Siphons.

Even pipelines that normally operate under positive pressures have air valves. These release air which accumulates at high spots along the line. So it is good practice to include an air valve at such locations. They can be automatic valves or just simple gate valves that are opened manually occasionally to release air.

It can sometimes be difficult to spot an air valve that is above the hydraulic gradient and this can lead to problems. An engineer visiting a remote farm saw what he thought was a simple gated air valve on a high spot on a pipeline supplying the farm with water. Air does tend to accumulate over time and can restrict the flow. So he thought he would do the farmer a favour and open the valve to bleed off any air that had accumulated. After a while he realised that the hissing sound was not air escaping from the pipe but air rushing in. The pipe was in fact above the hydraulic gradient and was working as a siphon at that point and the valve was only there to let air out during the priming process. The pressure inside the pipe was in fact negative and so when he opened the valve air was sucked and this de-primed the siphon. Realising his mistake he quickly closed the valve and went on down to the farmhouse. The farmer was most upset. What a coincidence – just as a water engineer had arrived, his water supply had suddenly stopped and an engineer was on hand to fix it for him!

If your car ever runs out of petrol a siphon can be a useful means of taking some fuel from a neighbour's tank. Insert a flexible small diameter tube into the tank and suck out all the air (making sure not to get a mouthful of petrol). When the petrol begins to flow catch it in a container and then transfer it to your car. Make sure that the outlet is lower than the liquid level in the tank otherwise the siphon will not work.

Another very practical use for siphons is to detect leakage in domestic water mains (sometimes called rising mains) from the supply outside in the street to a house (Figure 4.5b). This can be important for those on a water meter who pay high prices for their water. A leaky pipe in this situation would be very costly. The main valve to the house must first be closed. Then seal the cold water tap inside the house by immersing the outlet in a pan of water and opening the tap. If there is any leakage in the main pipe then water will be siphoned back out of the pan into the main. The rate of flow will indicate the extent of the leakage.

Siphons can be very useful in situations where the land topography is undulating between a reservoir and the water users. It is always preferable to locate a pipe below the hydraulic gradient by putting it in a deep trench but this may not always be practicable. In situations where siphoning is unavoidable the pipeline must not be more than 7 m above the hydraulic grade line. Remember atmospheric pressure drives a siphon and the absolute limit is 10 m head of water. So 7 m is a safe practical limit. When pipelines are located in mountainous regions the limit needs to be lower than this due to the reduced atmospheric pressure.

The pressure inside a working siphon is less than atmospheric pressure and so it is negative when referred to as a gauge pressure (measured above or below atmospheric pressure as the datum), for example, a −7 m head. Sometimes siphon pressures are quoted as absolute pressures (measured above vacuum pressure as the datum). So −7 m gauge pressure is the same as +3 m absolute pressure. This is calculated as follows:

$$\begin{aligned}\text{gauge pressure} &= -7 \text{ m head}\\ \text{absolute pressure} &= \text{atmospheric pressure} + \text{gauge pressure}\\ &= 10 - 7 = 3 \text{ m head absolute}\end{aligned}$$

4.8 Selecting pipe sizes in practice

The development of λ as a pipe roughness coefficient is an interesting story and this nicely leads into the use of the Darcy–Weisbach formula for linking energy loss with the various pipe parameters. There are several examples using this formula in the boxes and they demonstrate well the effects of pipe length, diameter and velocity on energy loss. So it is a useful learning tool.

Engineers in different industries and in different countries have also used other formulae often developed empirically to fit their particular circumstances. But these are gradually being abandoned and replaced by the Colebrook-White formula which accurately deals with most commercially available pipes. The task of using the formula, which is a rather complicated one, is made simple by the fact that it is now available as a set of design charts (Figure 4.6). The charts are also easier to use because discharge can be related directly to pipe diameter whereas Darcy–Weisbach formula only links to velocity and so requires an extra step (continuity equation) to get to discharge.

The boxes provide examples of the use of Darcy–Weisbach formula and design charts based on Colebrook-White formula.

EXAMPLE: CALCULATING PIPE DIAMETER USING DARCY–WEISBACH FORMULA

A 2.5 km long pipeline connects a reservoir to a smaller storage tank outside a town which then supplies water to individual houses. Determine the pipe diameter when the discharge required between the reservoir and the tank is 0.35 m^3/s and the difference in their water levels is 30 m. Assume the value of λ is 0.03.

This problem can be solved using the energy equation. The first step is to write down the equation for two points in the system. Point 1 is at the water surface of the main reservoir and point 2 is at the surface of the tank. Friction losses are important in this example and so these must also be included:

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

This equation can be greatly simplified. p_1 and p_2 are both at atmospheric pressure and are zero. The water velocities v_1 and v_2 in the two tanks are very small and so the kinetic energy terms are also very small and can be assumed to be zero. This leaves just the potential energy terms z_1 and z_2 and the energy loss term h_f so the equation simplifies to:

$$h_f = z_1 - z_2$$

Using the Darcy–Weisbach formula for h_f:

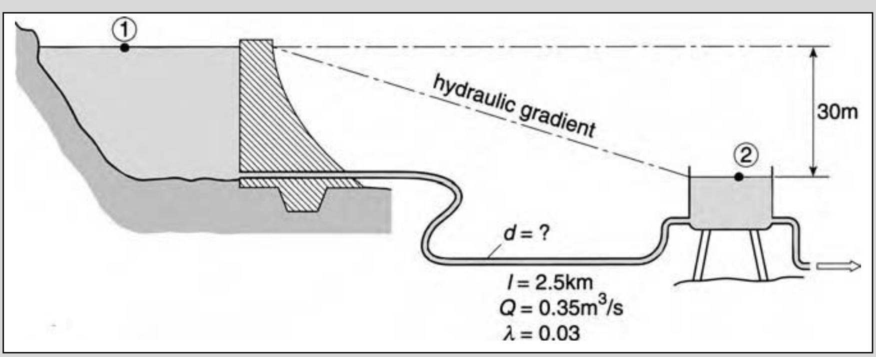
$$h_f = \frac{\lambda l v^2}{2gd}$$

And so:

$$\frac{\lambda l v^2}{2gd} = z_1 - z_2$$

Diameter d is unknown but so is the velocity in the pipe. So first calculate velocity v using the continuity equation:

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



4.6 Calculating the pipe diameter.

Calculate area *a*:

$$a = \frac{\pi d^2}{4}$$

And use this value to calculate *v*:

$$v = \frac{4Q}{\pi d^2} = \frac{4 \times 0.35}{3.14 \times d^2} = \frac{0.446}{d^2}$$

Note that as *d* is not known it is not yet possible to calculate a value for *v* and so this must remain as an algebraic expression for the moment.

Put all the known values into the Darcy–Weisbach equation:

$$\frac{0.03 \times 2500 \times 0.198}{2 \times 9.81 \times d \times d^4} = 0.35$$

Rearrange this to calculate *d*:

$$d^5 = \frac{0.03 \times 2500 \times 0.198}{2 \times 9.81 \times 30} = 0.025$$

Calculate the fifth root of 0.025 to find *d*:

$$d = 0.47 \text{ m} = 470 \text{ mm}$$

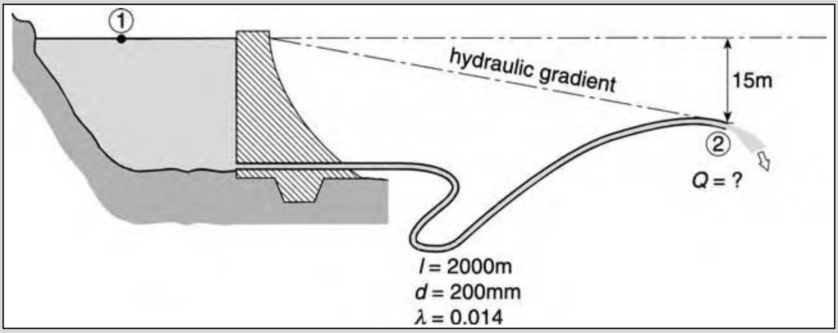
The nearest pipe size to this would be 500 mm. So this is the size of pipe needed to carry this flow between the reservoir and the tank.

This may seem rather involved mathematically but another approach, and perhaps a simpler one, is to guess the size of pipe and then put this into the equation and see if it gives the right value of discharge. This ‘trial and error’ approach is the way most engineers approach the problem. The outcome will show if the chosen size is too small or too large. A second or third guess will usually produce the right answer. If you are designing pipes on a regular basis you soon learn to ‘guess’ the right size for a particular installation. The design then becomes one of checking that your guess was the right one.

Try this design example again using the design chart in Figure 4.6 above to see if you get the same answer.

EXAMPLE: CALCULATING DISCHARGE FROM A PIPELINE

A 200 mm diameter pipeline 2000 m long is connected to a reservoir and its outlet is 15 m below the reservoir water level and discharges freely into the atmosphere. Calculate the discharge from the pipe when the friction factor is 0.014.



4.7 Measuring the discharge.

To solve this problem use the energy equation between point 1 at the surface of the reservoir and point 2 just inside the water jet emerging from the pipe outlet:

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

This equation can be greatly simplified because p_1 is at atmospheric pressure and so is zero. Also p_2 is very near atmospheric pressure because the position of 2 is in the jet as it emerges from the pipe into the atmosphere. If it was above atmospheric pressure then the jet would flow laterally under the pressure. It does not do this and so the pressure can be assumed to be close to atmospheric pressure. Therefore p_2 is zero. The water velocity v_1 is zero in the reservoir and v_2 at the outlet is very small in comparison with the potential energy of 30 m and so this can be assumed to be zero also. This leaves just the potential energy terms z_1 and z_2 and the energy loss term h_f so the equation simplifies to:

$$z_1 - z_2 = h_f = \frac{\lambda l v^2}{2gd}$$

Put in the known values and calculate velocity v:

$$15 = \frac{0.014 \times 2000 \times v^2}{2 \times 9.81 \times 0.2}$$

$$v^2 = \frac{15 \times 2 \times 9.81 \times 0.2}{0.014 \times 2000} = 2.1$$

$$v = 1.45 \text{ m/s}$$

Use the continuity equation calculate the discharge:

$$Q = va$$

Calculate area a:

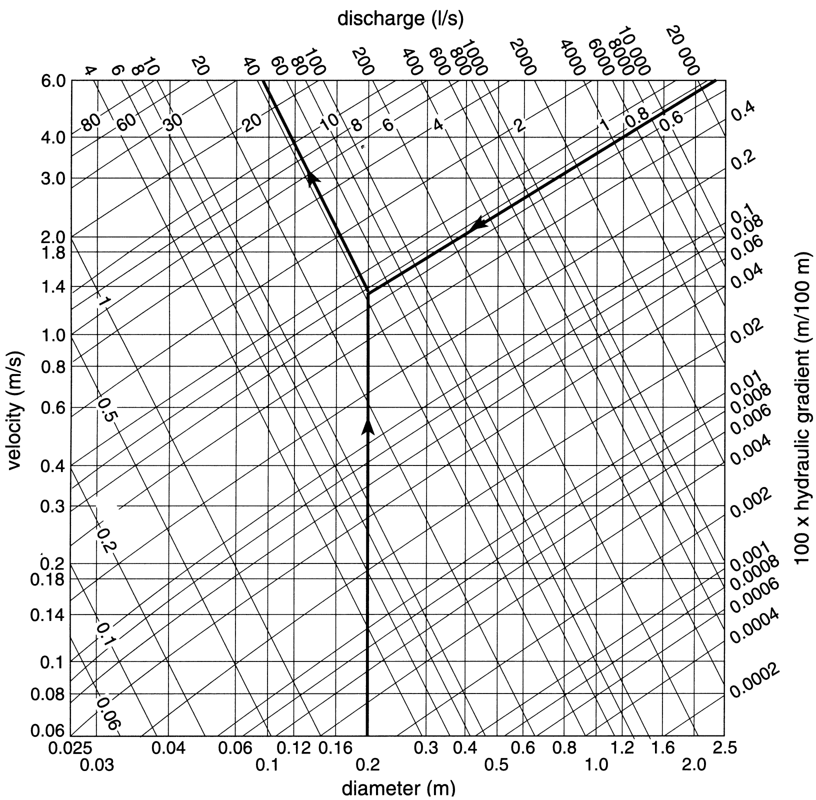
$$a = \frac{\pi d^2}{4} = \frac{\pi 0.2^2}{4} = 0.031 \text{ m}^2$$

$$Q = 1.45 \times 0.031$$

$$Q = 0.045 \text{ m}^3\text{/s or } 45 \text{ l/s}$$

4.8.1 Using hydraulic design charts

Pipe charts are now an increasingly common way of designing pipes. An excellent and widely used source is *Hydraulic Design of Channels and Pipes* by Peter Ackers (see references for details). This is a book of design charts based on the Colebrook-White equation. The equation best describes the transitional flow between smooth and rough pipe flow referred to in Section 4.4.1 which covers all commercially available pipes. An example of one of these charts is shown in Figure 4.8. It does not use λ values but expresses friction as the height of the roughness on the inside of a pipe. So for this chart, surface roughness is $k = 0.03$ mm and this is representative of asbestos cement and PVC pipes in reasonably good condition. The chart's range of flows is considerable; from less than 0.1 l/s to 20 000 l/s (or 20 m^3/s) with pipe diameters from 0.025 m to 2.5 m. This should satisfy most pipe designers. In the box is an example showing what a design chart looks like.



4.8 Typical pipe design chart.

EXAMPLE: CALCULATING DISCHARGE FROM A PIPELINE USING A DESIGN CHART

Using the same example as for the Darcy–Weisbach equation. A 200 mm diameter pipeline 2000 m long is connected to a reservoir and the outlet is 15 m below the reservoir water level. Calculate the discharge from the pipe when the pipe roughness value k is 0.03 mm.

The design chart uses the hydraulic gradient to show the rate of head loss in a pipeline. So the first step is to calculate the hydraulic gradient from the information given above. The pipe is 2000 m long and the head loss from the reservoir to the pipe outlet is 15 m so:

$$\text{hydraulic gradient} = \frac{h}{l} = \frac{15}{2000} = 0.0075$$

$$\text{or} \qquad = 0.75\ \text{m}/100\ \text{m}$$

Using the chart locate the intersection of the lines for a hydraulic gradient of 0.75 m/100 m and a diameter of 200 mm. This locates the discharge line and the value of the discharge:

$$Q = 0.045\ \text{m}^3/\text{s} \quad \text{or} \quad 45\ \text{l/s}$$

Note that the chart can also be used in reverse to determine the diameter of a pipe and head loss for a given discharge.

There are four important practical points to note from the examples.

The first point refers to the first worked example which showed how mathematically cumbersome it can be to determine the diameter by calculation. The easier way is to do what most engineers do; they guess the diameter and then check by calculation that their chosen pipe is the right one. This might seem a strange way of approaching a problem but it is quite common in engineering. It always helps to know approximately the answer to a problem before beginning to solve it. An experienced engineer usually knows what answer to expect; the calculation then becomes just a way of confirming this. (This is one of the basic unwritten laws of engineering – that you need to know the answer to the problem before you begin so that you know that you have the right answer when you get there.) This foresight is important because when you calculate a pipe diameter how will you know that you have arrived at the right answer? There will not be any answers available to the real problems in the field as there are in this text book and there may not be anyone around to ask if it is the right answer. You need to know that you have the right answer and this comes largely from experience of similar design problems. New designers are unlikely to have this experience, but they have to start somewhere and one way is to rely initially on the experience of others and to learn from them. This is the apprenticeship that all engineers go through to gain experience and become competent designers.

The second point to note is that there is no one unique pipe diameter that must be used in any given situation. If, for example, calculations show that a 100 mm pipe is sufficient then any pipe larger than this will also carry the flow. The question of which one is the most acceptable is usually determined by several design criteria. One is that the velocity should not exceed 1.6 m/s. Another might be a limit on the head that can be lost through friction. A third could be a limit on the pipe sizes available. There are only certain standard sizes which are manufactured and not all these may be readily available in some countries. A final deciding factor is cost – which pipe is the cheapest to buy and to operate?

The third point to consider is the value for pipe roughness. It is easy to choose the value for a new pipe but how long will it be before the roughness increases? What will the value be in, say, 10 years from now when the pipe is still being used? The roughness will undoubtedly increase through general wear and tear. If there has been lime scale deposits or algae slime build up on the inside of the pipe or the pipe has been misused and damaged then the roughness will be significantly greater. So when choosing the most appropriate value for design it is important to think ahead to what the roughness might be later in the life of the pipe. This is where engineering becomes an art and all the engineer's experience is brought to bear in selecting the right roughness value for design purposes. If the selected roughness is too low then the pipe may not give good, long service. If it is too high then this will result in the unnecessary expense of having pipes which are too big for the job in hand.

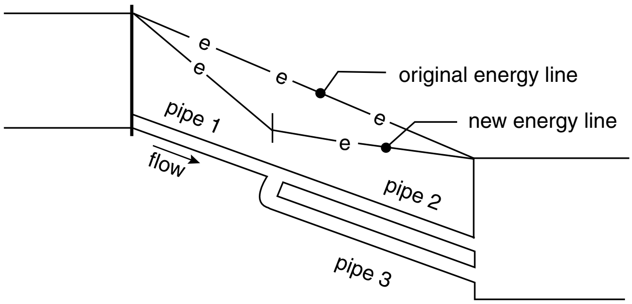
The final point, and often the most important to consider, is cost. Small diameter pipes are usually cheaper than larger diameter ones but they require more energy to deliver the discharge because of the greater friction. This is particularly important when water is pumped and energy costs are high. The trade-off between the two must take account of both capital and operating costs if a realistic comparison is to be made between alternatives. This aspect of pipeline design is covered in more detail in Chapter 8 Pumps.

4.8.2 Sizing pipes for future demand

Pipe sizes are often selected using a discharge based on present water demands and little thought is given to how this might change in the future. Also there is always a temptation to select small pipes to satisfy current demand simply because they are cheaper than larger ones. These two factors can lead to trouble in the future. If demand increases and higher discharges are required from the same pipe, the energy losses can rise sharply and so a lot more energy is needed to run the system.

As an example, a 200 mm diameter pumped pipeline 500 m long supplies a small town with a discharge of 50 l/s. Several years later the demand doubles to 100 l/s. This increases the velocity in the pipe from 1.67 to 3.34 m/s (i.e. it doubles) which pushes up the energy loss from 5 m to 20 m (i.e. a four-fold increase). This increase in head loss plus the extra flow means that eight times more energy is needed to operate the system and extra pumps may be required to provide the extra power input. A little extra thought at the planning stage and a little more investment at the beginning could save a lot of extra pumping cost later.

Increasing the energy available is one way of increasing the discharge in a pipeline to meet future demand. But another way is to increase the effective diameter of the pipe. The practical way of doing this is to lay a second pipe parallel to the first one. It may not be necessary to lay the second pipe along the entire length. Pipes are expensive and so from a cost point of view only the minimum length of parallel pipe should be laid to meet the demand (Figure 4.9). The discharge in pipe 1 will equal the discharge in pipe 2. This is the original pipeline carrying the (inadequate) discharge between the tanks – note the energy line which represents the uniform energy loss along the pipeline. Pipe 3 is the new pipe laid parallel to the original pipe and so the combined effect of pipes 2 and 3 is to increase the cross-sectional area carrying the discharge and decrease the energy loss along the parallel section of the pipeline (the velocity is lower because of the increased area). The effect of reducing the energy loss in the parallel section is to make more energy available to move water through pipeline 1 and so the overall discharge is increased. Note how the energy line for pipe 1 in the new system is steeper showing that it is carrying a higher discharge. The energy line for the parallel pipes has a more gentle gradient due to the overall reduction in velocity in this section. The length of parallel pipe depends on the required increase in discharge. Should the discharge demand increase further in the future the length of parallel pipeline can be extended to suit. The job for the designer is to decide on the diameter and length of pipe 3.



4.9 Parallel pipes can increase discharge with the same energy.

EXAMPLE: CALCULATING LENGTH OF A PARALLEL PIPE

A 1000 m long pipeline 150 mm diameter supplies water from a reservoir to an offtake point. Calculate the discharge at the offtake when the head available is 10 m. Since the pipeline was installed the water demand has doubled and so a parallel 250 mm dia pipeline is to be installed alongside the original pipeline (Figure 4.9). Calculate the length of new pipe required to double the discharge. Assume the friction factor for the pipelines is $k = 0.03$ mm. Use the pipe design chart in Figure 4.8 above.

First calculate the original discharge. Calculate the hydraulic gradient (100 h/l) and together with the pipe diameter determine the discharge from the pipe design chart. The data and the results are tabulated as follows:

Pipe	*Pipe dia (mm)*	*Length (m)*	*Friction k (mm)*	*Hyd grad (m/100 m)*	*Discharge (l/s)*
Original pipe	150	1000	0.03	1.0	23

The demand has now doubled to 46 l/s. So the additional 23 l/s is supplied by introducing a pipe of 250 mm diameter – pipe 3 – alongside the original pipeline but as yet of unknown length. This length cannot be calculated directly and requires some iteration. In other words, some intelligent guess work. It is convenient at this stage to divide the original pipeline into two parts – pipe 2 which has the same length as pipe 3, and pipe 1 which is from the reservoir to the point where the two parallel pipes join.

First determine the hydraulic gradient in the two parallel pipes – pipes 2 and 3 – so that the two pipelines carry a combined discharge of 46 l/s. The gradient will be the same for each pipeline as they have the same pressure at the points of connection and discharge. Mark on the pipe chart vertical lines representing the two pipe diameters. Look for a hydraulic gradient that intersects the pipe 'lines' so that the sum of the two discharges is 46 l/s. This occurs at a hydraulic gradient of 0.18 m/100 m which results in discharges of 10 and 35 l/s totalling 45 l/s. This is close enough to 46 l/s.

Next determine the hydraulic gradient for pipe 1 for a discharge of 46 l/s. From the chart this is 3.2 m/100 m.

Using this information it is now possible to set up an equation to calculate the length of pipe 3.

The sum of the head loss in pipes 1 and 3 (remember the loss in pipe 3 will be the same as pipe 2) is 10 m. So:

$$h_1 + h_3 = 10 \text{ m}$$

Now:

$$\frac{100h_1}{L_1} = \text{hydraulic gradient} = 3.2$$

$$h_1 = \frac{L_1 \times 3.2}{100}$$

Similarly:

$$h_3 = \frac{L_3 \times 0.18}{100}$$

So:

$$\frac{L_1 \times 3.2}{100} + \frac{L_3 \times 0.18}{100} = 10$$

Both L_1 and L_3 are unknown. So to 'eliminate' one of the unknowns substitute for L_1 in terms of L_3:

$$L_1 = 1000 - L_3$$

Substitute this into the above equation:

$$\frac{(1000 - L_3) \times 3.2}{100} + \frac{L_3 \times 0.18}{100} = 10$$

$$(1000 - L_3)\ 3.2 + 0.18L_3 = 1000$$

$$3200 - 3.2L_3 + 0.18L_3 = 1000$$

$$3.38L_3 = 2200$$

$$L_3 = \frac{2200}{3.38} = 650 \text{ m}$$

So the length of pipe 3 is 650 m. This is also the length of pipe 2. Pipe 1 will be 350 m. The following table summarises the results:

Pipe	*Pipe dia (mm)*	*Friction k (mm)*	*Hyd grad (m/100 m)*	*Discharge (l/s)*	*Length (m)*
Pipe 1	150	0.03	3.20	46	350
Pipe 2	150	0.03	0.18	10	650
Pipe 3	250	0.03	0.18	35	650

This is one example where using a formula makes the problem easier to solve as it avoids the iterative approach. Try to solve the problem using the Darcy–Weisbach formula and the continuity equation with a value of $\lambda = 0.04$ for the pipes.

4.9 Pipe networks

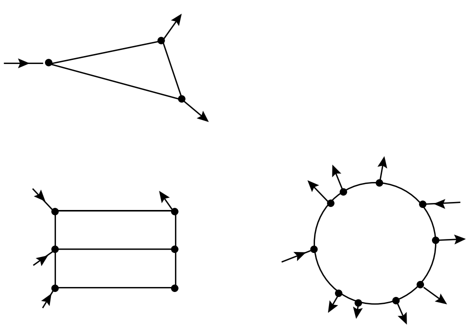
Most water supply systems are not just single pipes but involve several branching pipes or a pipe network. These supply water to several reservoirs or dwellings in a village or town (Figure 4.10). Some networks are simple and involve just a few pipes but some are quite complicated involving many different pipes and connections. Sometimes the pipes form a ring or loop and this ensures that if one section of pipe fails for some reason then flow can be maintained from another direction. It also has hydraulic advantages. Each offtake point is supplied from two directions and so the pipe sizes in the ring can be smaller than if the point was fed from a single pipe.

The simplest example of a ring or loop network is a triangular pipe layout (Figure 4.11). Water flows into the loop at point A and flows out at points B and C. So the water flows away from point A towards B and C. But there will also be a flow in pipe BC and this could be in the direction BC or CB depending on the pressure difference between B and C. If the pressure at B is higher than C then there will be a flow from B to C. Conversely if the pressure at C is higher than the pressure at B then the flow will be from C to B.

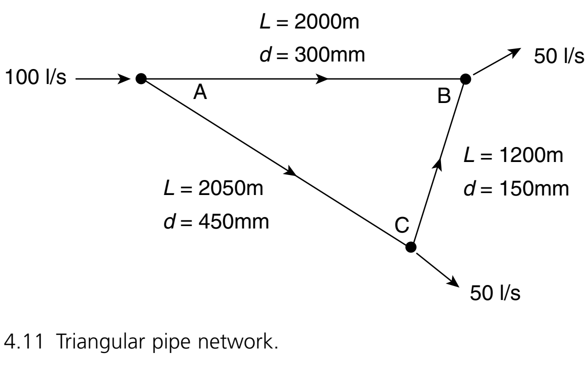
There are three rules to solving the problem of pressures and discharges in a network:

1 The sum of all the discharges at a junction is zero;
2 The flow in one leg of a network will be in the direction of the pressure drop;
3 The sum of the head losses in a closed loop will be zero for a start and finish at any junction in the loop.

These rules apply to all networks, not just the simple ones. However, the calculations can get rather involved. An example of how the rules are applied to a simple network is shown in the box.



4.10 Pipe networks.



4.11 Triangular pipe network.

EXAMPLE: CALCULATING DISCHARGES AND PRESSURES IN A PIPE NETWORK

A simple network of three pipes forms a triangle ABC. The following data are available

Pipe	*Diameter (mm)*	*Length (m)*	*Friction factor k (mm)*
AB	300	2000	0.03
BC	150	1200	0.03
CA	450	2050	0.03

The discharge entering the system at point A is 100 l/s and this meets the demand at B of 50 l/s and at C of 50 l/s. Calculate the discharges in each pipe and the pressures at B and C if A is supplied at a pressure of 100 m head of water. Use the pipe design chart in Figure 4.8.

Start by looking at the data, apply some common sense, and rule (1) get an assessment of the likely discharges in each pipe. AB and AC are approximately the same in length and AC has a large diameter. So assume Q_{AC} is 60 l/s, Q_{AB} is 40 l/s and Q_{CB} is 10 l/s. Note the assumed flow directions indicated by the arrows.

Next calculate the head loss in each pipe using the pipe chart in Figure 4.8. Notice how the discharges in pipes BC and CA are listed as negative. The sign comes from considering the discharges positive in a clockwise direction around the loop.

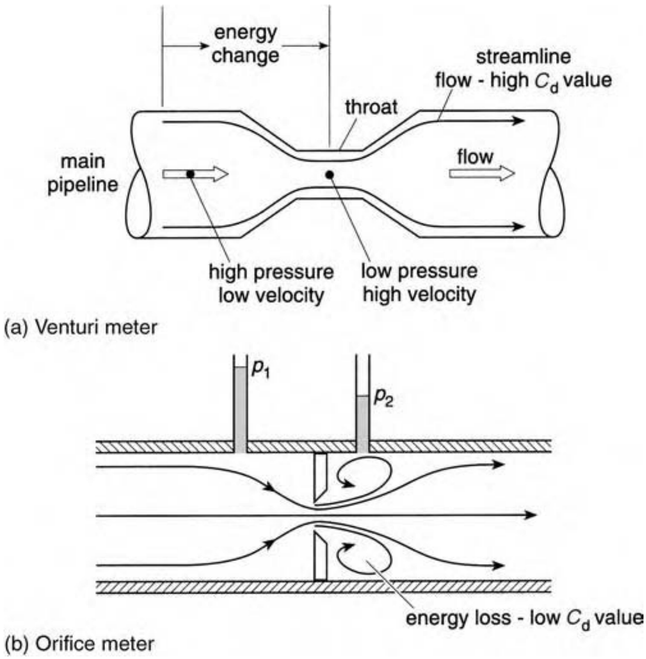
Pipe	*Diameter (mm)*	*Discharge (l/s)*	*Hydraulic gradient (m per 100 m)*	*Length (m)*	*Head loss (m)*
AB	300	40	0.14	2000	+2.80
BC	150	−10	0.2	1200	−2.40
CA	450	−60	0.025	2050	−0.51

Applying rule (3) from the start point A and moving in a clockwise direction add up all the head losses in the pipes, that is, 2.80 – 2.40 – 0.51 = 0.11 m. This sum should come to zero. To make the above sum come to zero slightly decrease the discharge in pipe AB, recalculate the head losses again and see if the sum of the head losses comes to zero. In this case the value is close to zero so this suggests that the discharge values chosen are close to the right ones. So there is no need for further iteration.

4.10 Measuring discharge in pipes

Discharges in pipelines can be measured using a *venturi meter* or an *orifice plate* (Figure 4.12). Both devices rely on changing the components of the total energy of flow from which discharge can be calculated (see Section 3.7.6). The venturi meter was developed by an American, Clemens Herschel (1842–1930) who was looking for a way to measure water abstraction from a river by industrialists. Although the principles of this measuring device were well established by Bernoulli it was Herschel who, being troubled by unlicensed and unmeasured removal of water by pipelines from a canal by paper mills, developed it into the device used today.

A venturi meter comprises a short, narrow section of pipe (throat) followed by a gradually expanding tube. This causes the flow velocity to increase (remember continuity) and so the kinetic energy increases also. As the total energy remains the same throughout the system it follows that there must be a corresponding reduction in pressure energy. By measuring this change in pressure using a pressure gauge or a manometer and using the continuity and energy



4.12 Measuring discharge in pipelines.

equations, the following formula for discharge in the pipe can be obtained:

$$Q = C_d a_1 \sqrt{\frac{2gH}{m^2 - 1}}$$

$$m = \frac{a_1}{a_2}$$

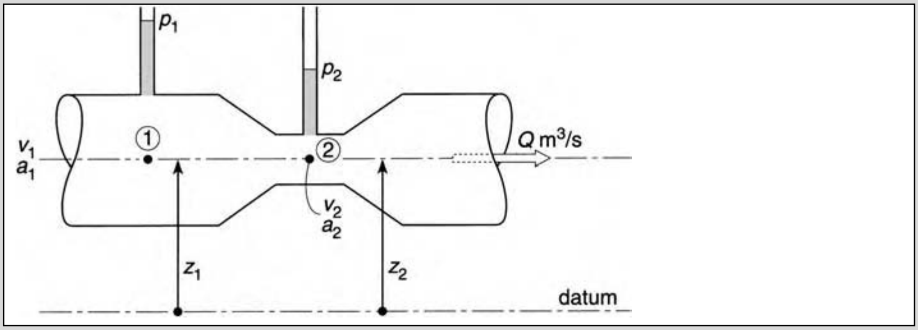
where a_1 is area of main pipe (m^2); a_2 is area of venturi throat (m^2); H is the head difference between pipe and throat (m); g is gravity constant (9.81 m/s^2); C_d is coefficient of discharge.

This is the theory, but in practice there are some minor energy losses in a venturi meter, so a coefficient discharge C_d is introduced to obtain the true discharge. Care is needed when using this formula. Some textbooks quote the formula in terms of a_2 rather than a_1 and this changes several of the terms. It is the same formula from the same fundamental base but it can be confusing. The safest way is to avoid the formula and work directly from the energy and continuity equations. A derivation of the formula and an example of calculating discharge working from energy and continuity are shown in the boxes.

DERIVATION: FORMULA FOR DISCHARGE IN A VENTURI METER

First write down the energy equation for the venturi meter. Point 1 is in the main pipe and point 2 is located in the throat of the venturi. It is assumed that there is no energy loss between the two points. This is a reasonable assumption as contracting flows suppress turbulence which is the main cause of energy loss.

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



4.13 Venturi meter for measuring discharge.

As the venturi is horizontal:

$$z_1 = z_2$$

And so:

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

Now rearrange this equation so that all the pressure terms and all the velocity terms are brought together:

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

The left-hand side of this equation is the pressure difference between points 1 and 2 which can be measured using pressure gauges or a differential manometer. This is a manometer with one limb connected to the pipe and the other limb connected to the throat (see Section 2.9.4).

At this point it is not possible to calculate the velocities because both v_1 and v_2 are unknown. A second equation is needed to do this – the continuity equation.

Write the continuity equation for points 1 and 2 in the venturi:

$$a_1 v_1 = a_2 v_2$$

Rearrange this:

$$v_2 = \frac{a_1}{a_2} v_1$$

Now a_1 and a_2 are the cross-sectional areas of the pipe and venturi respectively and can be calculated from the pipe and venturi throat diameters respectively. Substituting for v_2 in the energy equation

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = \left(\frac{a_1^2}{a_2^2}\right) \frac{v_1^2}{2g} - \frac{v_1^2}{2g}$$

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = \frac{v_1^2}{2g}\left(\frac{a_1^2 - a_2^2}{a_2^2}\right)$$

Put:

$$H = \frac{p_1}{\rho g} - \frac{p_2}{\rho g}$$

where H is the difference in head between the pipe (point 1) and the venturi throat point 2)

Rearrange the equation for v_1

$$v_1 = \sqrt{2gH}\left(\frac{a_2}{\sqrt{a_1^2 - a_2^2}}\right)$$

Use the continuity equation to calculate discharge:

$$Q = a_1 v_1$$

And so:

$$Q = \sqrt{2gH}\left(\frac{a_1 a_2}{\sqrt{a_1^2 - a_2^2}}\right)$$

Put:

$$m = \frac{a_1}{a_2}$$

And so:

$$Q = a_1\sqrt{\frac{2gH}{m^2 - 1}}$$

Introduce a coefficient of discharge C_d:

$$Q = C_d a_1\sqrt{\frac{2gH}{m^2 - 1}}$$

In the case of the venturi meter $C_d = 0.97$ which means that the energy losses are small and the energy theory works very well ($C_d = 1.0$ would mean the theory was perfect). For orifice plates the same theory and formula can be used but the value of C_d is quite different at $C_d = 0.6$. The theory is not so good for this case because there is a lot of energy loss (Figure 4.12b). The water is not channelled smoothly from one section to another as in the venturi but is forced to make abrupt changes as it passes through the orifice and expands downstream. Such abruptness causes a lot of turbulence which results in energy loss. (The C_d value is similar to that for orifice flow from a tank – see Section 3.6.3.)

EXAMPLE: CALCULATING DISCHARGE USING A VENTURI METER

A 120 mm diameter venturi meter is installed in a 250 mm diameter pipeline to measure discharge. Calculate the discharge when the pressure difference between the pipe and the venturi throat is 2.5 m of head of water and C_d is 0.97.

Although there is a formula for discharge it can be helpful to work from first principles. Not only does this reinforce the principle but it also avoids possible errors in using a rather involved formula which can easily be misquoted. So this example is worked from the energy and continuity equations.

First step write down the energy equation:

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

As the venturi is horizontal:

$$z_1 = z_2$$

And so:

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

Now rearrange this equation so that all the pressure terms and all the velocity terms are brought together:

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

But:

$$\frac{p_1}{\rho g} - \frac{p_2}{\rho g} = 2.5 \text{ m}$$

Remember that the pressure terms are in m head of water and it is the difference that is important and not the individual pressures:
And so:

$$\frac{v_2^2}{2g} - \frac{v_1^2}{2g} = 2.5 \text{ m}$$

It is not possible to solve this equation directly as both v_1 and v_2 are unknown. So use continuity to obtain another equation for v_1 and v_2:

$$a_1 v_1 = a_2 v_2$$

Rearrange this:

$$v_2 = \frac{a_1}{a_2} v_1$$

Next step calculate the areas a_1 and a_2.

Area of pipe:

$$a_1 = \frac{\pi d_1^2}{4} = \frac{\pi 0.25^2}{4} = 0.05 \text{ m}^2$$

Area of venturi:

$$a_2 = \frac{\pi d_2^2}{4} = \frac{\pi 0.12^2}{4} = 0.011 \text{ m}^2$$

$$v_2 = \frac{0.05}{0.011} \quad v_1 = 4.55\ v_1$$

Substitute this value for v_2 in the energy equation:

$$\frac{4.55^2 v_1^2}{2g} - \frac{v_1^2}{2g} = 2.5 \text{ m}$$

$$20.7\ v_1^2 - v_1^2 = 2.5 \times 2 \times 9.81 = 49.05$$

$$v_1 = \sqrt{\frac{49.05}{19.7}} = 1.57 \text{ m/s}$$

Calculate Q:

$$Q = C_d\ v_1\ a_1$$
$$= 0.97 \times 1.57 \times 0.05$$
$$Q = 0.076 \text{ m}^3\text{/s}$$

4.11 Momentum in pipes

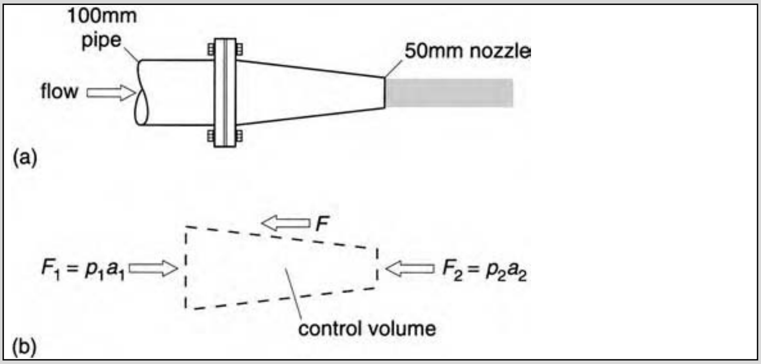
The momentum equation is used in pipe flow to calculate forces on pipe fittings such as nozzles, pipe bends and valves. In more advanced applications it is used in the design of pumps and turbines where water flow creates forces on pump and turbine impellers.

To solve force and momentum problems a concept known as the *control volume* is used. This is a way of isolating part of a system being investigated so that the momentum equation can be applied to it. To see how this works an example is given in the box below showing how the force on a pipe reducer (or nozzle) can be calculated.

EXAMPLE: CALCULATING THE FORCE ON A NOZZLE

A 100 mm diameter fire hose discharges 15 l/s from a 50 mm diameter nozzle. Calculate the force on the nozzle.

To solve this problem all three hydraulic equations are needed; energy, continuity and momentum. The energy and continuity equations are needed to calculate the pressure in the 100 mm pipe and momentum is then used to calculate the force on the nozzle.



4.14 Calculating the force on a nozzle.

The first step is to calculate the pressure p_1 in the 100 mm pipe. Use the energy equation:

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

The pressure in the jet as it emerges from the nozzle into the atmosphere is p_2. The jet is at the same pressure as the atmosphere and so the pressure p_2 is zero. The potential energies z_1 and z_2 are equal to each other because the nozzle and the pipe are horizontal and so they cancel out.

So the energy equation becomes:

$$\frac{p_1}{\rho g} = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

The value of p_1 is unknown and so are v_1 and v_2. So the next step is to calculate the velocities from the discharge equation:

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

Area of pipe:

$$a_1 = \frac{\pi d_1^2}{4} = \frac{\pi 0.1^2}{4} = 0.0078 \text{ m}^2$$

Area of jet:

$$a_2 = \frac{\pi d_2^2}{4} = \frac{\pi 0.05^2}{4} = 0.0019 \text{ m}^2$$

Next calculate the velocities:

$$v_1 = \frac{Q}{a_1} = \frac{0.015}{0.0078} = 1.92 \text{ m/s}$$

And:

$$v_2 = \frac{Q}{a_2} = \frac{0.015}{0.0019} = 7.9 \text{ m/s}$$

Put all the known values into the energy equation:

$$\frac{p_1}{\rho g} = \frac{7.9^2}{2 \times 9.81} - \frac{1.92^2}{2 \times 9.81} = 2.99$$

$$p_1 = 2.99 \times 1000 \times 9.81 = 29\,332 \text{ N/m}^2$$

The final step is to calculate the force F on the nozzle using the momentum equation. To do this the concept of the *control volume* isolates that part of the system being investigated. All the forces which help to maintain the control volume are then identified. F_1 and F_2 are forces due to the water pressure in the pipe and the jet and F is the force exerted on the water by the nozzle. Although the force F_2 is shown acting against the flow remember that there is an equal and opposite force acting in the direction of the flow (Newton's third law).

Use the momentum equation:

$$F_1 - F_2 - F = \rho Q\,(v_2 - v_1)$$

F is the force we wish to calculate but at this stage both F_1 and F_2 are also unknown. So the next step is to calculate F_1 and F_2 using known values of pressure and pipe area:

$$\begin{aligned} F_1 &= p_1 \times a_1 \\ &= 29\,332 \times 0.0078 = 228.8 \text{ N} \end{aligned}$$

And:

$$F_2 = p_2 \times a_2$$

But p_2 is zero and so:

$$F_2 = 0$$

Finally all the information for the momentum equation is available to calculate F:

$$\begin{aligned} 228.8 - F - 0 &= 1000 \times 0.015\,(7.9 - 1.92) \\ F &= 228.8 - 89.7 \\ F &= 139 \text{ N} \end{aligned}$$

As this is a fire hose, a firm grip would be required to hold it in place. If a fireman let go of the nozzle, the unbalanced force of 139 N would cause it to rapidly shoot backwards and this could do serious injury if it hit someone. A larger nozzle and discharge would probably need two firemen to hold and control it.

Forces, often large ones, also occur at pipe bends and this is not always appreciated. For example, a 90° bend on a 0.5 m diameter pipe operating at 30 m head carrying a discharge of 0.3 m^3/s would produce a force of 86 kN. This is a large thrust (over 8 tons) – which means that the bend must be held firmly in place if it is not to move and burst the pipe (Figure 4.15). A good way to deal with this is to bury the pipe below ground and encase the bend in concrete to stop it moving. The side of the trench must also be very firm for the concrete to push against it.

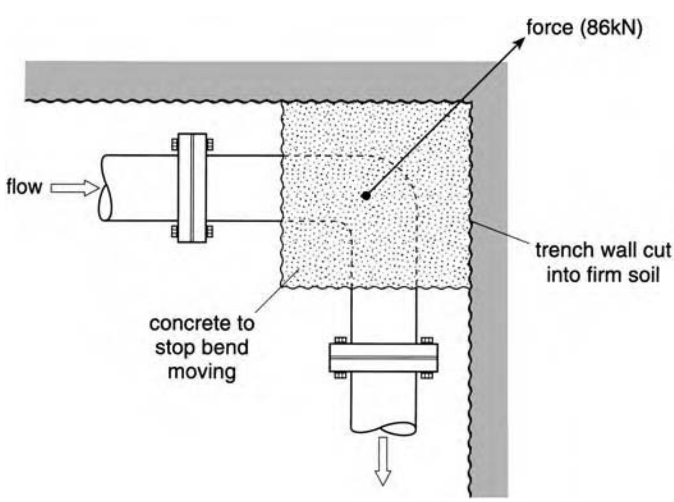
4.12 Pipe materials

Pipes are a major cost in any water supply or irrigation scheme. They must carry the design flow, resist all external and internal forces, be durable and have a long useful life. So it is important to know something about what pipes are available, how they are installed and used, what valves and fittings are needed and how to test them once they are in place.

4.12.1 Specifying pipes

Pipe manufacture is normally controlled by specifications laid down by national and international standards organisations. In the UK the British Standards Institute (BSI) has developed its own standards for pipes although a notable exception to this is the aluminium pipe used in many irrigation systems which has an international standard adopted by UK. The standards were developed for the water supply industry which demand high quality manufacture and rigorous testing of all pipes and fittings. The International Standards Organisation (ISO) does a similar job on an international scale. Many standards are the same and some BSI publications for example also have an ISO number.

Although the internal bore of a pipe determines its flow capacity it is not generally used to specify pipe size. The reason for this is that pipes are also classified according to pressure which means they have different wall thicknesses. If the outside diameter of a pipe is fixed, which is often the case from a pipe joining point of view, the internal bore will be different for different wall thicknesses. To add to this confusion different pipe materials (e.g. PVC, steel) will have different wall thicknesses. To overcome this problem manufacturers quote a *nominal pipe size*, which in many cases is neither the inside nor the outside diameter! – it is just an indication of



4.15 Force on a pipe bend.

the pipe diameter for selection purposes. It is also normal to specify the safe working pressure at which the pipes can be used.

4.12.2 Materials

Pipes are made from a wide variety of materials but the most common in water systems are steel, asbestos cement and various plastics. Steel pipes tend to be used only where very high pressures are encountered or in conjunction with other pipe materials where extra strength is needed, such as under roads or across ditches. Larger pipes are made from steel plate bent or rolled to shape and butt welded. But small sizes up to 450 mm are made from hot steel ingots which are pierced and rolled into a cylinder of the right dimensions. Corrosion is a problem with steel and so pipes are wrapped with bituminous materials or galvanising. Buried pipes often use cathodic protection. Corrosion is an electrolytic process set up between the pipe and the surrounding soil. It is a bit like a battery with the steel pipe acting as an anode which gradually corrodes. The process is reversed by making the pipe a cathode either by connecting it to an expendable anodic material such as magnesium or by passing a small electric current through the pipe material.

Asbestos cement pipes are still used in many countries for underground mains and are made from cement and asbestos fibre mixed in a slurry and deposited layer upon layer on a rolling mandrel. The pipe is then dipped in cold bitumen for protection and the end turned down to a specific diameter for jointing purposes. Unlike steel, asbestos cement does not corrode easily but it is easily damaged by shock loads and must be handled with care. Sometimes damage is not always apparent as in the case of hairline fractures. These only show up when the pipe is being site tested and leaks occur. Joints are made using an asbestos cement collar and rubber sealing rings similar to those used for PVC pipes. Bends and fittings cannot be made from asbestos cement and so ductile iron is used. An important criterion is that the outside diameter of the ductile steel must be the same as the asbestos cement so they can be effectively joined together.

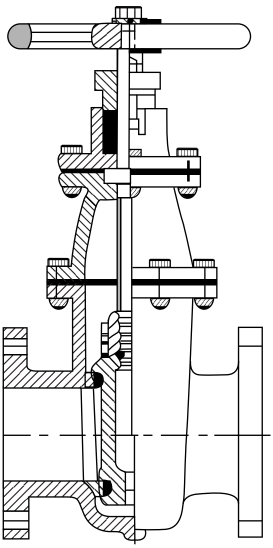
Several plastic materials are in common use for making pipes. Unplasticised polyvinyl chloride (UPVC) is a rigid material and an alternative to asbestos cement, which is falling out of favour because of the health risks associated with asbestos fibres. Pipes come in various pressure classes and are colour coded for easy recognition. They are virtually corrosion free, light in weight, flexible and easily jointed by a spigot and socket system using either a chemical solvent or rubber ring to create a seal. But laying PVC pipes needs care. They are easily damaged by sharp stones in the soil and distorted by poor compaction of material on the bottom and sides of trenches. Hydraulically PVC has a very low friction characteristic, but it can be easily damaged internally by sand and silt particles in the flowing water.

In contrast polythene pipe is very flexible and comes in high density and low density forms. Low density pipe is cheaper than high density and is used extensively for trickle and some sprinkler irrigation systems where pipes are laid out on the ground. The pipe wall is thin and so it is easily damaged by sharp tools or animals biting through it. However, it will stand up to quite high water pressures. High density pipes are much stronger, but more expensive and are used extensively for domestic water supply systems.

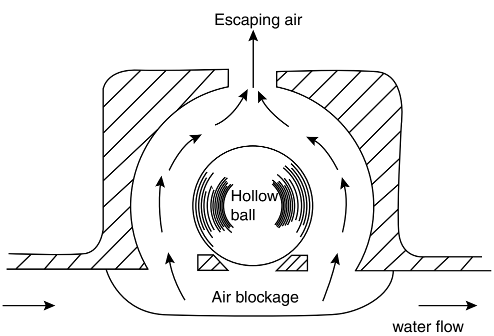
4.13 Pipe fittings

Pipelines require a wide range of valves and special fittings to make sure the discharge is always under control. Some of the more important are sluice valves, air valves, non-return valves and control valves (Figure 4.16).

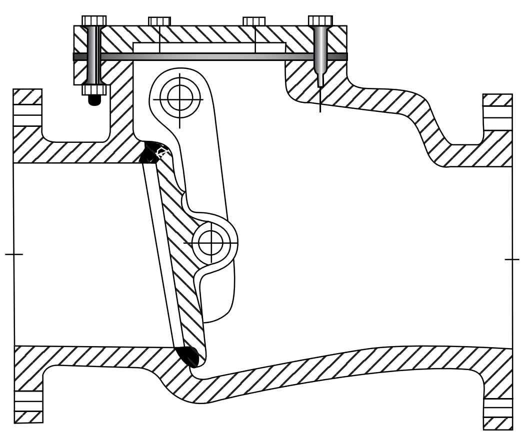
Sluice valves are essentially on-off valves. They can be used to control pressure and discharge but they are rather crude and need constant attention. It is only the last 10% of the gate opening



(a) Sluice valve.

