UNIVERSIDAD AUTÓNOMA DE QUERÉTARO

FACULTAD DE INGENIERÍA

ÁREA INGENIERÍA CÍVIL

HIDRAULICA I

GUIA DEL MAESTRO

Como parte del requisito para obtener la licenciatura en:

INGENIERÍA CIVIL

Presenta:

Victor Hugo Castro Saavedra

UNIVERSIDAD AUTÓNOMA DE QUERÉTARO

FACULTAD DE INGENIERÍA

ÁREA INGENIERÍA CÍVIL

Curso: Hidráulica 1

Código de curs**o:**

Semestre: 5

Duración: 18 semanas

Horas/Semanas: 4

Créditos:

Requisitos:

Curso	Código del curso
Física	
Mecánica de fluidos	

Objetivo:

El propósito de este curso es el proporcionar lo básico y conocimientos prácticos sobre la hidráulica, incluyendo hidrostática, hidrodinámica, flujo en conductos cerrados e hidráulica modelada

UNIDADES

UNIDAD	DESCRIPCIÓN
1	CONCEPTOS BÁSICOS
2	HIDROSTÁTICA (AGUA EN REPOSO)
3	HIDRODINÁMICA (FLUJO DE AGUA)
4	TUBERIAS
5	TEORIA DE MODELADO FÍSICO
6	ORIFICIOS, COMPUERTAS Y VERTEDORES

PROFESOR: DR. Eusebio Jr. Ventura Ramos

Clasificación:

Exámenes Parciales: 25% cada uno 75%

Tareas 25%

Horario: Lunes 10:00 – 12:00 y Jueves 09:30 – 11:30 a

CONTENTS

Page

Bibliografía

Kay, M. 2008. Practical Hidraulics. 2nd Edition. Taylor & Francis. New York, USA. 122 pp.

1. **SOME BASIC MECANICS**

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 behavior 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 that a pipeline is 6 long is not enough. It might be 6 centimeters, 6 meters or 6 kilometers. 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 centimeters, grams 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.

SI Units

The System International of 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 meters, 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	Meter	M
Mass	Kilogram	Kg
Time	Second	S

Table 1.2 Some useful derived units.

Measurement	*Dimension*	*Measurement*	*Dimension*
Area	*m2*	*Force*	*N*
Volume	*m3*	*Mass*	*Density kg/m3*
Velocity	*m/s*	*Specific weight*	*N/m3*
Aceleration	m/s2	Pressure	N/m2
Viscosity	Kg/ms	Momentum	Kgm/s
Kinematic Viscosity	m2/s	Energy for solids	Nm/N
		Energy for fluids	Nm/N

Note there is no mention of centimeters in Table 1.1.

Centimeters are part of the “CGS” units and not SI and so play no part in hydraulics or in this text. Millimeters are acceptable for very small measurements and kilometers for long lengths – but *not* centimeters.

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 meter 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. centimeters or inches instead of meters, 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.

<u>Velocity and Acceleration</u>

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 meters/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 meters/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:

Velocity (m/s)= $\frac{Distance\ (m)}{Time\ (s)} \frac{Distance\ (m)}{Time\ (s)}$

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 meters/second/ second (m/s). 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:

Aceleration (m/s) = $\frac{change\,in\,velocity\ (\frac{m}{s})}{time\ (s)} \frac{change\,in\,velocity\ (\frac{m}{s})}{time\ (s)}$

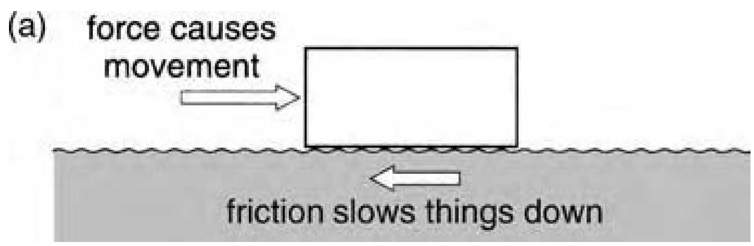
EXAMPLE: CALCULATING VELOCITY AND ACCELERATION An object is moving along at a steady velocity and it takes 150 s to travel 100 m. Calculate the velocity. Velocity = $\frac{Distance\ (m)}{Time\ (s)} \frac{Distance\ (m)}{Time\ (s)} = \frac{100}{150} \frac{100}{150}$ = 0.67 m/s If the object starts from rest, calculate the acceleration if its final velocity of 1.5 m/s is reached in 50 s: Acceleration = $\frac{Change\ in\ velocity\ (\frac{m}{s})}{Time\ (s)} \frac{Change\ in\ velocity\ (\frac{m}{s})}{Time\ (s)} = \frac{1.5-0}{50} \frac{1.5-0}{50}$ = 0.03 m/ $s^2\ s^2$

1.3. 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).

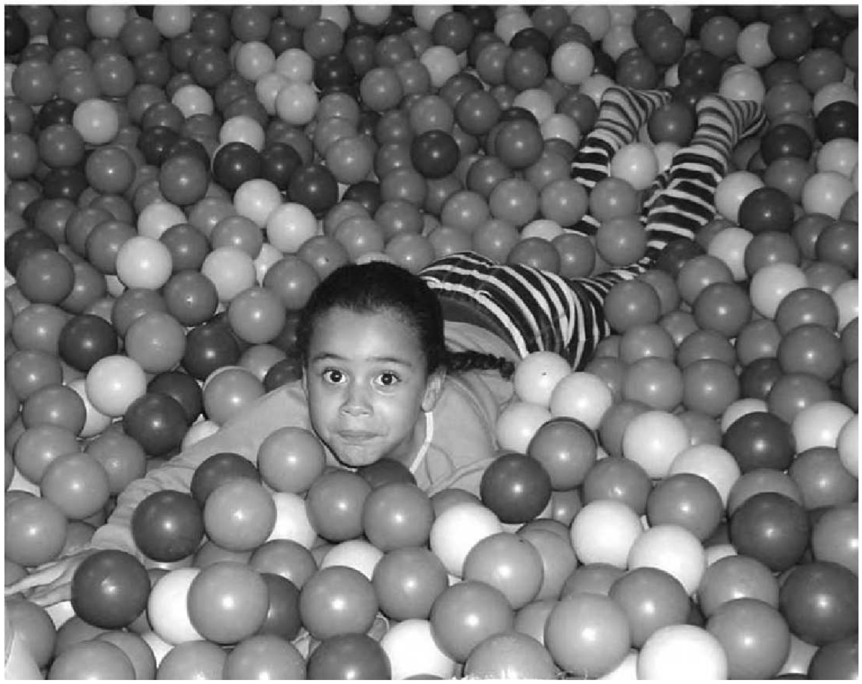
Friction is an essential part of our existence but sometimes it can be a nuisance. In car engines, for example, friction between the moving parts would cause them to quickly heat up and the engine would seize up. But oil lubricates the surfaces and reduces the friction.

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).



Cha 1 Fig 1 - Friction

(b)



Cha 1 Fig 2 (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).

1.4. 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/s2 – 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:

Weight (N) = Mass (kg) X Gravity constant (m/s2)

EXAMPLE: CALCULATING THE WEIGHT OF AN OBJECT
Calculate the weight of an object when its mass is 5 kg.
Using Newton's second law:
Weight = Mass X Gravity constant
Weight = 5 X 9.81 X 49.05 N
Sometimes engineers assume that the gravity constant is 10 m/s2 because it is easier to multiply by 10 and the error involved in this is not significant in engineering terms.
In this case:
Weight = 5 X 10 = 50 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!

1.5. 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 m3 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.

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.

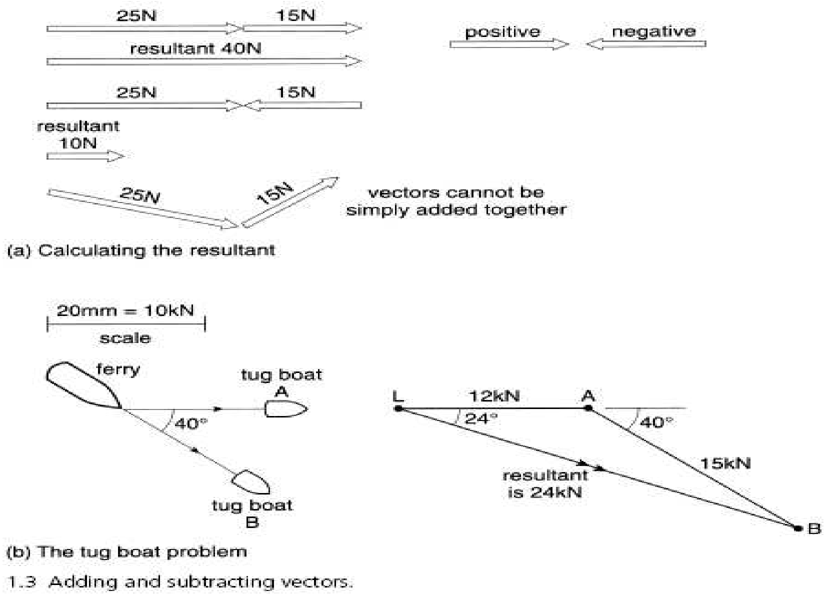
EXAMPLE: CALCULATING THE RESULTANT FORCE USING A VECTOR DIAGRAM

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) 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.*



Cha1 Fig 3 Adding and subtracting vectors

1.6. 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.

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) X 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).

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:

$$\boxed{\text{Energy Required} = \text{Work Done}}$$

so:

$$\boxed{\text{Energy Required (Nm)} = \text{Force (N)} \times \text{Distance (m)}}$$

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

$$\boxed{1\ \text{Ws} = 1\ \text{Nm}}$$

But this is a very small quantity for engineers to use and so rather than calculate energy in large numbers of Newton-meters 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:

$$\boxed{1\ \text{Wh} = 3600\ \text{Nm}}$$

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

$$\boxed{\begin{array}{c} 1\ \text{kWh} = 3\ 600\ 000\ \text{Nm} \\ 1\ \text{kWh} = 3600\ \text{kNm} \end{array}}$$

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:

$$\boxed{1\ \text{joule} = 1\ \text{Nm}}$$

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.

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.

And so:

$$\text{Power} = \frac{Energy\ (kWh)}{Time\ (h)} \quad \frac{Energy\ (kWh)}{Time\ (h)}$$

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.

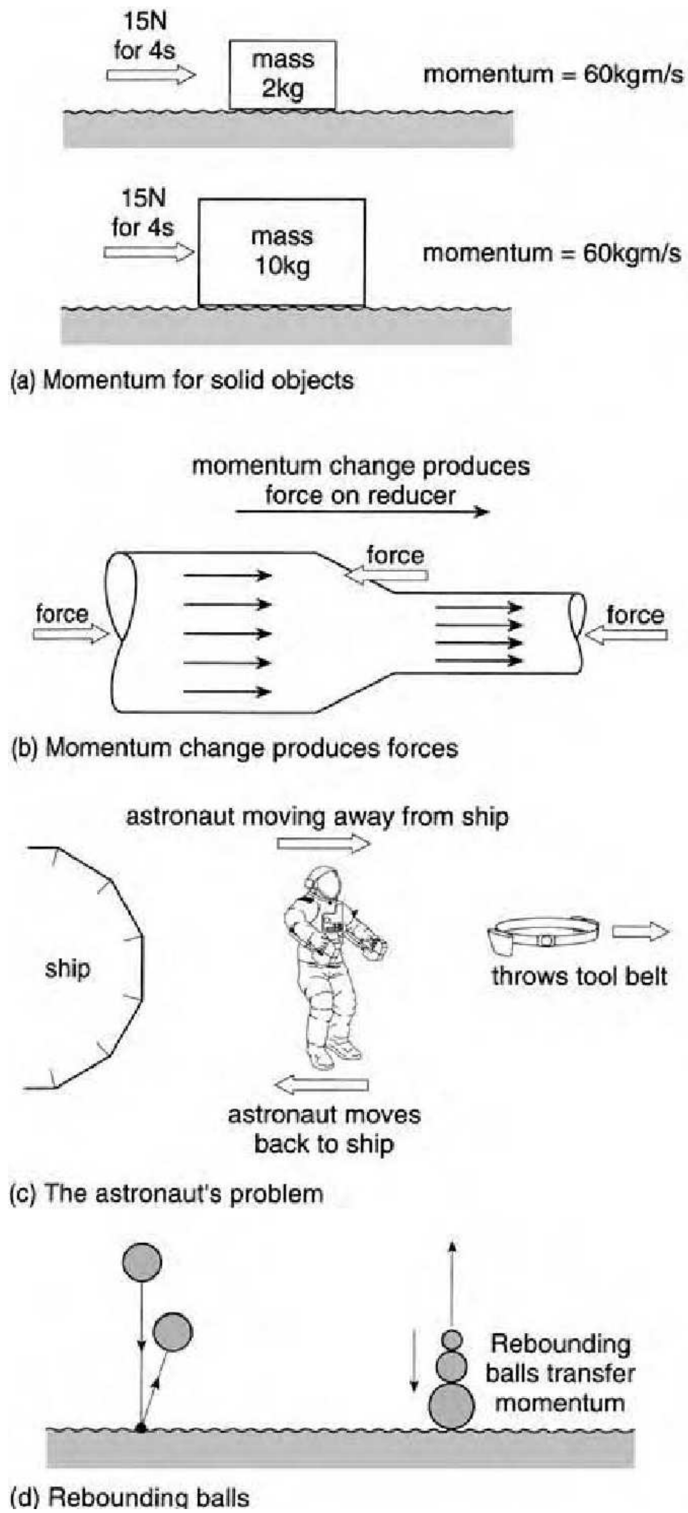
1.7. Momentum

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.



Cha 1 Fig 4 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:

$$\text{Force} = \text{Mass X Acceleration}$$
$$15 = 2 \text{ X } f$$
$$f = 7.5 \text{ m/s2}$$

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:

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

Calculate the momentum of the block:

$$\text{Momentum} = \text{Mass X Velocity}$$
$$\text{Momentum} = 2 \text{ X } 30$$
$$\text{Momentum} = 60 \text{ kgm/s}$$

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:

$$15 = 10 \text{ X } f$$
And so:
$$f = 1.5 \text{ m/s2}$$

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:

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

Now calculate momentum of this block:

$$\text{Momentum} = 10 \text{ X } 6$$
$$= 60 \text{ kgm/s}$$

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:

$$\text{Force X Time} = 15 \text{ X } 4$$
$$\text{Force X Time} = 60 \text{ Ns}$$

But the dimension for Newtons can also be written as kgm/s2. And so:

$$\text{Force X Time} = 60 \text{ kgm/s}$$

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:

Impulse = Momentum

And:

Force X Time = Mass X 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 X Time = Mass X 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:

Force= $\frac{Mass}{Time}\,\frac{Mass}{Time}$ X 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) X 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.

1.8. Properties of Water

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

1.8.1. 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 meter of the material and is a fixed value for the material concerned. For example, the mass density of air is 1.29 kg/m3, steel is 7800 kg/m3 and gold is 19 300 kg/m3.

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

Density (kg/ m^3 m^3) = $\frac{Mass\,(kg)}{volume\,(m^3)}$ $\frac{Mass\,(kg)}{volume\,(m^3)}$

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

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

r= 1000 kg/m3

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 meter of water.

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

Weight Density (kN/m3) = Mass Density (kg/m3) X Gravity Constant (m/s2)

For Water:
Weight Density = 1000 X 9.81
Weight Density = 9810 N/m3 (or 9.81 kN/m3)
Weight Density = 10 kN/m3 (approximately)

Sometimes weight density for water is rounded off by engineers to 10 kN/m3. 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 _*g* to describe the weight density. They are in effect the same but for clarity, ***"rg"*** is used throughout this book.

1.8.2. 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.

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

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:

$$\text{Specific Gravity} = \frac{\text{weight of an object}}{\text{weight of an equal volume of water}} \; \frac{\text{weight of an object}}{\text{weight of an equal volume of water}}$$

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:

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

Table 1.3 Some values of specific gravity.

Material/fluid Specific gravity Comments

Material	*Specific Gravity*	*Comments*
Water	*1.0*	*All other specific gravity measurements are made relative to that of eater*

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 manometer for measuring pressure*

So:

Mass density of mercury = SG of mercury X mass density of water
Mass density of mercury = 13.6 X 1000
Mass density of mercury = 13 600 kg/m3

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.

1.8.3. 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.

Cha 1 Fig 5 Underestanding 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:

$\mu\ \mu$ = 0.00114 kg/ms at a temperature of 15°C

μ = 1.14 X $10^3\ 10^3$ kg/ms

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

1.8.3.1. *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:

Kinematic viscosity (μ μ) = $\frac{viscosity\,(\mu)}{density\,(\rho)}$ $\frac{viscosity\,(\mu)}{density\,(\rho)}$

For water:

n= 1.14 X 10^{-2} 10^{-2} m2/s at a temperature of 15°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} 10^{4} Stokes = 1 m2/s For water: n= 1.14 X 10^{-2} 10^{-2} Stokes

1.8.3.2. *Dinamic Viscosity*

The usual symbol for dynamic viscosity used by mechanical and chemical engineers (as well as fluid dynamicists) is the Greek letter mu (μ), The symbol "η" is also used by chemists, physicists, and the IUPAC.

The SI physical unit of dynamic viscosity is the pascal-second (Pa·s), (equivalent to N·s/m^2, or kg/(m·s)). If a fluid with a viscosity of one Pa·s is placed between two plates, and one plate is pushed sideways with a shear stress of one pascal, it moves a distance equal to the thickness of the layer between the plates in one second.

The "cgs" physical unit for dynamic viscosity is the *poise*[8] (P), named after Jean Louis Marie Poiseuille. It is more commonly expressed, particularly in ASTM standards, as *centipoise* (cP).

Water at 20 °C has a viscosity of 1.0020 cP or 0.001002 kg/(m·s).

1 P = 1 g·cm^{-1}·s^{-1}. 1 Pa·s = 1 kg·m^{-1}·s^{-1} = 10 P.

The relation to the SI unit is

1 P = 0.1 Pa·s, 1 cP = 1 mPa·s = 0.001 Pa·s

1.8.4. 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.

1.8.5. 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. **HIDROSTATICS (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 meter 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:

$$\text{Pressure} = \frac{Force}{Area}\frac{Force}{Area}$$

Introducing the units of measurement:

$$\text{Pressure (kN/}m^2\,m^2\text{)} = \frac{Force\ (kN)}{Area(m^2)}\ \frac{Force\ (kN)}{Area(m^2)}$$

Force is in kilo-Newton's (kN), area is in square meters (m2) and so pressure is measured in kN/m2. Sometimes pressure is measured in *Pascal's* (Pa) in recognition of Blaise Pascal (1620–1662) who clarified much of modern-day thinking about pressure and barometers for measuring atmospheric pressure.

1 Pa = 1 N/m2

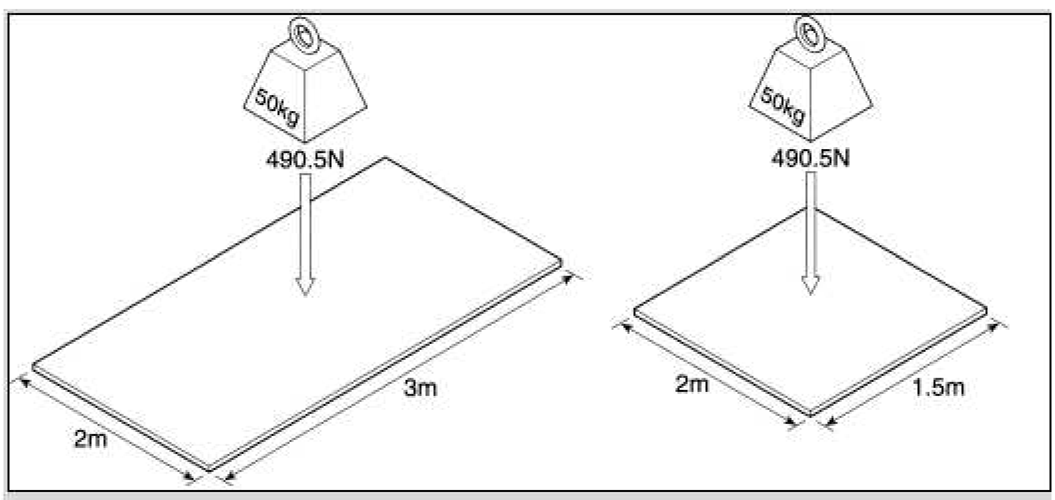
One Pascal is a very small quantity and so kilo-Pascal's are often used so that:

1 kPa = 1 kN/m2

Although it is in order to use Pascal's, kilo-Newton's per square meter 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.
Mass on plate = 50 kg
Weight on the Plate = Mass X Gravity constant
Weight on the Plate = 50 X 9.81 = **490.5 N**
Plate Area = 3 X 2 = 6 m2
Pressure on Plate = $\frac{Force}{Area}$ $\frac{Force}{Area}$ = $\frac{490.5}{6}$ $\frac{490.5}{6}$
Pressure on Plate **= 81.75 N/m2**
When the Plate is reduced to 1.5 m by 2 m:
Plate Area = 1.5 X 2 = 3 m2
Pressure on Plate = $\frac{490.5}{3}$ $\frac{490.5}{3}$
Pressure on Plate **=163.5 N/m2**
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.



