NUMERICAL MODELLING OF FLOW IN A 90° CHANNEL CONFLUENCE

Shafiqul Islam Shakil*[1], Md. Jahir Uddin[2] and Chandan Mondol[3]

[1]*Department of Civil Engineering, Khulna University of Engineering & Technology, Khulna, Bangladesh, e-mail: shakil021295@gmail.com*
[2]*Professor, Department of Civil Engineering, Khulna University of Engineering & Technology, Khulna, Bangladesh, e-mail: jahiruddin@ce.kuet.ac.bd*
[3]*Ph.D. Scholar, Department of Civil Engineering, Khulna University of Engineering & Technology, email: chandance2k7@gmail.com*

***Corresponding Author**

ABSTRACT

A confluence can be found where two streams join to form a river with a new name, or where two separate channels of a river join together at the downstream end, or at a point where a tributary meets a larger river. Open channel confluences are seen in many natural and man-made channels. There is a complex flow behavior in or around the junction. On the inner side immediately downstream of the junction, a separation zone is developed. The complex flow is a function of channel geometry, confluence angle, flow rate, boundary roughness and intensity of turbulence. A major influence on bed erosion and bank scouring is caused by channel confluence. A numerical simulation was performed in a 90-degree channel junction using River2D solver in iRIC to investigate the flow behavior. Velocity and surface elevation of water were computed for different flow ratios. From numerical simulation it was found that at upstream of the junction, water surface elevation was higher followed by a sudden drop of water levels immediately downstream of the junction. At outer bank velocity was higher and decreased towards the inner wall. The length and width of separation zone were decreased with increasing flow ratio. Numerical results of this investigation was compared with the experimental results of Kalyani Dissanayake (2009). The new data found from the current study can be used for the detailed analysis of flow dynamics. The application of this new knowledge can be used in improved design of river bank protection works and urban flood and erosion control structures.

Keywords: *Confluence, Water Surface Elevation, Separation zone, Flow ratio.*

1. INTRODUCTION

According to geography, a confluence is found where at least two streams of water join together to form a single water flow. A confluence can be found where two streams join for the formation of a river with a new name, or where two separate channels of a river join together at the downstream end. Confluences are studied in a variety of sciences. The flow pattern characteristics of bars, erosion and scour pools were studied in Hydrology (James L. Best,1986). The consequences and characteristics of water flow are often analyzed with mathematical models (Laurent Schindfessel et. al, 2015). The living organism distribution (i.e. ecology) depends on the channel confluence as well; "the general pattern (downstream of confluences) of increasing stream flow and decreasing slopes drives a corresponding shift in habitat characteristics (Beechie et. al., 2012). The overall aim of this research is to obtain a detailed description of flow behavior at a 90-degree channel confluence.

The specific objectives are to conduct a critical literature review of existing knowledge in open channel junction flows without sediment transport and identify the knowledge gaps in the research area, to perform Numerical simulations of junction flow, to compare experimental results with numerical simulations and to analyze the validity of the numerical simulations and predict the velocity profile. To achieve the above-mentioned objectives this study was carried out step by step. The literature reviews mainly focused on studies presented in journals, books and conference publications, to find out how previous studies in junction flows were conducted, and what conclusions were drawn from their results and experiments. This information helped to identify the knowledge gaps in junction flows and to set out the objectives for current study. Experimental result was taken from the experiments conducted by Kalyani Dissanayake, 2009. iRIC (International River Interface Co-operative) was used to simulate 90° open channel junction flow behavior with clean water. A steady-state two-dimensional numerical simulation was carried out. The water surface elevation and velocity are subsequently computed for different flow. Result of the numerical simulation is compared with the experimental result to analyze the validity.

Previous studies of channel junction flows were mainly based on laboratory scale model experiments with simplified flow conditions in which variables were controlled rather easily. Most laboratory experiments used fixed- bed channels and the key characteristics investigated were depth ratio, separation zone, effect of discordant beds, and location of shear layers, stagnation point and secondary flow current. In the most common and traditional approach, confluence characteristics have been determined in prismatic channels in laboratories (Taylor, 1944). However, subsequent predictions were based on several assumptions related to idealistic flow conditions using simplified channel geometries and small-scale physical models. Taylor studied the flow characteristics at a junction of two horizontal channels with rectangular cross sections. He applied the momentum equation to analyze the combined flow and verified the predictions with experimental data for junction angles of 45° and 135°. However, his theoretical predictions were applicable only for smaller junction angles. There was no acceptable agreement for large junction angle such as 135°. It was believed that this was due to the velocity distribution downstream of the junction that was distorted and the flow did not remain parallel to the channel walls. In Taylor's study (1944) he recognized that failure to measure the pressure on the walls of the branch channel or failure in the estimation of the momentum transfer from branch to main channel constitutes an important shortcoming.