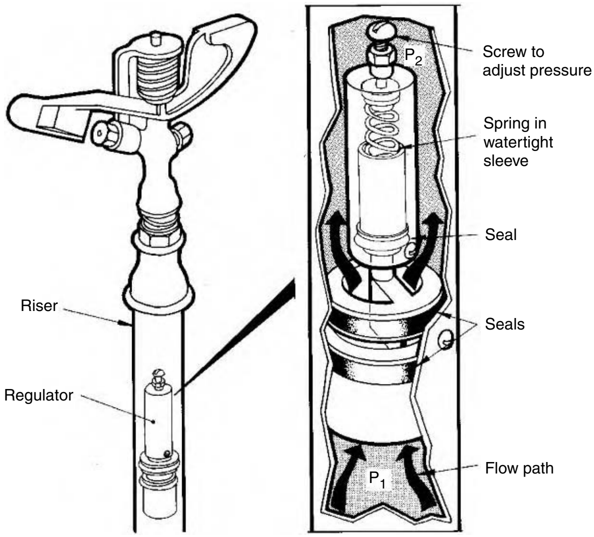


(b) Air valve.



(c) Non-return valve.

4.16 Pipe fittings.



(d) Pressure control valve.

4.16 Continued.

that has any real controlling influence. Moving the gate over the remaining 90% does not really influence the discharge, it just changes the energy from pressure to kinetic energy so the flow goes through the narrow opening faster. The valve body contains a gate which can be lowered using a screw device to close off the flow. The gate slides in a groove in the valve body and relies on the surfaces of the gate and the groove being forced together to make a seal. It was developed over 100 years ago for the water industry and has not changed materially since then. The gate can be tapered so that it does not jam in the gate guide but the taper gate must be opened fully otherwise it may start to vibrate once it is partially open. Sluice valves do require some effort to open them as water pressure builds up on one side producing an unbalanced head. The term 'cracking open' a valve is sometimes used to describe the initial effort needed to get the valve moving. A 100 mm valve fitted with a hand wheel is difficult to open when there is an unbalanced pressure of over 8 bar. Larger valves often used gearing mechanisms to move them and so opening and closing valves can be a slow business. This is an advantage as it reduces the problems of water hammer (see Section 4.14).

Butterfly valves do much the same job as sluice valves – they consist of a disc which rotates about a spindle across the diameter of the pipe. They are easier to operate than sluice valves requiring less force to open them. In spite of this they have not always been favoured by designers because the disc in an obstacle to the flow where debris can accumulate. It is also possible to close a small butterfly valve quite rapidly and this can cause water hammer problems.

Air valves are a means of letting unwanted air out of pipelines. Water can contain large volumes of dissolved air which can come out of solution when the pressure drops, particularly at high spots on pipelines (see Section 4.7). If air is allowed to accumulate it can block the flow or at least considerably reduce it. Air valves allow air to escape. There are two types depending on whether the pipeline is above or below the hydraulic gradient. For pipes below the hydraulic gradient where the water pressure is positive, the most common situation, a ball valve is used. When air collects in the pipe the ball falls onto its seating allowing air to escape. As the air escapes water rises into the valve which pushes the ball up to close off the valve. When the pipeline is above the hydraulic gradient the pressure inside the pipe is negative (below atmospheric pressure). In this case the ball valve would not work as air would be sucked in and this would allow the pipe to fill with air and break the siphon. A manual valve is needed to remove the air but a bellows fitting can also be used to suck out unwanted air. This process is very much like the priming described both for siphons (see Section 4.7) and pumps (see Section 8.4.1).

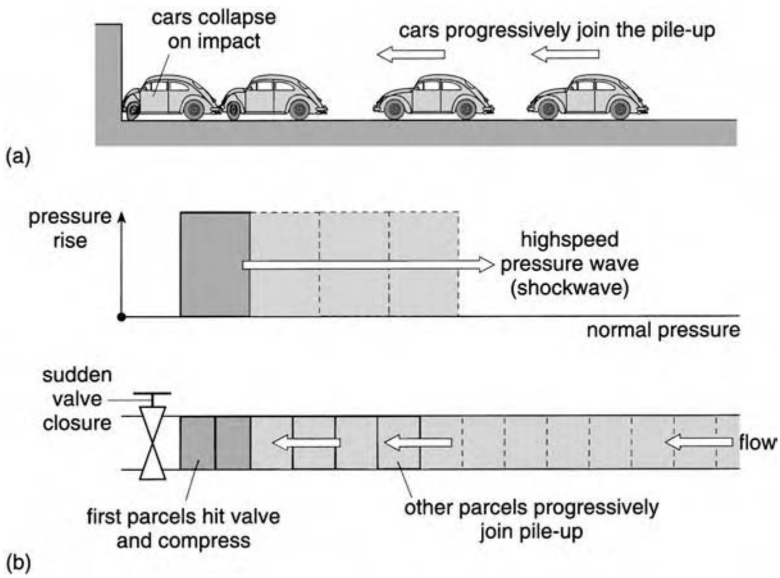
The non-return valve – sometimes called a reflux valve – does what it says. It allows water to flow one way only and prevents return flow once the main flow stops. These are essential fittings on pump delivery pipes to prevent damage from water hammer, and on water supply pipelines to prevent contamination from being 'sucked' into the pipe when it is being shut down. An example is an outside tap for watering the garden with a hose pipe. If for some reason another tap on the same domestic supply system is opened at the same time, the flow in the hose pipe can stop and even be reversed. This can suck soil particles and harmful bacteria into the pipeline and contaminate the flow. To avoid this problem all outside taps should be fitted with a non-return valve – in the UK it is a legal requirement.

Control valves are available for a wide range of control issues such as discharge and pressure control, pressure reducing, pressure sustaining and surge control. Some valves now are very sophisticated and expensive but each works on a simple principle. One simple example is a valve to regulate pressure under agricultural sprinklers. Sprinklers work best when the operating pressure is constant and at the value recommended by the manufacturer. A small regulator placed under each sprinkler ensures that each sprinkler operates at the desired pressure. Normally pressures p_1 and p_2 are equal and close to the sprinkler operating pressure (Figure 4.16d). But if the pressure p_1 rises then p_2 also starts to rise. This pushes down the sleeve closing up the waterway and reducing the flow to the sprinkler. The effect of this is to reduce the pressure p_2. If p_1 starts to fall then p_2 also falls, the sleeve opens and allows more flow through to maintain the pressure p_2. The regulated pressure p_2 is controlled by a spring in compression inside the sleeve. This can be adjusted using a small screw. For example, if a higher pressure p_2 is required then the screw is turned clockwise increasing the spring compression. This stops the movement of the sleeve until the higher pressure p_2 is reached. Although this is a simple mechanism most pressure regulating valves work on this principle. Note that pressure control valves will only reduce the pressure, they cannot increase it beyond the working pressure in the pipeline.

4.14 Water hammer

Most people will already have experienced this but may not have realised it nor appreciated the seriousness of it. When a domestic water tap is turned off quickly, there is sometimes a loud banging noise in the pipes and sometimes the pipes start to vibrate. The noise is the result of a high pressure wave which moves rapidly through the pipes as a result of the rapid closure of the tap. This is known as *water hammer* and although it may not be too serious in domestic plumbing it can have disastrous consequences in larger pipelines and may result in pipe bursts.

Water hammer occurs when flowing water is suddenly stopped. It behaves in a similar way to traffic flowing along a road when suddenly one car stops for no clear reason (Figure 4.17). The car travelling close behind then crashes into it and the impact causes the cars to crumple. The next one crashes into the other two and so on until there is quite a pile up. Notice that all the cars do not crash at the same time. A few seconds pass between each impact and so it takes several seconds before they all join the pile up. If you are watching this from a distance it would appear as if there was a wave moving up the line of cars as each joins the pile up. The speed of the wave is equal to the speed at which successive impacts occur. It is worth pointing out that cars are designed to collapse on impact so as to absorb the kinetic energy. If they were built more rigidly then all the energy on impact would be transferred to the driver and the passengers and not even seat belts would hold you in such circumstances.



4.17 Water hammer.

Now imagine water flowing along a pipeline at the end of which is a valve that is closed suddenly (Figure 4.17b). If water was not compressible then it would behave like a long solid rod and would crash into the valve with such enormous force (momentum change) that it would probably destroy the valve. Fortunately, water is compressible and it behaves in a similar manner to the vehicles it squashes on impact. Think of the flow being made up of small 'parcels' of water. The first parcel hits the valve and compresses (like the first car); the second crashes into the first and compresses and so on until all the water is stopped. This does not happen instantly but takes several seconds before all the water feels the impact and stops. The result is a sudden, large pressure rise at the valve and a pressure wave which travels rapidly along the pipe. This is referred to as a *shock wave* because of its suddenness.

The pressure wave is not just one way. Once it reaches the end of the pipeline it reflects back towards the valve again. It is like a coiled spring that moves back and forth and gradually stops. This oscillating motion can go on for several minutes in a pipe until friction slowly reduces the pressure back to the normal operating level.

The extent of the pressure rise depends on how fast the water was travelling (velocity) and how quickly the valve was closed. It does not depend on the initial pipeline pressure as is often thought. It can be calculated using a formula developed by Nicholai Joukowsky (1847–1921) who carried out the first successful analysis of this problem:

$$\Delta h = \frac{cv}{g}$$

where Δh is rise in pressure (m); c is velocity of the shock wave (m/s); v is water velocity (m/s); g is gravity constant (9.81 m/s^2).

The shock wave travels at very high velocity between 1200 and 1400 m/s. It depends on the diameter of the pipe and the material from which the pipe is made as some materials absorb the energy of compression of the water better than others. An example in the box shows just how high the pressure can rise.

EXAMPLE: CALCULATING PRESSURE RISE IN A PIPELINE DUE TO WATER HAMMER

Using the Joukowsky equation determine the pressure rise in a pipeline when it is suddenly closed. The normal pipeline velocity is 1.0 m/s and the shock wave velocity is 1200 m/s.

If the pipeline is 10 km long determine how long it takes for the pressure wave to travel the length of the pipeline.

Using the Joukowsky equation:

$$\Delta h = \frac{cv}{g}$$

$$= \frac{1200 \times 1.0}{9.81}$$

pressure rise = 122 m

So the pressure rise would be 122 m head of water or 12.2 bar. This is on top of the normal operating pressure of the pipe and could well cause the pipe to burst.

Calculate the time it takes the wave to travel the length of the pipe:

$$\text{time} = \frac{\text{distance}}{\text{velocity}} = \frac{10\,000}{1200} = 8\text{ s}$$

It takes only 8 s for the shock wave to travel the 10 km length of the pipe and 16 s for this to return to the valve.

The example of course, is an extreme one as it is difficult to close a valve instantaneously. It is also wrong to assume that the pipe is rigid. All materials stretch when they are under pressure and so the pipe itself will absorb some of the pressure energy by expanding. All these factors help to reduce the pressure rise but they do not stop it. Even if the pressure rise is only half the above value (say 6 bar) it is still a high pressure to suddenly cope with and this too is enough to burst the pipe. When pipes burst they usually open up along their length rather than around their circumference. They burst open rather like unzipping a coat.

Reducing water hammer problems is similar to reducing car crash problems. When cars are moving slowly then the force of impact is not as great. Also when the first car slows down gradually then the others are unlikely to crash into it. Similarly, when water is moving more slowly the pressure rises when a valve closes is reduced (see Joukowsky equation). This is one of the reasons why most pipeline designers restrict velocities to below 1.6 m/s so as to reduce water hammer problems. Also, when valves are closed slowly, water slows down gradually and there is little or no pressure rise along the pipe.

In summary to reduce the effects of water hammer:

- Make sure water velocities are low (below 1.6 m/s).
- Close control valves slowly.

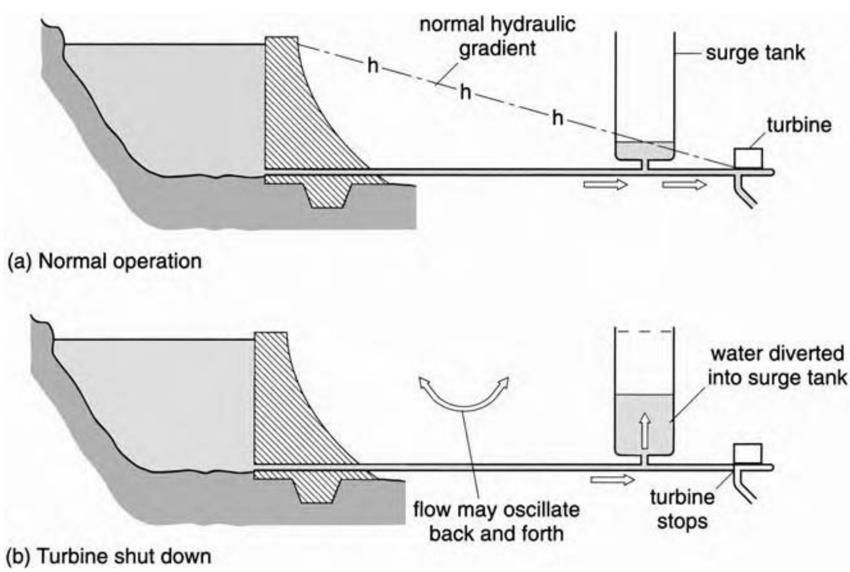
In some cases it is not always possible to avoid the sudden closure of a pipeline. For example, if a heavy vehicle drives over pipes laid out on the ground, such as might occur with fire hoses, it will squash them and stop the flow instantly. This will immediately cause the pressure to rise rapidly and this could burst the pipes. A similar situation can occur on farms where mobile irrigation machines use flexible pipes. It is easy for a tractor to accidentally drive over a pipe without realising that the resulting pressure rise can split open the pipeline and cause a lot of damage. In such situations where there are pipes on the ground and vehicles about, it is wise to use pipe bridges.

Some incidents though are not always easy to foresee. On a sprinkler irrigation scheme in east Africa pipe bridges were used to allow tractors to cross pipelines. But an elephant got into the farm and walked through the crop. It trod on an aluminium irrigation pipe and squashed it flat resulting in an instantaneous closure. This caused a massive pressure rise upstream and several pipes burst open!

4.15 Surge

Surge and water hammer are terms that are often confused because one is caused by the other. Surge is the large mass movement of water that sometimes takes place as a result of water hammer. It is much slower and can last for many minutes whereas water hammer may only last for a few seconds.

An example of the difference between the two can be most easily seen in a hydro-electric power station (Figure 4.18). Water flows down a pipeline from a large reservoir and is used to turn a turbine which is coupled to a generator that produces electricity. Turbines run at high speeds and require large quantities of water and so the velocities in the supply pipe can be very high. The demand for electricity can vary considerably over very short periods and problems occur when the demand falls and one or more of the turbines have to be shut down quickly. This is done by closing the valve on the supply pipe and this can cause water hammer. To protect a large part of the pipeline a *surge tank* is located as close to the power station as possible. This is a vertical chamber many times larger than the pipeline diameter. Water no longer required for the turbine is diverted into the tank and any water hammer shock waves coming up from



4.18 Surge in pipelines.

the valve closure are absorbed by the tank. Thus water hammer is confined to the pipeline between the turbine and the tank and so only this length of pipe needs to be constructed to withstand the high water hammer pressures. Gradually the tank fills with water and the flow from the reservoir slows down and eventually stops. Usually, the rushing water can cause the tank to overfill. In such cases water may flow back and forth between the tank and the reservoir for several hours. This slow but large movement of water is called *surge* and although it is the result of water hammer it is quite different in character.

Surge can also cause problems in pumping mains and these are discussed more fully in Section 8.14.

4.16 Some examples to test your understanding

1 A 150 mm pipeline is 360 m long and has a friction factor $\lambda = 0.02$. Calculate the head loss in the pipeline using the Darcy–Weisbach formula when the discharge is 0.05 m^3/s. Calculate the hydraulic gradient in m/100 m of pipeline (19.46 m; 5.4 m/100 m).

2 A pipeline 2.5 km long and 150 mm diameter supplies water from a reservoir to a small town storage tank. Calculate the discharge when the pipe outlet is freely discharging into the tank and the difference in level between the reservoir and the outlet is 15 m. Assume $\lambda = 0.04$ (0.012 m^3/s).

3 Water from a large reservoir flows through a pipeline, 1.8 km long and discharges into service tank. The first 600 m of pipe is 300 mm in diameter and the remainder is 150 mm in diameter. Calculate the discharge when the difference in water level between the two reservoirs is 25 m and $\lambda = 0.04$ for both pipes (0.02 m^3/s).

4 A venturi meter is fitted to a pipeline to measure discharge. The pipe diameter is 300 mm and the venturi throat diameter is 75 mm. Calculate the discharge in m^3/s when the difference in pressure between the pipe and the venturi throat is 400 mm of water. Assume the coefficient of discharge is 0.97 (0.07 m^3/s).

5 A pipeline is reduced in diameter from 500 mm to 300 mm using a concentric reducer pipe. Calculate the force on the reducer when the discharge is 0.35 m^3/s and the pressure in the 500 mm pipe is 300 kN/m^2 (37.4 kN).

6 A 500 mm diameter pipeline is fitted with a 90° bend. Calculate the resultant force on the bend if the normal operating pressure is 50 m head of water and the discharge is 0.3 m^3/s. Calculate the resultant force when there is no flow in the pipe but the system is still under pressure (138 kN; 137 kN).

7 Calculate the pressure rise in a 0.5 m diameter pipeline carrying a discharge of 0.3 m^3/s when a sluice valve is closed suddenly (187 m).

5 Channels

5.1 Introduction

Natural rivers and man-made canals are open channels. They have many advantages over pipes and have been used for many centuries for water supply, for transport and for agriculture. The Romans made extensive use of channels and built aqueducts for their sophisticated water supply schemes. Barge canals are still an important means of transporting heavy bulk materials in Europe and irrigation canals bring life and prosperity to the arid lands of North Africa, Middle East, India and Australia as they have done for thousands of years.

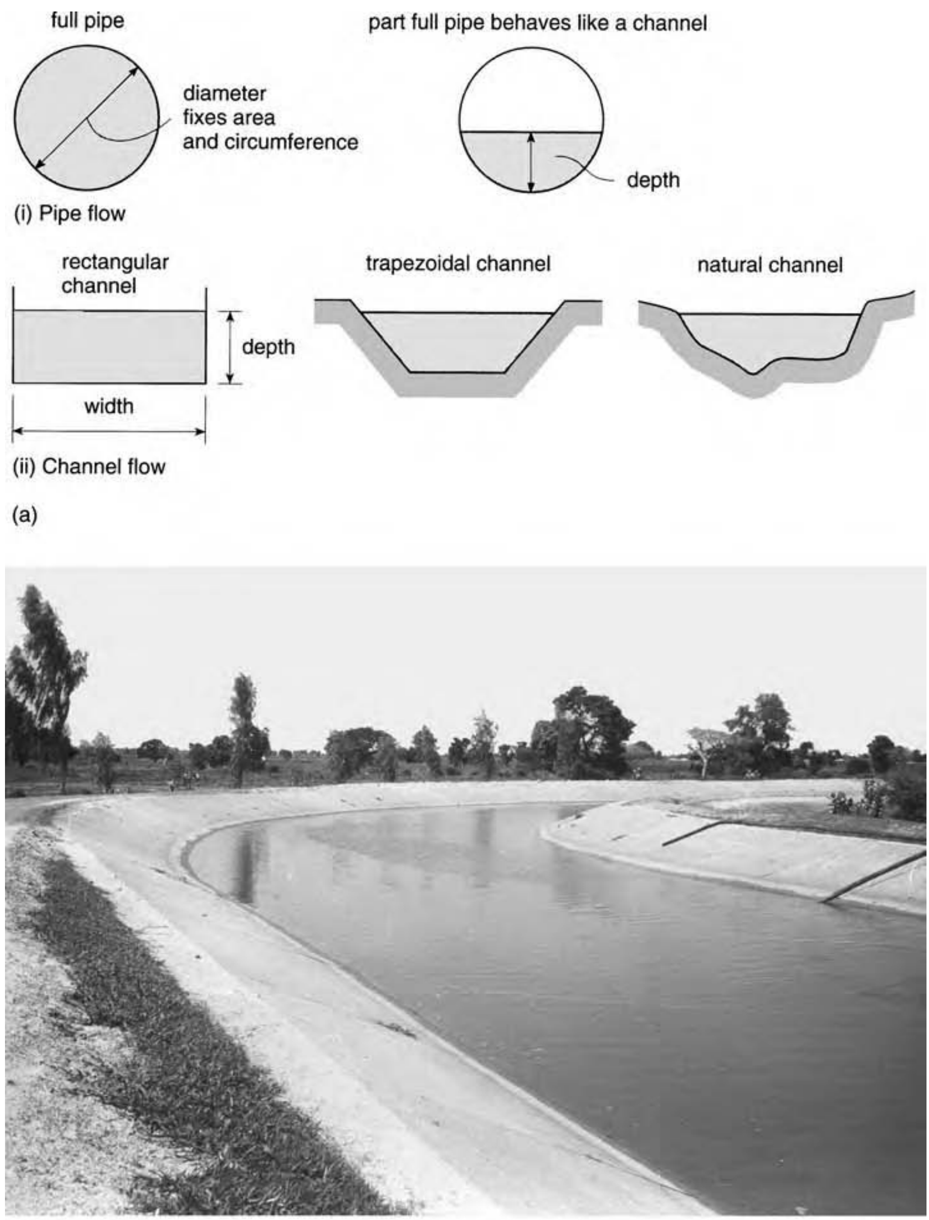
5.2 Pipes or channels?

The choice between pipes and open channels is most likely to depend on which provides the cheapest solution both in terms of capital expenditure and the recurrent costs of operation and maintenance. However, there are advantages and disadvantages with each which may influence or restrict the final choice.

Channels, for example, are very convenient and economical for conveying large quantities of water over relatively flat land such as in large irrigation systems on river flood plains. It is hard to imagine some of the large irrigation canals in India and Pakistan being put into pipes, although 5 m diameter pipes are used in Libya to transport water across the desert to coastal cities and irrigation schemes. In hilly areas the cost of open channels can rise significantly because the alignment must follow the land contours to create a gentle downward slope for the flow. A more direct route would be too steep causing erosion and serious damage to channels. Pipes would be more suitable in such conditions. They can be used in any kind of terrain and can take a more direct route. Water velocities too can be much higher in pipes because there is no risk of erosion.

Although there are obvious physical differences between channels and pipes, there are several important hydraulic differences between them:

- Open channels have a free or open water surface whereas pipes are enclosed and always flow full.
- Water can only flow downhill in channels but in pipes it can flow both uphill and downhill. Flow in pipes depends on a pressure difference between the inlet and outlet. As long as the pressure is higher at the inlet than at the outlet then water will flow even though the



(b) A lined irrigation canal in Nigeria

5.1 Channels can have many different shapes and sizes.

pipeline route may be undulating. Channels depend entirely on the force of gravity to make water move and so they can only flow downhill.

- Man-made channels can have many different shapes (circular, rectangular or trapezoidal) and sizes (different depths, widths and velocities). Natural river channels are irregular in shape (Figure 5.1). Pipes in contrast are circular in section and their shape is characterised by one simple dimension – the diameter. This fixes the area of the water way and the friction from the pipe circumference.

- Water velocities are usually lower in channels than in pipes. This is because channels are often in natural soils which erode easily. So channels are usually much larger than pipes for the same flow.
- Channels need much more attention than pipes. They tend to erode and weeds grow in waterways and so regular cleaning is required. Water losses from seepage and evaporation can also be a problem.

These differences make channels a little more complicated to deal with than pipes but most open channel problems can be solved using the basic tools of hydraulics; discharge and continuity, energy and momentum.

The study of open channels is not just confined to channel shapes and sizes. It can also include waves; the problem of handling varying flood flows down rivers and sediment transport associated with the scouring and silting of rivers and canals. Some of these issues are touched on in this chapter but waves are discussed more fully in Chapter 6.

5.3 Laminar and turbulent flow

Laminar and turbulent flow both occur in channels as well as in pipes. But for all practical purposes, flow in rivers and canals is turbulent and, like pipes, laminar flow is unlikely to occur except for very special conditions. For example, laminar flow only occurs in channels when the depth is less than 25 mm and the velocity is less than 0.025 m/s. This is not a very practical size and velocity and so laminar flow in channels can safely be ignored. The only time it does become important is in laboratory studies where physical hydraulic models are used to simulate large and complex channel flow problems. Scaling down the size to fit in the laboratory often means that the flow in the model becomes laminar. This change in flow regime will affect the results and care is needed when using them to assess what will happen in practice.

5.4 Using the hydraulic tools

Continuity and energy are particularly useful tools for solving open channel flow problems. Momentum is also helpful for problems in which there are energy losses and where there are forces involved.

5.4.1 Continuity

Continuity is used for open channels in much the same way as it is used for pipes (Figure 5.2a). The discharge Q_1 passing point 1 in a channel must be equal to the discharge Q_2 passing point 2.

$$Q_1 = Q_2$$

Writing this in terms of velocity and area:

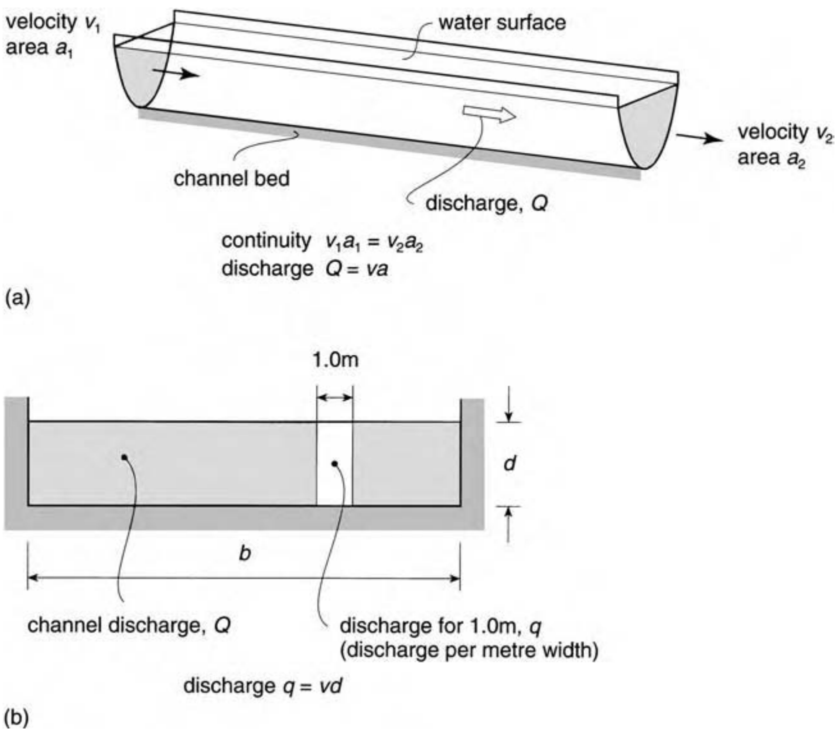
$$v_1 a_1 = v_2 a_2$$

The term discharge per unit width (q) is often used to describe channel flow rather than the total discharge (Q). This is the flow in a 1.0 m wide portion of a channel (Figure 5.2b).

To calculate the discharge per unit width (q) for a rectangular channel:

Use the continuity equation $Q = va$

where $a = bd$



5.2 Continuity in channels.

To calculate q assume $b = 1.0$ m. Therefore:

$$a = 1.0 \times d = d$$

And so:

$$q = vd$$

So when a channel width (b) is 7 m and it carries a discharge (Q) of 10 m^3/s, the flow per unit width is calculated as follows:

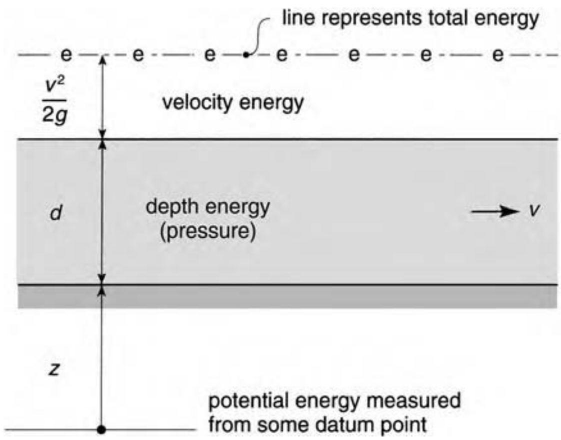
$$q = \frac{Q}{b}$$
$$= \frac{10}{7} = 1.43 \text{ m}^3\text{/s/m width of channel}$$

The discharge and continuity equations for channels are often written in terms of discharge per unit width as follows:

$$q = vd$$
$$v_1d_1 = v_2d_2$$

5.4.2 Energy

The idea of total energy being the same at different points in a system is also useful for channel flow. So when water flows between two points 1 and 2 in a channel, the total energy at 1



5.3 Energy in channels.

will be the same as the total energy at 2. As in the case of pipe flow this is only true when there is no energy loss. For some channel problems this is a reasonable assumption to make but for others an additional energy loss term is needed.

The energy equation for channel flow is a little different to the equation for pipe flow (Figure 5.3). For pipe flow the pressure energy is $p/\rho g$. For channel flow this term is replaced by the depth of water d. Remember that $p/\rho g$ is a pressure head and is already measured in metres and so for channels this is the same as the pressure on the channel bed resulting from the depth of water d. The potential energy z is measured from some datum point to the bed of the channel. The velocity energy $v^2/2g$ remains the same. Note that all the terms are measured in metres and so they can all be added together to determine the total energy in a channel.

Writing the total energy equation for an open channel:

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

Sometimes the velocity v is written in terms of the discharge per unit width q and depth d. This is done using the discharge equation:

$$q = vd$$

Rearrange this for velocity:

$$v = \frac{q}{d}$$

So the velocity energy becomes:

$$\frac{v^2}{2g} = \frac{q^2}{2gd^2}$$

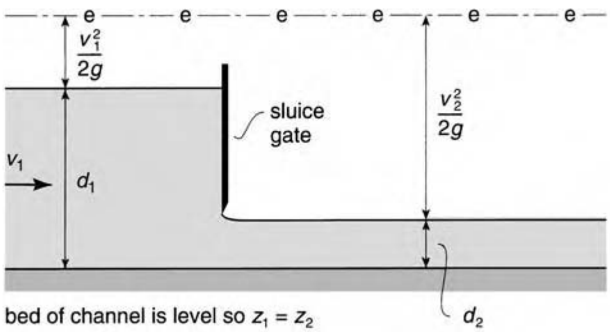
Substitute this in the energy equation:

$$\text{total energy} = d + \frac{q^2}{2gd^2} + z$$

Total energy can be represented diagrammatically by the *total energy line* e—e—e (Figure 5.3). This provides a visual indication of the total energy available and how it is changing. Notice that the line only slopes downwards in the direction of the flow to show the gradual loss of energy from friction. The water surface is the channel equivalent of the hydraulic gradient for pipes; it represents the pressure on the bed of the channel.

5.4.3 Using energy and continuity

One example of the use of the continuity and energy equations in an open channel is to calculate the discharge under a sluice gate (Figure 5.4). The sluice gate is a common structure for controlling flows in channels and it can also be used to measure flow if the water depths upstream



(a)

(b) Sluice gates controlling discharge into irrigation canal, Iraq

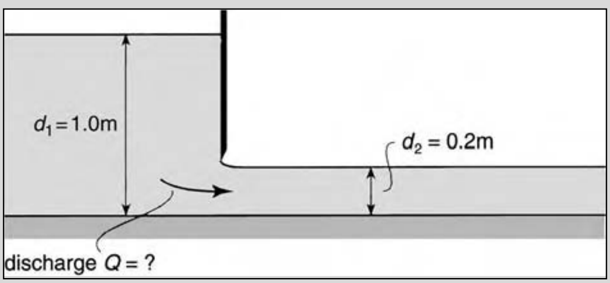
5.4 Using energy and continuity.

and downstream of the gate are measured. The approach is very similar to the venturi problem in pipe flow but in this case a gate is used to change the energy conditions in the channel. Notice how the energy line has been drawn to indicate the level of total energy. Firstly it shows there is no energy loss as water flows under the gate. This is reasonable because the flow is converging under the gate and this tends to suppress turbulence which means little or no energy loss. Secondly it shows that there is a significant change in the components of the total energy across the gate even though the total is the same. Upstream the flow is slow and deep whereas downstream the flow is very shallow and fast. The discharge is the same on both sides of the gate but it is clear that the two flows are quite different. In fact they behave quite differently too – but more about this later in Section 5.7.

The example in the box illustrates how to calculate the discharge under a sluice gate when the upstream and downstream water depths are known.

EXAMPLE: CALCULATING DISCHARGE UNDER A SLUICE GATE

A sluice gate is used to control and measure the discharge in an open channel. Calculate the discharge in the channel when the upstream and downstream water depths are 1.0 m and 0.2 m respectively.



5.5 Calculating discharge under a sluice gate.

When the flow is contracting as it does under a sluice gate, turbulence is suppressed and the flow transition occurs smoothly. Very little energy is lost and so the energy equation can be applied as follows:

total energy at point 1 = total energy at point 2

$$d_1 + \frac{v_1^2}{2g} + z_1 = d_2 + \frac{v_2^2}{2g} + z_1$$

As the channel is horizontal:

$$z_1 = z_2$$

And so:

$$d_1 + \frac{v_1^2}{2g} = d_2 + \frac{v_2^2}{2g}$$

Bring the d terms and v terms together and put in the values for depth:

$$1.0 - 0.2 = \frac{v_2^2}{2g} - \frac{v_1^2}{2g}$$

Both velocities v_1 and v_2 are unknown and so the continuity equation is needed to solve the problem:

$$v_1 d_1 = v_2 d_2$$

Put in the depths:

$$v_1 \times 1.0 = v_2 \times 0.2$$

And so:

$$v_1 = 0.2 v_2$$

Substitute for v_1 in the energy equation:

$$0.8 = \frac{v_2^2}{2g} - 0.04 \frac{v_2^2}{2g}$$

Rearrange this to find v_2:

$$v_2^2 = \frac{0.8 \times 2 \times 9.81}{1 - 0.04} = 16.35$$
$$v_2 = 4 \text{ m/s}$$

Calculate the discharge:

$$q = v_2 d_2$$
$$q = 4 \times 0.2$$
$$q = 3.27 \text{ m}^3\text{/s/m width of channel.}$$

5.4.4 *Taking account of energy losses*

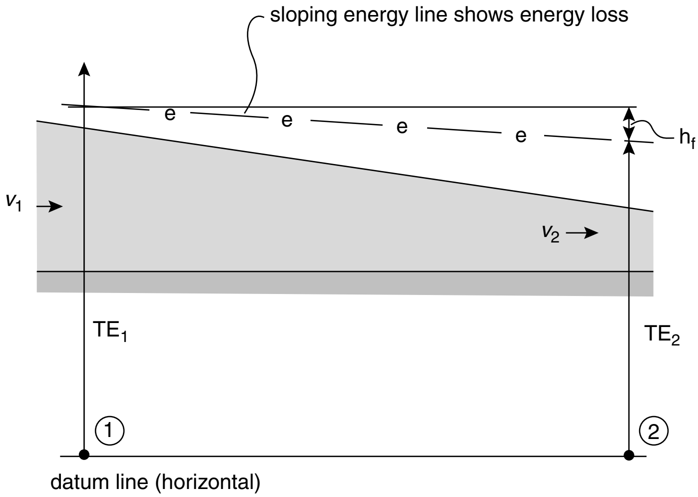
Energy loss occurs in channels due to friction. In short lengths of channel such as in the sluice gate example this is very small and so it is not taken into account in any calculations. But energy loss in long channels must be taken into account in the energy equation to avoid serious errors (Figure 5.6). Writing the energy equation for two points in a channel:

total energy at 1 (TE_1) = total energy at 2 (TE_2) + h_f

$$d_1 + \frac{v_1^2}{2g} + z_1 = d_2 + \frac{v_2^2}{2g} + z_2 + h_f$$

where h_f is energy loss due to friction (m).

This equation is very similar to that for pipe flow and in that case the Darcy-Weisbach formula was used to calculate the energy loss term h_f. This was the link between energy losses and pipe size. Similar formulae have been developed to calculate h_f for open channels and these link energy loss both to the size and shape of channels needed to carry a given discharge. But as with pipe flow there are a few important steps to take before getting to the formula. The first of these steps is the concept of uniform flow.



5.6 Energy losses in channels.

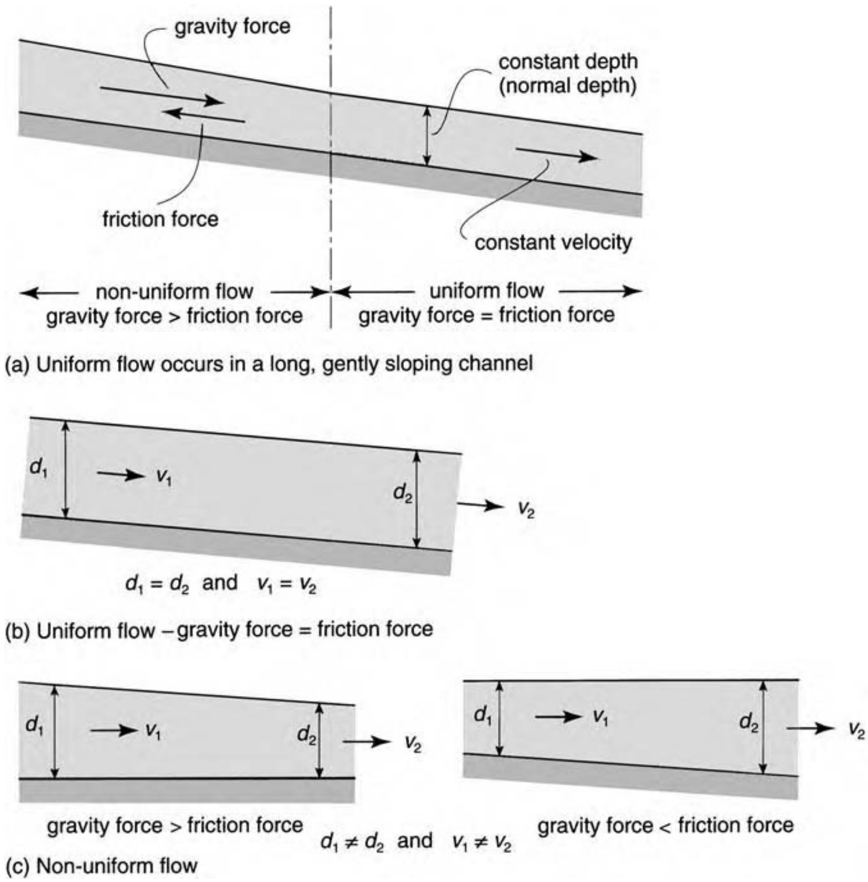
5.5 Uniform flow

All the energy loss formulae for open channels are based on *uniform flow*. This is a special condition that only occurs when water flows down a long, straight, gently sloping channel (Figure 5.7). The flow is pulled down the slope by the force of gravity but there is friction from the bed and sides of the channel slowing it down. When the friction force is larger than the gravity force it slows down the flow. When the friction force is smaller than the gravity force the flow moves faster down the slope. But the friction force is not constant, it depends on velocity and so as the velocity increases so does the friction force. At some point the two forces become equal. Here the forces are in balance and as the flow continues down the channel the depth and velocity remain constant. This flow condition is called *uniform flow* and the water depth is called the *normal depth*.

Unfortunately, in most channels this balance of forces rarely occurs and so the depth and velocity are usually changing gradually even though the discharge is constant. Even in long channels where uniform flow has a chance of occurring there is usually some variation in channel shape or slope or a hydraulic structure which changes the depth and the velocity (Figure 5.7c). So most channels have *non-uniform flow*. It is also called *gradually varied flow* because the changes take place gradually along the channel.

So if non-uniform flow is the most common and uniform flow rarely occurs, why are all the formulae based on uniform flow? Why not accept that non-uniform flow is the norm and develop formulae for this condition? The answer is quite simple. Uniform flow is much easier to deal with from a calculation point of view and engineers are always looking for ways of simplifying problems but without losing accuracy. There are methods of designing channels for non-uniform flow but they are much more cumbersome to use and usually they produce the same shape and size of channel as uniform flow methods. So it has become accepted practice to assume that channel flow is uniform for all practical purposes. For all gently sloping channels on flood plains (i.e. 99% of all channels including rivers, canals and drainage ditches) this assumption is a good one. Only in steep sloping channels in the mountains does it cause problems.

Remember too that great accuracy may not be necessary for designing channels and dimensions do not need to be calculated to the nearest millimetre for construction purposes. The nearest 0.05 m is accurate enough for concrete channels and probably 0.1 m for earth channels is more than enough. From a practical construction point of view it will be difficult to



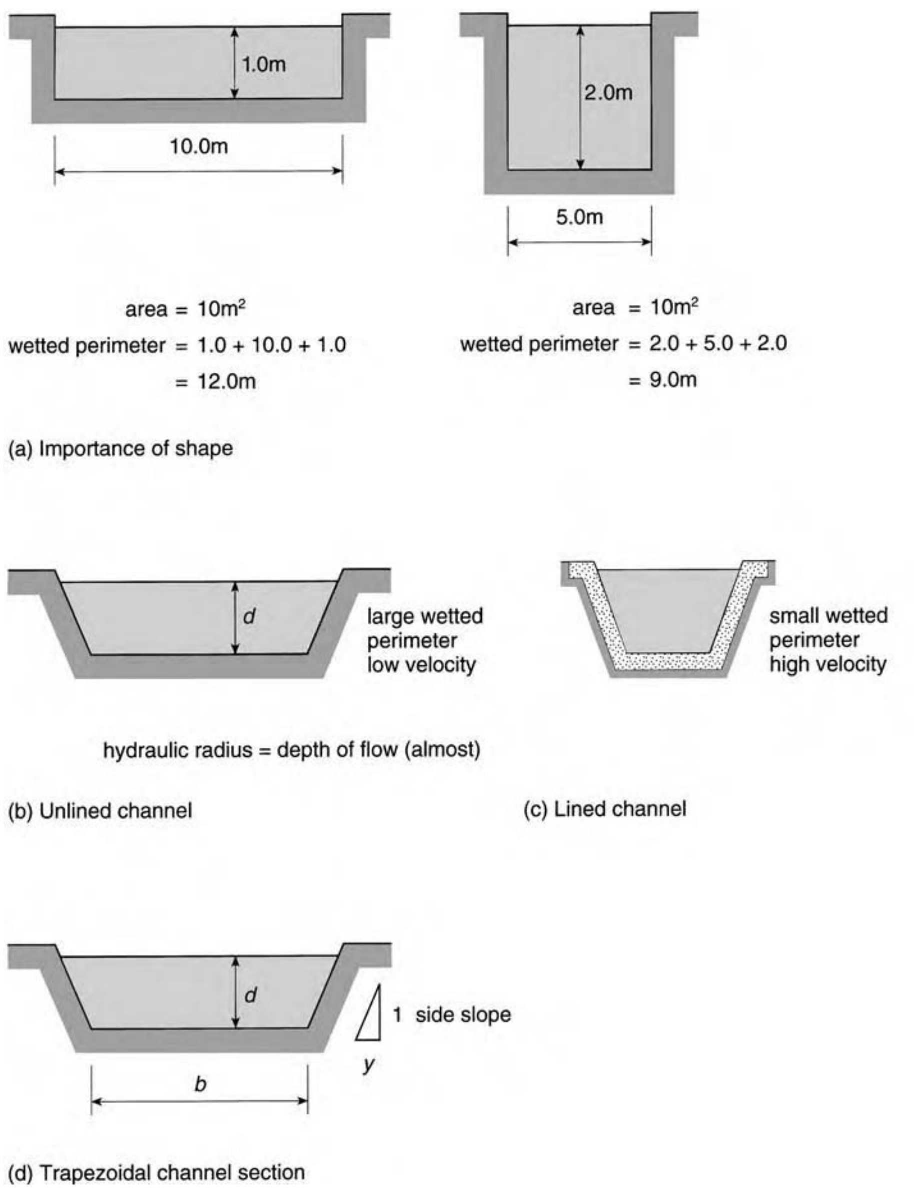
5.7 Uniform and non-uniform flow.

find a hydraulic excavator operator who can (or who would want to) trim channel shapes to an accuracy greater than this.

5.5.1 Channel shapes

For pipe flow the issue of shape does not arise. Pipes are all circular in section and so their hydraulic shape is determined by one dimension – the pipe diameter – and so both shape and size are taken into account in the Darcy-Weisbach formula. But channels come in a variety of shapes, the more common ones being rectangular, trapezoidal or semi-circular (Figure 5.8). They also come in different sizes with different depths and widths. So any formula for channels must take into account both shape and size.

To demonstrate the importance of shape consider two rectangular channels each with the same area of flow but one is narrow and deep and the other is shallow and wide (Figure 5.8a). Both channels have the same flow area of 10 m^2 and so they might be expected to carry the same discharge. But this is not the case. Friction controls the velocity in channels and when the friction changes the velocity will also change. The channel boundary in contact with the water (called the *wetted perimeter*) is the main source of friction. In the narrow channel the length of the boundary in contact with the water is 9 m whereas in the wide channel it is 12 m. So the



5.8 Channel shapes.

wide channel produces more friction than the narrow one and as a result the velocity (and hence the discharge) in the wide channel will be less than in the narrow one. So channels with the same flow area have different carrying capacities depending on their shape.

Understanding this can be very useful when deciding on the general shape of channels. Suppose you are constructing a channel in natural soil and there are worries about erosion of the bed and sides from a high water velocity. By making the channel wide and shallow the increased friction will slow down the water and avoid the problem (Figure 5.8b). Many natural river channels have a shallow, wide profile as they have adapted over many years to the erosion of the natural soils in which they flow.

Table 5.1 Minimum wetted perimeters for different channel shapes.

Channel shape	*Wetted perimeter*	*Hydraulic radius*
Rectangle	$4d$	$0.5d$
Trapezoid (half a hexagon)	$3.463d$	$0.5d$
Semi-circle	πd	$0.5d$

Note
d is the depth of flow.

For lined channels (e.g. concrete) the main issue is one of cost and not erosion. Lined canals are very expensive and so it is important to minimise the amount of lining needed. This is done by using a channel shape that has a small wetted perimeter (Figure 5.8c). Friction will be low which means the velocity will be high but this is not a problem as the lining will resist erosion. Minimum wetted perimeters for selected channel shapes are shown in Table 5.1.

5.5.2 Factors affecting flow

Channel flow is influenced not just by the channel shape and size but also by slope and roughness.

5.5.2.1 Area and wetted perimeter

The cross-sectional area of a channel (a) defines the flow area and the wetted perimeter (p) defines the boundary between the water and the channel. This boundary in contact with the water is the source of frictional resistance to the flow of water. The greater the wetted perimeter, the greater is the frictional resistance of the channel.

The area and wetted perimeter for rectangular and circular channels are easily calculated but trapezoidal channels are a bit more difficult. Unfortunately they are the most common and so given below are formulae for area and wetted perimeter (Figure 5.8d).

Area of waterway (a) $= (b + yd)d$
Wetted perimeter (p) $= b + 2d\,(1 + y^2)^{1/2}$

Where b is bed width (m); d is depth of flow (m); y is side slope.

5.5.2.2 Hydraulic radius

As the wetted perimeter can vary considerably for the same area, some measure of the hydraulic shape of a channel is needed. This is called the *hydraulic radius* and it is determined from the area and the wetted perimeter as follows:

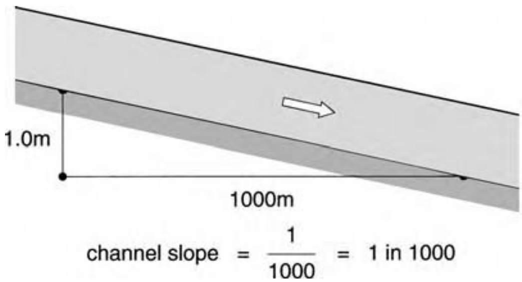
$$\text{Hydraulic radius (m)} = \frac{\text{area (m}^2\text{)}}{\text{wetted perimeter (m)}}$$

In the two channels in Figure 5.8, the hydraulic radius would be 1.11 m and 0.83 m respectively and this shows numerically just how hydraulically different are the two channels.

5.5.2.3 Slope

Water only flows downhill in channels and the steepness of the slope affects the velocity and hence the discharge. As the slope gets steeper the velocity increases and so does the discharge.

(remember $Q = va$)



5.9 Channel slope.

Slope is measured as a gradient rather than an angle in degrees. So a channel slope is expressed as 1 in 1000, that is, 1.0 m drop in 1000 m of channel length (Figure 5.9).

Slopes need not be very steep for water to flow. Many of the irrigation canals in the Nile valley in Egypt have slopes of only 1 in 10 000. This is the same as 1.0 m in 10 km or 100 mm per kilometre. This is a very gentle slope and you would not be able to detect it by just looking at the landscape, but it is sufficient to make water flow as the evidence of the Nile shows.

