2004

Numerical modeling of flow at an open-channel confluence

Muttucumaru Sivakumar
University of Wollongong, siva@uow.edu.au

Kalyani Dissanayake
kd09@uow.edu.au

Ajit Godbole
University of Wollongong, agodbole@uow.edu.au

Publication Details

Sivakumar, M., Dissanayake, K. & Godbole, A. (2004). Numerical modeling of flow at an open-channel confluence. In M. Mowlei, A. Rose & J. Lamborn (Eds.), Environmental Sustainability Through Multidisciplinary Integration (pp. 97-106). Australia: Environmental Engineering Research Event.

NUMERICAL MODELING OF FLOW AT AN OPEN-CHANNEL CONFLUENCE

M. SIVAKUMAR [1], K. DISSANAYAKE [1] AND A.GODBOLE [2]

[1]*Sustainable Earth Research Center (SERC), School of Civil, Mining and Environmental Engineering, University of Wollongong, NSW 2522, Australia*
[2]*School of Mechanical, Materials and Mechatronics Engineering, University of Wollongong, NSW 2522, Australia*

SUMMARY: This paper presents the first part of a 3D numerical simulation of a horizontal-bed open-channel water flow with a 90° equal-width junction. A commercially available CFD package is used. The results of the numerical simulations are compared with the experimental data published by previous researchers. The numerical simulation was carried out in two steps: (a) using a Cartesian mesh to determine the shape of the free surface, and (b) using a body-fitted mesh conforming to the free-surface shape. The first step yielded a fair comparison between simulated and experimentally determined free-surface profiles. Further work on the simulation in the second step is continuing. This study is also the first stage of a project involving numerical modeling of open channel junction flows with suspended sediment transport.

1. INTRODUCTION

Open channel junctions are a common occurrence in many natural waterways and man-made hydraulic systems such as in water and wastewater units. Confluences are critical elements, especially in the drainage geometry, of any hydraulic system (Weerakoon, 1990). Study of channel junction flows has considerable importance in the design of hydraulic structures. Flow through an open channel junction is a function of numerous variables, such as angle of confluence, channel width and cross-section, channel bed slope, flow direction and discharge, bed and wall roughness and Froude number of the downstream flow. This makes an adequate theoretical description difficult.

The entry of the lateral flow into the main channel throttles the main channel flow, causing in an increase in the hydraulic resistance to the flow. Turbulent mixing of the two streams results in considerable energy losses at the junction due to local turbulence. The upstream water level in the main channel rises, and the flow detaches from the channel sidewall immediately downstream of the junction, on the same side as the branch channel. The resulting three-dimensional separation zone reduces the available channel capacity for the combined flow downstream of the junction. Sediment particles trapped in the separation zone can gradually build up in this 'recirculation zone', changing the channel cross-section geometry and aggravating the scour and bank erosion effects.

The separation zone is a critical factor in channel junction design when considering both sediment flow and erosion problems. Water surface superelevation in confluences give rise to greater flood risk to the society (Weerakoon 1990). Therefore a thorough understanding of junction flow hydraulics is of considerable importance in river engineering in controlling local sedimentation processes, channel scouring and sidewall erosion.

With increasing discharge from the main channel, the separation zone decreases in width and length (Weber, et al, 2001). Also, the size of the separation zone and the surface depression within the separation zone increase with increasing the junction angle (J. Huang et al, 2002). The separation zone also shows a marked depression in the free surface immediately downstream of the junction. A second distinguishing feature of open-channel junction flows is the appearance of a shear plane skewed to a lesser or greater extent, depending upon the difference in flow velocities in the channels as shown in Figure 1 (Weber et al, 2001).

