The Institution of Engineers, Australia
Conference on Hydraulics in Civil Engineering
Hobart 28 –30 November 2001

Modelling of Bends and Hydraulic Structures in a Two-Dimensional Scheme

W.J. Syme
B.E. (Hons), M.Eng.Sc. (Res)
Associate, WBM Pty Ltd, Australia

Abstract: The majority of flood modelling software is based on the one-dimensional (1D) and/or two-dimensional (2D) forms of the shallow water free-surface flow equations. In addition to bed friction, the equations need to model energy or form losses where rapid changes to flow velocities occur, such as at bends and hydraulic structures. The 1D approach typically uses special structure flow equations requiring specification of contraction and expansion loss coefficients. This approach is not applicable or readily applied in the 2D domain, as 2D schemes simulate form losses by modelling the rapid changes in velocity within the horizontal plane. However, a 2D model is not an exact representation of reality and may not give a completely accurate representation of the true losses.

Resolution of the model mesh, relevance of the third dimension, viscosity formulation and other factors affect how well a 2D scheme models form losses. 2D models, particularly where the mesh is coarse, are likely to underestimate form losses and may need to have extra form losses applied as an additional term in the momentum equation. Modelling of structures by inserting a 1D element in the 2D domain tends to overestimate the losses due to form loss duplication. A reduction in the 1D element's form losses may be required in this instance. Despite their better accuracy over 1D models, 2D models still need to be calibrated, checked and fine-tuned. The paper discusses these issues in relation to flow round a bend, through culverts and over weirs. Outcomes from real-world applications are also discussed.

Keywords: One-dimensional, 1D, Two-dimensional, 2D, Hydraulic, Flood, Modelling, Bend, Structure

1 INTRODUCTION

The occurrence of energy losses associated with the contraction and expansion of water flow are well known and documented. The losses occur as water is forced into and out of constrictions, causing it to speed up, slow down, form circulations and generally flow in rapidly varying directions and speeds. These sudden changes in velocity (either in magnitude and/or direction) generate large-scale turbulence that dissipates energy as heat. The losses (referred herein as form losses, but also known as eddy or turbulence losses) are most pronounced at hydraulic structures and around sharp bends.

One-dimensional (1D) hydraulic modelling schemes (eg. ESTRY, HEC-RAS, ISIS, MIKE 11, RUBICON, SOBEK, SWMM) cannot, by virtue of using the 1D form of the shallow water equations, accurately model these rapid changes in velocity and hence the associated form losses. They typically apply form losses through special 1D structure flow equations or as additional energy loss built into the 1D momentum or energy equation.

Two-dimensional (2D) schemes (eg. TUFLOW, MIKE 21, DELFT-FLS, RMA, FESWMS, TELEMAC) can model rapidly varying velocities in the horizontal plane. These schemes simulate the horizontal formation of streamlines and associated form losses. How well a 2D scheme models these form losses fundamentally needs to be known and understood by software users. As 2D schemes are not three-dimensional (3D), and their resolution may not be sufficiently fine to represent all flow formations (eg. the vena-contracta), it is incorrect to assume that a 2D scheme does not require some adjustment (calibration).

2 FORM LOSSES

2.1 Definition

The primary types of energy loss (ie. the dissipation of energy as heat) are those associated with bed friction, and those that rapidly change the stream flow (form losses). Energy loss or head loss as referred to in this paper is the combination of bed friction losses and form losses.

Bed resistance (typically Manning's equation) is the dominant term in the momentum equation where the Manning's n is high and the depths are shallow. The head loss, Δh, is a function of the velocity (*V*), Manning's n value (*n*), hydraulic radius (*R*) and travel length (*L*) as given by Equation 1.

$$\Delta h = \frac{Ln^2 V^2}{R^{4/3}} \qquad (1)$$

Form losses result from rapid changes in velocity (magnitude or direction) such as when water is forced to contract, expand or flow round a bend. The head loss is typically expressed as a function of the dynamic head as given by Equation 2, where z is the form loss coefficient and g is the acceleration due to gravity (9.81m^2/s). Form losses are typically the dominant energy loss mechanism through hydraulic structures of short length and high velocities. Typically z varies from 0.5 to 1.5 of a dynamic head ($V^2/2g$).

$$\Delta h = z \frac{V^2}{2g} \qquad (2)$$

2.2 One-Dimensional (1D) Schemes

Form losses at bends can be incorporated into the 1D equations by applying the loss over the length of the 1D element in combination with the bed resistance using Equations 1 and 2. This approach can also used to model losses from bridge piers or similar where the bridge deck is not submerged and there is no significant contraction or expansion of flow. Alternatively, the bed resistance value (Manning's n) is increased so that it includes any form losses. There is little guidance in the literature on suitable values, and so the modeller relies on calibration data and experience (for example, see p11-11, Paterson River Flood Study, WBM 1997).

For hydraulic structures, 1D schemes typically replace the momentum equation with special equations describing the structure flow. These equations normally require contraction and expansion form loss coefficients to be specified by the user. For long structures (eg. a culvert), bed/wall resistance also needs to be incorporated using the Manning's equation or similar.

In the simplest approach, Equation 3 gives the head loss due to form losses. z_1, the contraction form loss coefficient typically ranges from 0.0 to 0.7, and z_2, the expansion form loss coefficient, from 0.0 to 1.0. V_s is the velocity in the structure.

$$\Delta h = (z_1 + z_2)\frac{V_s^2}{2g} \qquad (3)$$

In some schemes, the contraction and expansion loss coefficients are modified to take into account the upstream (approach) and downstream (exit) velocities of the flowing water. In a limiting case where, for example, the approach velocity, V_1, is the same as the velocity in the structure, V_s, it can be argued that there is no form loss, ie. z_1 should reduce to zero. In a 1D sense this is correct, although in reality unless the cross-section upstream and through the structure are identical in shape and perfectly aligned, there would be some form losses at the transition even if the flow areas are the same.

