HENRY
Hydraulic Engineering Repository

Ein Service der Bundesanstalt für Wasserbau

Conference Paper, Published Version

Jia, Yafei; Wei, Zhangping; Wang, Sam S. Y.; Blanckaert, K.; Ribeiro, M. L.

A Study of Flow Characteristics Near a Channel Confluence Using CCHE 2D/3D Models

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
Kuratorium für Forschung im Küsteningenieurwesen (KFKI)

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/109603

Vorgeschlagene Zitierweise/Suggested citation:
Jia, Yafei; Wei, Zhangping; Wang, Sam S. Y.; Blanckaert, K.; Ribeiro, M. L. (2012): A Study of Flow Characteristics Near a Channel Confluence Using CCHE 2D/3D Models. In: Hagen, S.; Chopra, M.; Madani, K.; Medeiros, S.; Wang, D. (Hg.): ICHE 2012. Proceedings of the 10th International Conference on Hydroscience & Engineering, November 4-8, 2012, Orlando, USA.

Standardnutzungsbedingungen/Terms of Use:

Die Dokumente in HENRY stehen unter der Creative Commons Lizenz CC BY 4.0, sofern keine abweichenden Nutzungsbedingungen getroffen wurden. Damit ist sowohl die kommerzielle Nutzung als auch das Teilen, die Weiterbearbeitung und Speicherung erlaubt. Das Verwenden und das Bearbeiten stehen unter der Bedingung der Namensnennung. Im Einzelfall kann eine restriktivere Lizenz gelten; dann gelten abweichend von den obigen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Documents in HENRY are made available under the Creative Commons License CC BY 4.0, if no other license is applicable. Under CC BY 4.0 commercial use and sharing, remixing, transforming, and building upon the material of the work is permitted. In some cases a different, more restrictive license may apply; if applicable the terms of the restrictive license will be binding.

Proceedings of the 10th Intl. Conf.on Hydroscience & Engineering, Nov. 4-7, 2012, Orlando, Florida, U.S.A.

A STUDY OF FLOW CHARACTERISTICS NEAR A CHANNEL CONFLUENCE USING CCHE2D/3D MODELS

Yafei Jia[1], Zhangping Wei[2], Sam, S.Y. Wang[3], K. Blanckaert, and M. L. Ribeiro[4]

ABSTRACT

Channel confluences connect channels in fluvial networks. Because the flow and sediment with significant different conditions are mixed at the confluence, abrupt bed morphology change often occurs in responds to the highly turbulent and dynamic conditions. The bed change will in turn vary the general flow pattern and cause subsequent lateral morphologic adjustment near the confluence. Research on this subject is of practical importance because sediment transport and morphologic change will affect channel stability and water quality.

In this paper, numerical simulations of the turbulent flows near a river confluence are presented. Both CCHE2D and CCHE3D model were applied and the simulations are based on data from a physical experiment. The 2D model provided a quick and adequate solution and a good initial condition for the 3D simulations. The 3D model predicted the flow stagnation, recirculation and the two-layer turbulent flow structure in the immediate vicinity of the tributary confluence very well. The simulated velocity fields of both models have good agreements when comparing to the measured longitudinal velocity vector field.

1. INTRODUCTION

Natural rivers are mostly dendritic or trellis channel networks formed in their watersheds. The lower order branch channels merge into higher order and larger stem channels with more water and sediment. The confluence where a branch channel merges with a larger one (main channel), often has pronounced characteristics of the hydrodynamics, sediment transport and water quality conditions, because the water flows and sediment sources in the branch channel and those in the main channel often differ significantly. Geomorphic response typically occurs near the river confluence due to these dynamic change, as well and sediment deposition or bank erosion (Benda et al. 2004, Unde and Dhakal, 2009). Because the waters of the two channels are from totally different source areas, the water quality of the two channels will have a large adjustment at the confluence as well. Because of its unique characteristics, channel confluence has been extensively studied in the past.

---

[1] Research Professor, Associate Director, National Center for Computational Hydroscience and Engineering, The University of Mississippi, Oxford, MS 38677. (Jia@ncche.olemiss.edu)

[2] Ph.D Student, same address.(zpwei@ncche.olemiss.edu)

[3] F.A.P. Barnard Distinguished Professor, Emeritus Director and Founder of NCCHE.(wang@ncche.olemiss.edu)

[4] Stucky SA. Renens, Switzerland

Near a river confluence, the magnitude and distribution of the flows in the two channels are significantly different; the sudden mix of the flows stimulates a strong momentum exchange and strong turbulence. Due to the interaction of two different flows, a stagnation point and recirculation zone will be formed near the confluence (Fig. 1). Sediments will normally deposit in the recirculation zone where the transport capacity is low, therefore, forming a sedimentation bar which further affects the overall flow pattern. In the main channel, a shear layer is formed which dominates the flow mixing and the formation of the sediment bar in the recirculation zone.