Previous researchers on open channel junction flows have focused mostly on a theoretical description in terms of a simplified mathematical model (e.g. Taylor, 1942; Hsu, 1998). The gross features of the flow are adequately predicted by the mathematical model (Weber et al, 2001), but not the detailed three-dimensional features of the flow. Experimental approaches or physical models are troublesome not only due to the measuring techniques available but also due to their inherent scale effects (Weerakoon, 1990). Recent developments in the techniques of computational fluid dynamics (CFD) offer the promise that these details could also be predicted (Olsen, 1999). However, the results of CFD simulations are themselves dependent on the adequacy of the mathematical models and the boundary conditions that represent the actual flow situation. They must therefore be validated against detailed experimental results. Weber et al (2001) have made available such a set of detailed experimental results, including free-surface mapping. This paper presents a 3D numerical simulation of this particular feature of open-channel junction flows, using a commercially available CFD package, PHOENICS (version 3.5).

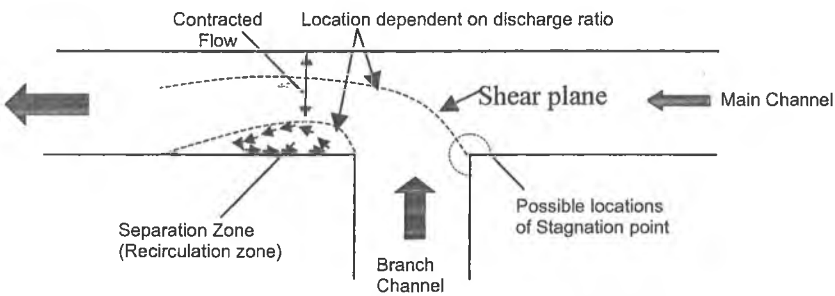


Figure 1. Flow Characteristics in Open Channel Junction (Weber, et al, 2001)

2. EXPERIMENTAL ARRANGEMENT (WEBER ET AL, 2001)

Weber et al (2001) performed laboratory experiments in a 90° combining flow flume as shown in Figure 2. Header tanks on both the main and branch channels supplied the varying discharge. Perforated plates and 100 mm thick honeycomb were placed at the main and branch channel inlets to cut down turbulence from pumped flows. To minimize losses on bends the channel transition piece were made smooth from vertical to horizontal and the floor of the entire facility was kept horizontal. The main channel is 21.95m long and the junction occurs 5.49 m downstream of the flume entrance. The branch channel is 3.66 m long. The branch channel and the downstream combined flow channel are all 0.914 m in width and 0.51 m in depth. The total combined flow, 0.170 m^3/s, and the tail-water depth, 0.296 m, were held constant, yielding a constant downstream Froude number (0.37), and a constant tail-water average velocity (0.628 m/s).

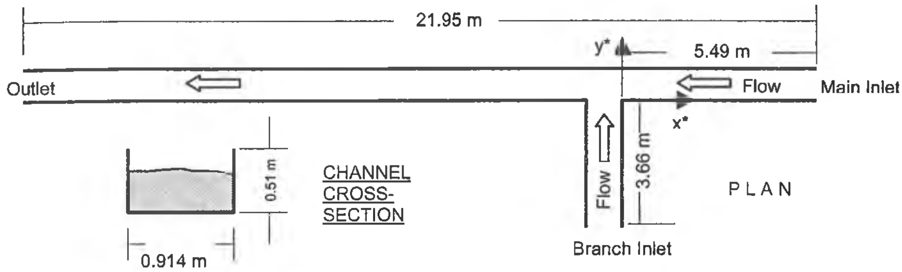


Figure 2. Schematic of Experimental Arrangement (Weber et al, 2001)

3. NUMERICAL SIMULATION

3.1 Computational Domain

To solve this flow problem in open channel junction numerically, the physical dimensions of the channels have to be represented in a computational domain as shown in figure 3. The dimensions of the computational domain are slightly greater than the physical dimensions of the flow in the experiment: 22.95 m, 5.07 m and 0.51 m in the x, y and z directions, respectively. This allows the simulated flows in the two channels to be uniform at locations sufficiently upstream of the junction, and is analogous to the experimental method of using perforated plates and honeycomb screens to achieve the same effect (Weber, et al, 2001). The blocks shown in figure 3 ensure that the numerical simulation of flows are confined to the main and branch channels.

