

7 Hydraulic analysis and design

7.1 Overview

Hydraulic analysis is an essential prerequisite for any project involving the implementation of works in a river.

Much analysis of hydraulics – for the purpose of design – can be carried out by applying basic theory rather than resorting to numerical or physical modelling techniques. The basic theory is fundamentally the same for all methods, as are the data inputs, but the results of a quick manual analysis can often be used to give the designer a ‘feel’ for the problem or to confirm that model results are in the right order. Such approaches are discussed later in the chapter.

The approach to hydraulic analysis determines the level of uncertainty that can be incorporated into the design; hence selection of the appropriate hydraulic approach is of critical importance in terms of the robustness and cost-effectiveness of the design and the level of residual risk associated with it.

There are two distinct categories of hydraulic analysis:

- physical analysis
- numerical analysis (deterministic and probabilistic)

The analysis depends on the type and quality of the input data. It follows that limitations and uncertainties in the input data should be understood (and reported) and treated sensibly.

Physical analysis usually involves construction of reduced scale laboratory models; full-scale prototypes are used in a few rare cases. Physical analysis should be used where:

- the complexity of the hydraulics is beyond the practical application of numerical analysis;
- the assumptions in numerical models are not valid and would lead to wrong solutions.

Physical models are typically used for complex three-dimensional (3D) problems such as pumping station intakes; they are often ignored or discounted for many other situations due to the cost, time and space required. This can be a false economy, as there can be significant benefits from undertaking a physically based analysis.

The huge developments in computer capability and applications mean that many problems that would previously have been analysed using a physical model are now addressed thorough the use of numerical (computational) models.

Numerical analysis – subsequently referred to as ‘hydraulic analysis’ – is built upon fundamental hydraulic principles. These principles lead to mathematical equations (‘deterministic’) which are resolved to derive the physical properties of flow depth and flow velocity upon which all design works rely. Having a basic understanding of the hydraulic principles is important, because simplifications have to be applied to the equations and assumptions have to be adopted to resolve the equations for practical applications. These simplifications and assumptions lead to inherent uncertainties that need to be allowed for.

To simplify and speed up the process of undertaking hydraulic analysis, many hydraulic software packages have been developed (including HEC-RAS, ISIS, MIKE 11 and InfoWorksRS). The hydraulic design process often relies on the application of such software packages, which are classed as deterministic models. Therefore it is important to use a software package appropriate to the design requirements and the level of risk.

There is a move towards probabilistic flood modelling. This covers a range of modelling approaches that ideally allow for:

- uncertainty in input data (for example, defence crest level);
- inaccuracy in model structure (for example, inexact model representation of a physical process);
- inherent randomness of a natural process (for example, rainfall).

It is usual to contrast probabilistic methods with deterministic methods, though most probabilistic methods rely on the use of deterministic models to transform probabilistic inputs to probabilistic outputs. Discussion on the details of probabilistic methods is beyond the scope of this guide.

The software packages are often treated as a black box into which, if the right data are put in, produce the right answer. This approach is not acceptable and care is necessary in the application of such models, especially where new or specialist methods are being adopted such as integrated one-dimensional (1D) – two-dimensional (2D) models. A project undertaken as part of the joint Defra/Environment Agency Flood and coastal risk management programme (<http://www.defra.gov.uk/enviro/fcd/research/default.htm>) has benchmarked a number of 1D river modelling software packages to assess their accuracy and appropriateness for application to certain situations (Defra and Environment Agency, 2004a). As new software features and releases are made available, a steady stream of new publications assesses the models.

The type and level of detail required by a hydraulic analysis should be guided by understanding:

- the functional requirements for a design;
- the design risks;
- the design standards.

For example, if designing a floodwall in a highly urbanised area where there is a high level of residual risk associated with the design, a detailed analysis that fully represents the hydraulics should be favoured. But if the design is a short length of embankment along a drainage channel, then application of basic theory may suffice.

Application of basic theory at the planning stage of a more detailed study should always be considered, as it will often:

- help determine the level of detail appropriate for the study;
- identify any new data collection that may be required;
- provide an opportunity to consider wider benefits and additional uses of these models (for flood mapping, water resources, water quality or geomorphological assessment needs);
- lead to significant cost savings and improved decision-making.

7.2 The main issues

When undertaking hydraulic analysis and design, it is important to have a good understanding of the hydraulic processes that could occur under a range of flow conditions including typical flows as well as flood (design) flows and extreme flows.

It is essential to determine, early on in the analysis, the degree to which storage of floodwater will be important (that is, whether flows will be contained within the channel or whether there will be extensive flooding of floodplains). Knowing this determines the type(s) of analysis that are appropriate and helps identify data requirements.

When using software modelling tools to assist with hydraulic analysis, it is important to use the most appropriate modelling tool rather than merely the tool that is available. Sometimes the most appropriate tool is not necessarily the best technically. This could be due to cost or time constraints. In

this case it is imperative that the limitations of the chosen approach and the long-term implications are fully understood and clearly communicated.

The advent of sophisticated modelling software allows hydraulic analysis to be undertaken by those who are not necessarily experienced or educated in hydraulic theory. However, this guide recommends the use of competent modellers with a good grounding in basic hydraulics. At all stages of the hydraulic analysis and design, a suitably trained and experienced professional should define the scope of work, review progress and approve the final output. Failure to do this can have significant cost, time and technical implications and can ultimately result in failure to achieve a solution that is fit for purpose.

The availability of modelling software provides the temptation to undertake very detailed analysis, which can often be beyond the level of detail that is actually needed (or justified by the input data). This can add unnecessary cost and time to the project. The ability to produce very detailed outputs, which look very realistic when presented as graphs, can produce a false level of confidence. At all stages of the hydraulic analysis and design, it is therefore important to challenge the level of detail and the validity of the results.

Undertaking of hydraulic analysis can be both simple and complex. It can be done quickly or may take many months, and can require subjective interpretation. When undertaking any hydraulic analysis, it is therefore vital to keep detailed records of key decisions, changes made and checks undertaken. Failure to do this makes it difficult for a third party to pick up the analysis at a later stage and, more importantly, will make it almost impossible to provide the robust evidence needed to support the final design.

All hydraulic analyses have some level of uncertainty associated with them. For this reason it is important that an understanding of the uncertainty in the analyses is developed and communicated. In some cases, this may significantly alter design decisions.

7.3 Basic hydraulic concepts

In the fluvial environment, hydraulic concepts are principally considered in the context of open channels and structures that either constrain or control the flow of water.

An open channel is a watercourse, canal or conduit in which the water flows with a free surface. In most practical applications, the air above the water is at rest (or close to) and at standard atmospheric pressure; this assumption is applied in the development of the basic hydraulic theory. There are many books explaining hydraulic theory (for example, Henderson 1966) and its practical application.

When designing engineering works in the fluvial environment, the fluid is generally considered as either a fluid that is at rest (static) or a fluid that is in motion. Therefore, an understanding of both these concepts is needed in fluvial design.

7.3.1 Water level control

When designing structures to control water level (for example, embankments and gates) the designer needs to calculate the hydrostatic forces exerted to ensure the structure can carry the applied loads safely.

Fundamental hydraulic principles can be readily used to determine the hydrostatic forces on structures. For water at rest it is the weight of the water that is of primary importance in the design. [Figures 7.1 and 7.2](#) illustrate the static forces on a tidal barrier and radial sluice gates respectively.



Figure 7.1 Static forces on a tidal barrier

This large gated structure prevents tidal surges from entering the River Hull from the Humber estuary. Very large hydrostatic forces are generated by the differential head (difference in water level) across the vertical lift gate and these are resisted by massive monolithic foundations under the towers.

The photograph shows the gate being raised after equalisation of the water levels. It is parked in a ‘turned over’ position at the top of the towers to allow clearance for shipping.



Figure 7.2 Static forces on radial sluice gates

Radial gates like this one on the River Medway have certain advantages over vertical lift gates, including reduced lifting effort, because the hydrostatic force is conveyed to a pivot. The disadvantages of radial gates are that they require longer structures to accommodate the radial arms, and the pivots require heavily reinforced concrete to carry the large concentrated load.

7.3.2 Flow in open channels

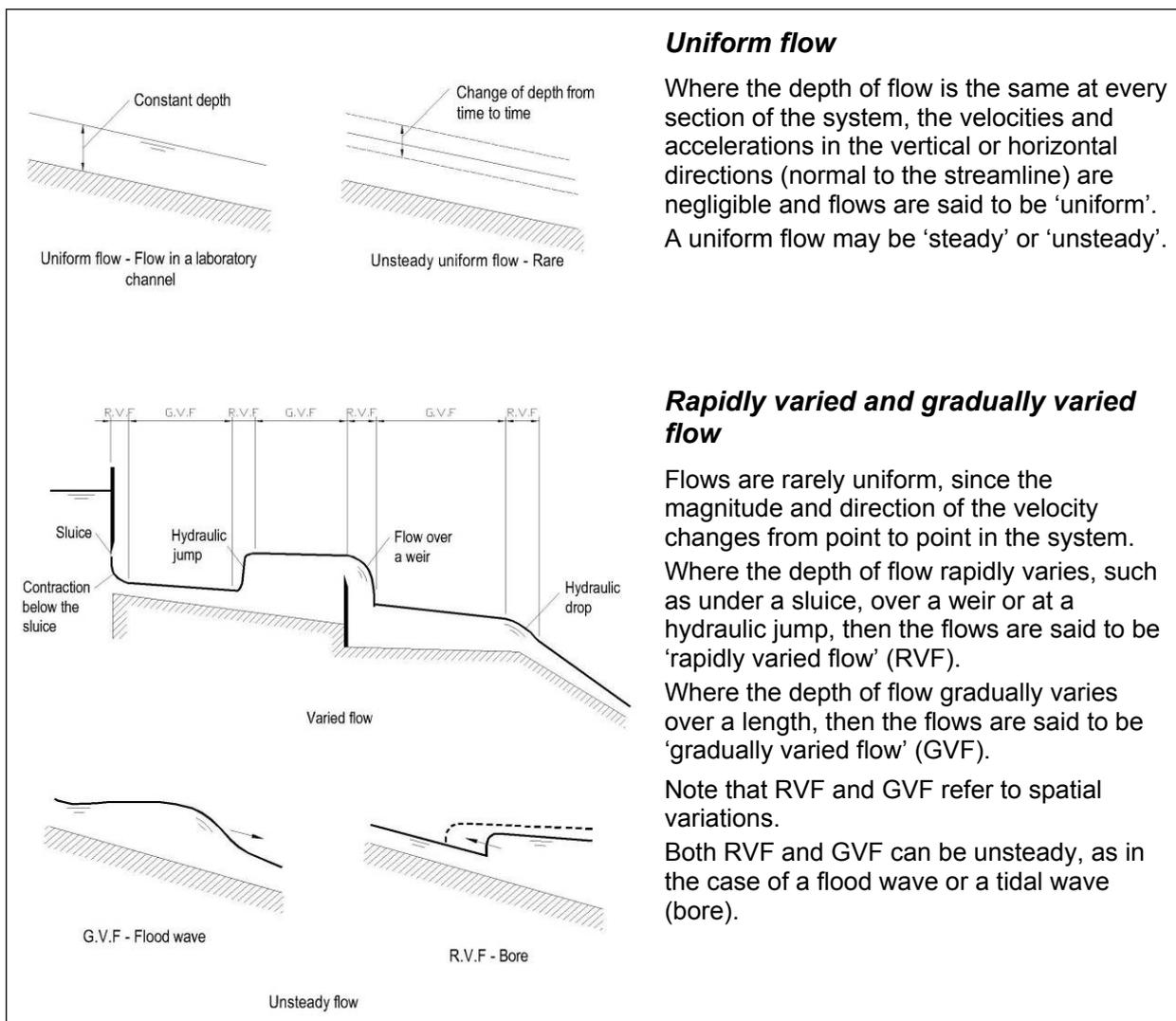
When undertaking any fluvial design, it is necessary to determine the flow of water (in motion) in open channels and the flood path. The flow is classified as being either one of two time-related types – steady or unsteady. For steady flow conditions, the flow at a given section remains constant with time, whereas for unsteady flow conditions the flow changes with time. Within both of these flow types the flow can be further classified as shown in Table 7.1.

Table 7.1 Classification of flow type (steady or unsteady)

Class	Description
Uniform	Where the velocity and depth of flow is reasonably constant with distance; typically where there are no hydraulic structures or significant changes in channel cross section.
Gradually varied	Where the velocity and depth of flow change with distance; typically due to changes in channel width or depth or local influences from structures. For example, immediately upstream of a weir.
Rapidly varied	Where velocity and depth of flow change rapidly, typically due to the influence of hydraulic structures. For example, a hydraulic jump downstream of a weir or when flow is out of the main channel.

Being able to categorise and conceptualise the flow classifications within open-channel systems is a useful means of understanding the hydraulics and the needs of design. **Box 7.1** uses a conceptual building block to explain these classifications (based on Chow, 1959).

Box 7.1 Understanding flow classifications



The most common hydraulic flow type occurring – and hence the problem most analysed in fluvial design – is steady uniform flow (see Figure 7.3).

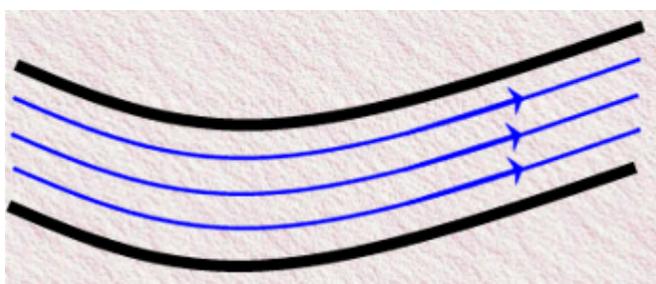


Figure 7.3 Steady uniform flow

In many practical fluvial design situations the flow depth, bed slope, velocity, and channel cross section can be assumed to remain constant over a given length of channel. This results in both steady and uniform flow conditions, more commonly referred to as steady uniform flow. In this case the energy grade line, hydraulic grade line, and channel bottom are all parallel, which is an important assumption in the development of hydraulic theory.