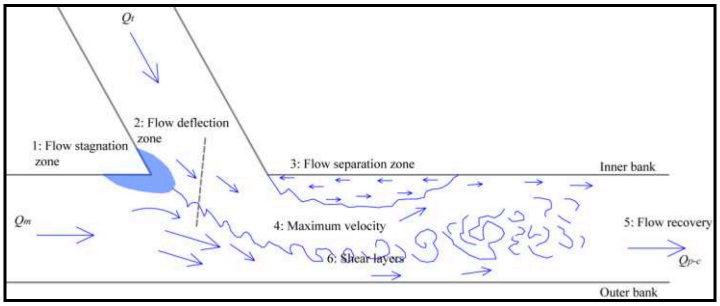

Figure 1 Illustration of flow pattern near a river confluence (copied from Leite Ribeiro et al. 2011, modified from Best 1987)

Demuren and Rodi (1983) studied the flow discharge mixing in the main channel of a confluence using a 3D steady state turbulent flow model with k-ε turbulence closure. The free surface was treated as a frictionless rigid-lid symmetry plane. Bonakdari, et al. (2011) studied the confluence side discharge mixing using a commercial3D CFD code with the free surface being approximated by the Volume of Fluid (VOF) method. The Reynolds stress turbulence model was used for computing experiment cases with side discharges angled 30 degree to the main channel flow. A quite comprehensive study of fluvial processes at river confluences has been conducted by Best (1987) using experimental flumes and field data. The fluvial processes at a confluence were found to be dominated by the angle of the junction and the ratio of flow discharges. Jia et al. (2002a) simulated the confluence flow distribution in the main channel using a finite volume model. It was found that the predicted size and shape of the recirculation downstream of the tributary channel are sensitive to the accuracy of the advection scheme. Best results were obtained with a second order, bounded monotonic scheme. Baranya, S., (2010) simulated the flow at the confluence of a curved natural river channel. Both approaches of Reynolds equations with k-ε closure and large eddy simulation were conducted and the computed results were compared with observed ADCP data. Roca, et al. (2009) studied the flow at a real world river confluence using the FESWMS-2DH numerical model developed by the US Federal Highway Administration (FHWA, 2002). The influence of several discharge ratios from the main and the branch channel to the free surface at the highly irregular confluence were studied using the numerical model. Leite Ribeiro et al. (2010) and Leite Ribeiro (2011) conducted experiments on the morphological response at a confluence due to flow mixing and sedimentation process. The width of the tributary channel was widened near the main channel to explore the ecological effect of morphologic adjustment. Sediments with a wide range of size distribution were fed into the side channel and were transported in the main channel and formed a large point bar due to sedimentation. In a more recent publication (Leite Ribeiro et al. 2012), another confluence experiment with sediment from a uniform side branch channel (no width expansion) was presented. In both cases the point bar was developed to the equilibrium stage. Because the tributary channel has a much higher slope than the main channel and carries a high sediment load, a point bars was formed immediately downstream of the confluence.

Open channel flow and sediment transport can be studied using flume experiment, field experiment and numerical simulation. Numerical simulation is an effective and efficient method but the numerical models have to be fully verified and validated before applying to a field investigation. In this paper, CCHE2D/3D computational models have been validated using the turbulent flow measured at a schematized experimental channel confluence which is characterized by a narrow and steep tributary and a wider and deeper main channel. The three-dimensional velocity and water surface data (Leite Ribeiro et al. 2012) were kindly provided for a flume experiment in a straight rectangle main channel connected with a straight rectangle side discharge channel angled at 90 degrees. A 3D free surface turbulent flow model, CCHE3D, with k-ε turbulence closure scheme and dynamic pressure, has been applied for the numerical study. CCHE2D model was also used to simulate the channel flow and the results were used for initial condition of the 3D simulation. The simulations in the study focused only on the turbulent flow measured after the sediment transport process reached equilibrium. Sediment transport and bed changes were also observed and measured in the experiments; modeling of these processes will be reported in the near future.

2. CCHE2D/3D MODEL

CCHE2D/3D are finite element based models using the colocation approach. Governing equations of turbulent flows are discretized using the Efficient Element Method on a quadrilateral structured mesh. Unsteady three dimensional Reynolds stress equations, free surface kinematic equation are solved for CCHE3D with options of hydrostatic and dynamic pressure and several turbulence closure schemes. The 3D governing equations are as follows:

$$u_{i,t} + u_j u_{i,j} - (-\overline{u_i u_j})_{,j} + \frac{p_i}{\rho} + f_i = 0 \tag{1}$$

$$u_{i,j} = 0 \tag{2}$$

the free surface is computed using the free surface kinematic equation:

$$\frac{\partial \eta}{\partial t} + u_\eta \frac{\partial \eta}{\partial x} + v_\eta \frac{\partial \eta}{\partial y} - w_\eta = 0 \tag{3}$$

and the *k*-*ε* model

$$k_{,t} + u_j k_{,j} - (\frac{\nu_k}{\sigma_k} k_{,j})_{,j} = P - \varepsilon \tag{4}$$

$$\varepsilon_{,t} + u_j \varepsilon_{,j} - (\frac{\nu_k}{\sigma_\varepsilon} \varepsilon_{,j})_{,j} = c_{\varepsilon 1} P \frac{\varepsilon}{k} - c_{\varepsilon 2} \frac{\varepsilon}{k^2} \tag{5}$$

where

$$P = \nu_t (u_{,j} + u_{,i}) u_{,i} \tag{6}$$

$$\nu_t = c_\mu \frac{k^2}{\varepsilon} \tag{7}$$

and standard coefficients: c_μ=0.09, σ_k=1.0, σ_ε=1.3, $c_{\varepsilon 1}$=1.44, $c_{\varepsilon 2}$=1.92, were applied.