Cha 2 Fig 1 Different areas produce different pressures for the smane forces

2.2.1. 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) tires 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)?

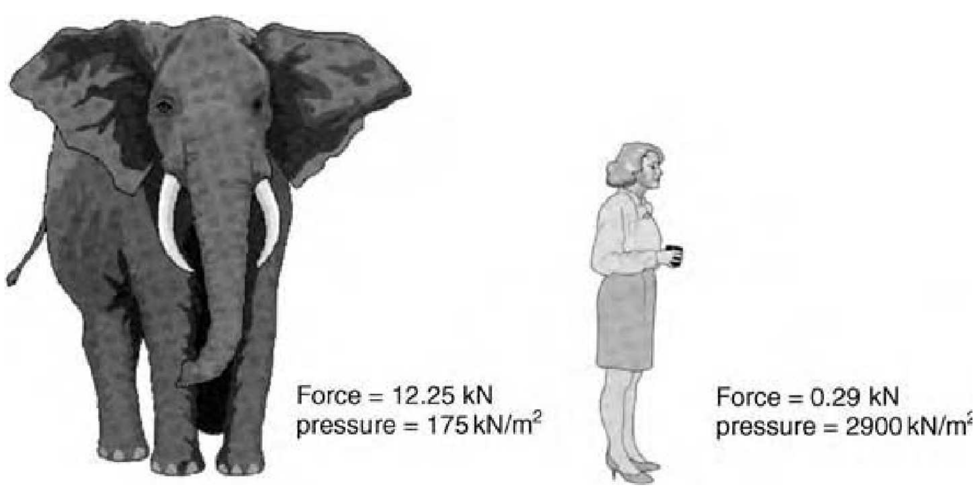
First calculate the pressure under the elephant's foot:

Elephant's mass = 5000 kg

Elephant's weight = 5000 X 9.81

= 49 050 N = 49 kN

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



2.2 Which produces the greater pressure?

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

= 0.07 m2

Pressure under foot = $\frac{force}{area}$ $\frac{force}{area}$ = $\frac{12.5}{0.07} = \frac{12.5}{0.07} =$

= 175 kN/m2

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

Woman's mass = 60 kg

Woman's weight = 589 N = 0.59 kN

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

Area of shoe heel= $\frac{\pi d^2}{4}$ $\frac{\pi d^2}{4}$ = = $\frac{\pi\ 0.1^2}{4}$ $\frac{\pi\ 0.1^2}{4}$
= 0.0001 m2

Pressure under heel = $\frac{Force}{Area}$ $\frac{Force}{Area}$ = $\frac{0.29}{0.0001}$ $\frac{0.29}{0.0001}$
= 2900 kN/m2

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.

2.2.2. 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:

$$p = rgh$$

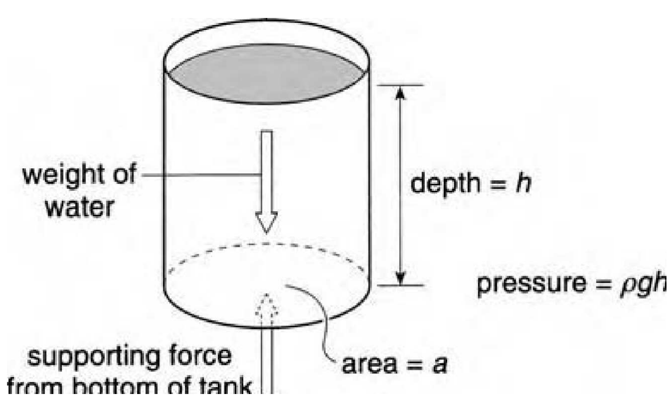
Where *p* is pressure (kN/m2); _ is mass density of water (kN/m3); *g* is gravity constant (m/s2); *h* is depth of water (m).

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).



Cha 2 Fig 2 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:

Volume of water = cross-sectional area X depth
Volume of water = a X h

And so:

Weight of water in tank = volume X density X gravity constant
Weight of water in tank = a X h X r X g

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

Supporting force = pressure X area
Supporting force = p X a
Now put these two forces equal to each other:
p X a = a X h X r X g

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

$$p = \mathrm{r}gh$$

Pressure = mass density X gravity constant X 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:

$p = rgh$

= 1000 X 9.81 X 3.0

= 29 430 N/m2

= 29.43 kN/m2

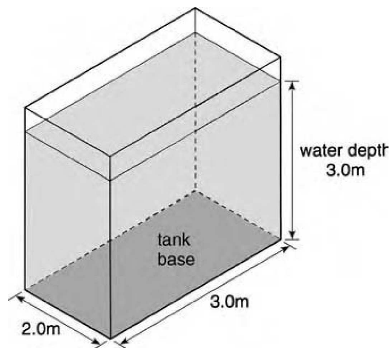
Calculate the force on the tank base using the pressure and the area:

Force = pressure X area

Base area = 3 X 2 = 6 m2

Force = 29.43 X 6

= 176.6 kN

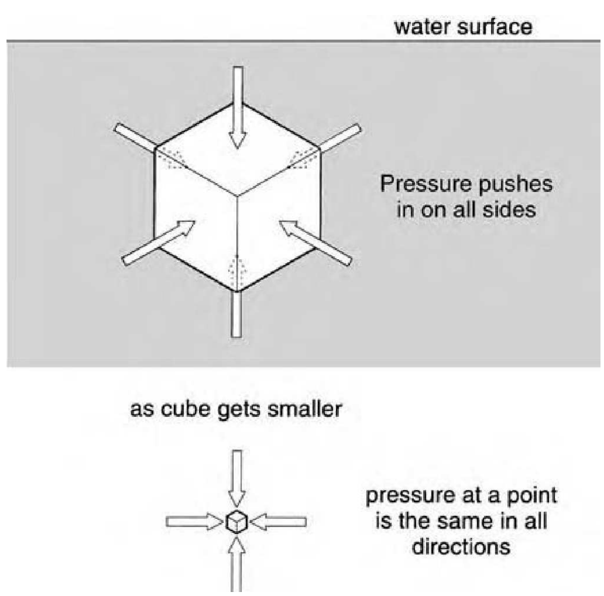


Cha 2 Fig. 3 Calculating force and pressure on tank base.

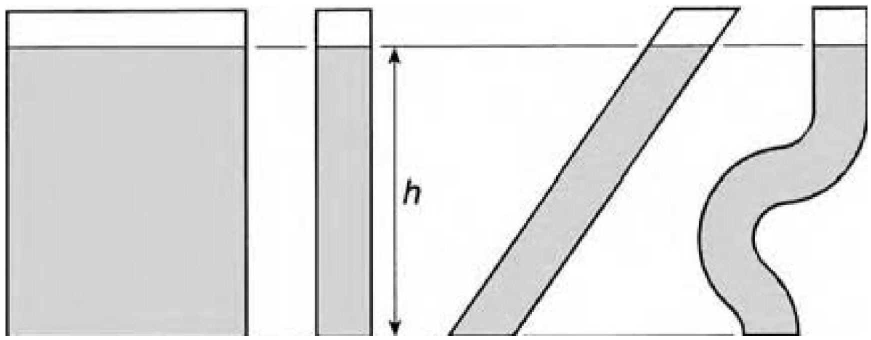
2.2.3. Pressure is Same in all Directions

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



Cha 2 Fig 3 Pressure is the same in all directions.



Cha 2 Fig 4 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.

2.2.4. 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.

2.2.5. Pressure Head

Engineers often refer to pressure in terms of meters of water rather than as a pressure in kN/m2.

So, referring to the pressure calculation in the box, instead of saying the pressure is 29.43 kN/m2 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 gh). It is called the *pressure head* or just *head* and is measured in meters. 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 meters 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 _ is different. So the rule is – say what liquid is being measured, for example, 3 meters head of water or 3 meters 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/m2). Specific gravity (SG) of mercury is 13.6.

First calculate the density of mercury:

ρ (mercury) = ρ (water) X SG (mercury)
ρ (mercury)= 1000 X 13.6
ρ (mercury)= 13 600 kg/m3

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:
29 430 = 13 600 X 9.81 X h
h = 0.22 m of mercury

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.

2.2.6. 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/m2. The average person has a skin area of 2 m2 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 airplane, even though the cabin is pressurized, 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 equalizes with the pressure of the atmosphere.

It was in the 17th century that scientists such as Evangelista Torricelli (1608–1647), a pupil of Galileo Galilee (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 realized 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 equalize 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/m2 – or approximately 10 m head of water.

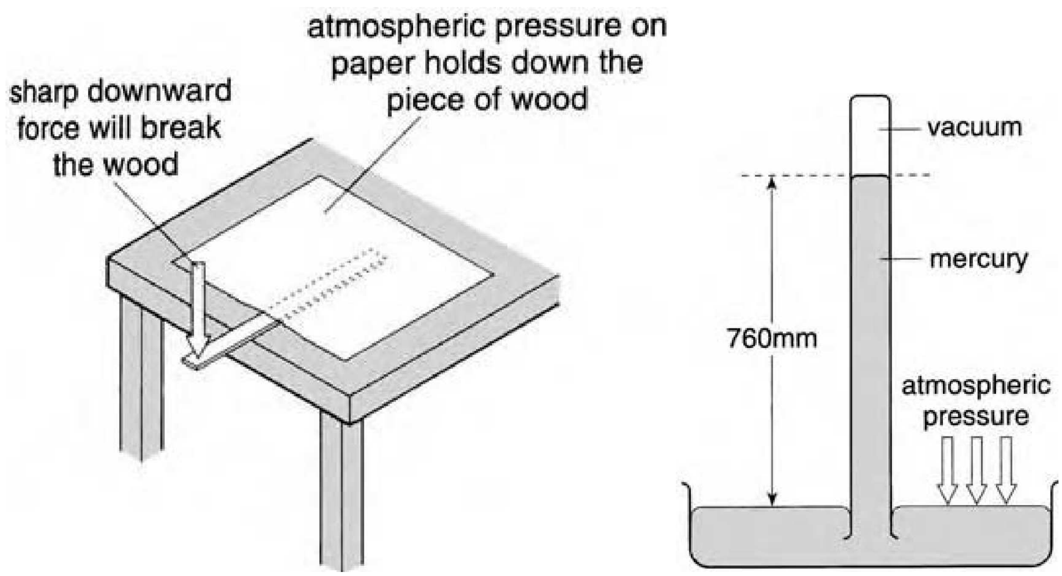
EXAMPLE: EXPERIENCING ATMOSPHERIC PRESSURE

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

If the paper is 1.0 m2 then the force holding down the paper can be calculated as follows:

Force = Pressure X Area

In this case:
Pressure = atmospheric pressure
Pressure = 100 kN/m2
And so:
Force = 100 X 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.



Cha 2 Fig 5 Measuring atmospheric pressure.

2.2.7. 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:

Atmospheric pressure = r*gh*

h is 760 mm and r for mercury is 13 600 kg/m3 (13.6 times denser than water)

So:

Atmospheric pressure = 13 600 X 9.81 X 0.76

Atmospheric pressure = 101 400 N/m2 or 101.4 kN/m2

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

Atmospheric pressure = *rgh*

This time the fluid is water and so:

101 400 = 1000 X 9.81 X *h*

h = 10.32 m

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/m2.

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

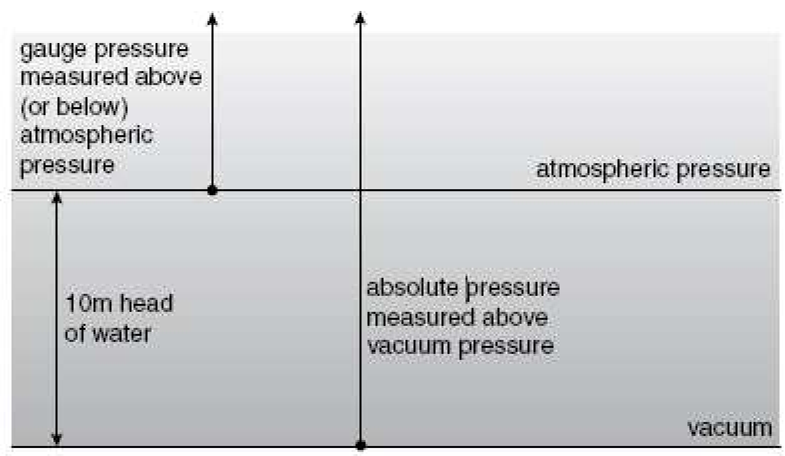
So:

1 millibar = 0.1 kN/m2 = 100 N/m2

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

Atmospheric pressure = 1 bar
or = 100 kN/m2
or = 10 m of water
or = 760 mm of mercury

EXAMPLE: CALCULATING PRESSURE HEAD
A pipeline is operating at a pressure of 3.5 bar. Calculate the pressure in meters head of water.
1 bar = 100 kN/m2 = 100 000 N/m2
And so:
3.5 bar = 350 kN/m2 = 350 000 N/m2
Use the pressure-head equation:
p = *rgh*
350 000 = 1000 X 9.81 X *h*
Calculate head *h*:
h = 35.67 m
Round this off: 3.5 bar = 36 m of water (approximately)



2.11 Gauge and absolute pressures.

Cha 2 Fig 6 Gauge and absolute pressures

2.3. Measuring Pressure

A barometer is a device that uses a liquid in a tube to measure the atmospheric pressure. A glass tube is filled with the liquid and inverted with its open end in a dish of the same liquid. The liquid that is used in real barometers is mercury (Hg), however we will be discussing a "Water Barometer" physlet (see the illustration on the right) to explain the principle of operation

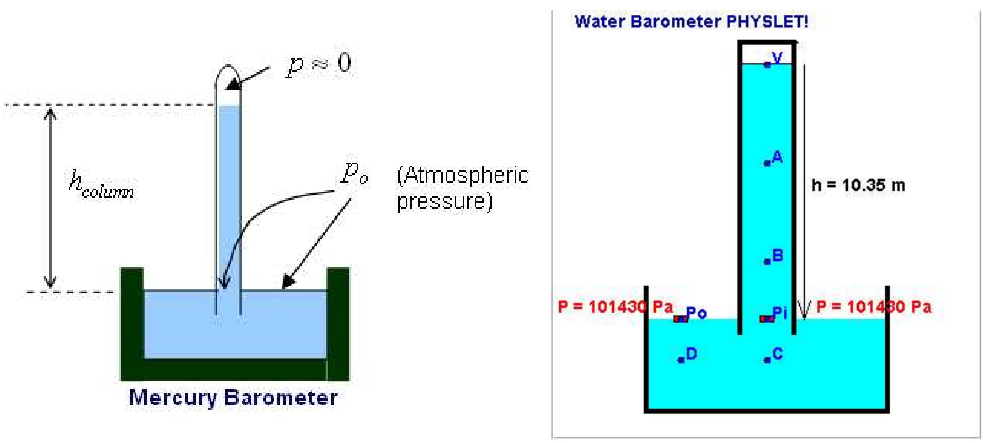
There are two pressure sensors (small red rectangles, one in the middle and one on the left) in the physlet which can be moved to show the pressure at a given point. Move these pressure sensors to measure the pressures at various points. You will find (using the middle sensor) that the pressure at point V (interface between the 'vacuum' and the water), p_V is equal to *zero*. And as you move the sensor down, the pressure at points A , B and C increases linearly with depth according to the formula we derived in the last section for fluid at rest:

$$p = p_V + \rho g h = \rho g h \quad p = p_V + \rho g h = \rho g h$$

Where h h is the depth below the point V.

Within the *same* (continuous) liquid, the pressure is the same at the same depth (level). Check the pressures at Po with that at Pi and at D with that at C (pressures at O and D are measured using the right side sensor). This tells us the height of the column of the liquid h_{column} h_{column} measures the atmospheric pressure as $P_o = P_i = \rho g h_{column}$. Approximate average pressure at sea level is given as 1.01 x 10^5 Pa, which corresponds to 10.3 m of water column in the water barometer!

When mercury is used as the liquid we have the real Barometer and h_{column} for mercury is 760 mm (76.0 cm). Therefore the atmospheric pressure is given as 760 torr (1 torr = 1 mm Hg (millimeter of mercury)).



Cha 2 Fig 7 Mercury Barometer

2.3.1. 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 summaries:

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.

2.3.2. 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/m2 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.

2.3.3. Piezometers

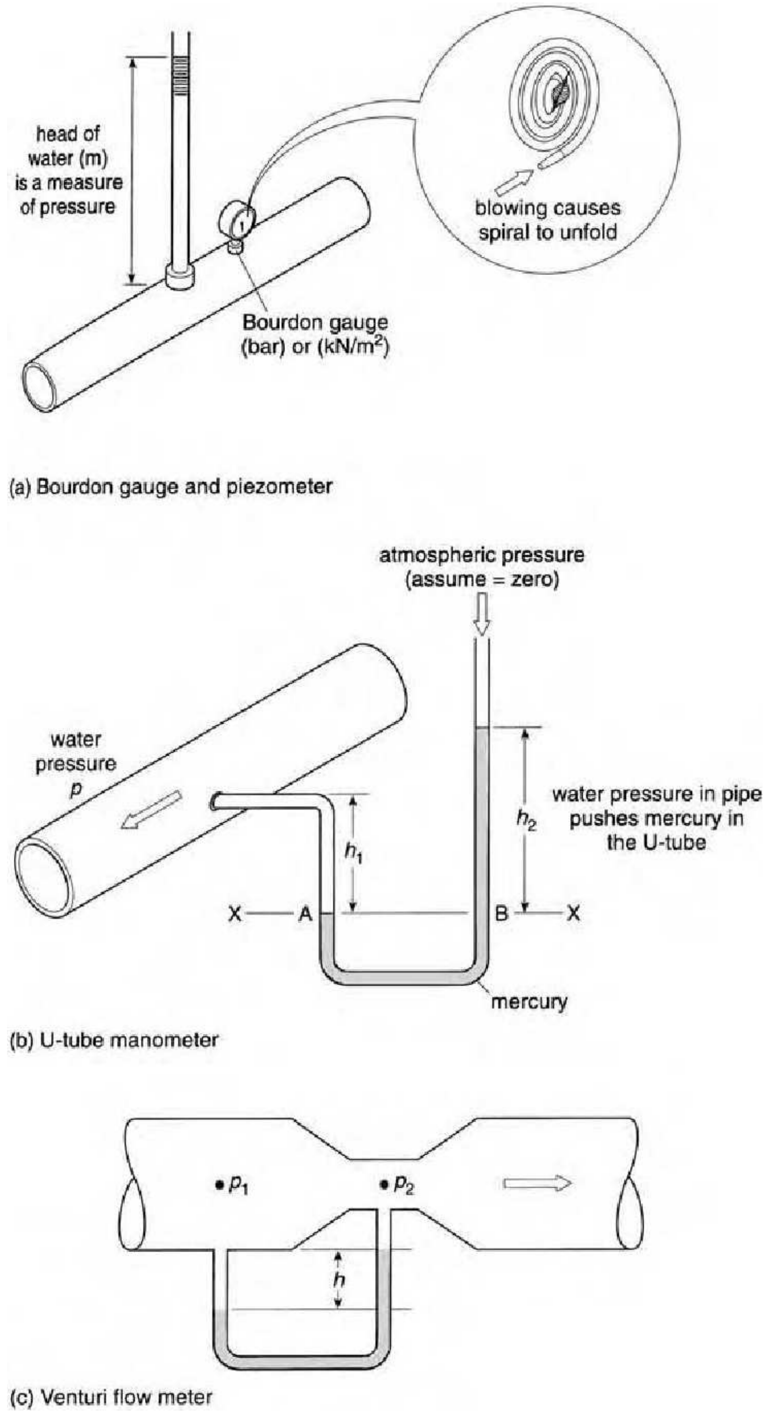
This is another device for measuring pressure. A vertical tube is connected to a pipe so that water can rise up the tube because of the pressure in the pipe (Figure 2.12a). This is called a *piezometer* or *standpipe.* The height of the water column in the tube is a measure of the pressure in the pipe, that is, the pressure head. The pressure in kN/m2 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/m2 in a water pipe.
Using the pressure-head equation:

$p = rgh$
200 000 = 1000 X 9.81 X h

Note in the equation pressure and density are both in N – not kN.
h = 20.4 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.



