Defra/Environment Agency
Flood and Coastal Defence R&D Programme

Afflux at bridges and culverts

Review of current knowledge and practice

Annex 2:
Hydraulic Model Implementation of Bridge and Culvert Afflux and Blockage

R&D Project Record W5A-061/PR2

Defra/Environment Agency
Flood and Coastal Defence R&D Programme

AFFLUX AT BRIDGES AND CULVERTS

Review of current knowledge and practice

Annex 2:
Hydraulic Model Implementation of Bridge and Culvert Afflux and Blockage

R&D Project Record W5A-061/PR2

Dr P G Samuels, HR Wallingford Ltd.

Research Contractor:
JBA Consulting – Engineers & Scientists

Publishing Organisation
Environment Agency, Rio House, Waterside Drive, Aztec West, Almondsbury,
Bristol, BS32 4UD.

Tel: 01454 624400 Fax: 01454 624409 Website: www.environment-agency.gov.uk

ISBN 1 8443 2291 2
Product Code SCHO0305BIYD-E-P

Dissemination Status

Internal:	Released Internally
External:	Released to Public Domain

Statement of Use
This Technical Report contains the results of the first phase of a study to improve the estimation of afflux at river structures in high flows. The information in this document will be used in developing improved software and guidance for flood defence and land drainage practitioners, and is made available for reference and use.

Keywords
Afflux, backwater, blockage, bridges, culverts, channel structures.

Research Contractor
This document was produced under R&D Project W5A-061 by P. G. Samuels, HR Wallingford Ltd.
The research contractor was JBA Consulting – Engineers & Scientists, South Barn,
Broughton Hall, Skipton, North Yorkshire, BD23 3AE.
Tel: 01756 799919 Fax: 01756 799449 Website: www.jbaconsulting.co.uk

Environment Agency's Project Manager
The Environment Agency's Project Manager for Project W5A-061 was:
Andrew Pepper, External Advisor, Engineering Theme.
Further copies of this report are available from the Environment Agency's science dissemination service

EXECUTIVE SUMMARY

This review paper forms part of the Environment Agency R&D scoping study on the Afflux and Blockage of Bridges and other Structures.

The paper considers the types and roles of computational models in the Environment Agency's flood defence activities both generally and specifically for the one-dimensional models in common use in the UK. Broadly these one-dimensional models have the same capability of modelling bridge afflux and the effects of culverts, with the MIKE11 model offering to greatest number of standard methods, but all models being capable of calibration of the afflux. The HEC-RAS software alone includes an option for modelling the partial blockage of a bridge by floating debris. None of the models appear to contain an option for including the head losses at any screens (whether obstructed or not) over the upstream or downstream face of a culvert.

Two and three dimensional models include explicitly the physical processes which cause afflux rather than by the use of empirical formulae and thus offer alternatives to physical modelling for complex sites where accurate afflux determination is an important issue.

The paper concludes with some implications for further research and development.

CONTENTS

1 INTRODUCTION

The purpose of the expert paper is to describe how the current generation of computational hydraulic river models include the so-called "head-losses" through bridges and culverts and account for the blockage of the structure by floating debris. The paper is intended to inform later stages of the scoping study on afflux and blockage of structures, particularly in identifying limitations of current approaches and thus potential research and development needs. The R&D needs may be either to make best use of current knowledge on this subject or to generate new knowledge and understanding to reduce the uncertainties in the current modelling undertaken by the Environment Agency and its consultants.

It is assumed that the reader of this expert review paper is familiar with the general process of river modelling as practised in the UK flood defence sector. This process has several key steps including:

- model specification
- data assembly and verification
- model building
- model calibration and verification
- sensitivity testing
- model application to generate the required outputs.

These steps are generic to the application of any type of hydrodynamic model whether steady or unsteady and in 1, 2 or 3 dimensions. The dimension in question is in space, with 1-D models representing variations in the primary flow variable in the downstream direction. For flood modelling 2-D models represent variations streamwise and laterally (i.e. in the two horizontal plan spatial dimensions). Three dimensional models include, to varying degrees of complexity, flow processes in all spatial dimensions. For steady flow models the variations of the flow in time are not considered in the computation, whereas for unsteady models some of the time dependence of the flow is resolved. In this context, the timescale of flow variation which is resolved is that of the rise and fall of a flood or tide, and possibly the operation of flood defence infrastructure such as gates or pumps.

Key choices are made at the model specification stage; these include:

- the extent of the computational domain
- the selection of model grid or nodal points (i.e. the definition of the spatial mesh)
- the representation of boundaries and physical processes in the model
- the choice of empirical relationships for representing features of the prototype which act as specific controls over-riding the normal free-surface flow computation.