Dynamic pressure is an important characteristic for highly turbulent three-dimensional flows. In this study case, the hydrostatic pressure assumption is no longer valid in the near field of the confluence. The dynamic pressure is computed by using the velocity correction method. A Poisson's equation formulated with the velocity correction method and the continuity equation is solved on a staggered grid to obtain dynamic pressure and forcing the simulated flow to satisfy the divergence free condition. The system of equations is solved implicitly by using the SIP method with the first order Euler's scheme. Wall boundary conditions have been used for the momentum equations of the k-ε model. The depth averaged version of these equations is solved for the CCHE2D model (Jia et al. 2002).

3. EXPERIMENT

The confluence experiment was conducted in a flume with a main channel (8.5m long and 0.5m wide) and a perpendicular tributary channel (4.9m long and 0.15m wide). The tributary channel intersects with the main channel at 3.6m from the inlet; both channels were of a rectangle cross-section. The initial bed elevation of the tributary channel is sharply higher that the main channel at the junction.

The initial slope of the tributary channel was 0.005 which was insufficient to transport the fed sediments. Sever sedimentation occurred which adjusted the channel to equilibrium slope (~ 0.019). The initial slope of the main channel bed was set to be zero. Because of the high slope the flow in the tributary is supercritical (*Fr*=1.3); the flow in the main channel is subcritical in general but it becomes supercritical near the right bank after a large sediment bar has been developed and the water flow is pushed to one side of the channel. The flow discharges from the tributary and the main channel inlets are 0.002 and 0.018m^3/s, respectively. More detailed information for the experiment facility, measurement techniques and results can be found in Leite Ribeiro, et al. (2012).

The velocity and water surface elevation of the flow were measured after the sediment transport process has reached to equilibrium. Since this paper studies only the flow field over the equilibrium bed rather than the sediment transport and sedimentation process, the map of the final bed topography of the channel was used to generate the computational mesh. The channel banks are considered as smooth and the beds were rough considering the sediment deposited bed and the main channel was covered with poorly sorted sediment mixture. d_{90} of the sediment mixture was used to specify the bed roughness for the CCHE3D model.

Figure 2 shows the configuration and dimension of the experiment flume; measured water surface, equilibrium bed topography along several longitudinal profiles in the main channel and one profile along the tributary channel. The equilibrium bed topography was formed by sediment deposition with sediment fed from the tributary channel. This paper focuses only on hydrodynamics, sediment transport process was not considered.