Cha 2 Fig 8 Piezometer

2.3.4. 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
Pressure at A = *p* + r*(water)g* h_1 h_1
Pressure at A = *p* + (1000 X 9.81 x *h*1)
Pressure at A = *p +* 9810 X *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 = r*(mercury) g h*2 + 0
Pressure at B = 1000 X 13.6 X 9.81 X *h*2
Pressure at B = 133 430 *h*2

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

$p + 9810\ h1 = 133\ 430\ h2$

Rearrange this to determine the pressure in the pipe *p*:

$p = 133\ 430\ h2 - 9810\ h1$

Note that *p* is in N/m2.

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
Pressure at A = *p* + *r*(water)*gh*1
Pressure at A = *p* + 1000 X 9.81 X 1.5
Pressure at B = pressure due to mercury column *h*2
+ atmospheric pressure
Pressure at B =*r*(mercury)*gh*2 + 0
Pressure at B = 1000 X 13.6 X 9.81 X 0.8
Note that as all the pressures are gauge pressures, atmospheric pressure is assumed to be zero.

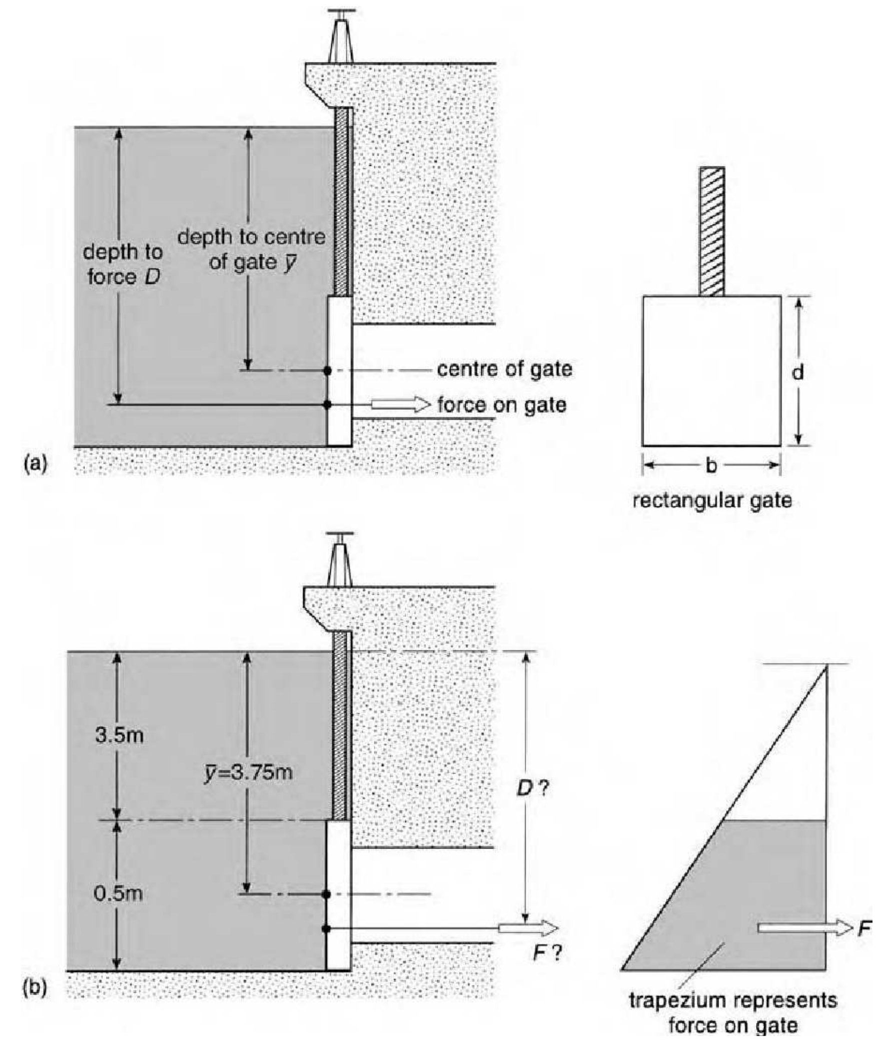
Putting the pressure at A equal to the pressure at B:
$p + 1000 \times 9.81 \times 1.5 = 1000 \times 13.6 \times 9.81 \times 0.8$
Rearrange this to determine p:
$p = (1000 \times 13.6 \times 9.81 \times 0.8) - (1000 \times 9.81 \times 1.5)$
$p = 106\ 732 - 14\ 715$
$p = 92\ 017$ N/m2
$p = 92$ kN/m2

2.4. Forces on Sluice Gates

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

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.


depth to force D
depth to centre of gate $\bar{y}$
centre of gate
force on gate
(a)
d
b
rectangular gate
3.5m
$\bar{y}$=3.75m
0.5m
D ?
F ?
(b)
F
trapezium represents force on gate

Cha 2 Fig 9 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:

$$F=rgay$$

In this case the area is of the gate and 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:

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

Where *d* is depth of gate (m), 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.

$$D=\frac{r^2}{12y}=\frac{r^2}{12y}=\;y$$

For circular gates:

Where "*r*" is radius of the gate (m).

The depth *D* from the water surface to the force *F* must not be confused with. *D* is the depth to the point where the force acts on the gate. It is always greater than. 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 X 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=rgay

Here:
a = area of the gate = 0.5 X 0.5 = 0.25 m2
y = depth from water surface to the centre of the gate
y = 3.5 + 0.25 = 3.75 m

F = 1000 X 9.81 X 0.25 X 3.75

F = 8580 N or 8.58 kN
Next calculate depth from water surface to where force *F* is acting:

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

$= \frac{0.25}{12X3.75} \frac{0.25}{12X3.75} + 3.75$

***D* = 3.76 m**

2.5. Buoyancy (Flotation)

In physics, buoyancy upward acting force exerted by a fluid, that opposes an object's weight. If the object is either less dense than the liquid or is shaped appropriately (as in a boat), the force can keep the object afloat. This can occur only in a reference frame which either has a gravitational field or is accelerating due to a force other than gravity defining a "downward" direction (that is, a non-inertial reference frame). In a situation of fluid statics, the net upward buoyancy force is equal to the magnitude of the weight of fluid displaced by the body [1] This is the force that enables the object to float.

For example if one places a copper ball in a pail of water it will sink, whereas a wooden ball will float. Whether or not a given object will sink or float in a fluid is determined by the buoyant force on the object. The buoyant force is essentially caused by the difference between the pressure at the top of the object, which pushes it downward, and the pressure at the bottom, which pushes it upward. Since the pressure at the bottom is always greater than at the top, every object submerged in a fluid necessarily feels an upward buoyant force. Of course, objects also feel a downward force due to gravity, and the difference between the gravitational force and buoyant force on a submerged object determines whether that object will sink, or rise to the surface. If the weight is greater than the buoyant force, the object sinks, and vice versa. It was Archimedes (supposedly while in his bath), who realized that submerged objects always displace fluid upwards (the level of water in the bathtub rose when Archimedes got in). Thus, he reasoned that the buoyant force on an object must be equal to the weight of fluid that object displaces. If the weight of an object is greater than the weight of displaced fluid, it will sink, whereas if the weight of the object is less than the weight of displaced fluid, it will rise. Moreover, it is evident that the volume of displaced fluid is precisely equal to the volume of the submerged part of the object, so that the difference between the buoyant force and the weight is determined by the relative density of the object and the fluid

2.5.1. Archimedes's 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 realized 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.

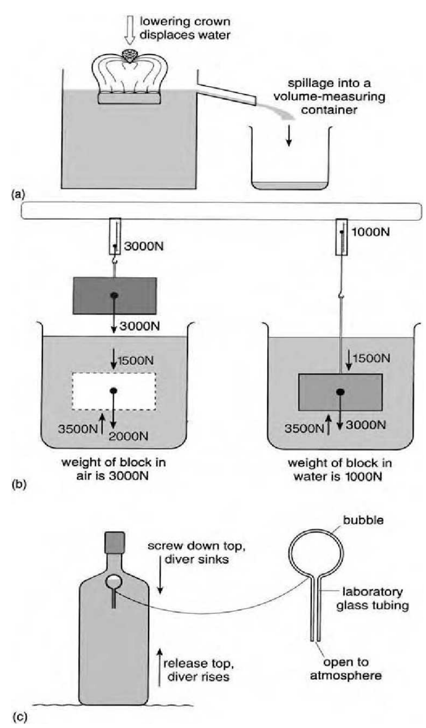
Perhaps the king had two crowns that looked the same in every way but one was made of gold and he suspected that someone had short-changed him by making the other of a mixture of gold and some cheaper metal. The problem that he set Archimedes was to tell him which was the gold one. Weighing them on a 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/m3) 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 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/m3) 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.



Cha 2 Fig 10 Archimedes' Principle

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:

$$\text{Specific gravity}=\frac{weight\ of\ crown}{weight\ loss\ when\ \ inmersed\ in\ water}\frac{weight\ of\ crown}{weight\ loss\ when\ \ inmersed\ in\ water}$$

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:

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

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 m3 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 m3 of water. As water weighs 10 000 N/m3 (approximately) the displaced water weighs 2000 N (i.e. 0.2 m3 X 10 000 N/m3). 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 m3) will occupy when it is lowered into the water (Figure 2.16b). The 'space' is currently occupied by 0.2 m3 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 m3 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.

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, 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.

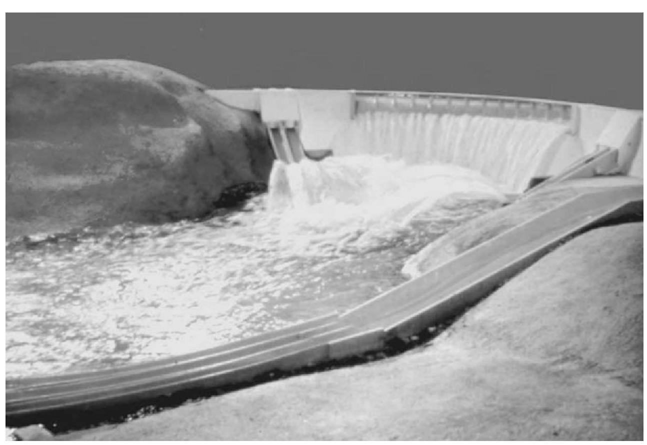
These do have significant influence on the choice of pipe size and so it is important to focus attention on them. This is why engineering is often regarded as much an art as a science.

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



Cha 3 fig 1 Laboratory model of a dam spillway

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 meters (m3) and so discharge is measured in cubic meters per second (m3/s). Alternative units are liters per second (l/s) and cubic meters per hour (m3/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:

$$\text{Discharge (m3/s)} = \frac{Volume\ (m^3)}{Time\ (s)} \frac{Volume\ (m^3)}{Time\ (s)}$$

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:

$$\text{Discharge } (Q) = \text{Velocity } (v) \text{X Area } (a)$$
$$Q = va$$

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).

$$\text{Inflow} = \text{Outflow}$$

And so:

$$Q1 = Q2$$

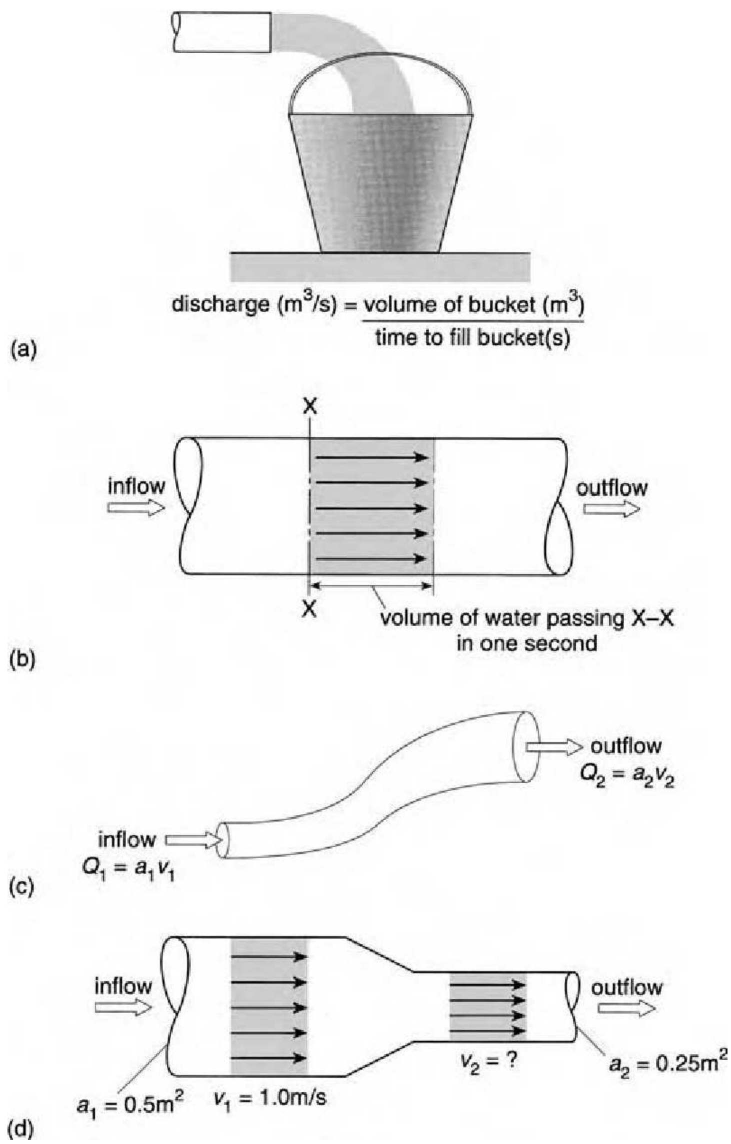
But from the discharge equation:

$$Q = va$$

And so:

$$v1a1 = v2a2$$

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.



Cha 3 Fig 2 Discharge and Continuity.

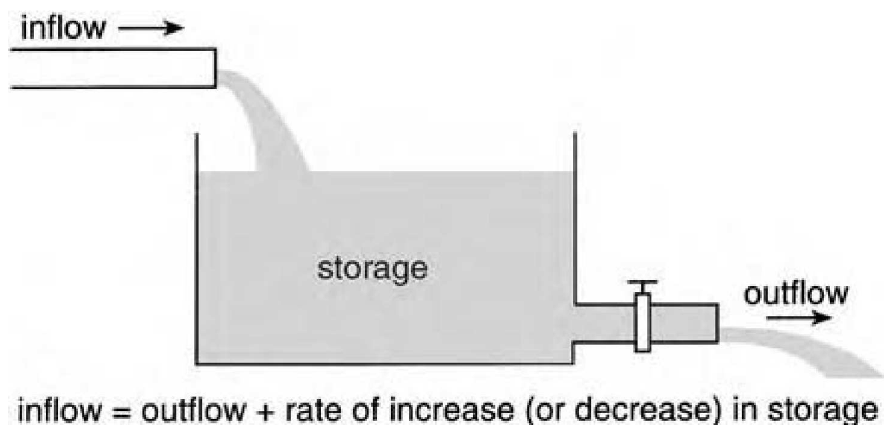
EXAMPLE: CALCULATING VELOCITY USING THE CONTINUITY EQUATION

A pipeline changes area from 0.5 to 0.25 m2 (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:

$v1a1 = v2a2$

$1 \times 0.5 = v2 \times 0.25$
$v2 = 2$ m/s
Note how water moves much faster in the smaller pipe.



Cha 3 Fig 3Continuity 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 meters (m). This make energy look similar to pressure head as both are measured in meters.

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:

Pressure Energy = $\frac{p}{rg}$ $\frac{p}{rg}$ where: r=r

Where *p* is pressure (kN/m2); is mass density (kg/m3); *g is* gravity constant (9.81 m/s2).

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 meters (m).

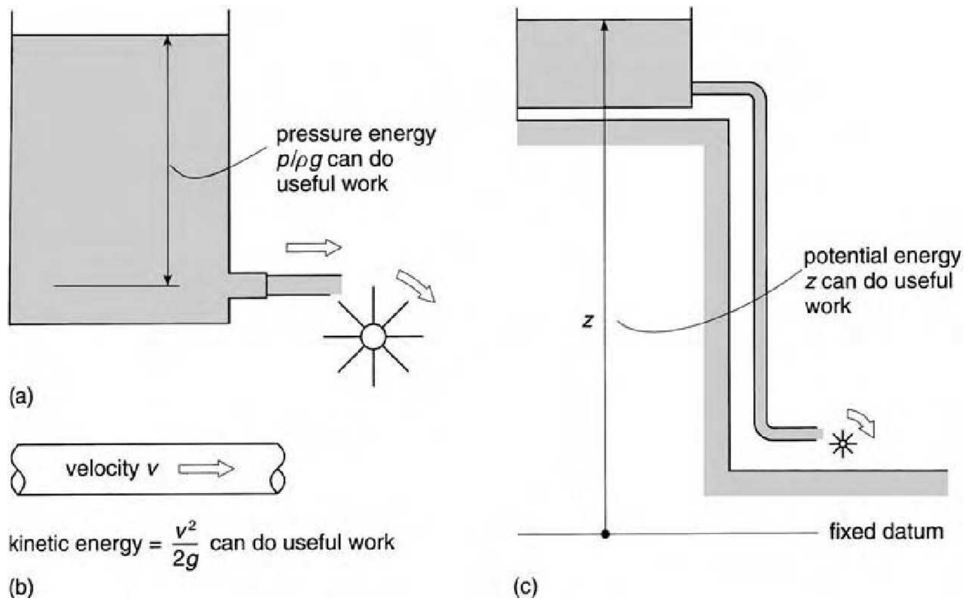
3.5.2. Kinetic Energy

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:

Kinetic Energy = $\frac{v^2}{2g}$ $\frac{v^2}{2g}$

Where *v* is velocity (m/s); *g* is gravity constant (9.81 m/s2).

Kinetic energy is also measured in meters (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.



Cha 3 Fig. 4 Pressure, kinetic and potential energy.

EXAMPLE: CALCULATING KINETIC ENERGY
Calculate the kinetic energy in a pipeline when the flow velocity is 3.7 m/s.

Kinetic Energy = $\frac{v^2}{2g}$ $\frac{v^2}{2g}$

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

This can also be thought of as a velocity head so calculate the equivalent pressure in kN/m2 that would produce this kinetic energy.

To calculate velocity head as a pressure in kN/m2 use:

Pressure =rgh
= 1000 X 9.81 X 0.7
= 6867 N/m2 = 6.87 kN/m2

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 meters above some fixed datum point:

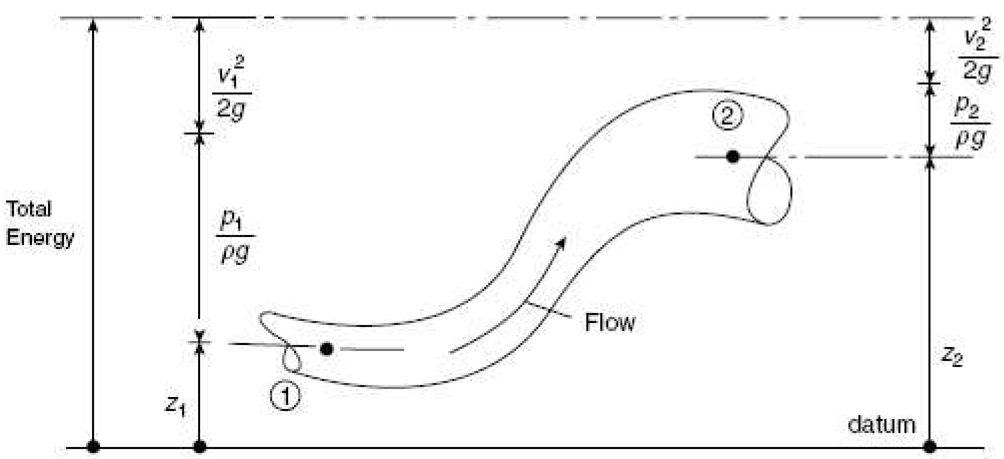
Potential Energy = z

Where "*z*" is the height of the water in meters (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.