Weber, et al. (2001) conducted experiments in an equal width, equal depth, and flatbed 90° laboratory open channel junction with rectangular geometry for different flow conditions. Velocities and water depth were measured in the vicinity of the junction using the Acoustic Doppler Velocimetry (ADV) technique and a point gauge respectively. A data set was compiled which fully describes the complex three-dimensional flow conditions present in an open channel junction for the selected flow conditions.

Compared to laboratory experiments, there is a shortage of data available from field studies on natural confluences. Field investigations are largely based on point measurements of the velocity field. Mamedov (1989) conducted field investigations measuring velocity field and sediment concentration of the flow. He identified major characteristic zones such as separation zone and stagnation zone in the Kura River in Russia. Kenworthy and Rhoads, (1995) found that patterns of normalized sediment concentrations at a cross-section near the exit of the confluence are a function of the ratios of momentum flux and mean sediment concentration in the upstream channels. These patterns reflected a shift in the location of the shear layer toward the outer bank with increase in the momentum ratio. However, the data collected in this study consists of only depth-integrated sediment samples and measurements of bulk upstream hydraulic variables. Therefore, a rigorous analysis of the mixing process in terms of flow mechanics was not possible.

In numerical simulation, partial differential equations expressing the governing physical laws are solved incorporating the constitutive models based on numerical methods. It is a technique that allows the alteration of one variable at a time, so assessment of the relative importance and interaction of different controls becomes possible.Duan, J.G. and Nanda, S.K. (2006) used a two-dimensional depth-averaged hydrodynamic model to simulate suspended sediment concentration distribution in the Groyne-River. The governing equations were depth-averaged two-dimensional Reynolds' averaged momentum equations and the continuity equation in which the density of sediment laden-flow varied with the concentration of suspended sediment.

2. METHODOLOGY

2.1 Experimental Setup

An open channel junction was designed and constructed at the Hydraulics Laboratory of the University of Wollongong as part of a PhD project by K. Dissanayake (2009). The size of the experimental facility was adjusted to suit the constraints on the available space. The experimental facility consists of a 90° junction of two equal-width, equal-depth flatbed channels with two separate water recirculation systems, water height measuring system and a data acquisition system using the LabVIEW program, along with other experimental control devices.

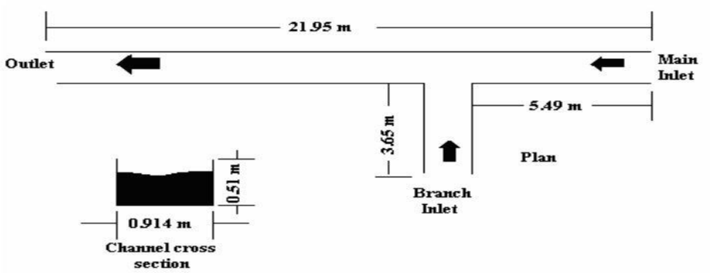


Figure 1: Experimental set up (K. Dissanayake, 2009)

The experimental setup was designed and constructed according to the dimensions given in Figure 1. The flume was fabricated using 6 mm thick Perspex sheets. It has a low Manning's value, around 0.009. The sharp corners at the junction were not rounded off. Clean water flow was established in the main channel.

2.2 Calibration

Dissanayake (2009) performed laboratory experiments in a 90° combining flow flume. The experimental facility was capable of establishing different flow conditions. The varying discharge on both channels (main and branch) were supplied by header tanks. Perforated plates and 100 mm thick honeycomb were placed at the main and branch channel inlets in order to reduce eddy generated at inlets. The transition pieces of channel were made smooth from vertical to horizontal while the entire floor of the facility was kept horizontal on bends in order to minimize losses. The length of main channel and branch channel were 21.95m and 3.66m respectively. The junction was 5.49m downstream of the flume entrance. The width of branch channel, main and the downstream combined flow channel was 0.914 m and the depth were 0.51m. The total combined flow was 0.170 m^3/s, and the tail-water depth was 0.296m and both were held constant, yielding a constant downstream Froude number (0.37), and a constant average velocity (0.628 m/s) of tail-water.