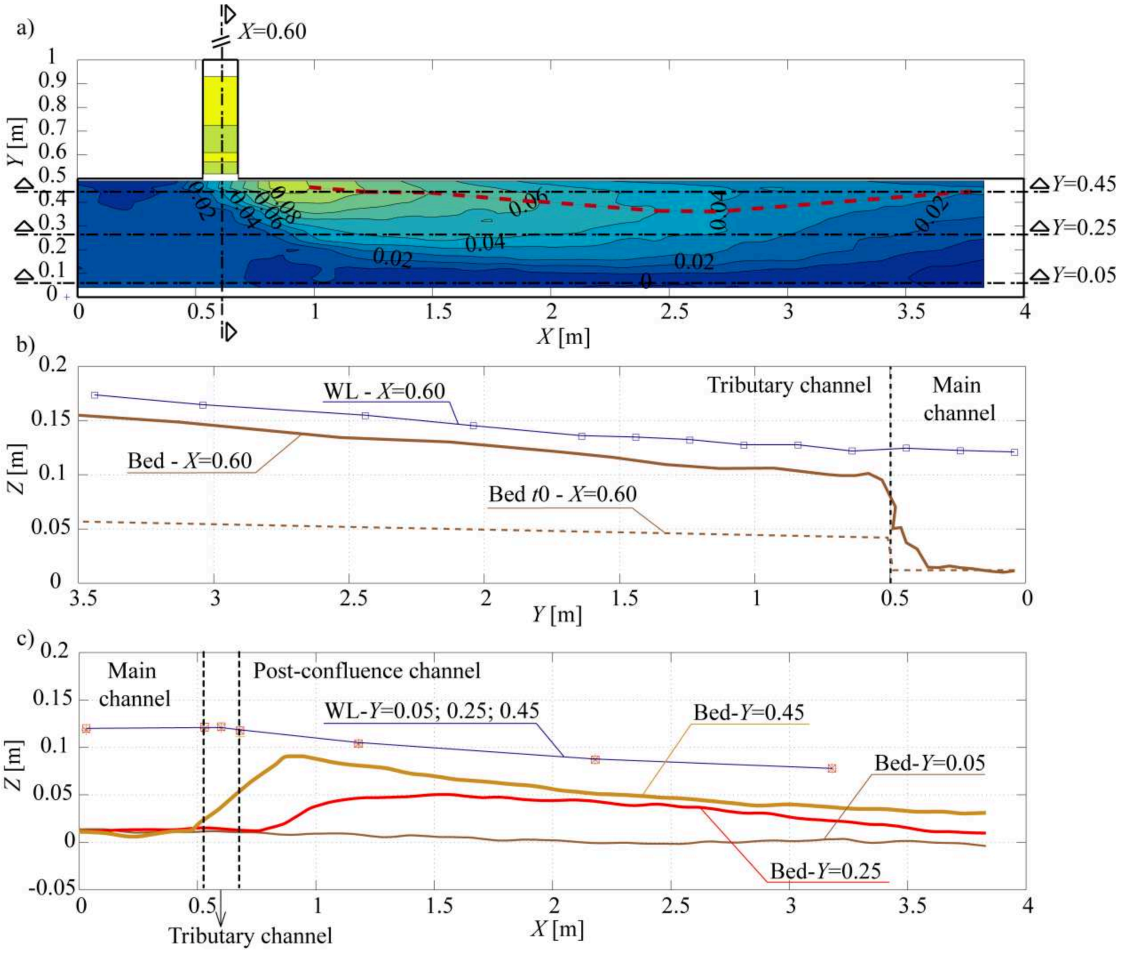

Figure 2 (a) Equilibrium bed topography in the main and tributary channel. (b) Longitudinal profiles of the water surface elevation, the equilibrium bed elevation and initial bed elevation (dashed line) at X = 0.60 m (axis of the tributary) and (c) Y = 0.05 m (near the outer bank), Y = 0.25 m (axis of the channel) and Y = 0.45 m (near the inner bank). (copied from Leite Ribeiro et al. 2012)

4. NUMERICAL SIMULATION

In this paper, the numerical simulations have been conducted for the aforementioned river confluence physical experiment. The simulation results were compared with the measured data and presented in the following sections. Both of the CCHE2D and CCHE3D model were applied. To save computation time, the 2D model was also used as a calibration tool and its results were initial conditions for the 3D model.

A mesh with 146 sections in main channel, 156 sections in transverse direction and 16 levels in the vertical was used for the simulation. In those for the transverse direction 59 sections are in the main channel, the rest are in the tributary. Mesh density around the confluence is higher for better resolution. The horizontal mesh was used also for the 2D simulation which is used as an initial condition for the 3D simulation. Bed elevation formed by the deposited sediment was measured and was used for generate the mesh for the numerical simulation. The flow velocity data was measured over a fixed bed after the sediment transport has reached equilibrium.

4.1 Boundary Conditions

According to the experiments, all the vertical walls of the flume were set to be smooth, sediment particle size, $d_{90}\approx$5.7mm, was used as a reference for bed roughness. The flow discharges for the main channel and side channel, 0.018 and 0.002m^3/s, used in the experiment, was set for the numerical simulation. Because the flow in the branch channel is super critical, water surface elevation of the branch channel was also used as boundary condition.

4.2 Simulation Results

The bed elevation of the simulation was generated based on the established equilibrium bed. The side channel flow is super critical with a mean depth of 0.02m and F_r=1.3, while the main channel is sub-critical flow with F_r=0.35. Under this condition a weak undulated hydraulic jump was observed. The high velocity flow from the tributary intrudes into the main channel and forms a two layer flow structure. Because of its higher location, near the confluence zone, the side discharge flows over the main channel flow which dives under the intruding side flow and passes through. The main channel flow also intrudes into the tributary forming a vertical roller which is bounded by the high sediment slope inside the tributary. The flow structure is reproduced by the CCHE3D model; Figure 3 shows the simulated vector field in a vertical section through the tributary channel using k-ε turbulence closure scheme and dynamic pressure.

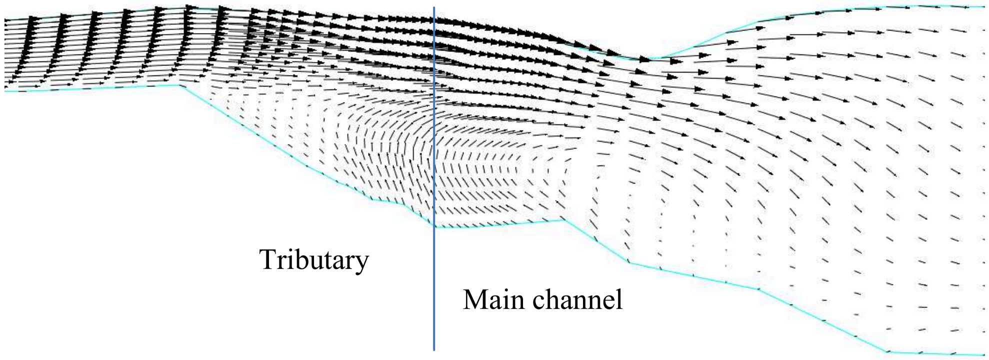

Figure 3 Simulated flow field in the confluence. The tributary side discharge is supercritical flow; it basically rides on the top of the main channel flow which dives under the side discharge and intrudes into the tributary channel with high slope.