In the application of a 1-D or 2-D model, the computational domain may in fact be chosen to start or end at a bridge or culvert, since at these sites the flow is well contained and sometimes is measured by current meter gauging. This practice then begs the question of how the boundary condition on water level or head-discharge rating should be estimated for higher flows than those measured.

The model node selection will depend upon the location of flow control structures with each model type and modelling package having individual requirements. Certainly the model nodes are likely to be refined in the neighbourhood of a structure.

The issues related to the representation of the boundaries and processes depend upon the dimensionality of the model being used. For 1-D models the river bed and flood plain will be represented by a series of cross-sections, in 2-D and 3-D models a regular, structured or unstructured grid will be employed depending upon the numerical methods in the model, as tabulated below.

Table 1.1 Types of numerical methods

Method	**2-D Model**	**3-D model**
Finite Difference	Regular	Regular
Curvilinear Finite Difference	Structured	Structured
Finite Element	Potentially Unstructured, but can be structured	Unstructured (H) Structured (V)
Finite Element (H) Sigma Coordinate (V)	-	Unstructured (H) Structured (V)
Finite Volume	Structured (usually)	Structured

The physical process within the model will either be included explicitly, through physically based equations expressing the basic laws of fluid motion, or, through parameterised relationships for empirical (or semi-theoretical) representations of processes which cannot be included explicitly within the confines of the dimensionality or grid resolution of the computational model. As the model dimensionality increases and the grid resolution is refined so more physical processes can be represented explicitly rather than by empirical relationships. This, however, is at the expense of increasing complexity of the model and computing requirements.

This paper will concentrate on so-called 1-D models for two main reasons:

- they are the most widespread in use for flood defence applications within the Environment Agency and by the Agency's consultants and
- they contain the traditional bridge and culvert afflux formulae which are the subject of this scoping study.

This paper considers only those 1-D models included on the Environment Agency's Current Best Interim System for Hydraulic Models – List A – namely ISIS, MIKE-11 and HEC-RAS. There are of course other models available but for the sake of clarity only these three are considered in detail.

However, some reference is made to the estimation of bridge afflux using 2-D and 3-D models.

This expert paper complements and draws on material collected in the other expert papers prepared under this current project on the theoretical basis of afflux calculations by Professor Knight (2002) and on international practice by Kirby and Guganesharajah (2001) and Benn and Bonner (2001). Two of the expert papers on the earlier Agency R&D scoping study on reducing the uncertainty in river and floodplain conveyance are also pertinent to the current project; those of Knight (2001) and Wright (2001) on conveyance estimation in 1-D models and 2-D & 3-D models respectively.

2 TYPES AND ROLES OF MODELS

2.1 1-D steady flow

The hydrodynamic models used for river flow simulation all represent the basic dynamics of the flow (downstream acceleration, longitudinal surface gradient and bed stresses). One-dimensional steady flow models have been used for flood defence design and planning for many years starting from the tabular and step backwater methods introduced over 100 years ago. These calculation procedures were automated with the advent of digital computers in the 1950's and 1960's by government institutions such as the US Army Corps of Engineers (ACE), large consulting engineering practices and specialist hydraulics laboratories. Two steady flow models continue to be in widespread use by the Environment Agency and its consultants, the HEC-RAS software from the US ACE and ISIS-Steady, which forms part of the ISIS river modelling suite from the HR Wallingford and Halcrow joint venture. Both these 1-D steady flow models originally covered just single branches of river but now have the capability to simulate flows in more complex channel networks.

To be of use for general application, 1-D models need to represent all features of the river system which can exert influence on the water level. Thus the models include means of estimating the effects of bridges and culverts under normal flow conditions and users also can approximate the behaviour of these structures in a partially blocked condition by suitable amendment of the data. Despite their simplicity, steady flow models still play an important role in many open channel flow projects. They are suitable for the computation of flood levels over reasonable lengths of rivers (related to the attenuation rate of the flood hydrograph) and variants are available which deal with transitional and supercritical flows in steep rivers. Some limited-length design projects can be appraised entirely with steady flow models of the estimated flood flows in the watercourse concerned.

Normally steady flow models are based upon an ordered (downstream to upstream) calculation of water levels at a series of cross-sections, for known discharge rates. The water levels are calculated from a numerical approximation to the solution of the dynamic equation gradually varied flow for free-surface flows; the most important factors usually being estimates of the river resistance and sufficiently closely spaced cross-section to resolve variations in the river geometry. The steady flow assumption reduces the relationship for conservation of mass to discharge being either uniform or varying in space according to the lateral runoff rates. Where a bridge, culvert, sluice or weir controls or influences the water level, then the dynamic equation is either replaced or augmented to take account of the effects of the structure. The formulae used to represent the structures will generally be derived from analysis of laboratory experiments or field measurements, often deriving from hydraulic measurement programmes several decades old. Whilst the accuracy of the measurements need not be questioned, the form of analysis often reflects the era when hydraulic calculations were undertaken manually rather than with a computational model.