The above-mentioned experimental condition is simulated by solver River2D of iRIC. In this numerical solution, Bousinessq type eddy viscosity is used for the transverse shear modeling. Secondary flows are not considered during calculation, which is the main reason for losing energy of flow near the channel boundary. Total numbers of grids are 2119. In the present study, concept of geometric similarity is applied for better result. During simulation by River2D, very shallow depth of flow as compared with (Dissanayake, 2009) brings erroneous result. The condition of geometric similarity is expressed in Figure 2.

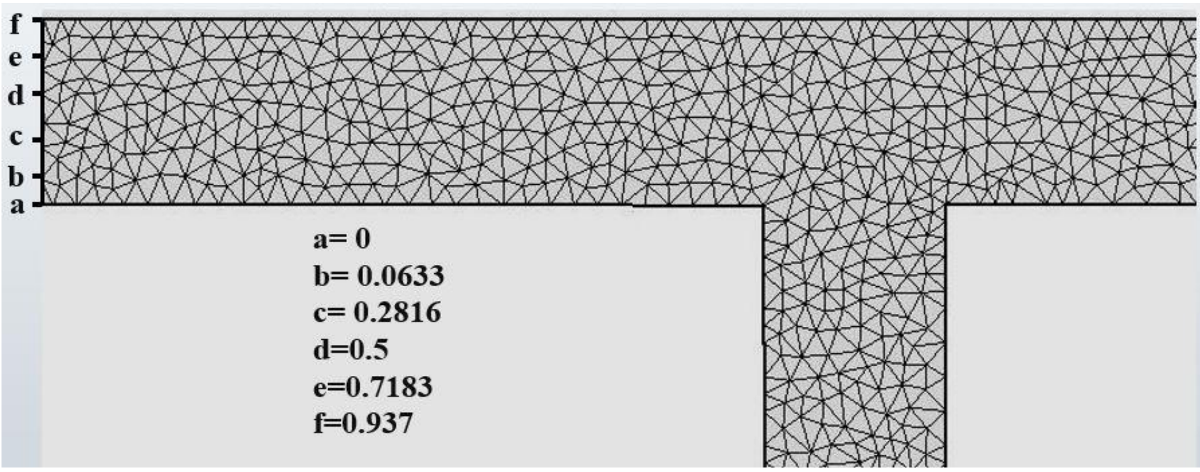


Figure 2: Computational Mesh for River2D Solver in iRIC

Discharge ratio was determined by $q^*= Q_{main}/Q_{total}$. Water height was normalized by $h^*=h/w$. Distance along x-direction was normalized by $x^*=x/w$. Distance along y-direction was normalized by $y^*=y/w$. Here, “w” is the channel width.

3. RESULTS AND DISCUSSIONS

Water heights were measured at set locations using five-point gauges which were mounted on a simple sliding base. For each of two different flow conditions, the sliding base was moved along the channels recording the water heights at five different locations across the channels simultaneously. Figure 3 shows the Normalized water depth (h*) contours in the main channel.

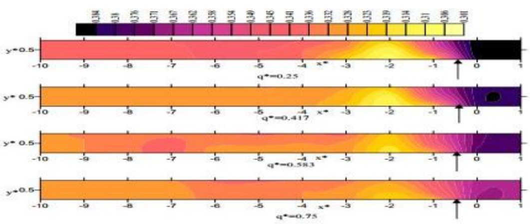


Figure 3: Normalized water depth (h*) contours in the main channel

Shown above water depth contour maps and water surface profiles in the main channel (Figure 3) it was observed that higher water depths were generated before the junction. The free surface profile along the main channel showed a sudden depression immediately after the junction, followed by recovery after about 7-8 channel widths from junction. Upstream to downstream depth deference was higher for lower discharge ratios and highest difference was observed for q*=0.25 flow condition. Super elevation exists adjacent to outer bank after the junction for all flow conditions. This water depths pattern is generated due to the obstruction effect caused by the lateral stream associated with turbulence mixing and energy losses at the junction. Measured water depths for flow conditions q*=0.25 and q*=0.75 were then compared with iRIC simulation and found that they are in good agreement showing the similar pattern of free surface profile changes at the junction.