The main channel flow and the tributary flow affect each other strongly. Due to the push of the side flow, a stagnation point is formed at the upstream corner of the tributary channel; the main channel flow was pushed toward the outer bank, a recirculation zone is thus formed immediately downstream of the confluence outlet. This flow pattern has been observed in the Lab and simulated by the CCHE3D model (Figure 4). The simulated flow patterns clearly indicate the mixing process of the side discharge into the main flow.

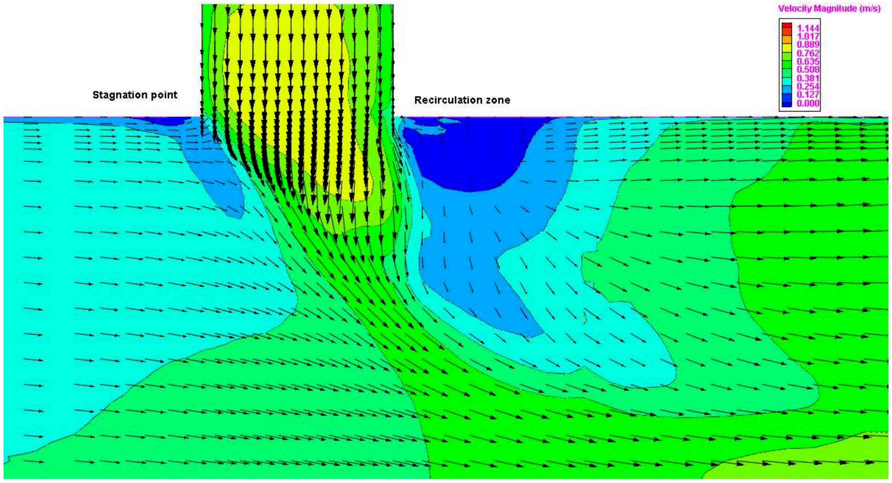

Figure 4 Simulated flow pattern at the water surface, the stagnation point and recirculation are both reproduced.

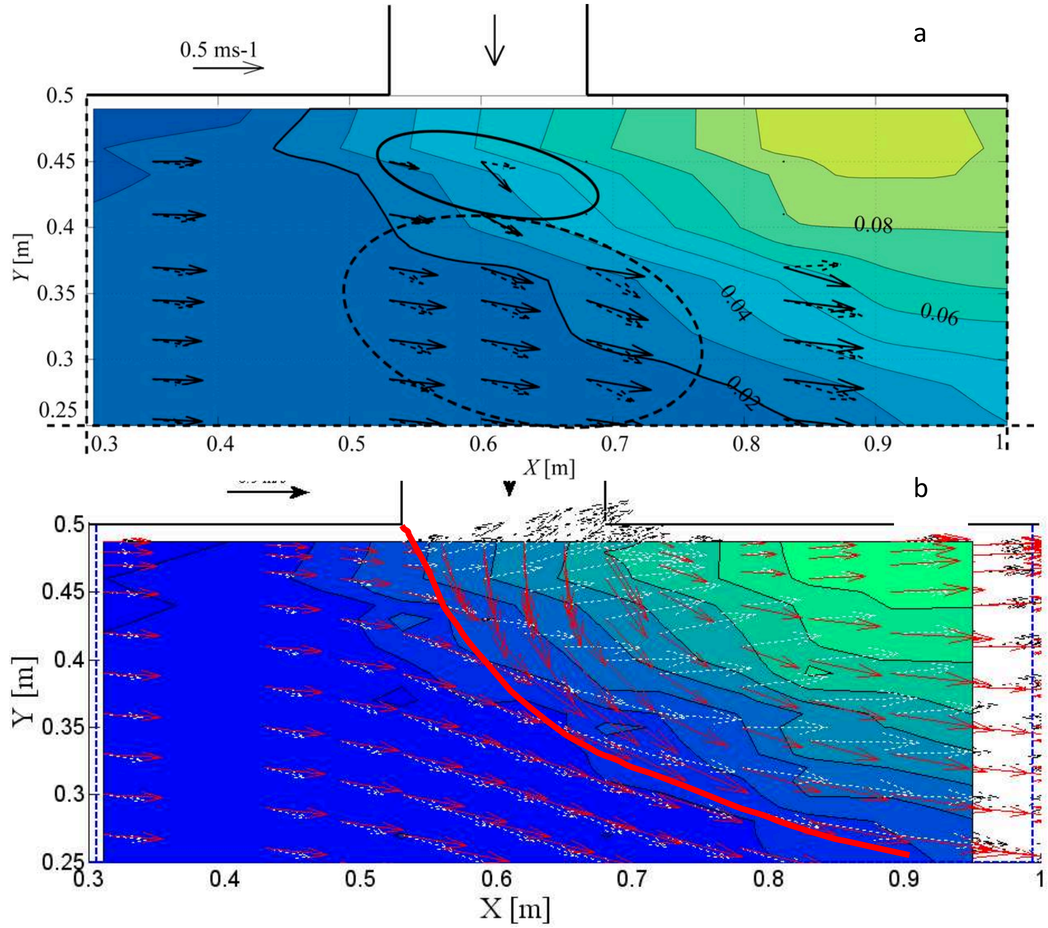

Figure 5 A comparison of observed and simulated 3D flow structure near the confluence.