Cha 3 Fig 5 Total energy is the same throughout the system

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

Total energy = $\frac{p}{rg}\frac{p}{rg} + \frac{v^2}{2g}\frac{v^2}{2g} + z$

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:

$$\frac{p_1}{rg}\frac{p_1}{rg} + \frac{v_1^2}{2g}\frac{v_1^2}{2g} + z_1\, z_1 = \frac{p_2}{rg}\frac{p_2}{rg} + \frac{v_2^2}{2g}\frac{v_2^2}{2g} + z_2\, z_2 \quad \text{where r = r}$$

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.

3.5.5. Bernoulli's Equation

The Bernoulli equation states that:

$$P + \frac{1}{2}\rho\frac{1}{2}\rho\ V^2\, V^2 + \rho\,\rho\ gh = \text{constant}$$

Where:

$p = p =$ Pressure $\rho = \rho =$ Density V = Velocity h = Elevation g = Gravitational acceleration

Where:

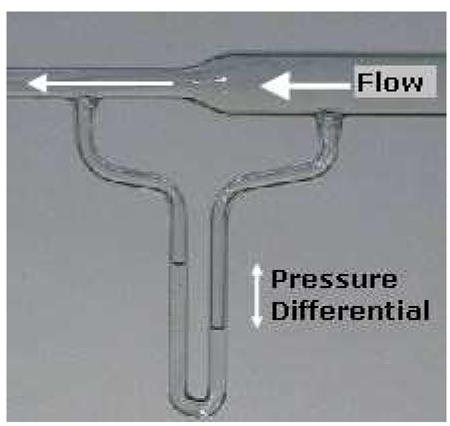
+Points 1 and 2 lie on a streamline,
+ The fluid has constant density,
+ The flow is steady, and
+ There is no friction.

In fluid dynamics, Bernoulli's principle states that for an inviscid flow, an increase in the speed of the fluid occurs simultaneously with a decrease in pressure or a decrease in the fluid's potential energy. Bernoulli's principle is named after the Dutch-Swiss mathematician Daniel Bernoulli who published his principle in his book *Hydrodynamica* in 1738.

Bernoulli's principle can be applied to various types of fluid flow, resulting in what is loosely denoted as Bernoulli's equation. In fact, there are different forms of the Bernoulli equation for different types of flow. The simple form of Bernoulli's principle is valid for incompressible flows and also for compressible flows (e.g. gases) moving at low Mach numbers. More advanced forms may in some cases be applied to compressible flows at higher March numbers (see the derivations of the Bernoulli equation).

Bernoulli's principle can be derived from the principle of conservation of energy. This states that, in a steady flow, the sum of all forms of mechanical energy in a fluid along a streamline is the same at all points on that streamline. This requires that the sum of kinetic energy and potential energy remain constant. If the fluid is flowing out of a reservoir the sum of all forms of energy is the same on all streamlines because in a reservoir the energy per unit mass (the sum of pressure and gravitational potential $\rho g h$) is the same everywhere.

Fluid particles are subject only to pressure and their own weight. If a fluid is flowing horizontally and along a section of a streamline, where the speed increases it can only be because the fluid on that section has moved from a region of higher pressure to a region of lower pressure; and if its speed decreases, it can only be because it has moved from a region of lower pressure to a region of higher pressure. Consequently, within a fluid flowing horizontally, the highest speed occurs where the pressure is lowest, and the lowest speed occurs where the pressure is highest



Cha 3 Fig 6 A flow of air into a venturi meter. The kinetic energy increases at the expense of the fluid pressure, as shown by the difference in height of the two columns of water

3.5.6. Some Useful Applications of the Energy Equation

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

3.5.6.1. *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.

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

$$\frac{p_1}{rg}\,\frac{p_1}{rg} + \frac{v_1^2}{2g}\,\frac{v_1^2}{2g} + Z_1\;Z_1 = \frac{p_2}{rg}\,\frac{p_2}{rg} + \frac{v_2^2}{2g}\,\frac{v_2^2}{2g} + Z_2\;Z_2 \quad \text{where r = r}$$

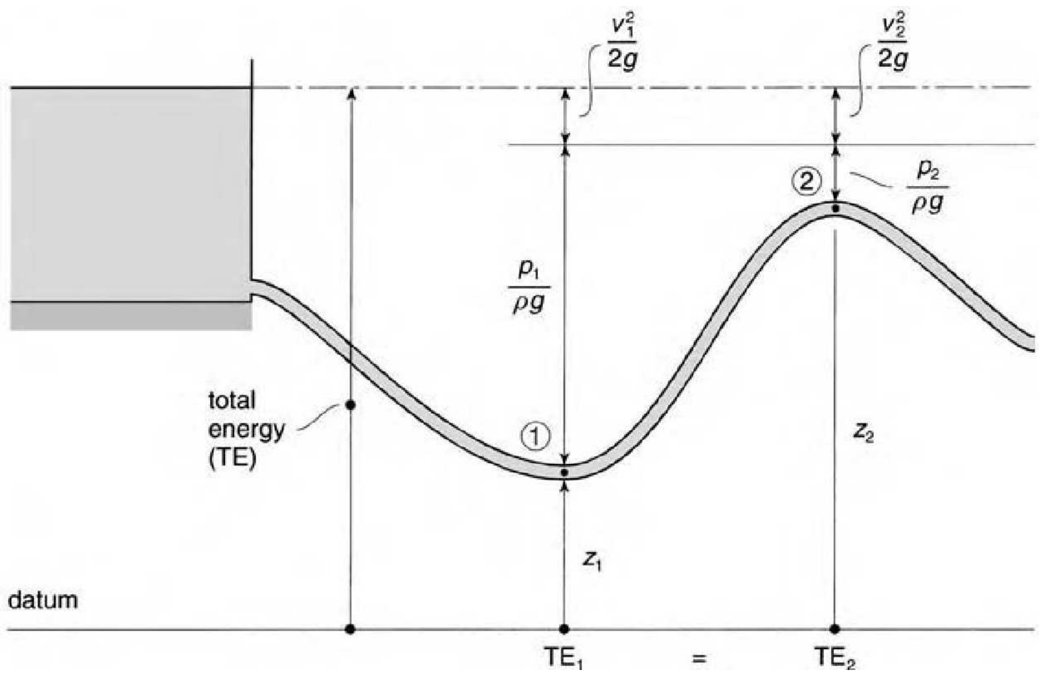
"*z*1"and "*z*2" are measured from some chosen horizontal datum.

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:

$$\frac{p_1}{rg}\,\frac{p_1}{rg} + Z_1\;Z_1 = \frac{p_2}{rg}\,\frac{p_2}{rg} + Z_2\;Z_2 \quad \text{where r = r}$$

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

$$\frac{p_1}{rg}\,\frac{p_1}{rg} - \frac{p_2}{rg}\,\frac{p_2}{rg} = Z_2\;Z_2 - Z_1\;Z_1 \quad \text{where r = r}$$



Cha 3 Fig 7Pressure 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 meters) 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/m2 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

$\frac{p_1}{rg}\frac{p_1}{rg} + \frac{v_1^2}{2g}\frac{v_1^2}{2g} + z_1\ z_1 = \frac{p_2}{rg}\frac{p_2}{rg} + \frac{v_2^2}{2g}\frac{v_2^2}{2g} + z_2\ z_2$ where r = r 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:

$$\frac{p_1}{rg}\frac{p_1}{rg} + z_1\ z_1 = \frac{p_2}{rg}\frac{p_2}{rg} + z_2\ z_2$$ where r = r

Rearranging this gives:

$$\frac{p_1}{rg}\frac{p_1}{rg} - \frac{p_2}{rg}\frac{p_2}{rg} = z_2\ z_2 - z_1\ z_1$$ where r = r

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

$z2 - z1 = 7.5$ m

This means that:

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

And so:

$p1 - p2 = 1000 \times 9.81 \times 7.5 = 73\ 575$ N/m2 = 73.6 kN/m2

Known pressure at point 1 = 150 kN/m2

And so:
Pressure at point 2 = 150 - 73.6 = 76.4 kN/m2

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

3.5.6.2. *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:

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

Assuming the flow is horizontal:

$z1 = zs$

As the water stops:

$$V_s \, V_s = 0$$

And so:

$$\frac{p_1}{rg}\,\frac{p_1}{rg} + \frac{v_1^2}{2g}\,\frac{v_1^2}{2g} = \frac{p_s}{rg}\,\frac{p_s}{rg}$$ Where r = r

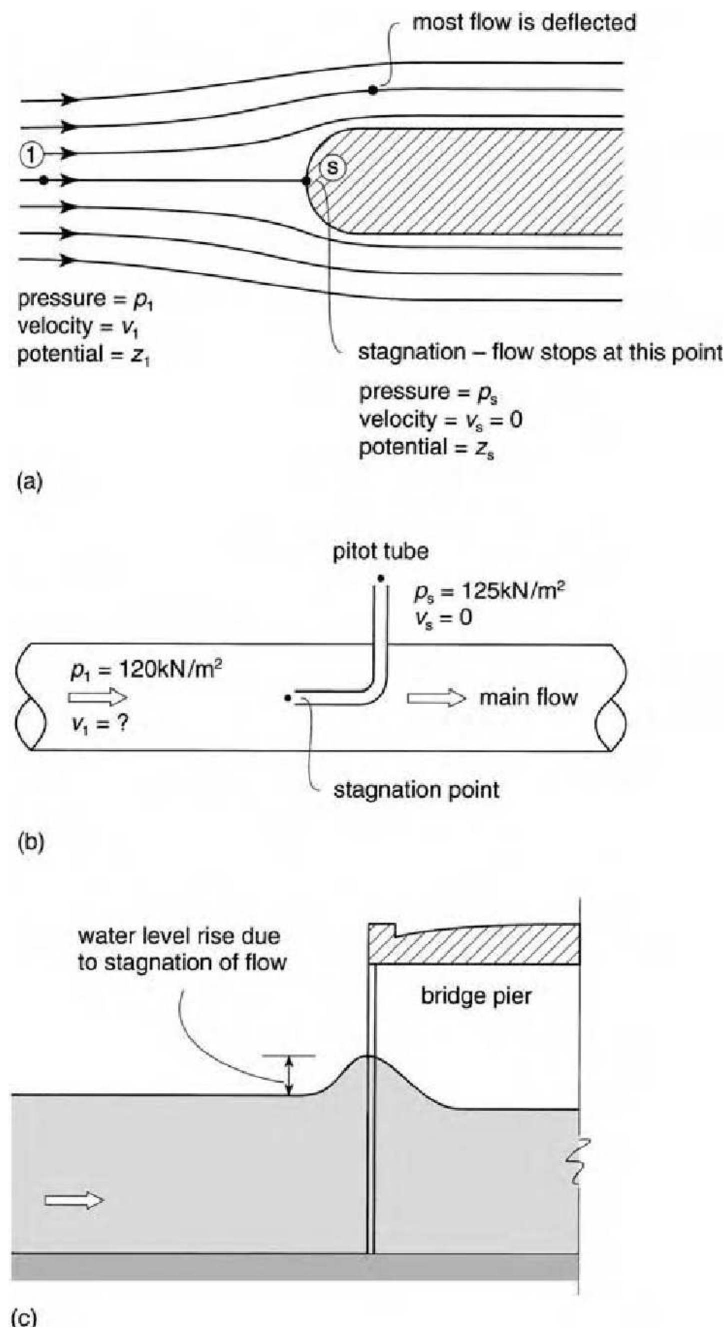
Rearranging this equation to bring all the velocity and pressure terms together:

$$\frac{v_1^2}{2g}\,\frac{v_1^2}{2g} = \frac{p_s}{rg}\,\frac{p_s}{rg} - \frac{p_1}{rg}\,\frac{p_1}{rg}$$ **W**here r = r

Rearranging it again for an equation for velocity *v*1:

$$V_1 = V_1 = \sqrt{2\left(\frac{p_s - p_1}{\rho}\right)}\,\sqrt{2\left(\frac{p_s - p_1}{\rho}\right)}$$

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 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.



Cha 3 Fig 8 *Measuring Velocity*

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.

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/m2 and the pitot pressure is 125 kN/m2 (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:

$$\frac{p_1}{rg}\frac{p_1}{rg} + \frac{v_1^2}{2g}\frac{v_1^2}{2g} + Z_1\,Z_1 = \frac{p_s}{rg}\frac{p_s}{rg} + \frac{v_s^2}{2g}\frac{v_s^2}{2g} + Z_s\,Z_s \quad \text{where r= r}$$

At the stagnation point:

$v_s\ v_s = 0$

And as the system is horizontal:

$Z_1\ Z_1 = Z_s\ Z_s = 0$

This reduces the energy equation to:

$$\frac{p_1}{rg}\frac{p_1}{rg} + \frac{v_1^2}{2g}\frac{v_1^2}{2g} = \frac{p_s}{rg}\frac{p_s}{rg} \quad \text{where r= r}$$

All the values in the equation are known except for *v*1 so calculate *v*1:

$$\frac{120000}{1000X9.81}\ \frac{120000}{1000X9.81} + \frac{V_1^2}{2\,x\,9.81} = \frac{125000}{100\,x\,9.81}\ \frac{V_1^2}{2\,x\,9.81} = \frac{125000}{100\,x\,9.81}$$

$$12.23 + \frac{V_1^2}{2g}\frac{V_1^2}{2g} = 12.74$$

$$\frac{V_1^2}{2x\,9.81}\ \frac{V_1^2}{2x\,9.81} = 12.74$$

$$V_1 = \sqrt{2\,x\,9.81\,x\,0.51}\ V_1 = \sqrt{2\,x\,9.81\,x\,0.51}$$

$V_1 =\ V_1 =$ 3.16 m/s

3.5.6.3. *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:

$$\frac{p_1}{rg}\frac{p_1}{rg} + \frac{v_1^2}{2g}\frac{v_1^2}{2g} + z_1\, z_1 = \frac{p_0}{rg}\frac{p_0}{rg} + \frac{v_0^2}{2g}\frac{v_0^2}{2g} + z_0\, z_0 \quad \text{where r= r}$$

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.

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:

$$z_1\, z_1 = \frac{V_0^2}{2g}\frac{V_0^2}{2g}$$

Rearranging this equation:

$$\frac{V_0^2}{2g}\frac{V_0^2}{2g} = z_1 - z_0\, z_1 - z_0$$

Put:

$$z_1 - z_0\, z_1 - z_0 \;\_\; h$$

Now rearrange again to obtain an equation for *v*0:

$$V_0\, V_0 = \sqrt{2gh}\,\sqrt{2gh}$$

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. 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:

$$Q = a\,v$$

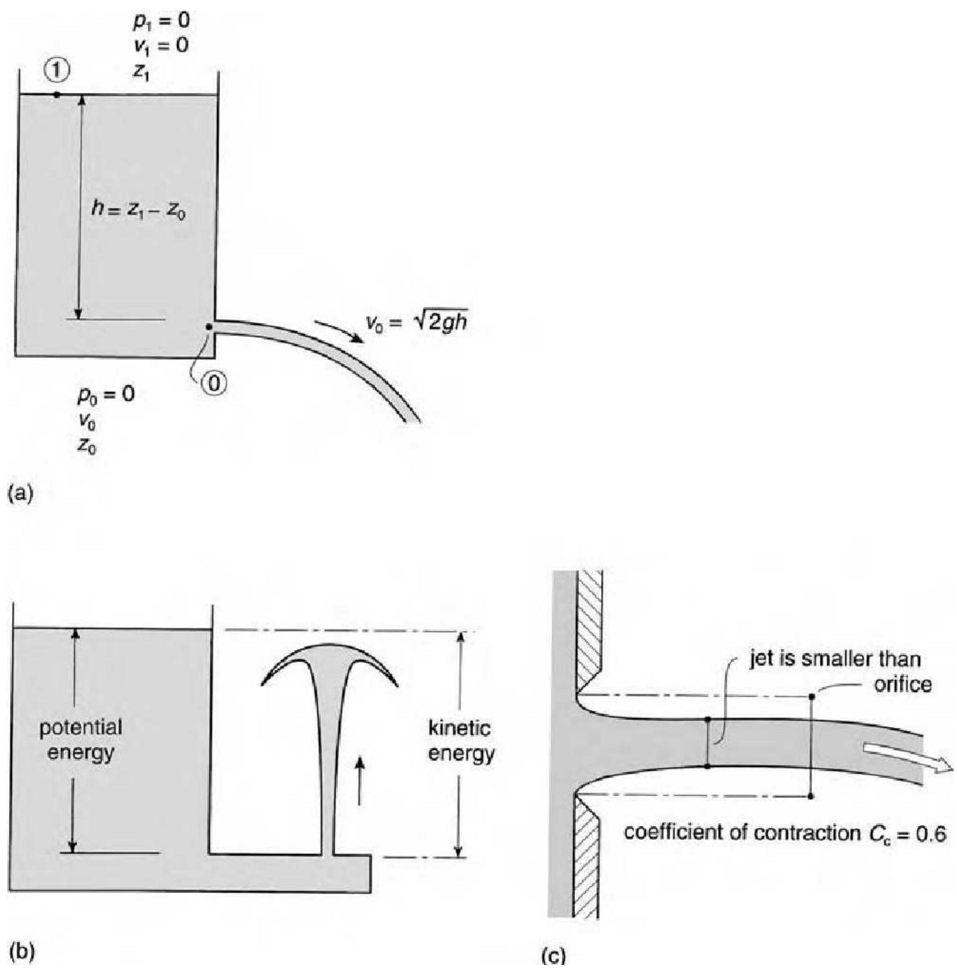
So:

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

The area of the orifice 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* Cc. This has a value of approximately 0.6. So the discharge formula now becomes:

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

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.



Cha 3 Fig 9 Flow through orifices.

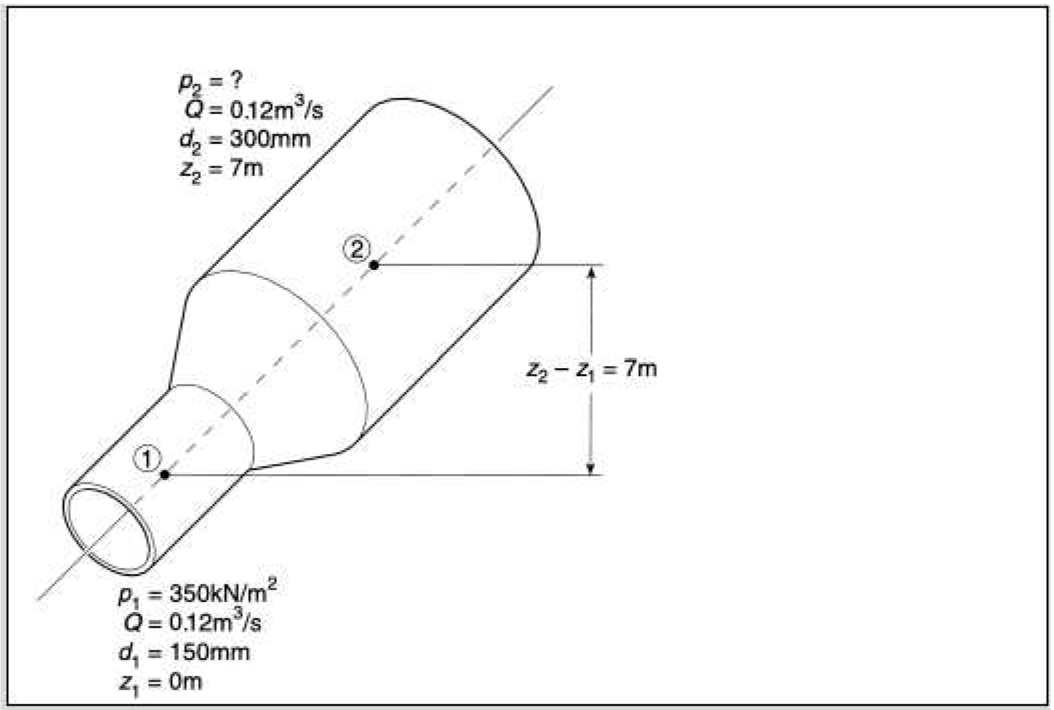
3.5.6.4 *Pressure and Velocity Changes in a Pipe*

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 m3/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/m2.
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:

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



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:*

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

And so:

$$V_1\ V_1 = \frac{Q}{a_1}\frac{Q}{a_1} \quad \text{and} \quad V_2\ V_2 = \frac{Q}{a_2}\frac{Q}{a_2}$$

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

$$a1 = \frac{\pi d_1^2}{4}\frac{\pi d_1^2}{4} = \frac{\pi 0.15^2}{4}\frac{\pi 0.15^2}{4}\ 0.018\text{ m2}$$

$$a2= \frac{\pi d_2^2}{4}\frac{\pi d_2^2}{4} = \frac{\pi\, 0.3^2}{4}\frac{\pi\, 0.3^2}{4} \quad 0.07 \text{ m2}$$

Now calculate the velocities:

$$v1= \frac{Q}{a}\frac{Q}{a} = \frac{0.120}{0.018}\frac{0.120}{0.018} = 6.67 \text{ m / s}$$

$$v2= \frac{Q}{a}\frac{Q}{a} = \frac{0.120}{0.07}\frac{0.120}{0.07} = 1.71 \text{ m / s}$$

Putting all the known values into the energy equation:

$$\frac{350000}{1000 X 9.81}\frac{350000}{1000 X 9.81} + \frac{6.67^2}{2X9.81}\frac{6.67^2}{2X9.81} + 0 = \frac{p_2}{rg}\frac{p_2}{rg} + \frac{1.71^2}{2 X 9.81}\frac{1.71^2}{2 X 9.81} + 7$$ Where r= r

Note although pressures are quoted in kN/m2 it is less confusing to work all calculations in N/m2 and then convert back to kN/m2. The equation simplifies to:

$35.68 + 2.26 = \frac{p_2}{rg}\frac{p_2}{rg} + 0.15 + 7$ Where r= r

Rearranging this equation for *p*2:

$\frac{p_2}{rg}\frac{p_2}{rg} = 35.68 + 2.26 - 0.15 - 7$ Where r= r

= 30.8 m head of water

To determine this head as a pressure in kN/m2 use the pressure-head equation:

Pressure :

Presión = rgh Where r= r

p2= 1000 X 9.81 X 30.8

p2= 302,000 N/m2 = 302 kN/m2

3.5.6.5 *Meters Venturi*

The Venturi effect is the reduction in fluid pressure that results when a fluid flows through a constricted section of pipe. The Venturi effect is named after Giovanni Battista Venturi (1746 1822), an Italian physicist.