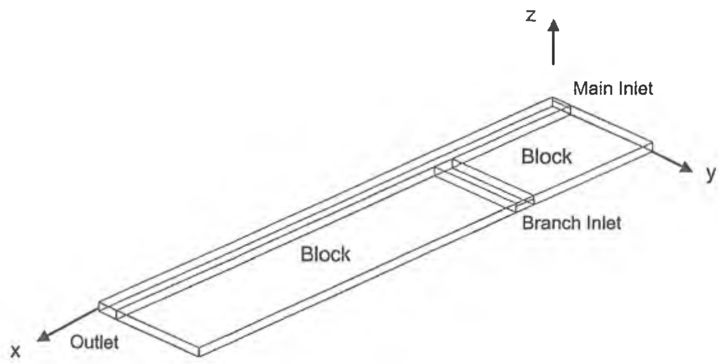


Figure 3. Computational Domain

3.2 Computational Grid

The confluence angle of 90° and channels of rectangular cross sections make it possible for the entire computational domain to be shaped like a single 'box'. This allows a structured Cartesian mesh to be used, provided the space not occupied by the flow itself is rendered impervious to the flow ('Block's in Figure 3). The mesh itself has a total of 155, 60 and 8 cells in the x, y, and z directions respectively, with a denser cell population around the channel junction, especially in the downstream direction, where significant gradients in the flow parameters are expected (Figures 4 and 5).

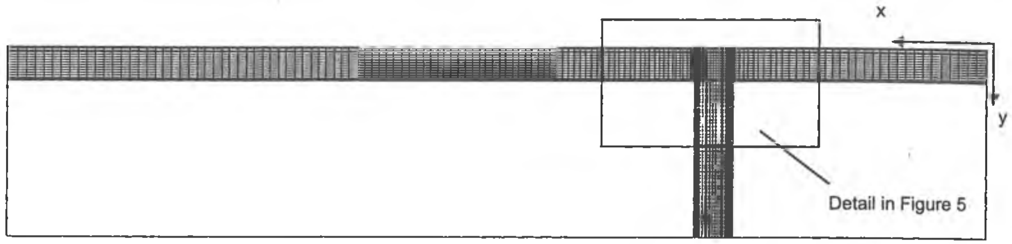


Figure 4.Computational Mesh

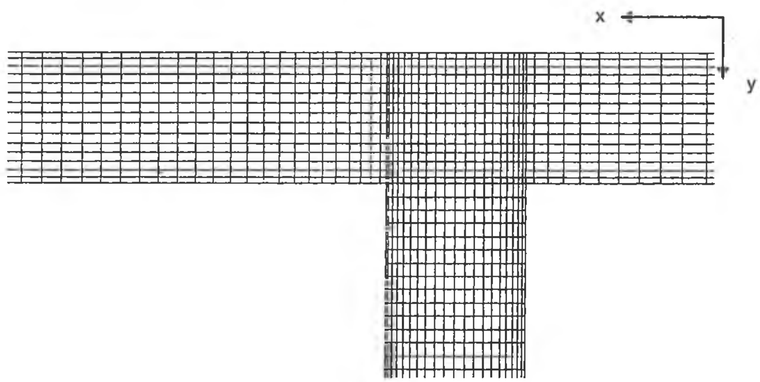


Figure 5. Computational Mesh – Detail near junction

3.3 Numerical Technique

Disregarding the slight temporal fluctuations (due to turbulence) in the actual flow parameters, a *steady-state* numerical simulation is carried out. Also, no 'overturning' of the free surface is expected or seen, so that in any given vertical (z direction) column of cells, only the lower cells will be occupied by water (Figure 6). This allows use of the Height of Liquid (HOL) technique, a built-in free-surface tracking algorithm in PHOENICS. The technique involves treating the flow of water (and the air above the water surface), as a single-phase incompressible flow. The free surface is located on the basis of fluid density (assumed as 998.0 kg/m^3 for water and 1.189 kg/m^3 for air, at 20° C), the total mass of water in any vertical column of cells, and that in all cells below any specific cell in the column. The equations of continuity and momentum, discretised according to the control-volume technique, are solved iteratively from an initially guessed flow field, until mass and momentum balances are achieved for each computational cell. Since the cells are contiguous, this automatically ensures mass and momentum balance for the entire computational domain.