For example, MIKE 11 (DHI 2000) adjusts the contraction and expansion form losses (z_1 and z_2) to account for the channel velocities upstream and downstream of the structure using Equations 4 and 5 (*A* is the flow area).

$$z_{1a} = z_1\left(1 - \frac{A_s}{A_1}\right) \quad \textit{or alternatively} \quad z_{1a} = z_1\left(1 - \frac{V_1}{V_s}\right) \qquad (4)$$

$$z_{2a} = z_2\left(1 - \frac{A_s}{A_2}\right)^2 \qquad or\,alternatively \qquad z_{2a} = z_2\left(1 - \frac{V_2}{V_s}\right)^2 \tag{5}$$

Equation 5 is derived from balancing momentum across an abrupt expansion, while Equation 4 is an approximation to experimental results in the literature (eg. Massey 1983, p219). Where the upstream and downstream velocity approaches zero, $z_{1a} = z_1$ and $z_{2a} = z_2$. Conversely, if the upstream and downstream velocity approach the structure velocity, $z_{1a} = 0$ and $z_{2a} = 0$; representing a no form loss situation.

Different solution schemes offer different alternatives for specifying and adjusting the contraction and expansion form loss coefficients. They may also allow for additional form losses within the structure (eg. piers, pipe bends). Reference to the software manuals, testing using simple, hypothetical models, and comparison with other software and desktop calculations help to understand and develop a "feel" for the software.

2.3 Two-Dimensional (2D) Schemes

2D schemes (in the X-Y plane) model sudden changes in velocity associated with the bending, contraction and constriction of flow, and the occurrence of large-scale turbulence; thereby accounting for some or all of the form losses. How well a 2D model reproduces the form losses depends largely on how accurate the streamlines are reproduced and the influence of any 3D (in the vertical) effects. Barton (2001) demonstrates that the resolution of the mesh, whether finite difference or finite element, influences the streamlines through a constriction, which in turn effects the resulting form losses.

The need to specify additional form losses in a 2D model (to account for poor streamline reproduction or any effects in the vertical) is an area not well understood. The following section demonstrates the performance of the TUFLOW finite difference 2D software (Syme 1991, WBM 2001) based on the scheme of Stelling (1984). Techniques used to correctly model structures in 2D are discussed.

3 COMPARING 1D AND 2D SCHEMES

3.1 Right Angle Bend

An ESTRY 1D and two TUFLOW 2D models of different mesh resolutions were developed for a channel with a right angle bend as illustrated in Figure 1. The channel is 50m wide and 5m deep with a Manning's n of 0.03. The 1D model network consists of 4 nodes connected by 3 channels, with Channel 2 being that around the bend. The Coarse 2D model has mesh of 10m square cells while the Fine 2D model has a mesh of 2m cells.

All models have the same boundaries and were simulated using a range of steady-state flow conditions. The downstream outlet was held constant at 5m deep, which corresponds to a water level of zero. The 2D models used a Smagorinsky viscosity formulation with a coefficient of 0.2 (this is the case for all 2D results presented).

Figure 2 shows the water surface profiles for an outlet velocity of 2m/s. The profiles for the 2D models are along the channel centreline. They illustrate the superelevation of the water surface at the bend and the following draw-down that occurs downstream of the bend. These affects cannot be modelled in the 1D model. As shown in the figure, the Fine 2D model produces the greatest head drop and therefore has the highest form losses. The 1D model, which has no form losses, follows the Manning's equation. The Coarse 2D model lies between the Fine 2D and 1D models.

Figure 3 illustrates the flow velocity vectors and water surface contours at the bend in the two 2D models.

Observations are:

(a) The 1D model requires specification of form losses or increased bed resistance to reproduce the head drop in either of the 2D models. A form loss coefficient of 0.4 to 0.5 is required on Channel 2 to reproduce the Coarse 2D head drop, while around 1.3 is required to reproduce the Fine 2D model. (By comparison, the right angled bend loss for pipe flow is estimated to have a coefficient of ~1.0, Massey 1983.)

(b) Separation of the flow on the inside of the bend is better represented in the Fine 2D Model than in the Coarse 2D Model which shows no separation. The flow separation in the Fine 2D Model results in a wake region and eddy formation that constricts the flow downstream resulting in greater head loss.

(c) It is possible that the Fine 2D model is overestimating (or even possibly underestimating) the head drop, however, until experimental data is available for model validation it is difficult to ascertain the accuracy of different models (any information on good model validation data is welcomed by the author; email: wjsyme@wbmpl.com.au).

(d) If bed resistance is the sole energy loss parameter used in calibration (ie. it accounts for bed resistance and form losses), it can be expected that in calibrating models around bends that:

a. 1D models require the highest Manning's n values;

b. Fine resolution 2D models the lowest Manning's n values; and

c. Coarse 2D models lie between the above.