A question which sometimes arises about channel slopes is – does slope refer to the water surface slope, the channel bed or the slope of the energy line? For uniform flow the question is irrelevant because the depth and velocity remain the same along the entire channel and so the water surface and the bed are parallel and have the same slope. For non-uniform flow it is the slope of the energy line that is important as is the driving force for the flow. Even when the bed is flat, water will still flow provided there is an energy gradient.

5.5.2.4 Roughness

The *roughness* of the bed and sides of a channel also contribute to friction. The rougher they are the slower will be the water velocity. Channels tend to have much rougher surfaces than pipes. They may be relatively smooth when lined with concrete but they can be very rough when excavated in the natural soil or infested with weeds. Roughness is taken into account in channel design formula and this is demonstrated in the next section.

5.5.3 Channel design formulae

There are two commonly used formulae which link energy loss in channels to their size, shape, slope and roughness: the *Chezy formula* and the *Manning formula*. Both are widely used and were developed on the assumption that the flow is uniform.

5.5.3.1 Chezy formula

This formula was developed by Antoine Chezy, a French engineer who, in 1768, was asked to design a canal for the Paris water supply.

The Chezy formula, as it is now known, is usually written as follows:

$$v = C\sqrt{RS}$$

where v is velocity (m/s); R is hydraulic radius (m); S is channel slope (m/m); C is the Chezy coefficient describing channel roughness.

This may not look much like a formula for friction loss in a channel but it is derived from the energy equation allowing for energy loss in the same way as was done for pipe flow. For pipe flow the outcome was the Darcy-Weisbach formula, for channel flow the outcome is the Chezy formula. For those interested in the origins of this formula, which are interesting both from a mathematical and historical point of view, a derivation is shown in the box. The formula shown above is the more familiar way of presenting the Chezy formula in hydraulic textbooks.

Once the velocity has been calculated the discharge can be determined using the discharge equation:

$$Q = va$$

DERIVATION: CHEZY FORMULA

To see how Chezy developed his formula look first at the energy equation describing the changes which take place between two points in a long channel (Figure 5.7b) but taking into account energy loss in the channel h_f.

$$d_1 + \frac{v_1^2}{2g} + z_1 = d_2 + \frac{v_2^2}{2g} + z_2 + h_f$$

He suggested that the energy loss h_f could be determined by:

$$h_f = \frac{Lv^2}{C^2R}$$

where L is the length of channel over which the energy loss occurs (m); v is velocity (m/s); R is hydraulic radius (m); C is Chezy coefficient describing roughness.

Notice how similar this equation is to the Darcy-Weisbach equation. Friction depends on the length and the square of the velocity. The Chezy coefficient C describes the friction in the channel and is similar to λ in the Darcy-Weisbach formula. Like λ it does not have a constant value. C depends on the Reynolds Number and also on the dimensions of the channel.
Put this into the energy equation:

$$d_1 + \frac{v_1^2}{2g} + z_1 = d_2 + \frac{v_2^2}{2g} + z_2 + \frac{Lv^2}{C^2R}$$

Now uniform flow is defined by the depths and velocities remaining the same along the whole length of a channel and so:

$$d_1 = d_2$$

And:

$$v_1 = v_2$$

This reduces the energy equation to:

$$\frac{Lv^2}{C^2R} = z_1 - z_2$$

Now divide both sides of the equation by L:

$$\frac{v^2}{C^2R} = \frac{z_1 - z_2}{L}$$

But $\frac{z_1 - z_2}{L}$ is the slope of the channel bed *S*. It is also the slope of the water surface. Remember that the two are parallel for uniform flow. Hence:

$$\frac{z_1 - z_2}{L} = S$$

And so:

$$\frac{v^2}{C^2 R} = S$$

Rearrange this to calculate the velocity:

$$v = C\sqrt{RS}$$

This is the familiar form of the Chezy equation that is quoted in hydraulic text books.

5.5.3.2 Manning formula

The Manning formula is an alternative to Chezy and is one of the most commonly used formulae for designing channels. This was developed by Robert Manning (1816–1897) an Irish civil engineer. It is an empirical formula developed from many observations made on natural channels.

$$v = \frac{R^{2/3} S^{1/2}}{n}$$

where *v* is velocity (m/s); *R* is hydraulic radius (m); *S* is channel bed slope (m/m); *n* is Manning's roughness coefficient.

Manning's *n* values depend on the surface roughness of a channel. Typical values are listed in Table 5.2.

The value of Manning's *n* is not just determined by the material from which the channel is made but it is also affected by vegetation growth. This can make it difficult to determine with any accuracy. The *n* value can also change over time as weeds grow and it can also change with changes in flow. At low discharges weeds and grasses will be upright and so cause great roughness but at higher discharges they may be flattened by the flow and so the channel becomes much smoother. There is an excellent book, *Open Channel Flow* by Ven Te Chow (see references), which has a series of pictures of channels with different weed growths and suggested *n* values. These pictures can be compared with existing channels to get some indication of *n*. But

Table 5.2 Values of Manning's *n*.

Channel type	*Manning's n values*
Concrete lined canals	0.012–0.017
Rough masonry	0.017–0.030
Roughly dug earth canals	0.025–0.033
Smooth earth canals	0.017–0.025
Natural river in gravel	0.040–0.070

how is Manning's n selected for a natural, winding channel with varying flow areas; with trees and grasses along its banks (perhaps also including the odd bicycle or super-market trolley) and flowing under bridges and over weirs? Clearly, in this situation, choosing n is more of an art than a science. It may well be that several values are needed to describe the roughness along different sections of the river.

5.5.4 Using Manning's formula

Manning's formula is not the easiest of formula to work with. It is quite straightforward to use when calculating discharge for a given shape and size of channel but it is not so easy to use the other way round, that is, to calculate channel dimensions for a given discharge. Unfortunately this is by far the most common use of Manning. One approach is to use a trial and error technique to obtain the channel dimensions. This means guessing suitable values for depth and width and then putting them into the formula to see if they meet the discharge requirements. If they do not then the values are changed until the right dimensions are found. Usually there can be a lot of trials and a lot of errors. Modern computer spreadsheets can speed up this painful process.

Another approach for those who do not have spreadsheet skills is the method developed by HW King in his *Handbook for the Solution of Hydrostatic and Fluid Flow Problems* (see references). This is a very simple and useful method and is ideally suited to designing trapezoidal channels, which are the most common. He modified Manning's formula to look like this:

$$Q = \frac{1}{n} jkd^{8/3} S^{1/2}$$

where d is depth of flow (m); S is channel slope; j and k are constants.

The values of j and k depend on the ratio of the bed width to depth and the channel side slope. This is the slope of the side embankments and not the longitudinal slope of a channel S. King's book has a very comprehensive range of j and k values. A selection of the most common values is shown in Table 5.3.

To use the method, values of the ratio of channel bed width to depth and side slope are first chosen. Values of j and k are then obtained from Table 5.3 and put into the formula from which a value of depth can be calculated. As the bed width to depth ratio is known, the bed width can now be calculated. If the resulting channel shape or its dimensions appear to be unsuitable for any reason (e.g. the velocity may be too high) then another ratio of bed width to depth ratio can be chosen and the calculation repeated. As well as providing j and k values, King also

Table 5.3 Values of j and k for Manning's formula.

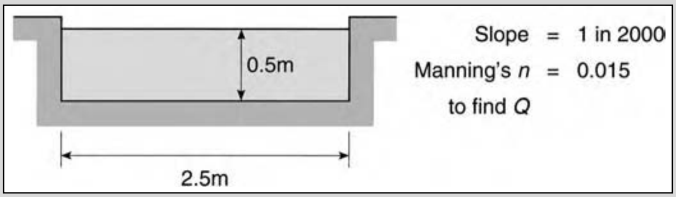
	Ratios of bed width (b) to water depth (d)					
	Values of j			*Values of k*		
Side slope	*b = d*	*b = 2d*	*b = 3d*	*b = d*	*b = 2d*	*b = 3d*
vertical	1	2	3	0.48	0.63	0.71
1 in 1	2	3	4	0.64	0.73	0.77
1 in 1.5	2.5	3.5	4.5	0.66	0.73	0.77
1 in 2	3	4	5	0.66	0.72	0.76
1 in 3	4	5	6	0.67	0.71	0.74

supplies values of the power function 8/3 so that it is easy to calculate the depth of flow. Remember that his book was written some years ago before everyone had a calculator.

Examples of the use of Manning's formula are shown in the boxes.

EXAMPLE: CALCULATING DISCHARGE USING MANNING'S FORMULA

Calculate the discharge in a rectangular concrete lined channel of width 2.5 m and depth 0.5 m with a slope of 1 in 2000 and a Manning's *n* value is 0.015.



5.10 Calculating discharge using Manning's equation.

The first step is to calculate the velocity but before this can be done the area, wetted perimeter and hydraulic radius must be determined:

Area *(a)* = depth × width
= 0.5 × 2.5 = 1.25 m²

Wetted perimeter *(p)* = 0.5 + 2.5 + 0.5 = 3.5 m

And so:

$$\text{Hydraulic radius } (R) = \frac{a}{p}$$

$$= \frac{1.25}{3.5} = 0.36 \text{ m}$$

Next calculate velocity:

$$v = \frac{0.36^{2/3} \times \left(\frac{1}{2000}\right)^{1/2}}{0.015}$$

$$= 0.75 \text{ m/s}$$

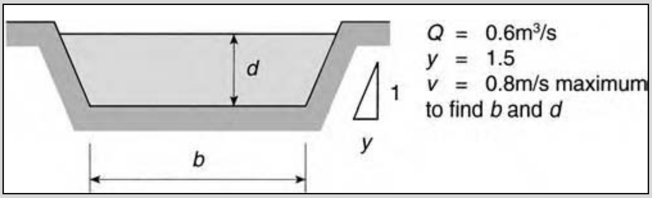
Now calculate discharge:

$Q = va$

$Q = 1.25 \times 0.75 = 0.94 \text{ m}^3/\text{s}$

EXAMPLE: CALCULATING DEPTH OF FLOW AND BED WIDTH USING MANNING'S FORMULA (KING'S METHOD)

Calculate a suitable bed width and depth of flow for an unlined trapezoidal channel to carry a discharge of 0.6 m^3/s on a land slope of 1 in 1000. The soil is a clay loam and so the side slope will be stable at 1.5:1 and the maximum permissible velocity is 0.8 m/s (Table 5.4).



5.11 Calculating depth of flow and bed width using King's method.

The first step is to select a suitable value for Manning's n ($n = 0.025$ for natural soil) and then select a bed width to depth ratio (try $b = d$).

Now obtain values for j and k from Table 5.3.

$j = 2.5$ and $k = 0.66$

Calculate d using the Manning formula:

$$Q = \frac{1}{n} jkd^{8/3}s^{1/2}$$

$$0.6 = \frac{1}{0.025} \times 2.5 \times 0.66 \times d^{8/3} \times 0.001^{1/2}$$

Rearrange this for d:

$$d^{8/3} = 0.286$$
$$d = 0.63 \text{ m}$$

As the ratio of bed width to depth is known calculate b. In this case:

$$b = d$$

And so:

$$b = 0.63 \text{ m}$$

All the channel dimensions are now known but do they comply with the velocity limit? Check the velocity using the discharge equation:

$$Q = av$$

For a trapezoidal channel:

$$a = (b + yd)d$$
$$= (0.63 + 1.5 \times 0.63)\ 0.63$$
$$= 0.993 \text{ m}^2$$

Substitute this and the value for discharge into the discharge equation:

$$0.6 = 0.993 \times v$$

Calculate velocity:

$$v = 0.6 \text{ m/s}$$

This is less than the maximum permissible velocity of 0.8 m/s and so these channel dimensions are acceptable.

Note that there are many different channel dimensions that could be chosen to meet the design criteria. This is just one answer. Choosing another *b*:*d* ratio would produce different dimensions but they would be acceptable provided they met the criteria. Increasing the *b*:*d* ratio would reduce the velocity whereas decreasing the *b*:*d* ratio would increase the velocity. The latter would not be an option in this example as the velocity is close to the maximum permissible already.

A freeboard would normally be added to this to ensure that the channel is not over-topped.

5.5.5 Practical design

In engineering practice the usual design problem is to determine the size, shape and slope of a channel to carry a given discharge. There are many ways to approach this problem but here are some guidelines.

Whenever possible the channel slope should follow the natural land slope. This is done for cost as it helps to reduce the amount of soil excavation and embankment construction needed. But when the land slope is steep, high water velocities may occur and cause erosion in unlined channels. The most effective way to avoid erosion is to limit the velocity. Maximum non-scouring velocities for different soil types are shown in Table 5.4. Channels can also be lined for protection. Slope can also be reduced to lower the velocity to an acceptable level by using drop structures to take the flow down the slope in a series of steps – like a staircase.

It is important to understand that there is no single correct answer to the size and shape of a channel, but a range of possibilities. If three people were each asked to design a trapezoidal channel for a given discharge, it is likely that they would come up with three different answers, and all could be acceptable. It is the designer's job to select the most appropriate one. Usually the selection is made simpler because of the limited range of values that are practicable. For example, land slope will limit the choice of slope and the construction materials will limit the velocity. But even within these boundaries there are still many possibilities.

One of the problems of channel design is that of choosing suitable values of depth and bed width. King's method gets around this problem by asking the designer to select a ratio between them rather than the values themselves. Another way to simplify the problem is to assume that the hydraulic radius *R* is equal to the depth of the water *d*. This is a reasonable assumption to make when the channel is shallow and wide. Referring to the example in Figure 5.8a, the hydraulic radius was 1.11 m in the wider channel when the depth was 1.0 m. This is close enough for channel design purposes. The depth can then be calculated using the Manning formula and the bed width determined using the area and discharge.

Table 5.4 Maximum permissible velocities.

Material	*Maximum velocity (m/s)*
Silty sand	0.30
Sandy loam	0.50
Silt loam	0.60
Clay loam	0.8
Stiff clay	1.10

The depth and width of a channel also influence velocity. For lined channels, which are expensive, it is important to keep the wetted perimeter (*p*) as small as possible as this keeps the cost down. This results in channels which are narrow and deep (Figure 5.8c). For unlined channels the velocity must be kept well within the limits set in Table 5.4. Making channels wide and shallow increases the wetted perimeter and channel resistance and this slows down the flow (Figure 5.8b). Look at any stream or river flowing in natural soil. Unless it is constrained by rocks or special training works it will naturally flow wide and shallow. So new channels which are to be constructed in similar material should also follow this trend.

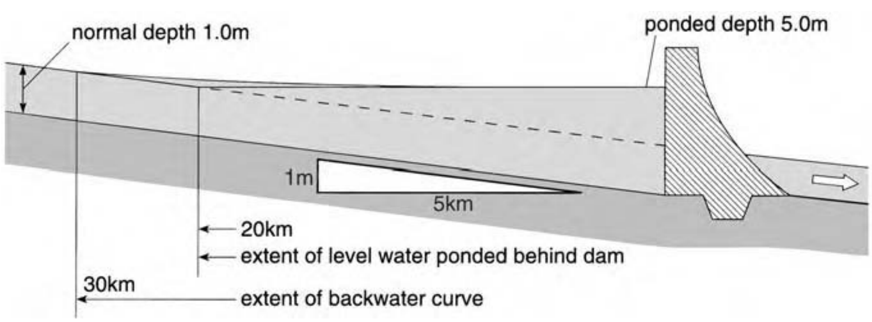
5.6 Non-uniform flow: gradually varied

There are two kinds of non-uniform flow. The first is *gradually varied flow*. This is the most common type of flow and has already been described earlier in this chapter. It occurs when there are gradual changes taking place in the depth and velocity due to an imbalance of the force of gravity trying to make the flow go faster down a slope and the channel friction slowing it down. The gradual changes in depth take place over long distances and the water surface follows a gradual curve.

Engineers recognise 12 different surface water curves depending on the different gradually varied flow conditions that can occur in channels but the most common is the *backwater curve*. This occurs when a channel is dammed (Figure 5.12). For example, a river flowing at a normal depth of 1.0 m down a gradient of 1 m in 5 km is dammed so that the water level rises to a depth of 5.0 m. For a level water surface behind the dam its influence extends 20 km upstream. But because the river is flowing there is a backwater curve which extends the influence of the dam up to 30 km. This effect can be important for river engineers who wish to ensure that a river's embankments are high enough to contain flows and for landowners along a river whose land may be flooded by the dam construction. The backwater curve can be predicted using the basic tools of hydraulics but they go beyond the scope of this book. One problem is that they depend largely for their accuracy on predicting the value of Manning's *n* which can be very difficult in natural channels.

5.7 Non-uniform flow: rapidly varied

The second type of non-uniform flow is *rapidly varied flow*. As its name implies, sudden changes in depth and velocity occur and this is the result of sudden changes in either the shape or size of channels. The change usually takes place over a few metres, unlike gradually varied flow where changes take place slowly over many kilometres. Hydraulic structures are often the cause of rapidly varied flow and the sluice gate in Section 5.4.3 is a good example of this. In this case



5.12 Backwater curve.

the gate changed the flow suddenly from a deep, slow flow upstream to a fast, shallow flow downstream. Building a weir or widening (or deepening) a channel will also cause sudden changes to occur. But unfortunately, all flows do not behave in the same way. For example, a weir in a channel will have quite a different effect on the deep, slow flow than on the shallow, fast flow. So a further classification of channel flow is needed, this time in terms of how flow behaves when channel size or shape is changed suddenly.

The two contrasting types of flow described above are now well recognised by engineers. The more scientific name for deep, slow flow is *sub-critical flow* and for shallow, fast flow is *super-critical flow*. This implies that there is some *critical point* when the flow changes from one to the other and that this point defines the difference between the two flow types. This is indeed the case. At the critical point the velocity becomes the *critical velocity* and the depth becomes the *critical depth*. The critical point is important not just to classify the two flow types. It plays an important role in the measuring discharges in channels. This is discussed later in Chapter 8.

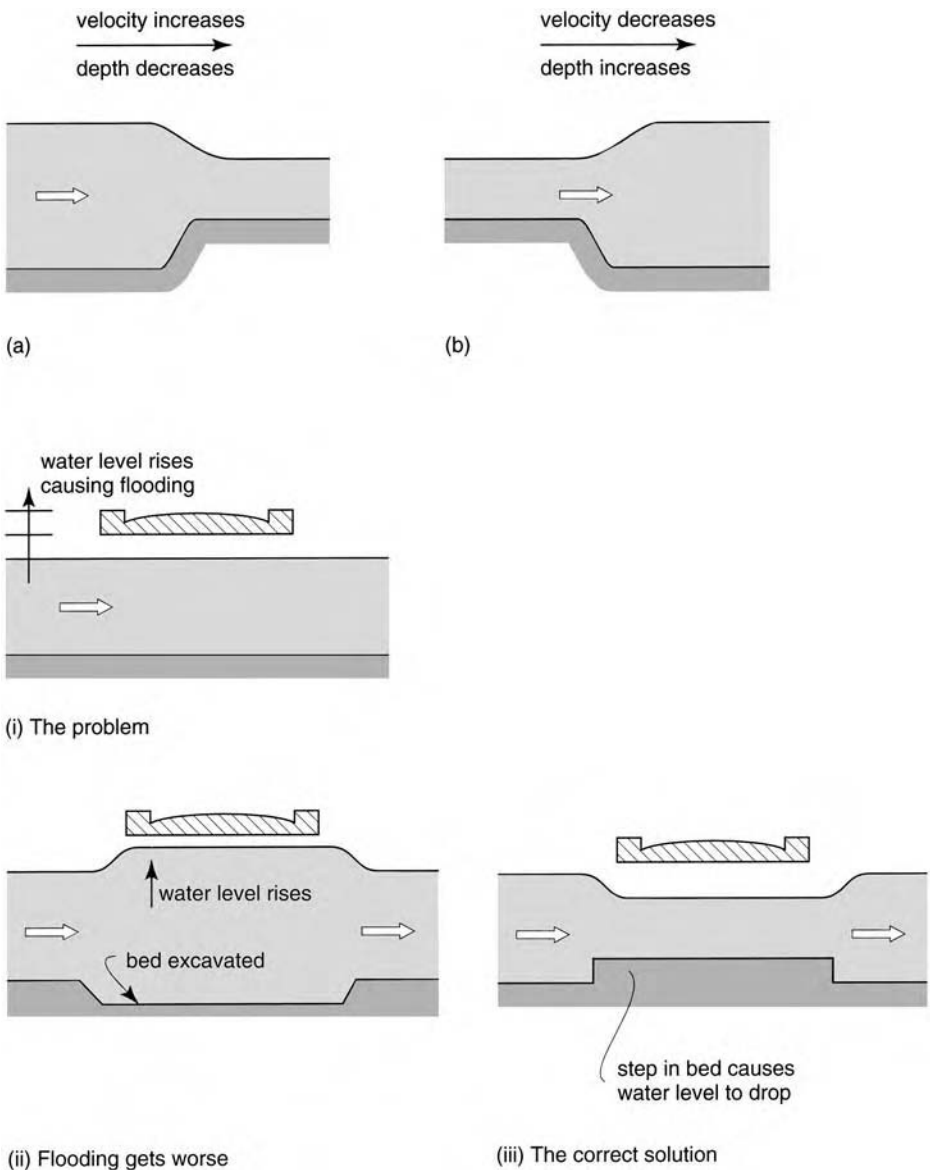
5.7.1 Flow behaviour

Just how do sub-critical and super-critical flows behave when there are sudden changes in the channel?

5.7.1.1 Sub-critical flow

This is by far the most common flow type and is associated with all natural and gently flowing rivers and canals. The effect of a sluice gate on this kind of flow has already been described. The effect of a weir is very similar (Figure 5.13a). A weir is like a step up on the bed of a channel. Such a step causes the water level to drop and the velocity to increase. The step up reduces the flow area in a channel but the water does not slow down because of this. Its velocity increases (remember the way flow behaves in constrictions – Section 3.7.1). This increases the kinetic energy, but as there is no change in the total energy this is at the expense of the depth (pressure) energy. So the depth is reduced causing a drop in the water level. Weirs are a common sight on rivers and most people will have seen this sudden but smooth drop in water level over a weir. A similar, though not so dramatic, drop in water level occurs when water flows under a bridge. The water level drops because the reduced width of the river increases its velocity.

Interestingly, the converse is true. When a channel is made larger by increasing its width or depth, the water level rises. This is not so easy to believe. But it follows from the energy equation and it actually happens in practice (Figure 5.13b). This was highlighted by a problem facing engineers who were troubled by flooding from a river flowing through a town and under the town bridge (Figure 5.13c). During stormy weather, the river level rises and reaches the underside of the bridge. The extra friction from the bridge slows the flow causing the water level upstream of the bridge to rise even further and flood the town. The problem was how to increase the carrying capacity of the river through the town, and particularly under the bridge, to avoid the flooding. The engineers decided that the most obvious solution was to make the channel deeper – but this made flooding worse, not better. Clearly the engineers did not understand the hydraulic tools of continuity and energy. The increase in channel depth reduced the velocity and hence the kinetic energy. As the total energy remained the same, the depth energy increased causing the river level to rise and not fall as expected. They eventually opted for the correct solution which was to reduce the flow area under the bridge by constructing a step on the bed of the river. This increased the velocity energy and reduced the depth of flow. So even when the river was in flood, the flow was able to pass safely under the bridge. The local engineers did not believe the solution at first and insisted on building a hydraulic model in the laboratory to test it. Seeing is believing and this convinced them!

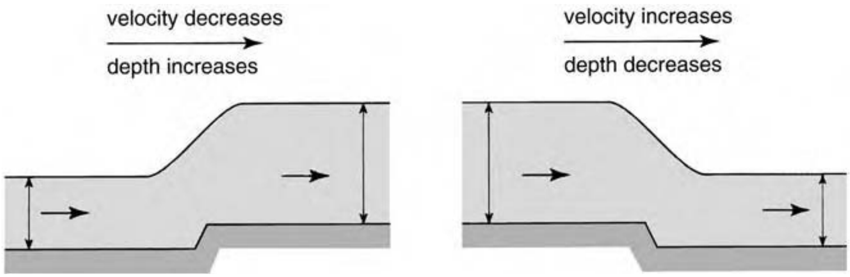


5.13 Rapidly varied flow – sub-critical.

Canoeists are well aware of the way in which rapid changes in water surface levels are a direct result of changes on the river bed. They are wary of those parts of a river where the current looks swifter. It may be tempting to steer your canoe into the faster moving water but it is a sign of shallow water and there may be rocks just below the surface which can damage a canoe. The slower moving water may not be so attractive but at least it will be deep and safe.

5.7.1.2 Super-critical flow

This type of flow behaves in completely the opposite way to sub-critical flow. A step up on the bed of a channel in super-critical flow causes the water depth to rise as it passes over it and a



5.14 Rapidly varied flow – super-critical.

channel which is excavated deeper causes the water depth to drop (Figure 5.14). Super-critical flow is very difficult to deal with in practice. Not only does the faster moving water cause severe erosion in unprotected channels, it is also difficult to control with hydraulic structures. Trying to guide a super-critical flow around a bend in a channel, for example, is like trying to drive a car at high speed around a sharp road bend. It has a tendency to overshoot and to leave the channel. Fortunately super-critical flows rarely occur and are confined to steep rocky streams and just downstream of sluice gates and dam spillways where water can reach speeds of 20 m/s and more. When they do occur engineers have developed ways of quickly turning them back into sub-critical flows so they can be dealt with more easily (see Chapter 6, Section 6.7.6).

5.7.1.3 General rules

So another way of classifying channel flow is in terms of how a flow behaves when the size or shape of a channel is changed suddenly:

For sub-critical flow the water depth decreases when a channel flow area is reduced either by raising its bed or reducing its width. Conversely, the depth increases when the flow area is increased.

For super-critical flow the water depth increases when the flow area is reduced and decreases when it is increased.

5.7.1.4 Spotting the difference

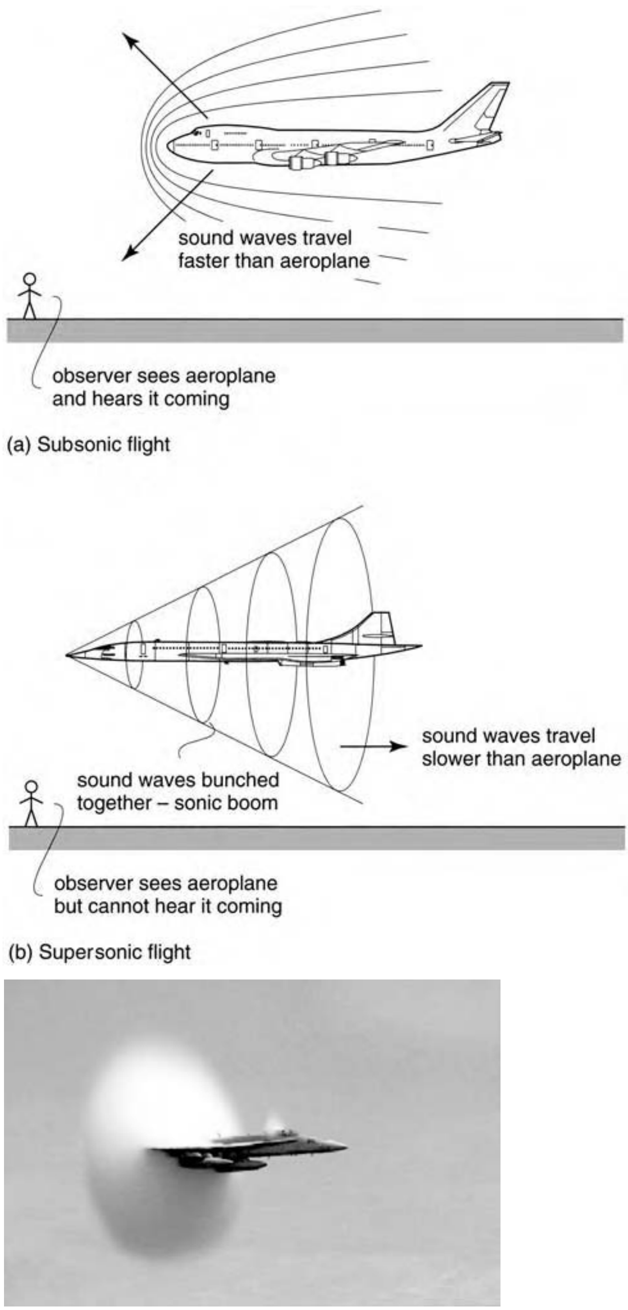
Sometimes the difference between sub- and super-critical flow is obvious. The sluice gate example demonstrates both sub-critical and super-critical flow in the same channel at the same discharge and the same total energy. In this situation there is a clear visual difference between them. But if the two flows occurred in separate straight channels it would be more difficult to tell them apart just by looking. However, if some obstruction is put into the two flows such as a bridge pier or a sharp bend the difference between them would be immediately obvious.

A more scientific way of distinguishing between the two flows is to establish the point of change from sub-critical to super-critical flow – the critical point. There are several very practical ways of doing this but before describing these it might be helpful to look first at another critical point which is similar to, but perhaps more familiar than channel flow.

5.7.1.5 An airflow analogy

Aeroplanes are now an everyday part of our lives. Although you may not be aware of it, and you certainly cannot see it, the airflow around an aeroplane in flight is, in fact very similar to water flow in a channel. They are both fluids and so a look at air flow may help to understand some of the complexities of water flow, and in particular the critical point.

Most people will have noticed that jumbo jets and Concorde have very different shapes (Figure 5.15). This is because the two aeroplanes are designed to travel at very different speeds. Jumbo jets are relatively slow and travel at only 800 km/h whereas Concorde travels at much higher speeds of 2000 km/h and more. But the change in aircraft shape is not a gradual one; a sudden



(c) Aircraft going through the sound barrier from subsonic to supersonic flight

5.15 Using waves to determine flow type.

change is needed when aeroplanes fly over 1200 km/h. This is the speed at which sound waves travel through still air. Sound waves move through air in much the same way as waves travel across a water surface and although they cannot be seen, they can be heard. When someone fires a gun, say 1 km away it takes 3 seconds before you hear the bang. This is the time it takes for sound waves to travel through the air from the gun to your ear at a velocity of 1200 km/h. Notice how you see the gun flash immediately. This is because light waves travel much faster than sound waves at a velocity of 300 000 km/s. This is the reason why lightening in a storm is seen long before the thunder is heard even when the storm is several kilometres away.

When an aeroplane is flying the noise from its engines travels outwards in all directions in the form of sound waves. When it is travelling below the speed of sound, the sound travels faster than the aeroplane and so an observer hears the aeroplane coming before it reaches him (Figure 5.15a). This is known as subsonic flight and aeroplanes which fly below the speed of sound have large rounded shapes like the jumbo jets. When an aeroplane is flying faster than the speed of sound, the sound travels slower than the aeroplane and is left far behind. An observer will see the aeroplane approaching before hearing it (Figure 5.15b). This is known as supersonic flight and aeroplanes travelling at such speeds have slim, dart-like shapes. When the observer does eventually hear it, there is usually a very loud bang. This is the result of a pressure wave, known as the sonic boom, which comes from all the sound waves being bunched up together behind the aeroplane.

So there are two types of airflow, subsonic and supersonic, and there is also a clear point at which the flow changes from one to the other – the speed of sound in still air.

5.7.1.6 Back to water

The purpose of this lengthy explanation about aeroplanes in flight is to demonstrate the close similarity between air flow and water flow. Sub-critical and super-critical flow are very similar to subsonic and supersonic flight. The majority of aeroplanes travel at subsonic speeds and there are very few design problems. In contrast, very few aeroplanes travel at supersonic speeds and their design problems are much greater and more expensive to solve. The same is true for water. Most flows are sub-critical and are easily dealt with. But in a few special cases the flow is super-critical and this is much more difficult to deal with.

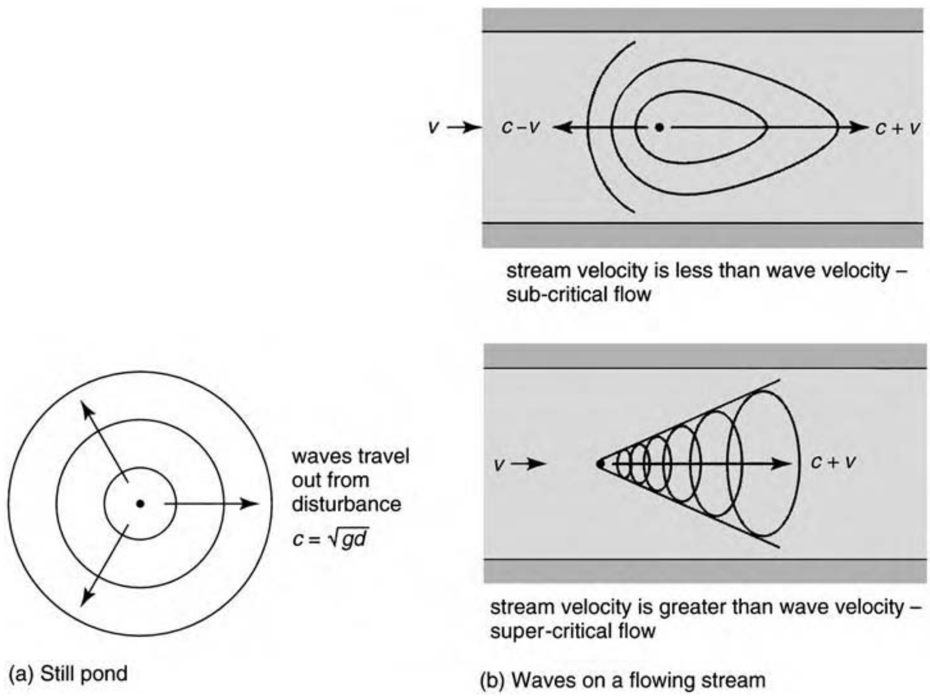
The change from sub- to supersonic occurs when the aeroplane reaches the speed of sound waves in still air. But this is where water is a little different. The change does not occur at the speed of sound in water, although sound does travel through water very effectively. It occurs when the flow reaches the same velocity as waves on the water surface. To avoid confusion between wave velocity and water velocity, waves velocity is often referred to as *wave celerity* (see Section 6.2).

Waves occur on the surface of water when it is disturbed. When you throw a stone into a still pond of water, waves travel out across the surface towards the bank (Figure 5.16a). Although there seems to be a definite movement towards the bank, it is only the waves that are moving outwards and not the water. The water only moves up and down as the waves pass. A duck floating on the pond would only bob up and down with the wave motion and would not be washed up on the bank! But the wave celerity is not a fixed value like the speed of sound. It depends on the depth of water and it can be calculated using the equation (see Section 6.4):

$$c = \sqrt{gd}$$

where c is wave celerity (m/s); g is gravity constant (9.81 m/s^2); d is depth of water (m).

So wave celerity sets the boundary between sub-critical and super-critical flow for a given flow. When the flow velocity is less than the wave velocity the flow is sub-critical. When it is greater, the flow is super-critical.



5.16 Using waves to determine flow type.

As water waves are easily seen they provide a good visual way of determining the type of flow. When you disturb there is a disturbance in a stream, waves move out in all directions from the point of disturbance but the pattern is distorted by the velocity of the water *v* (Figure 5.16b). Some waves move upstream but struggle against the flow and so appear to move more slowly than on the still pond. Others move downstream and are assisted by the flow and so they move faster. Because waves can still move upstream of the disturbance it means that the stream velocity *v* is less than the wave celerity *c* and so the flow must be sub-critical. When the stream velocity *v* is increased and becomes greater than the wave celerity *c*, then waves can no longer travel upstream against the flow. They are all swept downstream and form a vee pattern. This means the flow must be super-critical (Figure 5.16b).

When the stream velocity *v* is equal to the wave celerity *c* then the flow is at the change over point – the *critical point*. At this point the depth of the flow is the critical depth and the velocity is the critical velocity.

Notice the similarity between the sound waves around an aeroplane and the water wave patterns around the disturbance in a stream. There is even an equivalent of the sonic boom in water although it is much less noisy. This is the hydraulic jump which is described in more detail in Section 5.7.6.

5.7.1.7 The finger test

Dropping stones into streams is one way of deciding if the flow is sub-critical or super-critical. Another way is to dip your finger into the water. If the waves you produce travel upstream then the flow is sub-critical. If the waves are swept downstream and the water runs up your arm (you have created a stagnation point in the flow and your wet sleeve is a sign of the high velocity energy) then you are seeing super-critical flow.

5.7.2 *Froude Number*

Another way of determining whether a flow is sub- or super-critical is to use the *Froude Number* (*F*). Returning to the airflow analogy for a moment, aircraft designers use the *Mach Number* or *Mach speed* to describe subsonic and supersonic flight. This is a dimensionless number and is the ratio of two velocities:

$$\text{Mach No.} = \frac{\text{velocity of aircraft}}{\text{velocity of sound in still air}}$$

This dimensionless number was developed by an Austrian Physicist Ernst Mach (1838–1916). A Mach No. less than 1 indicates subsonic flight and a Mach No. greater than 1 indicates supersonic flight. It follows that a Mach No. of 1 means that the aircraft is travelling at the speed of sound.

A dimensionless number similar to the Mach No. was developed by William Froude (1810–1879) to describe sub- and super-critical flow in channels and is now referred to as the Froude Number. It is a ratio of the stream velocity to the wave celerity and is calculated as follows:

$$\text{Froude No. (F)} = \frac{\text{stream velocity (m/s)}}{\text{wave celerity (m/s)}}$$

Note that Froude Number is dimensionless. Now wave celerity:

$$c = \sqrt{gd}$$

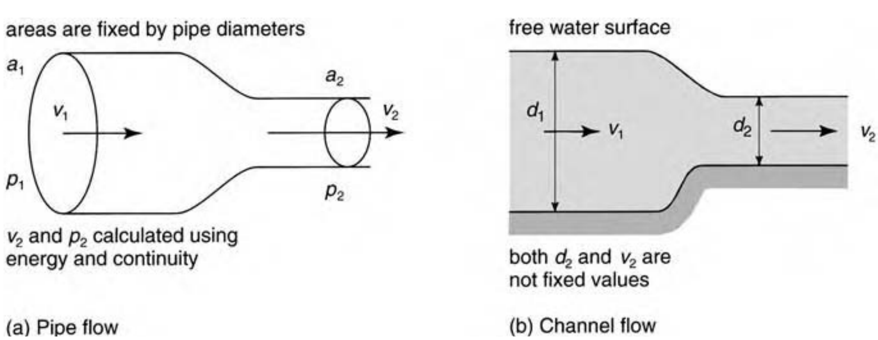
And so:

$$F = \frac{v}{\sqrt{gd}}$$

A Froude Number of less than 1 indicates sub-critical flow and a Froude Number greater than 1 indicates super-critical flow. It follows that a Froude Number of 1 means that the channel is flowing at critical depth and velocity. So calculating the Froude Number is another way of determining when the flow is sub- or super-critical.

5.7.3 *Specific energy*

It should be possible to quantify the changes in depth and velocity resulting from sudden changes in a channel by making use of the energy and continuity equations. Remember that a similar problem occurred in pipes when a venturi meter was inserted to measure discharge (see Section 4.10). The equations of energy and continuity were used to work out the pressure and velocity changes as a result of the sudden changes in the size of the pipe (Figure 5.17a). The solution was



5.17 Predicting changes in depth and velocity in a channel.

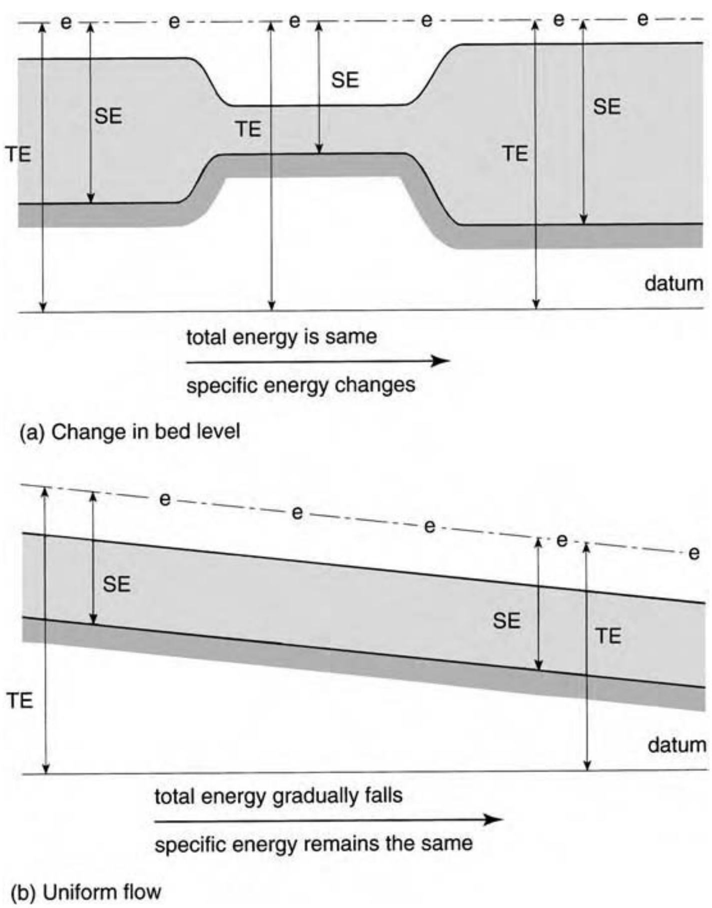
straight forward because the area of flow was fixed by the pipe diameter and so only the velocity needed to be calculated. But for a channel the flow area is not fixed. It is open to the atmosphere and so it can flow at many different depths (Figure 5.17b). So both the flow area and the velocity are unknown. If the energy and continuity equations are applied to this problem the result is a cubic equation which means there are three possible answers for the downstream depth and velocity. One answer is negative and this can be dismissed immediately as impracticable. But the two remaining answers are both possibilities, but which one is the right one?

To help solve this problem Boris Bakhmateff (1880–1951) introduced a very helpful concept which he called *specific energy* (*E*). Simply stated: *Specific energy is the energy in a channel measured from the bed of a channel*.

Writing this as an equation:

$$E = d + \frac{q^2}{2gd^2}$$

It is important at this point to draw a clear distinction between total energy and specific energy. They are linked but they are quite different (Figure 5.18a). Total energy is measured



5.18 Specific energy.

from some fixed datum and its value can only reduce as energy is lost through friction. When there is a change in the bed level of a channel (e.g. when water flows over a weir) there are also changes in the energy components but the total energy remains the same. Specific energy, in contrast, is measured from the bed of a channel and so when the bed level changes the specific energy also changes. It also means that specific energy can rise as well as fall depending on what is happening to the channel bed. When the flow moves from the channel over a weir the specific energy falls and when it comes off the weir it rises again.

The difference between total and specific energy is highlighted by uniform flow (Figure 5.18b). Total energy falls gradually as energy is lost through friction. But specific energy remains constant along the channel because there are no changes in depth and velocity.

The physical significance of specific energy beyond its simple definition is not so obvious and many engineers still struggle with it. However, it is a very easy and very practical concept to use. Rather than be too concerned about what it means, it is better to think of it as a simple mechanism for solving a problem. It is like a key for opening a lock. You do not need to know how the lock works in order to use it. You just put the key in and turn it. In the same way, specific energy unlocks the problem of quantifying the effects that sudden changes in a channel have on depth and velocity. It also helps to establish whether the flow is sub-critical or super-critical and it also unlocks the problem of how to measure discharges in channels (see Section 7.5). So on the whole it is a pretty effective key.

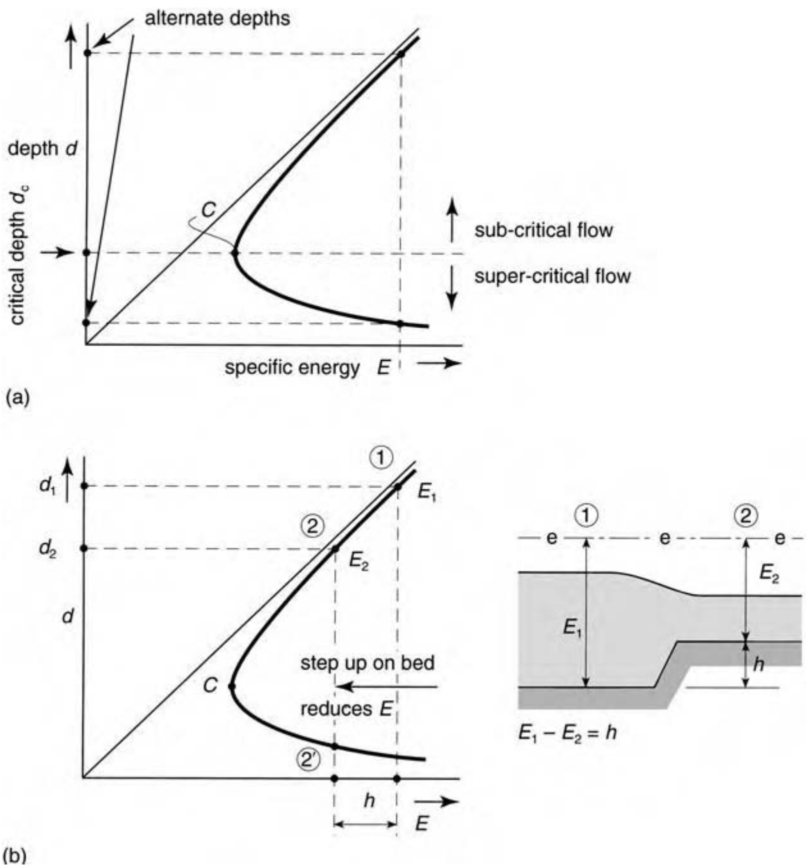
The best way to show how this works is with an example. A channel is carrying a discharge per unit width q. From the specific energy equation it is possible to calculate a range of values for specific energy by putting in different values of d. When the results of the calculations are plotted on a graph (Figure 5.19a) the result is the *specific energy diagram*. There are two limbs to the specific energy diagram and this shows the two possible solutions for depth for any given value of E. These are called the *alternate depths* and are the sub- and super-critical depths described earlier. The upper limb of the curve describes sub-critical flow and the lower limb describes super-critical flow. The change over between the two types of flow occurs at the point C on the graph and this is the *critical point.* It is the only place on the graph where there is only one depth of flow for a given value of E and not two.

So the specific energy diagram defines sub-critical and super-critical flow by defining the critical depth. It also confirms the earlier descriptions of the effects on depth and velocity of sudden changes in a channel. Take any point E_1 on the sub-critical part of the curve and look what happens when the value of E changes as a result of raising or lowering the channel bed. A step up on the bed reduces E and the graph shows that d decreases also (remember E is measured from the bed of the channel). A step down on the bed increases E and the graph shows that d increases. A similar example can be applied to the super-critical part of the diagram to show the effects of changing E on the depth of flow. In this case we see the opposite effect. A step up reduces E and increases d.

So the depth and velocity change, but by how much? This is where the specific energy diagram becomes very useful for quantifying these changes. To see how this is done consider what happens when there is a step up of height h on the bed of a channel from point 1 to point 2 (Figure 5.19b). This reduces the specific energy E_2 by an amount h. So:

$$E_1 - E_2 = h \quad \text{and} \quad E_2 = E_1 - h$$

E_2 on the curve can be found by subtracting h from the value of E_1 and this represents the condition on the step from which it is possible to determine the depth d_2 and velocity v_2. The same logic can also be applied to super-critical flow. An example in the box shows how this is done in practice for both sub-critical and super-critical flows.



5.19 Specific energy diagram.

One question this raises is: why does the flow stay sub-critical when it moves from point 1 to point 2? Why does it not go to point 2′, which has the same value of specific energy, and so become super-critical? The answer is in the specific energy diagram. Remember the diagram is for a given value of discharge. So suddenly 'leaping' across the diagram from 2 to 2′ would mean a change in discharge and this is not possible. The only way to get from 2 to 2′ is down the specific energy curve and through the critical point. This can only be done by increasing the value of *h* (i.e. raising the step even further) so that E_2 gets smaller until it reaches the critical point. At this point the flow can go super-critical. As *h* in this case is not high enough to create critical conditions, the flow moves from point 1 to point 2 and stays sub-critical.

The same argument can be applied to changes in super-critical flow also. The flow can only go sub-critical by going around the specific energy curve and through the critical depth.

To summarise some points about specific energy:

- Specific energy is used to quantify the changes in depth and velocity in a channel as a result of sudden changes in the size and shape of a channel. It also helps to determine which of the two possible answers for depth and velocity is the right one.
- Specific energy is different from total energy. It can increase as well as decrease. It depends on what happens to the bed of the channel. In contrast, total energy can only decrease as energy is lost through friction or sudden changes in flow.

- The specific energy diagram in Figure 5.19 is for one value of discharge. When the discharge changes a new diagram is needed. So for any channel there will be a whole family of specific energy diagrams representing a range of discharges.
- Specific energy is the principle on which many channel flow measuring devices such as weirs and flumes are based. They depend on changing the specific energy enough to make the flow go critical. At this point there is only one value of depth for one value of specific energy and from this is possible to develop a formula for discharge (see Section 7.5). Such devices are sometimes referred to as *critical depth structures*.
- For uniform flow the value of specific energy remains constant along the entire channel. This is because the depth and velocity are the same. In contrast, total energy gradually reduces as energy is lost along the channel.

5.7.3.1 Is the flow sub-critical or super-critical?

The answer comes from calculating the normal depth of flow, using a formula such as Manning's, and then comparing it with the critical depth. If the depth is greater than the critical depth the flow will be sub-critical and if it is less it will be super-critical.