According to the laws governing fluid dynamics, a fluid's velocity must *increase* as it passes through a constriction to satisfy the conservation of mass, while its pressure must *decrease* to satisfy the conservation of energy. Thus any gain in kinetic energy a fluid may accrue due to its increased velocity through a constriction is negated by a drop in pressure. An equation for the drop in pressure due to the Venturi effect may be derived from a combination of Bernoulli's principle and the continuity equation.

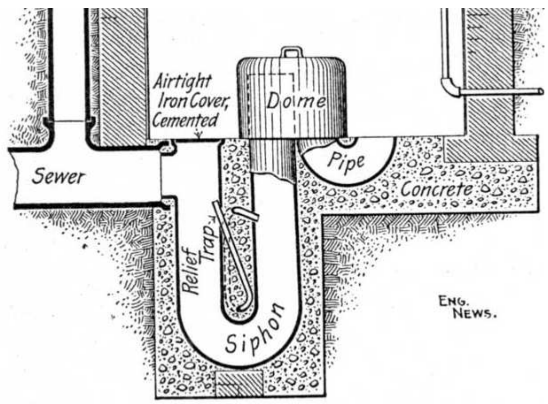
The limiting case of the Venturi effect is when a fluid reaches the state of choked flow, where the fluid velocity approaches the local speed of sound. In choked flow the mass flow rate will not increase with a further decrease in the downstream pressure environment. However, mass flow rate for a compressible fluid can increase with increased upstream pressure, which will increase the velocity of the fluid through the constriction (though the density will remain constant). This is the principle of operation of a de Laval nozzle. Referring to the diagram to the right, using Bernoulli's equation in the special case of incompressible flows (such as the flow of water or other liquid, or low speed flow of gas), the theoretical pressure drop ($p_1 - p_2$) at the constriction would be given by:

$$\frac{\rho}{2}(v_2^2 - v_1^2)$$

Where ρ is the density of the fluid, v_1 is the (slower) fluid velocity where the pipe is wider, v_2 is the (faster) fluid velocity where the pipe is narrower (as seen in the figure). This assumes the flowing fluid (or other substance) is not significantly compressible - even though pressure varies, the density is assumed to remain approximately constant.

3.5.6.6 *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 . 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. This is because the pressure in the section of pipe above the hydraulic gradient is negative.

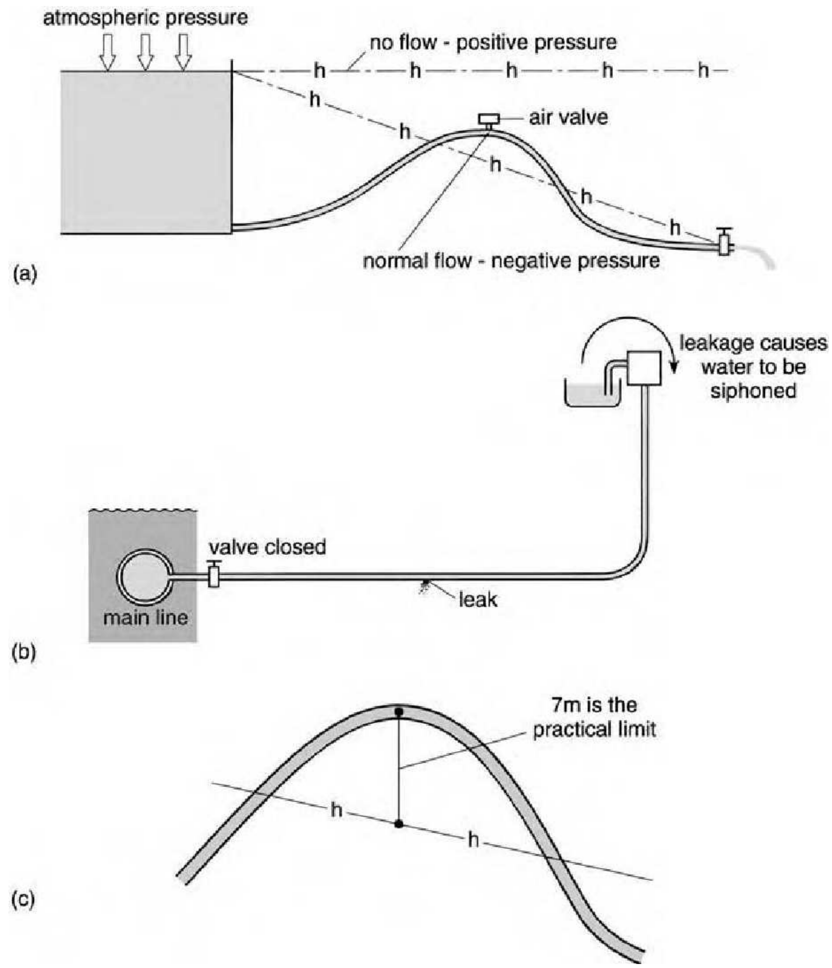


Automatic Siphon for Flush Tank.
Berry Flush Tank Co., Iowa City, Ia., Makers.

Cha. 3 Fig. 10 Siphons

<u>*How they work*</u>

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.



Cha 3 Fig 11 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:

Gauge pressure = 7 m head Absolute pressure = atmospheric pressure - gauge pressure = 10 - 7 = 3 m head absolute

3.5.6.7 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.

3.6 Momentum

In classical mechanics, momentum is the product of the mass and velocity of an object ($p = mv$). Like velocity, momentum is a vector quantity, possessing a direction as well as a magnitude. Momentum is a conserved quantity (law of conservation of linear momentum), meaning that if a closed system is not affected by external forces, its total momentum cannot change. Momentum is sometimes referred to as linear momentum to distinguish it from the related subject of angular momentum.

Although originally expressed in Newton's Second Law, the conservation of momentum also holds in special relativity and, with appropriate definitions, a (generalized) momentum conservation law holds in electrodynamics, quantum mechanics, quantum field theory, and general relativity. In relativistic mechanics, non-relativistic momentum is further multiplied by the Lorentz factor.

Linear momentum of a particle

Newton's apple in Einstein's elevator. In person A's frame of reference, the apple has non-zero velocity and momentum. In the elevator's and person B's frames of reference, it has zero velocity and momentum.

If an object is moving in any reference frame, then it has momentum in that frame. It is important to note that momentum is frame dependent. That is, the same object may have a certain momentum in one frame of reference, but a different amount in another frame. For example, a moving object has momentum in a reference frame fixed to a spot on the ground, while at the same time having 0 momentum in a reference frame attached to the object's center of mass.

The amount of momentum that an object has depends on two physical quantities: the mass and the velocity of the moving object in the frame of reference. In physics, the usual symbol for momentum is a bold p (bold because it is a vector); so this can be written

$$P = mv$$

Where p is the momentum, *m* is the mass and v is the velocity.

Example: a model airplane of 1 kg traveling due north at 1 m/s in straight and level flight has a momentum of 1 kg·m/s due north measured from the ground. To the dummy pilot in the cockpit it has a velocity and momentum of zero.

According to Newton's second law, the rate of change of the momentum of a particle is proportional to the resultant force acting on the particle and is in the direction of that force. The derivation of force from momentum is given below.

$$\sum F = \frac{dp}{dt}\frac{dp}{dt} = m\frac{dv}{dt}\frac{dv}{dt} = v\frac{dm}{dt}\frac{dm}{dt}$$

Given that mass is constant, the second term of the derivative is zero ($v\frac{dm}{dt}=0$). We can therefore write the following:

$$\sum F = ma$$

Or just simply

$$F = ma$$

Where "F" is understood to be the net force.

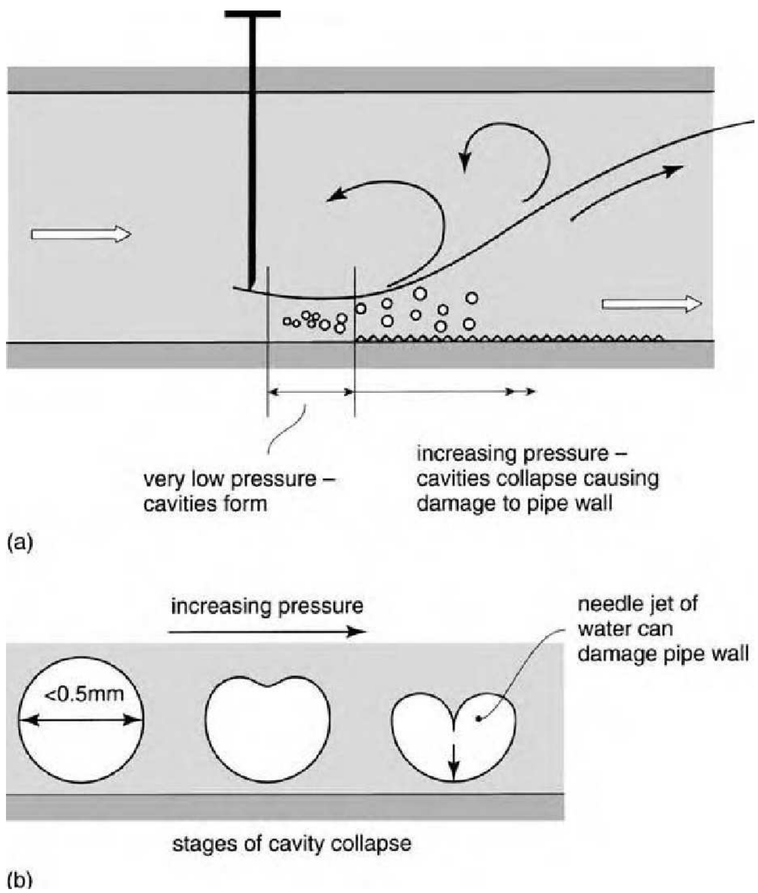
Example: a model airplane of 1 kg accelerates from rest to a velocity of 1 m/s due north in 1 s. The thrust required to produce this acceleration is 1 newton. The change in momentum is 1 kg·m/s. To the dummy pilot in the cockpit there is no change of momentum. Its pressing backward in the seat is a reaction to the unbalanced thrust, shortly to be balanced by the drag.

3.7 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

3.7.1. 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



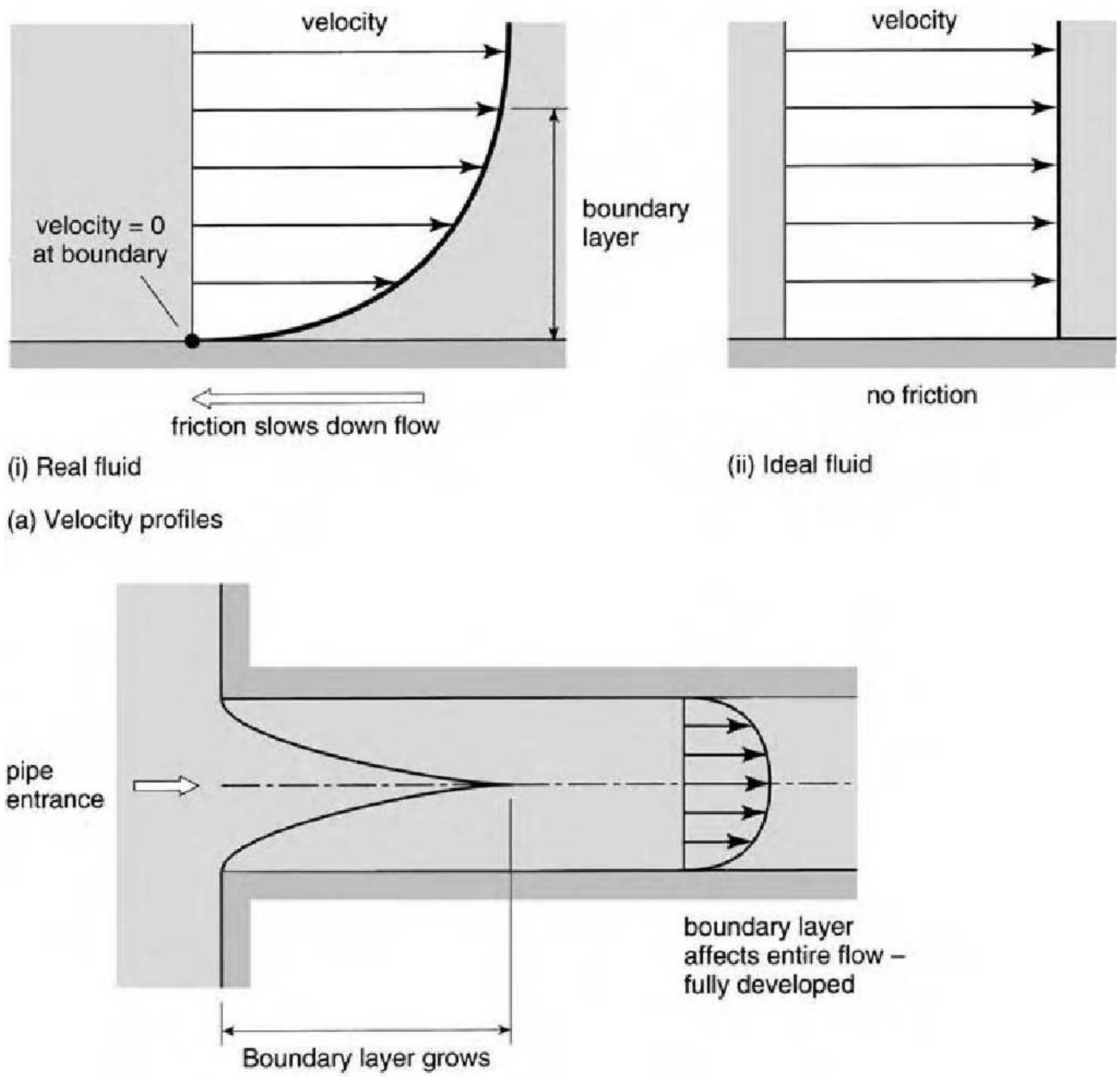
Cha 3 Fig 12Dangers of cavitation.

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 meters until it meets in the middle. From this point onwards the pipe boundary



Cha 3 Fig 13 Boundary effects.

influences the entire flow in the pipe. In channels the boundary effects of the bed and sides similarly grow over a few meters 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.

3.8. 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.

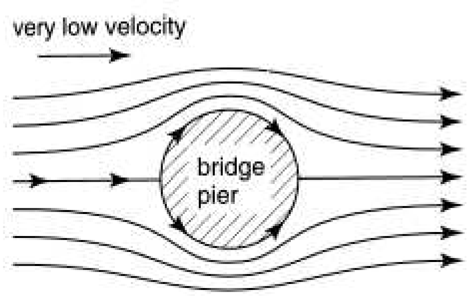
Stokes also demonstrated that for any object dropped in a fluid (or a stationary object placed in a flowing fluid which is essentially the same) there were two types of drag: *surface drag* or skin friction which resulted from friction between the fluid and the object, and *form drag* which resulted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

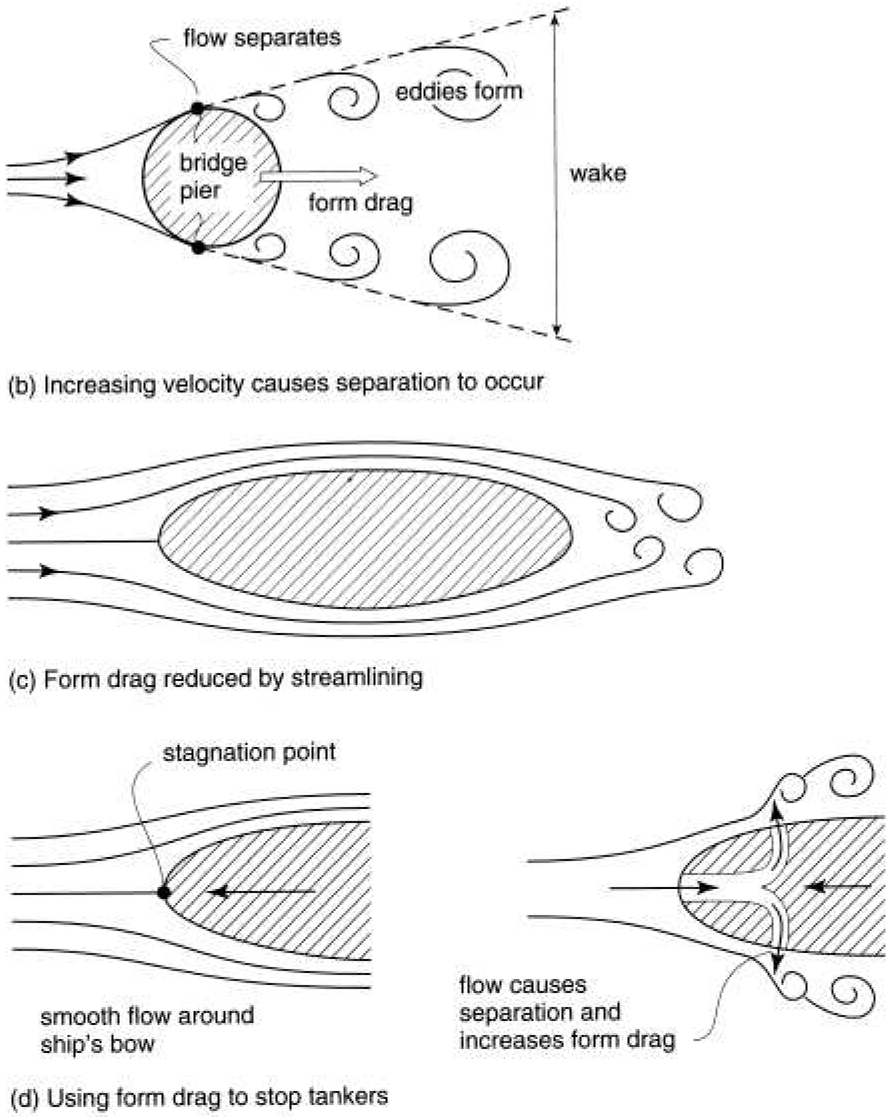
When the velocity is very low, the flow moves around the pier as shown in the next figures.

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. 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*. 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



(a) Surface drag only – no form drag



Cha 3 Fig 14

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*. 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 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:

Drag Force = $\frac{1}{2}$ $\frac{1}{2}$ Cra v^2 v^2

Where is fluid density (kg/m3), *a* is the cross-sectional area (m2), *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.

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!

4.1.1. 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* m3/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.