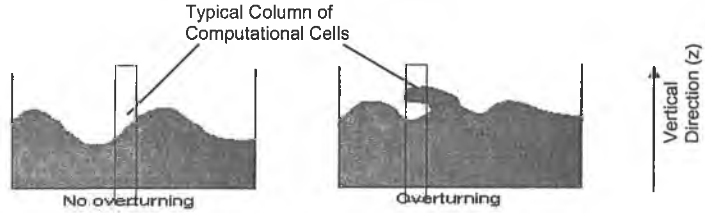


Figure 6. Nature of Free Surface

3.4 Boundary Conditions

Inlet velocities and inlet water heights for both upstream and the branch channels are determined solving the momentum equation in the main flow direction and assuming equal water depths in the branches upstream of the junction (equal-width channel confluence). Hence the main and branch channel inlet velocities and water depths are calculated as 0.139 m/s, 0.420 m/s and 0.33 m in both channels respectively and supplied as input data. At the solid walls (channel floors and sides), the no-slip boundary condition is applied. Zero gauge pressure boundary condition is applied at the vertical extremity of the computational domain in both channels, at vertical planes above the water inlets in both channels and at the outlet.

4. SIMULATION RESULTS

4.1 Water Surface

Figures 7 and 8 shows the computed water surface (surface with fluid density = 998.0 kg/m^3) in the vicinity of the junction. The sudden depression on the inner wall immediately downstream of the junction is clearly visible in figure 8 along with the variation in the recirculation zone.

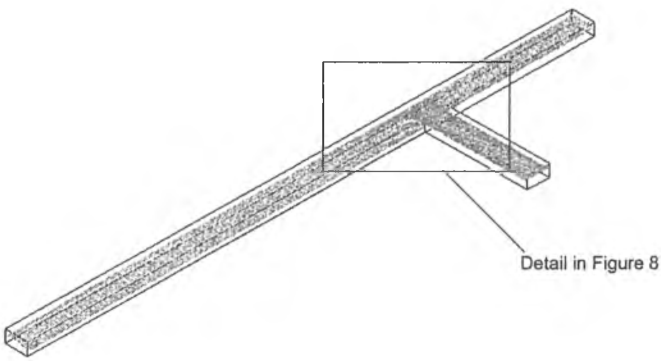


Figure 7.Water Surface – Entire flow field

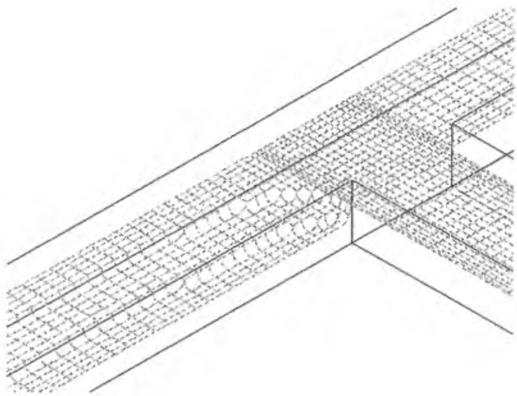
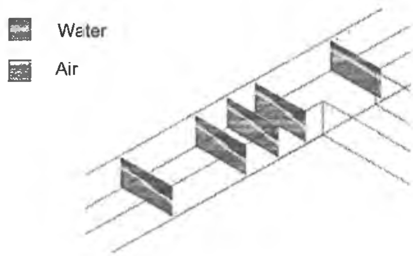
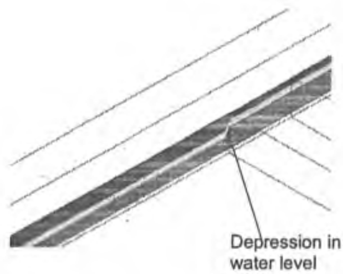


Figure 8.Water Surface – Detail near Junction

Figure 9 shows the water-air interface at various locations in the main channel near the junction. The effect of branch channel flow on the downstream of the channel junction is clearly visible from the density contours.