Representation of flow

The simplest representation of flow in a channel is one-dimensional. The HEC-RAS, ISIS and MIKE 11 software packages all reduce the theory to one dimension and resolve the flow depth and

discharge at defined river cross sections or at structures within the system. This approach (see Figure 7.4) is generally satisfactory where flows are predominantly in-bank, or where assumptions made for out-of-bank flows are appropriate and the uncertainty is not critical to the design process.

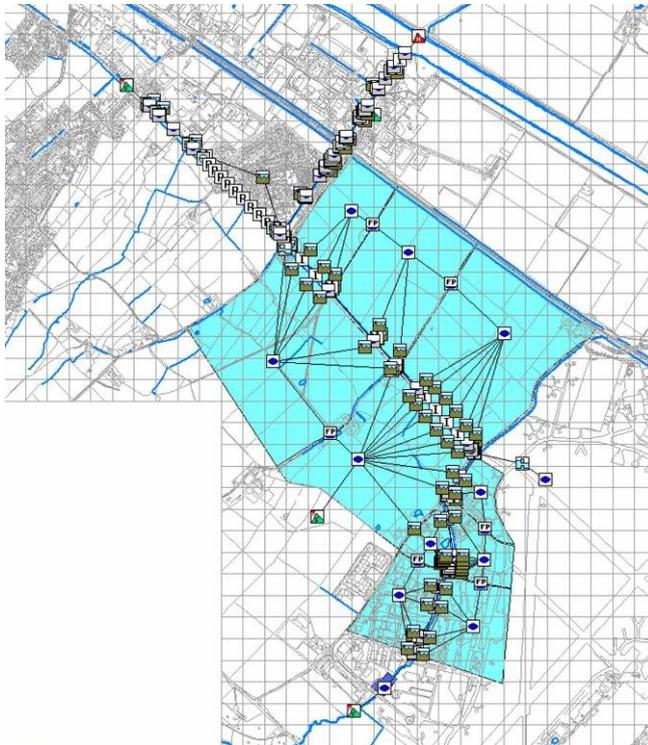
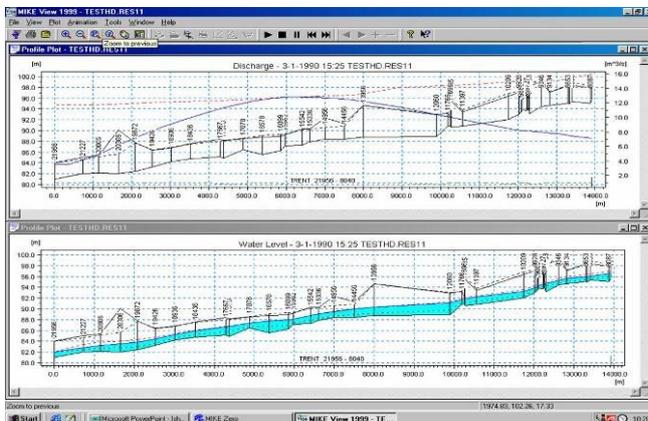


Figure 7.4 One-dimensional flow

The first illustration is a typical 1D model schematisation (build up) for ISIS. Here the main channel is modelled with 1D channel cross sections ('river section') and out-of-bank flow is modelled by a 1D link ('spill unit') connecting the channel cross sections with storage cells ('reservoir unit'). This essentially builds up a 2D network from a series of 1D connections. A limitation of this approach is the schematisation, which is determined by the modeller and which may not appropriately represent the flowpaths once it is out-of-bank.

It is important at the scoping stage of a project to understand the flow routes and the level of detail needed to represent these in an appropriate manner. Where the flowpaths could be complex or inefficient to set up, higher dimensional modelling may be more appropriate.



The second illustration shows the typical depth (of flow) and discharge (flow) outputs from MIKE 11. As the model is 1D, results data are determined only in a single horizontal direction for each modelled location ('node'). This provides a single depth and discharge (and thence velocity) value that is applicable for the whole cross section.

Basic hydraulic theory should be used to validate results (that is, to confirm that they are realistic).

Flow depths and velocities sometimes need to be known more accurately, for example:

- on complex floodplains;
- in urban areas where flowpaths need to be defined;
- where hazard analysis is needed;
- for geomorphological (erosion or deposition) analysis.

In these situations, the solution of the hydraulic theory in two dimensions is beneficial (see Figure 7.5). With increasing computing power, greater data availability and new software developments, the adoption of two-dimensional methods is becoming more common.

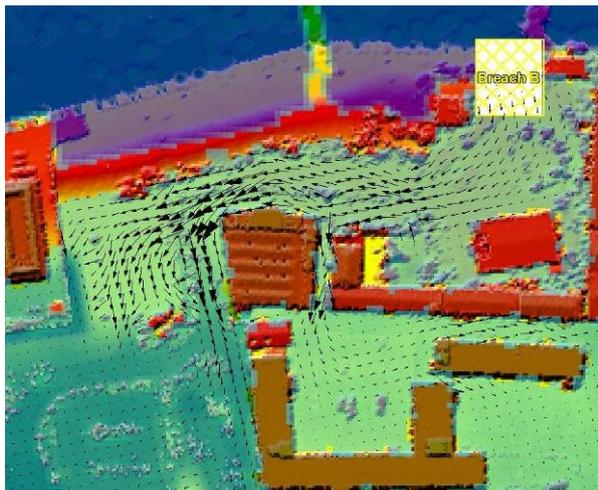


Figure 7.5 Two-dimensional flow

The illustration is a typical 2D velocity output for TUFLOW in an urban area. Here a digital terrain model is used to build up a 2D mesh across the whole modelled area at a suitable grid size, providing significantly greater resolution than a 1D model. The hydraulics are solved at the grid size level in two horizontal dimensions.

Results data provide the horizontal direction of flow velocity as well as flow depth at each modelled grid ('node') location.

Care needs to be taken, as many factors influence the validity of the results from 2D models, including the limitations of the data, the capability/suitability of the software and the model set-up. Just because the results look good it does not mean that they are right!

For most engineering purposes, basic hydraulic theory combined with 1D or 2D analysis is sufficient. There are a few occasions where the solution of the hydraulic theory in three-dimensions is needed (see Figure 7.6). The most frequent and practical application of 3D analysis is in short lengths (typically less than 200m) of open channel or small areas where a greater understanding of the flow phenomena is required. Three-dimensional analysis is generally avoided due to the computational power requirements, the extensive data requirements, the cost of the software and the complexity of the theory.

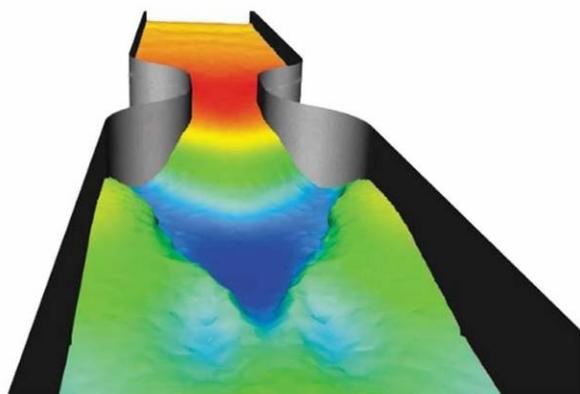


Figure 7.6 Three-dimensional flow

Flow through a flume exhibits significant velocity variations in the vertical dimension. As the flow passes through the throat of the flume, the water level is rapidly drawn down and then immediately further downstream of the throat there is a hydraulic jump.

These rapid velocity variations in the vertical combined with significant horizontal variations require 3D solution if a detailed understanding of the flow is needed.

The different colours show the change in velocity and hence flow depth.

High turbulence

Designers need to consider situations of high turbulence, as the flow is unstable and can lead to bed or bank erosion due to high frictional forces (see Figure 7.7). These forces should be avoided or appropriately designed for.

Typical areas to be aware of include:

- immediately downstream of structures;
- the interface between in-bank and out-of-bank flows;
- steep sections of river (although these are often where the bed is more resistant to erosion).



Figure 7.7 High turbulence and frictional forces

A spillway (a steep channel) should always be designed for high turbulence and high frictional forces, and for controlling the formation of a hydraulic jump at its toe.

The photograph illustrates how highly turbulent flows can contribute, (amongst other contributing factors in this case) to structural damage to a spillway.

The spillway damage and erosion of the ground beneath propagated up the spillway and into the dam embankment, causing significant damage.

Steady or unsteady flow?

If the time element is not of major concern (that is, the flow at a given section remains close to constant), then for many practical design applications, the assumption of steady flow can be considered and then applied to many unsteady flow conditions. Typical examples of such an approach include:

- the design of structures such as culverts, weirs and sluices which can be designed for a specific flow carrying capacity;
- the design of retaining structures (such as embankments and walls) where the flows remain in-bank so that the structure can be designed for a specific water level.

The time element and thence unsteady flow is vitally important to the design process in the case of:

- flood (and tidal) wave propagation and surge, where the depth of flow changes almost instantaneously as the waves pass by;
- flood storage (either online or offline) within the system.

7.4 Fundamental hydraulic principles

Three significant concepts or principles form the building blocks for all types of hydraulic analysis:

- energy (and conservation of energy);
- mass (and continuity);
- momentum.

This guide describes the basic hydraulic principles only for steady one-dimensional flow (see [Box 7.2](#)). Many textbooks provide detail on the theory and how the fundamental principles are developed to account for unsteady flow.

Box 7.2 Basic hydraulic principles for steady one-dimensional flow

Energy equation and conservation of energy

The principle of energy in fluid flow for practical design purposes can be considered in two parts:

Potential energy refers to the energy a fluid has due to its elevation:

$$PE = Wz$$

where:

W and z are respectively the weight of water and the distance the water is located above the reference point (or datum). The units for PE are $\text{kg m}^2/\text{s}^2$ or joules.

Kinetic energy refers to the energy possessed by fluid flow due to its velocity

$$KE = \left(\frac{1}{2}\right)(WV^2/g)$$

where:

V and g are respectively the velocity of the water and the acceleration due to gravity. The units for KE are $\text{kg m}^2/\text{s}^2$ or joules.

Hydraulic head

Potential and kinetic energy can be expressed in terms of 'head' (i.e. with the element of weight removed). The sum of potential (elevation) head and kinetic (velocity) head is referred to as 'total head' (H).

$$H = y + (V^2/2g)$$

where:

y is often referred to as the elevation head and is equal to depth of flow if the channel bed is the datum

There is also a pressure head term, but this is usually neglected.

$V^2/2g$ is referred to as the velocity head.

The units for H are metres (m).

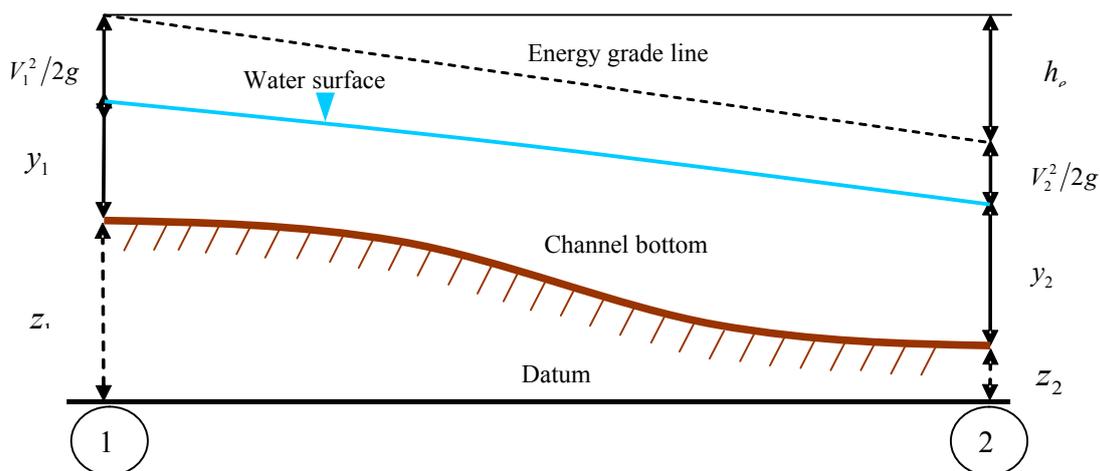
Energy equation

The principle of conservation of energy requires that the total energy head at the upstream section 1 should be equal to the total energy head at the downstream section 2 plus the loss of energy

$$z_1 + y_1 + V_1^2/2g = z_2 + y_2 + V_2^2/2g + h_e$$

where:

z , y and h_e are respectively the distance the channel bed is above datum, the average depth of water above the channel bed, and the loss of energy.



The energy grade line should always drop in the direction of flow except if energy is added to the system by a mechanical device.

Mass and continuity

The principles of conservation of mass are used to develop the equation of continuity.

For steady flow in open channels, this is expressed in terms of discharge (or flow rate).

$$Q = A_1 \times V_1 = A_2 \times V_2$$

where:

A and V are respectively the cross-sectional area and average velocity at locations 1 and 2. The units for discharge are m^3/s .

Momentum

Momentum can be defined as ‘mass in motion’ and is equal to the mass times the velocity.

Momentum has both magnitude and direction.

$$p = m \times V$$

where:

m is the mass and V is the velocity. The units for momentum are kg m/s .

To solve unsteady hydraulic problems, these equations need to be expanded to include the time component – although in doing so their solution becomes more complex. Fortunately software packages such as ISIS, InfoWorks RS, MIKE 11 and HEC-RAS have both steady and unsteady solvers for these equations.

The implementation of these equations differs from software to software; hence a software package may not be suitable for all hydraulic problems and they all have limitations. This is particularly true for software that solves the theory in two dimensions. For example, MIKE 21 and TUFLOW solve the principles of both mass and momentum, whereas JFLOW solves only the principles of mass. MIKE 21 and TUFLOW have a finite difference method of solution on a rectangular grid whereas InfoWorks 2D has a finite volume method of solution on a triangular grid.