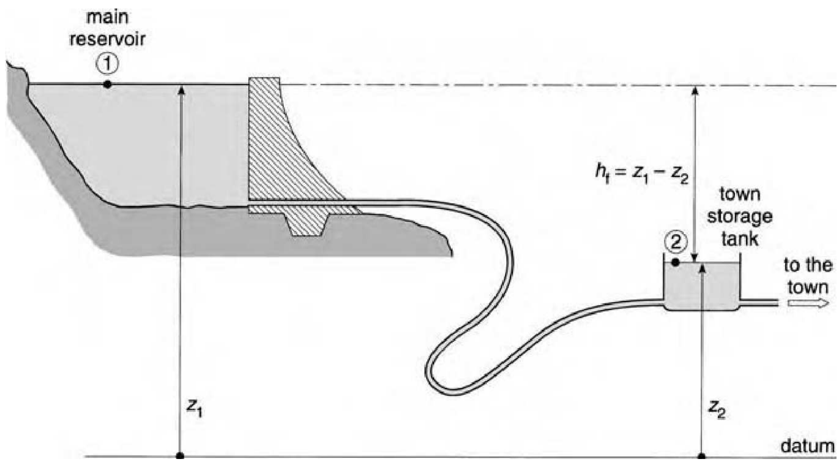
A formula to calculate pipe size would be ideal. However, to get there we first need to look at the energy available to 'push' water through the system, so the place to start is the energy equation.

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:

$$\frac{p_1}{rg}\frac{p_1}{rg} + \frac{v_1^2}{2g}\frac{v_1^2}{2g} + z_1\ z_1 = \frac{p_2}{rg}\frac{p_2}{rg} + \frac{v_2^2}{2g}\frac{v_2^2}{2g} + z_2\ z_2 + h_f\ h_f \qquad \text{where } = r$$

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



Cha 4 fig 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 $p2$ is zero as well. The water velocities $v1$ and $v2$ 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 $z1$ and $z2$ and the energy loss term hf. So the energy equation simplifies down to:

$$\boxed{hf = z1 - z2}$$

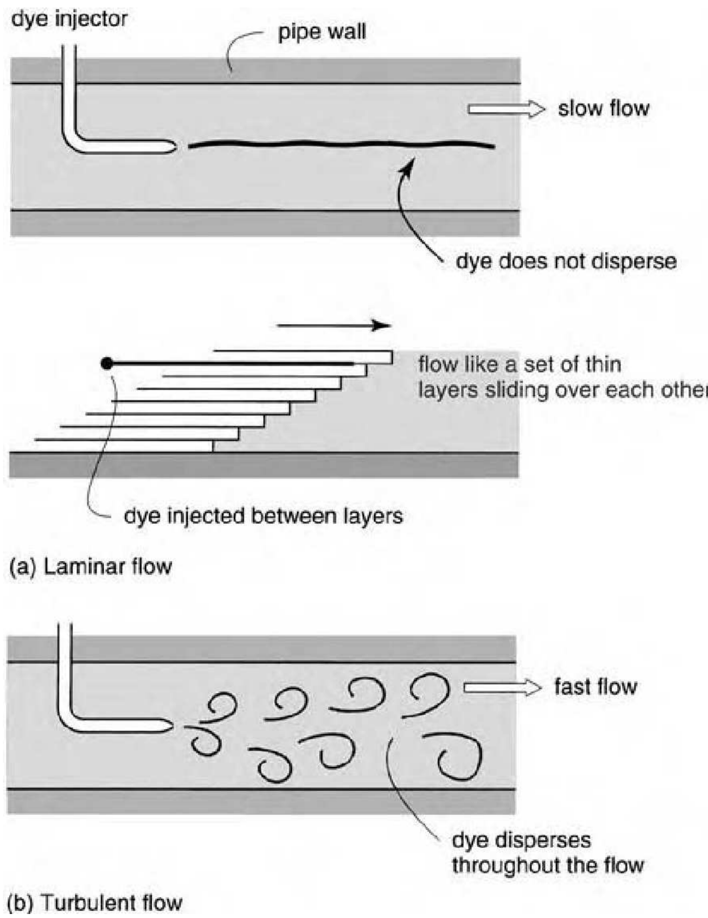
$z1$ - $z2$ is the difference in water levels between the reservoir and the storage tank and this represents the energy available to 'push' water through the system. "*h*f" is the energy loss due to friction in the pipe. The energy available is usually known and so this means we also know the amount of energy that can be lost through friction. The question now is 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.

4.2. 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.

4.2.1. 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 colored dye was injected into the flow so that the flow patterns were visible.



Cha 4 fig 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, viscosity and pipe diameter (d). He put these factors together in a way which is now called the *Reynolds Number* in recognition of his work.

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

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 μ) is large compared with the term ρ $\rho$$vd$. So viscosity is important in laminar flow and cannot be ignored. High Reynolds Number ($R > 4000$) means viscosity is small compared with the ρ $\rho$$vd$ 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 modeling (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 modeling 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.

4.2.1.1. *Reynolds Experiment*

Reynolds's experiment is to determine the factors that affect the movement of a fluid and how they affect it.

The fluid movement may be serpentine (turbulent) or direct (laminar) depending on:

+ The viscosity

+ Speed
+The characteristic length

Reynolds makes the following analogy:

"The circumstances which determine whether the movement of troops will be a march or confusion are very similar to those that determine whether the motion will be direct or sinuous. In both cases there is some influence to the order, with the troops is the discipline with water, its viscosity or clumping. The better the discipline of the troops, or more sticky fluid is less likely that the regular movement is altered on occasion. On the other hand, speed and size are both favorable to instability, the greater is the army and its movements faster the greater the chance of disorder, as well as the fluid, the wider the channel, the faster the speed the greater the chance of swirls".

This Reynolds concludes that the natural condition of a fluid is not the order but disorder. In a given length of horizontal pipe of constant diameter through which a fluid under pressure, energy loss occurs as the pressure head difference between the two points of interest.

$$\text{Loss of Energy } (hf) = h1 - h2$$

Where the pressure head at a point is given as the pressure at that point on the specific gravity (g) of the fluid

4.2.2. Chezy Equation and Darcy Weisbach

The flow of liquid through a pipe is resisted by viscous shear stresses within the liquid and the turbulence that occurs along the internal walls of the pipe, created by the roughness of the pipe material. This resistance is usually known as pipe friction and is measured is feet or meters head of the fluid, thus the term head loss is also used to express the resistance to flow.

Many factors affect the head loss in pipes, the viscosity of the fluid being handled, the size of the pipes, the roughness of the internal surface of the pipes, the changes in elevations within the system and the length of travel of the fluid.

The resistance through various valves and fittings will also contribute to the overall head loss. A method to model the resistances for valves and fittings is described elsewhere.

In a well designed system the resistance through valves and fittings will be of minor significance to the overall head loss, many designers choose to ignore the head loss for valves and fittings at least in the initial stages of a design.

Much research has been carried out over many years and various formulae to calculate head loss have been developed based on experimental data.

Among these is the Chézy formula which dealt with water flow in open channels. Using the concept of 'wetted perimeter' and the internal diameter of a pipe the Chézy formula could be adapted to estimate the head loss in a pipe, although the constant 'C' had to be determined experimentally.

Weisbach first proposed the equation we now know as the Darcy-Weisbach formula or
Darcy-Weisbach equation:

$$\boxed{hf = f\,(L/D) \times (v2/2g)}$$

Where:

hf = head loss (m)
f = friction factor
L = length of pipe work (m)
d = inner diameter of pipe work (m)
v = velocity of fluid (m/s)
g = acceleration due to gravity (m/s^2)

Or:

hf = head loss (ft)
f = friction factor
L = length of pipe work (ft)
d = inner diameter of pipe work (ft)

v = velocity of fluid (ft/s)
g = acceleration due to gravity (ft/s^2)

4.3. Friction Factor

Fanning did much experimentation to provide data for friction factors, however the head loss calculation using the Fanning Friction factors has to be applied using the hydraulic radius equation (not the pipe diameter). The hydraulic radius calculation involves dividing the cross sectional area of flow by the wetted perimeter. For a round pipe with full flow the hydraulic radius is equal to ¼ of the pipe diameter, so the head loss equation becomes:

$$hf= f_f \, f_f \, (L/ R_h \, R_h) \times (V^2/2g \; V^2/2g)$$

Where:
hf = ƒ f(L/Rh) x (v2/2g)
Where
Rh = hydraulic radius
ƒ f = Fanning friction factor

Darcy introduced the concept of relative roughness, where the ratio of the internal roughness of a pipe to the internal diameter of a pipe, will affect the friction factor for turbulent flow. In a relatively smoother pipe the turbulence along the pipe walls has less overall effect, hence a lower friction factor is applied.

The work of many others including Poiseuille, Hagen, Reynolds, Prandtl, Colebrook and White have contributed to the development of formulae for calculation of friction factors and head loss due to friction.

The Darcy Friction factor (which is 4 times greater than the Fanning Friction factor):

Used with Weisbach equation has now become the standard head loss equation for calculating head loss in pipes where the flow is turbulent. Initially the Darcy-Weisbach equation was difficult apply, since no electronic calculators were available and many calculations had to be carried out by hand.

The Colebrook-White equation which provides a mathematical method for calculation of the friction factor (for pipes that are neither totally smooth nor wholly rough) has the friction factor term f on both sides of the formula and is difficult to solve without trial and error (i.e. mathematical iteration is normally required to find f).

$$1 / \sqrt{f} \, \sqrt{f} = 1.14 - 2 \log 10 \, \log 10 \left(\frac{\varepsilon}{D} + \frac{9.35}{Re\sqrt{f}} \, \frac{\varepsilon}{D} + \frac{9.35}{Re\sqrt{f}} \right) \qquad \text{for } Re > 4000$$

Where:

f = friction factor
e = internal roughness of the pipe
D = inner diameter of pipe work

Due to the difficulty of solving the Colebrook-White equation to find f, the use of the empirical 'Hazen-Williams' formulae for flow of water at 60º F (15.5º C) has persisted for many years. To use the Hazen-Williams formula a head loss coefficient must be used.

Unfortunately the value of the head loss coefficient can vary from around 80 up to 130 and beyond and this can make the 'Hazen-Williams' formulae unsuitable for accurate prediction of head loss.

4.3.1. Moody´s diagram

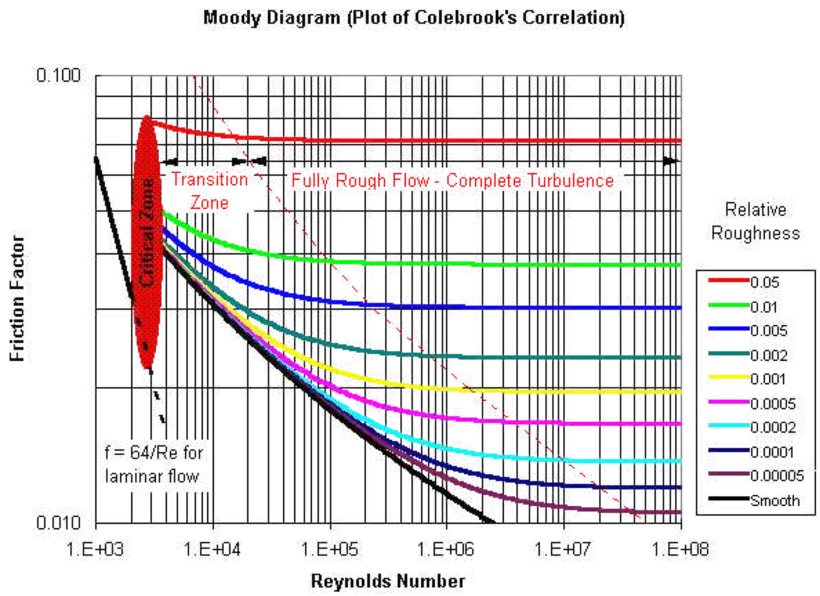
The Moody diagram is a graphical representation in doubly logarithmic scale of the friction factor as a function of Reynolds number and the relative roughness of a pipe.

On the Darcy-Weisbach equation appears the term λ that represents the Darcy friction factor, also known as the coefficient of friction. The calculation of this ratio is not immediate and there is no single formula to calculate it in all situations.

We can distinguish two different situations, the case where the flow is laminar and the case where the flow is turbulent. In the case of using a laminar flow of expressions of the Poiseuille equation, in the case of turbulent flow can use the Colebrook-White equation plus a few others how Barr equation, equation of Miller, Haaland equation.

For laminar flow the friction factor depends only on the Reynolds number. For turbulent flow, the friction factor depends on both Reynolds number and the relative roughness of the pipe, so in this case is represented by a family of curves, one for each value of the parameter k / D, where k is the absolute value of the roughness, the length (usually in millimeters) of roughness in the pipe directly measurable.

The following image shows the appearance of the Moody diagram



Cha 4 Fig. 3 Moody Diagram

4.4. Smooth and Rough Pipes

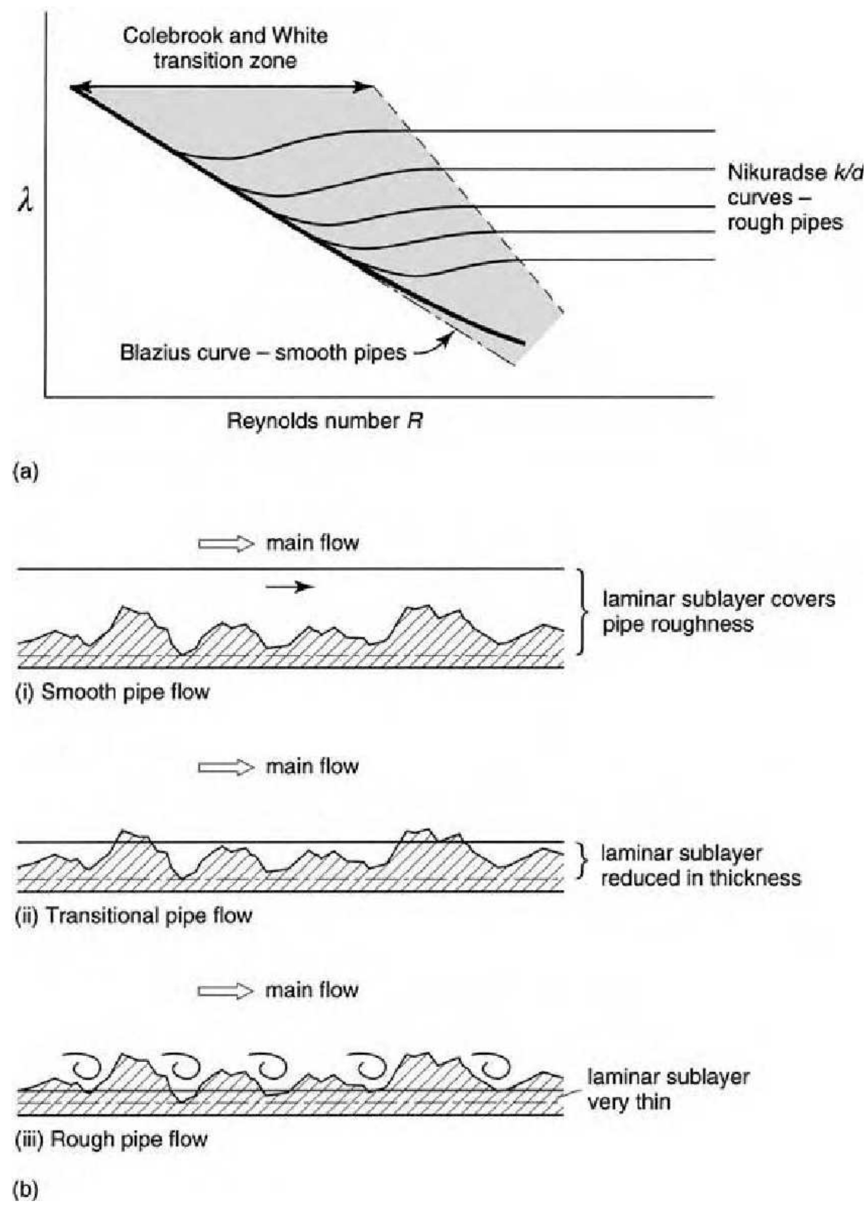
We now know that both investigators were right but they were looking at different aspects of the same problem. Blazius was looking at flows with relatively low Reynolds Numbers (4000 to 100 000) and his results refer to what are now called *smooth pipes*. Nikuradse's experiments dealt with high Reynolds 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 _ 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 l depends on both Reynolds Number and pipe roughness. 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 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 forall smooth and rough pipes and the transition is from smooth to rough pipe flow.



Cha 4 Fig 4 The l story.

To summaries 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

4.5. Hydraulic Gradient

One way of showing energy losses in a pipeline is to use a diagram. 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.

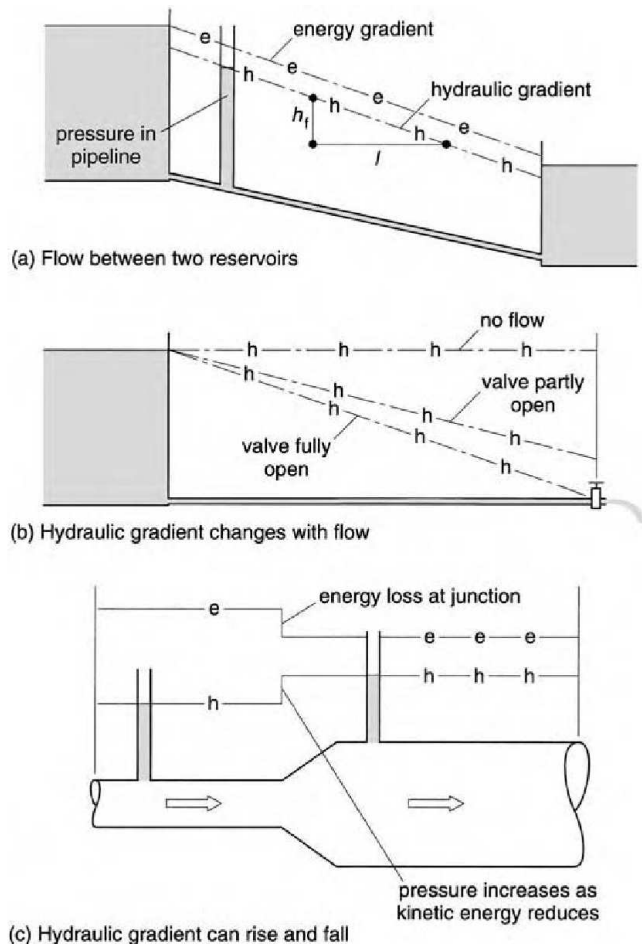
Although total energy is of interest, pressure is more important because this determines how strong the pipes must be to avoid bursts. For this reason a second line is drawn below the energy line, but parallel to it, to represent the pressure (pressure energy) and is marked *h—h—h*. This shows the pressure 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:

Hydraulic gradient = $\frac{h_f}{l}$ $\frac{h_f}{l}$

There *hf* 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 meters by a head difference in meters. However, it is often expressed in terms of meters head per meter length of pipeline. As an example a hydraulic gradient of 0.02 means for every one meter 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 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 X 2 = 10 m head.



Cha 4 Fig. 5 Hydraulic Gradient.

The hydraulic gradient is not a fixed line for a pipe; it depends on the flow.

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. 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. 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 incomparison 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. 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. Local Losses or Lower

In addition to the frictional energy losses, other losses "minor" problems associated with pipelines.

It is considered that such losses occur locally in the disturbance of flow. They occur due to any flow disturbance caused by curvature or changes in the section. They are called minor losses because they can often be neglected, particularly in long pipes where friction losses are high compared to local losses. However, in short pipes and a considerable number of accessories, the effect of local losses will be large and should be taken into account.
Minor losses are usually caused by changes in speed, whether magnitude or direction. Has been shown experimentally that the magnitude of losses is roughly proportional to the square of speed. It is common to express the minor losses as a function of head velocity in the tube, V2/2g:

$$h_L = h_L = K \frac{V^2}{2g} \frac{V^2}{2g}$$

With the loss “hL” minor loss coefficient K. K values for all types of accessories, are found in texts and hydraulic fluid.