(a) At various locations in Main Channel



(b) Adjacent to Main Channel Inner Wall

Figure 9.Density Contours near Junction

Figure 10 shows the computed variation in water heights contours in the vicinity of the junction, and Figure 11 the corresponding experimental (non-dimensionalised) water depth contours (Weber et al, 2001). It can be seen that the overall water depth patterns show similar trends. In particular, the depression downstream of the junction across and along the channels is shown clearly.

Further comparisons between computed and experimental water depths (in actual dimensions) are shown in Figure 12. (In the experimental channel x* = 0 corresponds to upstream end of the junction and x* =-1, one channel width downstream.)

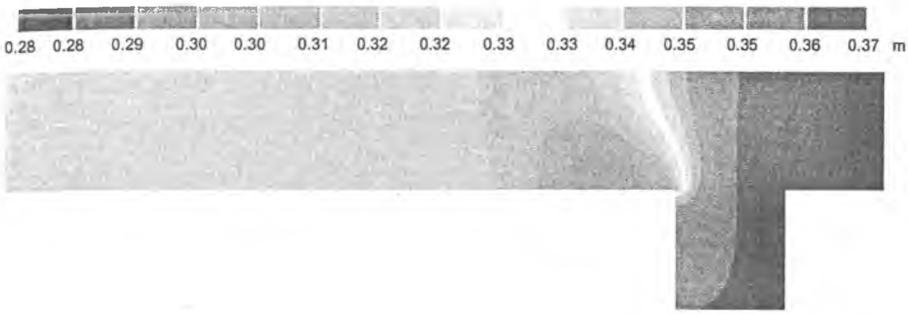


Figure 10. Computed Water Depth contours near Junction using PHOENICS

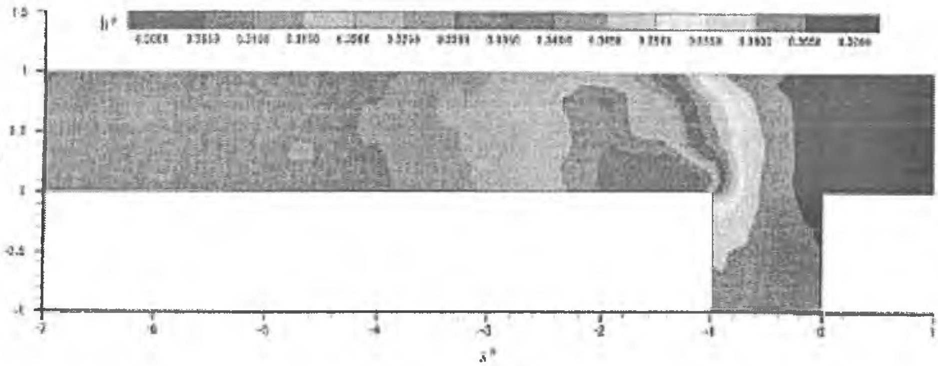


Figure 11. Experimental Water Depth Contours near Junction (Weber, et al, 2001)

It can be seen that the agreement between experiment and simulation is good at the upstream end of the junction. At further downstream locations, there is increasing discrepancy between experiment and simulation, although the shape of the free surface across the channel is accurately reproduced. Possible reasons for this include relative coarseness of the mesh in the vertical direction, leading to a 'smearing' in the density contours, and hence inaccuracy in free surface location. The coarse mesh may also result in inaccuracies in the computed flow parameters in the recirculation region. The turbulence intensity at the inlets was assumed to be 5%, which may not reflect the experimental situation. In addition, there is a slight mismatch between the exact locations of the experimental data collection points, and the midpoints of the computational cells. Ways of rectifying the above discrepancies are being investigated.