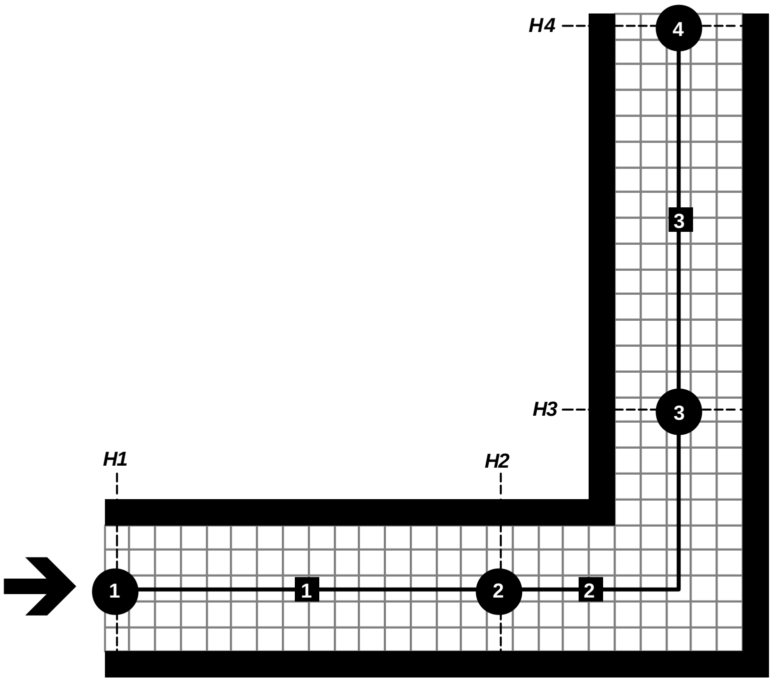

Figure 1 - 1D and 2D Layout of Right Angle Bend Model

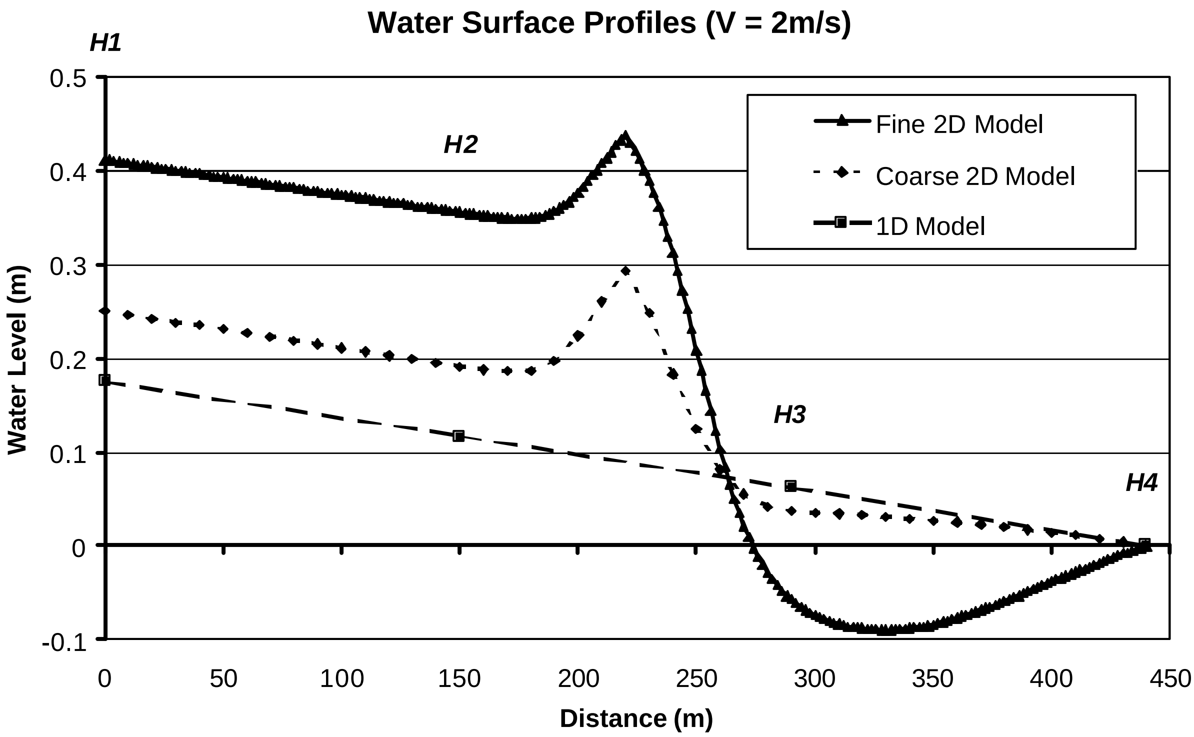

Figure 2 - Water Surface Profiles Along the Channel Centreline – Right Angle Bend Case