4.7. Selecting Pipe Sizes in Practice

The development of *l* 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 m3/s and the difference in their water levels is 30 m. Assume the value of 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:

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

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:

$$h_f \, h_f = z_1 \, z_1 - z_2 \, z_2$$

Using the Darcy–Weisbach formula for *h*f:

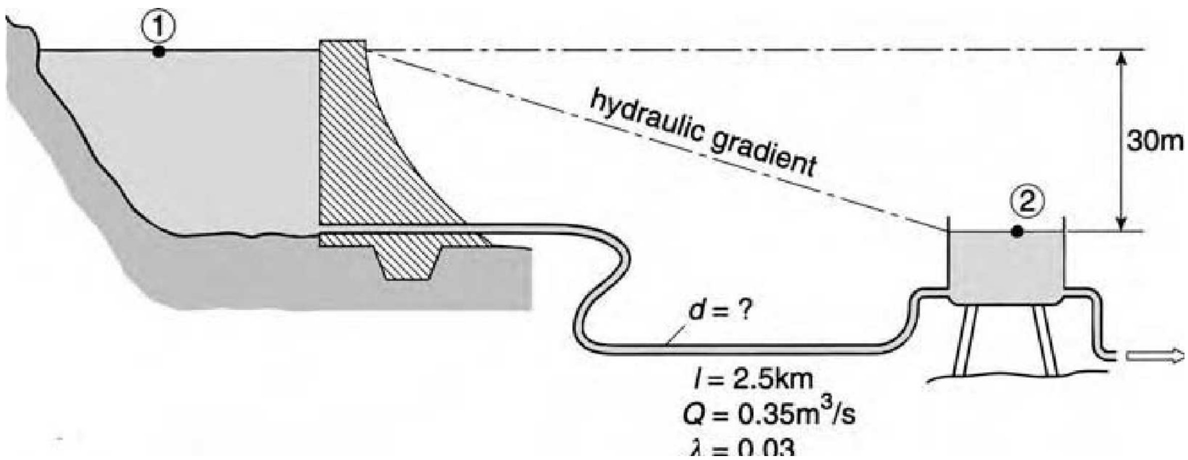
$$h_f \, h_f = \frac{\lambda \, l \, v^2}{2gd} \frac{\lambda \, l \, v^2}{2gd}$$

And so:

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

$$Q = va$$

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



Cha 5 Fig 6 Calculating the pipe diameter.

Calculate area *a*:

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

And use this value to calculate *v*:

$$V = \frac{4Q}{\pi d^4}\frac{4Q}{\pi d^4} = \frac{4 X 0.35}{3.14 X d^2}\frac{4 X 0.35}{3.14 X d^2} = \frac{0.446}{d^2}\frac{0.446}{d^2}$$

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:

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

Rearrange this to calculate *d*:

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

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 pipeson 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.

4.8. Other Friction Formulas

4.8.1. Manning

Manning's formula is an evolution of the Chezy formula for calculating the water velocity in open channels and pipes, given by the Irish engineer Robert Manning, in 1889:

$$V= \frac{1}{n}\frac{1}{n} R_h^{\frac{2}{3}} R_h^{\frac{2}{3}} S^{\frac{1}{2}} S^{\frac{1}{2}}$$

For some it is an expression of what is known Chezy coefficient C used in the Chezy formula:

$$V(h)= C \sqrt{R(h) * S} \sqrt{R(h) * S}$$

4.8.2. Hazen - Williams

The Hazen–Williams equation is an empirical formula which relates the flow of water in a pipe with the physical properties of the pipe and the pressure drop caused by friction. It is used in the design of water pipe systems[1] such as fire sprinkler systems[2], water supply networks, and irrigation systems. It is named after Allen Hazen and Gardner Stewart Williams.

The Hazen–Williams equation has the advantage that the coefficient *C* is not a function of the Reynolds number, but it has the disadvantage that it is only valid for water. Also, it does not account for the temperature or viscosity of the water

The general form of the equation relates the mean velocity of water in a pipe with the geometric properties of the pipe and slope of the energy line.

$$V = C\, R^{0.63}\, R^{0.63}\ \ S^{0.54}\, S^{0.54}$$

Where:

- V is velocity
- k is a conversion factor for the unit system (k = 1.318 for US customary units, k = 0.849 for SI units)
- C is a roughness coefficient
- R is the hydraulic radius
- S is the slope of the energy line (head loss per length of pipe or h_f/L)

Typical *C* factors used in design, which take into account some increase in roughness as pipe ages are as follows:

MATERIAL	FACTOR LOW	FACTOR HIGH	REFERENCE
ASBESTOS – CEMENT	140	140	-
CAST IRON	100	140	-
CEMENT-MORTAR LINED DUCTILE IRON PIPE	140	140	-
CONCRETE	100	140	-
COPPER	130	140	-
STEEL	90	110	
GALVANIZED IRON	120	120	
POLYETHYLENE	140	140	
POLYVINYL CHLORIDE(PVC)	130	130	-
FIBR-REINFORCED PLASTIC (FRP)	150	150	-

4.8.3. Chezy

The Chezy formula can be used to calculate mean flow velocity in conduits and is expressed as

$$v = c\,(R\,S)^{1/2}$$

Where:

v = Mean velocity (m/s, ft/s)

c = The Chezy roughness and conduit coefficient

R = Hydraulic radius of the conduit (m, ft)

S = slope of the conduit (m/m, ft/ft*)*

In general the Chezy coefficient - *c* - is a function of the flow Reynolds Number - Re - and the relative roughness - *ε/R* - of the channel.

4.9. Piping Systems

Whether you are pumping water to fill a pond or to aerate, it pays to do it as economically as possible. A key to economical operation is to minimize the work you have to do and to match your pump to the requirements. Both depend upon the piping system you move your water through. A suitable piping system for your operation can be determined by considering the three components that make up the total resistance to water movement in the pipe. This resistance, called the total dynamic head, determines the amount of work required to move each gallon of water. The total dynamic head is the sum of the lift, the velocity head and the friction head.

<u>Lift</u>

Lift is the vertical distance between the level of supply water's surface and point of discharge at the end of the pipe while the pump is running. It is the only component of the total dynamic head which is not directly affected by the piping system.

<u>Velocity Head</u>

The energy contained in a stream of water due to its velocity. This energy is lost when the water is discharged. The amount of work required to produce this velocity is equivalent to picking up the water high enough so that it would obtain the required velocity in falling. This height is called a "head" and is commonly measured in feet of water (the height the water has to be picked up). Numerically, it is equal to the square of the velocity (in feet per second) divided by 64.

Velocity Head = $\frac{V^2}{64}$ $\frac{V^2}{64}$ (V in feet

Most losses, and the work required to move the water in the pipe, vary with the velocity head. For a given flow rate, the velocity head is very sensitive to the size of the pipe. The velocity head depends upon the fourth power of the pipe diameter. For example, if the pipe diameter is halved, the velocity head is 16 times greater. Mathematically this relationship can be expressed as:

$$\frac{New\ velocity\ head}{Old\ velocity\ head}\ \frac{New\ velocity\ head}{Old\ velocity\ head} = \frac{Old\ diameter}{New\ diameter}^{4}\ \frac{Old\ diameter}{New\ diameter}^{4}$$

Example: Determine the velocity and velocity head for 1,200 gallons per minute (gpm) of water flowing in a 1-inch diameter pipe. The velocity is calculated by dividing the flow rate by the cross-sectional area of the pipe. Since there are 7.48 gallons in 1 cubic foot and 60 seconds in a minute, the flow rate in cubic feet per second (CFS) is obtained by dividing 1,200 gpm by 60 seconds per minute and 7.48 gallons per cubic foot. The flowrate is:

$$\frac{1200\ gpm}{(60\ \text{sec}\ min)\ (7.48\ \ gal\ /ft^{3})}\ \frac{1200\ gpm}{(60\ \text{sec}\ min)\ (7.48\ \ gal\ /ft^{3})} = 2.67\ \text{CFS}$$

The cross-sectional area, A, is equal to pi times the diameter squared divided by 4. Realizing that there are 144 square inches in 1 square foot, the area is calculated as:

$$\frac{3.14\ x\ 6\ INCHES\ x\ 6\ INCHES}{4\ x\ 114\ SQUARE\ INCHES\ PER\ SQUARE\ FOOT}\ \frac{3.14\ x\ 6\ INCHES\ x\ 6\ INCHES}{4\ x\ 114\ SQUARE\ INCHES\ PER\ SQUARE\ FOOT} = 0.20\ \text{SQUARE FEET}$$

The velocity, V, is:

$$V = \frac{FLOWRATE}{AREA}\ \frac{FLOWRATE}{AREA} = \frac{2.67\ CFS}{0.2\ SQUARE\ FEET}\ \frac{2.67\ CFS}{0.2\ SQUARE\ FEET} = 13.35\ \text{FT / SEC}$$

And the velocity head is:

$$\frac{v^2}{64}\ \frac{v^2}{64} = \frac{13.35\ x\ 13.35}{64}\ \frac{13.35\ x\ 13.35}{64} = 2.78\ \text{FEET}$$

As a rule of thumb, the velocity head should be less than 0.4 foot. The velocity head in feet for various diameter pipes and flow rates :

Table 1. Velocity head in feet.

Flow gpm → Dia. (inches)	100	200	400	600	800	1,000	1,500
4	0.10	0.41	1.62	3.65	6.48	10.13	22.80
6	0.02	0.08	0.32	0.72	1.28	2.00	4.50
8	0.01	0.03	0.10	0.23	0.41	0.63	1.42
10	0.00	0.01	0.04	0.09	0.17	0.26	0.58
12	0.00	0.01	0.02	0.05	0.08	0.13	0.28
14	0.00	0.00	0.01	0.02	0.04	0.07	0.15
16	0.00	0.00	0.01	0.01	0.03	0.04	0.09
18	0.00	0.00	0.00	0.01	0.02	0.02	0.06
20	0.00	0.00	0.00	0.01	0.01	0.02	0.04
24	0.00	0.00	0.00	0.00	0.01	0.012	0.02

Flow gpm → Dia. (inches)	2,000	2,500	3,000	4,000	5,000	7,500	10,000
4	40.53	63.32	91.19	162.11	253.29	569.91	1,013.17
6	8.01	12.51	18.01	32.02	50.03	112.57	200.13
8	2.53	3.96	5.70	10.13	15.83	35.62	63.32
10	1.04	1.62	2.33	4.15	6.48	14.59	25.94
12	0.50	0.78	1.13	2.00	3.13	7.04	12.51
14	0.27	0.42	0.61	1.08	1.69	3.80	6.75
16	0.16	0.25	0.36	0.63	0.99	2.23	3.96
18	0.10	0.15	0.22	0.40	0.62	1.39	2.47
20	0.06	0.10	0.15	0.26	0.41	0.91	1.62
24	0.03	0.05	0.07	0.13	0.20	0.44	0.78

4.9.1. Parallel Pipe

A parallel pipeline system consists of a set of pipes that are born and come together in one place. For a generic system of n parallel lines is verified that:
The total flow is the sum of the individual capacities of each pipe (continuity equation)

$$Q_T = \Sigma \sum_{i=1}^{n} Q_1$$

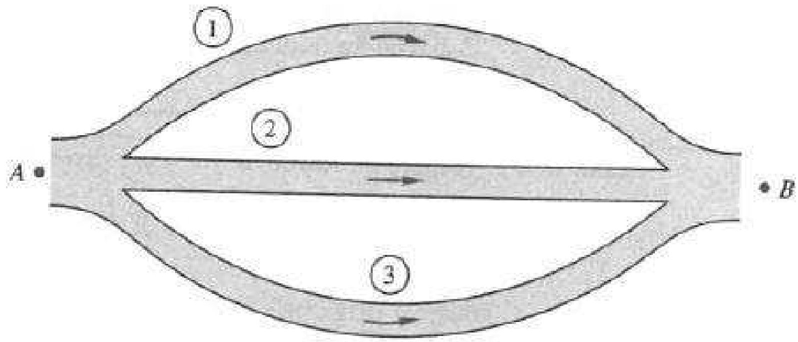
4.9.2. Cha 4 Fig 7 Parallel Pipe

The total pressure loss is equal to the loss in each of the piping system:

$$\Delta h_T \Delta h_T = \Delta h_i + \Delta h_{fi} + \Delta h_{mi} \quad i = 1,2, \ldots n. \Delta h_i + \Delta h_{fi} + \Delta h_{mi} \quad i = 1,2, \ldots n.$$

$h_{fi}\, h_{fi}$ and $h_{mi}\ h_{mi}$ Where are the primary and secondary losses in each of the system piping.

Set of 3 parallel lines between A and B



Cha 4 Fig 7 Parallel Pipe

4.9.2 Pipe Networks

In fluid dynamics, pipe network analysis is the analysis of the fluid flow through a hydraulics network, containing several or many interconnected branches. The aim is to determine the flow rates and pressure drops in the individual sections of the network. This is a common problem in hydraulic design.

In order to direct water to many individuals in a municipal water supply, many times the water is routed through a water supply network. A major part of this network may consist of interconnected pipes. This network creates a special class of problems in hydraulic design typically referred to as pipe network analysis. The modern solution for this is to use specialized software in order to automatically solve the problems. However, the problems can also be addressed with simpler methods like a spreadsheet equipped with a solver, or a modern graphing calculator.

Network Analysis

Once the friction factors are solved for, then we can start considering the network problem. We can solve the network by satisfying two conditions.

1.-At any junction, the flow into a junction equals the flow out of the junction.

2.-Between any two junctions, the head loss is independent of the path taken.

The classical approach for solving these networks is to use the Hardy Cross algorithm. In this formulation, first you go through and create guess values for the flows in the network. That is, if Q7 enters a junction and Q6 and Q4 leave the same junction, then the initial guess must satisfy Q7 = Q6 + Q4. After the initial guess is made, then, a loop is considered so that we can evaluate our second condition. Given a starting node, we work our way around the loop in a clockwise fashion, as illustrated by Loop 1. We add up the head losses according to the Darcy–Weisbach equation for each pipe if Q is in the same direction as our loop like Q1, and subtract the head loss if the flow is in the reverse direction, like Q4. In order to satisfy the second condition, we should end up with 0 about the loop if the network is completely solved. If the actual sum of our head loss is not equal to 0, then we will adjust all the flows in the loop by an amount given by the following formula, where a positive adjustment is in the clockwise direction.

$$\boxed{\Delta Q = \frac{\sum \text{head loss}_c - \sum \text{head loss}_{cc}}{n \cdot \left(\sum \frac{\text{head loss}_c}{Q_c} + \sum \frac{\text{head loss}_{cc}}{Q_{cc}}\right)},}$$

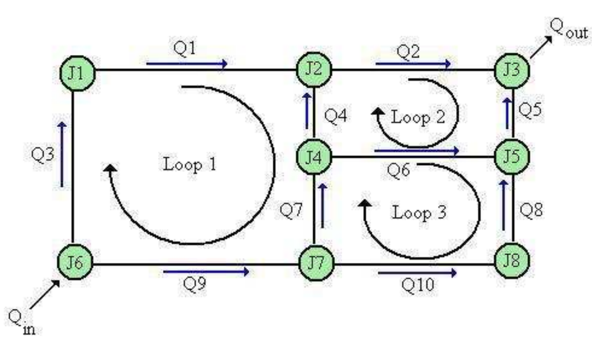
Where:

- *n* is 1.85 for Hazen-Williams and
- *n* is 2 for Darcy–Weisbach.

The clockwise specified (c) means only the flows that are moving clockwise in our loop, while the counter-clockwise specified (cc) is only the flows that are moving counter-clockwise.

This adjustment won't solve the problem, since with most networks we will have several loops. It is ok to do this adjustment, however, because our flow changes won't alter condition 1, and therefore, our other loops will still satisfy condition 1. However, we should use the results from the first loop if we progress to any other loops.

The more modern method is simply to create a set of conditions from your junctions and head-loss criteria. Then, use a Root-finding algorithm to find *Q* values that satisfy all the equations. The literal friction loss equations will use a term called Q^2, but we want to preserve any changes in direction. Create a separate equation for each loop where the head losses are added up, but instead of squaring *Q*, use $|Q|\cdot Q$ instead (with $|Q|$ the absolute value of *Q*) for the formulation so that any sign changes will reflect appropriately in the resulting head-loss calculation



Cha 4 fig 8 Network Analysis

5. **PHYSICAL MODELING THEORY**

Definition: the physical hydraulic model

A physical model is a scaled representation of a hydraulic flow situation. Both the boundary conditions (e.g. channel bed, sidewalls), the upstream flow conditions and the flow field must be scaled in an appropriate manner.

Physical hydraulic models are commonly used during design stages to optimize a structure and to ensure a safe operation of the structure. They have an important further role to assist non-engineering people during the `decision-making' process.

A hydraulic model may help the decision-makers to visualize and to picture the flow field, before selecting a `suitable' design.

In civil engineering applications, a physical hydraulic model is usually a smaller- size representation of the prototype (i.e. the full-scale structure).

Other applications of model studies (e.g. water treatment plant, flotation column) may require the use of models larger than the prototype. In any case the model is investigated in a laboratory under controlled condition

5.1. Dimensional Analysis

Taking into account all basic parameters, dimensional analysis yields:

$$F_1(\rho, \mu, \sigma, E_b, g, L, V, \Delta P = 0$$

There are eight basic parameters and the dimensions of these can be grouped into three categories: mass (M), length (L) and time (T). The Buckingham –theorem (Buckingham 1915) implies that the quantities can be grouped into five (5 . 8 ÿ 3) independent dimensionless parameters:

$$F_{2}\left(\frac{V}{\sqrt{gL}}; \frac{\rho V^2}{\Delta P}; \frac{\rho V L}{\mu}; \frac{V}{\sqrt{\frac{\sigma}{\rho L}}}; \frac{V}{\sqrt{\frac{E_b}{\rho}}}\right)$$

$F_2 = F_2 =$ Fr; Eu; R*e; We; Ma)*

The first ratio is the Froude number Fr, characterizing the ratio of the inertial force to gravity force. Eu is the Euler number, proportional to the ratio of inertial force to pressure force. The third dimensionless parameter is the Reynolds number Re which characterizes the ratio of inertial force to viscous force. The Weber number.

We is proportional to the ratio of inertial force to capillary force (i.e. surface tension).

The last parameter is the Sarrau - Mach number, characterizing the ratio of inertial force to elasticity force.

5.2. Similarity Dinamic

Traditionally model studies are performed using geometrically similar models. In a geometrically similar model, true dynamic similarity is achieved if and only if each dimensionless parameter (or II-terms) has the same value in both model and proto- type:

Frp . Frm; Eup . Eum; Rep . Rem; Wep . Wem; Map . Mam

Scale effects will exist when one or more II - terms have different values in the model and prototype.

In practice, hydraulic model tests are performed under controlled flow conditions.

The pressure difference ΔP may usually be controlled. This enables ΔP to be treated as a dependent parameter. Further compressibility effects are small in clear-water ows2 and the

Sarrau-Mach number is usually very small in both model and proto- type. Hence, dynamic similarity in most hydraulic models is governed by:

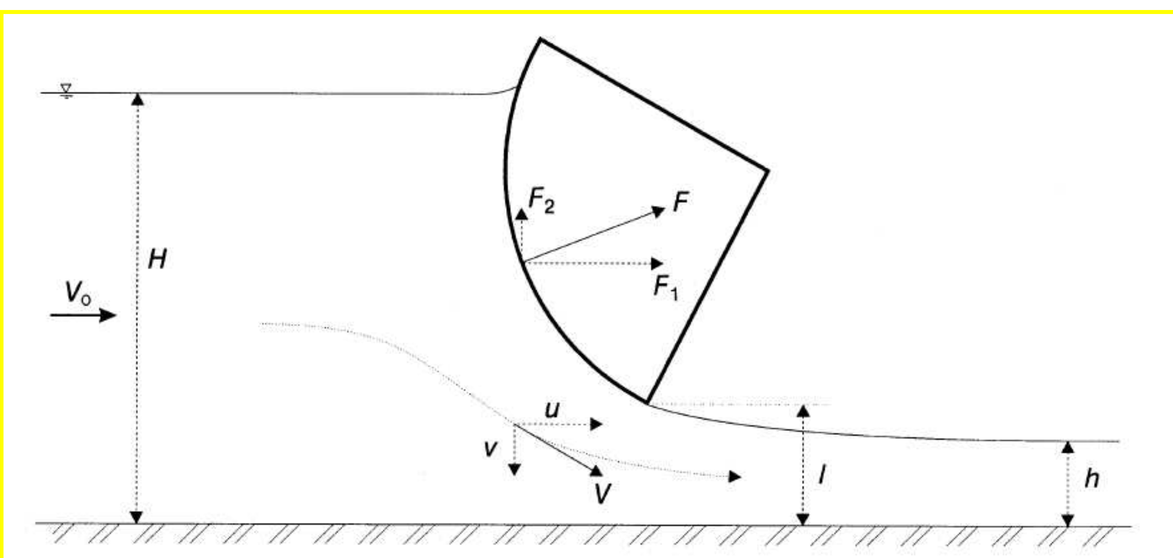
$$\frac{\Delta P}{\rho V^2} \frac{\Delta P}{\rho V^2} = F_3\left(\frac{V}{\sqrt{gL}}; \frac{\rho V L}{\mu}; \frac{V}{\sqrt{\frac{\sigma}{\rho L}}}\right) F_3\left(\frac{V}{\sqrt{gL}}; \frac{\rho V L}{\mu}; \frac{V}{\sqrt{\frac{\sigma}{\rho L}}}\right)$$

Eu= $F_3\ F_3$ (Fr; Re; We) Hydraulics Model Test

There are a multitude of phenomena that might be important in hydraulic flow situations: e.g. viscous effects, surface tension, gravity effect. The use of the same fluid on both prototype and model prohibits simultaneously satisfying the Froude, Reynolds and Weber number scaling criteria because the Froude number similarity requires Vr= $\sqrt{Lr}\,\sqrt{Lr}$, the Reynolds number scaling implies that Vr=. 1 / Lr and the Weber number similarity requires: Vr = 1 / $\sqrt{Lr}\,\sqrt{Lr}$

In most cases, only the most dominant mechanism is modeled. Hydraulic models commonly use water and/or air as flowing fluid(s). In fully-enclosed flows (e.g. pipe flows), the pressure losses are basically related to the Reynolds number Re. Hence, a Reynolds number scaling is used: i.e. the Reynolds number is the same in both model and prototype. In free-surface flows (i.e. flows with a free surface), gravity effects are always important and a Froude number modeling is used (i.e. Frm . Frp)

In a physical model, the flow conditions are said to be similar to those in the prototype if the model displays similarity of form (geometric similarity), similarity of motion (kinematic similarity) and similarity of forces (dynamic similarity)



Cha 5 Fig 1 Basic flow parameters.