EXAMPLE: CALCULATING THE EFFECT ON DEPTH OF CHANGING THE CHANNEL BED LEVEL

A 0.3 m step is to be built in a rectangular channel carrying a discharge per unit width of 0.5 m^3/s (Figure 5.20). Calculate the effect of this change on the water level in both sub-critical and super-critical flow conditions assuming the specific energy in the upstream channel $E_1 = 1.0$ m. Calculate the effect in the same channel of lowering a section of the channel bed by 0.3 m.

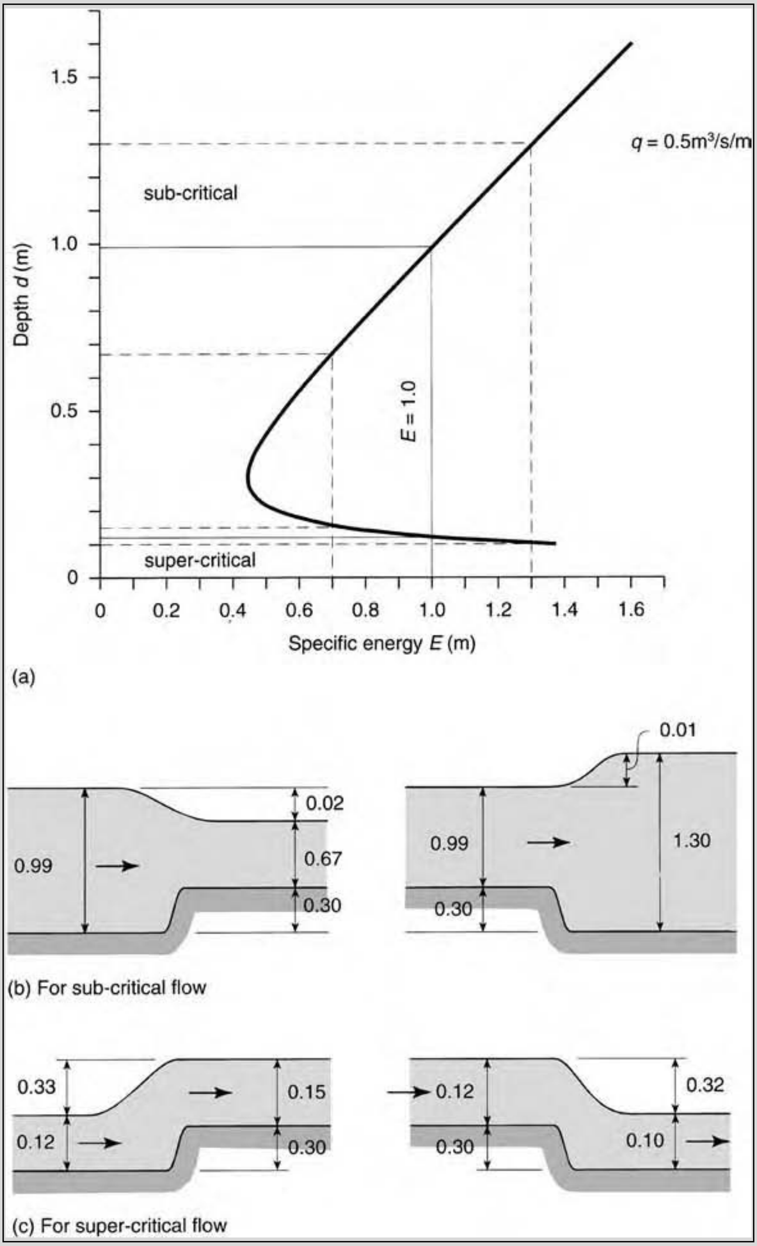
The first step is to calculate specific energy E for a range of depths for a discharge of 0.5 m^3/s:

Depth of flow (m)	*Specific energy (m)*
Sub-critical flow curve	
0.4	0.480
0.5	0.551
0.6	0.635
0.7	0.726
0.8	0.820
0.9	0.916
1	1.013
1.2	1.209
1.4	1.407
1.6	1.605
Super-critical flow curve	
0.1	1.374
0.12	1.005
0.15	0.716
0.2	0.519
0.3	0.442
0.25	0.454
0.35	0.454

Plot the results on a graph (See Figure 5.20a).

From the graph the alternate depths for E_1 = 1 m are d_1 = 0.99 m (sub-critical) and 0.12 m (super-critical). Consider the sub-critical flow case first.

For sub-critical flow – calculate the effect on water level of a 0.3 m high step up on the bed of the channel:



5.20 Calculating specific energy.

Specific energy in the channel is:

$$E_1 = 1.0 \text{ m}$$

And:

$$d_1 = 0.99 \text{ m}$$

Next calculate the specific energy on the step which is 0.3 m higher than the channel bed:

$$E_2 = E_1 - 0.3 = 0.7 \text{ m}$$

Now locate E_2 on the graph and read off the value for d_2:

$$d_2 = 0.67 \text{ m}$$

Depth d_2 is measured from the step and so the water level is 0.97 m above the original channel bed. This means that the water level has dropped 0.02 m as a result of raising the channel bed.

For sub-critical flow – calculate the effect on water level of a 0.3 m step down on the bed of the channel:

Specific energy in the channel is:

$$E_1 = 1.0 \text{ m}$$

And:

$$d_1 = 0.99 \text{ m}$$

Next calculate the specific energy where the channel has been excavated by 0.3 m:

$$E_2 = E_1 + 0.3 = 1.3 \text{ m}$$

Now locate E_2 on the graph and read off the value for d_2:

$$d_2 = 1.30 \text{ m}$$

Depth d_2 is measured from the bed of the excavated section and so the water level is now 1.30 m above the channel bed. This means that the water level rises by 0.01 m as a result of the step down in the channel bed.

For super-critical flow – calculate the effect on water level of a 0.3 m step up on the bed of the channel:

Specific energy in the channel is:

$$E_1 = 1.0 \text{ m}$$

And:

$$d_1 = 0.12 \text{ m}$$

Next calculate the specific energy on the step which is 0.3 m higher than the channel bed:

$$E_2 = E_1 - 0.3 = 0.7 \text{ m}$$

Now locate E_2 on the graph and read off the value for d_2:

$$d_2 = 0.15 \text{ m}$$

Now depth d_2 is measured from the top of the raised bed and so the water level is now 0.45 m above the original channel bed. This means that the water level rises 0.33 m as a result of raising the channel bed.

For super-critical flow – calculate the effect on water level of a 0.3 m step down on the bed of the channel:

Specific energy in the channel is:

$$E_1 = 1.0 \text{ m}$$

And:

$$d_1 = 0.12 \text{ m}$$

Next calculate the specific energy where the channel has been excavated by 0.3 m:

$$E_2 = E_1 + 0.3 = 1.3 \text{ m}$$

Now locate E_2 on the graph and read off the value for d_2:

$$d_2 = 0.10 \text{ m}$$

Now depth d_2 is measured from the bed of the excavated bed and so the water level is now 0.20 m below the original channel bed. This means that the water level drops 0.32 m as a result of the step down in the channel bed.

5.7.4 Critical depth

The critical depth d_c can be determined directly from the specific energy diagram but it can be difficult to locate its exact position because of the rounded shape of the curve close to the critical point. To overcome this problem it can be calculated using a formula derived from the specific energy equation:

$$d_c = \sqrt[3]{\frac{q^2}{g}}$$

This formula shows that the critical depth is influenced only by the discharge per unit width q. It has nothing to do with the slope or the normal depth. For the mathematically minded a proof of this is given in the box.

DERIVATION: CRITICAL DEPTH EQUATION

Derive a formula for the critical depth and its location on the specific energy diagram (Figure 5.20).

Use the specific energy equation:

$$E = d + \frac{q^2}{2gd^2}$$

At the critical point the specific energy E is at its lowest value and so the gradient of the curve $\mathrm{d}E/\mathrm{d}d$ is equal to zero. The equation for the gradient can be found using calculus and differentiating the above equation for the curve. If you are not familiar with calculus you will have to accept this step as given:

$$\frac{dE}{dd} = 1 - \frac{q^2}{gd^3} = 0$$

Depth d now becomes the critical depth d_c and so:

$$\frac{q^2}{gd_c^3} = 1$$

Rearrange this for d_c:

$$d_c^3 = \frac{q^2}{g}$$

$$d_c = \sqrt[3]{\frac{q^2}{g}}$$

So the critical depth d_c depends only on the discharge per unit width q.
To calculate the specific energy at the critical point first write down the specific energy equation for critical conditions:

$$E_c = d_c + \frac{q^2}{2gd_c^2}$$

But at the critical depth (see equation earlier in this proof):

$$\frac{q^2}{gd_c^3} = 1$$

Divide both sides by 2 and multiplying by d_c:

$$\frac{q^2}{2gd_c^2} = \frac{d_c}{2}$$

The left-hand side is now equivalent to the kinetic energy term so put this into the specific energy equation:

$$E_c = d_c + \frac{d_c}{2}$$

$$=\frac{3d_c}{2}$$

So for any given discharge the critical depth can be calculated as well as the specific energy at the critical point. These two values exactly locate the critical point on the specific energy diagram.
Note also from the above analysis that when:

$$\frac{q^2}{gd_c^3} = 1$$

$$v_c^2 = gd_c$$

And so:

$$v_c = \sqrt{gd_c}$$

This is the equation for the celerity of surface water waves and it shows that at the critical point the water velocity v_c equals the wave celerity $\sqrt{gd_c}$. Remember that the velocity of waves across water following a disturbance is often used to determine whether flow is sub-critical or super-critical.

EXAMPLE: CALCULATING CRITICAL DEPTH

Using information in the previous example calculate the critical depth and the step up in the bed level required to ensure that the flow will reach the critical depth. Assume the initial specific energy $E_1 = 1.0$ m.

First calculate the critical depth:

$$d_c = \sqrt[3]{\frac{q^2}{g}}$$

$$d_c = \sqrt[3]{\frac{0.5^2}{9.81}} = 0.29 \text{ m}$$

Now calculate the specific energy on the step up:

$$E_2 = d_2 + \frac{q^2}{2gd_2^2}$$

When this is critical flow:

$$d_2 = d_c$$

And so:

$$E_2 = 0.29 + \frac{0.5^2}{2 \times 9.81 \times 0.29^2}$$
$$= 0.44 \text{ m}$$

But:

$$E_1 - E_2 = h$$

that is, the change in specific energy is a direct result of the height of the step up h:
And so:

$$h = 1.0 - 0.44 = 0.56 \text{ m}$$

The bed level of the channel must be raised by 0.56 m to ensure that the flow goes critical.

5.7.5 Critical flow

Although critical flow is important, it is a flow condition best avoided in uniform flow. There is no problem when flow goes quickly through the critical point on its way from sub- to super-critical or from super to sub but there are problems when the normal flow depth is near to the critical depth. This is a very unstable condition as the flow tends to oscillate from sub to super and back to sub-critical again resulting in surface waves which can travel for many kilometres eroding and damaging channel banks. The explanation for the instability is in the shape of the specific energy diagram close to the critical point (Figure 5.20a). Small changes in specific energy E, possibly as a result of small channel irregularities, can cause large changes in depth as the flow oscillates between sub- and super-critical flow. As the flow *hunts* back and forth it sets up waves. So although critical flow is very useful in some respects it can cause serious problems in others.

5.7.6 Flow transitions

Changes in a channel which result in changes in flow from sub-critical to super-critical and vice-versa are referred to as *transitions*. The following are examples of some common transitions.

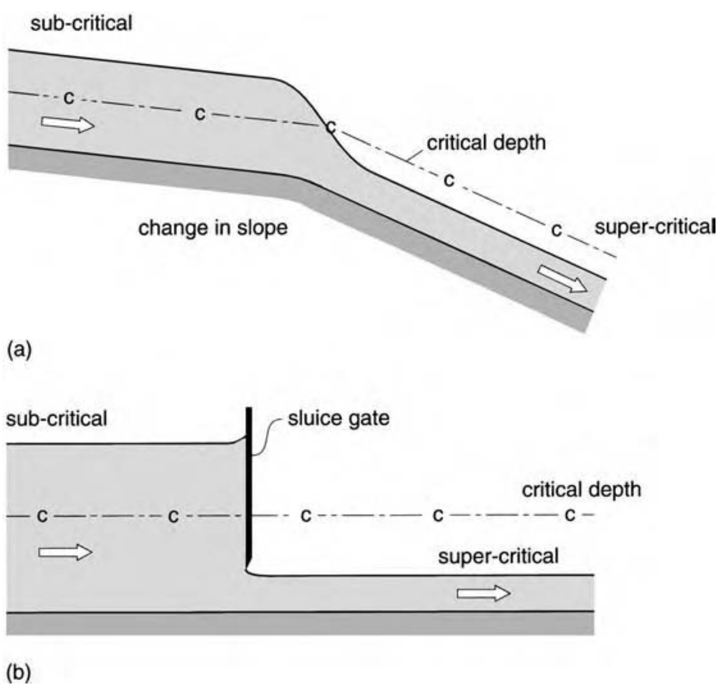
5.7.6.1 Sub- to super-critical flow

When flow goes from sub- to super-critical it does so smoothly. In Figure 5.21a the channel gradient is increased which changes the flow from sub- to super-critical. The water surface curves rapidly but smoothly as the flow goes through the critical point. There is no energy loss as the flow is contracting. Notice how the critical depth is shown as c—c—c so that the two types of flow are clearly distinguishable. Remember, the critical depth is the same in both sections of the channel because it depends only on the discharge and not on slope.

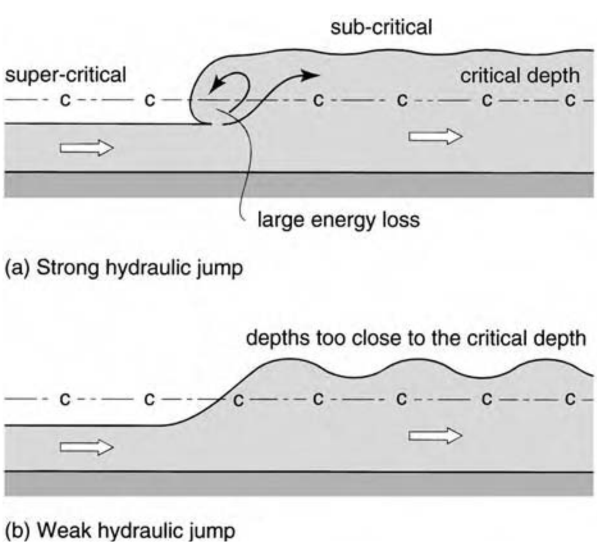
Another example of this type of transition is the sluice gate (Figure 5.21b). In this case a gate is used to force the change in flow. Again the transition occurs smoothly with no energy loss.

5.7.6.2 Super- to sub-critical flow (hydraulic jump)

The change from super- to sub-critical is not so smooth. In fact a vigorous turbulent mixing action occurs as the flow jumps abruptly from super- to sub-critical flow (Figure 5.22). It is aptly called a *hydraulic jump* and as the flow expands there is a significant loss of energy due to the turbulence.



5.21 Flow transitions – sub- to super-critical.



5.22 Flow transitions – super- to sub-critical.

Hydraulic jumps are very useful for many purposes:

- Getting rid of unwanted energy, such as at the base of dam spillways.
- Mixing chemicals in water. The vigorous turbulence ensures that any added chemical is thoroughly dispersed throughout the flow.

- Converting super-critical flow downstream of hydraulic structures into sub-critical flow to avoid erosion damage in unprotected channels.

Hydraulic jumps are usually described by their strength and the Froude Number of the super-critical flow. A *strong jump* is the most desirable. It is very vigorous, has a high Froude Number, well above one and the turbulent mixing is confined to a short length of channel. A *weak jump*, on the other hand, is not so violent. It has a low Froude No. approaching one, which means the depth of flow is close to the critical depth. This kind of jump is not confined and appears as waves which can travel downstream for many kilometres. This is undesirable because the waves can do a great deal of damage to channel banks.

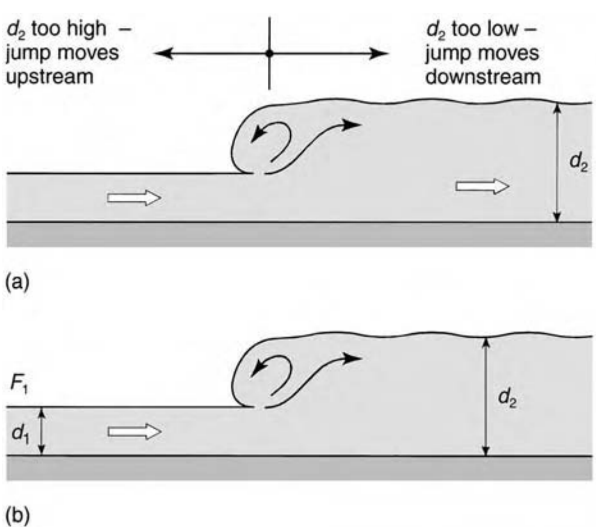
5.7.6.3 Creating a hydraulic jump

There are two conditions required to form a hydraulic jump:

- Flow upstream of the jump must be super-critical, that is, the Froude Number must be greater than 1.
- Downstream flow must be sub-critical and deep enough for the jump to form.

Once there is super-critical flow in a channel it is the downstream depth of flow that determines if a jump will occur. To create a jump the downstream depth must be just right. If the depth is too shallow a jump will not form and the super-critical will continue down the channel (Figure 5.23). Conversely, if the flow is too deep the jump will move upstream and if it reaches a sluice gate it may drown it out. This can be a problem as the high speed super-critical flow is not dispersed as it would be in a full jump and it can cause erosion downstream (see Section 7.32.1 for drowned flow from a sluice gate).

When the upstream depth and velocity are known it is possible to calculate the downstream depth and velocity which will create a jump by using the momentum equation. The energy equation cannot be used at this stage because of the large and unknown energy loss at a jump.



5.23 Forming a hydraulic jump.

The formula is derived from the momentum equation which links the two depths of flow d_1 and d_2:

$$\frac{d_2}{d_1} = \frac{1}{2}\left(\sqrt{1+8F_1^2} - 1\right)$$

Where d_1 is upstream depth; d_2 is downstream depth; F_1 is upstream Froude No.

Most hydraulic jump problems involve calculating the downstream depth d_2 in order to determine the conditions under which a jump will occur. But determining a value for d_2 is fine in theory, but in practice the formation of a jump is very sensitive to the downstream depth. This means that when the downstream depth is a little more or less than the calculated value, the jump does not stabilise at the selected location but moves up and down the channel *hunting* for the right depth of flow. In practice it is very difficult to control water depths with great accuracy and so some method is needed to remove the sensitivity of the jump to downstream water level and so stabilise it. This is the job of a *stilling basin* which is a concrete apron located downstream of a weir or sluice gate. Its primary job is to dissipate unwanted energy to protect the downstream channel and it does this by making sure the hydraulic jump forms on the concrete slab even though the downstream water level may vary considerably. This is discussed further in Section 7.10.

5.7.6.4 Calculating energy losses

Once the downstream depth and velocity have been calculated using the momentum equation, the loss of energy at a jump can be determined using the total energy equation as follows:

Total energy upstream = total energy downstream + energy loss at jump

That is:

$$d_1 + \frac{v_1^2}{2g} + z_1 = d_2 + \frac{v_2^2}{2g} + z_2 + \text{losses}$$

EXAMPLE: CALCULATING THE DOWNSTREAM DEPTH TO FORM A HYDRAULIC JUMP

Calculate the depth required downstream to create the jump in a channel carrying a discharge of 0.8 m^3/s per m width at a depth of 0.25 m.

The downstream depth can be calculated using the formula:

$$\frac{d_2}{d_1} = \frac{1}{2}\left(\sqrt{1+8F_1^2} - 1\right)$$

First calculate the velocity using the discharge equation:

$$q = v_1 d_1$$

$$v_1 = \frac{0.8}{0.25} = 3.2 \text{ m/s}$$

Next calculate the Froude No:

$$F_i = \frac{v_1}{\sqrt{gd_1}}$$

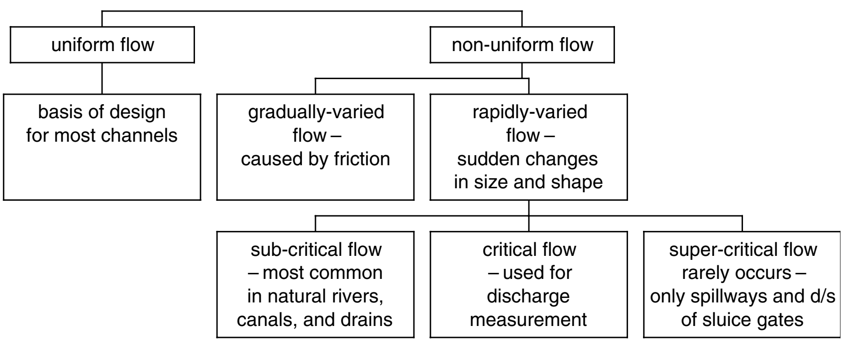
$$= \frac{3.2}{\sqrt{9.81 \times 0.25}} = 2.04$$

Substitute the values into the above formula:

$$\frac{d_2}{0.25} = \frac{1}{2}\left(\sqrt{1+(8\times 2.04^2)}-1\right)$$

$$d_2 = 0.25 \times 2.43 = 0.61 \text{ m}$$

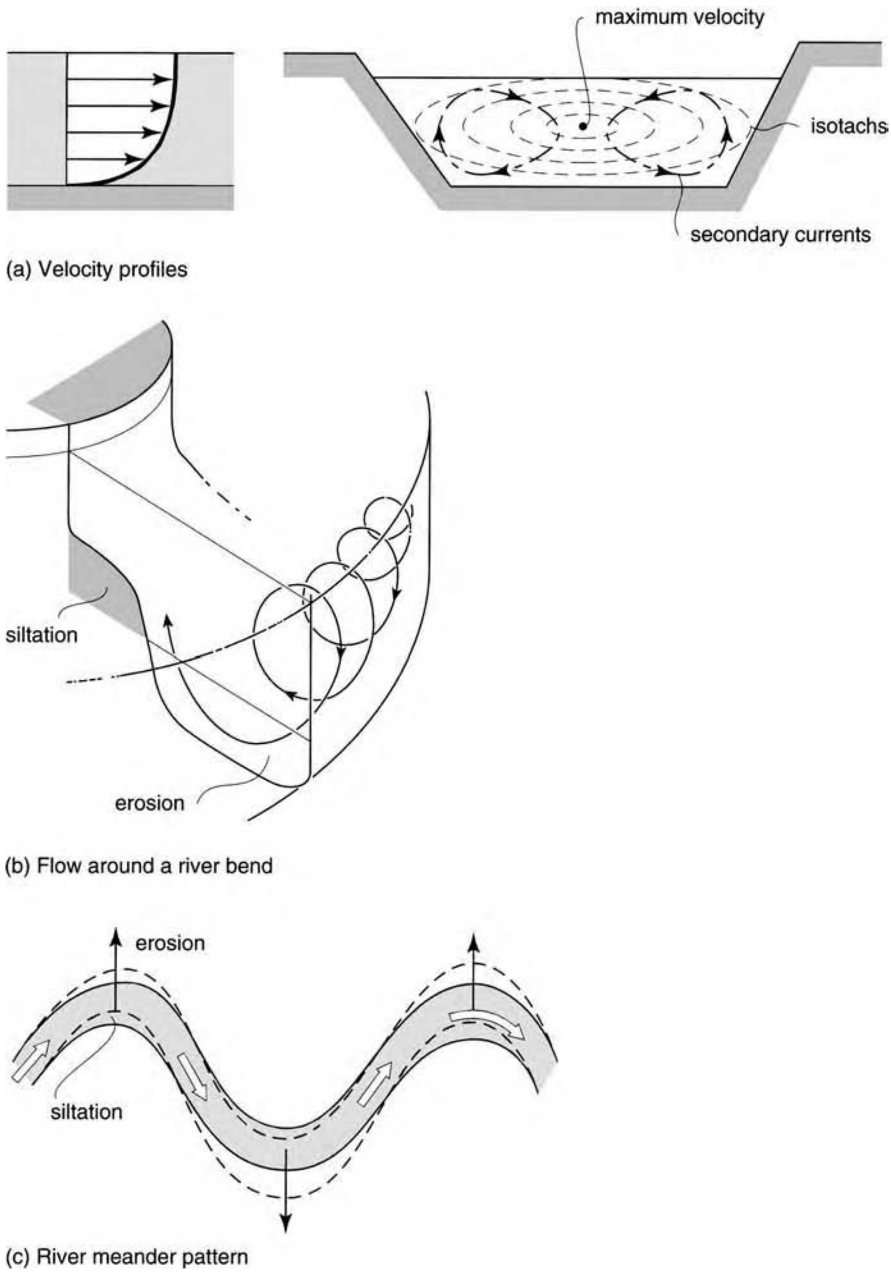
To summarise all the various types of flow that can occur in channels:



5.8 Secondary flows

One interesting aspect of channel flow (which all anglers know about) is *secondary flows*. These are small but important currents that occur in flowing water and explain many important phenomena.

In ideal channels the velocity is assumed to be the same across the entire channel. But in real channels the velocity is usually much higher in the middle than at the sides and bed. Figure 5.24 shows the velocity profile in a typical channel and in cross section, isotachs (lines of equal velocity) have been drawn. The changes in velocity across the channel cause small changes in pressure (remember the energy equation) and these are responsible for setting up cross currents which flow from the sides of the channel to the centre. As the water does not pile up in the middle of a channel there must be an equivalent flow from the centre to the edge. These circulating cross flows are called *secondary flows.* In very wide channels several currents can be set up in this way. So water does not just flow straight down a channel, it flows along a spiral path.



5.24 Secondary flows in channels.

5.8.1 Channel bends

An important secondary flow occurs at channel bends (Figure 5.24). This is created by a combination of the difference in velocity between the surface and the bed and the centrifugal forces as the water moves around the bend. A secondary current is set up which moves across the bed from the outside of the bend to the inside and across the surface from the inside to the outside. The secondary flow can erode loose material on the outside of a bend and carry it across a river bed and deposit it on the inside of the bend. This is contrary to the common belief that sediment on the bed of the river is thrown to the outside of a bend by the strong centrifugal forces.

One consequence of this in natural erodible river channels is the process known as *meandering.* Very few natural rivers are straight. They tend to form a snake-like pattern of curves which are called *meanders*. The outside of bends are progressively scoured and the inside silts up causing the river cross section to change shape. The continual erosion gradually alters the course of the river. Sometimes the meanders become so acute that parts of the river are eventually cut off and form what are called *ox-bow lakes.*

Note that it is not a good idea to go swimming on the outside bend of a river. The downward current can be very strong and pull the unwary swimmer down into the mud on the river bed!

5.8.2 Siting river offtakes

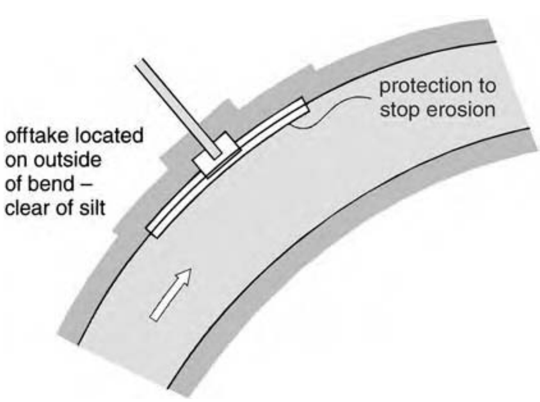
A common engineering problem is to select a site for abstracting water from a river for domestic use or irrigation. This may be a pump or some gated structure. The best location is on the outside of a river bend so that it will be free from (Figure 5.25). If located on the inside of a bend it would be continually silting up as a result of the actions of the secondary flows. The outside of a bend may need protecting with stone pitching to stop any further erosion which might destroy the offtake.

5.8.3 Bridge piers

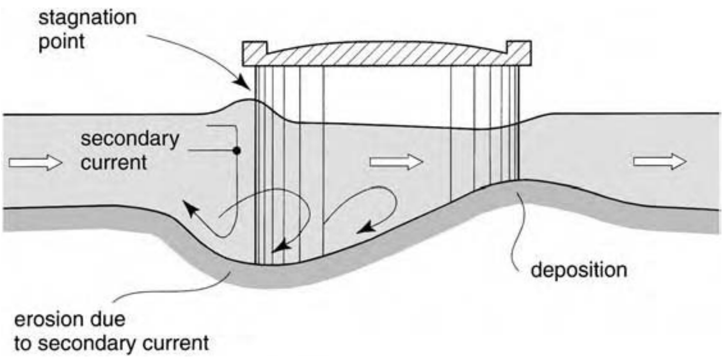
Scouring around bridge piers can be a serious problem. This is the result of a secondary flow set up by the stagnation pressure on the nose of a pier (Figure 5.26). The rise in water level as the flow is stopped causes a secondary downward current towards the bed. This is pushed around the pier by the main flow into a spiral current which can cause severe scouring both in front and around the sides of piers. Heavy stone protection can reduce the problem but a study of the secondary flows has shown that the construction of low walls upstream can also help by upsetting the pattern of the destructive secondary currents.

5.8.4 Vortices at sluice gates

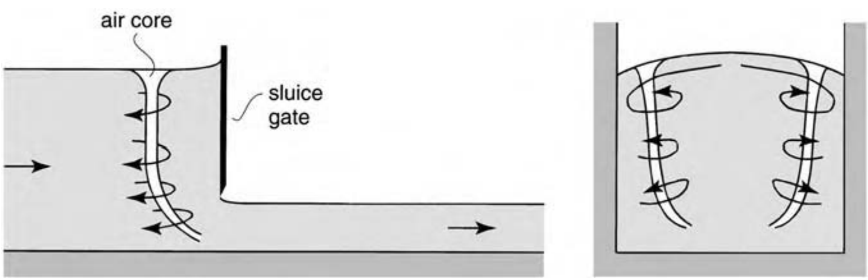
Vortices can develop upstream of sluice gates and may be so strong that they extend from the water surface right underneath the gate drawing air down into its core (Figure 5.27). They can



5.25 Siting river offtakes.



5.26 Flow around bridge piers.



5.27 Vortices at sluice gates.

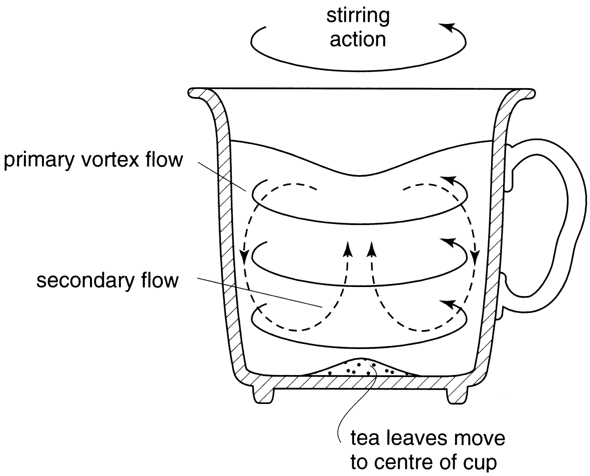
become so severe that they reduce the flow through the gate. They look like small tornadoes and are the result of secondary currents. They form when the surface flow reaches a sluice gate the water stops, that is, it stagnates. But the flow in the middle of the stream was moving faster than near the sides and so the rise in water level due to the stagnation is higher in the middle than at the sides. This causes a secondary flow from the middle of the stream to the sides and sets up two circulating vortices.

5.8.5 Tea cups

An interesting secondary flow occurs when stirring a cup of tea (Figure 5.28). First make sure there is no milk in the tea to obscure the view, then notice how the tea leaves at the bottom of the cup move in towards the centre when the tea is stirred. The stirring action sets up a vortex which is similar to the flow round a bend in a river. The water surface drops in the centre of the flow causing a pressure difference which results in a downward current on the outside of the cup which moves across the bottom and up the middle. Any tea leaves lying on the bottom of the cup are swept along a spiral path towards the centre of the cup.

5.9 Sediment transport

Sediment movement in channels is usually referred to as *sediment transport* and it is not a subject normally covered in texts on basic hydraulics. But a study of channels would not be complete without mentioning it. Most channels are either naturally occurring or excavated in



5.28 Stirring the tea.

the natural soil and so are prone to scouring and silting. Very few have the luxury of a lining to protect them against erosion. Natural rivers often carry silt and sand washed from their catchments, some even carry large boulders in their upper reaches. Man-made canals not only have to resist erosion but need to avoid becoming blocked with silt and sand. Indeed the sediment is often described as the sediment load, indicating the burden that channels must carry.

Channels carrying sediment are designed in a different way to those carrying clear water. Normally Manning or Chezy formulae are used for clear water but they are not well suited to sediment laden water. There is no simple accepted theory which provides a thorough understanding of sediment movement on which to base channel design. When engineers meet a problem for which there is no acceptable theory they do not wait for the scientists to find one. They try to find a way round the problem and develop alternative design methods. This is what happened in India and Pakistan at the turn of the century when British engineers were faced with designing and operating large canals which carried both water and silt from the Indus and Ganges rivers for the irrigation of vast tracts of land. They observed and measured the hydraulic parameters of existing canals that seemed to have worked well under similar conditions over a long period of time. From these data they developed equations which linked together sediment, velocity, hydraulic radius, slope and width and used them to design new canals. They came to be known as the regime equations. The word regime described the conditions in the canals where, over a period of time, the canals neither silted nor scoured and all the silt that entered the canals at the head of the systems was transported through to the fields. In fact an added benefit of this was the silt brought with it natural fertilisers for the farmers. But this is not claimed to be a perfect solution. Often the canals silted up during heavy floods and often they would scour when canal velocities were excessive. But on balance over a period of time (which may be several years) the canals did not change much and were said to be *in regime*. In spite of all the research over the past century, engineers still use these equations because they are still the best available today.

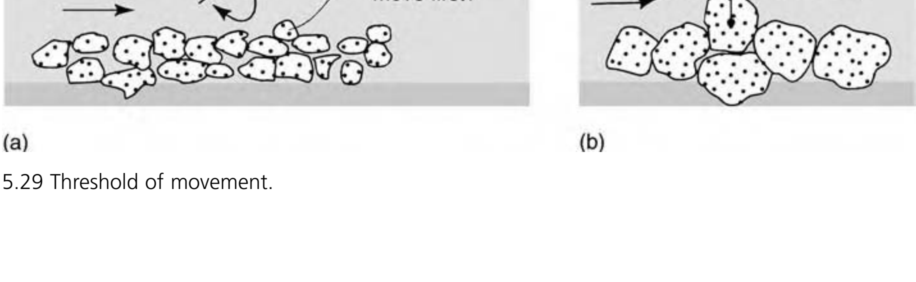
The lack of progress in our understanding of sediment transport comes from the nature of the problem. In other aspects of hydraulics there is a reasonably clear path to solving a problem. In channels one single force dominates channel design – the gravity force. All other factors such as viscosity can be safely ignored without it causing much error. But in sediment transport there are several factors which control what happens and no single one dominates. Gravity influences both water and sediment, but viscosity and density also appear to be important. This makes any fundamental analysis of the problem complex.

At the heart of the sediment transport problem is how to determine when sediment begins to move – the *threshold of movement*. Some investigators, particularly engineers, have tried to link this with average stream velocity because it is simple to measure and it is obviously linked to the erosive power of a channel. But research has shown that it is not as simple as this. Look at the particles of sand on the bed of a channel (Figure 5.29). Is it the average velocity that is important or the local velocity and turbulence close to the sand grains? Some grains are more exposed than others. Do they move first? Does grain size and density matter? Do the rounded grains move more easily than the angular ones? Is there a Bernoulli (energy change) lifting effect when flow moves around exposed grains which will encourage them to move? How important is the apparent loss in weight of sand grains when they are submerged in water? There are no clear answers to these questions but what is clear is that many factors influence the threshold of movement and this has made it a very difficult subject to study from an analytical point of view.

There is, however, some good news which has come from experimental work. It is in fact very easy to observe the threshold of movement and to say when it has been reached. Imagine a channel with a sandy bed with water flowing over it. When the flow is increased the sand will at some point begin to move. But the interesting point is that this begins quite suddenly and not gradually as might be expected. When the threshold is reached, the whole channel bed comes alive suddenly as all the sand begins to move at the same time. So if several observers, watching the channel, are asked to say when they think the threshold has been reached, they will have no problem in agreeing the point at which it occurs. What they will not be able to say for certain is what caused it.

Because of this clear observation of the threshold much of the progress had been made by scientists using experimental methods. Shields in 1936 successfully establish the conditions for the threshold of movement on an experimental basis for a wide range of sediments and these are the data that are still used today for designing channels to avoid erosion. It provides a much sounder basis for design than simply using some limiting velocity.

Working out the amount of sediment being transported once it begins to move is fraught with difficulties. The reason for this is that once movement begins the amount of sediment on the move is very sensitive to small changes in the factors which caused the movement in the first place. This means that small changes in what could be called the erosive power of a channel can result in very large changes in sediment transport. Even if it was possible to calculate such changes, which some experimenters have tried to do, it is even more difficult to verify this by measuring sediment transport in the laboratory and almost impossible to measure it with any



5.29 Threshold of movement.

accuracy in the field. For these reasons it seems unlikely that there will be any significant improvements in the predictions of sediment transport and that engineers will have to rely on the regime equations for some time to come.

5.10 Some examples to test your understanding

1 An open channel of rectangular section has a bed width of 1.0 m. If the channel carries a discharge 1.0 m^3/s calculate the depth of flow when the Manning's roughness coefficient is 0.015 and the bed slope is 1 in 1000. Calculate the Froude Number in the channel and the critical depth (0.5 m; 0.45 m; 0.29 m).

2 A rectangular channel of bed width 2.5 m carries a discharge of 1.75 m^3/s. Calculate the normal depth of flow when the Chezy coefficient is 60 and the slope is 1 in 2000. Calculate the critical depth and say whether the flow is sub-critical or super-critical (0.75 m; 0.37 m; flow is sub-critical).

3 A trapezoidal channel is to be designed and constructed in a sandy loam with a longitudinal slope of 1 in 5000 to carry a discharge of 2.3 m^3/s. Calculate suitable dimensions for the depth and bed width assuming Manning's n is 0.022 and the side slope is 1 in 2 (d = 0.98 m; b = 2.94 m (b = 3d)).

4 A trapezoidal channel carrying a discharge of 0.75 m^3/s is to be lined with concrete to avoid seepage problems. Calculate the channels dimensions which will minimise the amount of concrete when Manning's n for concrete is 0.015 and the channel slope is 1 in 1250 (for a hexagonal channel d = 0.37 m; b = 0.8 m. Note that other answers are possible depending on choice of side slope).

5 A hydraulic jump occurs in a rectangular channel 2.3 m wide when the discharge is 1.5 m^3/s. If the upstream depth is 0.25 m calculate the upstream Froude Number, the depth of flow downstream of the jump and the energy loss in the jump (2.78 m; 0.87 m; 0.3 m).

6 Waves

6.1 Introduction

Waves are very familiar to everyone, particularly those that toss about at sea during windy, stormy weather and those which roll in rhythmically on the beach. The tides too are waves but they are not so obvious as they slowly build up and subside twice a day and move many millions of tonnes of water across the oceans. But waves are not just confined to the sea. Earlier, in Chapter 5, waves were described on a still pond and on a flowing stream when a stone was thrown into the water to help determine critical flow conditions. Surge waves can sometimes be seen in rivers when control gates are suddenly closed and during high tides when tidal bores, some of which are quite famous, travel upstream from the mouth of rivers. Floods flowing down rivers too are another kind of wave which can be several hundreds of kilometres long.

Waves in rivers depend on gravity for their shape and size in much the same way that channels themselves depend on the force of gravity down the land slope for their energy. Sea waves are different as they depend on the circulation of the atmosphere and the resulting winds for their energy. Although it is possible for winds to create waves on lakes, they are not usually as large as those which can be generated by strong winds blowing over vast stretches of the oceans. These are the waves which play a large part in shaping our shorelines.

Sea waves tend to affect only the water surface. Indeed wave movement is different from the movement of the water in which it is travelling. Watching a single wave, it seems to travel a great distance at a steady speed. But observing the water closely, it hardly moves at all. Just look at any object floating – a seagull – on the water surface to appreciate this. When a wave passes the bird is not swept along with the wave it just bobs up and down. Leonardo da Vinci (1452–1519) compared water waves with the waves you see when the wind blows across a field of corn. The wave pattern seems to travel across the entire field and yet individual stalks only sway back and forth. Another example is that of a rope which is held at one end and shaken. A wave pattern travels along the rope but the rope only moves up and down. So if the water (or the corn or rope) is not being transported then what is? The answer is energy. It is energy which is being transported across the water and through the corn and along the rope.

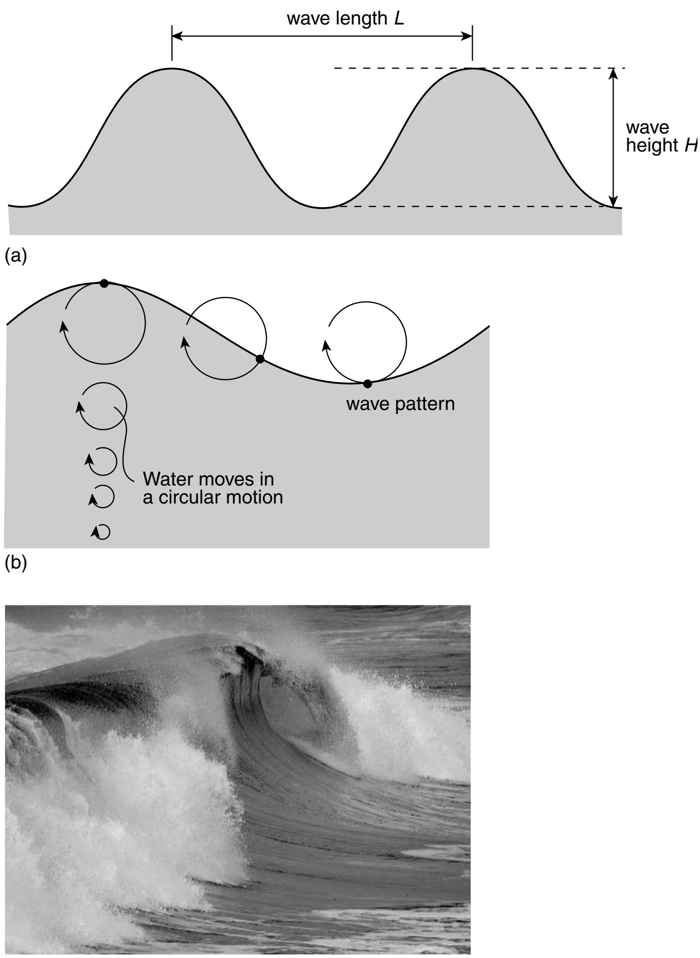
Our understanding of waves and how they are generated is far from complete. This is because is it difficult to observe waves at sea and also because all the mathematical formulae are based on ideal fluids and the sea does not always fit in with the ideal.

6.2 Describing waves

Wave *height H* is the dimension from the crest of a wave (peak) and the wave trough (Figure 6.1a). The height of a wave is twice its *amplitude* and the wave *length L* is the distance between two successive wave peaks.

As well as having dimensions in space, waves also have dimensions in time. So when waves move past an observer the time interval between successive peaks is called the wave *period* (*T*) and the number of peaks which pass an observer each second is the *frequency* (*f*).

The velocity of waves is called the *celerity* (*c*) and this word is used to clearly distinguish the wave velocity from that of the water. If the water is also moving then the wave velocity increases or *f* decreases depending on whether the wave is moving with against the flow. Although water waves have been likened to wave motion along a rope, water is a little different. While a rope is free to rise and fall leaving gaps around it, this is not possible with water and so as the water rises when a wave passes, water nearby flows into the space. This gives rise to a circular motion which extends some distance below the water surface but with diminishing effect (Figure 6.1b). At a depth of half the wave length the effects of surface waves are negligible. A submarine submerged at 150 m would be able to avoid all the severe waves that are produced by a storm at sea.



6.1 Wave dimensions and movement.

6.3 Waves at sea

Sea waves come in many different sizes (Figure 6.2). The babies in the family are the *capillary waves*, which are generated by wind and are only a few millimetres in length. Their shape and size is influenced by surface tension. This is a very small force and so for waves longer than this, the earth's gravity takes over control and these are often referred to as *gravity waves*. These also are generated by the wind. The longer waves are the tsunamis (see Section 6.6) and storm surges. But the longest of all are the tides which are controlled by the movement of the sun and moon. There are no simple dividing lines between the different waves – just a gradual change from one to the other.

The general formula for wave celerity is as follows:

$$C = \sqrt{\frac{gL}{2\pi}\tanh\left(\frac{2\pi d}{L}\right)}$$

where g is gravity constant (9.81 m/s^2); L is wave length (m); d is water depth (m).

The term tanh is a mathematical function known as the hyperbolic tangent and values can be obtained from mathematical tables in the same way as other functions such as sin and tan.

In deep water (defined as water deeper than half the wave length), as is the case of sea waves, the depth of the water has no effect on wave speed and so the formula simplifies to:

$$C = \sqrt{\frac{gL}{2\pi}}$$

This formula shows that the celerity of sea waves depends only on wave length. So waves of different length travel at different speeds. In an Atlantic storm, for example, waves of many different lengths are generated. As the waves start to move out of the storm area the longer waves move faster and so they begin to overtake the shorter waves. Waves can travel many hundreds of kilometres because there is very little friction to slow them down. When they reach the shores of Europe they are well sorted out with the longer waves arriving first followed eventually by the shorter waves. The rhythmic nature of waves arriving on a beach in this way is often referred to as *swell*. By timing the waves and noting the change in their frequency, it is possible to work out how far the waves have travelled since they were formed. In other words,

		gravity waves			
			long-period waves		
WAVE TYPE	capillary waves	wind waves	seiches and storm surges	tsunamis	tide waves
CAUSE	wind		wind, storms and other wind waves	storms and earth-quakes	Sun and Moon
SIZE	increasing →				

6.2 Types of sea waves.

it is possible to link waves to the particular weather events at sea that created them, perhaps several days before.

It was this kind of detailed study of waves and the weather patterns that form them that helped scientists to predict wave heights on the beaches of France during the very successful D-Day landings during the Second World War. Two scientists, Svedrup and Monk, developed a technique for predicting wave heights from weather patterns at sea. They also studied the celerity of waves travelling out from the storm area. From this they were able to predict the wave heights on the Normandy beaches resulting from bad weather conditions out in the Atlantic occurring several days earlier. This information was crucial for those landing on the beaches because the landing craft being used could only land safely at certain limited wave heights. Knowing the way in which the sea can change suddenly in the English channel it would have been a great risk to have gone ahead with such a major operation without some prior knowledge of what the sea was going to be like.

Similar analyses are regularly used today to predict sea conditions when tankers are refuelling ships at sea or ships are taking supplies to oil platforms. Predicting the wave heights that can be expected ensures successful and safe operations.

6.4 Waves in rivers and open channels

Waves which affect only the surface water can also occur in channels and are usually the result of some disturbance – like the waves which characterise super- and sub-critical flow. But in this case the water is shallow and so celerity of waves of this kind are related to the depth of the water. And the general formula is modified accordingly.
When depth *d* is significant:

$$\tanh\left(\frac{2\pi d}{L}\right) = \frac{2\pi}{L} \quad \text{and so } c = \sqrt{gd}$$

where *c* is wave celerity (m/s); *d* is depth of the water (m); *g* is the gravity constant (9.81 m/s^2).