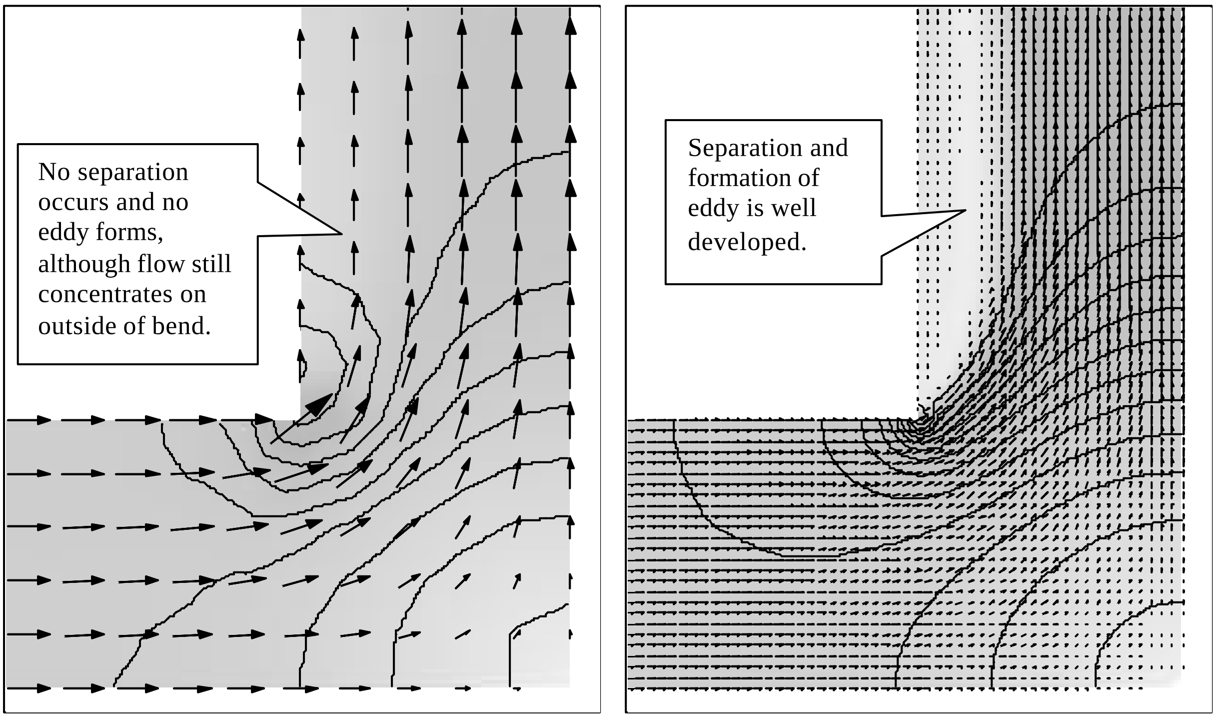

Figure 3 - 2D Flow Patterns and Water Surface Contours at Right Angle Bend

(Figure 3 illustrates the flow patterns and water surface contours at the bend for the coarse (10m) mesh on the left and the fine (2m) mesh on the right. The grey shades indicate the velocity magnitude (darker the shade the higher the velocity). Both figures use the same contour and shade palettes, although the velocity arrows are of different scales. The incoming velocity is ~1.9m/s in both figures. The development of flow separation is seen in the fine mesh resulting in eddy formation, contraction of flow and significant form losses. No eddy forms in the coarse mesh, although water is still forced to the right side, resulting in some form losses.)

3.2 Box Culvert Structures

The previous section illustrates the differences between 1D and 2D models in modelling form losses associated with flow round a bend. The magnitude of the form losses depends on how well developed the streamlines and eddy formations are; with finer (higher resolution) meshes producing more developed streamlines and greater losses.

The problem arises when modelling a hydraulic structure in 2D, of whether the 2D losses are appropriate and whether to manually specify additional form losses to satisfactorily reproduce the structure's afflux. It may even be necessary for structures with minimal energy loss to reduce the form loss in the 2D model by specifying negative additional form losses.

Figure 4 shows the 1D and 2D model discretisations for a straight channel with a bank of box culverts roughly one-third from the upstream end. The channel is a rectangular section, 100m wide with a bed at 0m and a Manning's n of 0.05. There are 16 culverts 2.4m wide and 1.8m high at an invert of 0m. The culverts in the 1D model are at Channel 2, whilst for the 2D model the four grey cells are modified to represent the culvert's obvert, walls, etc. The culvert Manning's n is 0.013 and the contraction and expansion losses were ~0.38 and ~0.50 according to Equations 4 and 5 using unadjusted coefficients of 0.5 and 1.0 respectively. The 2D model's cell size is 10m.

A second 2D model was developed with the four grey cells replaced by a dynamically nested 1D element representing the culverts. The flow patterns for the two 2D models are illustrated in Figure 5.

The 1D and two 2D models were simulated for a range of inflows and downstream water levels representing different unsubmerged and submerged flow regimes in the culverts. Figure 6 shows typical examples of the resulting water surface profiles at different flow stages. The profiles on the left present the results with no adjustment or addition of form losses.

For the Fully 2D Model (Culvert as 2D Cells) simulations, at the Unsubmerged and Outlet Controlled stages, the standard 2D equations are applied, while for the Inlet Controlled stages, the 2D equation terms in TUFLOW are automatically adjusted to represent the culvert flow equations for inlet control, unsubmerged outlet.

The equivalent profiles on the right in Figure 6 illustrate the effect of the following model adjustments:

(a) An additional form loss coefficient of 0.2 applied to the culvert 2D cells for the Fully 2D Model (Culvert as 2D Cells) model.

(b) For the dynamically nested 1D element, the culvert contraction loss coefficient reduced by 0.2 from 0.38 to 0.18.

(c) No adjustment to the 1D model coefficients.

As shown in Figure 6, these adjustments improve the agreement between the three models.