2.2 1-D unsteady network models

1-D network models may be viewed as a generalisation of steady flow models described above. They include both the variation of the flow conditions in time and the ability to simulate flows in an arbitrary network of channels, flood plains crossed coupled to a variety of hydraulic structures and other infrastructure (especially longitudinal and lateral embankments). These models can represent more complex cases than the simple backwater approach and are sometimes (mistakenly) called “quasi two-dimensional” based upon their flexibility to cover a flood plain area with a two-dimensional network of cells. Nevertheless, the flow linkages between these cells are still one-dimensional.

These models (such as HEC-RAS (Version 3), ISIS and MIKE11) are in widespread use internationally for the assessment of river flood management projects, and they form the cornerstone of the modelling undertaken by the Environment Agency in the flood defence sector.

The 1-D unsteady models generally include the influence of bridge and culvert afflux in a similar manner to the steady flow models, but the implementation of the traditional afflux formulae may need interpretation for the specific model structure. Thus it should not be assumed, as also in the case of Manning’s equation for flow resistance, that two model implementations of ostensibly the same method will deliver precisely the same answers in all situations. Ideally, any differences will be small and not significant in terms of the overall water level accuracy, however, in the Environment Agency (1997) model benchmarking project, the results of bridge afflux computations for the two tests in Benchmark 11 showed significant variations.

Depending upon the model and the detailed description of a particular case, the models may simulate the effects of flow by-passing the main channel control structure. The flow over the flood plain may be integrated with the channel flow or described separately. In either case it may be possible to simulate flow over approach embankments and through relief culverts under the embankment.

2.3 2-D steady and unsteady models

Two-dimensional models have been used in the research community for flood plain modelling for about 30 years, but so far this technology has not achieved widespread application in commercial or operational practice in the UK. Models which are in use in practice internationally include the finite element codes Telemac (developed by EDF) and RMA2 (developed originally by King). Bates *et al* (1995) have compared the performance of these models for typical UK flood plain studies, Cooper *et al* (1994) have demonstrated Telemac for modelling flooding at Gloucester (including some Bridges) and Hollinrake and Millington (1994) demonstrated Telemac on modelling structures on the River Frome. Bates *et al* (1998) also give further validation of Telemac for flood simulation over substantial lengths of river.

In general, regular, rectangular finite difference grids are not suitable for the representation of natural river geometry and so high dimensional models are normally built upon curvilinear, finite element or finite volume grids. However, De Roo (1998) has proposed a ratser-based flood plain flow model (LISFLOOD), which computes flow on a rectangular grid, but this appears to be a research prototype not in widespread use. Horrit and Bates (2001) have compared LISFLOOD with Telemac for flood plain definition. The unstructured finite element grid of Telemac has particular flexibility in representation of complex geometries.

In 2-D (in plan) models both the lateral (cross-stream) variation of velocity and of depth are resolved. This allows modelling of the separation of flow in the lee of corners in the boundary and of the additional friction losses due to increased length of streamlines caused by man-made structures. Both these factors are accounted for in the empirical formulae used in

1-D models to represent the afflux caused by bridges. Thus 2-D models resolve explicitly some factors that are impossible to represent deterministically with a 1-D model.

By the nature of the two-dimensional hydrodynamic calculations, it is difficult to incorporate a point "rating" within the grid to represent the head-discharge relationship for a culvert or bridge. Hence 2-D models represent afflux at structures by direct computation of the additional "losses" from convergence and divergence of the flow, separation and frictional resistance on increased streamline path lengths.

2.4 3-D steady models

These models are typically general purpose computational fluid dynamic (CFD) codes, developed for other application areas and implemented on riverine problems. Samuels *et al* (1998) describe the prospects for CFD for a variety of fluvial problems, especially for modelling conditions around hydraulic structures. The key choices in the codes are the representation of the free surface and the complexity of the turbulence closure. 3-D models will not be used for large river modelling studies but rather as an investigation tool for the hydrodynamics of a particular issue. Hence the issue is not how bridges and culverts are represented in these models but how well do these models account for the physics of the flows around bridges and culverts so that they can be used as an alternative to physical models for detailed design assessments or structure calibration.

Further discussion of these higher dimensional methods is given in the review paper by Wright (2001), commissioned in the Agency scoping study on reducing uncertainty in river and flood plain conveyance.