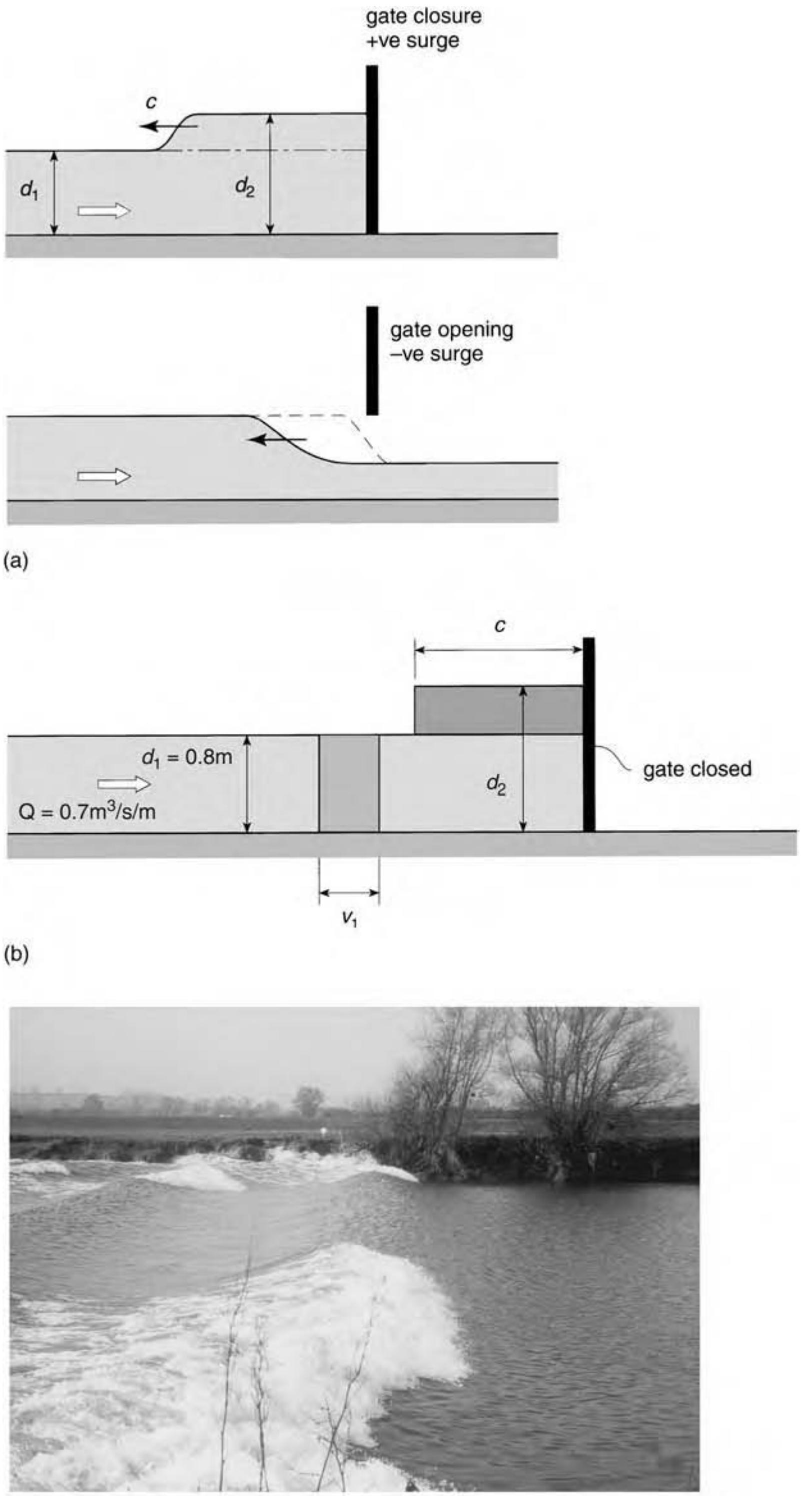
This formula was developed by Joseph Lagrange (1736–1813) and was further verified by John Scott Russel (1808–1882) who spent a great deal of time observing and measuring the speed of bow waves created on canals by horse-drawn barges.

But channels tend to be more associated with waves which have a significant effect on the whole flow and not just on the surface. Such waves include surges (or bores) which are sometimes seen moving upstream on tidal rivers or along channels after the closure of hydraulic gates. The hydraulic jump is sometimes described as a *standing wave.* It is similar to a surge but it stays in one place.

6.4.1 Surges

Surges occur when there is a sudden change in flow such as when a gate is closed or opened (Figure 6.3a). The sudden closure of a gate produces a positive surge whereas a sudden opening produces a negative surge. They are sometimes called *solitary waves*, that is, there is only one wave.

The strength of a surge depends on the change in the flow. When the change is very small the surge is undular (i.e. like a surface wave). When the change is large a much stronger surge develops. If it is a positive surge then it will have a vigorous rolling action and may look like a



(c) Tidal bore on R. Severn, UK

6.3 Surges in channels.

hydraulic jump but it is moving along the channel. A negative surge is much weaker and usually produces weak surface waves.

The celerity of a surge in a rectangular channel can be calculated using the formula:

$$C = \sqrt{\frac{gd_2(d_1+d_2)}{2d_1}} - v_1$$

where d_1 is upstream depth (m); d_2 is downstream depth (m); v_1 is upstream velocity (m/s).

This formula can be derived from the momentum equation, but this is not shown here. When the wave celerity is zero, that is, $c = 0$, the formula, if rearranged, is the same as that for the hydraulic jump (see Section 5.7.6). So one way of looking at a hydraulic jump is as a stationary (standing) wave or surge.

EXAMPLE: CALCULATING THE CELERITY AND HEIGHT OF A SURGE WAVE

Calculate the height and celerity of a surge wave resulting from the closure of a sluice gate on a canal when the discharge is 0.7 m^3/s/m width of channel and the normal depth of flow is 0.8 m (Figure 6.3b).

The formula for calculating the celerity is derived from the momentum equation:

$$c = \sqrt{\frac{gd_2(d_1+d_2)}{2d_1}} - v_1$$

but there are two unknown values, wave celerity c and the depth near the gate just after the sudden closure d_2. So another equation is needed before a solution can be found. This is the continuity equation but applied to surge (Figure 6.3b). This is done by calculating the volume of water coming down the channel in one second and equating this to the volume of water in the surge (the two volumes are shown shaded).

So:

$$v_1 d_1 = c(d_2 - d_1)$$

$$c = \frac{v_1 d_1}{d_2 - d_1}$$

Now:

$$d_1 = 0.8 \text{ m}$$

And:

$$v_1 = \frac{q}{d_1} = \frac{0.7}{0.8} = 0.88 \text{ m/s}$$

Put these values into the continuity equation:

$$c = \frac{0.88 \times 0.8}{d_2 - 0.88}$$

Put this equal to the wave celerity in the momentum equation:

$$\frac{0.88 \times 0.8}{d_2 - 0.88} = \sqrt{\frac{gd_2(0.8 + d_2)}{2 \times 0.8}} - 0.88$$

Solve this equation by trial and error, that is, by putting in different values of d_2 until one fits the equation:

$$d_2 = 1.14 \text{ m}$$

And so:

$$c = \frac{0.88 \times 0.8}{1.14 - 0.88 \times 0.8}$$

$$c = 2.07 \text{ m/s}$$

When a surge wave is not very high, that is, the depth d_2 is not significantly greater than the original depth in the channel d_1 then the wave celerity equation can be simplified by assuming that $d_1 = d_2$. So:

$$c = \sqrt{gd_1} - v_1$$

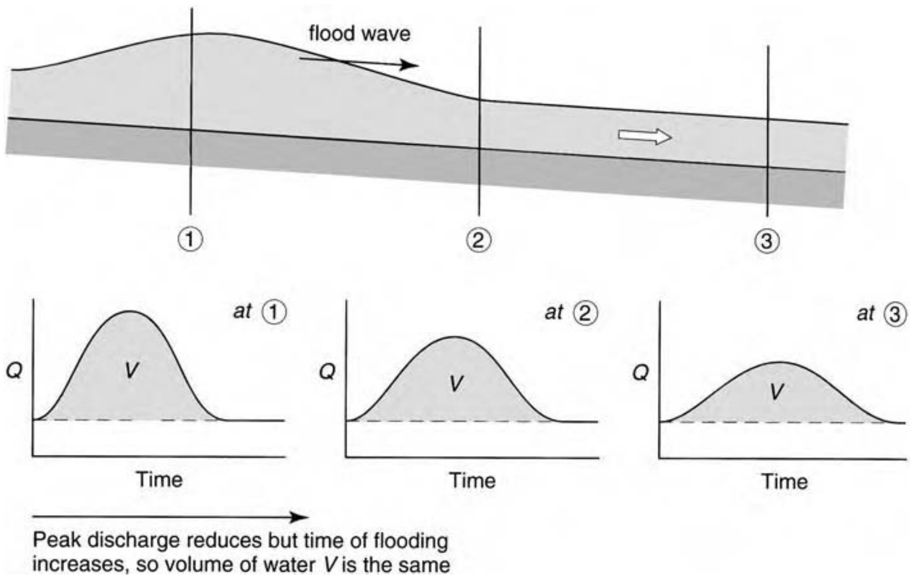
When the water velocity $v_1 = 0$ the celerity becomes $c = \sqrt{gd}$. This is the same formula used to calculate the celerity of gravity waves on a still pond (see Section 5.7.1).

6.4.2 Bores in tidal rivers

Some rivers regularly experience surges – sometimes called *bores* or *eagres*. They occur in tidal reaches close to estuaries. The most prominent surges occur where there is a high tidal range and where wide estuaries converge into a narrow river channel (Figure 6.3c). The rivers Severn and Trent in UK are famous for their bores. They are best observed during the spring tides when the tidal range is greatest and when river flows are at their lowest. The Severn bore starts in the estuary as high tide approaches and moves upstream reaching celerities of 5 m/s with a height of 1.5 m. Another famous surge, which attracts many surfers who like the challenge of riding big waves, is on the Amazon River in Brazil where it meets the Atlantic Ocean. It is up to 4 m high, running at up to 25 km/h and is known locally as the *pororoca* which is the local language for ‘the great destructive noise’.

6.5 Flood waves

Flood waves occur on rivers as a result of rainstorms on river catchments. They can be many hundreds of kilometres long; dealing with them is quite different from the much smaller waves already described. Instead of taking account of a whole wave along the entire length of a river, engineers deal with what happens to the discharge and water levels at particular sites over a period of time. A site may have been chosen because it is prone to flooding and there is a need to know what the effects of this will be, or it may be a convenient site for accurately measuring



6.4 Flood waves.

the discharge and water levels. Such observations lead to a graph of the discharge and the water level at that point on the river; it is called a *hydrograph* (Figure 6.4). Hydrographs can be very useful for dealing with river flow and flooding problems. For example, the area under the curve represents the volume of water that has passed the site during the flood. If the normal flow was also known it is possible to determine how much water was brought down by the flood.

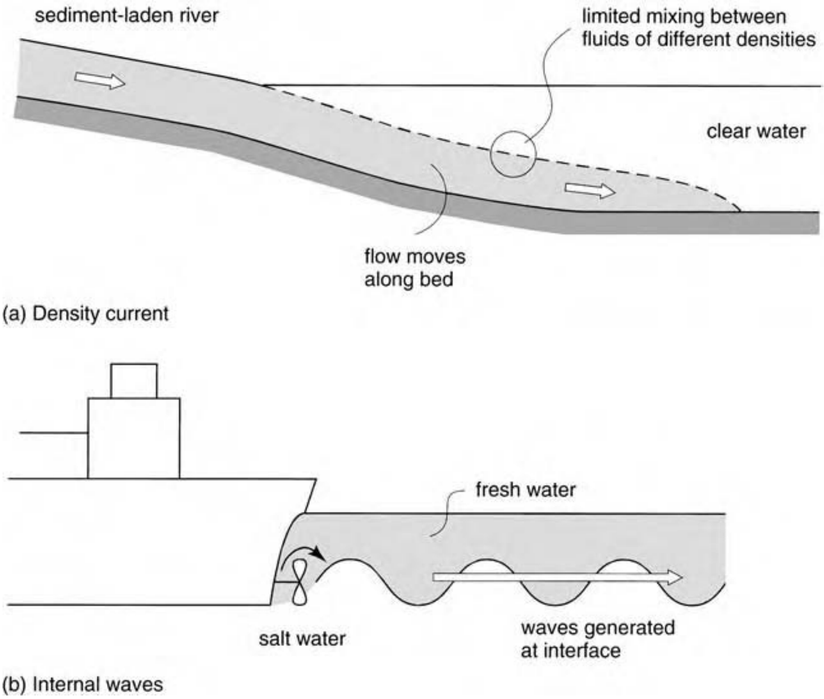
If another site is chosen further downstream and another hydrograph drawn, the two will have different shapes. They will both have the same area under the curve as they are both passing the same amount but the downstream curve will be longer and flatter – remember continuity. This means the maximum discharge is less but the flood duration is longer.

Gathering information on floods in this way provides valuable knowledge for river engineers who will be able to predict the effects of different rainstorms on river catchments and so design and construct engineering works which will help to alleviate and control flooding.

6.6 Some special waves

6.6.1 Density currents

When two fluids of different densities flow together they do not mix easily. The slow movement of sea water up estuaries during a rising tide under the outflowing fresh water from a river is an example of this. Fresh water has a slightly lower density than sea water and so the fresh water sits on the top and does not mix. Similarly, the flow of sediment-laden river water along the bed of the sea or a reservoir does not disperse into the main body of water (Figure 6.5a). Such flows, which occur when there are small differences in density, are called *density currents*. Another example is hot and cold water which do not mix easily because they have slightly different densities. Power stations sometimes have this same problem. They use a lot of cooling water; if the intake is close to the outlet then there is the risk of sucking hot water back into the station before it has had a chance to cool down through turbulent mixing with cooler water.



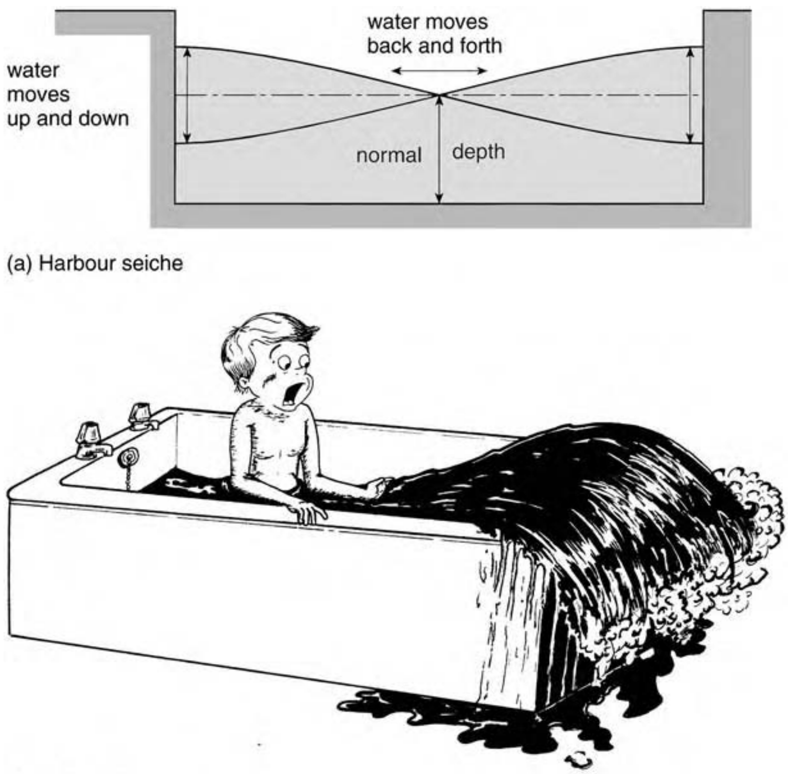
6.5 Density currents.

Waves can also occur at the interface between two fluids of different densities and can be quite intriguing. Oceanographers call these *internal waves* because the waves do not affect the water surface at all. Such waves seem to move in slow motion as if the force of gravity, which is controlling them, has been significantly reduced. This can be explained by the apparent loss in weight of the fluid as it lies on top of the denser lower fluid. The wave celerity equation still works but the value of g is much reduced by this buoyancy effect. When g is reduced the celerity is also reduced which means the waves move very slowly.

One internal wave system which caused some consternation occurred in the north Atlantic when a scientist was observing icebergs. The weather was mild and the ice was melting and there was about one metre of fresh water lying on top of the salt sea water (Figure 6.5b). Because of the slight density difference the two did not mix easily. When the observation ship's engines were started, to the surprise of the crew, the ship did not move, although the propellers were turning. The reason was the ship's propellers just happened to be located at the interface between the fresh and salt water and all the energy was being dissipated as wave energy along the interface. The sea surface was quite calm but just below, the waves were moving up and down at the interface and absorbing all the ship's energy.

6.6.2 Waves in harbours

Harbours are normally designed to keep out waves so that an area of calm water is created for ships to shelter from the sea for servicing and for loading cargo. This is done by having a narrow entrance. Most harbours do this successfully but there are cases when narrowing the harbour entrance has had the opposite effect – known as the *harbour paradox*. The narrow entrance has the effect of tuning certain incoming wave frequencies so rather than stopping the waves, the harbour amplifies them and they become worse than those outside. This is known as *harbour*



(a) Harbour seiche

(b) In the bath

6.6 Waves in harbours.

seiche. When this happens the waves reflect back and forth as there is very little in harbours to absorb the wave energy. It is very much like sliding up and down in the bath tub at home. The sliding action makes the water in the bath slosh back and forth (Figure 6.6). The bath tub has solid vertical sides which are very good at reflecting the waves but not so good at absorbing the wave energy. The waves can continue for some time until they finally settle down as the small amount of friction from the bath and your body take their toll. Harbours are very similar. They have lots of solid concrete surfaces and so waves tend to bounce back and forth rather than be absorbed.

The main concern in harbours is not so much with wave celerity but with the movement of water. As waves are reflected back and forth there is a great deal of water movement from one end of the harbour to the other (remember the bath again). The water velocity at the middle of the harbour may be as much as 0.5 m/s as water flows from one side to the other following the wave. This can create lots of problems. A ship moored at the mid-point could move by as much as 15 m back and forth following the movement of the wave. It is clearly undesirable to have a large ship moving about so much on its moorings when it is being loaded or unloaded. In harbours prone to this problem it is better to moor ships near the edge of the harbour where the water only moves up and down. From experience of harbour operation the most dangerous waves are those with a period of about two minutes. Such waves have been known to resonate in harbours which means that the frequency of the waves and the characteristics of the harbour, ships and their mooring systems combine to amplify the waves which makes the situation very much worse. All these factors are now taken into account when designing new harbour works or in repairing existing ones. Harbour designers wisely opt for model testing before construction to avoid these problems.

6.7 A tsunami wave coming ashore in Thailand 2004.

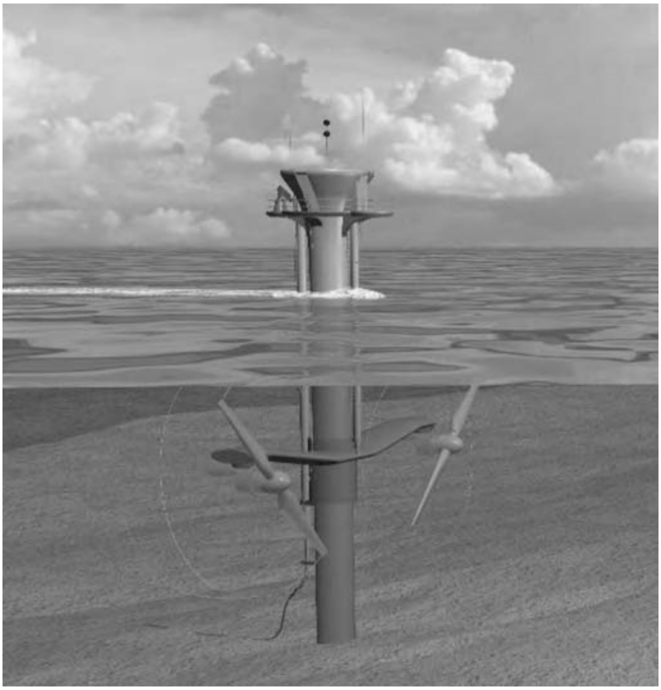
6.6.3 Tsunami wave

The *tsunami* is perhaps the most famous wave of all following the disastrous events in the Indian Ocean on 26 December 2004 (Figure 6.7). The word *tsunami* is Japanese for a long wave caused by earthquakes or landslides under the ocean. There is a common misconception that tsunamis behave like waves or swells which are generated by wind. They may have a similar appearance but they have a very different origin. The sudden movement of the sea bed during an earthquake causes a sudden movement of many millions of tons of water which can literally lift the sea level by a metre or more. The surge that comes from this weight of water has far greater force than any wind generated wave. This was evident from the devastating consequences shown in the many photographs and videos taken of the 2004 event.

Tsunamis are most common in the Pacific ocean. They travel at very high celerity with devastating consequences when they reach land. In the open ocean the wave may be only 0.5 m or so high but the wave celerity can be around 200 m/s when the ocean is say 5000 m deep. This might appear to be a surface wave but because of its great length its celerity is controlled by the depth of the ocean (remember the wave celerity equation $c = \sqrt{gd}$). Even so there is not much water displacement as it passes over the open ocean. A ship meeting this wave would hardly notice its passing. This is one of the reasons why many Indonesian fishermen survived the 2005 tsunami. They were fishing in boats off the coast and hardly noticed the tsunami as it passed underneath them. But nearer the shore was not a good place to be. As the tsunami runs into shallow water, say 100 m deep, the wave slows down to 30 m/s but as the energy in the wave is still the same the wave height increases substantially. At 30 m/s this is still quite fast and with the height increasing further as it runs up the beach it can do a great deal of damage to shore installations. The more destructive tsunamis are those which are funnelled into a narrow space when the width of a beach or harbour is restricted in some way. This has the effect of funnelling the energy of the wave into the harbour with destructive consequences.

6.7 Tidal power

Waves can be very useful for generating power. The most promising method is to hold incoming and outgoing tides behind a dam (or barrage) and use the head to drive turbines to generate



6.8 A new form of tidal turbine for generating electricity.

electricity. A power station using this idea was built on the estuary of the Raunce in Brittany, France in 1966 and there are plans to build a large barrage in the Severn estuary on the west coast of Britain. Although it has been estimated that this station could provide up to 6% of Britain's power needs the scheme is hotly debated because it would affect the pattern of the tides and sediment movements in the estuary and many ecologically sensitive wetland sites could be adversely affected.

Tidal barrages are an economic possibility when the tidal range exceeds 5 m but taking energy out of the tides may cause problems. In estuaries that naturally absorb tidal energy through friction and very little energy is reflected back out to sea there is the chance that a power station can take out the energy from the tides without serious effect. But in cases where there is very little friction and all the tidal energy is reflected back out to sea there may be dangers in extracting tidal energy. The effects of the energy not being reflected back again cannot be predicted.

More recent developments are looking at harnessing tidal energy without having to build large, expensive and environmentally risky structures. One such development is a turbine that sits on a tower in the sea which can be lifted out for routine maintenance (Figure 6.8).

In the 1920s one enterprising engineer undertook a study to establish the feasibility of constructing tidal power stations across the Irish Sea between Ireland and Wales and Ireland and Scotland. Because of the difficulties and high costs of measuring velocities across this large stretch of water he looked around for a hydraulically similar model on which to make measurements. He worked out that that Salisbury plain is hydraulically similar to the Irish sea and that the winds blowing across it produced similar Reynolds Numbers to the flow of water in the Irish sea. The study concluded that the Irish sea was more reflective than energy absorbing and so there were great doubts about the prospects for power generation.

7 Hydraulic structures for channels

7.1 Introduction

Hydraulic structures in channels have three main functions:

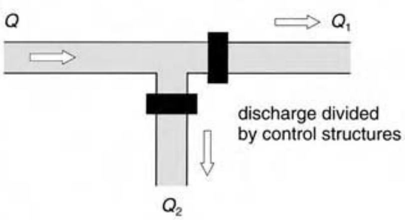
- measuring and controlling discharge
- controlling water levels
- dissipating unwanted energy.

The measurement and control of discharge in channels are perhaps one of the more obvious uses of hydraulic structures (Figure 7.1). Large irrigation networks, for example, require structures at each canal junction to measure and control discharge so that there is a fair and equitable distribution of water. It is not enough just to construct a canal junction and hope that the flow will divide itself properly between the two. Natural rivers too need regular flow measurement so that engineers can make sure there is an adequate supply to meet the growing demands for domestic and industrial uses as well as maintaining base flows for environmental purposes. It is important to ensure that minimum base flows are maintained in dry summer periods to protect fish stocks and environmentally sensitive wetlands. Flood flows are also measured so that adequate precautions can be taken to avoid or control flooding particularly in urban areas where damage can be very costly.

The need to control water levels in channels may not be so obvious. On irrigation schemes water level control is just as important as discharge control. The canals are built up higher than the surrounding ground level so there is enough energy for water to flow by gravity through pipes from the channels into the farms (Figure 7.2a). The water level (known as the command) must be carefully controlled if each farm is to receive the right discharge. But all too often the water level drops because of low flows or seepage losses. This reduces the discharge through the pipes making it difficult for farmers to irrigate properly. To avoid this problem, hydraulic structures are built across the canals to raise the water levels back to their command levels. Such structures are called *cross regulators* (Figure 7.2).

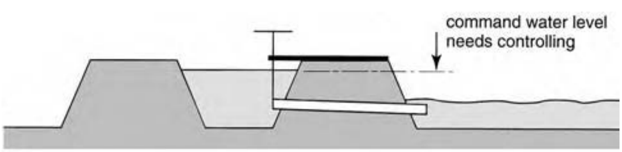
Another example of the need for water level control might be on a river close to a natural wetland site which is valued for its bird population or its special plants (Figure 7.2b). Water might enter the site either by the flooding from the river as it overtops its banks or through seepage from its bed and sides. Either way the wetland is very dependent on the water level in the river as well as its flow to avoid it drying out and causing irreparable damage

(a) Discharge regulator in a canal system in France

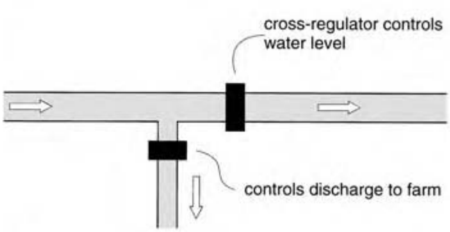


(b)

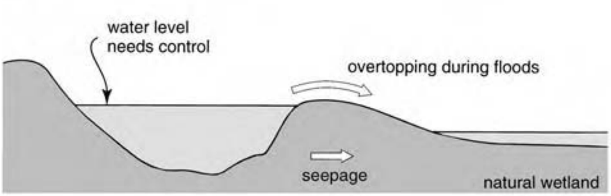
7.1 Measuring and controlling discharge.



(a)



(b)



(c)

7.2 Controlling water levels.

to flora and fauna. Fluctuations in water level can be avoided by building a structure across the river to hold the water at the desired level throughout the year even though the discharge may vary significantly.

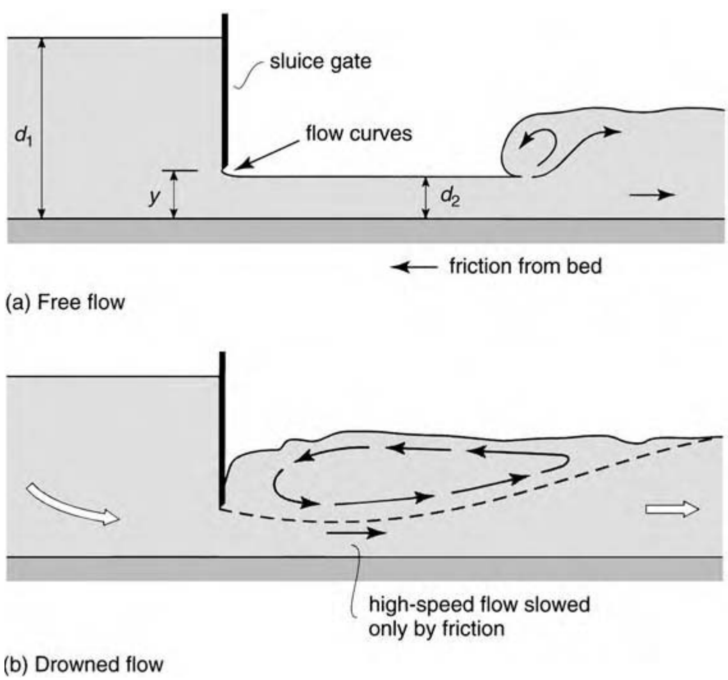
Hydraulic structures are also very useful for getting rid of unwanted energy. When water flows down dam spillways it can reach speeds of 60 km/h and more and is capable of doing a lot of damage. Hydraulic structures are used to stop such high speed flows and dissipate the kinetic energy by creating hydraulic jumps.

Some hydraulic structures only carry out one of the functions described above whilst others perform all three functions at the same time. So a hydraulic structure may be used for discharge measurement, and at the same time may be performing a water level control function and dissipating unwanted energy.

From the point of view of measuring discharge and controlling water levels, there are only two types of structure. Some structures allow water to flow through them and these are called *orifice structures*. Others allow water to flow over them and these are called *weirs or flumes*. Hydraulically, they behave in quite different ways and so each has certain applications for which they are best suited. The energy dissipating function can be attached to both of these structure types.

7.2 Orifice structures

The principles of orifice flow were described earlier in Chapter 3. In practice, orifice structures have fixed or movable gates rather than just a simple opening. The sluice gate described in Section 5.4.3 is a good example of this type of structure. The flow under the gate is very similar to orifice flow but not quite (Figure 7.3a). First, the flow contracts only on its upper surface as it goes under the gate and second there is additional friction from the bed of the



7.3 Orifice structures.

channel. So to find a formula for discharge for this structure the orifice formula is a good starting point, but it needs modifying.

The formula for discharge from an orifice is:

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

Modifying this for a sluice gate:

$$Q = C_d a\sqrt{2gd_1}$$

Where Q is discharge (m/s^3); a is area of gate opening (m^2); C_d is a coefficient of discharge; d_1 is the water depth upstream of the orifice (m).

This looks to be a simple formula. It is an attempt to relate the discharge to the area of the gate opening a and the up-stream water depth d_1 because both are easy to measure. However, the relationship is not so simple; as a result, C_d is not a simple coefficient of discharge as defined earlier in Chapter 3. It takes account of the contraction of the flow under the gate, but, in addition, it allows for energy losses and the effects of the size of gate opening and the head on the gate. So C_d is not a simple constant number like the coefficient of contraction C_c. Usually the manufacturer of hydraulic gates will supply suitable values of C_d so that 'simple' discharge formulae can be used.

7.2.1 Free and drowned flow

The sluice gate example shows the flow freely passing under the gate with a hydraulic jump downstream. The downstream depth d_2 has no effect on the upstream depth d_1. This is referred to as *free flow* and the formula quoted above for calculating discharge is based on this condition.

In some circumstances the jump can move upstream and drown out the gate and is referred to as *drowned flow* (Figure 7.3b). The flow downstream may look very turbulent and have the appearance of a jump but inside the flow the action is quite different. There is very little turbulent mixing taking place and the super-critical flow is shooting underneath the sub-critical flow. This high speed jet is not stopped quickly as it would in a jump but slows down gradually over a long distance through the forces of friction on the channel bed. This flow can do a lot of damage to an unprotected channel even though the water surface may appear to be quite tranquil on the surface. Under drowned conditions the formula for discharge must be modified to take account of the downstream water level which now has a direct influence on the upstream water level.

7.3 Weirs and flumes

Weirs and flumes are both overflow structures with very different characteristics to orifices. Many different types of weirs have been developed to suit a wide range of operating conditions, some comprise just a thin sheet of metal across a channel (*sharp-crested weirs*), whereas others are much more substantial (*solid weirs*). Both are based on the principle of changing the energy in a channel and using the energy and continuity equations to develop a formula for discharge based on depth (pressure) measurements upstream. But solid weirs rely on an energy change which is sufficient to make the flow go through the critical point. Because of this they are sometimes called *critical-depth structures*. Sharp-crested weirs do not have this constraint – but they do have others.

Flumes are also critical-depth structures but rely on changing the energy by narrowing the channel width rather than raising the bed. Sometimes engineers combine weirs and flumes by both raising the bed and narrowing the width to achieve the desired energy change. It does not really matter which way critical flow is achieved so long as it occurs.

7.4 Sharp-crested weirs

Sharp-crested weirs are used to measure relatively small discharges (Figure 7.4a). They comprise a thin sheet of metal such as brass or steel (sometimes wood can be used for temporary weirs) into which a specially shaped opening is cut. This must be accurately cut leaving a sharp edge with a bevel on the downstream side. When located in a channel, the thin sheet is sealed into the bed and sides so that all the water flows through the opening. By measuring the depth of water above the opening, known as the head on the weir, the discharge can be calculated using a formula derived from the energy equation. There is a unique relationship between the head on the weir and the discharge and one simple depth measurement determines the discharge.

7.4.1 Rectangular weirs

This weir has a rectangular opening (Figure 7.4b). Water flows through this and plunges downstream. The overflowing water is often called the *nappe*. The discharge is calculated using the formula:

$$Q = \frac{2}{3} C_d L \sqrt{2g} H^{1.5}$$

where C_d is a coefficient of discharge; L is length of weir (m); H is the head on the weir measured above the crest (m).

C_d allows for all the discrepancies between theory and practice.

7.4.2 Vee-notch weirs

This weir has a triangular shaped notch and is ideally suited for measuring small discharges (Figure 7.4c). If a rectangular weir was used for low flows, the head would be very small and difficult to measure accurately. Using a vee weir, the small flow is concentrated in the bottom of the vee providing a reasonable head for measurement.

The discharge is calculated using the formula:

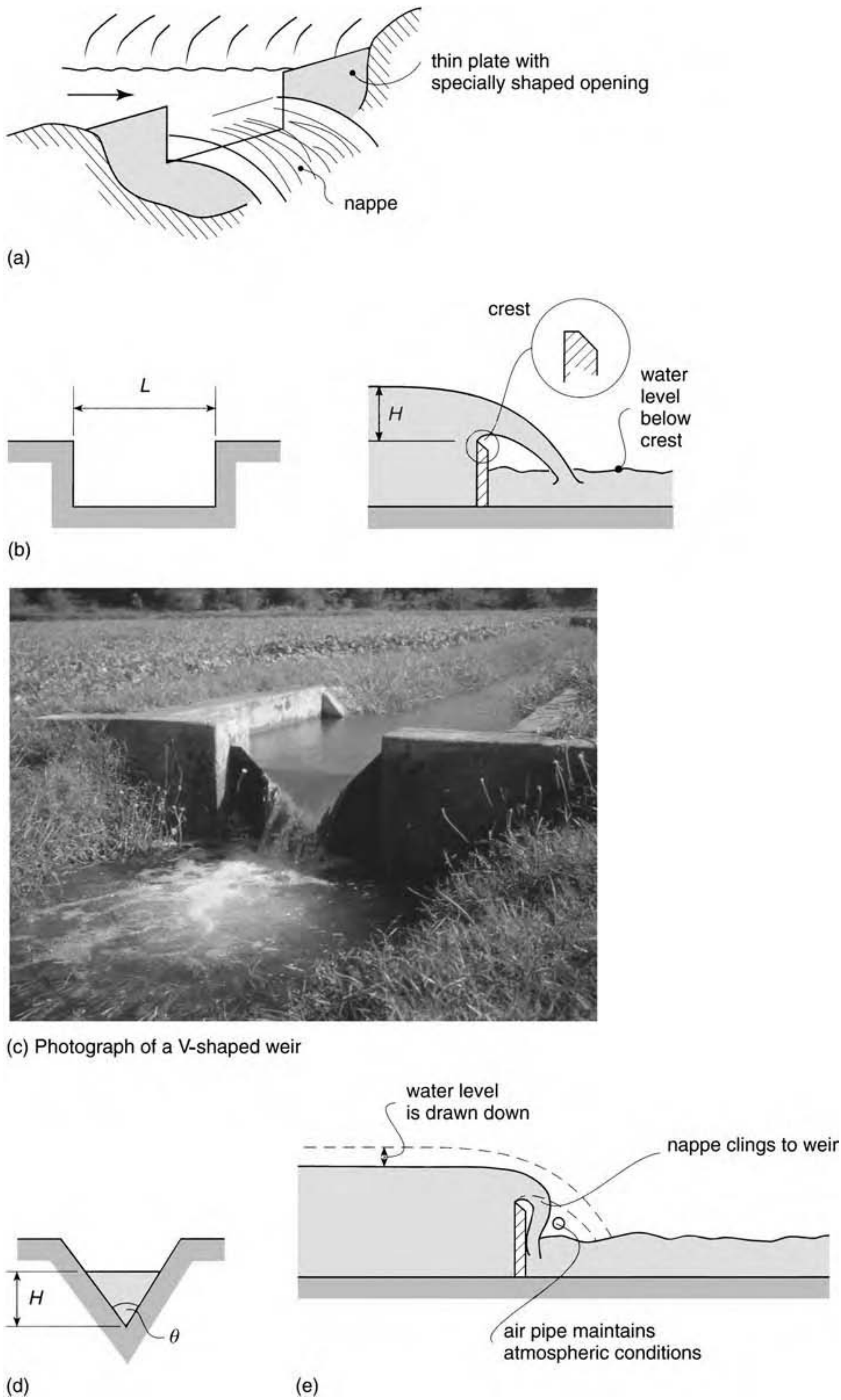
$$Q = \frac{8}{15} C_d \sqrt{2g} \tan\left(\frac{\theta}{2}\right) H^{2.5}$$

where θ is the angle of the notch.

7.4.3 Some practical points

There are several conditions that must be met for these weirs to work properly. These are set out in detail in British Standard BS3680 (see references for details). The following are some of the key points:

- The water must fall clear of the weir plate into the downstream channel. Notice the bevelled edge on the crest facing downstream which creates the sharp edge and helps the water to spring clear. If this did not happen the flow clings to the downstream plate and draws down the flow reducing the head on the weir (Figure 7.4d). Using the above formula with the reduced value of head would clearly not give the right discharge.



7.4 Sharp-crested weirs.

- Flow over the weir must be open to the atmospheres so that the pressure around it is always atmospheric. Sometimes the falling water draws air from underneath the weir and unless this air is replaced a vacuum may form which causes the flow to cling to the downstream face (Figure 7.4d). This draws down the upstream water level reducing the head on the weir and giving a false value of discharge when it is put into the formula. To prevent this, air must be allowed to flow freely under the nappe.
- The weir crest must always be set above the downstream water level. This is the *free flow* condition for sharp-crested weirs. If the downstream level rises beyond the crest, it starts to raise the upstream level and so the weir becomes *drowned*. Another word that is used to describe this condition is *submerged flow*. The formula no longer works when the flow is drowned and so this situation must be avoided by careful setting of the weir crest level.
- The head H must be measured a few metres upstream of the weir to avoid the draw-down effect close to the weir.
- When deciding what size of weir to use it is important to make sure there is a reasonable head so that it can be measured accurately. This implies that you need to have some idea of the discharge to be measured before you can select the right weir size to measure it. If the head is only say, 4 mm then 1 mm error in measuring it is a 25% error and will result in a significant error in the discharge. However, if the head is 100 mm then a 1 mm error in measuring the head is only a 1% error and so is not so significant.

Sharp-crested weirs can be very accurate discharge measuring devices provided they are constructed carefully and properly installed. However, they can be easily damaged, in particular the sharp crest. If this becomes rounded or dented through impact with floating debris, the flow pattern over the weir changes and this reduces its accuracy. For this reason they tend to be unsuited for long-term use in natural channels but well suited for temporary measurements in small channels, in places where they can be regularly maintained and for accurate flow measurement in laboratories.

7.5 Solid weirs

These are much more robust than sharp-crested weirs and are used extensively for flow measurement and water level regulation in rivers and canals (Figure 7.5a).

All solid weirs work on the principle *that the flow over the weir must go through the critical depth*. The idea of critical depth was discussed in Chapter 6 where it was shown that it was the height of a weir which determined whether or not the flow goes critical. Once this happens a formula for discharge can be developed using the concept of specific energy and the special conditions that occur at the critical point.

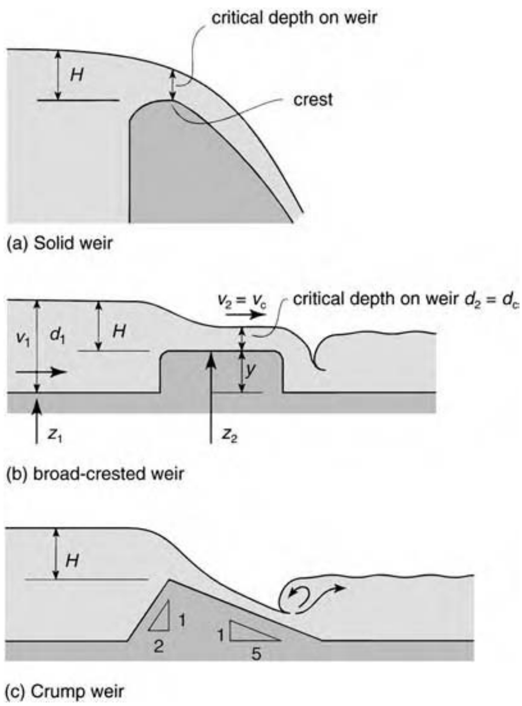
The formula links the channel discharge (Q) with the upstream water depth *measured above the weir crest* (H).

$$Q = CLH^{1.5}$$

where C is weir coefficient; L is length of the weir crest (m); H is head on the weir measured from the crest (m).

To see how this formula is developed see the box.

As there is some draw down close to the weir, the head is usually measured a few metres upstream where the water level is unaffected by the weir.



7.5 Solid weirs.

DERIVATION: FORMULA FOR DISCHARGE OVER A CRITICAL FLOW STRUCTURE

Derive a formula for discharge for a critical flow structure in an open channel. Use a broad-crested weir as an example of a critical flow structure, although the analysis would be the same for any similar structure (Figure 7.5b).

First write down the total energy equation for the flow in the channel (point 1) and the flow on the weir (point 2):

total energy at point 1 = total energy at point 2

$$d_1 + \frac{v_1^2}{2g} + z_1 = d_2 + \frac{v_2^2}{2g} + z_2$$

Now:

$$z_2 - z_1 = y$$

That is, the height of the weir is equal to the difference in the potential energy and so:

$$d_1 + \frac{v_1^2}{2g} = d_2 + \frac{v_2^2}{2g} + y$$

But at the critical depth:

$$d_2 = d_c$$

And:

$$v_2 = v_c$$

Substitute these into the energy equation:

$$d_1 + \frac{v_1^2}{2g} = d_c + \frac{v_c^2}{2g} + y$$

But at the critical depth:

$$\frac{v_c^2}{2g} = \frac{d_c}{2}$$

(See derivation of the formula for critical depth in Section 5.7.4.) Put this into the equation:

$$d_1 + \frac{v_1^2}{2g} = d_c + \frac{d_c}{2} + y$$

Rearrange this:

$$d_1 + \frac{v_1^2}{2g} - y = \frac{3}{2}d_c$$

But critical depth can be calculated from the formula:

$$d_c = \sqrt[3]{\frac{q^2}{g}}$$

(See derivation of the formula for critical depth in Section 5.7.4.) Also put:

$$d_1 + \frac{v_1^2}{2g} - y = H$$

This means that *H* is measured from the weir crest and so:

$$H = \frac{3}{2}\sqrt[3]{\frac{q^2}{g}}$$

Rearrange this for *q*:

$$q = \left(\frac{2}{3}\right)^{3/2} g^{1/2} H^{3/2}$$

$$q = 1.71H^{3/2}$$

This is the theoretical flow and an allowance now needs to be made for minor energy losses. This is usually combined with the 1.17 and introduced as a coefficient *C*. So:

$$q = CH^{1.5}$$

Here q is the discharge per unit width and so the full discharge Q is calculated by multiplying this by the length of the weir L:

$$Q = CLH^{1.5}$$

Note that strictly speaking H is the measurement from the weir crest to the energy line as it includes the kinetic energy term. In practice H is measured from the weir crest to the water surface. The error involved in this is relatively small and can be taken into account in the value of the weir coefficient C.

As the formula is based on critical depth it is not dependent on the shape of the weir. So the same formula can be used for any critical-depth weir, not just for broad-crested weirs. Only the value of C changes to take account of the different weir shapes.

7.5.1 Determining the height of a weir

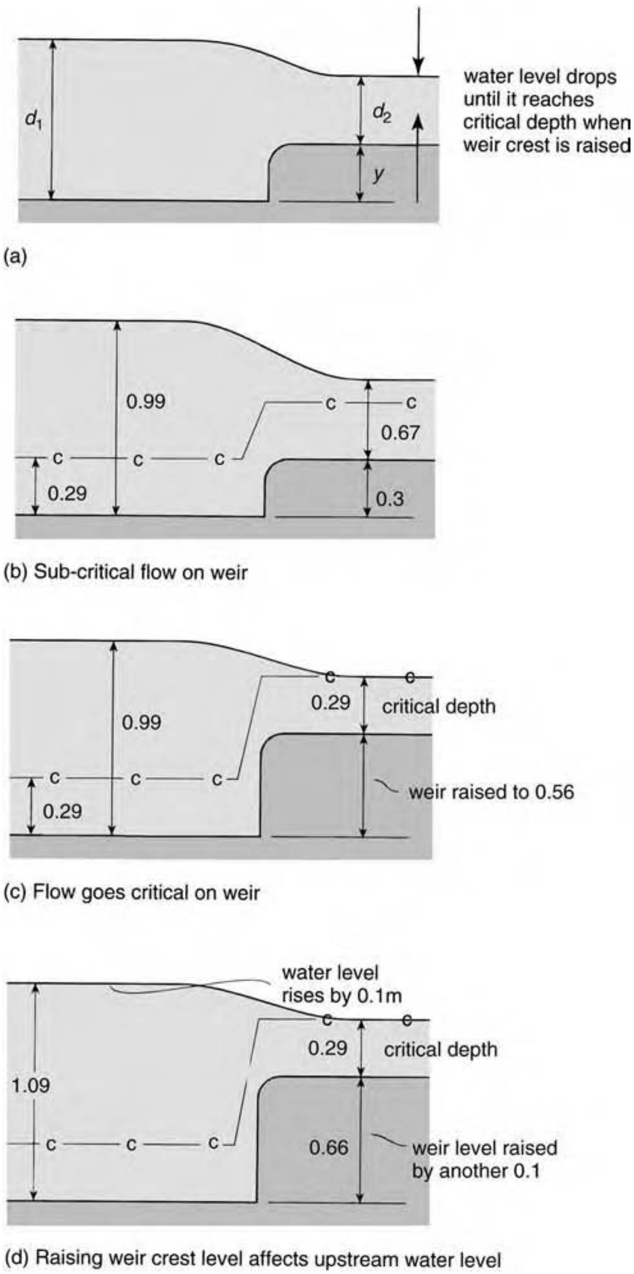
Just how high a weir must be for the flow to go critical is determined from the specific energy diagram. The effect of constructing a weir in a channel is the same as building a step up on the bed as described in Section 5.7.3. This was concerned only with looking at what happens to the depth when water flows over a step. In the case of a step up, the depth on the step decreased and the velocity increased (Figure 7.6a). At that time no thought was given to making the flow go critical. A worked example showed that for a 0.3 m high step up, the depth of water was reduced from 0.99 m upstream to 0.67 m on the step (this is summarised in Figure 7.6b). This is still well above the critical depth of 0.29 m (see calculation in Section 5.7.4).

Now assume that the step up on the bed is a weir and the intention is to make the flow go critical on the weir crest. This can be achieved by raising the crest level. Raising it from 0.3 m to 0.56 m further reduces the depth on the weir from 0.67 m to 0.29 m which is the critical depth (Figure 7.6c) (this can be worked out using the specific energy diagram in Section 5.7.3). This is the minimum weir height required for critical flow. Note that although the weir height has increased by 0.26 m, the upstream depth remains unchanged at 0.99 m. If the weir height is increased beyond 0.56 m the flow will still go critical on the crest and remain at the critical depth of 0.29 m. It will not and cannot fall below this value. The difference will be in the upstream water level upstream which will now rise. Remember there is a unique relationship between the head on a weir and the discharge. So if the weir is raised by a further 0.1 m to 0.66 m the upstream water level will also be raised by 0.1 m to maintain the correct head on the weir (Figure 7.6d).

The operation of weirs is often misunderstood and it is believed that they cause the flow to back up and so raise water levels upstream. This only happens once critical conditions are achieved on the weir. When a weir is too low for critical flow it is the water level on the weir which drops. The upstream level is unaffected. But once critical flow is achieved, raising the weir more than is necessary will have a direct effect on the upstream water level.

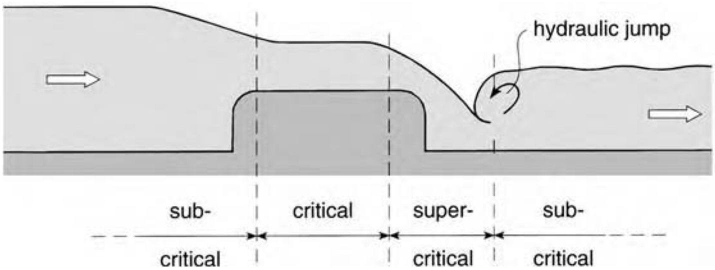
7.5.1.1 Being sure of critical flow

Critical flow must occur for the discharge formula to work. But in practice it is not always possible to see critical flow and so some detective work is needed. Figure 7.7 shows the changing flow conditions as water flows over a weir. Upstream the flow is sub-critical, it then



7.6 Determining the height of a weir.

goes critical over the weir and then super-critical downstream. It changes back to sub-critical through a hydraulic jump. When this sequence of changes occurs it can be reasoned that critical flow must have occurred and so the weir is working properly. The changes are best verified in reverse from the downstream side. Remember a hydraulic jump can only form when the flow is super-critical and so if there is a hydraulic jump in the downstream channel, the flow over the



7.7 Being sure of critical flow.

weir must be super-critical. If the upstream flow is sub-critical, which can be verified by the water surface dropping as water flows over the weir, then somewhere in between the flow must have gone critical. So a hydraulic jump downstream is good evidence that critical flow has occurred.

Note that it is not important to know exactly where critical flow occurs. It is enough just to know that it has occurred for the formula to work.

7.5.2 Broad-crested weirs

These are very common structures used for flow measurement. They have a broad rectangular shape with a level crest rounded at the edge (Figure 7.5b). The value of *C* for a broad-crested weir is 1.6 and so the formula becomes:

$$Q = 1.6\ L\ H^{1.5}$$

One disadvantage of this weir is the region of dead water just upstream. Silt and debris can accumulate here and this can seriously reduce the accuracy of the weir formula. Another is the head loss between the upstream and downstream levels. Whenever a weir (or a flume) is installed in a channel there is always a loss of energy particularly if there is a hydraulic jump downstream. This is the hydraulic price to be paid for measuring the flow.