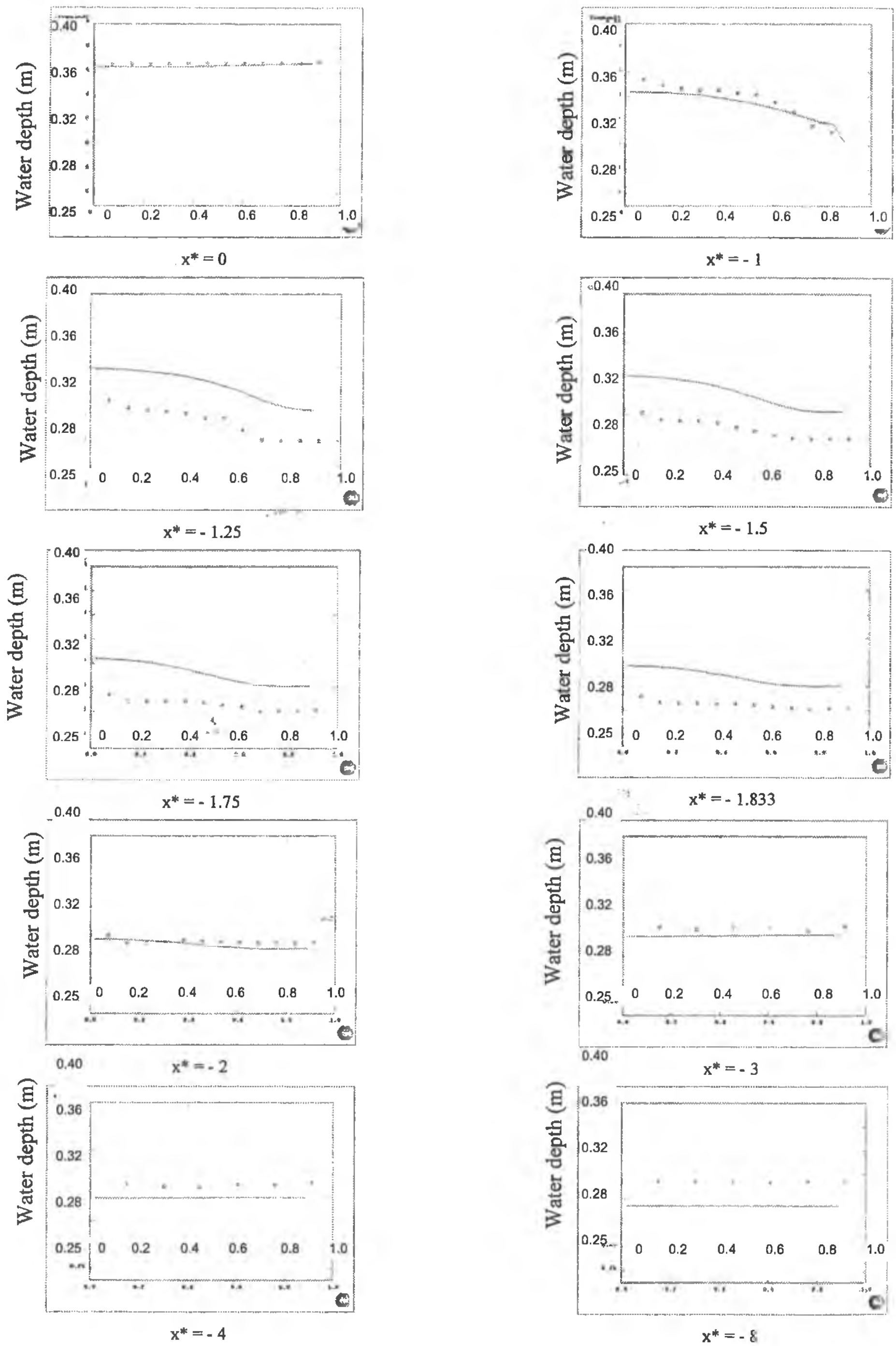


Figure 12. Water Depth Comparisons at Various Locations
(□ Experiment; — Simulation); $x^* = x/w$, where w = width of the channel and x = distance along the main channel in the experimental channel.