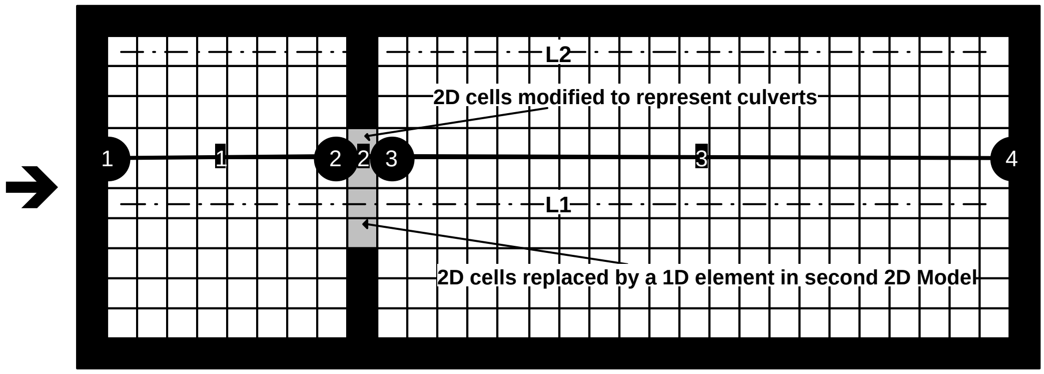

Figure 4 - 1D and 2D Layouts for Box Culvert Models

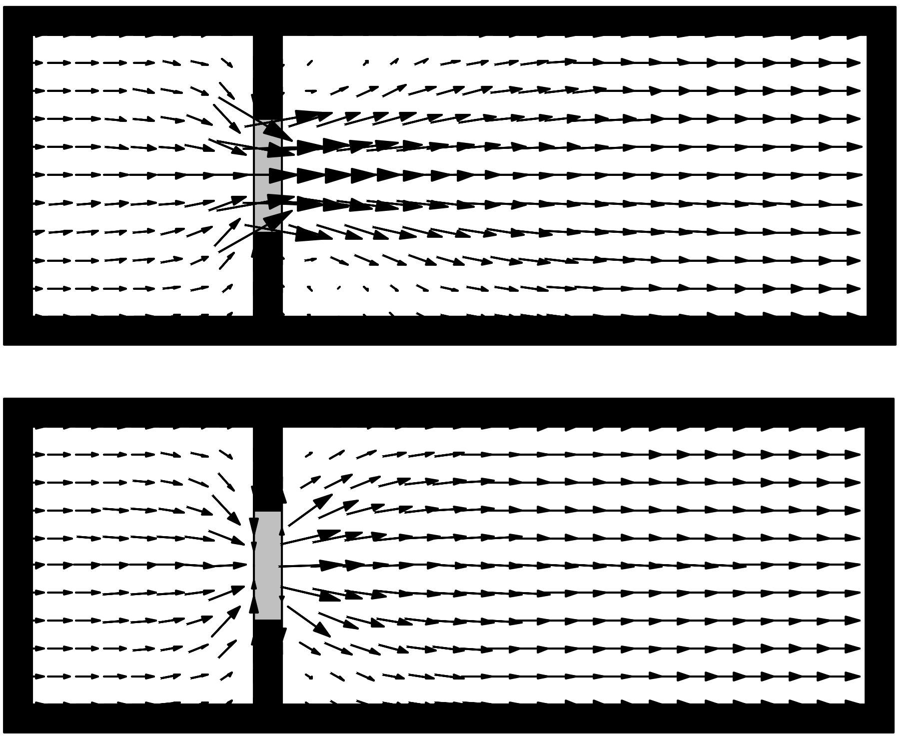

Figure 5 shows the flow patterns for the two 2D models. At the top is the Culvert as 2D Cells case that shows the momentum of the water continuing through the structure and forming small eddies on the downstream side.

For the Culvert as 1D Element case (bottom flow pattern), the momentum is broken with no eddies forming.

Figure 5 - Flow Patterns for 2D Box Culvert Models
(Culvert as 2D Cells on Top and Culvert as 1D Element at Bottom)