Visualizing the simulation results, the recirculation zone only occupies the upper part of the flow depth. This is because the intruding side-discharge is flowing at the upper part of the main channel depth, the main channel flow dives at the stagnation point and passes to the downstream side of the side jet underneath. This is also observed in the experiment: the colored main channel flow from upstream passed under the side flow jet to the recirculation zone (Figure 9 in Leite Ribeiro et al. 2012).

A qualitative comparison of the two layer flow structure is also presented in Figure 5. Fig. 5a illustrates the observed velocity distributions near the bed (0.2h) and near the surface (0.7h) level: inside the shear layer (left side when looking downstream) the flow appears to rotate clockwise, while outside the shear layer (right side) the flow rotates counter-clockwise. Fig. 5b shows the simulated flow field plotted in a way similar to Fig. 5a. The two layer flow structure is seen to be consistent to the measured vector field. A red curve is drawn in Fig 5b separating the two rotation zones, it appears this boundary line in the simulated flow is located slightly closer to the right bank than the observed one, but the general trend is very close.

Finally, quantitative comparisons were made for measured and simulated longitudinal velocity fields in the main channel. Good agreements between the computed water surface elevations and velocity profiles using both CCHE2D and CCHE3D models have been obtained. Figure 6 shows the comparisons of the depth-averaged flow data and CCHE2D simulation results. Results of four transects, including that upstream (x=0.53m), in the middle (x=0.60m) and downstream of the confluence (x=1.33m, x=3.33m), are presented. It can be seen the 2D model can reasonably simulate the complex flow in the main channel, although 3D mixing and vertical two layers structure near the confluence cannot be predicted.

Figure 7 shows the comparisons of the simulated longitudinal velocities in two transects. One is in the middle of the confluence (x=0.6m) and the other is downstream of the confluence in the main channel (x=1.33m). As can be seen, the vertical velocity profile has been simulated very well: the vertical distribution lines of the observation and simulations are almost overlapping each other. Due to the development of the gravel bar, the flow in the main channel has a much narrower path. The velocity near the right bank of the downstream section (x=1.33m) is about 0.8m/s, the water depth is about 0.09m. The flow here is very close to super critical and turns to super-critical further downstream. The velocity data was measured after the sediment transport reached equilibrium and the form of sediment bar was stable. There is a slight difference, however, between the equilibrium bed and that during the measurement of the velocity (Fig. 7). In the 3D simulations, both of the bed data were used, so that simulated velocities in the measurement transects have better accuracy.

5. CONCLUSIONS

Both CCHE2D and CCHE3D model were applied to simulate the turbulent flow near a river confluence. The 2D model was used as a calibration tool and to prepare for the initial condition for the 3D model. The experimental data of Leite Ribeiro et al.(2012) was used for the numerical simulations. The flows of the tributary and in part of the main channel are supercritical. Both models produced satisfactory results. In particular, CCHE3D predicted the effect of the stagnation point of the confluence, the recirculation zone and the two layer flow structure near the confluence. Quantitative comparisons of the simulated water surface and velocity distributions also show good agreements between measurement and simulations. This is study validated the capabilities of both CCHE2D and CCHE3D of simulating highly turbulent free surface flows in river channels with tributary confluences.