3.1 Validity Analysis & Velocity Prediction

Two-dimensional numerical modelling was carried out using River2D for 90° open channel junction to simulate flow conditions q*=0.25 and q*=0.75. Results of the numerical simulations were compared with experimental data from previous researchers (Dissanayake, 2009). Higher water depths were generated before the junction. The free surface profile along the main channel showed a sudden depression immediately after the junction, followed by recovery after about 7-8 channel widths from junction. Upstream to downstream depth deference was higher for lower discharge ratios and highest difference was observed for q*=0.25 flow condition. There is a drop-in surface elevation at the right corner of the junction. This zone is called the Stagnation zone. The relative velocity of water in this zone is zero. Figure 4 shows the color contour of water surface profile. The experimental result (Dissanayake, 2009) was compared with the result from River2D and both results show a very little variation (Figure 5 to 14).
(q*=0.25) (q*=0.75)

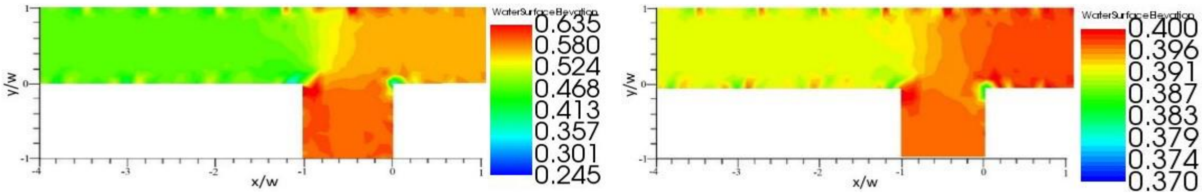
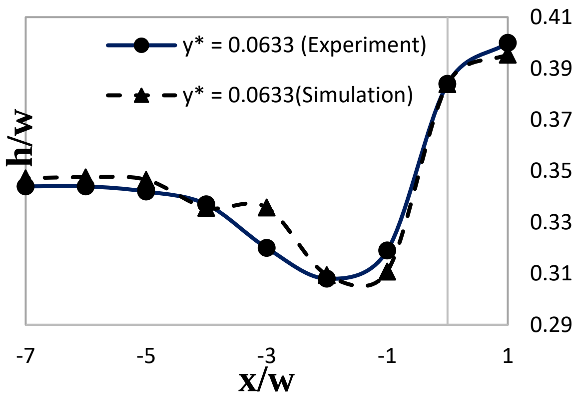


Figure 4: Water Surface Profile along the junction

It can be seen that the overall water depth patterns show very similar trends in both simulation and experiment. The depression of water at the junction downstream along and across the channels is shown clearly. The figures show further comparisons between experimental and computed dimensional water depths. At the upstream end, the agreement between experiment and simulation is very good. At further downstream locations, there is an increasing discrepancy between experimental observation and simulation within the high turbulence zone, although the free surface shape is accurately reproduced. A possible reason for this is a slight mismatch between the exact locations of the experimental data collection points, and the mid-points of the computational cells where the data is stored after calculations.



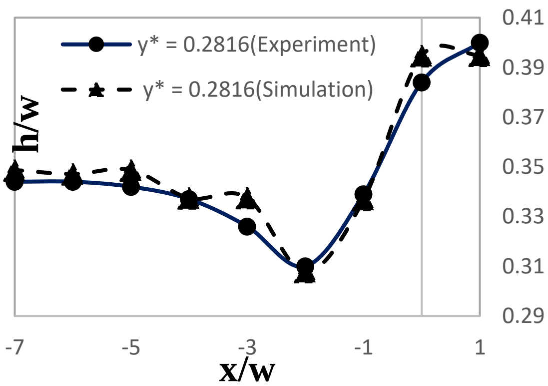


Figure 5: Water Surface Profile (q*=0.25, y*=0.0633) Figure 6: Water Surface Profile (q*=0.25, y*=0.2816)

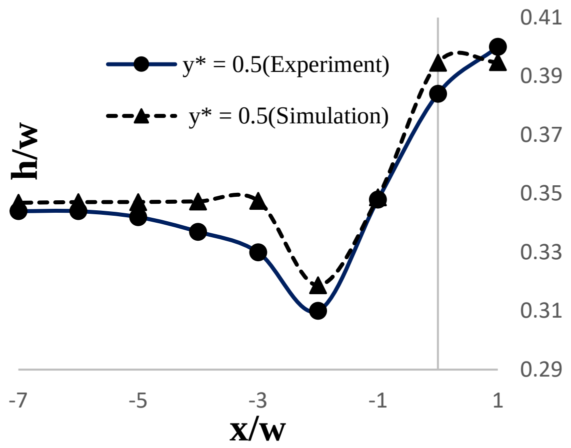


Figure 7: Water Surface Profile (q*=0.25, y*=0.5)