3 AFFLUX AND BLOCKAGE REPRESENTATION IN 1-D MODELS

3.1 Choices facing model developers

In a 1-D model the head-discharge relationship at a bridge or culvert is introduced either by:

- overwriting the (partial) differential equation for the flow dynamics with an algebraic equation for the structure rating, or
- by adding an slope term to the equation of the flow dynamics to provide the correct head difference between the river sections upstream and downstream of the structure.

The commercial river models reviewed below all use finite difference schemes, though not the same method. Specialist models for dam-break analysis are being developed (Morris, 2000) which use the finite volume method because of its superior numerical qualities for rapidly varying flows, but these are not yet implemented in flood modelling practice. The two finite difference methods in common use are the staggered (or 6-point) scheme used in MIKE11 and the box (4-point) scheme used in HEC-RAS, HYDRO and ISIS. With the staggered scheme water level, H, and discharge, Q, computation points are interleaved (or staggered) whereas in the box scheme water level and discharge are calculated at all points of the mesh. This difference has implications for the detailed implementation of structure afflux in the models, since in the staggered scheme the discharge in the structure is readily identified at a Q-point as are the water levels on either side of the structure at the corresponding H-points. Models based upon the Box scheme need to provide an additional relationship between the discharge values on either side of the structure.

A separate issue is how the model developers have incorporated features of the prototype which are not covered in the standard afflux calculation methods. These may include bypassing flows over the flood plain, structures skewed to the main flow directions, structures on river bends etc. The representation of conveyance in the river model may differ from that used in the derivation of the standard afflux formulae. In this case, adopting the method in the afflux formulae locally may imply inconsistencies of approach whether this is included in the main river flow representation or just for the bridge. Likewise the use of the main model conveyance methods within the afflux formulae also implies and inconsistency with the derivation of the method. Some afflux formulae (e.g. Yarnell's equation), were produced from experiments in rectangular channels with the formula implying a linear relationship between depth, flow area and pier area. In a non-rectangular section this is not the case and so the models developer faces decisions on how best to extend the method to general geometries. Such issues cannot be resolved without access to the original datasets for the afflux method development to check the sensitivity of the results to these different assumptions. At best this remains another modelling uncertainty, which although unquantified should, be small.

The hydraulic models HEC-RAS, ISIS and MIKE11 allow the user to define a headloss or a rating at any particular point within the system. Thus the user is able to include a hydraulic relationship for any structure derived from theoretical analysis, physical model experiment or field data.

3.2 Common issues between models

3.2.1 Survey data required

All models require a similar level of survey detail of the bridge or culvert site. Typically the models need a survey of the face of the structure, the pier shapes and dimensions, the bridge soffits and the approach and exit alignments. From this information the user will assess the values of the various coefficients according to the guidance given for the application of the various afflux formulae. The commercial models all have a user interface designed to facilitate the entry of the survey data. There should be no major difference between the models in their requirements for number or spacing of sections in terms of delivering the same numerical accuracy in the end calculation since all the models use methods with similar accuracy characteristics.

3.2.2 Culvert blockage representation

HYDRO-1D and ISIS have the capability of modelling trash screens over the faces of a culvert where the user specifies the geometry and proportion of the screen blocked by debris. From the user documentation supplied, HEC-RAS and MIKE11 do not appear to require data on any such screens. Thus losses associated with screens in a clear or partially blocked culvert cannot be included automatically within these simulations. Blockage of structures may be included manually within the models, through appropriate manipulation of the structure data. Alternatively, it is possible to simulate, albeit crudely, the transient effects of blockage by modelling a sluice gate in conjunction with the structure and partially closing the gate to simulate blockage in the upper part of the structure.

3.2.3 Uncertainty in the results

The Agency benchmarking project for river models included two benchmark tests (11A and 11B) for bridge afflux. The project report (Environment Agency, 1997) demonstrated considerable variation between the different estimates of afflux for both cases; for test 11A the range was 0.01 m to 0.209 m and for Test 11B the range was 0.02 m to 0.563 m. This indicates that the application of hydraulic modelling formulae without the benefit of field or experimental calibration data is subject to significant uncertainty.

3.3 HEC-RAS

This review has been based upon the HEC-RAS Hydraulic Reference Manual, Version 3.0 dated January 2001 (USACE, 2001). The major advance of this program over previous versions of the HEC-2 and HEC-RAS software is that unsteady flow is now supported. Future versions of the software are planned which will also include sediment routing. The documentation makes no distinction between facilities available in the steady and unsteady simulations and it is assumed that both offer the same description of the river system.