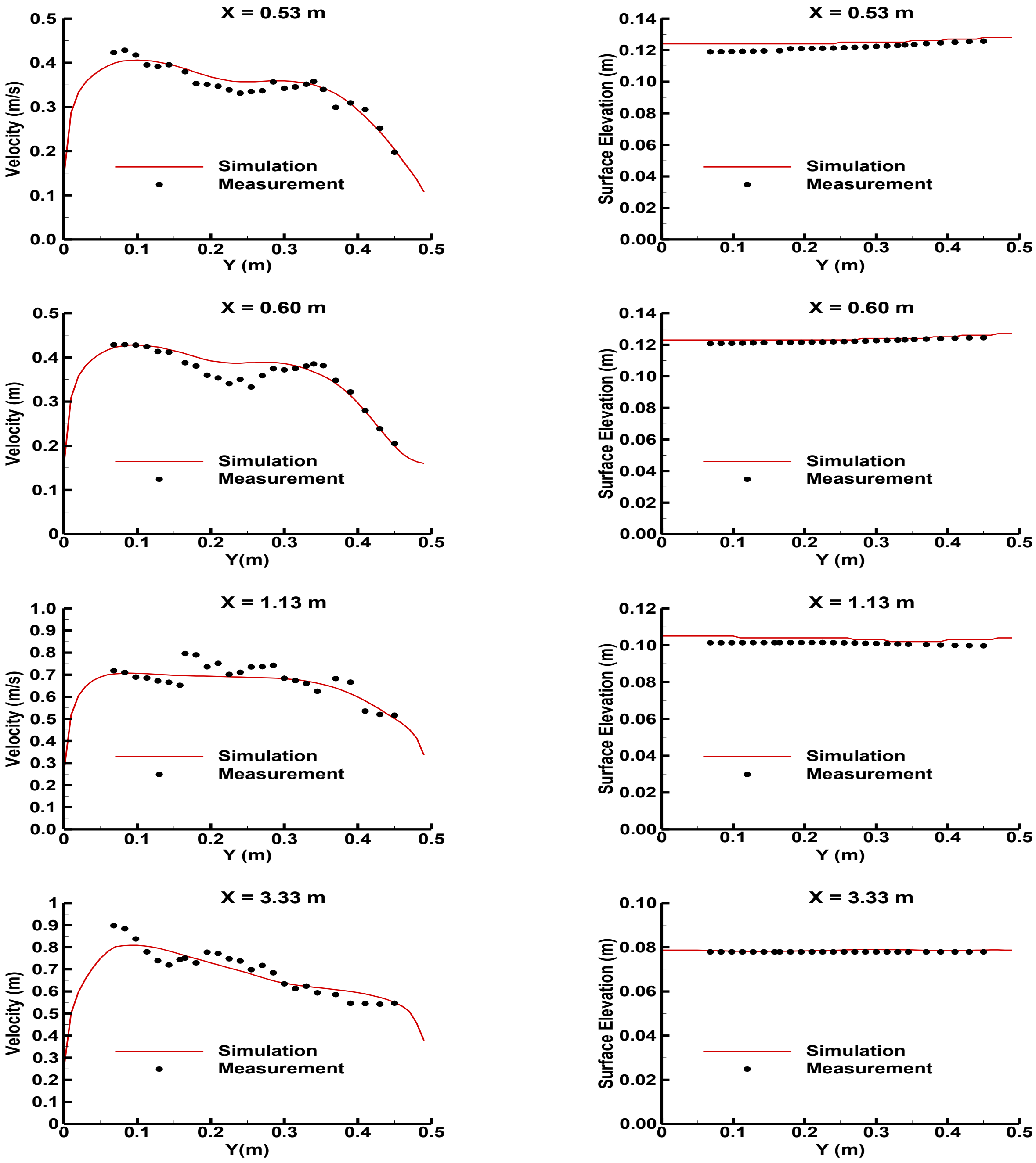

Figure 6 Comparisons of measured and simulated 2D longitudinal flow velocities and water surface in the main channel

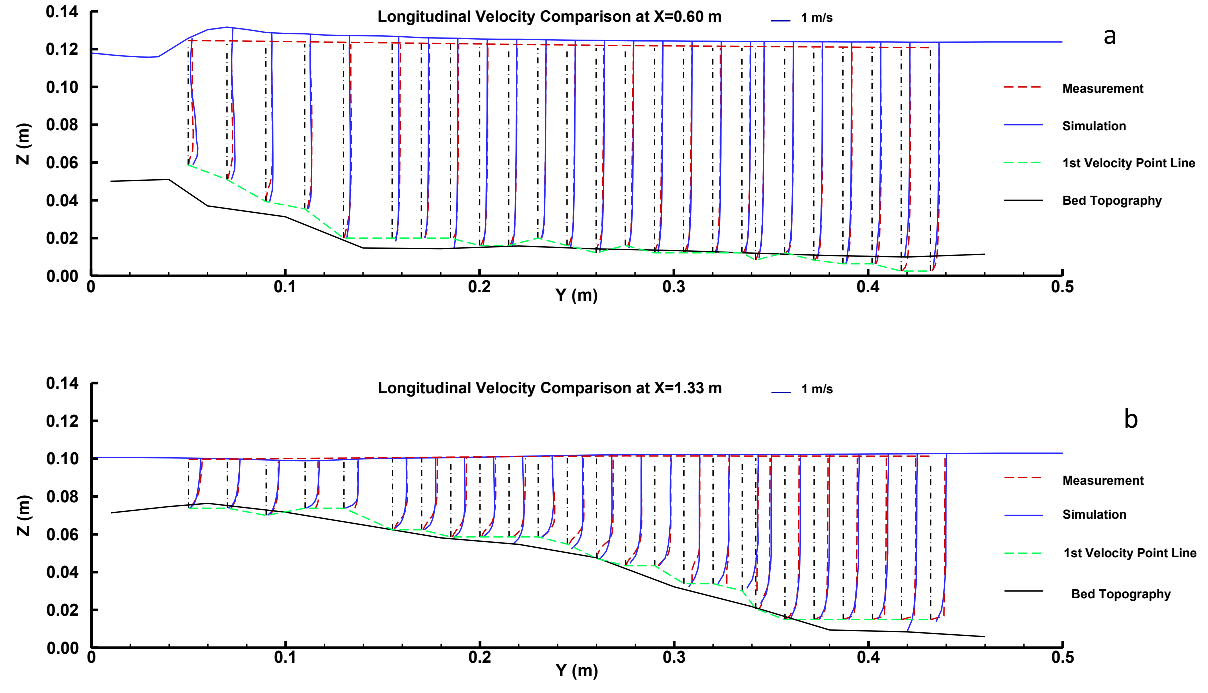

Figure 7 Comparison of computed and measured 3D velocity profiles in one cross-section at the confluence (a) and one downstream of the confluence (b). The bed topography was formed in the physical experiment.