7.4.1 Normal, subcritical and supercritical flows

The **normal flow** is the flow in an open channel in which the slope of the water surface and channel bottom is the same, and the water depth remains constant along the channel reach. Normal flow gives rise to uniform velocity and depth (normal depth). **Subcritical flow** occurs when the flow velocity is less than ‘critical’ and is encountered where the channel is not steep.

Supercritical flow (see Figure 7.8) occurs when the flow velocity is greater than the critical velocity, and occurs where the bed slope is steep or where potential energy is converted to kinetic energy over a short distance.



Figure 7.8 Supercritical flow

Supercritical flow can be maintained over a long length of channel, for example in a spillway channel carrying flood flow past a dam. Here the supercritical flow has high velocity, is highly turbulent, and has high frictional forces. In this instance, the under-design of this spillway (channel) is resulting in erosion of the bed and the bank.

Subcritical flow can only change to supercritical flow by going through the point of critical depth, and supercritical flow can only change back to subcritical flow by going through a hydraulic jump (see

below). This has practical importance in that, if the flow goes through the critical depth, the discharge can be calculated using a simple expression based on the upstream water level and the channel geometry at the point of critical depth (see [Box 7.9](#)).

For subcritical flow, downstream changes in flow depth impact on water levels upstream. This is an important hydraulic principle in fluvial design and is discussed further in the section on weir hydraulics (see [Section 7.4.5](#)).

Hydraulic jumps

Hydraulic jumps form when flows at high velocity enter a zone of the river or engineered structure that can only sustain a lesser velocity. They also form when a rapid flow encounters a submerged object which then throws the water upwards. When this occurs, rapidly flowing water is abruptly slowed and rapidly increases in height in the form of a step or standing wave, converting some of the flow's initial kinetic energy into an increase in potential energy, with some energy irreversibly lost through turbulence to heat. The jump is also commonly accompanied by eddying, air entrainment and surface undulations. [Figure 7.9](#) shows an example of jump formation.



Figure 7.9 Changes in flow regime at a weir

The uniform channel upstream of this weir exhibits subcritical flow. At the weir crest, the flow passes through critical depth. On the downstream face of the weir, the flow is supercritical with a high velocity. Immediately downstream of the weir, part of the kinetic energy of the flow is converted to potential energy and a hydraulic jump is formed. Downstream of the hydraulic jump, the flow returns to subcritical flow.

Note the provision of erosion protection on the banks in the area of high turbulence downstream of the weir.

It is important to know where and under what conditions a jump will occur and the point at which the normal depth of flow (see [Section 7.4.3](#)) is achieved downstream, so that bed protection can be appropriately placed and designed. With proper hydraulic design, the formation of hydraulic jumps can be an effective means of dissipating energy at hydraulic structures.

Change from subcritical to supercritical

The change from subcritical to supercritical occurs whenever the *Froude number*, Fr , exceeds unity (that is, $Fr > 1.0$) ([Box 7.3](#)).

Box 7.3 Froude number

If $Fr < 1$, we call the flow subcritical.
If $Fr > 1$, we call the flow supercritical.

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

where:

V , g and y_m are respectively the velocity of the flow, acceleration due to gravity, and the average depth of flow relative to the water surface (cross-sectional area of flow divided by surface width).

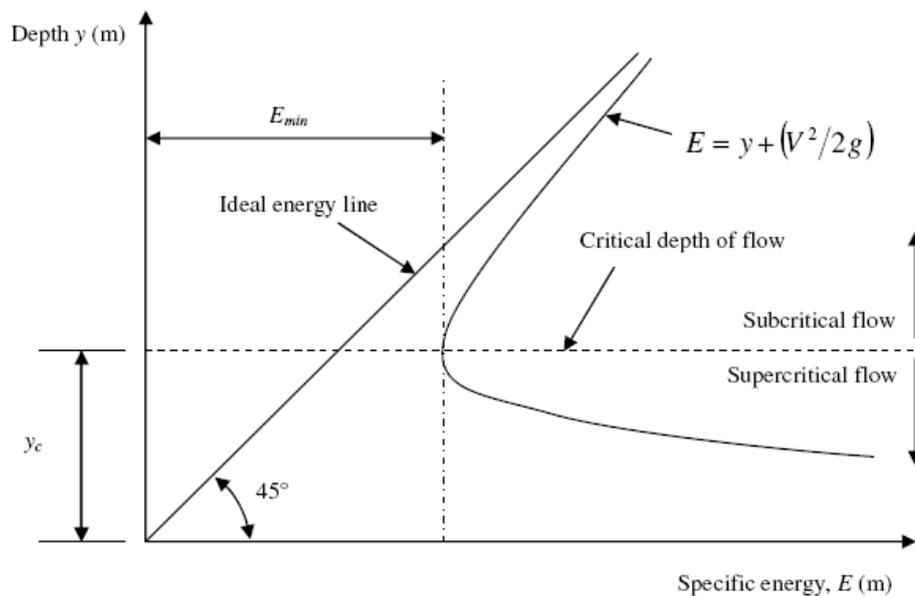
When the Froude number is unity (1.0), then flow is at the critical depth of flow. This is also referred to as the depth of control.

If the depth of flow is plotted against the specific energy for a given channel section and discharge, Q , then the specific energy in the channel section is a function of the depth of flow only (Box 7.4).

Box 7.4 Specific energy curve

Specific energy is derived from the energy equation. It is defined as the energy per unit weight relative to the bottom of the channel.

$$E = y_1 + V_1^2/2g$$



At the minimum energy, the curve has only one depth solution – the critical depth of flow; at all other points, the curve shows that there are two possible depths. For depths above the critical depth, the flow is termed subcritical flow (or tranquil) and, below critical depth, it is termed supercritical flow (or shooting). By convention the ideal energy line represents subcritical flow and is drawn at 45 degrees (see graph).

7.4.2 Equations for open-channel flow

The equations described up to this point do not take into account any frictional energy losses. Several empirical formulae have been developed to enable solution of the energy equation taking account of frictional energy losses. These equations allow calculation of the conveyance capacity of channels.

One of the most commonly used formulae was introduced by the Irish engineer Robert Manning in 1889 and is known as Manning's equation. This empirical equation applies to uniform flow in open channels, as described in Box 7.5. The conveyance estimation system (CES; see Section 7.4.3) provides advice on surface friction or 'roughness', and should be used as the latest source of information.

The Colebrook-White equation (HR Wallingford and Barr, 2006) is seldom used in open-channel fluvial flow conditions, but has wider application in closed conduit flow and in artificial channels, such as water treatment works.

Box 7.5 Manning’s equation

Manning’s equation defines the mean channel velocity:

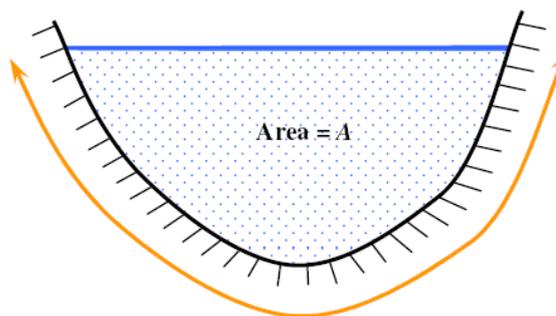
$$V = (1/n)R^{2/3}S^{1/2}$$

where:

V , R , S , and n are respectively the mean velocity (m/s), hydraulic radius (m), slope of the energy line (m/m) and the Manning coefficient of roughness.

The hydraulic radius is defined as:

$$R = A/P$$



To determine the energy gradient:

In an idealised case, the slope of the energy line (also known as the friction slope) is the same as the water surface slope and the bed slope. Hence for most practical purposes, the energy gradient can be taken as the water surface slope. The water surface slope can be determined from surveying the water level over a significant distance.

To determine Manning’s ‘ n ’:

The Manning ‘ n ’ coefficient is a function of bed material, vegetation growth, channel irregularities, obstructions and shape and size of channel (see typical values in Table 7.2).

Table 7.2 Typical Manning’s ‘ n ’ values

Type of channel and description	Minimum	Normal	Maximum
<i>Main channels:</i>			
▪ clean, straight, full stage, no riffles or deep pools	0.025	0.030	0.033
▪ clean, winding, some pools and shoals	0.033	0.040	0.045
▪ sluggish reaches, weedy, deep pools	0.050	0.070	0.080
▪ very weedy reaches, deep pools	0.075	0.100	0.150
<i>Floodplains:</i>			
▪ short grass	0.025	0.030	0.035
▪ mature field crops	0.030	0.040	0.050
▪ medium to dense brush, in summer	0.070	0.100	0.160
▪ dense willows, summer, straight	0.110	0.150	0.200

Note: This is a simplified version of a table originally proposed by Chow (1959). Much more comprehensive information is available from the roughness advisor in the CES (see below).

7.4.3 Calculation of depth flow or discharge

Steady uniform flow equation

A common method of calculating the normal depth of flow or the normal flow (discharge) is to use the steady uniform flow equation (Box 7.6), which combines Manning’s equation with the continuity equation.

Box 7.6 Steady uniform flow equation

Combining Manning’s equation with the continuity equation, the expression for ‘steady uniform flow’ can be derived.

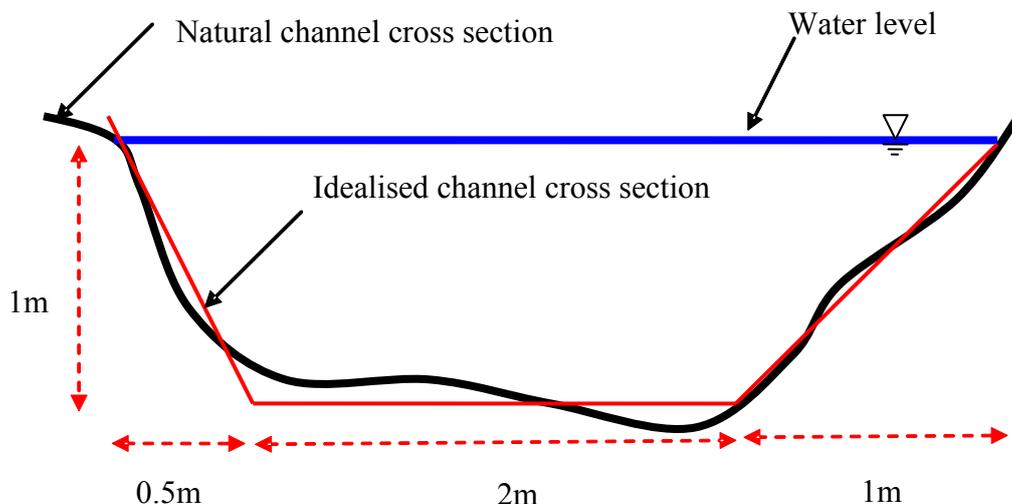
$$Q = A(1/n)R^{2/3}S^{1/2}$$

Manning’s equation is a useful and quick method of determining either the depth of flow (if discharge is known) or discharge (if depth is known) when in the field and when checking the validity of results from a hydraulic software package. Box 7.7 shows an example calculation of discharge.

Box 7.7 Example hand calculation of discharge

Determine the discharge when:

- the uniform depth of flow is 1m for a natural channel;
- where the shape has been reduced to a trapezoidal channel with dimensions as given below;
- has a water surface slope of 1 in 500;
- has a channel roughness equivalent to a Manning’s ‘n’ of 0.050.



(1) Calculate the area (m²).

$$A = (1 \times 0.5 \times 0.5) + (1.0 \times 2.0) + (1 \times 1 \times 0.5) = 2.75$$

(2) Calculate the wetted perimeter (m).

$$P = \sqrt{1^2 + 0.5^2} + 2 + \sqrt{1^2 + 1^2} = 4.53$$

(3) Calculate the hydraulic radius (m).

$$R = 2.75 / 4.53 = 0.61$$

(4) From steady uniform flow equation, determine the discharge (m³/s).

$$Q = 2.75 \frac{1}{0.050} 0.61^{2/3} (1/500)^{1/2} = 1.76$$

Although the area and water surface width are similar, the idealised channel will have a smaller wetted perimeter, and therefore the calculated flow will be slightly over estimated. In addition, the selection of Manning’s ‘n’ can have a significant impact

Character of channel	n	Q (m ³ /s)
Clean, straight channel	0.025	3.53
Clean, winding channel	0.033	2.67
Sluggish weedy channel	0.050	1.76
Very weedy channel	0.075	1.18

on the discharge as illustrated in the table to the right.

The steady uniform flow equation can be expressed in terms of a parameter known as ‘conveyance’ (Box 7.8).

Box 7.8 Conveyance equation

Conveyance relates the total discharge to a measure of the gradient or slope of the channel.

$$Q = KS^{1/2}$$

where K is conveyance (m^3/s).

$$K = A(1/n)R^{2/3}$$

Manning’s equation, which is empirically derived, is not based on rigorous physics and can provide unreliable results in cases where the overall shape of the flow cross section is complex, an example being a river in flood, with much shallower flow on the floodplain than in the main channel. Various techniques have been developed to account for this, one such example being the ‘divided channel method’ as used in the HEC-RAS software package (US Army Corps of Engineers, 2008). In this method, the flow rate is calculated separately for the floodplain and main channel zones (as illustrated in Figure 7.10) and subsequently summed, though the lateral shearing and consequent momentum transfer between the vertical divisions or ‘slices’ is ignored.

Manning’s equation thus represents all the energy losses through the ‘ n ’ value; it lumps all the physical flow processes such as shearing and momentum transfer into one ‘catch-all’ parameter, with little understanding and interpretation of the independent energy ‘loss’ mechanisms.