3.3.1 Bridge Afflux formulae

HEC-RAS3 contains four methods for estimating afflux at bridges:

- The Energy method
- The Momentum method
- WSPRO
- Yarnell's equation

All four of these methods can be applied to class A flow (subcritical) through the bridge. For Class B flow (critical depth occurs in the bridge constriction) the Momentum method is used, with the program automatically selecting the Energy method for those supercritical cases where the Momentum method does not converge. For Class C flow (entirely supercritical) the program may use either the Energy or Momentum methods

For the Momentum and Energy methods the frictional resistance between the cross-sections upstream and downstream of the bridge is included. However, when Yarnell's equation is used to estimate the losses associated with bridge piers, the head difference from Yarnell's equation is used directly to obtain the water level at the upstream section from that at the downstream section, without addition of friction loss. In this case the documentation implies that the two sections should be located close to the bridge structure.

HEC-RAS includes orifice and pressurised (surcharged) flow under a bridge deck where the upstream soffit is in contact with the water surface. The discharge coefficients vary according to the flow case that occurs (downstream face clear of water, $C_d = 0.27\text{-}0.5$, or submerged, $C_d \approx 0.8$) and the depth of submergence. This approach is limited for the case where the ratio of the upstream depth to the bridge soffit height is in the range 1.0-1.1, as the flow behaviour is unpredictable and the pressure equation is not applicable. Road overflow is modelled as a combination of pressure flow and the standard weir equation developed for rectangular channels. The weir equation incorporates the approach velocity through the use of the energy grade line elevation in place of the upstream water surface elevation in determining the upstream energy relative to the road crest. The flow is divided between the pressure and weir equations and hence iterated until the upstream energies correspond. At high tailwater elevations, a flow reduction factor is applied to the weir equation, which is dependent on the degree of submergence. Submergence in this instance is the water depth above the minimum weir elevation on the downstream side relative to the height of the energy grade line above the minimum weir elevation on the upstream side. For a submergence of 95% or greater, the pressure and weir method is automatically switched back to the energy method.

The implementation of the Federal Highway Administration (FHWA) WSPRO method in HEC-RAS accounts for the additional flow resistance due to the lengthening of the streamlines as the flood plain flow converges towards the bridge openings. This is achieved through an estimation of the plan shape of the streamlines within the WSPRO procedure. The HEC-RAS documentation includes several figures showing the coefficient variation for various elements of the FHWA method, but is not clear on how this graphical information is encapsulated within the code.

The model can accommodate several openings at a site with any combination of arches and flood relief culverts. The user needs to specify stagnation points giving the division between the various elements of the hydraulic computation.

The model also includes an option to simulate the partial blockage of the bridge opening through the accumulation of floating debris against piers and also the user can specify a siltation depth within culverts. HEC-RAS documentation provides further information on how floating debris is incorporated into the model.

3.3.2 Culvert formulae

HEC-RAS has substantial options for modelling the flow in culverts. The model includes:

- a selection of nine common culvert sections shapes
- single and multiple barrels
- free surface flow

- inlet control
- outlet control
- pressurised flow
- horizontal and adverse slopes
- entry and exit losses

The hydraulic computations are based upon the US FHWA (1985) publication "Hydraulic Design of Highway Culverts" for inlet controlled flow. For outlet controlled flow the choice of appropriate equation in the software depends on the results of several comparisons between possible cases. For full flow the head losses are calculated from the FHWA equations which include entry, exit and friction losses all of which vary with the square of the discharge through the structure. For culvert overflow the HEC-RAS performs an iterative procedure to determine the amount of flow over the weir and through the culvert. As with the bridge case, the program iterates until the split is such that the upstream energies correspond.

3.4 HYDRO-1D

HYDRO-1D is the proprietary river flow model developed by Mott-MacDonald Ltd, which has been applied in many cases in the UK and internationally for river flood simulations. The expert paper by Kirby & Guganesharajah (2001) gives a short description of the treatment of bridge afflux in HYDRO-1D, including a summary of the flow equations used but not the detailed numerical implementation. The following brief description of HYDRO-1D is taken from their expert paper.

3.4.1 Bridge Afflux formulae

Bridge structures are represented in HYDRO-1D by a "*bridge reach*" which comprises a cross section representing the open flow area of the bridge at its downstream face and a corresponding set of hydraulic parameters. Bridge piers are thus explicitly represented as a reduction in available flow area and both arch and flat soffits can be represented. Bridge Afflux is calculated through defining expansion and contraction losses in the energy equation and the loss coefficients are chosen by the user with guidance from published values in a variety of sources. The effects of bypassing flow are incorporated through the use of "*weir reach*" to simulate flow over approach embankments and the bridge deck.

3.4.2 Culvert formulae

Hydro has a separate computational unit to represent the flow through culverts which accounts for friction losses through the culvert as well as the entry and exit losses. Where friction losses are important through the bridge structure, this may be modelled more effectively as a culvert.