ACKNOWLEDGEMENT

This work is a result of research supported in part by the USDA Agriculture Research Service under the Specific Research Agreement No. 58-6408-1-609 monitored by the USDA-ARS National Sedimentation Laboratory (NSL) and The University of Mississippi (UM).
The experimental research was supported by the Swiss Federal Office for the Environment (FOEN) in the framework of the project “Integrated management of river systems.” Blanckaert was partially funded by the the Chinese Academy of Sciences Visiting Professorship for Senior International Scientists, grant number 2011T2Z24.

REFERENCES

Baranya, S., J. Józsa, E. Napoli, 2010, “Field and numerical study of river confluence flow structures”, *River Flow 2010 - Dittrich, Koll, Aberle & Geisenhainer (eds) - © 2010 Bundesanstalt für Wasserbau ISBN 978-3-939230-00-7.*

Benda, L., K., Andras, D., Miller, and P., Bigelow, 2004, “Confluence effects in rivers: Interactions of basin scale, network geometry, and disturbance regimes”, Water Resources Research, Vol. 40, W05402, doi:10.1029/2003WR002583, 2004

Best, J., 1987, “Flow dynamics at river channel confluences; implications for sediment transport and bed morphology.” Recent developments in fluvial sedimentology; contributions from the Third International Fluvial Sedimentology Conference, Volume 9, 1987, pp25-37.

Bonakdari, H., G. Lipeme-Kouyi and X. Wang, 2011, “Experimental Validation of CFD Modeling of Multiphae Flow Trough Open Channel Confluence”, World Environmental and Water Resources Congress 2011, Bearing Knowledge for Sustainability © ASCE 2011, pp 2176-2183

Czernuszenko, W., Jia, Y. and Wang, S.S.Y., 1995, "3D Turbulent Side Discharge into Open Channel Flow," Proceedings of the XXVI IAHR Congress, Hydra 2000, Vol. 1, pp. 171-176, Thomas Telford, London, UK, Sept. 1995.

Demuren, A.O. and W. Rodi, 1983, “Side Discharges Into Open Channels: Mathematical Model” *Journal of Hydraulic Engineering,* Vol. 109, No. 12, December, 1983. ©ASCE, ISSN 0733-9429, pp1707-1722

Jia, Y., Czernuszenko, W. and Wang, S.S.Y, 2002a, ”Simulation of three-dimensional side discharge into open channels” Archives of Hydro-engineering and Environmental Mechanics, Polish Academy of Sciences. Vol. XLIX No. 1, 2002, pp 3-21.

Jia, Y., Wang, S.Y.Y., & Xu, Y. (2002b). Validation and application of a 2D model to channels with complex geometry, *International Journal of Computational Engineering Science*, 3(1) (March 2002), 57-71.

Jia, Y. and Wang, S.S.Y., 1999, "Numerical model for channel flow and morphological change studies", *Journal of Hydraulic Engineering*, ASCE, Vol. 125, No. 9, pp. 924-933.

Jia, Y., Steve Scott, Yichun Xu, Suiliang Huang, Sam S.Y. Wang, 2005, “Three-Dimensional Numerical Simulation and Analysis of Flow around a Submerged Weir in a Channel Bendway,” *Journal of Hydraulic Engineering*, 131(8), 682-693.

Leite Ribeiro, M, 2011, “Influence of Tributary Widening on Confluence Morphodynamics”, PhD dissertation, Lausanne, Swiss, 2011.

Leite Ribeiro, M., J.-L. Boillat, A. J. Schleiss and K. Blanckaert, 2010, “Experimental study on a widening tributary channel and its influence on the confluence morphology”, *River Flow 2010 - Dittrich, Koll, Aberle & Geisenhainer (eds) - © 2010 Bundesanstalt für Wasserbau ISBN 978-3-939230-00-7*

Leite Ribeiro, M., Blanckaert, K., Roy, A.G., and Schleiss, A. J., 2012, “Flow and sediment dynamics in channel confluences”, Journal Of Geophysical Research, Vol. 117, F01035, doi:10.1029/2011JF002171, 2012

Roca, M, J.P. Martín-Vide, P.J.M. Moreta, (2009), “Modelling a torrential event in a river confluence”, Journal of Hydrology 364 (2009) 207–215, doi:10.1016/j.jhydrol.2008.10.020

Speziale, C.G., Sarkar, S. and Gatski, T.B. (1991). “Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach.” *J. Fluid Mech.,* 277, 245-72.

Unde, M.G., and S. Dhakal, 2009, “Sediment characteristics at river confluences: a case study of the Mula-Kas confluence, Maharashtra, India”, *Progress in Physical Geography* 33(2) (2009) pp. 208–223

DOI: 10.1177/0309133309338655

Zhaowei Liu, Z.W., Chen, Y.C., and Zhu, D.J., 2007, “Study on the concentration distribution in a trapezoidal open-channel flow with a side discharge”, Environ Fluid Mech (2007) 7, pp509–517. DOI 10.1007/s10652-007-9042-7