Acknowledgments

This research was funded by the National Highway Institute and was undertaken with the collaboration of BMT WBM Pty Ltd. The authors would like to thank all of those who contributed to this research.

References


K. J. Leister, and M. A. Jempson

BMT WBM Pty Ltd

PO Box 604, Collins Street West
Melbourne VIC 8007
AUSTRALIA

E-mail: joel.leister@bmtwbm.com.au

Abstract: Bridge embankments and structures can significantly influence flooding patterns and levels on floodplains. The literature (Austroads 1994; Bradley 1978) provides details of how to estimate contraction and expansion losses and pier losses of bridge embankments and structures for desktop analysis. However, limited guidance is provided as to the application of these losses in a 2D modelling environment.

BMT WBM is currently undertaking research that involves the replication of physical flume models tests (undertaken at Colorado State University by Liu, Bradley and Plate, 1957) in the 2D hydraulic model, TUFLOW. The data from these physical flume tests formed the basis of all current literature into the contraction and expansion losses and pier losses of bridges. This paper will present the research that has been undertaken by BMT WBM and discuss its implications for the representation of key structures in 2D flood models.

Keywords: TUFLOW, flume, abutments, piers, backwater, hydrodynamic model

1. INTRODUCTION

Backwater calculations for bridge design in Australia are based primarily on the Austroads publication “Waterway Design – A Guide to the Hydraulic Design of Bridges, Culverts and Floodway” (Austroads, 1994). The section on bridge design is based on the publication, “Hydraulics of Bridge Waterways” (Bradley, 1978) resulting from work undertaken by Bradley for the National Highway Institute in 1978. The findings published in Bradley are based on a series of flume tests undertaken by Liu et al at the Colorado State University and documented in the publication “Backwater Effects of Piers and Abutments” (Liu, Bradley, & Plate, 1957).

A two-dimensional (2D) modelling scheme will inherently model the energy losses associated with contraction and expansion, but the reliability of the representation of the losses is dependent on the scale of the contraction relative to the model element size and the model's ability to replicate the energy losses associated with the varying scales of turbulence from sub-grid to larger than grid. There may be other modelling imperatives that dictate an element size that is too large to reliably represent the losses, in which case additional losses should be built into the model. The losses in Austroads (1994), which are presented as coefficients of velocity head, could be useful in this regard, but there is no basis for the modeller to make such a judgement.

With regards to piers, the 2D model element will, in most situations, be too large to adequately represent the pier geometry and hence the flow patterns and pier losses may not be reliably represented. Further, 2D models generally have simplified turbulence models such that they are unable to reliably represent the energy losses around piers. Therefore additional losses are normally applied to the 2D model to represent the losses associated with piers. Austroads (1994) provides useful information in this regard in that pier losses are given as coefficients of velocity head. However, Austroads (1994) does not provide advice with regards to the application of these losses to 2D schemes.

The research undertaken by the authors aims to determine appropriate techniques for modelling energy losses associated with bridge constrictions and piers when using a 2D hydraulic modelling scheme. Specifically, the following hypotheses will be tested:

1. The energy losses associated with bridge constrictions can be accurately represented in a 2D model.
2. The energy losses associated with piers can be accurately represented in a 2D model.
3. Additional losses should be applied to the 2D model to represent the losses associated with piers.

The research undertaken by the authors aims to determine appropriate techniques for modelling energy losses associated with bridge constrictions and piers when using a 2D hydraulic modelling scheme. Specifically, the following hypotheses will be tested:

1. The energy losses associated with bridge constrictions can be accurately represented in a 2D model.
2. The energy losses associated with piers can be accurately represented in a 2D model.
3. Additional losses should be applied to the 2D model to represent the losses associated with piers.
that a 2D modelling scheme can reproduce, within reasonable bounds of uncertainty, the contraction and expansion losses associated with flow through a bridge opening as indicated by physical model tests undertaken by Liu et al (Liu, Bradley, & Plate, 1957); and

(ii) that the energy loss coefficients associated with bridge piers as reported in Liu et al can be applied in a 2D modelling scheme to reproduce, within reasonable bounds, the increase in water level reported by Liu et al.

2. METHODOLOGY

The computer modelling of the test flume used by Liu et al (1957) was undertaken using the hydraulic modelling package TUFLOW (BMT WBM, 2008). TUFLOW is a two-dimensional finite difference model that uses the 2D Shallow Water Equations to determine the water surface. The TUFLOW model is based upon a regular square grid of uniform elements that each contains information regarding the surface roughness (Manning's ‘n’ value) and topography.

The analysis was undertaken in three parts, broadly:

- calibration of the test flume under normal flow conditions;
- determination of the afflux due to the presence of abutments; and
- determination of the afflux due to the presence of piers in the flowpath.

The results from each of these three analyses were compared to the results obtained from the physical flume, as presented in Liu et al (1957) in order to determine the ability of TUFLOW to reproduce, within reasonable bounds, the results of the physical flume.