3.5 ISIS

The information in this review has been based upon the description of the hydraulic units given in the ISIS user reference manual (HR Wallingford, 1997). ISIS is also available as the hydraulic modelling component of InfoWorksRS which provides a broader range of support to the model user.

3.5.1 Bridge Afflux formulae

The ISIS manual describes the following options for modelling a bridge structure:

- the HR Wallingford arch bridge method
- the USBPR design method

- the use of the culvert (“Conduit”) options
- the use of a sluice-gate option to generate an orifice equation
- the use of discrete (“Bernoulli”) energy losses.

The HR Wallingford arch bridge method was originally derived from experiments in rectangular channels, with the calculation formulae being based upon the Froude number adjacent to the bridge. The ISIS model has an implementation for general river section shapes. Bridge arches are assumed to be parabolic in shape. Over and bypassing flows must be included separately using the “spill” units, which model an irregularly shaped weir crest. The method in ISIS contains a discretisation of the afflux curves in the original HR research report, with the accuracy of software implementation of the method being tested under the Quality Assurance programme of the ISIS development.

The implementation of the USBPR design method in ISIS was taken from the former FLUCOMP river model (Samuels & Gray, 1982). The details of the implementation are not, however, described in the ISIS documentation. The design curves for the various contributions to the overall loss coefficient are represented as piecewise continuous curves (generally quadratic and linear) fitted to the graphs given in the USBPR publication. These allow linear extrapolation above the range of data, any such extrapolation is reported to the user. The loss coefficients for the bridge piers permits continuous interpolation between the published reference curves in the USBPR publication to allow the user to calibrate the afflux against observations. The opening ratio for the bridge section (M) is adjusted to take account of any flow through flood relief culverts and weiring over the approach embankments.

The USBPR method incorporates three cases for partially inundated bridge structures:

- Upstream girder in contact with the flow with a free downstream discharge
- All girders in contact with the flow and subject to having submerged discharge downstream
- Flow over approach embankments

The first two cases are based on pressure and orifice equations respectively, where the discharge coefficients vary according to the submergence typically in the range C_d = 0.25-0.5. In the case of road overflow, a broad crested weir equation is applied, with a correction for downstream submergence at high tailwaters. This is combined with the pressure approach and an iterative approach is used to determine the upstream water level.

Thus ISIS manual states that ISIS does not include the frictional resistance at the bridge; it only represents the energy losses through the bridge structure.

3.5.2 Culvert formulae

ISIS contains several options for modelling culverts (or “Conduits”). These are:

- Circular
- Full arch
- Rectangular
- Sprung Arch
- Symmetrical (general, symmetrical section)

In each case the treatment of the inlet and exit losses is through the use of energy (“Bernoulli”) loss coefficients and the friction losses along the culvert are calculated from either the Manning or the Colebrook-White resistance equations. For hydraulically short culverts a sluice gate or orifice equation is recommended as a potential alternative. This orifice equation is based on a rectangular cross-section.

3.6 MIKE11

The information on MIKE11 has been taken from the latest version of the technical documentation of the bridge representation in the MIKE11 model, which was supplied by DHI Water and Environment (2001) specifically for the current project.

3.6.1 Bridge Afflux formulae

The representation of bridges within MIKE11 has recently been extended and now the user has the choice of the following afflux estimation formulae:

- Biery and Delleur method (arch bridges)
- HR Wallingford method (arch bridges)
- Nagler (piers)
- Yarnell (piers)
- D'Aubuisson (piers)
- WSPRO (through a modified version of the FHWA WSPRO method, see below)
- USBPR

Further details of these methods are given in the companion expert review paper by Knight (2002) and in the user documentation for the MIKE11 modelling package. In addition the MIKE11 model includes facilities for:

- pressure flow through surcharged arches
- road overflow around the structure
- submerged bridges

The discharge coefficient for the pressure equation is described as a tabular function of the upstream depth and the bridge soffit height, although the values are not listed in the manual. When the bridge structure bottom level is exceeded, the bridge type solution will be replaced with a submerged solution, which takes the form of the orifice equation (with $C_d = 0.8$). Overflow is only available in combination with submerged flow and occurs when the bridge structure top level is exceeded.

In the description of the implementation of the WSPRO method, the MIKE11 states that:

> *"The computations of the Federal Highway Administration's WSPRO computer program have been adapted for calculation of free surface flow. Some modifications have been necessary in order to fit the MIKE11 model."*

However, these modifications and adaptations are not disclosed in the documentation.

3.6.2 Culvert formulae

MIKE11 includes an orifice flow formula as one structure option and this can represent the head-discharge relationship for a hydraulically short culvert. The flow cases which are represented include:

- no flow
- inlet control
- outlet control
- orifice flow
- full culvert

3.7 Inter-comparison of model capabilities

Table 3.1 below summarises the capabilities of the 1-D models included in this review. Fuller details of the various methods are given in the documentation for each model and the review paper by Knight (2002) commissioned as part of this Scoping Study.