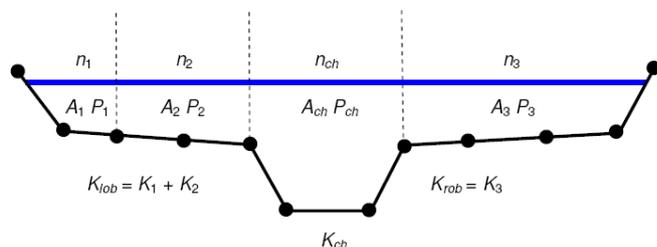


Figure 7.10 Default conveyance subdivision method as used in HEC-RAS

Conveyance estimation system

The development of the ‘conveyance estimation system’ (CES) was commissioned under the joint Defra/Environment Agency flood and coastal erosion risk management programme (Defra and Environment Agency, 2004b) to provide a new method of estimating the conveyance or carrying capacity of a channel. It incorporated recent advances in the understanding of shearing and momentum transfer flow mechanics and physical flow processes.

The CES has three primary components, which are used when determining site-specific stage-conveyance curves. These components are:

- roughness advisor – roughness values (see Figure 7.11);
- conveyance generator – stage-conveyance relationship (see Figure 7.12);
- uncertainty estimator – upper and lower bands for the stage-conveyance relationship.

CES methods have been integrated into the 1D river-modelling software packages ISIS and InfoWorks RS.

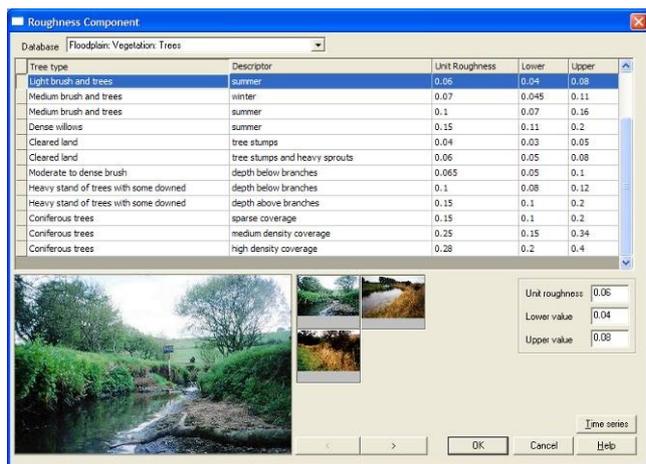


Figure 7.11 CES roughness advisor

The roughness advisor is used to determine ‘unit’ roughness values (these are independent of cross section and planform shape).

Photographs are provided to help the user select the most appropriate values based on vegetation, bank, bed and ground material types (sediment size) and local irregularities (such as trash or groynes).

The roughness advisor also includes information on seasonal variations in vegetation roughness that should be considered.

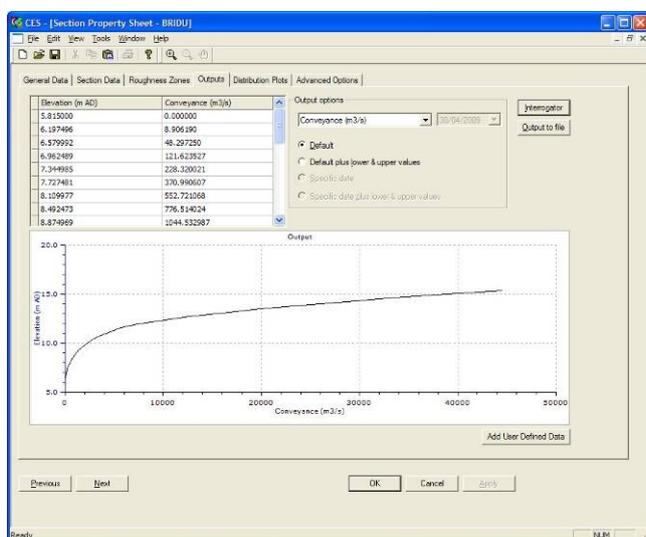


Figure 7.12 CES conveyance generator

The conveyance generator allows users to assign the unit roughness values to zones within cross sections and then calculate stage/conveyance relationships.

An inbuilt backwater calculator is provided to enable water levels to be determined for defined flows and downstream water levels.

7.4.4 Backwater profiles

A backwater profile forms in a channel where the depth is raised above the normal depth of flow and the effect is felt upstream. The backwater length is the distance upstream before normal depth is re-established.

The calculation of backwater effects is important because:

- it enables the determination of the upstream influence of works in a river channel;
- the backwater length may extend far upstream, potentially causing inundation in areas remote from the study reach;
- flow gauging stations should not be located within reaches influenced by backwater effects;
- the backwater profile is useful for the operation of land drainage pumps to avoid frequent switching on and off of the pumps, leading to increased wear and tear and thence reduced operational life.

In land drainage areas, the backwater length tends to be long and hence the extent over which water levels are affected is important.

Backwater profiles tend to occur upstream of structures such as bridges or weirs. Figure 7.13 shows a typical backwater profile.

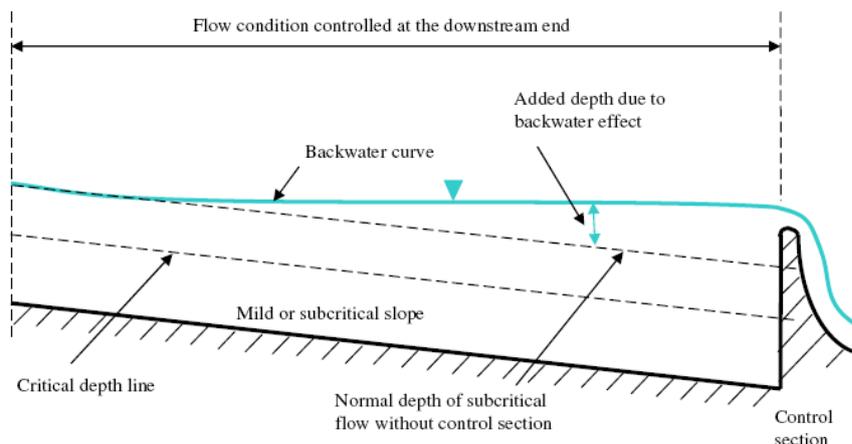


Figure 7.13 Typical backwater profile

As a guide the backwater length can be approximated from the equation $0.7 \times \text{depth}/\text{gradient}$ (using consistent units of measurement). Typical backwater lengths in the UK are shown in Table 7.3.

Table 7.3 Typical UK backwater lengths

River	Location	Depth, D (m)	Slope S (m/km)	Typical backwater length (km)	Estimated backwater length $0.7D/S$ (km)
Avon	Pershore	4.2	0.46	6.9	6.4
Cleddau	Haverfordwest	2.5	1.70	1.0	1.0
Lagan	Dunmurry	2.5	1.20	1.4	1.5
Lagan	Lisburn	3.0	0.29	7.2	7.2
Lagan	Banoge	2.0	1.30	1.1	1.1
Little Ouse	Lakenheath	2.5	0.05	35	35
Rhymney	Llanedeyrn	2.0	2.20	0.64	0.64
Severn	Shrewsbury	4.5	0.50	6.9	6.3
Severn	Tewkesbury	8.5	0.10	59	59
Soar	Loughborough	3.5	0.25	9.8	9.8
Stour	Christchurch	4.5	1.0	3.2	3.2
Taff	Cardiff	6.0	2.5	1.7	1.7
Wharfe	Otley	2.0	1.8	0.76	0.78

(Taken from Samuels, 1989)

The backwater can be calculated iteratively, starting from a downstream control or cross section of known depth and flow rate, and moving upstream applying energy balances at each consecutive section.

The backwater effect is present until normal depth is achieved at some backwater length upstream from this downstream control. Where either the flow rate or the flow depth is known, the backwater approach may be used to calculate the corresponding normal depth of flow or discharge respectively. In this latter case, the backwater profile is essentially a normal depth profile upstream of this known quantity. Further details and examples of how to undertake a backwater calculation can be found in most hydraulic textbooks.

Limitations of the backwater calculation method are as follows.

- Initial downstream conditions are needed (that is, the flow and depth are known at the ‘control point’).
- The method is for subcritical flow only.
- For subcritical flow where the channel bed is steep, the calculation may result in a negative upstream energy head (and hence a negative depth) being calculated.
- The method is limited to application in single reaches with no branches, confluences, junctions, loops or spills into floodplain storage areas.
- The method applies only to steady flow conditions (with the flow rate constant with time).

7.4.5 Flow at structures

Structures are commonly found in most open channels. Their impact on the flow can range from being significant to minimal depending upon the design, location, orientation and intended purpose. The most common structures in the UK are weirs, culverts, bridges and sluices.

Weirs

Weirs provide one of the most common forms of flow monitoring or water level control structures throughout the world.

There are several types of weir including the broad-crested, sharp-crested, ogee, Crump and flat-vee. Sharp-crested weirs are normally used in laboratories, but can be used as temporary flow measuring structures on small watercourses.

All weirs work on the principle that the flow over the weir must go through the critical depth. It is the height of a weir that determines whether or not the flow goes critical. Once this happens a formula for discharge can be developed using the concept of specific energy and the special conditions that occur at the critical point.

Figures 7.14 and 7.15 show free flow (sometimes called ‘modular flow’) and drowned flow over a weir respectively.

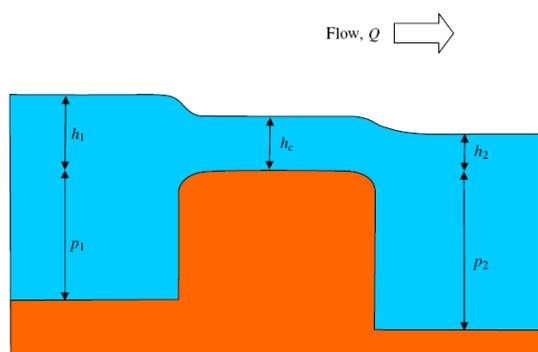


Figure 7.14 Free flow over a broad-crested weir

When flow passes over the weir under free flow conditions, the flow moves from subcritical to supercritical. This means that downstream changes in water level do not affect the upstream water level, so the flow over the weir can be assessed by measuring the upstream water level only. There is some drawdown close to the weir, hence the head h_1 is usually measured a few metres upstream or in a stilling chamber by the side of the channel.

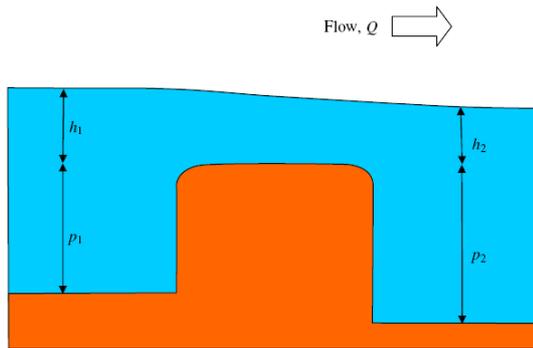


Figure 7.15 Drowned flow over a weir

The free flow condition disappears if the downstream water level rises to the point whereby supercritical flow over the weir crest no longer occurs. In this situation the weir is said to be drowned. The flow can only be estimated if both upstream and downstream water levels are available.

The flow over a weir can be calculated from the standard weir equation (Box 7.9).

Box 7.9 Standard weir equation

This equation links the channel discharge (Q) with the upstream water depth measured above the weir crest (h):

$$Q = CBh^{1.5} \text{ (m}^3\text{/s) where: } C, B \text{ and } h \text{ are respectively the weir coefficient, length of the weir crest (m), and head on the weir measured from the weir crest (m).}$$

Strictly speaking, h is the measurement from the weir crest to the energy line as it includes the velocity head (kinetic energy term); in practice h is measured from the weir crest to the water surface. The error involved in this is relatively small and can be taken into account in the value of the weir coefficient C . The value of C for a broad-crested weir is theoretically 1.705 in metre-second units and this is usually rounded to 1.70. This value applies if h includes the velocity head, so the practical value is normally greater.

As the formula is based on critical depth it is not dependent on the shape of the weir. Consequently the same formula can be used for any critical depth weir and not just for broad-crested weirs. The value of C changes to take account of the different weir shapes; higher values of C apply to various forms of short-crested weir, such as the Crump weir. The flat-vee weir is a special case, where the exponent also changes from the standard value of 1.5.

Practical guidance on the design of weirs is given in Sections 8.3.3 and 11.3.2.

Culverts

The hydraulic analysis of flow in culverts is complicated and the details are beyond the scope of this guide. The detailed hydraulic design of culverts is extensively covered in text books, British Standards and various design manuals such as CIRIA's *Culvert design guide* (Ramsbottom *et al*, 1997) which is currently being updated and revised. However, it is important to understand the principles of culvert hydraulics for the purpose of design.

A culvert consists of the following components (see Figure 7.16) which have specific hydraulic characteristics and performance:

- inlet, where water enters the culvert;
- barrel(s) of length L (m), and height D (m) through which water is passed;
- outlet, where water leaves the culvert.

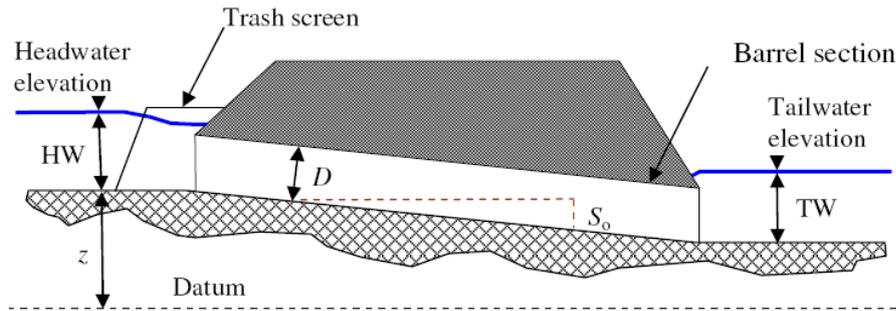


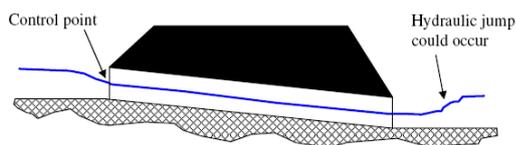
Figure 7.16 Culvert components

The barrel is normally laid on a slope (S_o) and the depth of water above the upstream and downstream invert levels is commonly known as the headwater depth (HW) and the tailwater depth (TW) respectively. Figure 7.16 shows the culvert flowing full (surcharged); in this case the barrel slope has little or no effect on the flow.