5. CONCLUSIONS

The first part of a 3D numerical simulation of a horizontal-bed open-channel water flow with a 90° equal-width junction is presented. A commercially available CFD package, PHOENICS (version 3.5) is used. The results of the numerical simulations are compared with the experimental data published by previous researchers (Weber et al, 2001). The free-surface model yields a fair comparison of the free-surface profiles around the confluence, showing a sudden depression immediately after the junction, followed by recovery. The extent of the recirculation zone, smaller near the bed and larger near the free surface, is also shown in the simulation. The relative coarseness of the computational mesh, especially in the vertical direction, combined with the hybrid differencing scheme, can lead to errors in the numerical simulation. This is particularly true in and near the recirculation zone. Further work with a finer mesh (15 cells) in the vertical direction is continuing. Demonstrating the use of the free-surface model, and its validation using available experimental data was essential, before the more challenging problem of sediment transport could be attempted. An experimental arrangement allowing introduction of sediment in the branch channel is being designed.

Further work is continuing, using a 'body-fitted' mesh conforming to the computed free water surface. It is hoped that this simulation will reveal more details of the flow, such as primary and secondary recirculation patterns.

ACKNOWLEDGEMENTS

One of the authors, K. Dissanayake, would like to gratefully acknowledge support from the University of Wollongong in the form of a post-graduate research scholarship and Sustainable Earth Research center (SERC). Dr Simon Beecham from the University of Technology, Sydney and Dr Jaya Kandasamy from the NSW Department of Infrastructure, Planning and Natural Resources provided valuable initial ideas for the project. The experimental data used for validation was obtained from IIHR, USA.

REFERENCES

Huang J., Webber L.J. and Lai Y.G. (2002) Three-dimensional numerical study of flows in open channel junctions, *J. Hydraulic Engineering*, 3, 268-279.

Olsen (1999) Computational Fluid Dynamics in Hydraulics and Sedimentation Engineering

Ramamurthy A.S., Tran D.M. and Carballada L.B. (1993) Increased hydraulic resistance in combining open channel flows, *Water Resources Engineering* 128, 6, 1505-1508.

Shumate E.D. and Webber L.J. (1998) Experimental Description of combining flows at open channel junction, Proceedings *of the ASCE Water Resources Engineering*, 1679-1684.

Taylor E. H., (1942) Flow characteristics at rectangular open channel Junctions, *Proceedings ASCE*, 893-902.

Weber L.J., Shumate E.D. and Mawer H. (2001) Experiments on flow at a 90° open channel junction, *J. Hydraulic Engineering*, May, 340-350.

Weerakoon S.B. and Tamai N. (1988) Three-dimensional calculation of flow in river confluences using boundary fitted co-ordinates. *J. Hydro science and Hydraulic Engineering*, 7, 51-62.

Weerakoon S.B. (1990) Flow structure and bed topography in river confluences. *PhD thesis, University of Tokyo*, Tokyo, Japan.

Weerakoon S.B., Kawahara Y., and Tamai. N. (1991) Three dimensional flow structure in channel confluence of rectangular section. *Proceedings XXIV Congress, International Association for Hydraulic Research, Part* A, 373-380.

Weerakoon S.B., Tamai N. and Kawahara Y. (2003) Depth-averaged flow computation at a river confluence, *Journal of Water and Maritime Engineering*, WM 156, 1, 73-83.