Table 3.1 Summary of methods included within commercial 1-D models

Capability / Method / Procedure	**HEC-RAS**	**HYDRO-1D**	**ISIS**	**MIKE11**
Bridge Afflux Methods				
Biery and Delleur method (arch bridges)				✓
D'Aubuisson (piers)				✓
Energy Equation Method	✓	✓	✓	✓
FHWA WSPRO	✓			✓
HR Wallingford method (arch bridges)			✓	✓
Momentum Equation Method	✓			✓
Nagler (piers)				✓
USBPR Design Method			✓	✓
Yarnell (piers)	✓			✓
Surcharged flow	✓	✓	✓	✓
Submerged bridges	✓	✓	✓	✓
Bypassing flow as weir	✓	✓	✓	✓
Friction losses included at bridge site	✓	✓		✓
Multiple openings at a site (arches and culverts)	✓	✓	✓	✓
Culverts				
Short – Entry and exit losses only	✓	✓	✓	✓
Long – friction losses included	✓	✓	✓	✓
Partially full culverts	✓	✓	✓	✓
Variety of culvert shapes	✓	✓	✓	?
Blockage				
Model Blockage of bridge opening	✓		*	
Manual estimation	✓	✓	✓	✓
Transient blockage as a sluice gate	✓	✓	✓	?
Trash/security screens		✓	✓	

✓	method or feature included
?	likely capability but insufficient information available
*	only available in the InfoWorksRS implementation of ISIS

Broadly these one-dimensional models have the same capability of modelling bridge afflux and the effects of culverts, with the MIKE11 model offering to greatest number of standard methods for bridges, but all models being capable of calibration of the afflux. The HEC-RAS software alone includes an option for modelling the partial blockage of a bridge by floating debris.

4 AFFLUX AND BLOCKAGE IN HIGHER DIMENSIONAL MODELS

From the discussion of 2-D and 3-D models in the flood defence sector (Sections 2.3 and 2.4 above) it is clear that the afflux at bridges and culverts is treated differently in higher dimensional models.

Two dimensional models account for the shape of the stream lines in plan and thus the frictional resistance along the path of the flow. By including simple turbulence models (eddy viscosity or at most the k-ε model), the separation of the flow on the downstream side of the structure can be reproduced giving areas of recirculation and the attendant energy losses (See the flow case at Maisemore Bridge in Cooper et al (1994)). However, these models do not include pressurised flow through drowned arches or culverts.

Three dimensional models may be based upon standard CFD packages such as CFX, PHOENICS or FLOW3D. These models incorporate a detailed representation of the physics of the fluid motion and are being explored as potential alternatives for scale hydraulic models to establish the stage-discharge relationships of structures, see Samuels *et al* (1998) and Nex & Samuels (1998). Thus they do not need any of the traditional afflux formulae in them, rather they are capable of demonstrating the validity (or otherwise) of idealised afflux formulae outside the range of derivation.

5 RESEARCH AND DEVELOPMENT IMPLICATIONS

From this review of the implementation of bridges and culverts in hydraulic models the following R&D implications can be drawn.

1. Afflux formulae were often developed for manual application rather than as a computational procedure, hence, model developers interpret afflux formulae to fit them within the algorithmic structure of their models.
2. There are several approaches available to modelling the afflux at a structure in HEC-RAS, ISIS and MIKE11, with choices left to the users' experience and judgement. Hence there is a need to benchmark the afflux formulae against each other – without any presumption as to which (if any) is correct and provide guidance on the most suitable approach to adopt. There have been significant enhancements of 1-D models since the Agency model benchmarking project was reported in 1997.
3. There is a need to include safety and trash screens in the data files to indicate potential for blockage and additional head losses.
4. Apart from bridges in the HEC-RAS model, there is a need to automate the representation of blockage of structures by debris, preferably transient rather than permanent to facilitate probabilistic risk analysis of the effects of blockage.
5. Since bridge sites are complex with non-standard approach and exit conditions and alignments, there may be value in practice for facilities to construct limited reach local modelling around structures as a calibration aid.
6. The uncertainty in afflux estimates from models appears not to be well understood. Further analysis of afflux estimates may lead to quantification of this uncertainty for informing decisions made on model simulations.
7. Two-dimensional modelling appears to be a viable alternative to 1-D modelling for cases where there the site topography is complex and outside that assumed in 1-D model afflux formulae. Further validation of 2-D modelling on this issue would be of value for typical UK sites.
8. Three dimensional modelling is a potential means of validating and extending the range of application of afflux formulae. The Environment Agency may wish to facilitate research in this area by the provision of data to research groups and letters of support to other funding agencies to accompany R&D grant applications.
9. Since afflux formulae include estimates of flood plain and channel conveyance, there is a link between the current project and the outputs of the Environment Agency R&D on improving conveyance estimation. Revision of bridge afflux calculation formulae may be needed following a move to use a new standard conveyance method for river and flood plain conveyance.
10. Guidelines for accounting for afflux is needed where a bridge sites is the downstream boundary of a computational model.