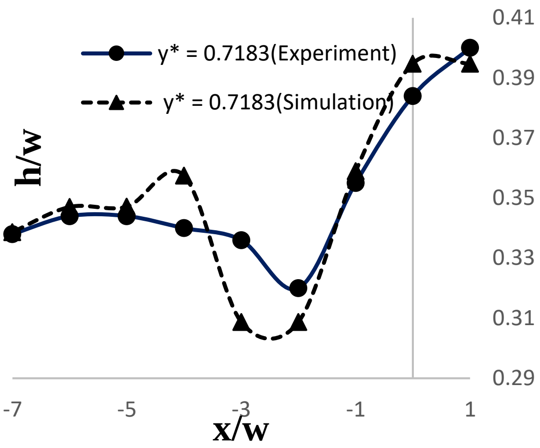


Figure 8: Water Surface Profile (q*=0.25, y*=0.7183)

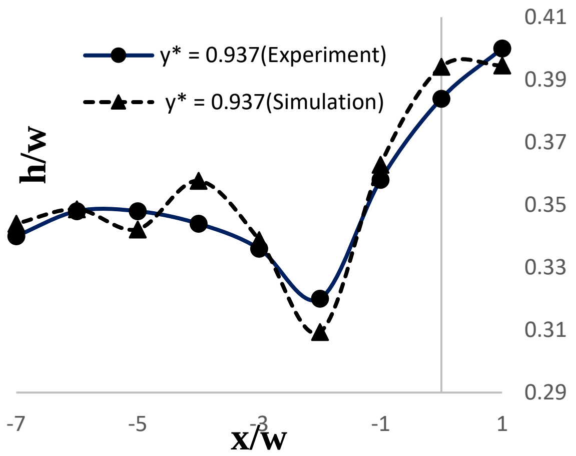


Figure 9: Water Surface Profile (q*=0.25, y*=0.937)

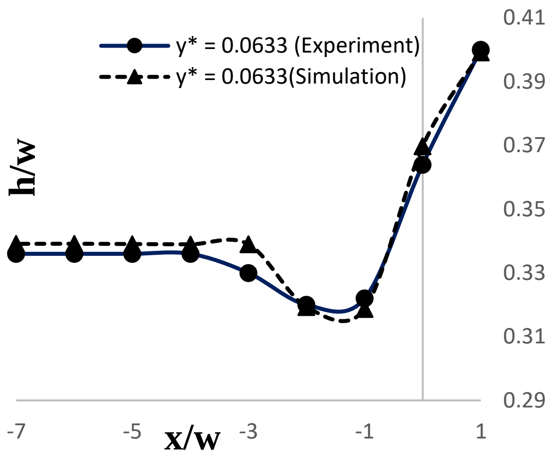


Figure 10: Water Surface Profile (q*=0.75, y*=0.0633)

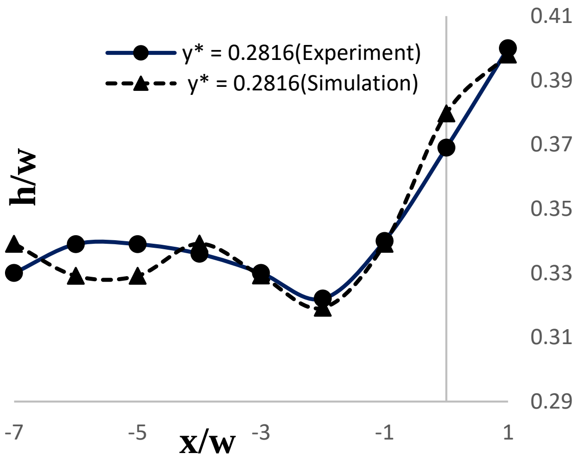


Figure 11: Water Surface Profile (q*=0.75, y*=0.2816)