It is common for the flow velocity through a culvert to be greater than that in the channel upstream and downstream. This is for reasons of economy (a smaller culvert is cheaper to construct) and to reduce the risk of sediment deposition in the culvert barrel. This though increases the risk of erosion at the culvert outlet and carries the penalty of increased headloss (the difference between headwater elevation and tailwater elevation). In fact the headloss through the culvert is a function of $V^2/2g$, so that doubling the velocity quadruples the headloss. This fact is critical when considering the impact of culverting a stream, since any rise in upstream water level may pose a flood risk. This is one of the reasons why the Environment Agency has a policy that discourages the culverting of streams.

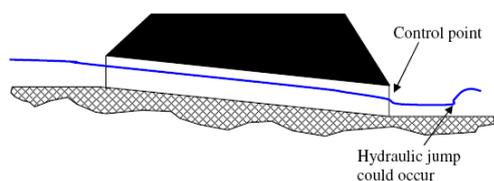
The flow through culverts is characterised by the type of flow that occurs at the inlet, outlet and through the culvert. Descriptions of the different types of flow are given in Box 7.10. Note that the flow type numbers are those used in *Culvert design guide* (Ramsbottom *et al*, 1997). Other references (including Chow, 1959) use different numbering systems.

Box 7.10 Types of flow through culverts



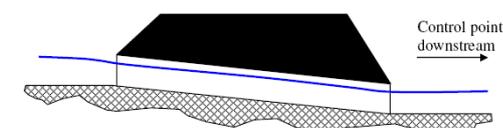
Type I – Critical depth at inlet

The culvert flows with a free surface and the flow is controlled at the inlet. Flow passes through critical depth slightly downstream of the inlet and remains supercritical for the full length of the barrel (see also Figure 7.17).



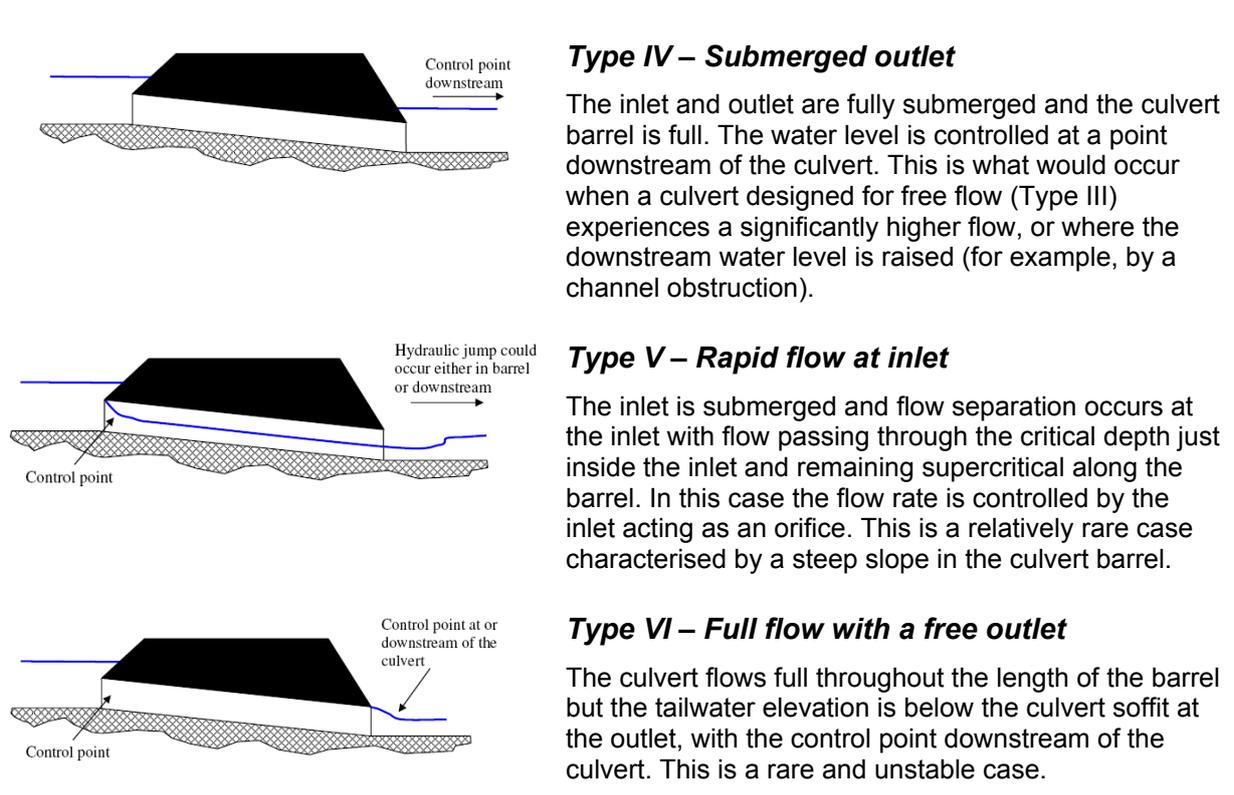
Type II – Critical depth at outlet

The culvert flows with a free surface and the flow is controlled at the outlet. The culvert barrel slope is less than critical slope and the water surface passes through critical depth at the outlet. This is a relatively rare case characterised by a drop at the outlet or a steepening of channel slope downstream of the outlet.



Type III – Subcritical flow throughout

The culvert flows with a free surface throughout and the water level is controlled downstream of the outlet. This is the most common flow type and is the case that should generally be adopted for the design of a new culvert, unless there are overriding reasons for a steep culvert or surcharged flow.



In culvert hydraulics, the concepts of outlet control and inlet control are used to simplify the analysis. With outlet control it is the channel downstream or the culvert barrel itself that acts to restrict flow. With inlet control it is the inlet that controls the flow, and changes to the water level downstream of the culvert do not affect the discharge through the culvert. Inlet control takes two forms, one where the inlet acts like a weir (Figure 7.17 and Type I flow in Box 7.10) and the other where the inlet acts as an orifice (Type V flow in Box 7.10).

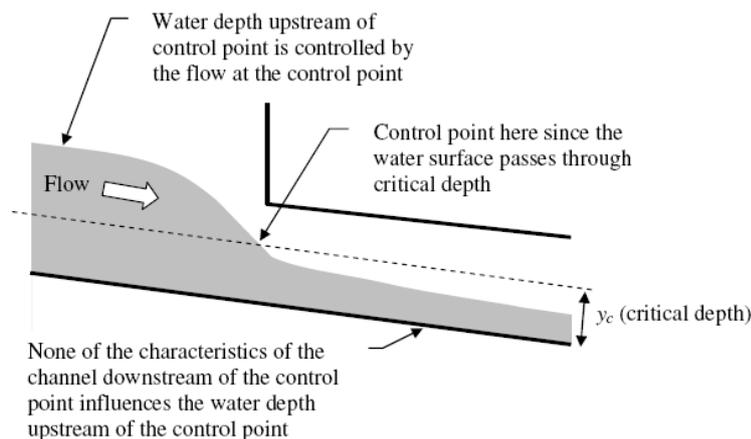


Figure 7.17 Inlet control

Inlet-controlled flow occurs when the flow carrying capacity of the culvert entrance is less than the flow capacity of the culvert barrel. The controlling factors are:

- inlet area, shape and edge configuration;
- allowable headwater elevation.

This form of control is a feature of steep culverts (that is, those with supercritical flow in the barrel).

The hydraulic design of a culvert is generally based on an estimated flood flow (for example, the 1% annual exceedance probability – see Section 2.4.1). However, it is important to check the hydraulic design for higher flows if only to see what the consequences would be (particularly in terms of flood risk and flow pathways). Wherever practical, a culvert should be designed for free flow (see Chapter 8). Culvert performance curves (see Figure 7.18) provide a useful check on the design and show if control changes from inlet to outlet or vice versa.

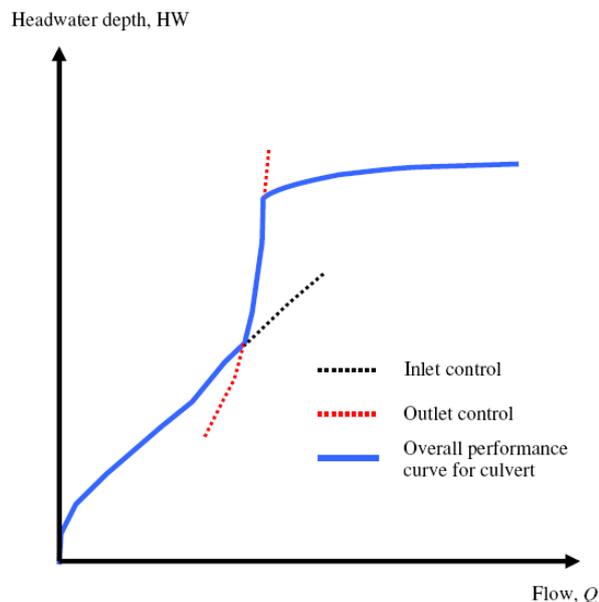


Figure 7.18 Culvert performance curves

One method of analysis which demonstrates the hydraulic performance/behaviour of culverts is to construct performance curves.

These are graphs of headwater depth against flow for the full range of flow conditions. They are calculated for inlet control and outlet control, and then combined to form an overall performance curve.

Laboratory and field measurements and hydraulic theory are used in combination to develop the hydraulic performance curves.

A simple and rapid ‘afflux estimation system’ (AES) has been developed (Defra and Environment Agency, 2007) for use with the commonest types of bridge and culvert structures in the UK. AES, which is designed to be used in conjunction with CES, supports the modelling of pipe, box and arch culverts. The parameters needed for the culvert are:

- barrel roughness, span, rise, arch springing level and length;
- inlet shape, material, type and edge type;
- structure crest elevation.

The culvert barrel roughness and inlet parameters are looked up in design tables. The culvert rating is then computed for both upstream and downstream culvert sections.

Other issues to consider in culvert design include:

- the potential for culvert blockage (see [Figure 7.19](#));
- the presence of trash and security screens (see [Figure 7.20](#)).



Figure 7.19 Culvert blockage

Culverts are often prone to blockage. In this illustration the blockage is at the inlet, but it can also be within the culvert which is much more difficult to remove.

Blockage reduces the area available for flow and results in a higher water level upstream, with consequent increase in flood risk or overtopping of the road (or other infrastructure that the culvert passes under).

It is vital to assess the risks and consequences of blockage when designing any culvert.



Figure 7.20 Trash and security screens

Screens are often installed on culvert entrances to stop debris entering the culvert or to prevent access by children.

The presence of the screen itself has a hydraulic effect. This may be small when the screen is clean but can be very significant when it is obscured by trash.

The case of a partially or fully blocked screen must be considered in the hydraulic design.

The Environment Agency's *Trash and security screens: a guide for flood risk management* (2009) gives detailed guidance on the design of screens.

Bridges

Bridges have many similarities to culverts with respect to their hydraulics, although the characteristics of their design are often different, as a bridge tends to have a larger opening than a culvert. The analysis of flow through a bridge is complicated, the detail of which is beyond the scope of this guide.

For simplicity, a structure is often defined as a bridge rather than a culvert if the ratio of its streamwise length to the height of its opening is less than about five, or the opening width (or span) is greater than about 2m. [Figure 7.21](#) shows an example of a three-arched bridge.

Figure 7.21 Arched bridge over the River Lee in Cork (Ireland)



For bridges (and culverts), 'afflux' is an important hydraulic impact. Afflux is defined as the maximum rise in water surface elevation above that which would exist if the bridge was not there. It is most important in flood conditions.

Afflux is illustrated in [Figure 7.22](#) for a bridge structure located in a uniform channel. The dashed line represents the normal water surface for the unstructured channel. The solid line represents the water surface when the structure is present. Afflux is shown as the maximum rise of water level above the normal depth (Y_1) of the undisturbed channel. Note that the afflux differs from the headloss across a structure, as the latter is a variable depending on the upstream and downstream locations of measurement.

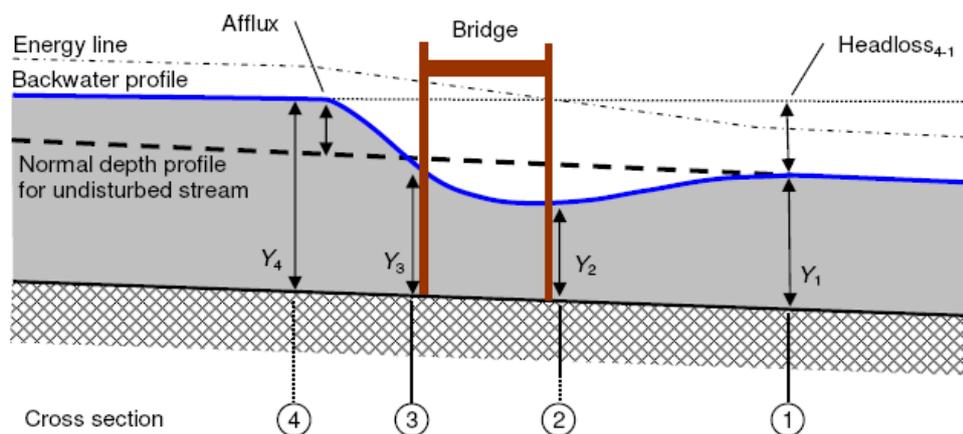


Figure 7.22
Bridge afflux

When a structure such as a bridge is placed in a stream, there is a local loss of stream energy. This is due to the fluid friction in contact with the structure and the stagnation zones that border the contracting (cross sections 4 to 3 in [Figure 7.22](#)) and expanding (cross sections 2 to 1) flow reaches upstream and downstream of the structure. To maintain a steady flow, this local loss of energy is compensated by an increase in stream potential energy immediately upstream of the structure. This creates a backwater that begins at the afflux location.