EXAMPLE: CALCULATING DISCHARGE USING A BROAD-CRESTED WEIR

A broad-crested weir is used to measure discharge in a channel. If the weir is 2 m long and the head on the crest is 0.35 m, calculate the discharge.
The discharge over a broad-crested weir can be calculated using the formula:

$$Q = 1.6\ L\ H^{1.5}$$

Put in values for length *L* and head *H*:

$$Q = 1.6 \times 2 \times 0.35^{1.5}$$

$$Q = 0.66\ m^3/s$$

7.5.3 Crump weirs

These weirs are commonly used in the UK for discharge measurement in rivers. Like the broad-crested weir it relies on critical conditions occurring for the discharge formula to work. It has a triangular-shaped section (Figure 7.5c). The upstream slope is 1 in 2 and the downstream is 1 in 5. The sloping upstream face helps to reduce the dead water region which occurs with broad-crested weirs. It can also tolerate a high level of submergence. Its crest can also be constructed in a vee shape so that it can be used accurately for both small and large discharges.

7.5.4 Round-crested weirs

Weirs of this kind are commonly used on dam spillways (Figure 7.5a). The weir profile is carefully shaped so that it is very similar to the underside of the falling nappe of a sharp-crested weir (compare the two shapes in Figure 7.5a and Figure 7.4b). Many standard designs are available which have been calibrated in the laboratory using hydraulic models to obtain the C values. An example is the standard weir designs produced by the US Bureau of Reclamation (see references for details). By constructing a weir to the dimensions given in their publications, the discharge can be measured accurately using their C values (usually between 3.0 and 4.0).

7.5.5 Drowned flow

Weirs which rely on critical depth are much less sensitive to being *drowned* (or *submerged*) than the sharp-crested type. This means that the downstream water level can rise above the weir crest without it affecting the performance of the structure *provided* the flow still goes critical somewhere on the weir. There are ways of using weirs even when they are completely submerged but they are far less accurate and both upstream and downstream water depths must be measured. It is better to avoid this situation if at all possible.

7.6 Flumes

Flumes also rely on critical flow for measuring discharges. They are sometimes called *throated flumes* because critical conditions are achieved by narrowing the width of the channel (Figure 7.8). Downstream of the throat there is a short length of super-critical flow followed by a hydraulic jump. This returns the flow to sub-critical. The formula for discharge can be determined in the same way as for solid weirs. The result is as follows:

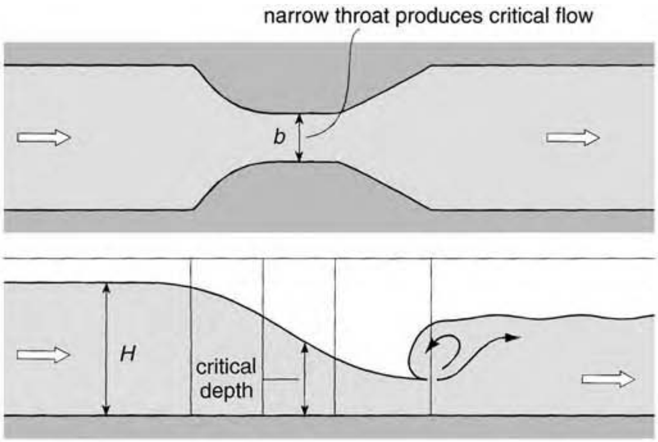
$$Q = 1.65\, b\, H^{1.5}$$

where b is width of the flume throat (m); H is upstream depth of water (m).

The head loss through flumes is much lower than for weirs and so they are ideally suited for use in channels in very flat areas where head losses need to be kept as low as possible.

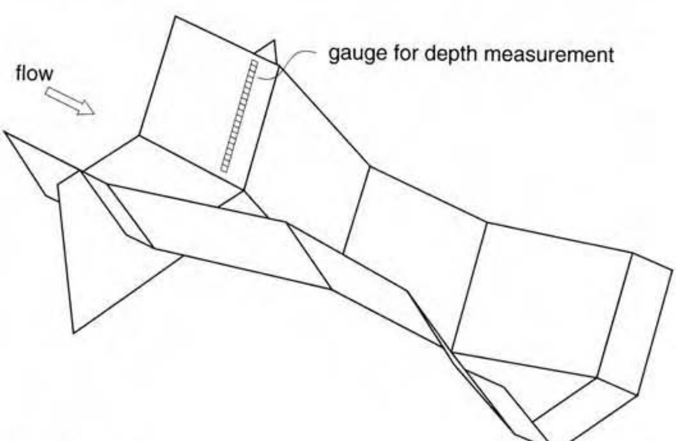
7.6.1 Parshall flumes

Parshall flumes are used extensively in the USA and were developed by R.L. Parshall in 1926 (Figure 7.8b). They gained popularity in many other countries because the construction details, dimensions and calibration curves relating upstream depth to discharge have been widely published. There are several different sizes available to measure flows up to 90 m^3/s. They are



(a) Venturi flume

(b) Parshall flume



(c) WSC flume

7.8 Flumes.

relatively simple to construct from a range of materials such as wood, concrete and metal because they have no curved surfaces.

If they are made and installed as recommended they provide accurate discharge measurement.

7.6.2 WSC flumes

This is a range of standard flumes for measuring small discharges from less than 1.0 l/s up to 50 l/s developed by Washington State University in USA (Figure 7.8c). They are vee-shaped so that they can be used to measure low flows accurately and, like Parshall flumes, they can be easily made up in a workshop from metal or glass fibre. They are particularly useful as portable flumes for spot measurements in small channels and irrigation furrows.

7.6.3 Combination weir-flumes

Sometimes both weir and flume effects are combined to achieve critical flow. The advantages of the vee for low flows can also be added. In such cases a laboratory model test is needed to determine the *C* value in the discharge equation.

7.7 Discharge measurement

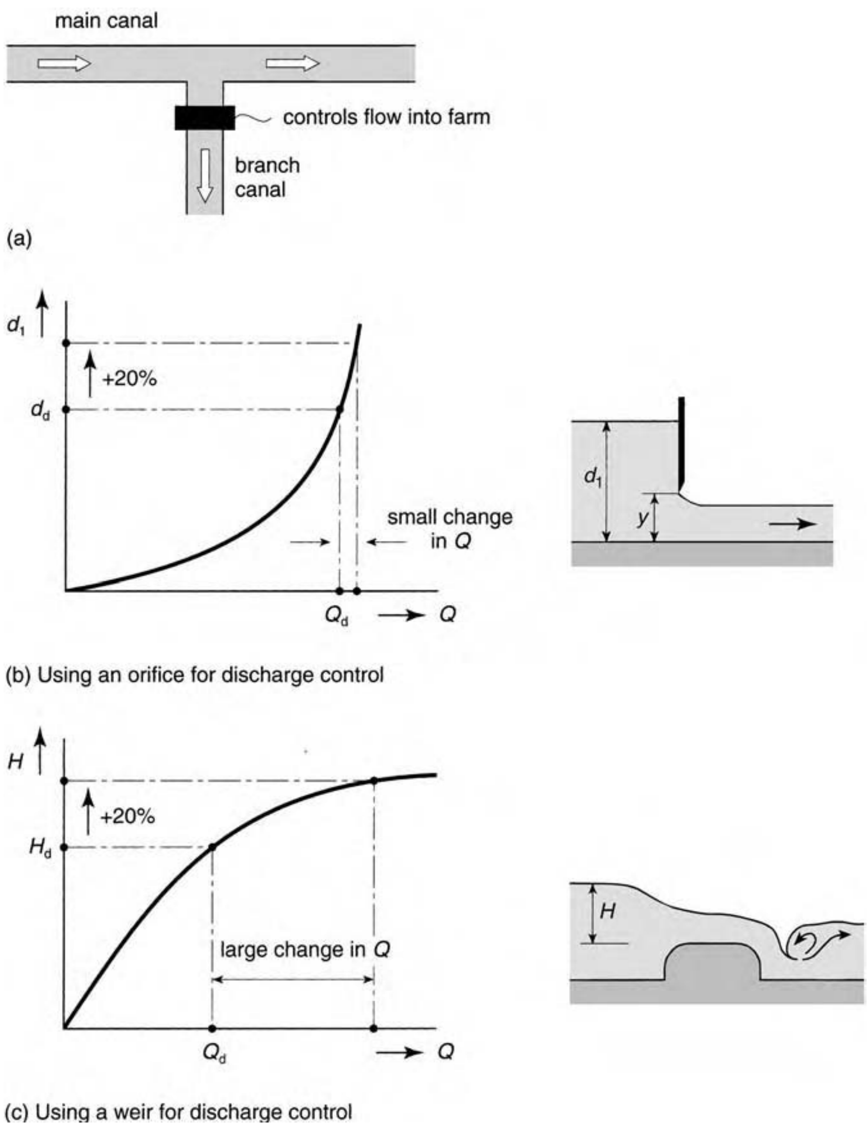
Weirs, flumes and orifices can all be used for discharge measurement. But weirs and flumes are better suited to measuring discharges in rivers when there can be large variations in flow. Weirs and flumes not only require a simple head reading to measure discharge but they can also pass large flows without causing the upstream level to rise significantly and cause flooding. Orifice structures too can be used for flow measurement but both upstream and downstream water levels are usually required to determine discharge. Large variations in flow also mean that the gates will need constant attention for opening and closing.

7.8 Discharge control

Although orifices are rather cumbersome for discharge measurement they are very useful for discharge control. This is because the discharge through an orifice is not very sensitive to changes in upstream water level. Consider as an example, an irrigation canal system where a branch canal takes water from a main canal (Figure 7.9a). The structure at the head of the branch controls the discharge to farmers downstream. The ideal structure for this would be an orifice and the reason lies in its hydraulic characteristic curve (Figure 7.9b). This is a graph of the orifice discharge equation:

$$Q = C\sqrt{2gd_1}$$

In this simple example, the orifice opening is assumed to be fixed and so discharge Q changes only when the upstream depth d_1 changes. The point Q_d and d_d on the graph represents the normal operating condition. Now suppose the main canal operating depth rises by say, 20%, because of changes in the water demand elsewhere in the system. The effect that this would have on the discharge into the branch (through the orifice) is to change it by only 5%. So even though there is a significant change in the main canal this is hardly noticed in the branch. This can be very useful for ensuring a reliable, constant flow to a farmer even though the main canal may be varying considerably due to changing demands.



7.9 Discharge control.

In contrast, if a weir (or flume) is installed at the head of the branch canal, it would be very easy to use for discharge measurement but it would not be so good for controlling the flow. Again look at the hydraulic characteristic curve for a weir (Figure 7.9). This is a graph of the weir discharge equation:

$$Q = CLH^{1.5}$$

Q_d and H_d represent the normal operating condition. Now if the water level in the main canal rises by 20% this almost doubles the discharge in the branch canal. So this type of structure is very sensitive to water level changes and would not make a very good discharge control structure.

These examples show the sensitivity of the two types of control structure. This is how they react to small changes in water level. But they refer to fixed structures. Most structures have gates (both weirs and orifices) which can be used to adjust the discharge but this adds to the burden of managing the channels. Better to use a structure which is hydraulically suited to the job to be done and which reduces the need for continual monitoring and adjustment.

7.9 Water level control

Orifices, weirs and flumes can all be used for water level control. But the very reason that makes an orifice a good discharge regulator make it unsuitable for good water level control. Conversely, a weir (or flume) is well suited to water level control but not for discharge control.

Imagine the water level in a river needs to be controlled to stabilise water levels in a nearby wetland site and it needs to be more or less the same both winter and summer even though the river flows change from a small flow to a flood. The structure best suited for this would be a weir as it can pass a wide range of flows with only small changes in head (water level). An orifice structure would not be suitable because large changes in water level would occur as the flow changed and the structure would require a great deal of attention and adjustment.

Another water level control problem occurs in reservoirs. When a dam is built across a river to store water, a spillway is also constructed to pass severe flood flows safely into the downstream channel to avoid over-topping the dam. The ideal structure for this is a weir because large discharges can flow over it for relatively small increases in water level. The rise in water level can be calculated using the weir discharge formula and this helps to determine the height of the dam. A freeboard is also added to the anticipated maximum water level as an added safety measure. The spillway can be the most expensive part of constructing a new dam. Many small dams are built on small seasonal streams in developing countries to conserve water for domestic and agricultural use. But even a small dam will need a spillway and although the stream may look small and dry up occasionally, they can suffer from very severe floods which are difficult to determine in advance, particularly when rainfall and discharge data for the stream are not available. Building a spillway for such conditions can be prohibitively expensive and far exceed the cost of the dam and so it may be cheaper in sparsely populated areas to let the dam be washed away and then re-build it on the rare occasions that this happens. There are of course many other factors to take into account besides dam reconstruction costs, such as the effects downstream of a dam break. It is a sobering thought that even the biggest and most important dams can fail because of severe flooding. They have large spillways to protect them from very severe floods which are carefully assessed at the design stage. But it is impossible to say that they will safely carry every flood. Nature seems to have a way of testing us by sending unexpected rainstorms but fortunately these events are few and far between.

One way of avoiding the expensive spillway problem is to construct a reservoir at the side of a river rather than use the river channel itself. This is called *off-stream storage*. Water is taken from the river by gravity or by pumping into a reservoir and only a modest spillway is then needed which would have the same capacity as the inlet discharge. When a flood comes down the river there is no obstruction in its path and so it flows safely pass the reservoir.

7.10 Energy dissipators

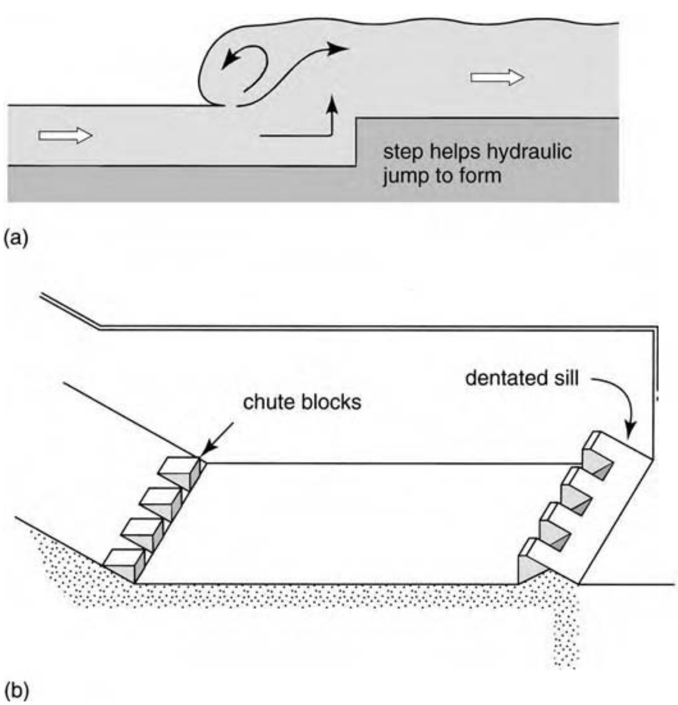
7.10.1 Stilling basins

When water flows over a weir or through a flume and becomes super-critical it can do a lot of damage to the downstream channel if it is left unprotected. This is particularly true when water rushes down an overflow spillway on a dam. The water can reach very high velocities by the time it gets to the bottom. Scour can be prevented by lining the channel but this can be an expensive option.

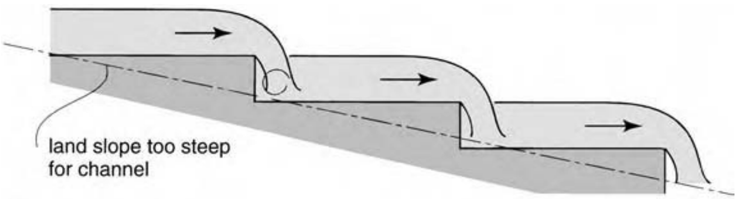
The alternative is to convert the flow to sub-critical using a hydraulic jump. The requirements for a jump are super-critical flow upstream and sub-critical flow in the downstream channel. The main problem is to create the right flow conditions in the downstream channel for a jump to occur even though the discharge may range from a small overspill flow to a large flood. Consider what happens when flow reaches the bottom of a spillway. If the tail-water is too shallow for a jump to form the super-critical flow will shoot off downstream and no jump will form. If the water is too deep the jump will be drowned and the super-critical flow will rush underneath and still cause erosion for some distance downstream. These problems can be resolved by building *stilling basins* which create and confine hydraulic jumps even though the tail-water may not be at the ideal depth for a jump to occur naturally. There are many different designs available but perhaps the simplest is a small vertical wall placed across the channel (Figure 7.10a). Other more sophisticated designs have been developed in laboratories using models as it is not possible to design them using formulae. The choice of stilling basin is linked to the Froude Number of the upstream super-critical flow (Figure 7.10b).

7.10.2 Drop structures

Drop structures are used to take flow down steep slopes step by step to dissipate energy and so avoid erosion (Figure 7.11). Channels on steep sloping land are prone to erosion because of the high velocities. One option to avoid this is to line channels with concrete or brick but another is to construct natural channels on a gentle gradient and to build in steps like a



7.10 Stilling basins.



(a)

(b) Drop structure on an irrigation canal in Iraq

7.11 Drop structures.

staircase. The water flows gently and safely along the shallow reaches of channel and then drops to the next reach through a drop structure. Often a drop is made into a weir so that it can also be used for discharge measurement. It can also be fitted with gates so that it can be used for water level and discharge control. Drop structures are usually combined with stilling basins so that unwanted energy is got rid of effectively.

7.11 Siphons

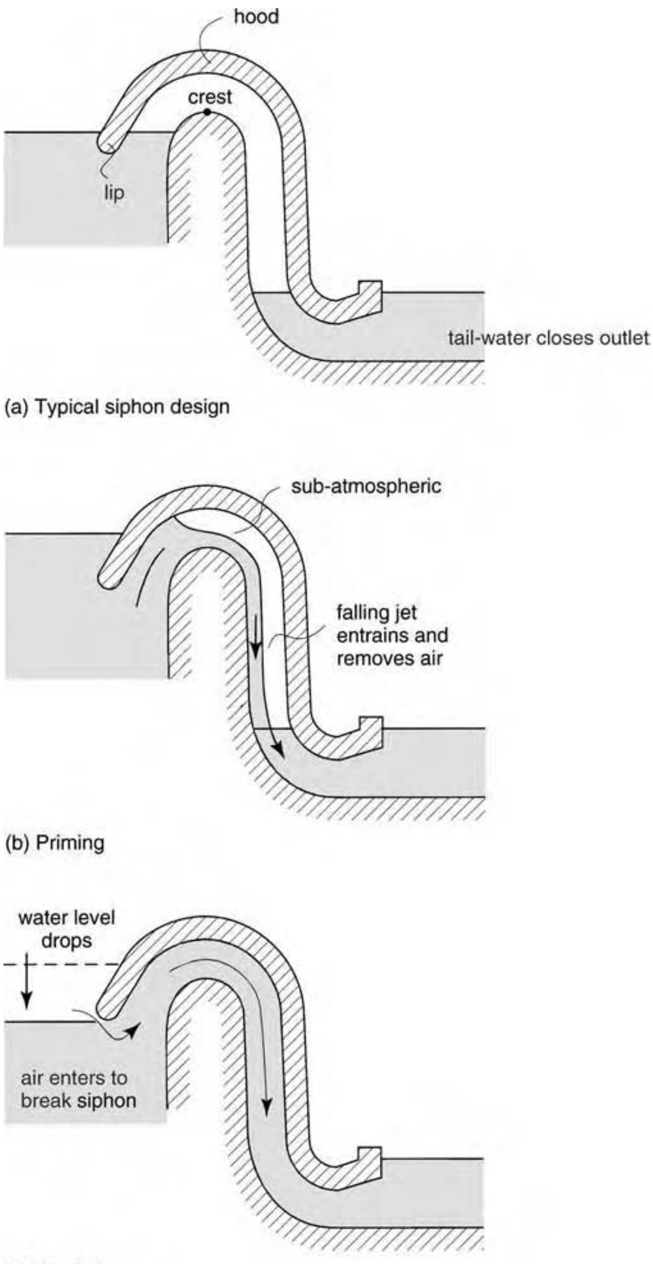
Siphons are hydraulic structures which have always fascinated engineers. Used long ago by ancient Greeks and Egyptians, they began to be used seriously in civil engineering in the mid-19th century as spillways on storage reservoirs. More recently, their special characteristics have been put to use in providing protection against sudden surges in hydropower intakes and in controlling water levels in rivers and canals subjected to flooding.

Although siphons have been successfully installed in many parts of the world, there is still a general lack of guidelines for designers. This is borne out by the wide variety of siphon shapes and sizes that have been used. Invariably design is based on intuition and experience of previous siphon structures, and few engineers would attempt to install such a structure without first carrying out a model study.

The principle of siphon operation is described in Section 4.7 in connection with pipe flow. In its very simplest form it is a pipe which rises above the hydraulic gradient over part of its length. The following siphon structures, although more sophisticated than a pipe, still follow this same principle.

7.11.1 Black-water siphons

These are the most common types of siphon; Figure 7.12 shows how one can be used as a spillway from a reservoir. It consists of an enclosed barrel which is sealed from the atmosphere by the upstream and downstream water levels. The lower part is shaped like a weir and the upper part forms the hood.



7.12 Black-water siphon.

Water starts to flow through a siphon when the upstream water level rises above the crest. As the flow plunges into the downstream water it entrains and removes air from inside the barrel. As the barrel is sealed, air cannot enter from outside and so the pressure gradually falls and this increases the flow rate until the barrel is running full of water. At this stage the siphon is said to be *primed* and flow is described as *black-water flow*. This is in contrast to the flow just before priming when there is a lot of air entrained and it has a white appearance. This is termed *white-water flow*.

Priming takes place rapidly until the discharge reaches the siphon's full capacity. This is determined by the size of the barrel and the difference in energy available across the siphon. The siphon eventually starts to draw down the upstream water level and this continues even when the water level falls below the crest. Only when it is drawn down to the lip on the hood on the inlet can air enter the barrel and break the siphonic action. Flow then stops rapidly and will only begin again when the siphon is reprimed.

Black-water siphons have several advantages over conventional spillways. There are no moving parts such as gates and so they do not need constant attention from operators. They respond automatically, and rapidly, to changes in flow and water levels and so floods which come unexpectedly in the night do not cause problems. Their compactness also means that they are very useful when crest lengths for conventional weir spillways are limited. But they are not without problems. The abruptness of priming produces a sudden rush of water which can cause problems downstream. They are also prone to *hunting* when the flow towards a siphon is less than the siphon capacity. They are continually priming and de-priming and this can cause surges downstream and vibration which is not good for the structure.

7.11.2 Air-regulated siphons

This type of siphon is a more recent development and offers many advantages over the more common black-water siphon. It automatically adjusts its discharge to match the approach flow and at the same time maintains a constant water level on the upstream side. This is achieved by the siphon passing a mixture of air and water.

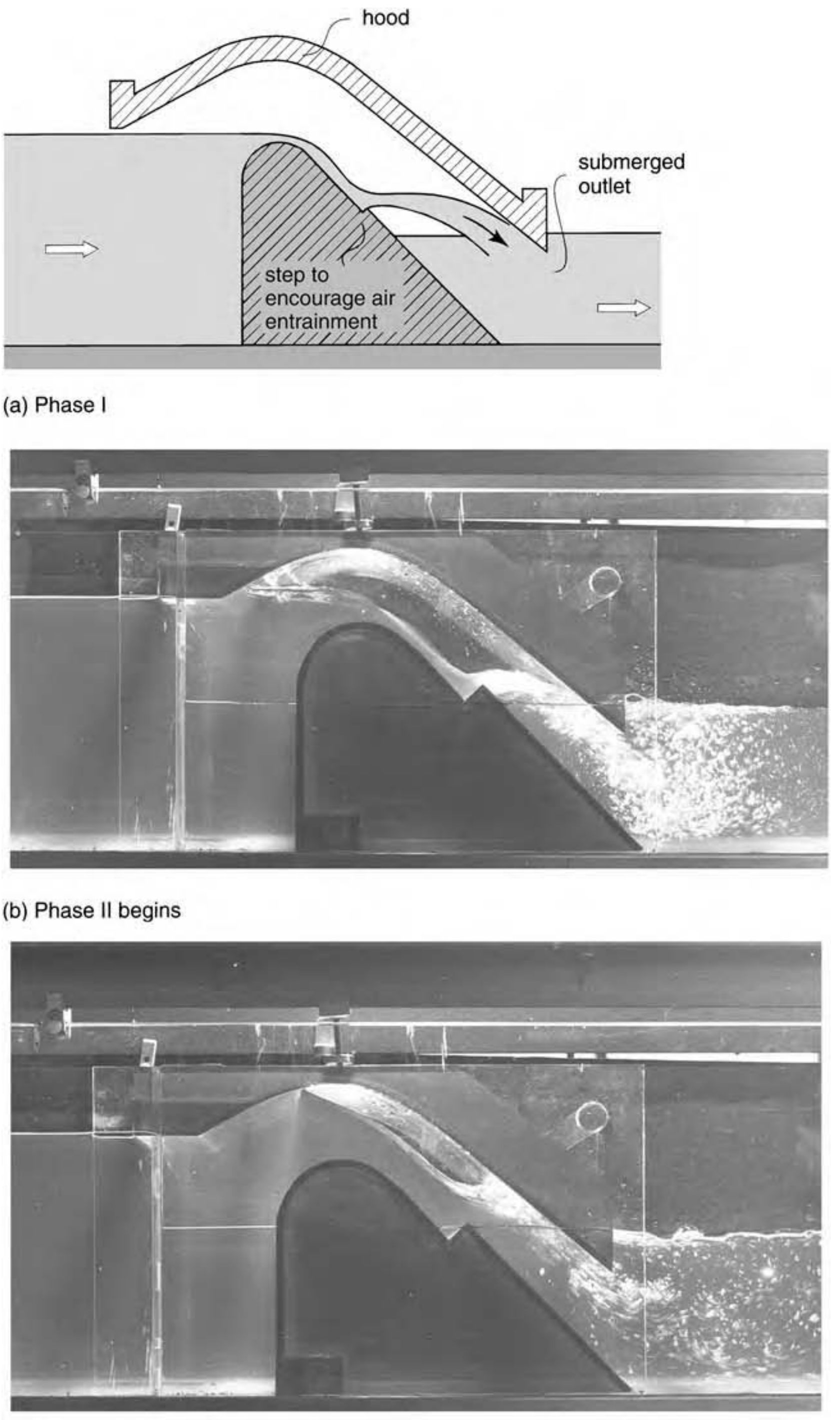
Air-regulated siphons are typically used for water level control in reservoirs and in rivers and canals. They will maintain a constant water level in a channel even though the discharge is changing. This would be an ideal structure for the wetland example discussed earlier.

Figure 7.13a shows a typical air-regulated siphon for a river. There needs to be sufficient energy available to entrain air in the barrel and take it out to prime the siphon. This one is designed to operate at very low heads of 1–2 m (difference between the upstream and downstream water levels). The siphon is shaped in many ways like a black-water siphon and relies on the barrel being enclosed and sealed by the upstream and downstream water levels. But the main difference is the inlet to the hood or upstream lip. This is set above the crest level whereas in a black-water siphon it is set below. A step is also included in the down-leg to encourage turbulence and air entrainment.

The operation of the siphon has several distinct phases (Figure 7.13 a–e):

Phase I – Free weir flow (Figure 7.13a) As the upstream water level starts to rise due to increased flow, water flows over the crest and plunges into the downstream pool. The structure behaves as a conventional free flowing weir. As the water level has not yet reached the upstream lip, air which is evacuated by the flow is immediately replaced and the pressure in the barrel remains atmospheric.

Phase II – Deflected flow (Figure 7.13b and c) As the flow increases, the upstream water level rises further and seals the barrel. The evacuation of air continues and a partial vacuum is created. This raises the head of water over the crest and so increases the discharge

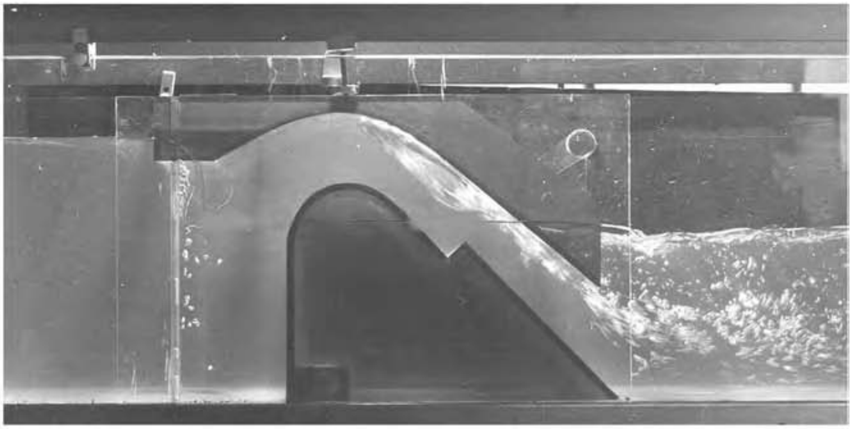


(a) Phase I

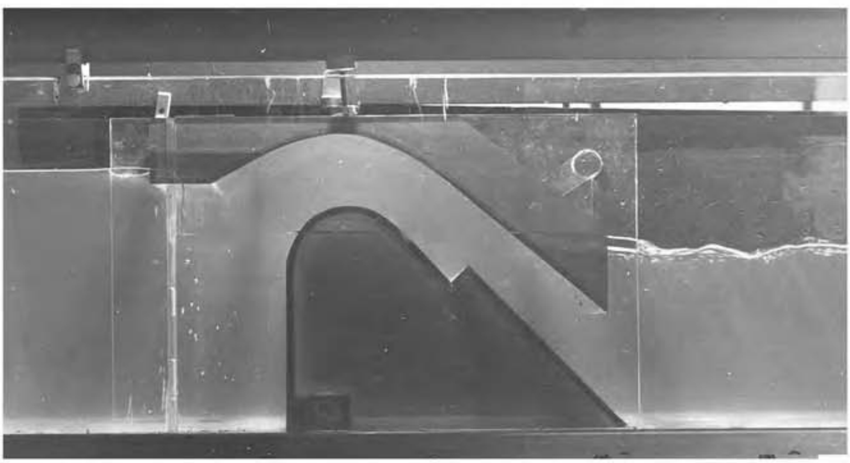
(b) Phase II begins

(c) Phase II

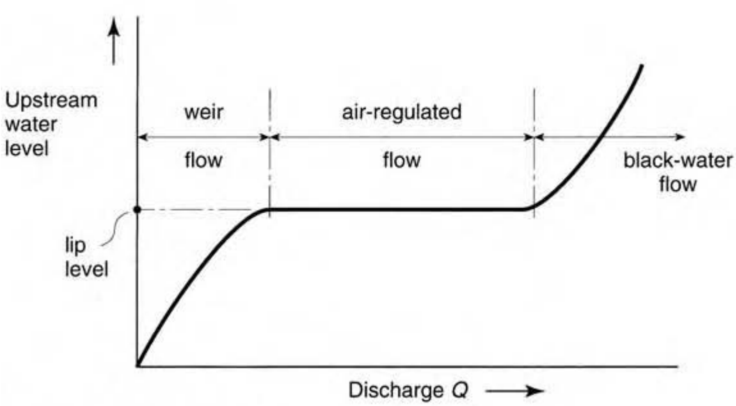
7.13 Air-regulated siphon.



(d) Phase III



(e) Phase IV



(f) Discharge characteristic

7.13 Continued.

through the siphon. A point is reached when the flow through the siphon exceeds the incoming flow. The water surface close to the lip is drawn down and air is sucked into the barrel to compensate for the evacuation taking place on the downstream side. This process is not cyclic but continuous. Both air and water are drawn continuously through the structure. In this manner the siphon adjusts rapidly and smoothly to the incoming flow and is said to be self-regulating. As the flow passes over the weir it is deflected by a step and springs clear of the structure. This encourages air entrainment and evacuation at low flows and it not intended to create an air seal as in the case of some black-water siphons.

Phase III – Air-partialised flow (Figure 7.13d) As the flow increases, the water level inside the hood rises to a point where the air pocket is completely swept out and the siphon barrel is occupied by a mixture of air and water. Changes in the discharge are now accommodated by variations in the quantity of air passing through the siphon and not by an increase in the effective head over the weir crest.

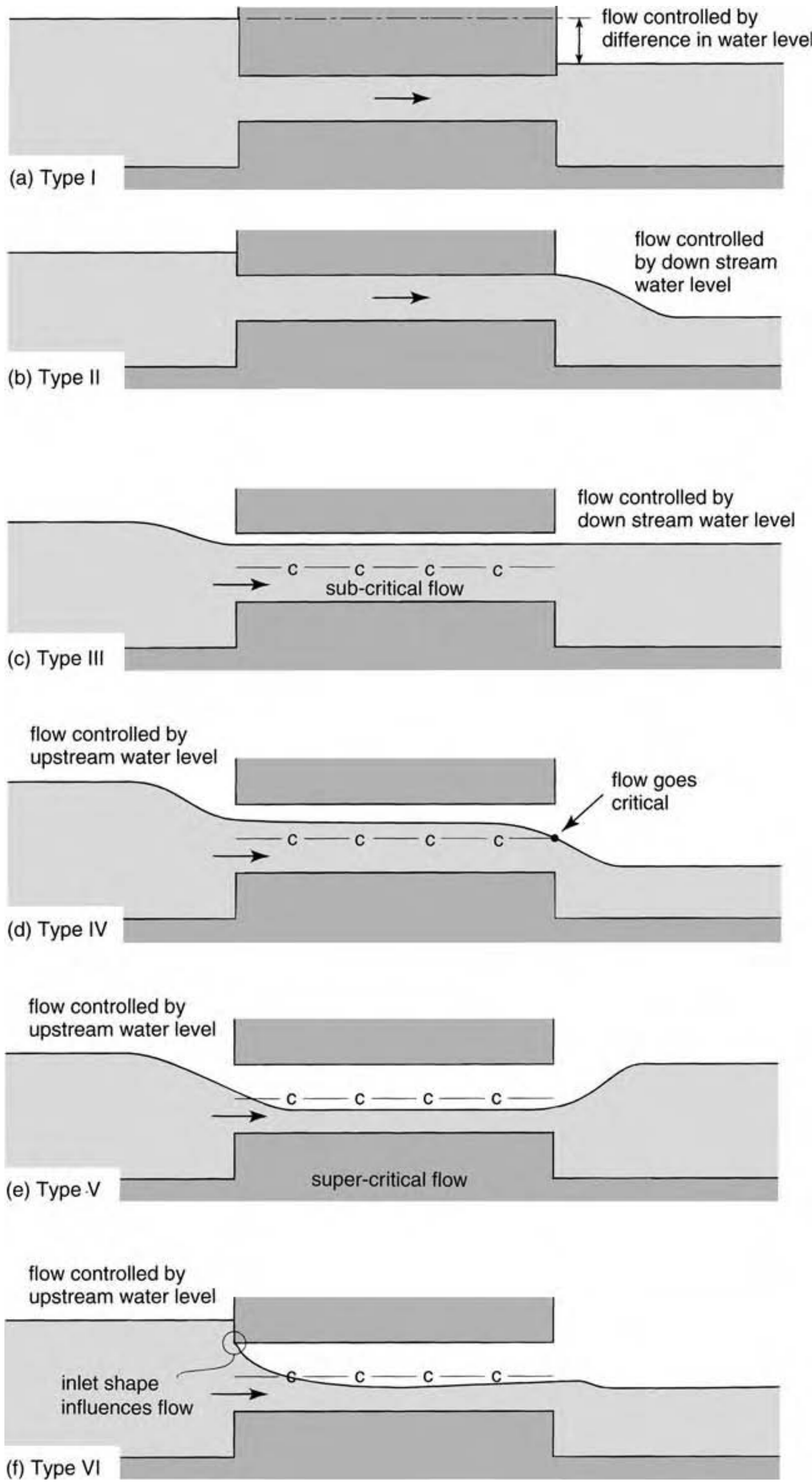
Phase IV – Black-water flow (Figure 7.13e) Increasing the flow beyond the air-partialised phase produces the more common black-water flow in which the barrel is completely filled with water. The discharge is now determined by the head across the siphon, that is, the difference between the upstream and downstream water levels.

The flow changes from one phase to another quite gradually and smoothly and there is no distinct or abrupt change over point. During phases II and III, the upstream water level remains relatively constant at a level close to that of the upstream lip. This feature makes this structure an excellent water level regulator. Only when the black-water phase is reached does an increase in discharge cause a significant rise in the upstream water level (Figure 7.13f).

7.12 Culverts

Culverts are very useful structures for taking water under roads and railways. They are circular or rectangular in shape and their size is chosen so that they are large enough to carry a given discharge, usually with minimum energy loss (Figure 7.14). They are important structures and can be as much as 15% of the cost of building a new road.

7.14 Culverts.



7.15 Culvert flow conditions.

Although they are very simple in appearance culverts can be quite complex hydraulically depending on how and where they are used. Sometimes they flow part full, like an open channel and other times they can flow full, like a pipe. Six different flow conditions are recognised depending on the size and shape of a culvert, its length and its position in relation to the upstream and downstream water level (Figure 7.15).

The simplest condition occurs when a culvert is set well below both the upstream and downstream water levels and it runs full of water (type I). It behaves like a pipe and the difference between the water levels is the energy available which determines the discharge. This full pipe flow can still occur even when the downstream water level falls below the culvert soffit (this is the roof of the culvert) (type II). The culvert is now behaving like an orifice and the flow is controlled by the upstream water level and the size of the culvert. In both cases the discharge is controlled by what is happening downstream. When the water level falls the discharge increases and when it rises the discharge decreases. If the flow in the channel does not change then any change in the downstream water level will have a corresponding effect on the upstream water level.

The four remaining flow conditions are for open channel flow. Three occur when both the upstream and downstream water levels are below the culvert soffit (type III, IV and V). The difference between III and IV is the slope of the culvert. III produces sub-critical flow and so the discharge is controlled by the downstream water level. When the downstream level rises the discharge reduces or the upstream level rises to accommodate the same flow. In IV, the flow is still sub-critical but the downstream water level is low and so the flow goes through the critical depth at the outlet. This means that the flow is controlled by the upstream level. Because the flow has gone through the critical depth any changes downstream do not affect the flow in the culvert or upstream. Condition V produces super-critical flow and so the culvert is again controlled by the upstream level. A rise in the downstream level will have no effect until it starts to drown the culvert.

The final condition occurs when the upstream water level is above the soffit, the downstream level is below and there is a sharp corner at the entrance causing the flow to separate (type IV). Open channel flow occurs in the culvert and is primarily due to the shape of the entrance. But if the entrance is rounded then this could change the flow to pipe flow. Clearly this would improve its discharge capacity.

Because of their complexity, and particularly the importance of the shape of the entrance, most culvert designs are based on model tests rather than on fundamental formulae. But this does not mean that every culvert must be tested in this way. There has already been extensive testing of culverts over many years and so designers look to the standard handbooks when deigning new culverts (see references for details).

7.13 Some examples to test your understanding

1 A broad-crested weir 3.5 m wide is to be constructed in a channel to measure a discharge of 4.3 m^3/s. When the normal depth of flow is 1.2 m, calculate the height of weir needed to measure this flow assuming that the flow must go critical on the weir crest (0.4 m).

2 A broad-crested weir is 5.0 m wide and 0.5 m high is used to measure a discharge of 7.5 m^3/s. Calculate the water depth upstream of the weir assuming that critical depth occurs on the weir and there are no energy losses (1.45 m).

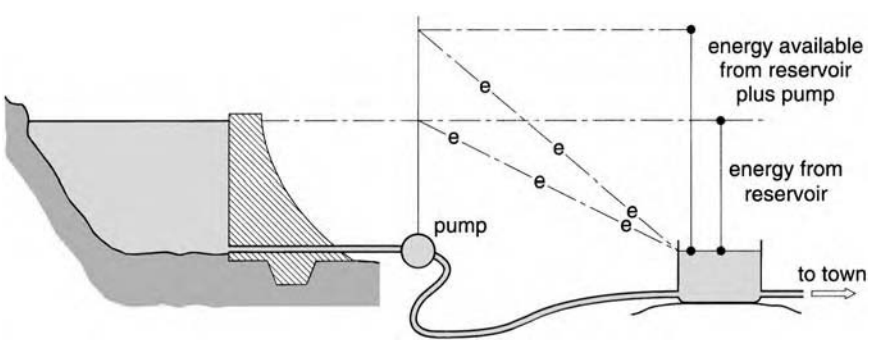
8 Pumps

8.1 Introduction

A few water supply systems have the benefit of a gravity supply but most require some kind of pumping. Pumps are a means of adding energy to water. They convert fuel energy, such as petrol or diesel, into useful water energy using combustion engines or electric motors. In the pipeline problem in Chapter 5 the energy to drive water along the pipeline to the town was obtained from a reservoir located high above the town. The energy line drawn from the reservoir to the town indicated the amount of energy available. Adding a pump to this system increases the available energy and raises the energy line so that the discharge from the reservoir to the town can be increased (Figure 8.1).

Pumps have been used for thousands of years. Early examples were largely small hand- or animal-powered pumps for lifting small quantities of water. It was not until the advent of the steam engine, only two centuries ago, that the larger rotating pumps were developed and became an important part of the study of hydraulics. Consequently there are two main types of pump:

- Positive displacement pumps – these are mainly small hand- and animal-powered pumps many of which are still used today;
- Roto-dynamic pumps – these are mainly the modern pumps driven by diesel or electric motors that are now in common use throughout the world.



8.1 Pumps add energy to pipe systems

Because of the similarities between pumps and turbines (used to generate energy) a small section is devoted to them at the end of this chapter.

8.2 Positive displacement pumps

Positive displacement pumps usually deliver small discharges over a wide range of pumping heads. Typical examples are hand-piston pumps, rotary pumps, air-lift pumps and Archimedean screws (Figure 8.2).

Hand-piston pumps are used extensively in developing countries for lifting groundwater for domestic water supplies. A pipe connects the pump to the water source – usually a well. At the end of this pipe is a non-return valve that only allows water to enter the pump and stops it from flowing back into the source. The pump itself comprises a piston and cylinder. These must have a very close fit so that when the piston is raised, it creates a vacuum in the cylinder and water is drawn up into the pump. When the piston is pushed down the water is pushed through a small valve in the piston to fill up the space above it. As the piston is raised again, it lifts the water above it so it pores out through the spout of the pump and into a tank or other water collecting device. The procedure is then repeated. The discharge from the pump depends on the energy available from those working the pump handle. You basically get out what you put in. The height to which water can be pumped in this way is fixed only by the strength of those pumping and the pump seals, which will start to leak when the pressure gets too high.

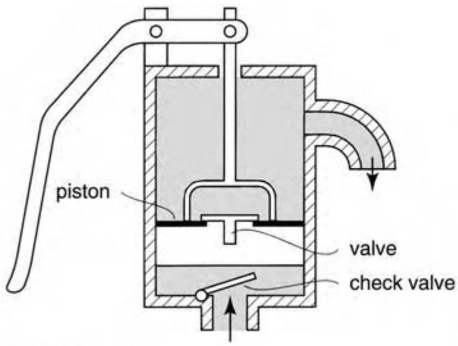
The rotary pump contains two gears that mesh together as they rotate in opposite directions. The liquid becomes trapped between the gears and is forced into the delivery pipe.

The air-lift pump uses an air compressor to force air down a pipe into the inlet of the water pipe. The mixture of air and water, which is less dense than the surrounding water in the well, then rises up above ground level. This pump is not very efficient but it will pump water that has sand and grit in it which would normally damage other pumps.

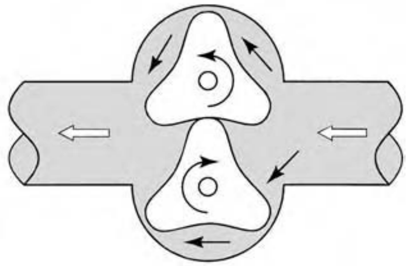
The Archimedean screw pump has been used for thousands of years and is still used today for pumping irrigation water in Egypt. It comprises a helical screw inside a casing. Water is lifted by turning the screw by hand. Some modern pumping stations also use this idea but the screws are much larger and are driven by a diesel or electric power unit.

The treadle pump is another example of positive displacement pump (Figure 8.3). It is growing in popularity in many developing countries because it enables poor farmers, who cannot afford a motor-driven pump, to use their human power to pump the large volumes of water needed for irrigating crops. It was first developed in Bangladesh in the early 1980s for irrigating flooded rice and it is now estimated that some two million pumps are in use throughout Asia. Their popularity has spread in recent years to sub-Saharan Africa. The designers of the treadle pump have skilfully adapted the principle of the hand-piston pump so that greater volumes of water can be lifted. Two pistons and cylinders are used to raise water instead of one. But the most important innovation was the change in driving power from arms and hands to legs and feet. Leg muscles are much more powerful than arm muscles and so they are capable of lifting much more water. The two cylinders 0.75 to 110 mm in diameter, are located side-by-side and a chain or rope, which passes over a pulley, connects the two pistons together so that when one piston is being pushed down, the other one is being raised. Each piston is connected to a treadle on which the operator stands and pushes them up and down in a rhythmic motion – like pushing peddles on a bicycle. This rhythmic method of driving the pump seems to have gained wide acceptance among farmers in the developing world.

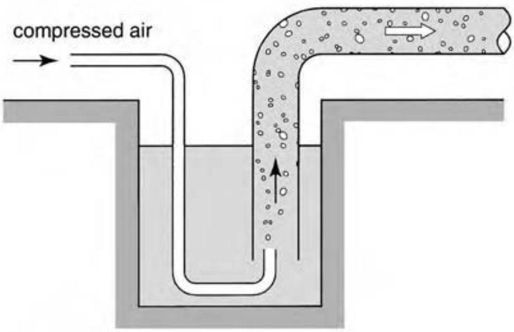
Although the pump is popular in several developing countries some exaggerated claims are often made about its physical performance, usually by those who have little knowledge of how



(a) Piston pump



(b) Rotary pump

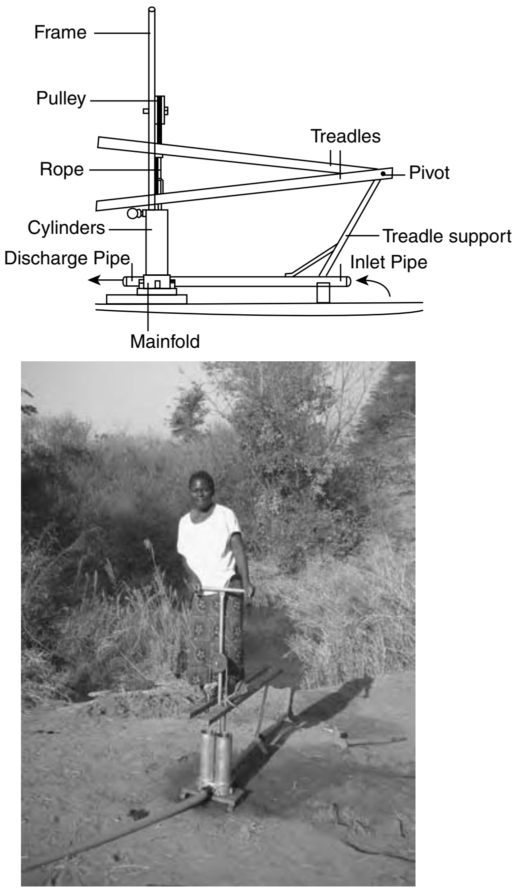


(c) Air-lift pump

(d) Archimedian screw pump in Egypt

(e) Hand pump in village in the Gambia

8.2 Positive displacement pumps.



8.3 Treadle pump.

the physical world works. Some claim it will deliver pressures up to 15 m and at the same time deliver discharges of 1–2 l/s and more as if it contains magical properties. Applying some simple hydraulic principles to this pump can dispel some of this magic and show just what it is capable of producing for a small farmer in a developing country (see box in Section 8.5). The reality is more like 0.6 l/s to 1.2 l/s with a head of 1–2 m. Like most things in life you can only get out of this pump what you are prepared to put into it in terms of power and energy – I am afraid there are no free lunches even in pumping.

Typically, one fit, male operator could irrigate up to 0.25 ha which can dramatically transform a family's livelihood from one of subsistence to one where there is a cash income from irrigated produce.

One final positive displacement pump worth mentioning is the heart which pushes blood around your body. The muscles around the heart contract and push blood into your arteries, which is your body's pipe system. It must create enough pressure to force blood to flow around your entire body with enough flow to carry all the nutrients your body needs. Problems of high blood pressure (known as hypertension) occur when the arteries (the pipe system) become restricted in some way. Your heart then increases the pressure to maintain circulation. Applying basic hydraulics to this – fatty deposits in your pipes (or arteries) increase pipe friction and can also reduce pipe diameters. Muscles surrounding the pipes are also known to contract and reduce pipe diameters. All these physical factors can push up the pumping pressure. Also the taller you are the more pressure your heart needs to produce to get blood to the top of your head and lift it back up from your feet to your heart. Flow in arteries is mostly laminar, although in the larger main artery near the heart it can be turbulent. This implies that friction loss in arteries is a function of viscosity rather than roughness and so anything that 'thins' blood (reduces its viscosity) should ease the flow. Some drugs are known to do this as well as others that stop blood clots and pipe blockages. Blood pressure-reducing drugs work in many different ways. Some slow down the heart rate, and others increase pipe diameters by reducing the squeezing effects of the muscles around them. But the process is not simply a physical one. It is a complex mixture of physical and biological processes and there is still a great deal not yet known about how these processes combine and work together. But basic hydraulics undoubtedly plays a part.