The work by Liu et al (1957) was undertaken using imperial measurements (feet and inches), whilst TUFLOW relies on metric dimensions. Consequently, both imperial and metric dimensions are used throughout the description of the methodology.

2.1. Abutment Tests

The abutment flume tests were based upon eight of the base case flume models. In total 40 abutment tests were undertaken which modelled five constrictions ranging in width from 2 feet to 6 feet. Each of the tests was conducted on the TUFLOW flumes which consist of 9 individual flumes of varying element size. The aim of these tests was to determine the influence of element size versus constriction width when determining the afflux, and more importantly, does TUFLOW replicate, within reasonable bounds, the results from the physical flume tests.

The constriction was modelled by varying the topography to create an opening within the abutments across the width of the flume of the required size. In each of the tests, a feature of TUFLOW known as a flow constriction (FC) was utilised. A flow constriction within TUFLOW allows the user to modify the properties of a given element to reduce the available flow width and therefore model a partially blocked cell (BMT WBM 2008).

2.2. Pier Tests

The pier tests were undertaken using three distinct methods to determine their applicability in the determination of the afflux as a consequence of the presence of piers. The pier tests were undertaken on piers only and did not include any influence of significant abutment constrictions or the presence of a bridge deck, both of which would be of importance in any real-world analysis.

A number of different pier types, including square shaft piers, round-ended narrow piers, single shaft piers and double shaft piers, were analysed using a number of the flumes modelled as part of the base case calibration. The pier were tested in a number of different configurations of pier size and pier numbers as documented in the various tests undertaken by Liu et al (1957).
The pier tests were undertaken using three different methodologies; the use of form loss coefficients (applied to both the entire cross sections (Method One) and only to individual cells containing a pier (Method Two)) and the blockage (or partial blockage) of elements containing piers (Method Three).

3. RESULTS AND DISCUSSION

3.1. Abutment Tests

3.1.1. Influence of Viscosity Coefficient

TUFLOW, by default, uses the Smagorinsky viscosity formulation to model the eddy viscosity which is used to approximate the effect of sub-grid scale turbulence. The work of Barton (2001) showed that the spatial resolution of a 2D model does have an impact on the ability of the model to predict the energy losses due to turbulent effects.

A series of abutment tests were simulated within TUFLOW to determine the influence of the viscosity coefficient. The tests were undertaken using viscosity coefficients of 0.1, 0.2 (the default) and 0.4 and were all run using the Smagorinsky viscosity formulation. The results from one of these tests, in this case using a 5 foot opening, are presented in Figure 1.

![Figure 1 Influence of Viscosity Coefficient](image)

The results from these tests indicate that the viscosity coefficient has minimal influence over the model’s predictive performance when the models are on a large grid (the left hand side of the figure). However, as the model’s grid size becomes finer, the results using the different viscosity coefficients start to diverge. The spread of values appears to increase as the viscosity coefficient increases, and this is particularly evident for the fine grid scale models (the right hand side of the figure).

These results are not surprising. As the grid becomes finer relative to the scale of the turbulence the model inherently represents more of the losses and hence relies less on the viscosity formulation and so a small coefficient is required. For the current modelling, these results suggest the adoption of a viscosity coefficient equal to 0.1 would be appropriate to obtain a better match between the predictive results of TUFLOW and the recorded results of Liu et al (1957), especially at a fine grid scale.

These results are consistent with those of Barton (2001) and BMT WBM (2008) that indicates caution should be used when using very fine grids as the influence of the viscosity term can be particularly relevant.
A viscosity coefficient equal to 0.1 was adopted for the current research. This coefficient is different to that recommended in BMT WBM (2008), however, the model results support its use in the current analysis.

3.1.2. Abutment Analysis

The results from the abutment tests were plotted to determine the influence of two components of the blockage on TUFLOW’s predictive ability. The first of this components was the influence of the number of grids contained within the constriction (Figure 2) and secondly the influence of the number of grids adjacent to the blockage (Figure 6).

In each of these figures, the plotted points can be used to determine some details about the particular scenario being modelled. The squares, diamonds, triangles, circles and dashes correspond to models with a 2 foot, 3 foot, 4 foot, 5 foot and 6 foot constriction opening respectively, whilst each colour indicates a series of models running under the same set of conditions (inflow, slope, roughness).

These results suggest that once 6 elements exist within the constriction, the TUFLOW models will, within reasonable bounds, replicate the results recorded in Liu et al (1957). It was thought that the main influence on the model’s ability to replicate the afflux would be related to its ability to represent the sub-grid scale turbulence. Whilst the model will never fully be able to represent the sub-grid scale turbulence due to limitations within the viscosity formulation, the representation of this turbulence would improve with finer grid scales. However, the expectation that at a large grid size, the TUFLOW model would under-predict the afflux when compared to the physical model is not supported by the results shown in Figure 2.