Important structural variables that affect afflux include:

- opening ratio – the ratio of the structure's open area to the flow area at a particular water level;
- skew – the angle normal to the structure's axis with the incident flow direction;
- eccentricity – the offset of the structure's centre line from the flow centre line;
- surface roughness – determines the frictional energy loss by the flow;
- bridge piers (the number and streamlining are important).

Important flow variables are:

- Froude number – ratio of the inertia force of the water to the weight of the water (when less than one, the flow is subcritical and surface waves can propagate both upstream and downstream; when greater than one the flow is supercritical and surface waves cannot propagate upstream);
- choking – occurs when the flow depth at the structure is at a condition of minimum energy and thus any discharge increase must incur an increased afflux;
- sediment transport – may lead to a larger opening being scoured at the structure, reducing the afflux;
- debris transport – may lead to blockage of the structure and increased afflux.

The common classifications for bridge structures for which afflux equations have been developed include:

- pier bridges;
- embankment bridges;
- arched bridges.

Manual methods are available for calculating the bridge afflux which use simplified equations (Bradley, 1978), though the afflux equations are more readily used within computer code which model both the stream and structure hydrodynamics. Various water surface profiling methods are used in the different one-dimensional hydraulic analysis codes recommended by the Environment Agency.

AES (Defra and Environment Agency, 2007) supports the modelling of six bridge types:

- arch bridge – elliptical, parabolic, semi-circular or user-defined;
- beam bridge – continuous or single pier.

Gates

The hydraulic analysis of flow over, through or under gates is similar to that of weirs and culverts in combination. Although the detail of this is beyond the scope of this guide, it is important to understand the principles of the different types of gates (Table 7.4) and their hydraulic performance.

Table 7.4 Main types of gates used in the UK

Type	Description
Flapgate	A fully automatic type controlled by the pressure head across it. It is a gate hinged at the top. When pressure is from the on-seating side, the gate is kept closed; a pressure from the off-seating side opens the gate when a threshold pressure is surpassed.
Vertical rising (or sluice) gate	A plate sliding in the vertical direction, controlled by machinery.
Radial gate	A structure where a small part of a cylindrical surface serves as the gate, supported by radial constructions going through the cylinder's centre of radius and controlled by machinery.
Rising sector gate (see Figure 7.23)	A part of a cylindrical surface that rests at the bottom of the channel and rises by rotating around its centre, and controlled by machinery.



Figure 7.23 Rising sector gate

Perhaps the most iconic gate structure in the UK is the Thames barrier, which is a rising sector gate.

Radial and rising sector gates have the advantage that the friction forces generated when the gate is moved are low.

Radial gates are often used if the hydrostatic forces are so large that the use of an undershot sliding gate would be impracticable because of the large amount of friction on the sliding surfaces.

The discharge under and over various types of gates can be characterised by several flow modes. A combination of hydraulic theory and experimentally determined coefficients applicable for specific situations provide methods to determine the flow through gate structures.

Other types of structure

Many other types of structures are found in watercourses such as pumps, orifices and siphons (see Chapter 11). The hydraulic design of these types of structure is covered in many textbooks.

7.5 Practical application of hydraulic modelling

7.5.1 Developing a scope of work

Before beginning any modelling study, a clear scope of work should be developed by a competent modeller with experience in hydraulic modelling. This project brief should define clearly the model requirements.

Model usage

The scope of work should describe the intended use of the model – whether it be strategy, flood alleviation scheme, outline design, or to support detailed design. It should also state whether there are future aspirations for further development of the model, or if there are any synergies or links with other projects. This helps to ensure that any future model uses are considered and, where possible, taken into account so as to maximise current and future value from the modelling.

The scope of work should outline the expected accuracy tolerances in the model; for example, whether it is important that the model represents accurately water levels, volumes or flows, or any combination of these.

Catchment understanding

The scope of work should identify the location of the modelling study, with a clear location plan showing the study area and with the upstream and downstream extents of the model labelled. All this information should be repeated in the text.

Ideally the extents of the model should be located at points of hydraulic control such as a tidal boundary or a weir. If this is not possible, the boundary should be located where the flow is likely to be contained within the main channel for all flow events to be considered.

To ensure all key areas are modelled correctly, the scope of work should set out existing knowledge of the catchment. For example, it is important to understand the locations of important flowpaths and complex hydraulics: if there are specific key flood risk areas that need to be considered, these should be clearly highlighted. The benefits of a walkover survey should not be overlooked.

Any known interactions between the fluvial, pluvial, groundwater and tidal sources of flooding should also be highlighted in the scope of work.

Resources

The anticipated cost of the modelling study should be given in the scope of work, along with a timescale in which the project is to be completed. In many cases it will be appropriate to state ‘milestones’ by which certain components of the model should be complete.

7.5.2 Data

For most modelling studies, some data will already exist for the catchment. These will need to be collected, while other data will need to be generated.

Sources

The main data source for most modelling studies is the Environment Agency. Information about recent studies is held on the National Flood and Coastal Defence Database (NFCDD). Records of older studies are likely to be held in more local databases. This information is supplied by the Environment Agency free of charge for its own projects and at a small cost for other types of project. Requests for

information should be made through the National Customer Contact Centre (see <http://www.environment-agency.gov.uk/contactus/default.aspx>).

Existing datasets

Some or all of the following data may need collecting. **Table 7.5** lists the commonly used datasets and why they are important.

One of the most important datasets is historic flood levels. This information is vital for the calibration of a model when the model is tested against a number of recorded flood events. In most circumstances, having more data is better – especially if they are recent and if they relate to more than one flood event. All data should have been checked and confirmed as fit-for-purpose before use.

When obtaining historic flood levels and other non-standard data, an understanding of how the river was operating is required so that these data can be used appropriately in the study. For example with flood levels, if a bridge or culvert was blocked during an event, the flood level could be artificially raised and not match modelled results.

The data collection process can take some time and needs to be incorporated into the programme. Allow a couple of weeks to obtain standard information that has already been collected. New datasets (see below) can take a significant amount of time to acquire and this need should be identified as early as possible.

Table 7.5 Commonly used datasets and their importance

Data	Why important
Mapping data: <ul style="list-style-type: none"> ▪ OS MasterMap (vector) ▪ OS LandLine (vector) ▪ OS 1:10k (raster) ▪ OS 1:25k (raster) ▪ OS 1:50k (raster) 	<ul style="list-style-type: none"> ▪ Mapping data are needed so that an understanding of the catchment can be gained. ▪ Model results eventually need to be presented on top of mapping. ▪ OS MasterMap can be used in 2D models to assign no-flow areas around properties and also in the application of Manning's 'n' to different areas.
Aerial photography	<ul style="list-style-type: none"> ▪ Aerial photography can be used to gain an understanding of the catchment. ▪ Model results can be presented on top of aerial photographs. ▪ Aerial photography is particularly useful in public consultation exercises.
Digital elevation data (bare earth and surface) <ul style="list-style-type: none"> ▪ SAR* (synthetic aperture radar) ▪ LiDAR (light detection and ranging) 	SAR or LiDAR is important in model construction and in the flood mapping process. <ul style="list-style-type: none"> ▪ 1D models – used for creating out-of-bank flow information – extended sections, secondary channels, reservoir unit storage-area curves. ▪ 2D models – used for the 2D domain which routes flows. ▪ Flood mapping – water surface is generally intersected with LiDAR to generate a flood extent.
Survey data of the channel and structures	<ul style="list-style-type: none"> ▪ Needed for representation of the channel and structures in the model.
Existing models and reports	<ul style="list-style-type: none"> ▪ Existing models can be potentially used or modified to provide all or part of a new hydraulic model. ▪ Existing model reports are useful to gain an understanding of how a catchment responds and operates.
Historic data (including water levels, photographs, reports and flood outlines)	<ul style="list-style-type: none"> ▪ Needed for calibrating the model.

Defence data (including operational information)	<ul style="list-style-type: none"> ▪ Important if it is necessary to consider the standard of protection afforded by existing defences in the model.
--	---

* SAR data should not be used for detailed models due to concerns over the accuracy of this type of data.

New datasets

New datasets requiring collection are likely to be either from LiDAR or a ground survey for river channels and assets. When specifying new datasets for use in hydraulic models, reference to the latest survey specification from the Environment Agency is recommended. As of January 2009, this is Part B: Delivery (Sections 2.6–2.9) of *Strategic flood risk management specification for flood risk mapping* (Environment Agency, 2006). This document is available on request from the Environment Agency.

When specifying a survey of river channels and structures, undertake a site visit with the surveyor. During the site visit, mark up detailed plans of the areas requiring survey and pay attention to potential health and safety concerns and access requirements for data collection. Use this information to prepare a brief.

LiDAR should ideally be collected during the winter months when vegetation cover is lower, but it also requires suitable weather conditions (that is, little cloud cover). Where possible, include some allowance for unfavourable conditions in the programme.

If new ground survey is required, consideration needs to be given to the time of year. During summer months, channels may be overgrown causing access problems and tree coverage may affect global positioning system (GPS) signals. During winter months, shorter daylight hours reduce the amount of time available on site and water levels will probably be higher in the river, potentially making the survey more difficult.

Management of data

Most modelling projects utilise significant amounts of data and it is important that these data are managed properly. It is recommended that a data register is used on all projects, recording when information is received and where it is located. On receiving the dataset, it should be checked to ensure it is complete and of sufficient quality for the project. Failure to check the quality of the data on receipt could cause significant delays at a later stage should problems with the data emerge. The data should then be stored in a central location on a file server, preferably using a file structure format common across projects.

It is also important to assess and record the quality of collected data. This information is termed ‘metadata’ (information about data) and is a vital part of the quality assurance (QA) process. Users of a model output should be able to gain access to metadata so that they can determine whether the model and its data are appropriate to their needs.

Copyright of data

Nearly all data are subject to copyright law and, in most instances, a licence agreement accompanies the data when received. This agreement should be read and its contents noted and adhered to. It should be recorded in the data register.

In general, data provided by the Environment Agency are made available on a project-by-project basis. Often it is a condition of their use that they can only be used on that particular project.

The licence details should be included on all maps obtained from the Ordnance Survey.

7.5.3 Model build

Conceptual review

A conceptual review should be carried out before modelling starts. This involves thinking through all aspects of what the modelling is trying to achieve to ensure the best possible method is chosen. If done well, it allows the correct decisions to be made from the outset.

Choice of model

The choice of model used in a modelling study depends on a number of factors typically including:

- the level of complexity of the out-of-bank flood flowpaths;
- if storage and flow control are relevant;
- the required level of model accuracy;
- the data available;
- the time and resources available.

Table 7.6 lists some of the most common modelling techniques and software used. In general terms, a 1D model is best for modelling in-channel flows and structures and a 2D model is best for considering out-of-bank flows. A dynamically linked 1D–2D model allows the transfer of flow between the channel and out-of-bank flow, and is currently the most accurate form of modelling in mainstream application. The development of linked 1D–2D models is a new technique and is subject to ongoing software development and R&D.

Table 7.6 Common modelling techniques and software

Model type	Details	
Unsteady 1D hydrological routing	Pros	<ul style="list-style-type: none"> ▪ Reasonable time-dependent flow prediction for natural channels of reasonable slope.
	Cons	<ul style="list-style-type: none"> ▪ No water level prediction unless stage/discharge relationship is known at required location. Often normal depth approximation (which balances surface slope with friction slope) is used. ▪ Cannot simulate flood storage or tidal effects or link accurately to floodplain modelling.
	Data	Coarse channel sections.
	Software	ISIS, Hec-RAS, MIKE 11
Unsteady 1D hydrodynamic	Pros	<ul style="list-style-type: none"> ▪ Accurate flow and level calculation even when normal depth does not occur. ▪ Can include standard empirical equations for weirs, sluices, bridges and other structures. ▪ Can simulate tidal effects.
	Cons	<ul style="list-style-type: none"> ▪ Cannot simulate floodplain flow unless flow routes are defined beforehand.
	Data	Channel sections, and structures.
	Software	ISIS, Hec-RAS, InfoWorks RS, MIKE 11

Quasi 2D floodplain only (also called 2D hydrological routing)	Pros	<ul style="list-style-type: none"> ▪ Will produce results for most situations even if accuracy is variable. ▪ Reasonable steady or quasi-steady broad scale (>50m) flow prediction for natural floodplains of reasonable slope and 'basin filling' situations. ▪ Can be statically linked to outflow from channel.
	Cons	<ul style="list-style-type: none"> ▪ Floodwave speed not accurately represented. ▪ Very long run times at fine scale relative to fully 2D model types. ▪ Not as accurate as fully 2D, as not all terms in the equations are considered. ▪ Usually not linked to channel model which dictates amount of flow on floodplain. ▪ Cannot represent structures or defences.
	Data	DEM
	Software	JFLOW, LISFLOOD
1D hydrodynamic linked to quasi 2D floodplain	Pros	<ul style="list-style-type: none"> ▪ Reasonable steady or quasi-steady broad scale (>100m) flow prediction for natural floodplains of reasonable slope and 'basin filling' situations. ▪ Can be dynamically linked with channel model types and can represent structures and defences. ▪ Generally much faster than quasi 2D floodplain only model type.
	Cons	<ul style="list-style-type: none"> ▪ Generally need to define floodplain flow routes before simulation is attempted. ▪ Not as accurate as fully 2D as not all terms in the equations are considered.
	Data	Channel sections and DEM
	Software	ISIS, Hec-RAS, InfoWorks RS, MIKE 11, JFLOW, LISFLOOD
1D hydrodynamic linked to fully 2D floodplain	Pros	<ul style="list-style-type: none"> ▪ Reasonable unsteady broad or fine scale flow prediction for natural floodplains of arbitrary slope, including 'basin filling' situations. ▪ Can be dynamically linked with channel model types and can represent structures and defences. ▪ Generally much faster than quasi 2D floodplain only model type. Can simulate tidal effects in both channel and floodplain.
	Cons	<ul style="list-style-type: none"> ▪ Can lead to long run times at fine scale.
	Data	Channel sections and DEM
	Software	ISIS-TUFLOW, ESTRY-TUFLOW, ISIS1D2D, MIKE Flood

In some complex modelling scenarios, 3D modelling may be required. This may be used where:

- an understanding of flow over, through and around structures is needed;
- physical phenomena such as superelevation and standing or reflective waves, which cannot be represented by a 1D or 2D approach, are likely to occur.