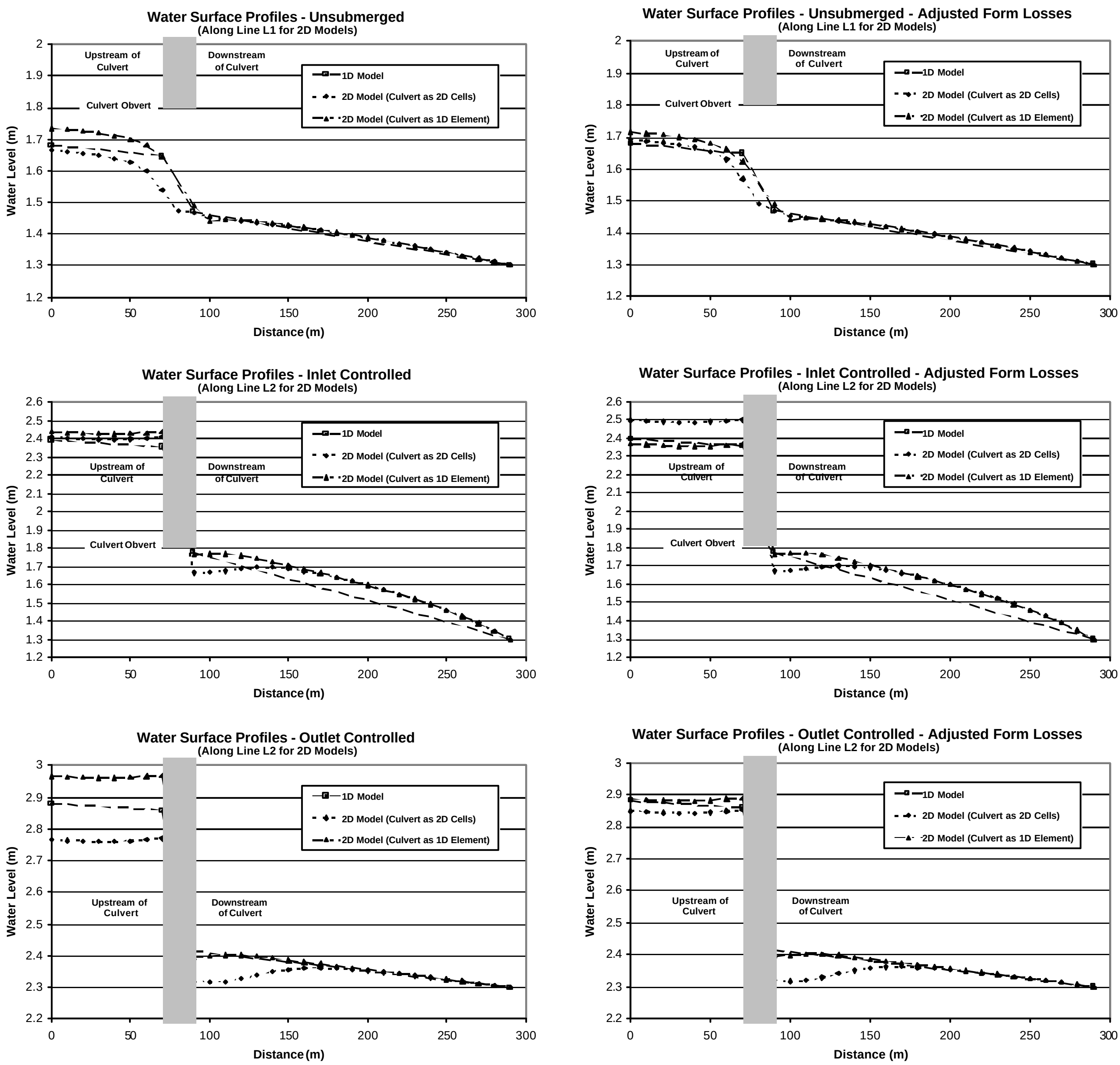

Figure 6 - 1D and 2D Water Surface Profiles for Box Culvert Model Simulations

3.3 Weirs

Flow over levees, road and rail embankments, bunds, etc is generally modelled using the broad-crested weir equation. The broad-crested weir flow equation (for an unsubmerged weir) is not based on form losses, but after simplifications takes the form shown in Equation 6, where q is the flow per unit width, g is 9.81m/s^2 and H is the energy head upstream relative to the weir crest (Henderson, 1966). Once the downstream H exceeds 0.75 to 0.85 of the upstream H (depending on the characteristics of the embankment), the weir flow starts to become submerged, and Equation 6 no longer applies.

$$q = \frac{2}{3} H \sqrt{\frac{2}{3} g H} \qquad (6)$$

Equation 6 is built into the TUFLOW 2D scheme to automatically switch in and out of unsubmerged weir flow (critical flow) at any point in the 2D domain. Equation 6 and the bed friction term (Equation 1) are equated giving an adjusted Manning's n value. The free-overfall algorithm (Syme 1991) is utilised to provide stability over dry and/or steeply sloping terrain. A weir calibration factor (default = 1), which can vary spatially over the 2D domain, is used for adjusting or calibrating the weir flow.

The box culvert model described previously was modified to represent a weir structure. The weir was modelled as a "Thin Weir" by setting the elevations along the side of the cells to 1.0m as shown in Figure 7. In this case the weir equation is automatically switched on and off depending on whether the flow across the cell sides is submerged or unsubmerged.

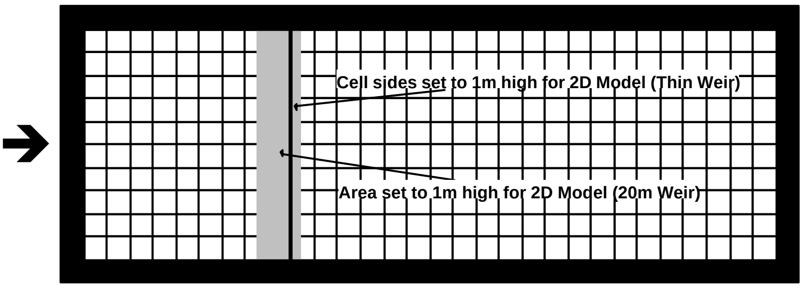

Figure 7 - Layouts for Thin and 20m Weir 2D Models

A triangular hydrograph (peak 200m^3/s) followed by a constant flow of 100(m^3/s) and a rising tailwater was simulated (see boundary time series in Figure 8). Figure 8 shows the resulting water level hydrographs at the upstream end of the model. The comparison with the 1D model results and theory (Equation 6) show excellent comparison with the results from the 2D model.

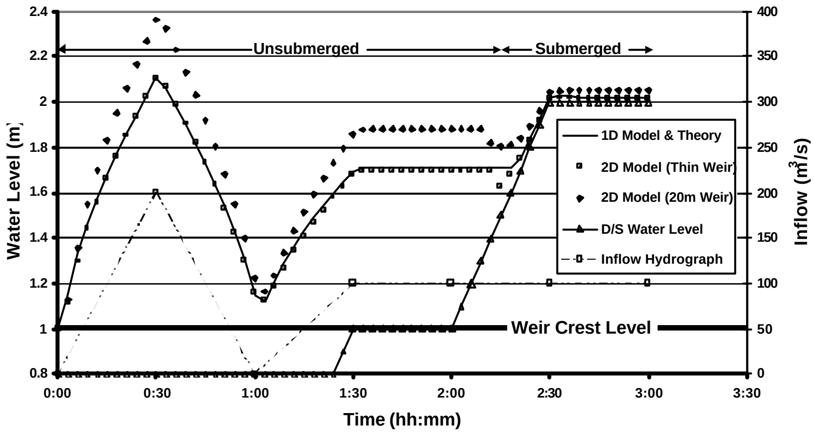

Figure 8 - Upstream Hydrographs – 2D Weir Models

An important issue is that an embankment maybe several cells in travel length (ie. in the direction of flow). For example, Figure 8 also shows that for a 20m long weir higher upstream water levels result. This is mainly due to the extra bed friction that results from high velocities over the raised cells. The modeller needs to decide whether this is an accurate depiction of the situation, or reduce the head drop by making the weir a thin weir or reducing the Manning's n values over the raised cells.