8.3 Roto-dynamic pumps

All roto-dynamic pumps rely on spinning an impeller or rotor for their pumping action. There are three main types that are described by the way in which water flows through them:

- centrifugal pumps
- mixed flow pumps
- axial flow pumps.

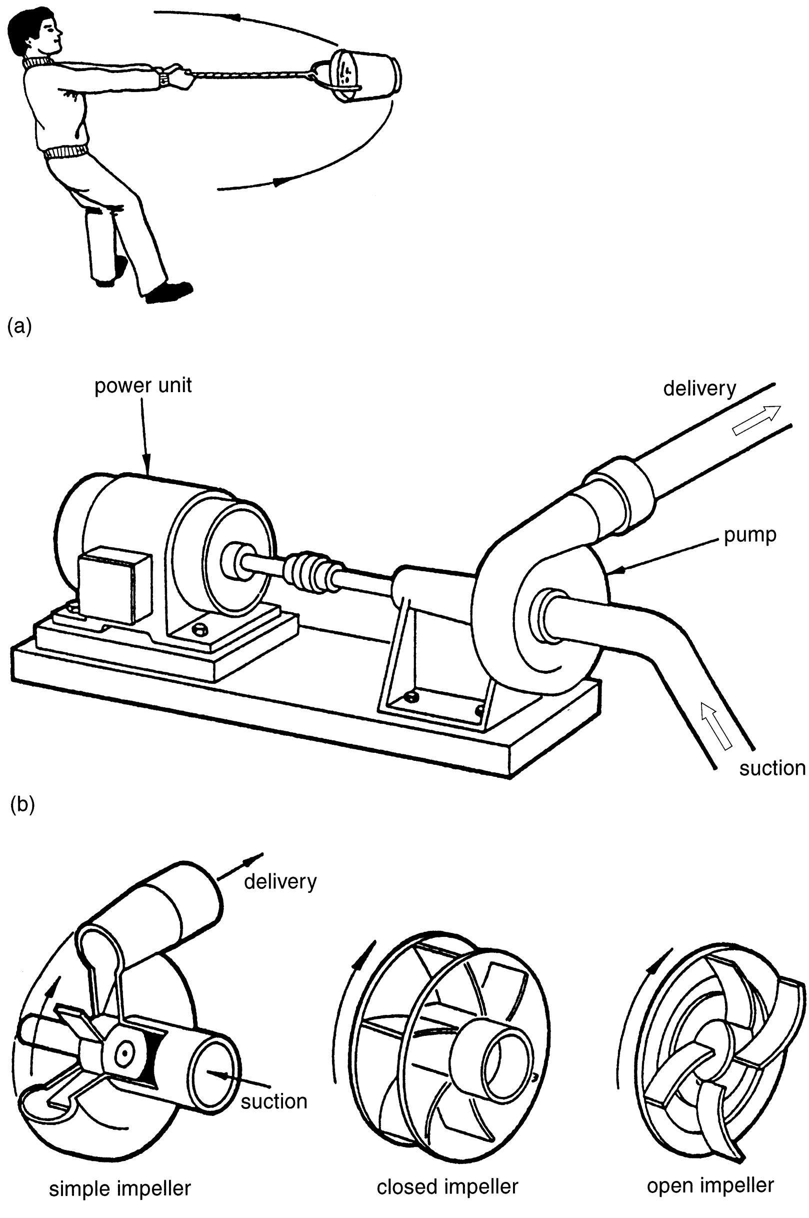
8.3.1 Centrifugal pumps

Centrifugal pumps are the most widely used of all the roto-dynamic pumps. The discharges and pressures they produce are ideally suited to water supply and irrigation schemes.

To understand how centrifugal pumps work, consider first how centrifugal forces occur. Most people will, at some time, have spun a bucket of water around at arm's length and observed that water stays in the bucket even when it is upside down (Figure 8.4a). Water is held in the bucket by the centrifugal force created by spinning the bucket. The faster the bucket is spun the tighter the water is held. Centrifugal pumps make use of this idea (Figure 8.4b). The bucket is replaced by an *impeller* which spins at high speed inside a spiral casing. Water is drawn into the pump from the source of supply through a short length of pipe called the *suction*. As the impeller spins, water is thrown outwards and is collected by a spiral-shaped pump casing and guided towards the outlet. This is the *delivery* side of the pump.

The design of the impeller and the casing is important for efficient pump performance. As water enters the pump through the suction pipe the opening is small so velocity is high and, as a consequence of this, the pressure is low (think about the energy equation). As the flow moves through the pump and up between the blades of the impeller the flow gradually expands. This causes the velocity to fall and the pressure to rise again. It is important at this stage to recover as much pressure energy as possible and not to lose energy through friction and turbulence. This is achieved by carefully shaping the impeller blades and the spiral pump casing so that the movement of water is as 'streamline' as possible. Any sudden change in change in shape would create a great deal of turbulence, which would dissipate the water energy instead of increasing the pressure.

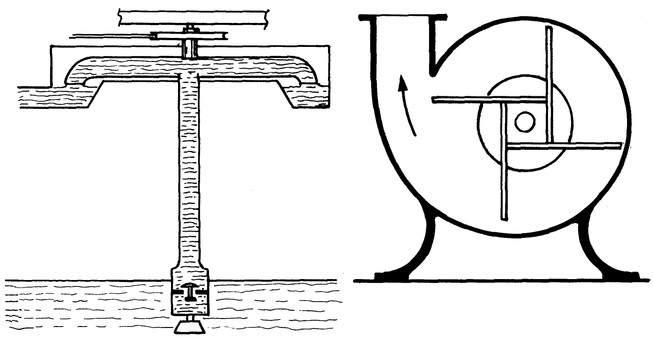
Some pumps have very simple impellers with straight blades (Figure 8.4c), but these create a lot of turbulence in the flow and so pressure recovery is not so good. Energy losses tend to be high and the efficiency is poor. From a practical point of view pumps like this are cheap to make and are used where efficiency is not so important such as in domestic washing machines.



8.4 Centrifugal pumps.

Larger pumps need more careful design and have curved vanes so that the water enters and leaves the impeller smoothly. This ensures that energy losses are kept to a minimum and a high level of efficiency of energy and power use can be achieved. Most impellers have side plates and are called *closed impellers*. When there is debris in the water *open impellers* are used to reduce the risk of blockage.

Centrifugal pumps are very versatile and can be used for a wide variety of applications. They can deliver water at low heads of just a few metres up to 100 m or more. The discharge range is



8.5 Early examples of roto-dynamic pumps.

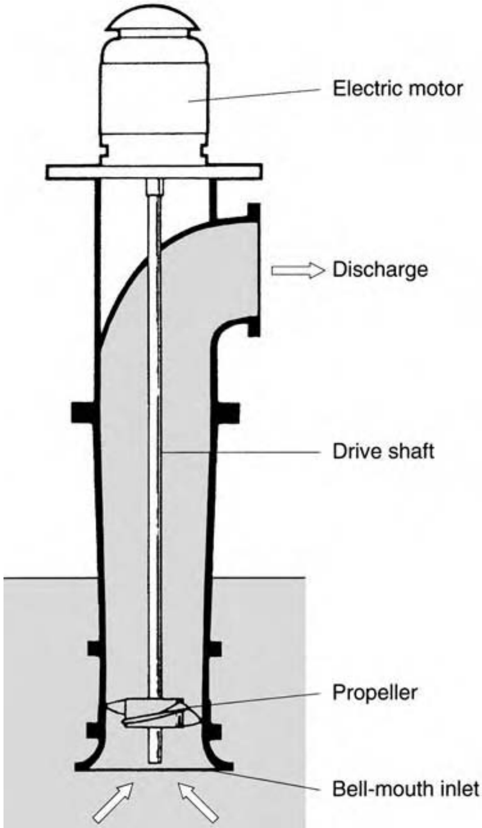
also high, from a few litres per second up to several cubic metres per second. Higher discharges and pressures are achieved by running several pumps together (see Section 8.10) or by using a multi-stage pump. This comprises several impellers on a shaft, driven by the same motor. Water is fed from the outlet of one stage into the inlet of the next impeller, increasing the pressure at each stage. Multi-stage pumps are commonly used in boreholes for lifting groundwater.

From a historical perspective one of the earliest centrifugal pumps was developed in the late 18th century and used a spinning pipe that discharged into a semi-circular collecting channel (Figure 8.5). Once the pipe was filled with water (primed) and was spinning, water was continually drawn up the tube and discharged into the collecting channel. Another, the Massachusetts pump built in 1818, comprised a set of simple straight blades mounted on a horizontal shaft surrounded by a collecting case that directed the velocity of the water up the discharge pipe. This looks similar in some respects to modern centrifugal pumps but it differs in one important aspect. In this and other earlier pumps the water leaves the pump impeller at high speed and friction is left to do the job of slowing down the water once it leaves the pump. This means that they were not very efficient at transferring energy and power to the water and so their performance was rather poor.

It was the application of the energy equation to pumps that revolutionised pump design and performance. This was the realisation that when water is flowing in a pipe, pressure can be converted to velocity and vice-versa. It is well known that pressure is needed to produce a high velocity jet of water but what was not so obvious was that the pressure can be recovered if the water jet is slowed down in a smooth manner. This was the understanding that was missing in the early pumps. Designers knew that the spinning action would speed up the water but they did not realise that pressure could be recovered again by smoothly slowing the water down. It is the recovery of pressure energy that marks out the design of modern pumps.

8.3.2 Axial flow pumps

Axial flow pumps consist of a propeller housed inside a tube that acts as a discharge pipe (Figure 8.6). The power unit turns the propeller by means of a long shaft running down the middle of the pipe and this lifts the water up the pipe. These pumps are very efficient at lifting large volumes of water at low pressure. They are ideally suited for lifting water from rivers or lakes into canals for irrigation and for land drainage where large volumes of water need to be lifted through a few metres. However, they tend to be very expensive because of the high cost of materials, particularly the drive shaft and bearings to support the shafted propeller. For this reason they tend only to be used for large pumping works.



8.6 Axial flow pump.

8.3.3 Mixed flow pumps

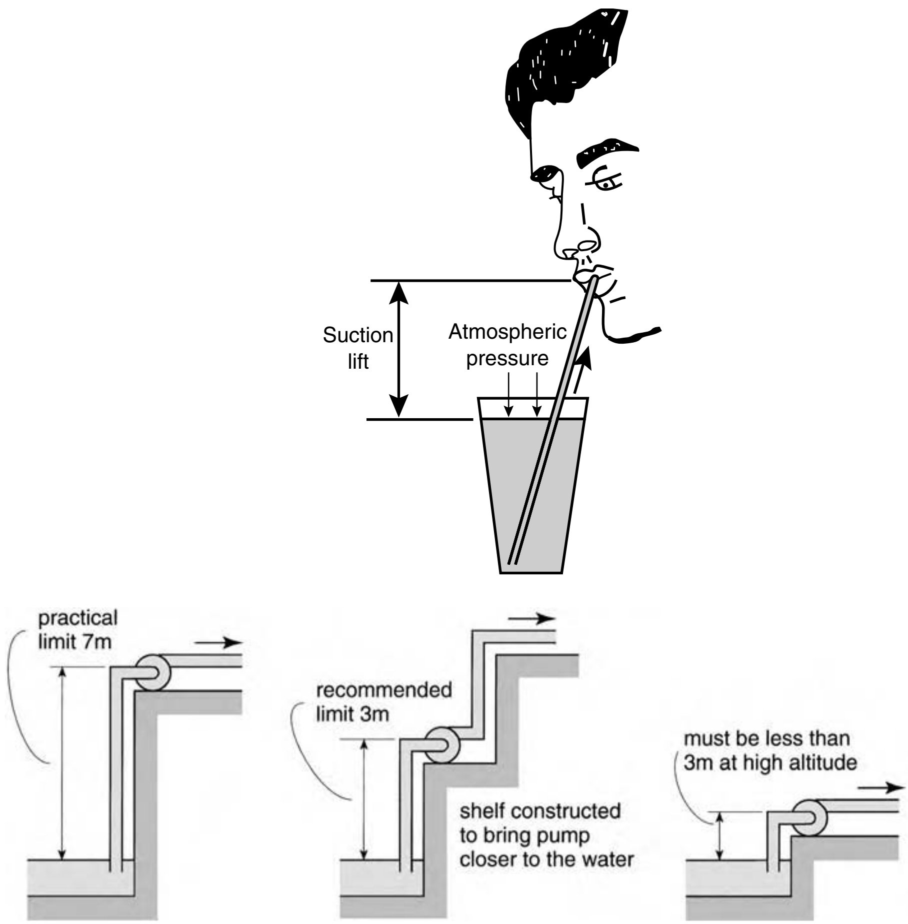
These pumps are a mixture of axial flow and centrifugal pumps and so combine the best features of both pump types. Mixed flow pumps are more efficient at pumping larger quantities of water than centrifugal pumps and are more efficient at pumping to higher pressures than axial flow pumps.

8.4 Pumping pressure

8.4.1 Suction lift

Pumps are usually located above the water source for the convenience of the users and this leads to the idea that pumps must 'suck' up water from the source before lifting it to some higher level. But this idea and the word 'sucking' can lead to misunderstandings about how pumps work (Figure 8.7).

'Sucking' water up a drinking straw is very similar to the way a pump draws water from a source. However, you do not actually suck up the water, you suck out the air from the straw and create a vacuum. Atmospheric pressure does the rest. It pushes down on the water surface in the glass and forces water up the straw to fill the vacuum. So atmospheric pressure provides the driving force but it also puts a limit on how high water can be lifted in this way. It does not depend on the ability of the sucker. At sea level, atmospheric pressure is approximately 10 m



8.7 Pump suction.

head of water and so if you were relying on a straw 10.1 m long for your water needs you would surely die of thirst! Even a straw 9 m long would cause you a lot of problems. It would be very difficult to maintain the vacuum in the straw and you would probably spend most of the day taking the small trickle of water that emerged at the top of the straw just to stay alive. The shorter the straw however, the easier drinking becomes and the more water you get.

This same principle applies to pumps. Ideally it should be possible to draw water from a source up to 10 m. But because of friction losses in the pipework an upper limit is usually set at 7 m. Even at this level there can be problems in keeping out air and maintaining a vacuum and so a more practical limit is 3 m. When water is more than 3 m below the pump a shelf can be excavated below ground level to bring the pump closer to the water. For pumps operating at high altitudes in mountainous regions where atmospheric pressure is less than 10 m the practical limit of 3 m will need to be lowered further for satisfactory pump operation.

The difference in height between the water source (usually referred to as the sump) and the pump is called the *suction lift*. It is an unfortunate name as it clearly does not describe what actually happens in practice. However, it is in common use and so we are stuck with.

Not all pumps suffer from suction problems. Some pumps are designed to work below the water level in the sump and are called *submersible pumps*. These are often used for deep boreholes and are driven by an electric motor, which is also submerged, connected directly to the pump drive shaft. The motor is well sealed from the water but being submerged helps to keep the motor from overheating.

Excessive suction lift can affect pump performance and this is demonstrated by the results of a test performed on a pump for a small rural water supply scheme. The pump delivered 6.5 l/s with a 3 m suction lift. But when the suction lift was increased to 8 m the discharge dropped to 1.2 l/s – a loss in flow of 5.3 l/s – only 15% of the original discharge! So the general rule is to keep the suction lift as short as possible.

Centrifugal pumps will not normally suck out the air from the suction and will only work when the pump and the suction pipe is full of water. If pumps are located above the sump then it will be necessary to fill the pump and the suction pipe with water before the pump is started up. This process is called *priming*. A small hand-pump, located on the pump casing is used to evacuate the air. When pumps are located below the sump they will naturally fill with water under the influence of gravity and so no additional priming is required.

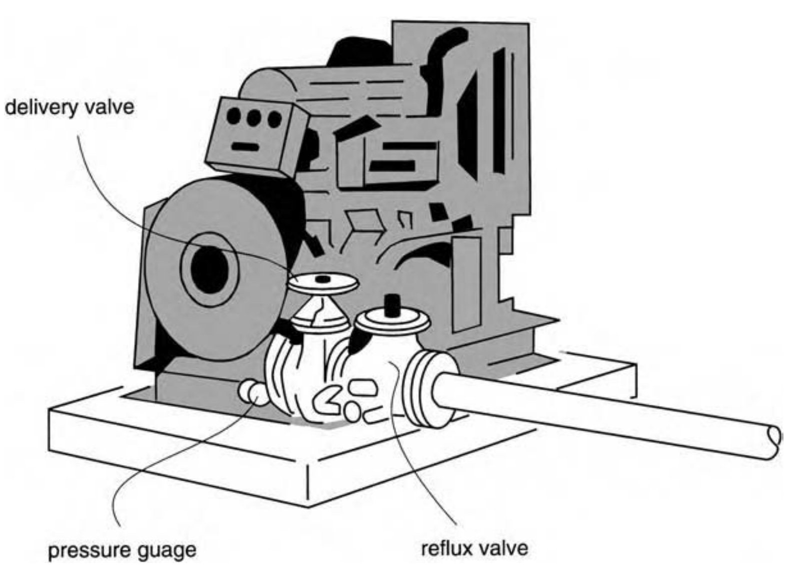
For axial flow pumps, the impeller is best located below the water level in the sump and so no priming is needed.

8.4.2 Delivery

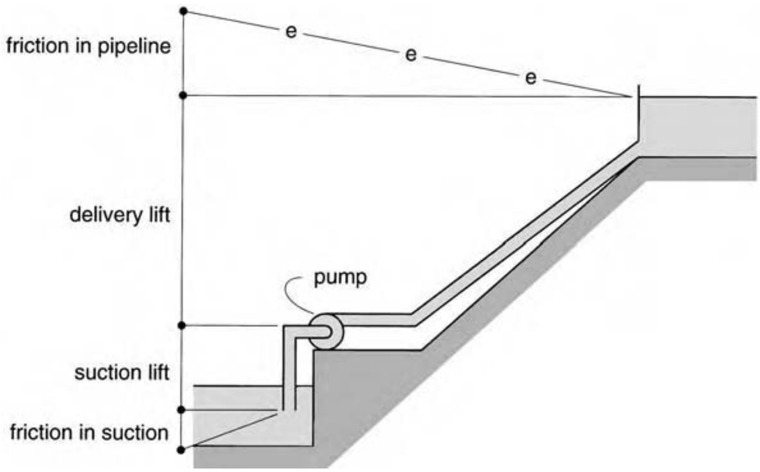
The delivery lift is the pressure created on the delivery side of the pump. The delivery side comprises pipes and fittings that connect the pump to the main pipe system and provide some control over the pressure and discharge.

For centrifugal and mixed flow pumps, a sluice valve is connected to the pump outlet to assist in controlling pressure and discharge (Figure 8.8). It is closed before starting so that the pump can be primed. Once the pump is running it is slowly opened to deliver the flow.

A reflux valve is connected downstream of the delivery valve. This allows water to flow one way only – out of the pump and into the pipeline. When a pump stops, water can flow back towards the pump and cause a rapid pressure rise which can seriously damage both the pump and the pipeline (Section 8.14). The reflux valve prevents the return flow from reaching the pump. Some reflux valves have a small by-pass valve fitted which allows water stored in the pipe to pass around the valve and to prime the pump.



8.8 Delivery side of centrifugal pump.



8.9 Pumping head.

8.4.3 Pumping head

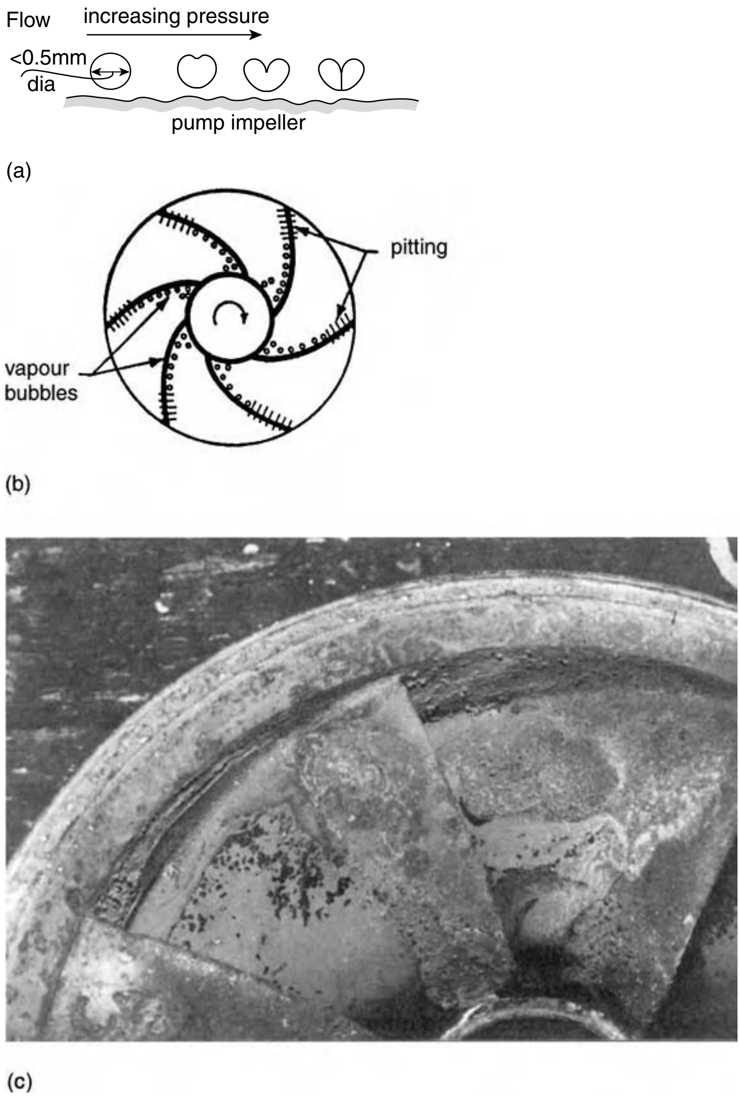
Pumping head should not be confused with the delivery head as is often the case. Pumping water requires energy not only to deliver water but also to draw it up from its source. So the pumping head is the sum of the delivery lift and suction lift and the friction losses in both suction and delivery pipes (Figure 8.9).

pumping head (m) = suction lift (m)
+ friction loss in suction (m)
+ delivery lift (m)
+ friction loss in pipeline (m)

The suction lift refers to the elevation change between the sump water level and the pump and the delivery lift is the elevation change between the pump and the point of supply, in this case it is the reservoir water level. Both are fixed values for a given installation. The friction losses in the sump and the delivery will vary depending on the discharge. There are also minor losses at the inlet and outlet to a pipeline but these tend to be very small in comparison to the main lift and other friction losses. To allow for all their minor losses it is common practice to add 10% to the pumping heads rather than try and work them out in detail.

So if the suction lift is 4 m and the pump then delivers 7 m head the pumping head would be 11 m. This represents the total height through which the water must be lifted from source to delivery point. If 11 m was the maximum that a pump can deliver then any change in the suction lift would affect the delivery lift. For example, if the suction lift increased to 6 m then the delivery lift would reduce to 5 m resulting in the same overall total pumping head of 11 m. So just quoting delivery lift without any reference to suction lift does not provide enough information about what a pump can do in pressure terms.

In many pumping installations, the pumping head also includes any losses in head resulting from friction in the suction pipes and the losses as the water flows through filters and valves and also friction and fittings losses on the delivery side.



8.10 Cavitation. (a) How cavitation bubbles collapse. (b) and (c) Cavitation damage to a pump.

8.4.4 Cavitation

A particular problem associated with excessive suction lift is a phenomenon known as *cavitation* which can not only damage the pump but also reduce its operating efficiency (Section 3.9.2). When water enters a pump near the centre of the impeller the water velocity is high and so the pressure is very low. This is exacerbated when the suction lift is high. In some cases the pressure is so low that it reaches the vapour pressure of water (approx. 0.5 m absolute) and cavities (similar to bubbles) begin to form in the flow. The cavities are very small – less than 0.5 mm diameter – but there are usually so many of them that the water looks milky in appearance. This must not be confused with air entrainment which looks similar. Rather they are 'holes' in the water that contain only water vapour. The problems occur when the cavities move through the pump into an area of higher pressure where they become unstable and begin to collapse (Figure 8.10a).

A small needle jet of water rushes across the cavity with such force that if the cavity is close to the pump impeller or casing then it can start to damage them. Each cavity collapse is not significant on its own but when many thousands of them continually collapse close to the metal surfaces then damage begins to grow. In some cases it has been known to completely wear away the impeller blades (Figure 8.10b).

Most pumps in fact cavitate but not all cavitation causes damage. Pump designers go to great lengths to ensure that the cavities collapse in the main flow, well away from the impeller blades. If this is not possible then stainless steel impellers are used as this is one of the few materials which can resist cavitation damage. But this is a very expensive option and not one to take up lightly.

Cavitation not only erodes the surfaces in contact with water but it can also be very noisy. It also causes vibration and can reduce pumping efficiency. If you have an opportunity to visit a water pumping station then place your ear against the pump casing and you will hear the cavitation. Above all the noise of the pump engines it should be possible to hear a sharp crackling sound. This is the sound of the cavities collapsing as the pressure rises in the pump. The level of the noise produced by the cavitation is a measure of how badly the pump is cavitating.

Cavitation also occurs in turbine runners which are discussed briefly later in this chapter. High velocities at the turbine inlet produce cavities which then collapse close to the runner blades near the exit.

A concept sometimes used for limiting suction lift and so avoiding cavitation is the Net Positive Suction Head (NPSH). Manufacturers normally specify a value of NPSH for each discharge to ensure a pump operates satisfactorily.

The NPSH for an installation can be calculated as follows:

$$\text{NPSH} = P_a - H_s - V_p - h_f$$

where P_a is atmospheric pressure (m); H_s is suction head (m); V_p is the vapour pressure of water (approx. 0.5 m absolute); h_f is the head loss in the suction pipe (approx. 0.75 m).

The value of NPSH calculated using this formula must be greater than that quoted by the pump manufacturer for the pump to operate satisfactorily.

8.5 Energy for pumping

Energy is needed to pump water. The amount required depends on both the volume of water pumped and the height to which it is lifted. It can be calculated using the following formula:

$$\text{water energy (kWh)} = \frac{\text{volume (m}^3) \times \text{head (m)}}{367}$$

A derivation of this formula is shown in the box for the more inquisitive reader.

DERIVATION: A FORMULA TO CALCULATE ENERGY FOR PUMPING

Derive a formula to calculate the amount of energy needed to lift a volume of water V m^3 to a height of H m.

Start with the basic equation for calculating energy from Section 1.10:

water energy = work done

So:

water energy (Nm) = force (N) × distance (m)

We now need to establish what the values of force and distance mean in terms of pumping water. Look first at force. This is the total weight of water to be lifted. Usually the volume is known and so we use Newton's second law to determine the weight:

force = mass (kg) × acceleration (m/s^2)

In this case acceleration is due to gravity and so:

mass of water = volume (m^3) × density (kg/m^3)

So:

force = volume (m^3) × density (kg/m^3) × gravity constant (m/s^2)

Now density ρ = 1000 kg/m^3 and the gravity constant g = 9.81 m/s^2 so:

force = volume (m^3) × 1000 × 9.81
= volume (m^3) × 9810

The 'distance' part of the energy equation is the pumping head and so:

distance = pumping head (m)

Putting the values for force and distance in the energy equation:

water energy (Nm) = volume (m^3) × 9810 × head (m)

But this is in Nm. We need to convert this to more useful and practical units of energy like kWh so:

1 kWh = 3 600 000 Nm (Section 1.10.2)

So:

$$\text{water energy (kWh)} = \frac{\text{volume (m}^3\text{)} \times 9810 \times \text{head (m)}}{3\,600\,000}$$

This simplifies to the formula which is most often used:

$$\text{water energy (kWh)} = \frac{\text{volume (m}^3\text{)} \times \text{head (m)}}{367}$$

An example of how to use this formula for calculating energy for pumping is shown in the next box.

8.6 Power for pumping

Remember from Chapter 1 that the terms power and energy are often used to mean the same thing. But power is the rate of energy use. So as energy is determined by volume and head, power is determined by discharge and head. Power is usually measured in Watts (W) but this is a relatively small amount of power and so it is more common to speak of pumping power in terms of kilo-Watts (kW).

One way of calculating power is to divide energy by the time taken to use the energy. So:

$$\text{water power (kW)} = \frac{\text{energy (kWh)}}{\text{time (h)}}$$

But another way of calculating power is to use the formula:

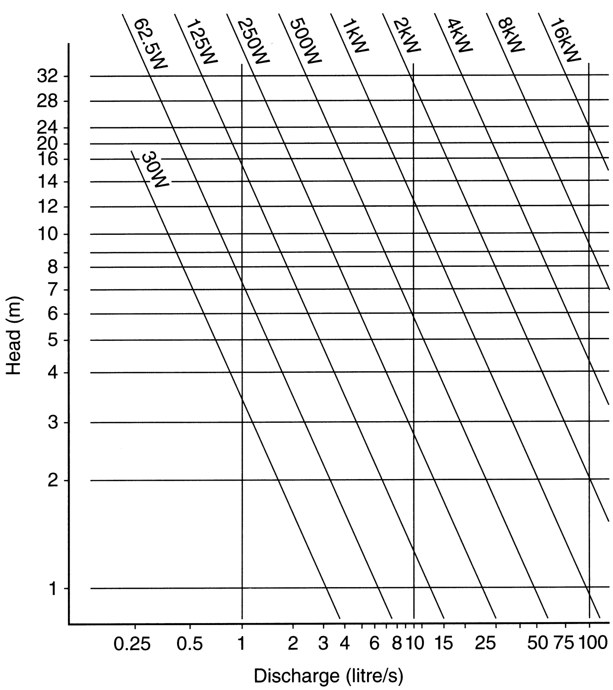
$$\text{water power (kW)} = 9.81 \times \text{discharge (m}^3\text{/s)} \times \text{pumping head (m)}$$
$$P = 9.81\ QH$$

A graph of water power and discharge for a wide range of pumping heads is a convenient way of showing the range of this formula (Figure 8.11).

Manufacturers often quote discharges in m^3/h rather than in m^3/s. In this case the formula becomes:

$$\text{water power (kW)} = \frac{\text{discharge (m}^3\text{/h} \times \text{head (m)}}{367}$$

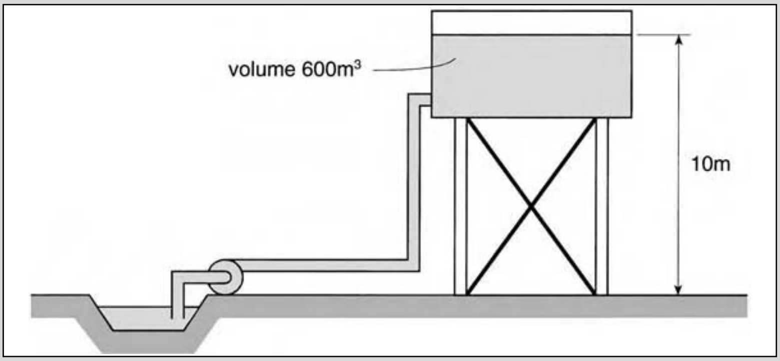
The derivation of this formula is similar to that for water energy but discharge is used instead of volume. Try to work this one out for yourself.



8.11 Power requirements for pumps.

EXAMPLE: CALCULATING THE ENERGY AND POWER REQUIRED TO FILL A TANK WITH WATER

A 600 m^3 tank 10 m high is filled with water each day. Calculate the amount of energy required to fill the tank. If the pump runs for 6 hours each day calculate the power required.



8.12 Calculating the energy and power required to fill a tank with water.

First calculate the amount of energy needed each day using the energy formula:

$$\text{energy (kWh)} = \frac{\text{volume (m}^3) \times \text{head (m)}}{367}$$

Put in values for volume and head:

$$\text{energy (kWh)} = \frac{600 \times 10}{367}$$
$$= 16.3 \text{ kWh}$$

This is the energy required each day to fill the tank. Notice how the time period over which the energy is used needs to be specified.
Now calculate the water power needed. First calculate the discharge:

$$\text{discharge (m}^3\text{/h)} = \frac{\text{volume (m}^3)}{\text{time (h)}}$$
$$= \frac{600}{6} = 100 \text{ m}^3\text{/h}$$

Now calculate power:

$$\text{power (kW)} = \frac{\text{discharge (m}^3\text{/h)} \times \text{head (m)}}{367}$$
$$= \frac{100 \times 10}{367} = 2.7 \text{ kW}$$

The energy can also be calculated from power by multiplying power by the time over which the power is used. In this case the time is 6 hours.

$$\begin{aligned} \text{energy (kWh)} &= \text{water power (kW) x operating time (h)} \\ &= 2.7 \times 6 \\ &= 16.3 \text{ kWh} \end{aligned}$$

Note the two approaches produce the same answer.

8.6.1 Efficiency

In the example in the box the 'water energy' and 'water power' are only part of what is needed to actually fill the water tank. Losses occur in the power unit that drives the pump and in the pump itself and these need to be taken into account when the total power needed for pumping is determined. Energy and power losses occur when fuel energy is converted into useful water energy and through turbulence and friction in the pump and fittings. Much of this is dissipated as heat energy. The losses that occur are expressed as an *efficiency*, which can be looked at from both a power and an energy point of view.

Energy and power efficiencies are often assumed to be the same. In practice this may not be the case. A seasonal assessment of energy use efficiency may not always give the same value as power use efficiency measured only one or two times during a lengthy period of operation.

Power efficiency is a measure of how well the power from the power unit is converted into useful water power in the pump and is calculated as follows:

$$\text{power efficiency (\%)} = \frac{\text{water power output}}{\text{actual power input}} \times 100$$

Energy use efficiency is used to judge how well a pump is performing over a longer period of time.

$$\text{energy efficiency (\%)} = \frac{\text{water energy output}}{\text{actual energy input}} \times 100$$

To calculate energy efficiency, the time over which the energy is used must be known, for example, a day, month or a season.

A system with no energy losses would have an efficiency of 100%, so all the power or energy input would be transferred to the water. But this does not happen in practice. There are always losses in the various components of the power unit and pump, as well as in the pipe system. The actual efficiency will be the product of the efficiency of each component. For example, a centrifugal pump with an efficiency of 80% being driven by an electric motor with an efficiency of 80% would have an overall efficiency of 64% ($0.8 \times 0.8 = 0.64$). The efficiency of the pipe system can be incorporated in a similar way. So it is not just the efficiency of the pump that matters – all the components of the system must be well matched to produce a high overall efficiency. Just to complicate matters, the efficiency of each component is not fixed – the values vary depending on the discharge and pressure in the system. The designer's objective is to match the various components so that together they produce the highest level of overall efficiency at the desired pressure and discharge. Usually this is not one fixed point but a range of pressures and discharges in which it is judged that the system is performing optimally with minimal energy losses.

This is true for all lifting devices and not just centrifugal pumps. Treadle pumps, for example, have an optimum treadling speed that minimises losses and power inputs from an operator and maximises pump output in terms of pressure and discharge.

EXAMPLE: ESTIMATING THE DISCHARGE AND PRESSURE THAT CAN BE ACHIEVED WITH A TREADLE PUMP

Estimate the discharge and pressures that can be expected from a treadle pump operated by a farmer in a developing country (Section 8.2). Make reasonable assumptions to complete the calculations.

The first step is to determine the power available for pumping. Once this is established it is possible to determine the pressure and discharge that could be expected from the pump.

A reasonably fit, well-fed male between 20 and 40 could produce a steady power output of 75 Watts for long periods. To give you some idea of what this means, it is like walking up and down a flight of stairs in about 20 s. Not so bad for the first few times but think about sustaining this over several hours. In a developing country where operators may not be so well fed, or operators are women and children, a more reasonable power value might be 30 Watts.

The pump unit is also not very efficient at converting human power into water power so we need to allow for an efficiency of say 40%.

So taking these figures into account assume the power input is 30 Watts (0.03 kW) and the pump efficiency is 40%:

$$\text{water power (kW)} = \frac{\text{discharge (m}^3\text{/h)} \times \text{head (m)}}{367}$$

$$0.03 \times 0.4 = \frac{\text{discharge (m}^3\text{/h)} \times \text{head (m)}}{367}$$

Power is directly related to both discharge and head so at the same power input level, if head increases then discharge will decrease and vice-versa.

So assume the suction lift is 2 m which is typical for many treadle pump applications. Now calculate the discharge using the above power formula:

$$\text{discharge} = \frac{0.03 \times 0.4 \times 367}{2} = 2.20 \text{ m}^3\text{/h}$$

$$= 0.6 \text{ l/s}$$

If the water lift is reduced to 1 m the discharge increases:

$$\text{discharge} = 1.2 \text{ l/s}$$

So when the pumping head is low the discharge ranges from 0. 6 to 1.2 l/s.

In some countries treadle pumps are used to pump water to much higher heads than this, but there is a limit to this which depends on the operator. The maximum pressure is determined by the weight of the operator who stands on the treadle and applies a downward force on the pistons.

So assume a typical 65 kg operator standing on a treadle immediately above a typical 100 mm diameter piston. Now calculate the pressure using:

$$\text{pressure (N/m}^2) = \frac{\text{force (N/m}^2)}{\text{area (m}^2)}$$

Calculate the force and the area:

$$\begin{aligned}\text{max. force from operator (N)} &= \text{mass of operator (kg)} \times \text{gravity constant (m/s}^2) \\ &= 65 \times 9.81 \\ &= 637 \text{ N}\end{aligned}$$

$$\text{area of piston} = \frac{\pi \times d^2}{4} = \frac{\pi \times 0.1^2}{4} = 0.007 \text{ m}^2$$

Put values for force and area into the formula to calculate pressure:

$$\text{pressure} = \frac{637}{0.007} = 91\,000 \text{ N/m}^2$$

Calculate this in terms of head (m) using the pressure-head equation:

$$P = \rho g h$$

$$\text{head (m)} = \frac{P}{\rho g}$$

$$= \frac{91\,000}{1000 \times 9.81} = 9.27 \text{ m}$$

So the maximum head this operator can achieve pressing his full weight onto each piston is 9 m. In practice not all operators put their full weight on the treadles and so the actual pressure may be much lower than this, say 5 m. The pressure can be increased by reducing the diameter of the pistons but this will also reduce the volume of water lifted each pump stoke. Another way of course, would be to get a heavier operator or perhaps use two operators at the same time, which is often done when children are using it.

If the maximum pressure that can realistically be achieved on this pump is 5 m then in such circumstances, the water power formula shows that the discharge would be reduced to 0. 24 l/s.

Three important points to note:

First, the pumping head is the sum of the suction lift and the delivery lift and so it is the pumping head that is 5 m. Treadle pump suppliers are notorious for quoting delivery and suction lifts separately and this can confuse the buyer who is trying to compare pump performance.

Second, it is clear that you cannot have both high pressure and a large discharge. You can have one or the other and the combination depends on the power available from operator. Basically there is no 'free lunch' – you get out of the pump what you are prepared to put into it.

Finally, this is not rocket science. But it does demonstrate how basic hydraulics can be applied usefully to design what is a simple but very effective water lifting device – something that is not always done in practice.

8.7 Roto-dynamic pump performance

Small centrifugal pumps are sometimes characterised by the power of their drive motors, for example, 3 HP pump or a 5 kW pump; or by their delivery diameter, for example, 50 mm pump (Table 8.1). This provides some guidance for selection but the full performance characteristics should be used for larger pumps.

All roto-dynamic pumps are factory tested and data are published by the manufacturers on the following characteristics:

- discharge and head
- discharge and power
- discharge and efficiency.

These data are usually presented graphically; typical characteristics for all three pump types are shown in Figure 8.13. The curves shown are only for one operating speed. But as pumps can run at many different speeds, several graphs are needed to show their full performance possibilities.

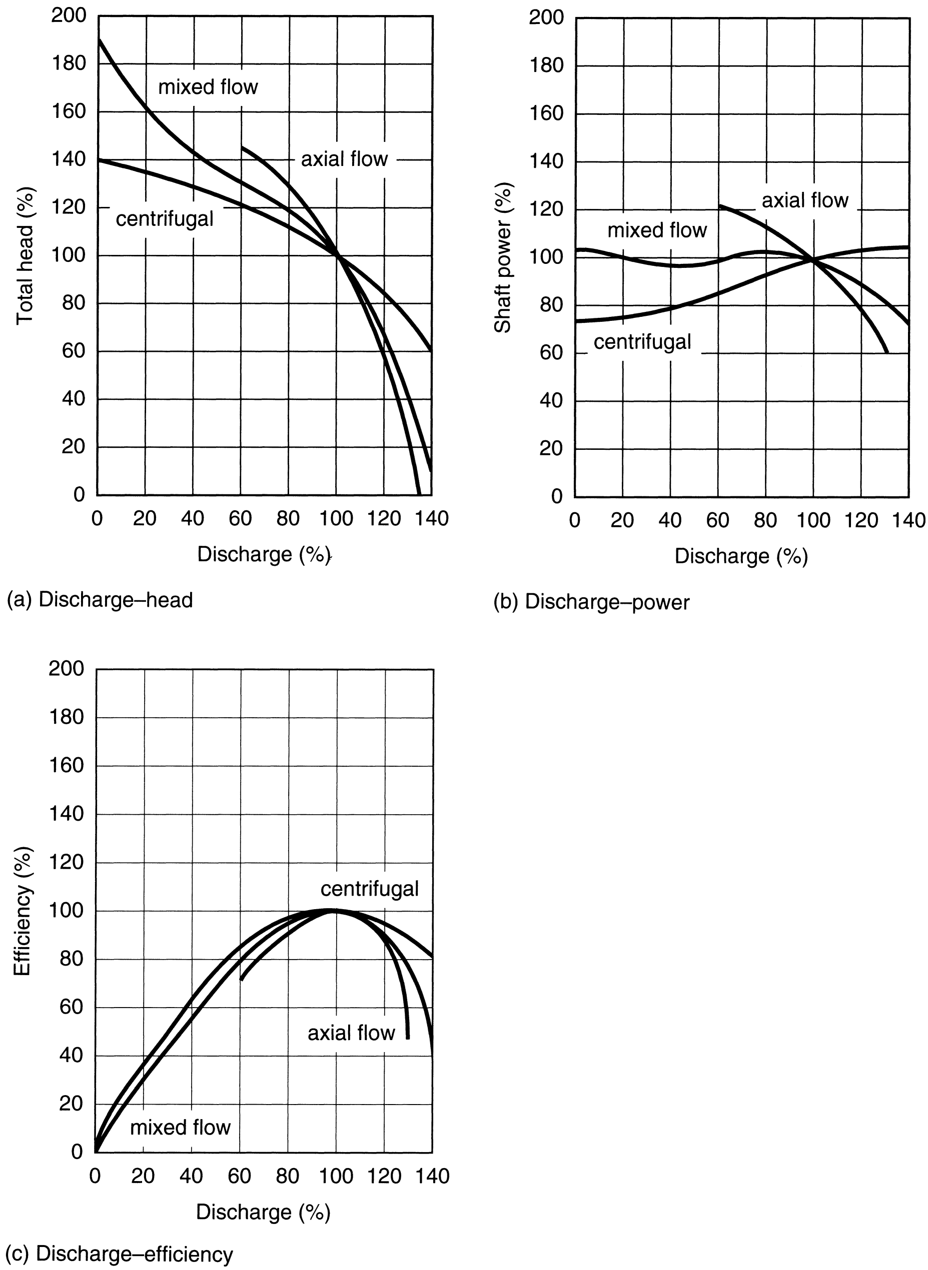
8.7.1 Discharge and head

The relationship between discharge and head is usually of most immediate concern to the user. Will the pump deliver the discharge at the required pressure? A pump can, in fact, deliver a wide range of discharges but there will be changes in pressure as the discharge changes. Pump speed can also be varied and this changes both head and discharge. The faster the speed the greater will be the head and discharge.

Figure 8.13a shows typical discharge – head curves for centrifugal, mixed flow and axial flow pumps for a given pump speed. The axes of the graph would normally show discharge (in m^3/s) and head (in m). But in order to show typical changes in performance the curves have been drawn to show the percentage changes when either the discharge or the head is changed from the normal operating condition represented by the 100% point. So for the centrifugal pump, when the discharge is reduced to 80% of its design flow the head increases to

Table 8.1 Typical discharges from small centrifugal pumps.

Pump size (mm)	*Discharge (litre/s)*
25	0–5
50	5–15
75	15–25
100	25–35
125	35–50



8.13 Pump characteristics.

112% of its design value. Note these values of change are only representative values. They would be different for different pumps and so reference would need to be made to manufacturer's data.

Consider the centrifugal pump first. When it is started up and the delivery valve is closed, there is no discharge and the head is at its maximum. The graph shows that for this pump the head reaches 140% of its design head. As the discharge increases there is a trade-off between discharge and head. When the discharge increases then less head is available.

The discharge-head curves for both the mixed flow and axial flow pumps are similar in shape to that of the centrifugal pump. When the head is high the discharge is low and when the head is low the discharge is high.

The curves for all three pumps are for a given speed of rotation. When the speed is changed the curve will also change. The greater the speed, the greater will be the head and the discharge. So there will be several discharge-head curves for each pump; one for each speed.

8.7.2 Discharge and power

All pumps need power to rotate their impellers. The amount of power needed depends on the pump speed and the head and discharge required. For centrifugal pumps the power requirement is low when starting up but it rises steadily as the discharge increases (Figure 8.13b). For axial flow pumps the power requirement is quite different. There is a very large power demand when starting up because there is a lot of water and a heavy pump impeller to get moving. Once the pump is running the power demand drops to its normal operating level. Mixed flow pumps operate in between these two contrasting conditions and have a more uniform power demand over the discharge range.

8.7.3 Discharge and efficiency

Power efficiency is a measure of the power input to the water power output and this varies over the operating range of all three pump types. Generally, efficiency increases as the discharge increases. But it rises to some maximum value and then falls again over the remaining discharge range (Figure 8.13c). The maximum efficiency is usually between 30 and 80%, and so there is only a limited range of discharges and heads over which pumps operate at maximum efficiency. Outside this range they will still work but they will be less efficient and so more power is needed to operate the same system. Smaller pumps are usually less efficient than larger ones because there is more friction to overcome relative to their size. But inefficiency is less important for small pumps.

8.8 Choosing the right kind of pump

Specific speed (N_s) is one way of selecting the right kind of pump to use – centrifugal, mixed flow or axial. This is a number that depends on the speed of the pump, the discharge and head required for a particular installation and provides a common baseline for comparing pumps.

It is the speed at which a pump will deliver 1 m^3/s at 1.0 m head and is calculated as follows:

$$N_s = \frac{NQ^{1/2}}{H^{3/4}}$$

where N is rotational speed of the pump (rpm); Q is pump discharge (m^3/s); H is pumping head (m).

The specific speed is independent of the size of the pump and so it describes the shape of the pump rather than how big it is. But specific speed is not dimensionless. So it is important to make sure that SI units are used so that the range of specific speeds and pump types is as shown in Table 8.2.

Table 8.2 Specific speeds for different pumps.

Pump type	*Specific speed N_s*	*Comments*
Centrifugal	10–70	High head – low discharge
Mixed flow	70–170	Medium head – medium discharge
Axial flow	above 110	Low head – large discharge

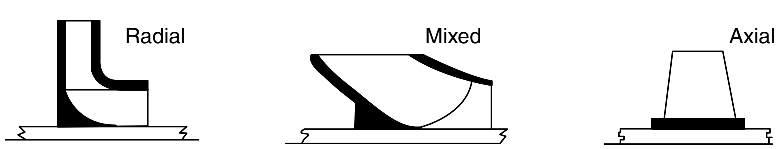
Beware of specific speeds that seem to be quite different to those used here. The main reason is likely to be the use of different units of measurement. USA still uses feet and pounds as its basic units of measurement and so any specific speed there will depend on these units.

The shape of the pump impeller also helps to define the different pump types (Figure 8.14 below). Centrifugal pumps produce high pressures using centrifugal force. To achieve this, the impeller is shaped to turn the flow from the pump inlet through a right angle so that it moves radially outwards towards the delivery pipe as the impeller spins (left-hand picture Figure 8.14). It is this radial flow and the large ratio between the pump inlet diameter and the outlet diameter that generates high velocities and hence the high pressures associated with centrifugal pumps. In contrast, axial flow pumps produce large discharges rather than high heads and so no centrifugal forces are needed. The impeller is propeller-shaped and is designed to move large volumes of water along the axis of the pump. It works in much the same way as a propeller pulls an aircraft through the air (right-hand picture Figure 8.14). Note the ratio of the inlet diameter to the outlet diameter is 1.0. In between these two extremes are mixed flow pumps. These variously have a mixture of radial flow and a larger outlet diameter – to produce some pressure, and axial flow – to produce flow.