The computational effort in undertaking this type of method can be very large and it is generally only undertaken when poor design has significant consequences or leads to an unsatisfactory level of risk.

The use of a physical model should also be considered for this type of application, particularly if the problem relates to a relatively small area (for example, at a hydraulic structure such as is illustrated in [Figure 7.6](#)).

It is important to use the most appropriate modelling tool for the project rather than merely the tool that is available. Inappropriate tool selection and use can have significant technical and cost implications for both current and future needs.

Model schematisation

Model schematisation is the representation of the river system in the hydraulic model. Depending on the choice of model and the accuracy required, there will be differing requirements for the model schematisation.

Typically the model schematisation is focused on how the in-bank flow links to the out-of-bank flow and storage. It is important to consider all the flood flow routes, together with how flood mechanisms interact with each other.

Model schematisation should be undertaken by an experienced modeller before modelling begins and reviewed regularly as model build progresses. The schematisation should also be discussed with hydraulic engineers who have knowledge of the site in question; they will be able to advise whether the assumptions made are realistic.

General model build considerations

A number of general model build considerations need to be taken into account in both 1D and 2D models. These are described below.

Channel sections

To help to ensure model stability, place channel sections at regular intervals throughout the model. The spacing should aim to satisfy the requirements that channel sections should:

- be no more than $20B$ apart, where B is the top width of the channel;
- generally not be more than $1/(2S)$ apart, where S is the mean slope (m vertical to m horizontal) of the river;
- generally not be more than $0.2D/S$ apart, where D is the typical depth of flow and S is the mean slope.

Where these requirements cannot otherwise be met, interpolated channel sections can be used in the model.

Structures

When modelling structures, a range of methods is available to represent the physical structure. Decide on an appropriate methodology for each structure. If necessary, use the model help files or seek advice.

It is important to consider what happens at the structure for a range of flow conditions. The structure can operate very differently under certain conditions and new flow routes can be created if the structure is bypassed.

The location of surveyed cross sections is important when modelling structures. These cross sections should be located at the upstream and downstream face of the structure. Extra cross sections should also be surveyed at a sufficient distance away from the structure where it does not affect flows. These distances are often referred to as the contraction and expansion lengths.

Roughness

All models require a roughness to be entered into the model. The most common representation of this is in the form of Manning's n coefficient, although alternatives exist in the form of the CES and the Colebrook–White equation. Examples of Manning's n are given in [Table 7.2](#); they are also found in popular hydraulic textbooks.

The roughness of the channel varies with season, flood level and maintenance regime, and hence can have a range of justifiable values. Sensitivity should be undertaken on this parameter to determine the effect it has on the flood level. It may be appropriate to have models configured for different seasons, especially when trying to undertake model calibration (see Section 7.5.5).

1D model build considerations

Out-of-bank flow

A number of methods can be used to simulate out-of-bank flows. The choice of method depends on the relative importance of the hydrodynamic effects and the resolution of accuracy required. Typical methods include:

- extended channel sections;
- storage areas;
- secondary channel sections.

Table 7.7 summarises the relative merits of these three methods while Box 7.11 describes their main features.

If out-of-bank flow representation is needed in detail (for flood hazard assessments, for example), a 2D approach is likely to be more appropriate.

Table 7.7 Relative merits of different out-of-bank flow methods

Attribute	Floodplain modelled using:		
	Extended channel sections	Secondary channel	Storage areas
Ease of modelling	PPP	P	PP
Suitable for washlands with embankments	O	P	PP
Differing water levels in main river and floodplains	O	P	P
Floodplain velocities calculated	P	PP	O

Box 7.11 Methods for simulating out-of-bank flows

Bank tops modelled using SPILL unit

Left floodplain modelled using separate RIVER unit

Right floodplain modelled using separate RIVER unit

Extended channel sections configuration

Time-dependent storage effects influence the flood extent and depth of flow on the floodplain. When using extended channel sections, a single water level is determined across the whole section. This can lead to an over-representation of the flood extent and floodplain flood depth.

Secondary channel sections configuration

By modelling the floodplain as separate channels connected to the main channel, the time-dependent storage effects can be considered. This approach, which provides an improved representation of velocities and flows on the floodplain (compared with extended channel

sections), requires the modeller to predefine the floodplain channels. This is a subjective process; it requires additional model set-up time and leads to increased model run times.

Storage area configuration

If the floodplain time-dependent storage effects and water levels need to be considered but the velocity is not required, an alternative to the secondary channel method is the use of storage units (or cells). This is still a subjective process, but reduces the model set-up and run times compared with the use of secondary channel sections.

2D model build considerations

Cell size

The 2D grid cell size is dictated by the accuracy of the modelling required. Typically for a broad-scale model, a cell size of 625m² (25m × 25m) can be used but, for more detailed studies such as in urban areas, a cell size of 100m² (10m × 10m) or less should be used in order to pick up features such as buildings, walls and roads.

Model stability: time and distance steps

The relationship between the time and distance steps can have an effect on the model stability and accuracy of the solution. A widely used parameter in numerical modelling to assess suitable timesteps and distance steps is the *Courant number* (see Box 7.12). The Courant number depends on the numerical scheme used within the software.

As a rule, the time step in seconds should typically be of the same numerical order as the half the cell size in metres. For steep models where high velocities and supercritical flow are experienced, a smaller timestep may be required.

Box 7.12 Courant stability for ISIS and TUFLOW

Courant number for 1D ISIS model is defined as:	$C_r = \frac{\Delta t \sqrt{gH}}{\Delta x}$	Target $C_r < 10$
Courant number for 2D TUFLOW model is defined as:	$C_r = \frac{\Delta t \sqrt{2gH}}{\Delta x}$	Target $C_r < 5$
where: Δt , Δx , g and H are respectively timestep(s), length of model element (m) (or distance between cross sections), acceleration due to gravity (m/s ²) and depth of water (m)		

Model run time

Model run time (the time it takes to run a model) depends heavily on the cell size chosen for the model, the timestep of the model and the duration of the model run. Halving the 2D cell size typically corresponds to increasing the simulation time by a factor of eight (four times as many cells and half the timestep).

Model boundary conditions

Linking hydrology

There are a number of approaches to link the hydrological inflows to the hydraulic model. The most commonly used approaches are:

- point inflows at the upstream hydraulic model extents;
- point inflows at strategic locations throughout the catchment (for example, tributaries, natural watercourses);
- lateral inflows – to enable inflows to be distributed over a length of the model;
- distributed areal rainfall, to model surface water runoff (2D only).

The approach or approaches to be used should be discussed by the hydrologist and the hydraulic modeller to determine the most applicable method. This will largely be based on the hydrological and hydraulic modelling methodology.

Downstream boundary

In models are used to estimate flood depths, it is important that the downstream boundary is far enough away from the area of interest so as not to affect the results. As a general rule this distance should be greater than the backwater length L , which can be estimated as $0.7 \times \text{depth}/\text{gradient}$ (using dimensionally consistent units) (see [Section 7.4.4](#)).

Critical storm duration

An appreciation of the critical storm duration is required as this can affect the water levels in the model. For most fluvial design or other modelling purposes, the highest water levels or largest flows are required, giving a worst case situation.

To determine the critical storm duration, multiple runs of the hydrology through the model need to be undertaken with different storm durations.

In most circumstances, the critical storm duration can be undertaken on one set of annual exceedance probability (AEP) inflows (see [Section 2.4.1](#)) and then be applied directly to other AEP inflows.

If a complex hydraulic process starts to operate between two events, a further check on the critical storm duration should be carried out.

In some models, there may be the need to use different critical storm durations in different parts of the model. This can complicate model processing significantly and so checks should be made on the water levels produced, as different storm durations might make only very small changes to the water levels. This inaccuracy can then be either accepted or included as part of the freeboard calculation.

Joint probability

Joint probability refers to the chance of two or more conditions occurring at the same time to produce a high water level (for example, a large river flow and high tidal level). High water levels are often caused by more than one environmental variable, so that the probability of a certain level occurring is related to the combined probability of occurrence of all the variables concerned. There is often a degree of dependence between the variables and an assessment of this dependence is required to evaluate the flood risk due to these extreme events.

Failure to consider joint probability in the design process can lead to significant under- or over-design.

The joint Defra/Environment Agency flood and coastal erosion risk management R&D programme has funded research into joint probability models for use in flood risk studies. *Joint probability – dependence mapping and best practice* (Defra and Environment Agency, 2006) provides the latest guidance.

Model construction order

If a model is constructed well, stability problems at the model run stage can be greatly reduced. As a general rule of thumb, the following construction order should be adopted:

- 1 Input channel sections for the main channel.
- 2 Add hydrology.
- 3 Gradually input structures into the main channel.
- 4 Gradually add out-of-bank flow areas as required, along with any necessary structures.
- 5 Add rules to any gates, pumps, orifices and the like in the model

The model should be run regularly during the model build process for both low and high flows. The high flow used should be the maximum flow which the model is required to simulate.

As the model is developed, the model data files should be backed up to:

- provide an audit trail;
- enable earlier versions of the model build data files to be returned to if necessary.

During the model construction phase, the model should be reviewed regularly to ensure:

- its performance represents the hydraulics of the system;
- its application remains ‘fit for purpose’.

Model simulations

Initial conditions

Before a model is simulated, it is usually important to set the initial conditions in the model. Initial conditions tell the model what the water levels should be at each point at the start of a model simulation. These can be important in some situations, specifically in storage areas, as a wrong initial condition may mean that incorrect volumes are calculated.

Most 1D models will not run if inappropriate initial conditions are used.

Steady and unsteady models

When running a model, it is vital to consider whether the model is run as a steady-state or unsteady model. Steady-state models are where a single flow rate is applied constantly at each inflow boundary and unsteady models are where the flow varies with time at each inflow boundary (see also [Table 7.8](#)).

The accuracy of steady-state models depends largely on the accuracy of the flow rate used whereas, for unsteady models, the accuracy also depends on the shape and hence the volume of the inflow hydrograph and its timing.

If electing to run an unsteady model, it should always be run in a steady-state first to establish the initial conditions. This also serves as a quick check that the model is constructed correctly.

Table 7.8 Application of steady-state and unsteady models

Run type	Application
Steady-state	Models where: <ul style="list-style-type: none"> ▪ the flood event has infinite or unknown volume; ▪ there is no reverse flow in the channel; ▪ floodplain flow is limited.
Unsteady	Models where: <ul style="list-style-type: none"> ▪ a tidal boundary is present; ▪ floodplain flow needs to be taken into account; ▪ there are pumped watercourses where the pumping rate varies depending on water levels; ▪ there are gates that open and close according to control rules.

Run parameters

Run parameters are used in most modelling packages to define the properties of the simulation (that is, the timestep, tolerances for the model or settings for the numerical solver). If these values are changed from the standard values, consider the potential impact of this on the model results. The default values are not always the most appropriate and should be reviewed as part of the model build and QA procedures.

Model run log

A model run log should be kept for all model runs (even failed runs) so that an audit trail can be maintained and simulations re-run later if necessary.

7.5.4 Verification and validation

Model verification should be undertaken to ensure the model is built correctly and competently. This procedure should be captured as part of the QA procedures and should start before model building begins. It should part of model build good practice (see [Section 7.5.6](#)) and involve model checks (see below).

Model checks

Model checks should be incorporated into the whole modelling process so that any problems are identified as early as possible. The major milestones when a model should be checked are:

- schematisation for survey;
- final schematisation and ‘raw’ data;
- data file before calibration;
- outputs from calibration;
- at each design modification.

It is essential to use appropriate QA procedures for modelling work to ensure that model checks are carried out thoroughly and at appropriate intervals. The outputs of the modelling checks should be reported in a standard way and included as part of the modelling report.

As the model is developed, validation should be undertaken to ensure the model is developed in accordance with the project brief. Validation should be carried out through calibration and sensitivity testing (see below).

7.5.5 Calibration and sensitivity testing

Understanding how a model performs is fundamental to the model build process. Calibration and sensitivity testing should be undertaken on every model (calibration data permitting) and the process should start early in the model process. A model that does not perform as intended or required is not ‘fit for purpose’.

Calibration

Calibration of the model is important to ensure that the model schematisation accurately represents the system being modelled. Calibration data are not always available and, in such circumstances, greater emphasis should be put on understanding the model sensitivity and model uncertainties.

The calibration process should include:

- both the hydrological and hydraulic processes;
- a number of different events.

When selecting calibration events, it is useful if they are for a range of flows (creating both in-bank and out-of-bank flow scenarios).

It is important to calibrate to an appropriate level of accuracy. It should be agreed at the start of the project (within the scope of works) the indicative water level accuracy that the model should achieve. Typical values are given in [Table 7.9](#).

In addition to the peak water level, the rising and falling limbs of the hydrograph should be similar to those observed and their timing should be as accurate as possible.

Table 7.9 Typical indicative model water level accuracy

Model type	Model accuracy (water level accuracy)	
	Acceptable	No worse than
Broad scale	±500mm	±750mm
Flood defence	±250mm	±350mm

It is also useful, as a further check, to ensure that the model results reproduce the observed flood envelope (that is, the extent of flooding on a plan). A check on the number of properties that are flooded can also be beneficial. It should be noted that checking the aerial extent of the flood envelope is of limited value where the edge of the floodplain is defined by a sudden rise in ground level. This is because the aerial extent will be similar for a wide range of flood levels.