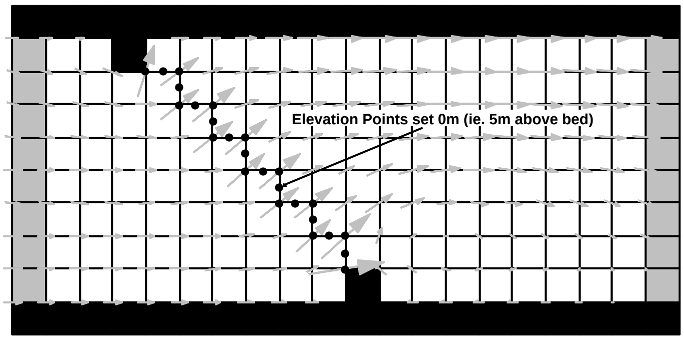

Figure 9 - 2D Model with Weir at 45° to Mesh Axes

Another issue is the performance of a fixed grid scheme such as TUFLOW where the weir flow is not perpendicular to the mesh axes. Figure 9 shows a test model with a weir at 45° to the mesh axes, and Figure 10 presents a graph of unit flow (m^2/s) across the weir versus upstream head (Equation 6), and that obtained from the 2D model. There is ~10% over prediction of the flow, which can be corrected by applying a weir calibration factor (the graph shows the result for a factor of 1.1).

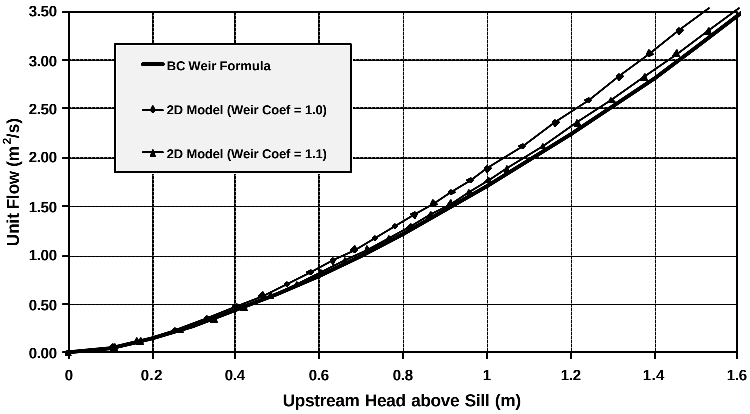

Figure 10 - Unit Flow Across an Unsubmerged Broad-Crested Weir at 45° to Mesh Axes

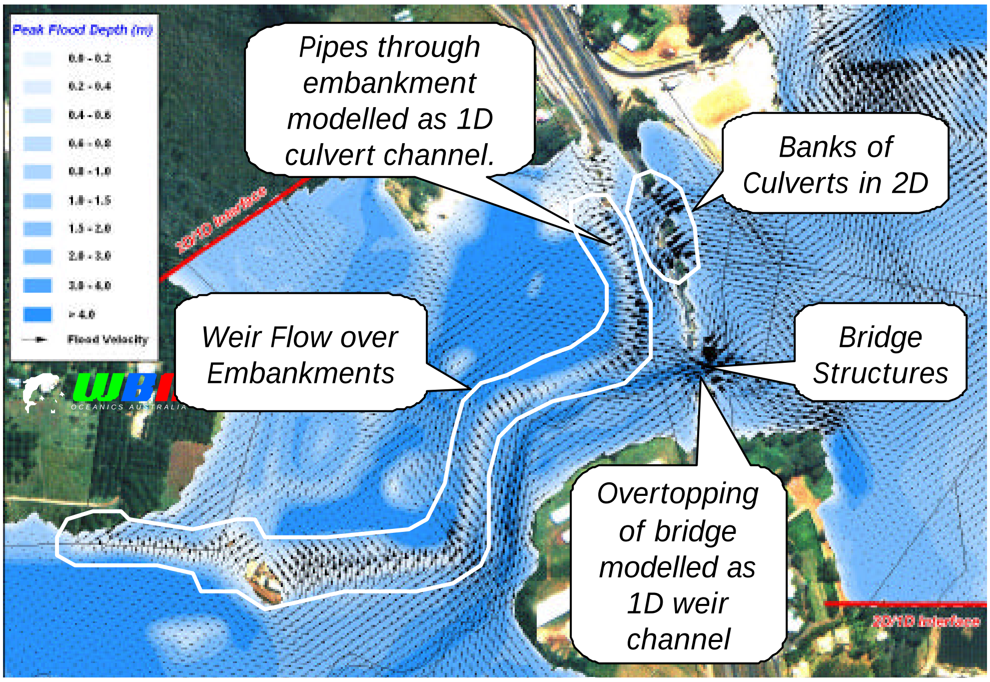

Figure 11 - Example of Complex 2D Hydraulic Structure Modelling (WBM 1999)

4 REAL-WORLD APPLICATIONS

The TUFLOW scheme has been applied to a wide range of real-world flooding applications involving complex flows and arrangements of hydraulic structures. A number of these have replaced sections of existing 1D models where it was interesting to compare the two styles of modelling. General observations are:

(a) Where significant bends occurred, there was little or no need to provide additional form losses or increased Manning's n values in the 2D model to reach calibration, whereas the 1D models required these additional losses.