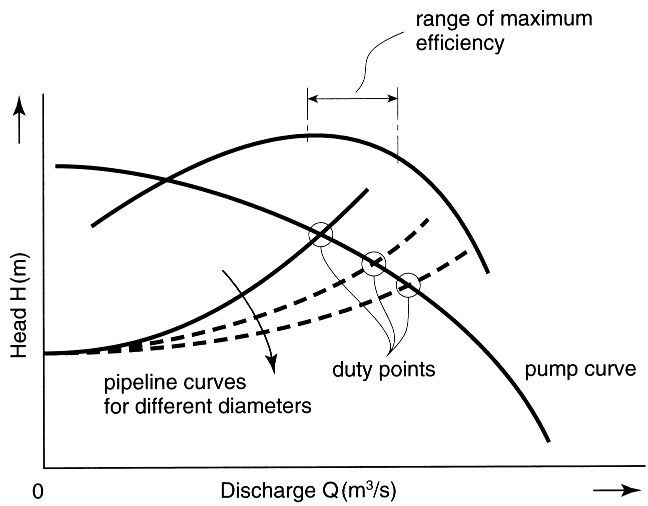
8.9 Matching a centrifugal pump with a pipeline

Selecting the right centrifugal pump and pipe system for a particular water supply installation depends both on hydraulics and cost. There is not just one unique solution to the problem. There will be several pump and pipeline combinations that will do the job from a hydraulic point of view but the final choice usually comes down to cost – which combination is the cheapest? Will a small diameter pipe with a large pump to overcome the high head losses be more or less costly than a larger and more expensive pipeline with a small pump that is less expensive to run? Working out the capital costs of each pump and pipeline together with the fuel and maintenance costs over the life time of the installation will provide the answer.

First the hydraulics. An example of matching a pump and pipeline is shown in Figure 8.15. The pump is represented by two curves – a discharge-head curve and a discharge–efficiency curve. The pipeline is represented by a head–discharge curve which shows how head losses in the pipe change when it is carrying a range of different discharges. This is constructed by putting discharge values into a head loss formula, such as the Darcy-Weisbach formula, and



8.14 Typical pump impeller shapes.



(a)

8.15 Matching a pump with a pipeline.

calculating the resulting head loss. The corresponding discharge and head loss values are then plotted onto the graph. Note the pipeline curve does not start at zero on the head axis but at some point which represents the total static head on the pump. This will be the elevation change from the water level in the sump to the point of delivery. This will be a fixed value for a given installation and not dependent on discharge. The intersection of the pump curve and the pipeline curve gives the pressure and discharge at which the combined pump and pipeline will operate. If this intersection occurs within the acceptable efficiency range for the pump then the combination is acceptable from a hydraulic point of view. A practical example of how to match a pump with a pipeline is shown in the box.

Next comes the cost bit. The hydraulic example above is for one pipeline and pump combination. But other pipe diameters could do the job just as well. Figure 8.15 above shows how several pipelines of different diameters can be assessed each producing a different duty point. More pump curves could also be introduced to see how each pipeline would interact with a range of different pumps. The end result is a wide range of possible pipeline and pump combinations that can do the job. But some will be cheaper than others and the task then is to find out which is the cheapest. This is done by comparing not just the capital costs of each pipeline and pump but a combination of both the capital costs and the running costs (energy and maintenance) over the economic life of the system. An example of how to do this is shown in a box.

EXAMPLE: MATCHING A PUMP WITH A PIPELINE

Water is pumped from a river through a 150 mm diameter pipeline 950 m long to an open storage tank with a water level 45 m above the river. A pump is available and has the discharge-head performance characteristics shown below. Calculate the duty point for the pump–when the friction factor for the pipeline $\lambda = 0.04$.

Total head (m)	30	50	65	80	87	94
Discharge (l/minute)	2000	1750	1410	800	500	0

The first step is to plot a graph of the pump discharge-head characteristic.

Next calculate the head loss in the pipeline for a range of discharge values using the Darcy–Weisbach formula:

$$h_f = \frac{\lambda l v^2}{2gd}$$

When $\lambda = 0.04$, length $l = 950$ m and diameter $d = 0.15$ m:

$$h_f = \frac{0.04 \times 950 \times v^2}{2 \times 9.81 \times 0.15} = 12.9\, v^2$$

But from the discharge equation:

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

And so

$$v^2 = \frac{Q^2}{a^2}$$

Calculate area a

$$a = \frac{\pi d^2}{4} = \frac{\pi \times 0.15^2}{4} = 0.017 \text{ m}^2$$

Use these values to calculate h_f:

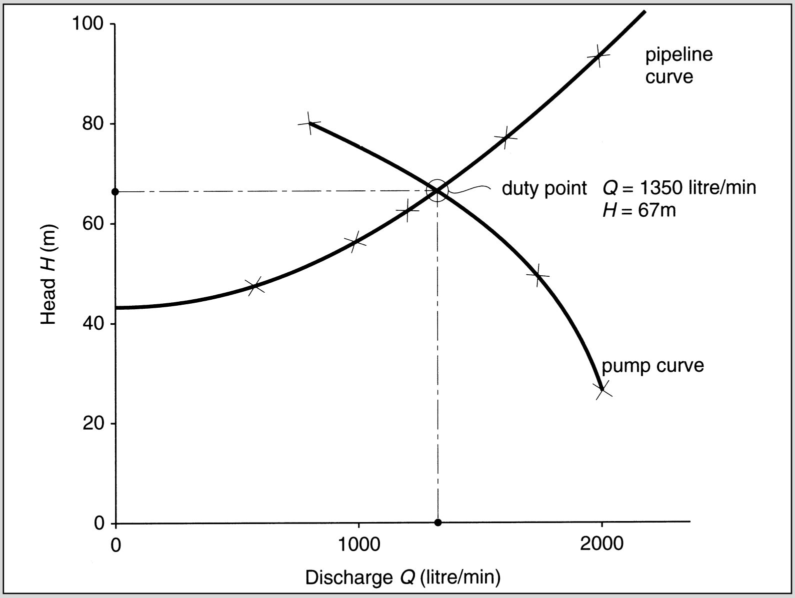
$$h_f = 12.9 \times \frac{Q^2}{0.017^2}$$

$$= 44\,636\, Q^2$$

Now calculate values of head loss for different values of Q. Remember the values of discharge need to be in SI units of m^3/s for the calculation. Also the static head of 45 m must be added to the calculated head loss to determine the total pumping head. The results are tabulated below:

Discharge (l/min)	*Discharge (m^3/s)*	*Head loss h_f (m)*	*Head loss + 45 m (m)*
2000	0.33	48.6	93.6
1600	0.027	30.17	75.15
1200	0.02	17.85	62.85
1000	0.017	12.30	57.30
500	0.006	1.60	46.60

Now construct a graph of the pump curve and the pipe curve:



8.16 Pump and pipeline characteristic curves.

From the graph the duty point is located where the two curves cross. This occurs when:

$Q = 1350$ l/min and $H = 67$ m

EXAMPLE: SELECTING THE CHEAPEST PIPELINE AND PUMP COMBINATION

A pumped water supply pipeline 1150 m long is to deliver 2700 l/min. The static lift is 20 m, the friction factor for the pipeline $\lambda = 0.03$ and the pumping plant operates at 80% efficiency. If the system runs for 17 hours each day 365 days per year, determine the cheapest combination of pipeline and pump for this installation given the following data:

Item	*Cost*	*Economic life*	*Maintenance costs*
Pumps		20 years	10% per annum
One kilowatt of installed power	£150		
Electricity	1.8 p/kWh		
Pipelines			0%
100 mm	£5.90/m	50 years	
150 mm	£8.00/m	50 years	
200 mm	£10.50/m	50 years	
250 mm	£14.80/m	50 years	
300 mm	£21/m	50 years	

Interest rate 5%

To calculate the cost of pumping first determine the power requirement of each pipe-pump combination. So use the power formula:

$$\text{power (kW)} = \frac{\text{discharge (m}^3\text{/h)} \times \text{head (m)}}{367 \times \text{efficiency}}$$

We know the discharge is 2700 l/min (162 m^3/h) and the efficiency is 80% (0.8 in the formula) but we do not know the head which is a combination of the static head and the head loss in the pipeline. So first calculate the head loss h_f using the Darcy–Weisbach formula:

$$h_f = \frac{\lambda l v^2}{2gd}$$

To calculate velocity use the discharge equation:

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

Calculate area a:

$$a = \frac{\pi d^2}{4}$$

Put this into the equation for v:

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

Now calculate the power for each of the pipe and pump combinations. The results are tabulated below.

Pipe dia (mm)	*Velocity (m/s)*	*Head loss (m)*	*Static head*	*Total pumping head (m)*	*Power (kW)*
100	5.73	577.84	20	597.84	329.89
150	2.55	76.09	20	96.09	53.03
200	1.43	18.06	20	38.06	21.00
250	0.92	5.92	20	25.92	14.30
300	0.64	2.38	20	22.38	12.35

In order to combine and compare capital costs and operating costs of the various pipeline-pump combinations calculate the equivalent annual costs for each system (if you are not familiar with discounting cash flows then please refer to any basic economics text book). First calculate the annual pump cost:

annual pump cost (£) = energy cost + annual replacement cost
+ maintenance cost + annual interest cost

Calculate each component of the annual pump cost:

energy cost (£) = operating time (h) × power (kW) × cost (£/kWh)
= 17 × 365 × power (kW) × cost (£/kWh)
= 6205 × power (kW) × cost (£/kWh)

The pump must be replaced every 20 years so the annual replacement cost is 5% of the capital cost of the pump.

$$\text{annual replacement cost} = \%\ \text{annual replacement} \times \text{capital cost (£)}$$
$$= 0.05 \times £150 \times \text{installed power (kW)}$$

$$\text{maintenance cost} = 10\%\ \text{of capital cost}$$
$$= 0.1 \times £150 \times \text{installed power (kW)}$$

$$\text{annual interest cost} = \text{pump capital cost (£)} \times \text{interest rate (\%)}$$
$$= £150 \times \text{installed power (kW)} \times \text{interest rate (\%)}$$

Add all these costs together for each pump to obtain the annual cost of pumping:

Pipe dia (mm) (1)	*Capital cost (£) (2)*	*Energy cost (£) (3)*	*Replacement cost (£) (4)*	*Maintenance cost (£) (5)*	*Interest cost (£) (6)*	*Total annual cost (£) (3)+(4)+(5)+(6)*
100	49 484	36 845	2474	4948	2474	46 741
150	7953	5922	397	795	397	7511
200	3150	2345	157	315	157	2974
250	2145	1597	107	214	107	2025
300	1852	1379	92	185	92	1748

Now calculate the annual cost of each pipeline over the economic life of the pipeline (note in this instance there is no maintenance cost):

$$\text{annual pipe cost (£)} = \text{annual replacement cost} + \text{annual interest cost}$$

Calculate each component of the annual pipeline cost:

The pipe must be replaced every 50 years so the annual replacement cost is 2% of the capital cost of the pipe.

$$\text{annual replacement cost} = \%\ \text{annual replacement} \times \text{capital cost (£)}$$
$$= 0.02 \times \text{capital cost (£)}$$
$$\text{annual interest cost} = \text{capital cost (£)} \times \text{interest rate (\%)}$$

Add all these together to determine the annual cost of the pipelines:

Pipe dia (mm)	*Capital cost (£) (1)*	*Replacement cost (£) (2)*	*Interest cost (£) (3)*	*Total annual cost (£) (2)+(3)*
100	6785	135	339	474
150	9200	184	460	644
200	12 075	241	603	845
250	17 020	340	851	1191
300	24 150	483	1207	1690

Now add the annual pumping costs to the annual pipeline costs to obtain the total cost for each pump and pipeline combination:

Pipe dia (mm)	*Pump costs (£) (1)*	*Pipeline costs (£) (2)*	*Combined cost (£) (1)+(2)*
100	46 741	474	47 215
150	7511	644	8155
200	2974	845	3819
250	2025	1191	3216
300	1748	1690	3438

The costs show that the 250 mm diameter pipe is the lowest cost solution.

8.10 Connecting centrifugal pumps in series and in parallel

There are many situations when one centrifugal pump is not enough to deliver the required head or discharge and so two or more pumps are needed. There may also be circumstances when discharge requirements vary widely, such as in meeting domestic water demand, and it is preferable to have several small pumps working together instead of one large one. Centrifugal pumps can be operated together either in *series* or in *parallel* (Figure 8.17).

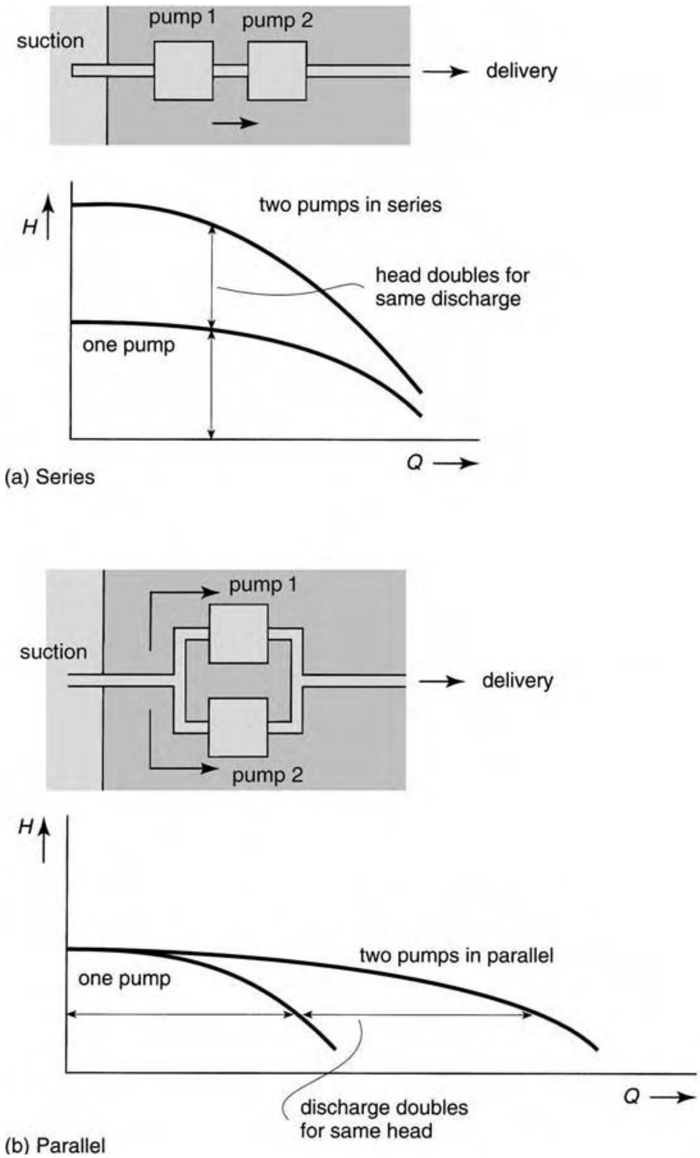
Pumps are connected in series when extra head is required. Note that the pumps need to be identical. They are connected together with the same suction and delivery pipe but are powered by different motors. The same flow passes through pump 1 and then through pump 2 and so the discharge is the same as it would be for one pump but the head is doubled. The discharge-head curve for two pumps is obtained by taking the curve for one pump and doubling the head for each value of discharge.

Pumps are operated in parallel when more discharge is required. Again the pumps must be identical. They each have separate suctions but they are connected into a common delivery pipe. With this type of connection the head is the same as for a single pump but the discharge is doubled. The discharge-head curve for the two pumps is obtained by taking the curve for one pump and doubling the discharge for each value of head.

But doubling the discharge on the pump curve does not mean that the system discharge is also doubled. The new combined pump curve must be matched with the pipe curve to determine the new discharge and pressure at which the new setup will work. The same is true when the pump head is doubled in series connections.

An example in the box illustrates how two pumps working in parallel affect the discharge and head in a pipeline.

Pumps in series work well but when one pump breaks down then the whole system is down. Pumps in parallel are useful when there are widely varying demands for water. One pump can operate to provide low flows and the second pump can be brought into operation to provide the larger flows.



8.17 Pumps in series and in parallel.

EXAMPLE: CALCULATING NEW DUTY POINT FOR PUMPS WORKING IN PARALLEL

In a previous box (Matching a pump with pipeline) the duty point – discharge 1350 l/min and head 67 m – was calculated for a centrifugal pump and a 150 mm diameter pipeline 950 m long supplying an open storage tank with a water level 45 m above the river. Determine the revised duty point when a second similar pump is connected in parallel into the system.

Characteristic of single pump:

Total head (m)	30	50	65	80	87	94
Discharge (l/minute)	2000	1750	1410	800	500	0

Determine the characteristic of two similar pumps working in parallel by doubling the discharge values for the same head. The new characteristic is:

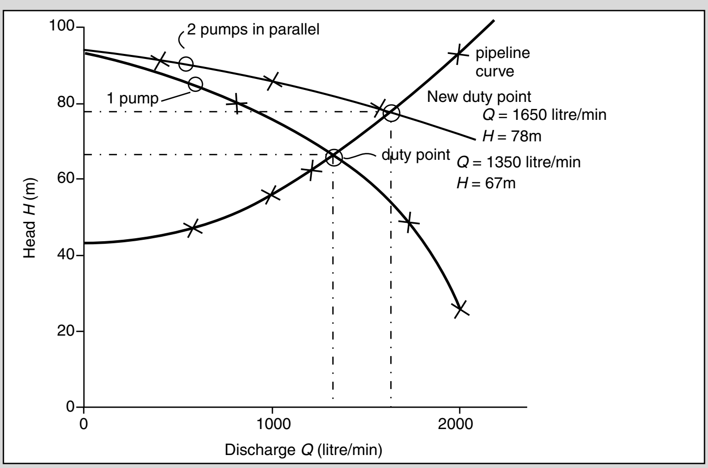
Total head (m)	30	50	65	80	87	94
Discharge (l/minute)	4000	3500	2820	1600	1000	0

The pipe curve is the same as in the previous example as this does not change. The formula for calculating the head loss is based on the Darcy-Weisbach formula:

$$h_f = 44\,636\,Q^2$$

Discharge (l/min)	*Discharge (m³/s)*	*Head loss h_f (m)*	*Head loss + 45 mm*
2000	0.33	48.6	93.6
1600	0.027	30.17	75.15
1200	0.02	17.85	62.85
1000	0.017	12.30	57.30
500	0.006	1.60	46.60

Now construct a graph of the pipe and two pumps curves to show the effect of pumping in parallel:



8.18 Characteristic curves for pumps operating in parallel.

From the graph the new duty point is located where the new parallel pump curve intersects the pipe curve.
This occurs when $Q = 1650$ l/min and $H = 78$ m

8.11 Variable speed pumps

Centrifugal pump installations are designed to the meet the maximum discharge at a particular running speed. Lower discharges are then dealt with by throttling the flow using various types of valve installed along the pipeline. This works fine as a control mechanism but the pump is still using up energy as if it was delivering the design discharge and so a lot of energy is wasted. A typical electric motor costing some £500 could consume over £50 000 in electricity charges over its useful life and so a single percentage point increase in efficiency can save as much as the capital cost of the motor itself.

One solution to the varying flow problem is to vary the pump speed to fit with the change in demand. But this is not always convenient and easy to do. Electric motors, which are increasingly used to drive pumps, are usually ac induction motors and they run at a fixed speed which is determined by the frequency of the alternating electricity supply. In the UK this is 50 Hz (50 cycles per second) so motors (and pumps) run at 3000 rpm. In some countries the ac supply is 60 Hz and so motors there run at 3600 rpm.

It is possible to vary motor speed by setting a number of predetermined speeds using different motor wiring systems. But another way is to vary the frequency of the electricity supply. This is a recent innovation for controlling pumps. The pump motor is fitted with a frequency converter and various electronic sensors so that the pump can be set to run at a constant pressure over a wide range of discharges or at a constant discharge over a wide range of pressures (Figure 8.19). They are often referred to as inverters but this describes only part of their function. The frequency converter adjusts the motor and hence the pump speed so that the various set demands are met. This can significantly reduce energy demands and the costs of pumping.

8.12 Operating pumps

Here are some good practical guidelines for operating pumps properly.

8.12.1 Centrifugal pumps

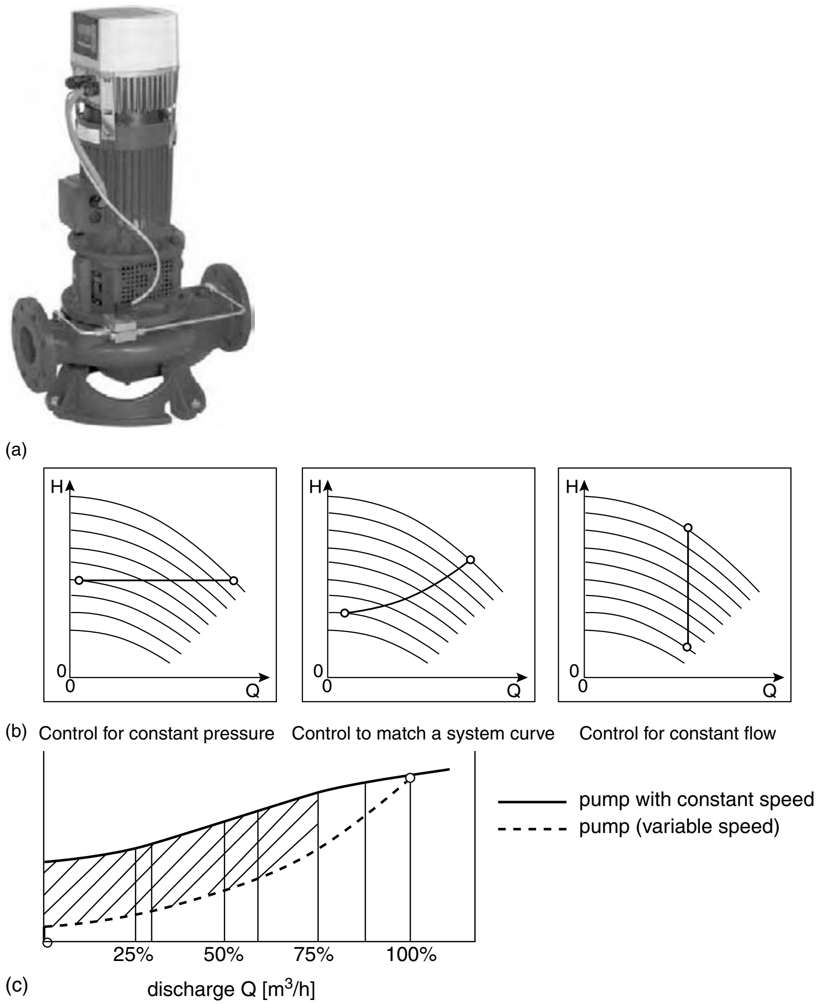
Centrifugal pumps always have control valves fitted on the suction and delivery sides of the pump. The valve on the delivery side is used to control pressure and discharge (Figure 8.8). It is closed before starting so that the pump can be primed. Once the pump is running it is slowly opened to deliver the flow.

A reflux (non-return) valve is also usually fitted after the delivery valve to stop reverse flows that can damage the pump (Section 8.14). Some reflux valves have a small by-pass valve fitted which allows water stored in the pipe to pass around the valve and be used for priming the pump.

A reflux valve is usually fitted on the end of the suction pipe. This keeps the suction pipe full of water and makes priming easier.

Before starting a pump, the delivery valve must be closed. The pump and the suction pipe are then filled with water for priming. When pumps are located below the sump water level they are primed automatically as water flows under gravity into the pump casing. Contrary to expectation starting a pump with a closed delivery valve does not cause problems. The pressure does not go on rising and damage the pump. Rather it reaches a steady speed and pressure. Remember the pressure depends only on the speed of rotation of the impeller. The faster it rotates the higher will be the pressure. The delivery valve is then slowly opened, water flows into the pipeline and as the discharge increases the pressure at the pump gradually falls. At the same time the power requirement increases. This gradual change continues until the delivery pipe is full of water and the system is operating at its design head and discharge.

These operating procedures are important for several reasons. Suppose for example, the delivery valve was left open as often happens. Not only would it be difficult to prime the pump



8.19 Variable speed pumps. (a) Pump fitted with electric motor power unit and frequency converter. (b) Pump output matched to demand by varying motor speed. (c) Difference in power consumption for fixed and variable speed pumps.

but also when it is started, water would surge along the delivery pipe. Such a sudden surge of water might damage valves and fittings along the pipeline. The rapid increase in discharge also causes a rapid increase in power demand and power units do not usually like this. It is like getting into a car when the engine is cold and suddenly trying to put it into top gear and accelerate fast. The power demand is too great and the engine will stall and stop. The most sensible way to get moving is to build up the power demand gradually through the use of the gears and clutch. The same idea applies to pumps by opening the delivery valve slowly.

8.12.2 Axial flow pumps

Axial flow pumps do not have any control valves on either the suction or the delivery. Because of the high power demand when starting up, it is not desirable to start a pump against a closed valve. What is needed, however, on some pumps is a siphon breaker. This stops water from siphoning back from the delivery side into the sump when the pump stops by allowing air into the pump casing.

8.13 Power units

There are two main types of power unit: internal combustion engines and electric motors.

8.13.1 Internal combustion engines

Many pumping installations do not have easy access to electricity and so rely on petrol (spark ignition) engines or diesel (compression ignition) engines to drive pumps. These engines have a good weight to power output ratio, and are compact in size and relatively cheap due to mass production techniques.

Diesel engines tend to be heavier and more robust than petrol engines but are more expensive to buy. However, they are also more efficient to run and if operated and maintained properly they have a longer working life and are more reliable than petrol. In some countries petrol-driven pumps have needed replacing after only three years of operation. Diesel pumps operating in similar conditions could be expected to last at least 6 years. However, it must not be forgotten that engine life is not just measured in years, it is measured in hours of operation and its useful life depends on how well it is operated and serviced. There are cases in developing countries where well maintained diesel pumps have been in continuous use for 30 years and more. A diesel pump can be up to four times as heavy as an equivalent petrol pump and so if portability is important a petrol engine may be the answer.

8.13.2 Electric motors

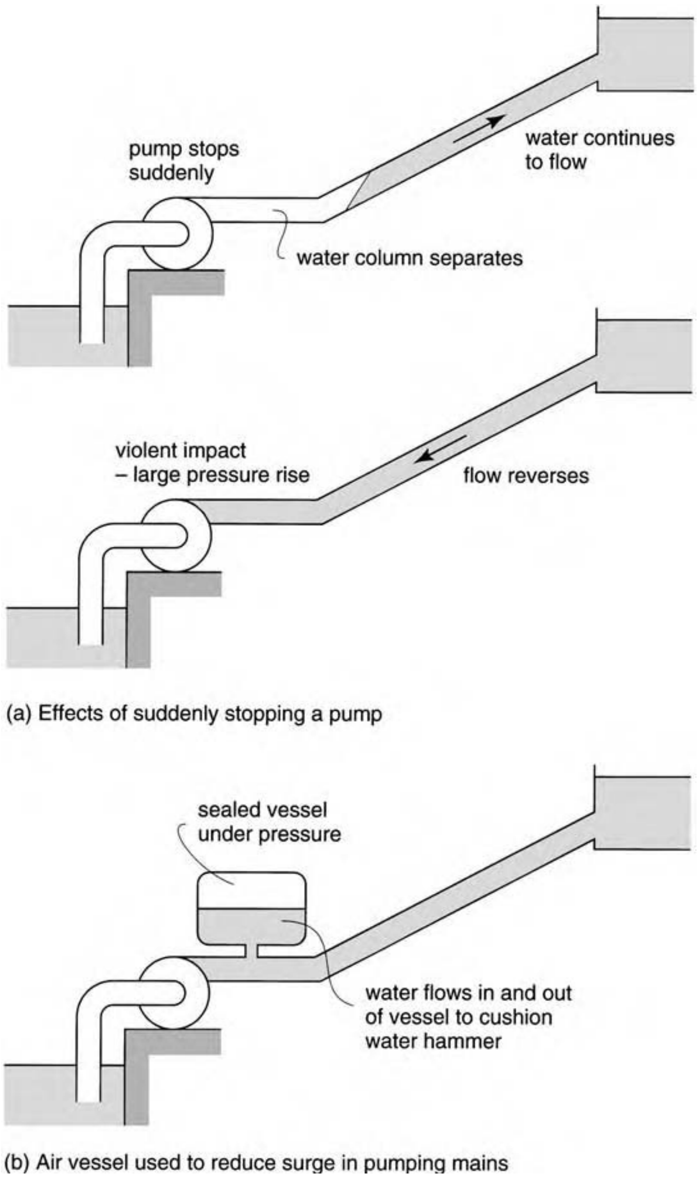
Electric motors are very efficient in energy use (75–85%) and can be used to drive all sizes and types of pumps. The main drawback is the reliance on a power supply which is outside the control of the pump operator and in some countries it is unreliable. Inevitably electrical power supplies usually fail when they are most needed and so backup generators may be needed driven by diesel engines.

8.14 Surge in pumping mains

Water hammer in pipelines has already been discussed in Section 4.14, but it can cause particular problems in pumping mains by creating *surges*. This is the slower mass movement of water often resulting from the faster moving water hammer shock waves.

In pumping mains the process is reversed and it is the surge of water that can set up the water hammer. To see how this happens imagine water being pumped along a pipeline when suddenly the pump stops (Figure 8.20). The flow in the pipe does not stop immediately but continues to move along the pipe. But as the main driving force has now gone, the flow gradually slows down because of friction. As the flow moves away from the pump and as no water enters the pipe, an empty space forms near the pump. This is called *water column separation* and the pressure in the empty space drops rapidly to the vapour pressure of water.

The flow gradually slows down through friction and stops. But there is now an empty space in the pipe and so water starts to flow back towards the pump gathering speed as it goes. It rapidly refills the void and then comes to a sudden and violent stop as it hits the pump. This is very similar to suddenly closing a valve on a pipeline and results in a high pressure shock wave



8.20 Surge and water hammer in pumping mains.

which moves up the pipe at high speed (approx. 1200 m/s). This can not only burst the pipe but it may also seriously damage the pump as well.

There are several ways of avoiding this problem:

Stop pumps slowly. When pumps stop slowly water continues to enter the pipe and so the water column does not separate and water hammer is avoided. Electric pumps are a problem because they stop very quickly when the power fails. Diesel pumps take some time to slow down when the fuel is switched off and this is usually enough time to avoid the problem.

Use a non-return valve. Using a non-return valve in the delivery pipeline will allow the flow to pass normally along the pipe but stop water from flowing back towards the pump. In this case the valve and the pipe absorb the water hammer pressures and the pump is protected.

Use an air vessel. This is similar in action to a surge tank but in reverse (Figure 8.20b). When a pump stops and the pressure starts to drop, water flows from a pressurised tank into the pipeline to fill the void and stop the water column from separating. When the flow stops and reverses, it flows back into the tank. The water then oscillates back and forth until it eventually stops through friction. Unlike a surge tank, an air vessel is sealed and the air trapped inside acts like a large coiled spring compressing and expanding to dampen the movement of the water in much the same way as a shock absorber on a car dampens the large bumps in the road. This device tends to be expensive and so would only be used on large, important pumping installations where the designers expect serious water hammer problems to occur.

8.15 Turbines

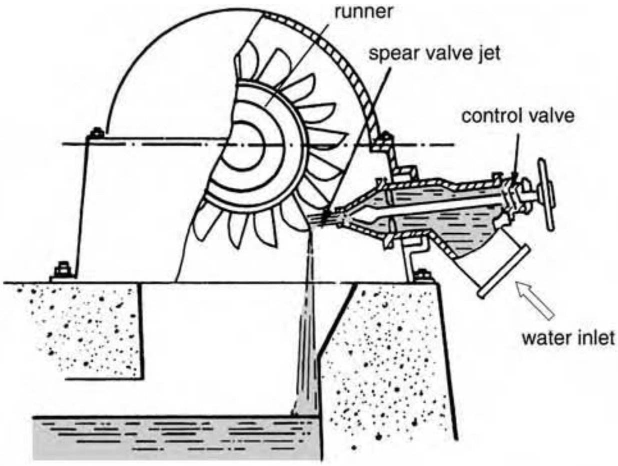
Water can also produce energy as well as absorb it. This idea has been exploited for centuries long before rotary pumps were invented. Water wheels were used by the Romans and throughout Europe to grind cereals. Today these wheels have been replaced by turbines that are connected to generators to produce electrical energy. These are used extensively in Scotland for power generation, in mountainous countries such as Switzerland and in many other countries where convenient dam sites can be located to produce the required head. Unfortunately there are not many sites around the world where there are sufficient renewable energy sources that can be exploited in this way. They are not usually the main source of electricity but provide a valuable addition to the main power source, such as coal, gas or oil-fired power stations, to meet peak demands.

Another development has been the combination of pumps and turbines in what is known as a pumped storage scheme. In Wales, for example, a small hydropower plant is used to generate electricity during the day to meet peak demands. During the night when power demand is low, surplus electricity from the main grid is used to pump water back up into the reservoir to generate electricity the next day. This is a way of 'storing' electricity by storing water.

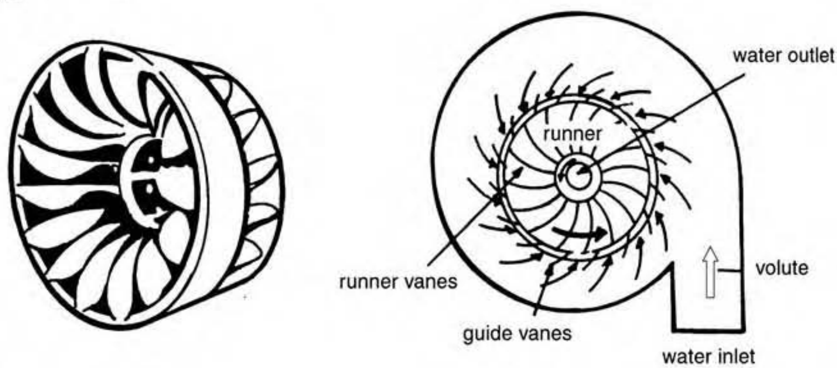
There are three main types of turbine (Figure 8.21): impulse turbines, reaction turbines and axial flow turbines.

Impulse turbines are similar in operation to the old water wheel. The most common is the Pelton wheel, named after its American inventor, Lester Pelton (1829–1908). This comprises a wheel with specially shaped buckets around its periphery known as the runner. Water from a high level reservoir is directed along a pipe and through a nozzle to produce a high speed water jet. This is directed at the buckets and causes the runner to rotate. The momentum change of the jet as it hits the moving buckets creates the force for rotation. So by knowing the speed of the jet it is possible to work out this force and the amount of electrical energy that can be generated. Pelton wheels can be very efficient at transferring energy from water to electricity and figures as high as 90% are quoted by manufacturers. They are best suited to high heads above 150 m. Some installations run with heads in excess of 600 m.

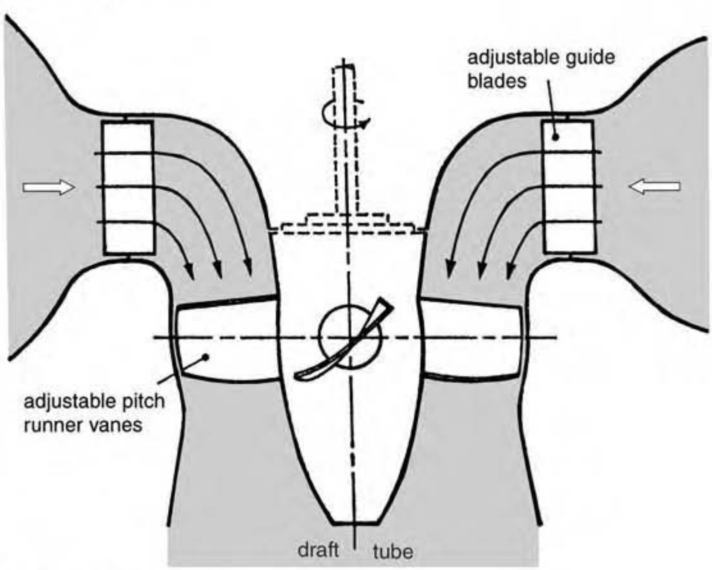
Reaction turbines are like centrifugal pumps in reverse. The most common design is the Francis turbine. Although James Francis (1815–1892) did not invent the turbine he did a great deal to develop the inlet guide vanes and runner blades to improve turbine efficiency and so his name became associated with it. The turbine resembles a centrifugal pump but instead of the rotating impeller driving the water as in a pump, the runner is driven by the water. Water is fed



(a) Pelton wheel



(b) Francis turbine



(c) Kaplan turbine

8.21 Turbines.

under pressure and at high velocity around a spiral casing onto the runner and as rotates the pressure and kinetic energy of the water is transferred to the runner. Francis turbines normally work at heads of 15–150 m.

Axial flow (or propeller) turbines are known as Kaplan turbines. They are named after Victor Kaplan (1876–1934), a German professor who developed this type of turbine with adjustable blades. Axial flow turbines operate like axial flow pumps in reverse. They operate at low heads (less than 30 m) and are used in tidal power installations. They have blades on their runners that can be twisted to different angles in order to work at high efficiency over a wide range of operating conditions. This kind of turbine is also used in pumped storage schemes. It is used as a turbine during the day when power demand is high and at night it is connected to an electric motor and used as a pump to lift water back into a high level reservoir in readiness for the next day.

8.16 Some examples to test your understanding

1 A centrifugal pump is connected to a 25 m length of 300 mm diameter suction pipe and discharges into a 500 m long pipe with a diameter of 150 mm. The water level in the river is 2 m above datum, the centre line of the pump is 3.75 m higher, and the discharge pipe enters a tank whose water level is 25 m above datum. Calculate the pressure head that the pump must produce to pump a discharge of 0.150 m^3/s. Assume a friction factor of $\lambda = 0.015$ (61.8m).

2 A pump supplies water to a large reservoir through a 2000 m long pipeline 400 mm in diameter. If the difference in water level between the sump and the reservoir is 20 m and the friction factor for the pipeline is $\lambda = 0.03$, calculate the pressure and power output required to deliver a discharge of 0.35 m^3/s (29.38 m; 101 kW).

3 A centrifugal pump lifts water from a well to a storage tank with an outlet 20 m above the water level in the well. The suction and delivery pipes are 50 mm in diameter and the total pipe length is 150 m. The friction factor for the pipeline is $\lambda = 0.035$. The pump performance at different heads is tabulated below:

Head (m)	0	12	25	30	32	33
Discharge (l/s)	9	8	6	4	2	0
Efficiency (%)	0	50	60	60	50	0

Plot the pump characteristic curves and the pipe system curve and determine the discharge, head, efficiency, and power required at the duty point (2.8 l/s; 31 m; 56%; 0.85 kW).
If a second pump is connected in series calculate the new duty point (4.8 l/s; 57 m; 60%; 2.64 kW).

9 Bathtub hydraulics

Having a bath is a good practical way to learn about hydraulics. Here are just some of the things to look out for.

Just filling the bath can be an experience in itself. The hot and cold water taps are running but then you notice that the two are not mixing well. One side of the bath is hot and the other cold. There is some mixing at the interface due to turbulence but this is all. The only way to get an even temperature is to stir the water vigorously with your hand. The reason for this mixing problem is that water density varies slightly with temperature and this density difference is enough to inhibit mixing. This is a major problem at most power generating stations that use vast quantities of water from rivers and the sea to cool their systems. High towers help to cool water before it is returned to the river or sea but any slight difference in temperature will stop it mixing fully with the receiving flow. Swimming downstream of a power station can be a pleasant, warm experience. The challenge for the engineers is to find ways of mixing the water thoroughly so that the receiving water returns to its original temperature as quickly as possible so as not to affect local aquatic plant and fish life. It also stops hot water short-circuiting the system and finding its way into the intake and back into the power station as can happen with coastal stations.

As water flows from the taps across the bottom of the bath the flow is usually super-critical – fast and shallow. But this state soon changes as a hydraulic jump forms at the far end and then quickly makes its way back towards the tap end as a travelling surge wave (Section 6.4). For a while a stable circular hydraulic jump can be seen just where the water plunges into the bath but eventually this is drowned out as the level in the bath rises. The incoming flow is still super-critical and this now shoots under the slow flow. The energy is not dispersed in the hydraulic jump, it is gradually absorbed as friction along the base of the bath slows the water down. When this occurs in natural channels it can cause severe erosion of the bed and sides (Section 7.2.1).

Once your bath is full to the right level and is at the right temperature, there is now that 'Eureka moment' that Archimedes experienced when he stepped into the water. Archimedes first discovered the significance of this some 2000 years ago when he realised that the water displaced when you get into the bath has the same volume as your body. He also noticed that if you float on the water instead of sitting on the bottom then the amount of water you displace is equal to your weight. From this he was able to solve the problem that the King of Syracuse had over what materials had been used to make his crown – was it gold or was it really lead (Section 2.12)? This was the beginning of our understanding of hydrostatics (water which is not moving) and led to formulae for the design and construction of water tanks, dams and submarines that we still use today. It is an almost perfect theory. It was unfortunate that the Greeks also tried their hand at hydrodynamics – water which is moving – but they got this bit wrong and sent science off in the wrong direction for almost 2000 years. We had to wait until the likes of Sir Isaac Newton came along to put things right.

Sitting still in a bath and soaking up the warmth is a good experience – but for children this is almost impossible. Sliding up and down quickly is much more fun as you can make waves and even make the water flow over the sides of the bath. This is because water is a real fluid with viscous properties. As you start to move up and down the bath you transfer your body momentum to the water by surface and form drag (Section 3.10) – the larger you are the more form drag you can create and the more water you can move. When you stop sliding about, the waves seem to continue for a while before they stop. This is because the only force available to absorb the wave energy is friction and as there is very little of this in a bath it takes some time to suppress the waves. Water can also slosh about in harbours in much the same way as the bath tub and it can cause lots of problems for ships. The wave energy comes from the sea, it enters the harbour and it is difficult to get rid of because, like the bathtub, the walls of the harbour reflect the energy rather than absorb it. The movement of the water can move ships back and forth on their moorings which can be a major problem if the ship happens to be a supertanker and you are trying to keep it still while loading it with oil. This is why harbour entrances are narrow and specially angled to stop wave energy from entering. You may have noticed that the sea is much calmer inside a harbour than outside. Some harbours though have been known to behave in quite the opposite way. When the entrance is narrow the waves inside seem to get larger. This is a resonance effect that harbour designers must guard against.

As you relax in the bath you decide to have a drink. A glass of whisky will do the trick but you want water with it. So here is a little puzzle to while away the time. Do you put the water in the whisky or the whisky in the water? Start with a glass of water and a glass of whisky. One spoonful of whisky is put into the water and mixed and then one spoonful of the mixture is put back into the whisky. Is there more whisky in the water or more water in the whisky? If you cannot work it out then have a look at the solution in the box.

It is now time to get out of your bath and take out the plug. Notice how this sets up a nice whirlpool around the plug hole and it seems to be hollow down the middle. This is a boundary layer effect similar to those described in Section 3.9.3. The boundary layer close to the outlet slows the flow velocity which makes the water swirl and the vortex then forms. We say that the boundary layer curls up. All water intakes at reservoirs and control gates along rivers suffer from vortices like this which draw in air and reduce the water discharge. If you put some floating object over the vortex, such as your plastic duck, it stops the swirling and the discharge down the plughole increases. This is what engineers do in practice to stop vortices from forming at off-takes and also pump station suction inlets. Setting the outlet or the pump intake deep below the water surface will also suppress the vortices.

Finally there is the inevitable question that I am sure will keep you awake at night – which way does the vortex go? Some say that the vortex goes in a clockwise direction in the northern hemisphere and anticlockwise in the southern hemisphere. There is a very practical demonstration of this by an enterprising young man who lives on the equator in Kenya and puts on demonstrations for the tourists. He has a large can filled with water which he sets up 20 m north of the equator. When he pulls a plug out of the bottom of the container the water slowly starts to swirl in a clockwise direction. He then does the same test 20 m south of the equator and the swirl starts in an anticlockwise direction. It works every time. Convinced? The reality is that no one is absolutely sure. Experiments to test the theory have been tried but they are very difficult to do in a laboratory. The problem is that the force which causes the movement – the Coriolis force which comes from the earth's rotation – is very small in comparison to other forces around such as minor vibrations due to traffic outside the laboratory or temperature changes in the room setting up convection currents in the tank. All these can significantly influence which way the water will begin to swirl and override the effects of the Coriolis force. A large tank of water is needed to get an appreciable Coriolis force but arranging this under laboratory conditions is not very practicable.

If you have an aversion to baths and you prefer a shower there is always something to learn here though it may not be as much fun. When you switch on the shower and draw the plastic curtain around you for a bit of privacy, have you noticed how it tends to cling to your body – it is cold and uncomfortable. This is because the fast downward flow of water from the shower causes a slight drop in air pressure within the curtained space (remember the energy equation – energy changes from pressure energy to kinetic energy). The pressure outside the curtain is still at atmospheric pressure – slightly greater than the air pressure inside – and so the pressure difference causes the curtain to move towards the water and to cling to you.

Once you start to appreciate what is going on in your bath, bath-times will never be the same again!

EXAMPLE: A MIXING PROBLEM

Take one glass of water and a equal glass of whisky. One spoonful of whisky is put into the water and mixed. One spoonful of the mixture is put back into the whisky. Is there more whisky in the water or more water in the whisky?

Start by assuming that each glass holds 10 spoonfuls – so the water glass holds 10 spoonfuls of water and the whisky glass holds 10 spoonfuls of whisky. Now follow the argument below under the water and whisky headings as liquid is moved from one to the other:

Water	*Whisky*
10_{water}	10_{whisky}
One 'spoonful' (one part) of water is taken from the water and added to the whisky	
9_{water}	$10_{whisky} + 1_{water}$
The whisky glass now holds 11 spoonfuls of the mix. Each spoonful of the mix comprises 10 parts water and one part whisky i.e. $(10_{whisky} + 1_{water})/11$. Now take one spoonful of the mix and return this to the water:	
$9_{water} + (10_{whisky} + 1_{water})/11$ $9_{water} + 1/11_{\ water} + 10/11_{\ whisky}$ $9\ 1/11_{water} + 10/11_{whisky}$	$10_{whisky} + 1_{water} - (10_{whisky} + 1_{water})/11$ $10_{whisky} + 1_{water} - 10/11_{\ whisky} - 1/11_{\ water}$ $9\ 1/11_{whisky} + 10/11_{water}$

So the result is the amount of water in the whisky is the same as the amount of whisky in the water. So whichever way you mix your drinks it make no difference.

References and further reading

BS3680 *Measurement of Liquid Flow in Open Channels*. British Standards Institution, London (also ISO 748 1979)

(1981) Part 4A *Thin Plate Weirs*

(1981) Part 4C *Flumes*

Part 4E *Rectangular Broad Crested Weirs*

These are the standards which are the basis for the design, installation and operation of flow measuring structures.

Chadwick, A., Morfett, J. and Borthwick, M. (2004) *Hydraulics in Civil and Environmental Engineering*. 4th edition E & FN Spon, London.

Similar in some ways to Webber, this is another excellent undergraduate text but its coverage of hydraulics is much wider and includes sediment transport, river and canal engineering, coastal engineering and hydrology.

Chow, V.T. (1981) *Open Channel Hydraulics*. International Student Edition. McGraw Hill.

Although an 'old' book it is still the bible of open channel hydraulics. It is a handbook with lots of information on all aspects of channel flow and design – it is not an elementary text.

Douglas, J.F. (1996) *Solving Problems in Fluid Mechanics Parts I and II*. Longman, Harlow, UK.

This is a good problem-solving-oriented text. It approaches the principles of hydraulics by showing how to solve numerical problems.

Hamill, L. (2001) *Understanding Hydraulics*. MacMillan Press Ltd, Basingstoke, UK.

This is a book written by a teacher of hydraulics who clearly has a lot of experience of teaching a difficult subject. It is written in a conversational style with the student asking questions and the teacher providing the answers. It is sympathetic to the needs of young engineers but makes no allowances for those with limited mathematical skills.

Hydraulics Research (1990) *Charts for the Hydraulic Design of Channels and Pipes*. 6th Edition. Thomas Telford Ltd.

An excellent and widely used book of design charts based on the Colebrook-White equation.

King, H.W. and Brater, E.F. (1996) *Handbook of Hydraulics for the Solution of Hydrostatic and Fluid Flow Problems*. 7th edition. McGraw Hill.

Very useful for the solution of Manning's equation for channel flow. His method is used in this text.

Portland Cement Association (1964) *Handbook of Concrete Culvert Pipe Hydraulics*. Chicago, IL.

Very comprehensive book on culvert hydraulics and their construction. Although it is primarily supporting concrete culverts they can be made out of other materials as well.

Rouse, H. and Ince, S. (1957) *History of Hydraulics*. Dover Publications, New York.

A comprehensive, well-researched and very readable history of hydraulics from the early work of Greek scholars to our modern notions of fluid behaviour. Rouse is a well-known authority on hydraulics.

USBR (1974) *Design of Small Dams*. United States Dept of the Interior, Bureau of Reclamation.

Lots of useful construction details of hydraulic structures and particularly weirs and spillways.

Vallentine, H.R. (1967) *Water in the Service of Man*. Penguin Books Ltd, Harmondsworth, UK.

Sadly this book is now out of print but copies are still available on the internet. It is an excellent, easy to read, introduction to the fundamentals of water flow in pipes, channels and pumps as well as providing a broader appreciation of water and its uses including its history. The text is very descriptive, anecdotal and entertaining in style with lots of good explanations and very little mathematics. Vallentine is clearly an engineer who knows how to communicate his ideas in a practical and interesting way. Penguin really ought to reprint this book.

Webber, N.B. (1971) *Fluid Mechanics for Civil Engineers*. Chapman Hall, London.

An excellent, comprehensive undergraduate civil engineering textbook covering both basic principles and practical applications. Now out of print but copies still available on the internet.

Index