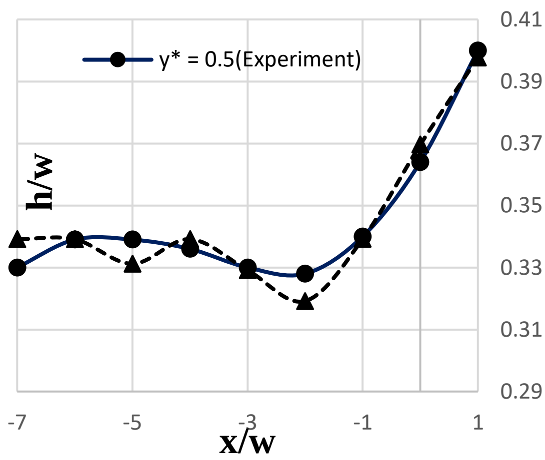
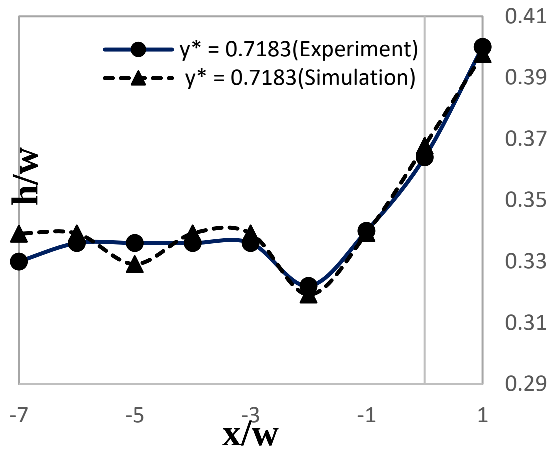


Figure 12: Water Surface Profile (q*=0.75, y*=0.5)



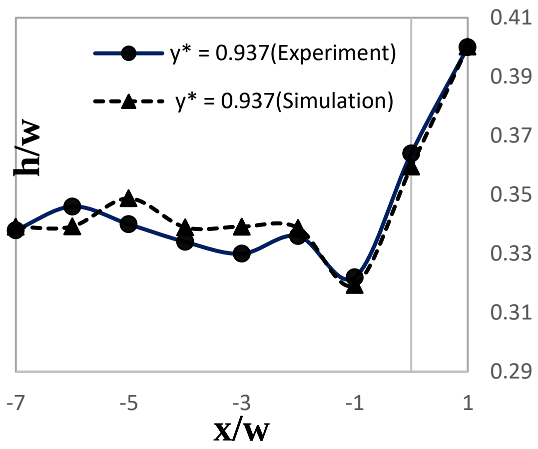
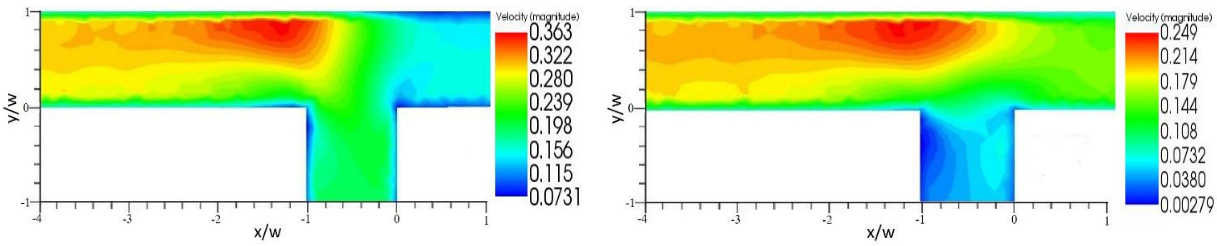


Figure 13: Water Surface Profile (q*=0.75, y*=0.7183)   Figure 14: Water Surface Profile (q*=0.75, y*=0.937)

The maximum depth difference between upstream and downstream in the simulation is 0.08 m (73mm) whereas the maximum depth difference in the experiment is 0.07 m (64 mm). Therefore, the simulation results show 14% discrepancy in water heights. In reality it is virtually impossible to simulate exactly the actual flow conditions which exist in real situations. Furthermore, flow through open channel junctions is inherently three dimensional and unsteady. Therefore, appropriate assumptions were made to simplify the problem which results in discrepancy between predictions and experimental observations.



q*=0.25   q*=0.75

Figure 15: Color Contour of velocity distribution

The maximum velocity is distant from the inner bank of the main channel. The flow remaining in the main channel develops within the section after passing the intake entrance, but because of the effects of curved flow lines at the intake entrance, the maximum velocity will be deviated back towards the inner wall. Figure 15 shows the color contour of velocity distribution along the junction for different flow ratios.