Figure 2 suggests that models with less than 2 grids within the constriction will result in a poor correlation to the afflux determined by the physical flume model (TUFLOW results in a higher afflux). As seen in this figure, the models with poor correlation were generally simulating either a 2 foot or 3 foot opening, suggesting that there may be other factors influencing the results, rather than simply the representation of the sub-grid scale turbulence through contraction and expansion of the constriction.

A subset of Figure 2 is shown in Figure 3 and is displaying the results from the flumes with a 3 foot opening only. This figure clearly demonstrates the poor predictive performance of TUFLOW for a number of the flume tests when on a large grid scale. Figure 3 (and Figure 2) shows that as the grid size decreases, the model’s predictive performance improves.
In exploring the poor correlation between TUFLOW and the physical flume when modelling a small opening on a large grid, the same results were plotted against the Froude Number (calculated at the location where the maximum afflux occurred). The results from this analysis are plotted in their entirety in Figure 4, whilst a subset showing only the results from the models with a 3 foot opening is displayed in Figure 5.

Figure 4 and Figure 5 both show that for flow conditions resulting in a Froude Number of less than 0.1, the predictive performance of TUFLOW will be poor. However, when the flow conditions result in a Froude Number at the point of maximum afflux greater than 0.20, the results from the TUFLOW model have, within reasonable bounds, reproduced the results of the physical flume tests.
The abutment analysis was also undertaken to determine the influence, if any, of the number of grids located adjacent to the blockage. The results from this analysis are presented in Figure 6.

Unlike the results seen in Figure 2, Figure 6 shows no clear trend that the number of cells located adjacent to the blockage is having a significant influence on the results. In general terms, the results appear to indicate that if more than 6 grids exist adjacent to the blockage, the TUFLOW model will reproduce the results of the physical model within reasonable bounds. However, there are also a number of results where there are less than 6 grids adjacent to the blockage that also provides a good replication of the physical model results.

3.2. Pier Analysis

The pier analysis was undertaken for four distinct types of piers; square shaft, single shaft, double shaft and round-ended narrow. As discussed previously, the pier losses were applied to the model in three distinct methods with the intention of determining an appropriate method to model piers within a 2D hydraulic model. The results from this analysis are presented in Figure 7 for the square shaft piers only.
Figure 7 shows the results from the various methods of modelling a square shaft pier. These results include the piers modelled through the application of a loss coefficient to the entire cross section (yellow squares), application of loss coefficient to individual elements (blue diamonds) and as a partial blockage of the model cell (light blue triangles). As shown in the figure, modelling a pier as a partial blockage of an individual element will almost certainly result in an afflux lower than that observed through the physical flume testing. Although the differences in this case are quite small, it would be expected that the differences would increase in real-world applications.

The square shaft piers are the only ones that result in a modelled afflux lower than the physical flume test when using either Method One (yellow squares) or Method Two (dark blue triangles) to apply the form loss coefficients to the hydraulic model.

The corresponding figures for the other modelled pier types, although not presented, demonstrated that in general, the results between the two methods involving the application of form loss coefficients (Method One and Method Two) will be identical when the number of elements per pier is less than 1; however, this is not always the case due to the way in which the piers are arranged across the cross section. Once the number of elements per pier is greater than 1, the results from Method Two provide a closer match to those observed in the physical model.

These results indicate that it is more appropriate to apply the form loss coefficient calculated from the literature (eg: Austroads, 1994), rather than model the piers as a blockage within the cell. The differences between applying the form loss coefficient to either the entire cross section or individual cells are relatively minor under the conditions tested, although Method 2 resulted in a marginally better prediction of the afflux. Under higher velocity regimes the differences would be greater, with Method 1 resulting in greater afflux. Method 1 is intuitively consistent with the original laboratory tests in that the measurements were at a location away from the localised effects around an individual pier and hence were an averaged value for the cross-section. Therefore, until further research is undertaken it is recommended that Method 1 be adopted.

4. CONCLUSIONS AND RECOMMENDATIONS

The research that has been presented in this paper has shown that for the majority of modelled flumes tested, these hypotheses can be considered true. However, a number of conclusions and recommendations have been determined based upon the results of the research and have been documented below.

- The importance of the viscosity coefficient increases as the grid size decreases and the turbulence associated with the flow conditions can be modelled as a grid scale rather than at a sub-grid scale.
The predicted afflux of small constrictions relative to the grid size should be checked against additional methods to ensure the afflux is not significantly over-predicted.

The results suggest that a modeller should try to include at least 6 model elements within a constriction to enable an accurate prediction of the afflux due to the contraction and expansion. The number of elements adjacent to the blockage is not a significant factor in the afflux predictions.

The research has shown that the modelling of a pier through the partial or complete blockage of individual elements will result in an under-prediction of the afflux due to the pier when compared against a physical flume result.

The application of form loss coefficients obtained from the literature to model the energy losses associated with a pier is an appropriate way to model piers within a 2D hydraulic model. Form loss coefficients should be applied across the entire cross section until further research is undertaken.

Additional research is recommended to further explore, subject to the availability of suitable data, the combined influences of abutments, piers and bridge decks on the upstream afflux.

5. REFERENCES