(b) 1D and 2D models generally agree well when the flow was mostly confined to the main river channel. Significant differences can occur when the flow patterns become complex (by taking different courses over the floodplains), and the fixed flow paths and/or the level of detail in the 1D model are inappropriate.

(c) Designing hydraulic structures using 2D models (where the flow patterns are complex) leads to more accurate modelling and savings in civil design and construction.

Figure 11 illustrates an example of complex hydraulic structure modelling carried out using TUFLOW.

Based on the results of test models and numerous real-world applications, the following are typical observations of the TUFLOW software.

(a) Box culvert structures modelled in 2D tend to require an additional form loss coefficient of from 0.1 to 0.3 to reach agreement with culvert design curves.

(b) Dynamically nested 1D structure elements in 2D models model tend to overestimate the form losses. This is thought to be due to some duplication of losses between the 2D domain and the 1D element. These structures need to have the combined contraction and expansion loss coefficients of the 1D element reduced by amounts varying from 0.0 to 1.0. Structures with widths less than the 2D model's cell size usually require no or minimal reduction in the loss coefficients, while larger structures with high velocities may require as much as a 1.0 reduction in the loss coefficient(s).

(c) Testing and checking of real-world applications has shown that culverts and weirs can be correctly modelled in 2D at an angle oblique to the mesh axes (TUFLOW uses a fixed grid mesh).

5 CONCLUSIONS

(a) 2D models, like 1D models require cross-checks and possible adjustment when modelling hydraulic structures, bends and other transitions. Adjustment can be in the form of applying additional form losses. For minimum energy structures, negative additional form losses may be necessary.

(b) 2D models produce different amounts of form loss depending on the resolution of the mesh.

(c) A 2D model of a bend produces form losses, superelevation effects, eddies and so forth. In this sense they are closer to reality than 1D models and produce a more accurate depiction of the flow behaviour and water surface. However, additional form losses or higher bed resistance may be required if the mesh is coarse.

(d) A 2D model of a hydraulic structure:

a. Offers over a 1D model improved flow descriptions and water surface predictions.

b. Cannot be assumed to produce the correct form losses.

c. May need to be "calibrated" using additional form losses to account for poor streamlines caused by mesh coarseness and 3D effects.

(e) Substitution of 1D elements in a 2D model to represent hydraulic structures needs to recognise:

a. Where the structure width spans several or more 2D cells, duplication of form losses and overestimation of the afflux results. The form losses applied to the 1D insert very likely need to be reduced.

b. Poor flow patterns, especially on the downstream side, can occur as the continuity of the water's momentum in the 2D domain is broken by the 1D element.

c. Is best suited to where the structure width is less than that of the 2D cells, or it is not practical or possible to use a 2D discretisation.

d. Where the structure flow is upstream (inlet) controlled, substitution of a 1D element is recommended, the exception being if the 2D software has specialised routines (eg. weir flow).

(f) Whilst 2D models produce more accurate flow patterns and water surfaces than 1D models, they still require to be calibrated to historical flood data. It is still just as important that data be collected during and after flood events for the calibration of 2D models.

(g) It is likely that the above conclusions apply to 2D schemes other than TUFLOW, especially implicit finite difference ones. However, schemes vary slightly in their behaviour and response to changes in mesh resolution and other parameters (Syme et al 1998, Barton 2001). It is therefore important the modeller develops a "feel" and an understanding so as to gain confidence in each scheme he/she uses.

6 RECOMMENDATIONS

(a) On-going research and testing of 2D models to develop guidelines for adjustment of form loss related parameters when modelling hydraulic structures in 2D.

(b) Establishment of guidelines and standard tests (preferably based on experimental results) for validation of 2D schemes.

7 REFERENCES

Barton, C.L. (2001). *Flow Through an Abrupt Constriction – 2D Hydrodynamic Model Performance and Influence of Spatial Resolution.* Thesis submitted as partial fulfilment for Master of Engineering Science, Environmental Engineering, Griffith University, July 2001.

DHI (2000). *MIKE 11 Reference Manual*. Danish Hydraulic Institute, Hørsholm, Denmark, 2000.

Henderson, F.M. (1966). *Open Channel Flow*. Macmillan Publishing Co., Inc, 1966.

Massey, B.S. (1983). *Mechanics of Fluids 5th Edition*. Van Nostrand Reinhold (UK) Co. Ltd., England, 1983.

Stelling, G.S. (1984). *On the Construction of Computational Methods for Shallow Water Flow Problems*. Rijkswaterstaat Communications, No. 35/1984, The Hague, The Netherlands.

Syme, W.J. (1991). *Dynamically Linked Two-Dimensional / One-Dimensional Hydrodynamic Modelling Program for Rivers, Estuaries & Coastal Waters*. William Syme, M.Eng.Sc (100% Research) Thesis, Dept of Civil Engineering, The University of Queensland, May 1991.

Syme, W.J., Nielsen, C.F., Charteris, A.B. (1998). *Comparison of Two-Dimensional Hydrodynamic Modelling Systems Part One - Flow Through a Constriction.* International Conference on Hydraulics in Civil Engineering, Adelaide, September 1998.

WBM (1997). *Paterson River Flood Study*. WBM Oceanics Australia, Document N0076.1.2, May 1997.

WBM (1999). *Bruce Highway Crossing – Eudlo Creek Hydraulic Analysis.* WBM Oceanics Australia, Document 11329.R1.2, July 1999.

WBM (2001). TUFLOW and ESTRY User/Reference Manual (Version 3). WBM Oceanics Australia, 2001.