The major cause of velocity loss near the outer boundary is the frictional resistance between the channel boundary and the flowing water. There is also some source of errors. The positions of the cross sections were not accurate. So, the result slightly deviates from the experimental one. There is a drop of velocity in the right corner of the branch channel at the junction. The velocity was nearly zero there. This zone is called the Stagnation zone. The highest static pressure is found at zero velocity and hence the maximum static pressure is at the stagnation points. This static pressure is known as the stagnation pressure. Flow separation occurs at the left corner of the branch channel at the junction where the boundary layer travels far enough against an adverse pressure gradient that the speed of the boundary layer relative to the object falls almost to zero. The fluid flow becomes detached from the surface of the object, and instead takes the forms of eddies and vortices. In this study, there is no significant result regarding the separation zone as the secondary flow was not considered. There were limitations on total number of elements in the computational domain based on the software type.

Therefore, there were limitations for further mesh refinements of the computational mesh. The quality of the mesh could effect on the accuracy of predictions. Dense cell population in areas of higher flow parameter changes were enabled to simulate accurate flow fields. The current study does not consider the effect of secondary current. The behavior of separation zone depends on the secondary current. Sediment transport was not taken into account in this present study. In natural river confluences sediment transportation is a major factor. The current study was based on a 90° open channel junction. Though this confluence angle produces maximum obstruction to the main channel flow among most channel configurations in nature are of small confluence angles such as 30º, 45º and 60º. Flow fields in such channel junctions are different. (Obtuse confluence angles also exist in nature. Most commonly those sites have hard rocks. Increased velocities will not be a crucial issue for such sites). Therefore, it is recommended to conduct further studies on such channel configurations investigating flow and sediment characteristics.

4. CONCLUSIONS

Considerable research on open-channel flows has been undertaken in the past and yet the description of flow behavior at channel junctions is incomplete. Therefore, the present study was aimed at further investigation of junction flow behavior without sediment transport through comprehensive numerical techniques. The current study provides new data, particularly on the geometry of the recirculation region immediately downstream of the junction, contributing to a better understanding of flow dynamics at open channel junctions.The studies conducted in this research provide greater insight towards the understanding of the flow and sediment transport characteristics. It is revealed that flow dynamics at open channel junctions may have important effects on the dispersal of dissolved or suspended substances in headwater areas of channel networks. Data obtained in this study is useful in controlling sediment erosion and deposition processors and flooding at channel junctions. The simulation can be perfected with the consideration of the secondary flow. Thus, a detailed investigation can be conducted on the separation zone characteristics. Sediment laden flows can also be investigated for a better understanding of the flow and sediment transport behavior on the open channel confluences.

REFERENCES

Beechie et al. (2012), who cite earlier work. Tim Beechie, John S. Richardson, Angela M. Gurnell, and Junjiro Negishi (2012) "Watershed processes, human impacts, and process-based restoration." In Philip Roni and Tim Beechie (eds.) (2012) Stream and Watershed Restoration: A Guide to Restoring Riverine Processes and Habitats, John Wiley & Sons

Best, J. L., (1986) "The morphology of river channel confluences",*Progress in Physical Geography 10:157–174*

Biron, P., Best, J. L., & Roy, A. G. (1996). "Effects of bed discordance on flow dynamics at open channel confluences",*Journal of Hydraulic Engineering, 122(12), 676-682*.

Dissanayake, K. (2009)"Experimental and numerical modelling of flow and sediment characteristics in open channel junctions", Doctorof Philosophy thesis, School of Civil Mining and Environmental Engineering, University of Wollongong, 2009. *https://ro.uow.edu.au/theses/3555*

Gurram, S. K., Karki, K. S., & Hager, W. H. (1997). "Subcritical junction flow", *Journal of Hydraulic Engineering, 123(5), 447-455.*

Laurent Schindfessel, Stéphan Creëlle and Tom De Mulder (2015) "Flow patterns in an open channel confluence with increasingly dominant tributary inflow",*Water 7: 4724–4751*.

Taylor, E. H. (1944). "Flow characteristics at rectangular open-channel junctions",*Transactions of the American Society of Civil Engineers, 109(1), 893-902.*

Weber L. J., Schumate E. D., and Nicola Mawer (2001). "Experiments on flow at a 90 0 open channel junction." *Journal of Hydraulic Engineering, 127 (5).*