5.3. Reynold´s Law

In fluid mechanics, the Reynolds number Re is a dimensionless number that gives a measure of the ratio of inertial forces $\rho v^2/L$ to viscous forces $\mu v/L^2$ and consequently quantifies the relative importance of these two types of forces for given flow conditions. The concept was introduced by George Gabriel Stokes in 1851,[1] but the Reynolds number is named after Osborne Reynolds (1842–1912), who popularized its use in 1883.

Reynolds numbers frequently arise when performing dimensional analysis of fluid dynamics problems, and as such can be used to determine dynamic similitude between different experimental cases. They are also used to characterize different flow regimes, such as laminar or turbulent flow: laminar flow occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion, while turbulent flow occurs at high Reynolds numbers and is dominated by inertial forces, which tend to produce chaotic eddies, vortices and other flow instabilities

Definition

Reynolds number can be defined for a number of different situations where a fluid is in relative motion to a surface (the definition of the Reynolds number is not to be confused with the Reynolds Equation or lubrication equation). These definitions generally include the fluid properties of density and viscosity, plus a velocity and a *characteristic length* or *characteristic dimension*. This dimension is a matter of convention - for example a radius or diameter are equally valid for spheres or circles, but one is chosen by convention. For aircraft or ships, the length or width can be used. For flow in a pipe or a sphere moving in a fluid the internal diameter is generally used today. Other shapes (such as rectangular pipes or non-spherical objects) have an *equivalent diameter* defined. For fluids of variable density (e.g. compressible gases) or variable viscosity (non-Newtonian fluids) special rules apply. The velocity may also be a matter of convention in some circumstances, notably stirred vessels.

$$\mathrm{Re} = \frac{\rho VL}{\mu} \frac{\rho VL}{\mu} = \frac{VL}{\nu} \frac{VL}{\nu}$$

Where:

- V is the mean velocity of the object relative to the fluid (SI units: m/s)
- L is a characteristic linear dimension, (travelled length of the fluid hydraulic diameter when dealing with river systems) (m)
- μ is the dynamic viscosity of the fluid (Pa·s or N·s/m² or kg/(m·s))
- ν is the kinematic viscosity ($\nu = \mu / \rho$) (m²/s)
- ρ is the density of the fluid (kg/m³)

Note that this is equal to the ratio between $\frac{\rho v^2}{L}$, which is the drag (up to a numerical factor, half the drag coefficient), and $\frac{\mu V}{L^2}$, which is the force due to viscosity (up to a numerical factor depending on the form of the flow).

<u>Significance</u>

$$\text{RE}= \frac{Total\ momentum\ transfer}{Molecular\ momentum\ transfer} \frac{Total\ momentum\ transfer}{Molecular\ momentum\ transfer}$$

<u>Flow in Pipe</u>

For flow in a pipe or tube, the Reynolds number is generally defined as:

$$\text{Re}= \frac{\rho V D_H}{\mu} \frac{\rho V D_H}{\mu} = \frac{V D_H}{v} \frac{V D_H}{v} = \frac{Q D_H}{v A} \frac{Q D_H}{v A}$$

Where:

- D_H is the hydraulic diameter of the pipe (m).
- Q is the volumetric flow rate (m^3/s).
- *A* is the pipe *cross-sectional* area (m^2).

5.4. Froude Law

The Froude number is a dimensionless number defined as the ratio of a characteristic velocity to a gravitational wave velocity. It may equivalently be defined as the ratio of a body's inertia to gravitational forces. In fluid mechanics, the Froude number is used to determine the resistance of an object moving through water, and permits the comparison of objects of different sizes. Named after William Froude, the Froude number is based on the speed/length ratio as defined by him.

The Froude number is defined as:

$$\text{Fr} = \frac{V}{c} \frac{V}{c}$$

Where *V* is a characteristic velocity , and *c* is a characteristic water wave propagation velocity. The Froude number is thus analogous to the Mach number. The greater the Froude number, the greater the resistance.

Some uses are:

Ship hydrodynamics

For a ship, the Froude number is defined as:

$$Fr = \frac{V}{\sqrt{gL}}, \frac{V}{\sqrt{gL}},$$

Where V is the velocity of the ship, g is the acceleration due to gravity, and L is the length of the ship at the water line level, or L_{wl} in some notations. It is an important parameter with respect to the ship's drag, or resistance, including the wave making resistance.

Shallow water waves

For shallow water waves, like for instance tidal waves and the hydraulic jump, the characteristic velocity V is the average flow velocity, averaged over the cross-section perpendicular to the flow direction. The wave velocity, c, is equal to the square root of gravitational acceleration g, times cross-sectional area A, divided by free-surface width B:

$$C= \sqrt{g\frac{A}{B}}, \sqrt{g\frac{A}{B}},$$

So the Froude number in shallow water is:

$$Fr= \frac{V}{\sqrt{g\frac{A}{B}}} \frac{V}{\sqrt{g\frac{A}{B}}}$$

For rectangular cross-sections with uniform depth d, the Froude number can be simplified to:

$$Fr= \frac{V}{\sqrt{gd}} \frac{V}{\sqrt{gd}}$$

For $Fr < 1$ the flow is called a subcritical flow, further for $Fr > 1$ the flow is characterised as supercritical flow. When $Fr \approx 1$ the flow is denoted as critical flow.

An alternate definition used in fluid mechanics is

$$Fr = \frac{V^2}{gd}, \frac{V^2}{gd},$$

Where each of the terms on the right have been squared. This form is the reciprocal of the Richardson number.

Stirred tanks

In the study of stirred tanks, the Froude number governs the formation of surface vortices. Since the impeller tip velocity is proportional to *Nd*, where *N* is the impeller speed (rev/s) and *d* is the impeller diameter, the Froude number then takes the following form:

$$Fr = \frac{N^2 d}{g} \frac{N^2 d}{g}$$

Densimetric Froude number

When used in the context of the Boussinesq approximation the densimetric Froude number is defined as

$$Fr = \frac{u}{\sqrt{g'h}} \frac{u}{\sqrt{g'h}}$$

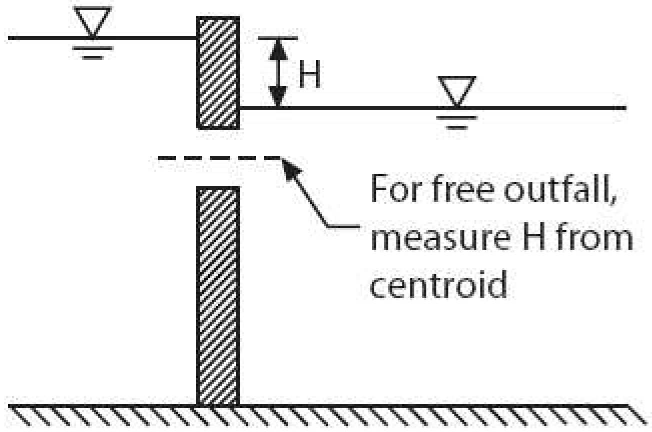
Where *g*' is the reduced gravity:

$$g' = g \frac{\rho_1 - \rho_2}{\rho} \frac{\rho_1 - \rho_2}{\rho}$$

The densimetric Froude number is usually preferred by modelers who wish to nondimensionalize a speed preference to the Richardson number which is more commonly encountered when considering stratified shear layers. For example, the leading edge of a gravity current moves with a front Froude number of about uni

6. **ORIFICES AND WEIRS**

6.1. Orifices



Cha 6 Fig 1 Orifices flow

The orifice equation is defined as:

$$Q=CA\ 2gH$$

Where:

Q = Flow (m^3/sec., ft^3/sec.)
C = Orifice coefficient
A = Flow area (m^2, ft^2)
g = Gravitational acceleration (m/$sec.^2$, ft/$sec.^2$)
H = Head (m, ft)

6.1.1. Orifice Coefficients

Although these coefficients vary with shape, size, and head depth, an average C coefficient of 0.60 is often used for storm water orifice openings. A list of orifice coefficients for various heads and sizes of circular, square, rectangular, and triangular shapes can be found in the Handbook of Hydraulics, by Brater et Al

6.1.2. Sluice Gate

Gates have the hydraulic properties of orifices. Therefore, the discharge through a
sluice gate is:

$$Q= CA \sqrt{2gH}\ \sqrt{2gH}$$

Where :

Q = Flow (m3/sec., ft3/sec.)
C = Orifice coefficient
A = Flow area (m2, ft2)
g = Gravitational acceleration (m/sec.2, ft/sec.2)
H = Head (m, ft)

Model a sluice gate by using the Generic Orifice in FlowMaster, entering the appropriate coefficient.

6.2. Weirs

Sharp-crested and non-sharp-crested weirs are the two profiles generally associated with weir flow.

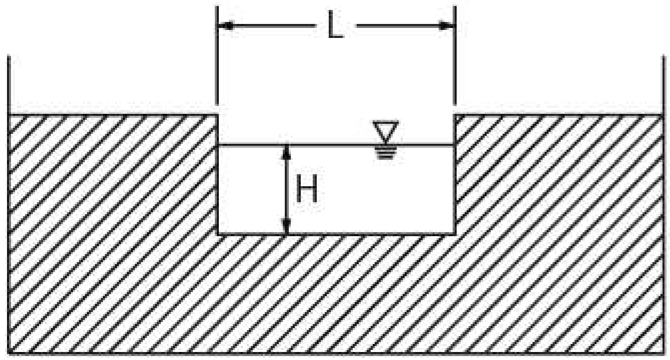
Sharp-crested weirs are usually used for measuring a discharge, based on the water height. Non-sharp-crested weirs are usually part of a hydraulic structure, such as an overflowing embankment or roadway.

6.2.1. *Sharp-Crested Weirs*

A sharp-crested weir has a sharp upstream edge formed so that the water flows clear of the crest. Flow Master handles weir calculations for unsubmerged (free discharge) and submerged (backwater effect) sharp-crested weirs.

Rectangular sharp – Crested Weir

Note: An error message will be displayed if the input data doses not satisfy these requirements



Cha 6 Fig 2 Rectangular sharp – Crested Weir

The discharge over an unsubmerged rectangular sharp-crested weir is defined as:

$$Q=C(L-0.1iH)\ H^{\frac{3}{2}} H^{\frac{3}{2}}$$

Where :
Q = Discharge over weir (m3/sec., ft3/sec.)
C = Weir coefficient (typical values for this kind of weir are C = 1.84 SI and C = 3.33 U.S. customary)
L = Weir opening width (m, ft)
i = Number of contractions (i = 0, 1, or 2)
H = Head above bottom of opening (m, ft)

i = 0 corresponds to the case of a *suppressed* rectangular weir, for which the channel width is equal to the weir opening length, and yields the equation:

$$Q = CL\ H^{\frac{3}{2}} H^{\frac{3}{2}}$$

i = 2 corresponds to the case of a *contracted* rectangular weir.

6.2.2. Broad Crested Weir

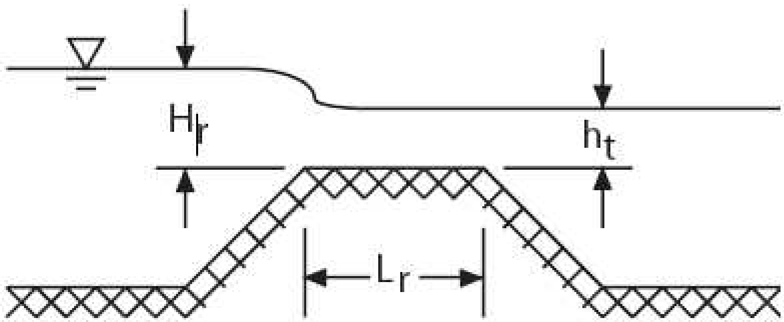
A broad-crested weir has a crest that extends horizontally in the direction of flow far enough to support the nappe (sheet of water flowing over the crest of the weir) so that hydrostatic pressures are fully developed for at least some short distance.

In order to model Embankment or Roadway overtopping, the Federal Highway Administration (FHWA) has developed a methodology that can be found in the manual FHWA, HDS No. 5, Hydraulic Design of Highway Culverts, 1985, which uses the general broad-crested weir equation.

$$Q = C_d L H_r^{\frac{3}{2}} C_d L H_r^{\frac{3}{2}}$$

Where:

Q = Discharge over weir (m3/sec., ft3/sec.)
Cd = Weir coefficient
L = Length of roadway crest (m, ft)
Hr = Overtopping depth (m, ft)

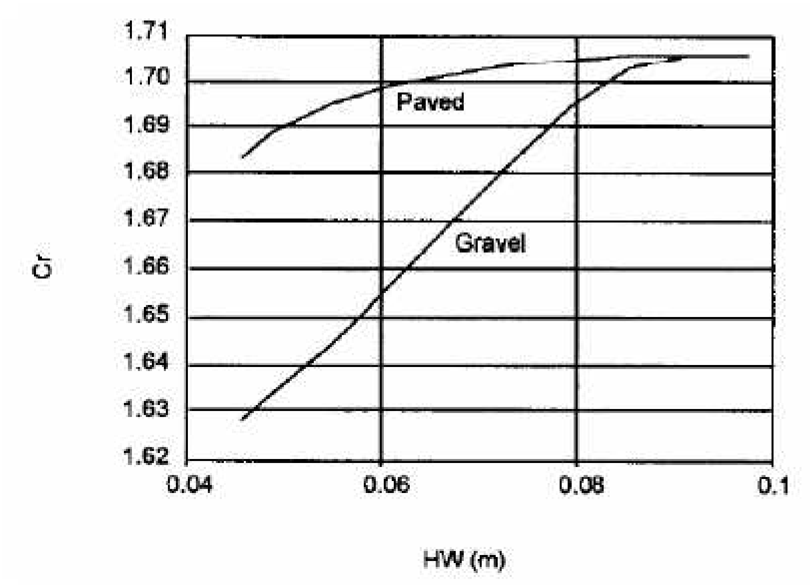


Cha 6 Fig 3 Broad Crested Weir

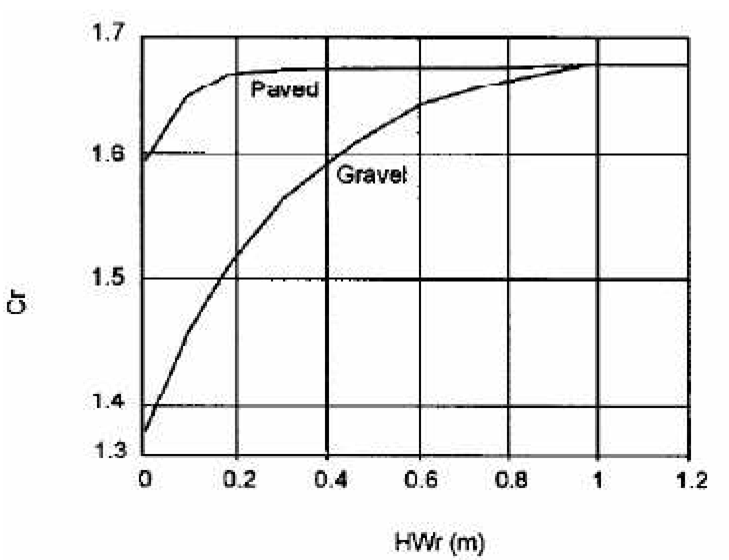
The overtopping discharge coefficient *Cd* is a function of the submergence using the equation:

$$Cd = KtCr$$

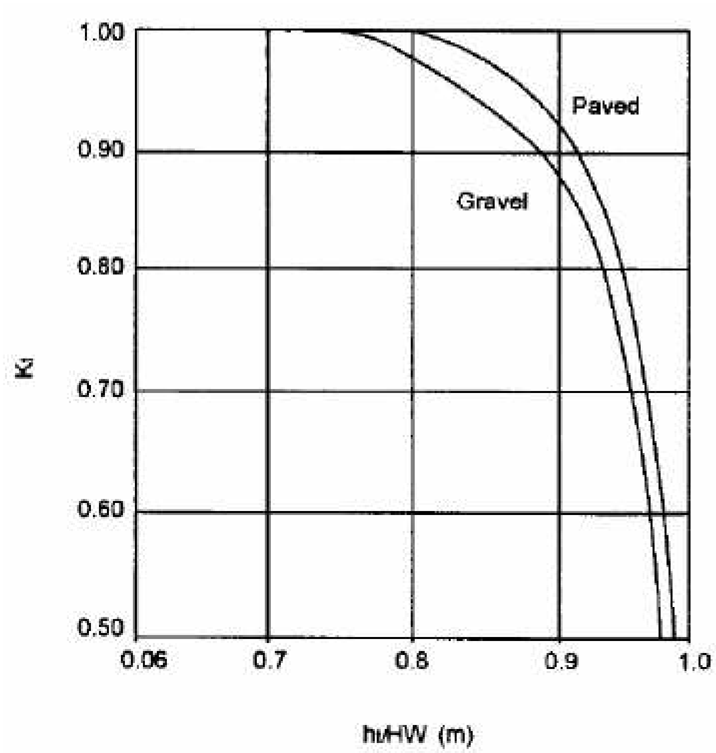
The variables *Kt* and *Cr* are defined in the following figures, reproduced from the manual FHWA, HDS No.5, Hydraulic Design of Highway Culverts, 1985. The first two figures are used by Flow Master to derive the base weir coefficient *Cr* resulting from deep and shallow overtopping, respectively. The submergence correction *Kt* is determined implicitly using the third figure.



Cha 6 Fig 4Discharge Coefficient *Cr*, for *Hr*/*L* > 0.15



Cha Fig 5 Discharge Coefficient *Cr*, for *Hr*/*L* ≤ 0.15



Cha 6 Fig 6 Submergence Factor, *k*

REFERENCIAS

Open Channel Hydraulics   Ven Te Chow   Mc Graw Hill International Book Company

Hydraulics Hand-book   R. H. Warring   Gulf Publishing Company   8th Edition

http://en.wikipedia.org/wiki/Viscosity#Dynamic_viscosity

http://www.princeton.edu/~asmits/Bicycle_web/Bernoulli.html

http://en.wikipedia.org/wiki/Archimedes#The_Golden_Crown

http://theoryx5.uwinnipeg.ca/mod_tech/node67.html

http://www.ustfa.org/Trout%20production/Piping%20Systems%20%20%20srac%201994.pdf

http://www.ustfa.org/Trout%20production/Piping%20Systems%20%20%20srac%201994.pdf

http://escuelas.fi.uba.ar/iis/Orificios%20y%20vertederos.pdf

http://www.pipeflow.co.uk/public/articles/Darcy_Weisbach_Formula.pdf (chazy And darcy)

Copia la lista de referencias del texto. Solo de internet no es bueno