If calibration is unsuccessful, a number of potential sources of error can be considered. These include:

- hydrometric data errors in stage datum, timing and rating curves;
- hydrological analysis;
- channel survey;
- changes in channel geometry;
- data handling errors;
- inappropriate schematisation (a common cause of error) and approximation errors;
- event specific interventions (for example, structure operations and structure blockages);
- software limitations.

It is fundamentally important to identify the likely source(s) of error before attempting to resolve the issue. Random adjustment of model parameters to improve calibration is not acceptable.

Sensitivity

Sensitivity analysis should be undertaken on the model in order to develop an understanding of the relationship between key input factors. This appreciation is fundamental to ensuring the correct use of the model.

Good modelling practice requires the modeller to provide an evaluation of the confidence in the model which assesses the uncertainties associated with the modelling process and with the outcome of the model itself. Sensitivity analysis is an essential part of the quality assurance and design process.

Sensitivity analysis reveals which parameters the model most depends upon for its accuracy. For example, if the sensitivity tests show that water level prediction varies significantly for relatively small changes in Manning's n , then the model developer, the designer and the user of the results need to be made aware that accurate estimation of channel roughness is crucial and that seasonal changes may have significant impacts. This information also helps the model developer and designer to be aware of the risks associated with model uncertainties and helps inform decisions on the freeboard to be allowed.

Sensitivity analysis should be undertaken on the following areas of the model:

- Manning's n ;
- coefficients in structures – bridges, weirs, spills and orifices;
- ponds/reservoirs – initial water levels;
- boundaries – flows and levels.

Other factors that could be considered include:

- structure blockage;
- wind effects;
- model run parameters.

As part of the sensitivity analysis, undertake mass balance checks to ensure the model is not gaining or losing inappropriate amounts of volume. Hydraulic models tend to gain or lose volume due to numerical rounding errors, model stability and model convergence. This gain or loss of volume is referred to as the 'mass error'.

As a rule of thumb, mass errors should be less than 2%. If the mass error is greater than 2%, the cause and location of the mass error within the model schematisation should be identified and the consequence of this error assessed and improvements to the model considered. If the mass error is greater than 5%, then it suggests that the model schematisation or type of model (software) is not robust and needs to be challenged.

Knowing the amount of mass error and relating this to the uncertainties in the hydrological analysis enables the designer to understand the uncertainties in the model that need to be accounted for in the design.

7.5.6 Model build good practice

Model management

When undertaking any modelling project, detailed records need to be kept. Having good documentation is essential. This is important for QA purposes but also so that someone not involved in the original modelling can pick up the model and understand:

- how the model was created;
- why certain decisions in the model build process were taken;
- which versions of the model should be used to continue development or extract results from.

Common ways of keeping such records include.

- create a calculations file where general assumptions and comments are kept;
- add comments to the model itself (a good sign of a competent modeller);
- use a model run log to record all model changes and runs (even failed attempts);
- label model files with a logical and common naming strategy;
- label objects in the model with a common naming strategy;
- use a common model file structure on the server;
- consider use of version control system to store multiple copies of evolving datafiles.

Shortcomings and limitations of hydraulic models

When applied to analysis and design, the shortcomings and limitations of a hydraulic model need to be understood and documented. This will help future users of the model to know if anything needs to be developed further and if the model is applicable for their intended use.

Model handover

As part of the handover of the model, a model handover document should be created. The Environment Agency's specification for flood risk mapping (see [Section 7.5.2](#)) provides guidance on this document, which should contain:

- a technical overview of the model and its background;
- notes on how the model operates under different flow conditions;
- an explicit statement of any concerns about the accuracy of the model or its ability to represent reality.

Client management of models

After the model is handed over to the client, the model should be archived so that it can be used in the future. The model should be reviewed before reuse to ensure that:

- modelling assumptions are still correct;
- the hydrology and hydraulics of the model are still current.

7.5.7 Risks and mitigation

A number of common risks should be mitigated against when undertaking the construction of a hydraulic model (see [Table 7.10](#)).

Table 7.10 Common modelling risks

Risk	Mitigation
Data	
Survey appropriateness	<ul style="list-style-type: none"> ▪ Make sure that the modeller has visited the site. ▪ Make an assessment of the cross section spacing needed. ▪ Mark up plan of where sections are required. ▪ Prepare a survey brief detailing the requirements.
Survey errors	<ul style="list-style-type: none"> ▪ Use experienced surveyors. ▪ Check survey drawings and electronic data on their receipt.
Use of out-of-date data	<ul style="list-style-type: none"> ▪ Check that the latest version is used.
Boundaries	
Hydrological boundary incorrectly used in model	<ul style="list-style-type: none"> ▪ Check with the hydrologist how to apply the hydrological boundary to the model.
Model build	
Poor model design	<ul style="list-style-type: none"> ▪ Gradually build complexity into the model and check it at each stage. ▪ Check that floodplains are accurately represented. ▪ Make sure that river reach lengths are correct.
Data entry errors	<ul style="list-style-type: none"> ▪ Plot all cross sections so that they can be subjected to a reality check. ▪ Incorporate data checking routines that, for example, highlight any apparent inconsistencies.
Floodplain	<ul style="list-style-type: none"> ▪ Make sure that the schematisation of the floodplain is appropriate. ▪ Undertake a mass balance check to understand the performance of the model better.
Structures	<ul style="list-style-type: none"> ▪ Check that the most appropriate modelling unit is being used to model the physical structure. ▪ Ensure that out-of-bank flow routes are modelled around the structures.
Storage areas	<ul style="list-style-type: none"> ▪ Ensure that the storage area has the correct area/stage/volume relationship. ▪ Set initial water levels appropriately.
Calibration	
Lack of communication between hydrologist and modeller	<ul style="list-style-type: none"> ▪ Allow project time for the hydrologist and modeller to review and reprocess results.
Calibration data	<ul style="list-style-type: none"> ▪ Ensure that there is an understanding of the validity of the calibration data and under what circumstances the data were gathered.
Other	
Model audit records	<ul style="list-style-type: none"> ▪ Ensure that the model development process is well documented. ▪ Use model run logs.

7.6 Modelling and the design process

Modelling fits into all stages of the design process but in a number of different ways depending on the stage of the design process, and hence the level of accuracy and understanding required.

Generally, as more detail is added to the design more modelling is required to understand the fluvial processes in more detail. For example, a study might start by looking at water levels in order to determine the height of a flood embankment, but additional analysis might be needed later to look at velocities to test for scour. A number of other pressures such as budget and programme will also influence modelling decisions. Cost versus quality is considered in [Figure 7.24](#).

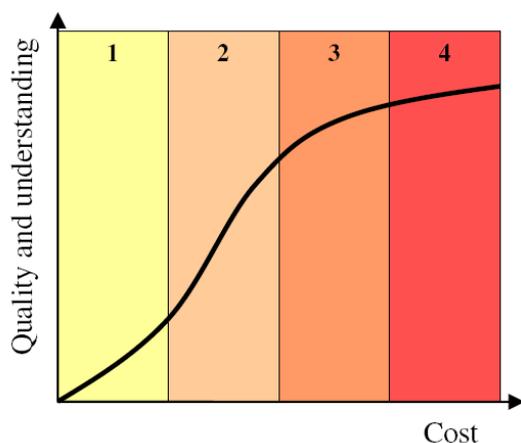


Figure 7.24 Cost versus quality

Cost versus quality for modelling doesn't quite fit the traditional curve found in many textbooks, assuming that you are starting from a point where you understand little about the river system.

Initially (stage 1) there can be a big gain from creating a simple model to understand the river, but it will remain relatively poor quality.

For a moderate cost, the undertaking of thorough modelling leads to a huge increase in quality (stage 2). Often in practice we combine stages 1 and 2 in order to make initial design decisions.

Once a thorough model is created, you enter an understanding phase (stages 3 and 4) where you get diminishing returns for the cost, but an increased level of detail and understanding is gained. In this phase you need to be pragmatic about what designs are being tested in order to maximise value for money.

Opportunities for added value and the longer term needs and potential uses of the model should always be sought at the outset. This can often lead to increased quality with either minimal initial cost or significant cost savings at a later stage.

Appropriate detail –starting with the end in mind

When writing the modelling brief and specification or working out how modelling should proceed, it is vital to identify the end goal. This enables early decisions to be made that benefit the project; for example, capturing more data initially to produce a more accurate model (stages 1 and 2 in [Figure 7.24](#)). An alternative (even though the end goal is known) is to take the view that a staged approach developing a simpler modelling first (stage 1) is more appropriate because there may be milestones or decisions to be made in the project which change the end goal.

When starting with the end in mind, it is also necessary to consider the starting point. In some instances, a detailed model may already exist which can be built upon (that is, the starting point may be at stage 2 or 3) but, in other cases, it may be necessary to start from scratch. A thorough review of existing work is required before a project commences.

Level of uncertainty and understanding

Depending on the stage of the design process, the user of the model outputs will require different levels of accuracy and understanding. At each of these stages, it is important to communicate an understanding of the uncertainty in model outputs as, in some cases, this can alter design decisions.

Inherent in this is a need to understand the limits of the model and what it is and is not capable of doing.

Interaction in the design

As the design progresses, there is a need for the modeller and the designer to work together to test solutions to the problem (stages 3 and 4 in Figure 7.24). It is critical that conceptual designs are investigated early on in the process and not considered further if it appears that they will not provide a solution. In undertaking this work, the modeller will often create many revisions of a model to test the design and it is important for the modeller to remember to keep a detailed log of all changes.

7.7 Summary of key tools

A key tool in hydraulic analysis is the application of the simple hand calculations highlighted in this chapter and referred to in textbooks. There are also numerous modelling tools in the marketplace that allow engineers to undertake hydraulic analysis. The choice and selection of the tool used depends on the model requirements and on the organisation carrying out the analysis.

The principal modelling tools used in the UK are listed in Table 7.11. Software is continuously being developed and enhanced, so the latest information on a particular package should be sought from the source given.

The merits and suitability of the software packages featured in Table 7.11 have been assessed in a number of benchmarking studies, reports and papers. These provide a useful source of information to understand the capabilities, the suitability and limitations of some of these software packages.

Table 7.11 Principal modelling tools used in the UK

Software	1D	2D	1D–2D	Source
HEC-RAS	P			http://www.hec.usace.army.mil/software
ISIS	P	P	P ¹	http://www.halcrow.com/isis
InfoWorks RS/2D	P	P	P	http://www.wallingfordsoftware.com
MIKE11/21	P		P	http://www.dhigroup.com/Software/WaterResources.aspx
JFLOW		P ²		http://www.jbaconsulting.co.uk
TUFLOW	P ³	P	P ^{1,4}	http://www.tufLOW.com

¹ Available through ISIS-TUFLOW link.

² Not fully hydrodynamic (does not solve momentum).

³ Available as ESTRY (provided with TUFLOW).

⁴ Links to ESTRY.

Key references

Chow, V T (1959). *Open-channel hydraulics*. McGraw-Hill.

A 'classic hydraulics textbook dealing with the design for flow in open channels and related channel structures. Covers both theory and practice; attempts to bridge the gap between the two.

Defra and Environment Agency (2004b). *Reducing uncertainty in river flood, conveyance, phase 2. Conveyance manual*, Project Record W5A-057/PR/1, Joint Defra/Environment Agency flood and coastal erosion risk management R&D programme project code SC000042. Environment Agency. Available from: http://www.river-conveyance.net/1_CES_UserGuide.pdf.

A software tool that enables the user to estimate the conveyance or carrying capacity of a channel. Includes the 'Roughness advisor', the 'Conveyance generator', and the 'Uncertainty estimator'.

Defra and Environment Agency (2007). *Hydraulic performance of river bridges and other structures at high flows – phase 2 (afflux estimation system)*, Science Summary SC030218/SS, Joint Defra/Environment Agency flood and coastal erosion risk management R&D programme project code SC030218.

Latest research methods applicable to UK channels for calculating afflux. Includes the 'Afflux advisor', which is ideal for occasional use by non-specialists, and the 'Afflux estimation system' (AES), which is a more rigorous tool for calculating afflux. Software and accompanying documentation are available from www.river-conveyance.net.

Ramsbottom, D, Day, R and Rickard, C (1997). *Culvert design manual*, Report R168. CIRIA. [Due to be superseded by the *Culvert design and operation guide* in 2009.]

This comprehensive guide covers the hydraulic design of new culverts and the hydraulic analysis of existing culverts. It also has comprehensive coverage of the practical aspects of designing and constructing culverts. It is due to be replaced in 2009 by the *Culvert design and operation guide*, an extended and updated guide which will cover the management of existing culverts, including repairs and rehabilitation.

Other references

Bradley, J (1978). *Hydraulics of bridge waterways*, 2nd edition. US Department of Transportation, Federal Highway Administration.

Defra and Environment Agency (2004a). *Benchmarking of hydraulic river modelling software packages*, R&D Technical Summary W5-105/TS, Joint Defra/Environment Agency flood and coastal erosion risk management R&D programme project code SC010058.

Defra and Environment Agency (2006). *Joint probability: dependence mapping and best practice*, Technical Summary: FD2308, Joint Defra/Environment Agency flood and coastal erosion risk management R&D programme project code FD2308.

Environment Agency (2006). *Strategic flood risk management framework, specification for flood risk mapping*, Release 1.2. Environment Agency.

Environment Agency (2009). *Trash and security screens: a guide for flood risk management*. Environment Agency.

Henderson, F M (1966). *Open channel flow*. Macmillan.

HR Wallingford and Barr, D I H (2006). *Tables for the hydraulic design of pipes, sewers and channels*, Volumes 1 and 2, 8th edition. Thomas Telford.

Samuels, P G (1989). Backwater lengths in rivers. *Proceedings of the Institution of Civil Engineers*, Part 2, Vol 87, pp 571–582. ICE.

US Army Corps of Engineers (2008). *HEC-RAS user's manual*. USACE.