6 REFERENCES

1. BATES P D, ANDERSON M G & HERVOUET J-M (1995), *Initial comparison of two two-dimensional finite element codes for river flood simulation*, Jnl. Water Marit. & Energy, Proc ICE, Vol 112 no 3, paper 10666, pp238-248.

2. BATES P D, STEWART M D, SIGGERS G B, SMITH C N, HERVOUET J-M & SELLIN R H J (1998), *Internal and external validation of a two-dimensional finite element code for river flood simulations*, Jnl. Water Marit. & Energy, Proc ICE, Vol 130 no 3, paper 11434, pp127-141.

3. BENN J & BONNER V R (2002), *A review of current practice in the USA*, Environment Agency R&D project Scoping Study into Hydraulic Performance of Bridges and other Structures, Environment Agency, Bristol

4. COOPER A, BAUGH, & MILLINGTON R (1995). *Two dimensional numerical modelling of the River Severn in Gloucester and the associated flood plain*, HR Wallingford, Report SR 429, September 1995

5. DHI Water & Environment (2001), *MIKE11 Hydrodynamic Reference Manual*, DHI, Horshølm, Denmark

6. DE ROO A P J (1998), *LISFLOOD: a rainfall-runoff model for large river basins to assess the influence on land use on flood risk*, Proceedings of the RIBAMOD Workshop 2 on Impact of climate change on flooding and sustainable river management, Eds R Casale, P G Samuels & A Bronstert, Directorate General for Research and Development, European Commission, Luxembourg, ISBN 92-828-7110-X

7. ENVIRONMENT AGENCY (1997), *Benchmarking and Scoping Study of Hydraulic Models*, Stage 2 Final Report Volume 1, Project 508, Environment Agency, Bristol

8. FHWA (1985), Hydraulic Design of Highway Culverts, Hydraulic Design Series No. 5, Federal Highways Administration, US Department of Transportation, Washington DC.

9. HOLLINRAKE P G & MILLINGTON R J (1994), *Discharge and Roughness Assessment at UK Gauging Stations*, HR Wallingford, Report SR 286, March

10. HORRITT M S & BATES P D (2001), *Predicting Flood Plain Inundation: Raster Based Approach versus the Finite Element Approach*, Hydrological Processes Journal

11. HR WALLINGFORD (1997), *ISIS user reference manual*, HR Wallingford, Wallingford, OXON

12. KIRBY A & GUGENESHARAJAH K (2001), *A review of current practice for Afflux Estimation*, Environment Agency R&D project Scoping Study into Hydraulic Performance of Bridges and other Structures, Environment Agency, Bristol

13. KNIGHT D W (2001), *Conveyance in 1-D river model*s, Expert Review Paper, Annex to final report on Environment Agency R&D project "Scoping Study for Reducing Uncertainty in River Flood Conveyance", Environment Agency, Bristol

14. KNIGHT D W (2002), *Expert Review Paper on Bridge and Culvert Afflux formulae,* Environment Agency R&D project

15. MORRIS M W (2000), *CADAM Concerted Action on Dam Break Flooding*, Final Report to the European Commission, HR Wallingford, available at internet address: http://www.hrwallingford.co.uk/projects/CADAM/CADAM/index.html

16. NEX A P and SAMUELS P G (1999). *The use of 3D CFD models in river flood defence*, HR Wallingford, Report SR 542, January 1999

17. SAMUELS P G & GRAY M P (1982), *The FLUCOMP river model Engineers Guide*, Report EX 999, HR Wallingford, March.

18. SAMUELS P G, MAY R W P & SPALIVIERO F (1998). *The use of 3-D computational models for river flow simulation*, MAFF Conference of River and Coastal Engineers, Keele University, July 1998, MAFF London.

19. US Army Corps of Engineers (2001), *HEC-RAS River Analysis System, Hydraulic Reference Manua*l, Version 3.0, USACE Hydrologic Engineering Center, Davis CA.

20. WRIGHT N G (2001), *Conveyance Implications for 2-D and 3-D modelling*, Expert Review Paper, Annex to final report on Environment Agency R&D project "Scoping Study for Reducing Uncertainty in River Flood Conveyance", Environment Agency, Bristol.