Numerical simulation of confluence flow in open channel with dynamic meshes techniques

Yang, Q. Y.; Liu, T. H.; Lu, W. Z.; Wang, X. K.

Published in:
Advances in Mechanical Engineering

Published: 01/01/2013

Document Version:
Final Published version, also known as Publisher’s PDF, Publisher’s Final version or Version of Record

License:
CC BY

Publication record in CityU Scholars:
Go to record

Published version (DOI):
10.1155/2013/860431

Publication details:
Yang, Q. Y., Liu, T. H., Lu, W. Z., & Wang, X. K. (2013). Numerical simulation of confluence flow in open channel with dynamic meshes techniques. Advances in Mechanical Engineering, 2013, [860431]. https://doi.org/10.1155/2013/860431

Citing this paper
Please note that where the full-text provided on CityU Scholars is the Post-print version (also known as Accepted Author Manuscript, Peer-reviewed or Author Final version), it may differ from the Final Published version. When citing, ensure that you check and use the publisher's definitive version for pagination and other details.

General rights
Copyright for the publications made accessible via the CityU Scholars portal is retained by the author(s) and/or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights. Users may not further distribute the material or use it for any profit-making activity or commercial gain.

Publisher permission
Permission for previously published items are in accordance with publisher's copyright policies sourced from the SHERPA RoMEO database. Links to full text versions (either Published or Post-print) are only available if corresponding publishers allow open access.

Take down policy
Contact lbscholars@cityu.edu.hk if you believe that this document breaches copyright and provide us with details. We will remove access to the work immediately and investigate your claim.
Research Article

Numerical Simulation of Confluence Flow in Open Channel with Dynamic Meshes Techniques

Q. Y. Yang,1,2 T. H. Liu,1 W. Z. Lu,3 and X. K. Wang2

1 Department of Hydraulics, Changjiang River Scientific Research Institute, Wuhan 430010, China
2 State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University, Chengdu 610065, China
3 Department of Civil and Architecture Engineering, City University of Hong Kong, Hong Kong

Correspondence should be addressed to X. K. Wang; wangxiekang@scu.edu.cn

Received 1 July 2013; Accepted 20 October 2013

Abstract

Due to the interaction between branch flow and main stream at confluence zone in open channel, the water level of free surface often varies dramatically. In three dimensional simulations of confluence flow, multi-phase models are usually adopted in treating the free-surface boundary, for example, the Volume of Fluid model. The major dilemma for adopting multi-phase modes is that the method consumes more time and computation resources. In this study, the new mesh technique, the dynamic meshes, is adopted to track the position of free surface. The simulation results show that simulation adopting dynamic meshes converges rapidly and is in good agreement with the experimental data. In addition, simulations and comparisons of different turbulence models coupled with dynamic meshes, rigid lid, or the Volume of Fluid method are carried out to investigate the impact by tracing the free-surface boundary. The simulated position of free surface, velocity distribution, and vector field are all compared to the data collected in the flume test. The results of numerical simulations of confluence flow using the dynamic meshes present much better accuracy than those of Volume of Fluid or rigid-lid method, although they take the same turbulence model.

1. Introduction

1.1. Confluence Flow. Open-channel confluence flow is common in natural rivers and receives intensive interest in environmental and hydraulic engineering. Previous studies have presented detailed descriptions of flow properties in the confluence region, indicating that the distinctive characteristics of confluence flow are the recirculation zone and secondary circumpulsion.

Figure 1(a) shows a schematic of confluence flow structure in the horizontal plane [1]. When two channel flows meet at the confluence, due to the difference of velocity field in main and tributary channels, there is usually a shear plane existing between them, which is recognized as the major source of turbulence generation [2, 3]. With the stagnation zone as the beginning of the shear plane, a separation zone is usually generated downstream of the branch channel and marked as the source of turbulence. In separation zone, velocity magnitude is often small, while the velocity gradient near the edge of the separation zone is large. So the edge of the separation zone performs as another shear plane in the confluence region. Figure 1(b) shows the three-dimensional view of the confluence flow [4]. Due to the differences of centrifugal forces along the vertical direction, a secondary circulation is generated [5–7]. The separation zone and secondary circulation have significant impacts on the sediment transport [2, 8, 9] and the pollutant dispersion [10–21] in confluence region.

Several issues affect scales of the separation zone and intensity of the secondary circulation. Best and Reid [22, 23] studied the effects of confluence angle and flow rate ratio between main and tributary channels on the flow structure and bed deformation in the confluence region. Ashmore and Parker [24] and Borghesi and Sahebari [25] presented the relationship between flow rate ratio and local scour in the vicinity of confluence. Best and Roy [10], Biron et al. [26, 27], Bradbrook et al. [28], Rhoads and Sukhodolov [29], and Wang et al. [30] pointed out the great importance
of bed discordance on the scale of separation zone near the bed and the flow acceleration behind the confluence. Bradbrook et al. [31, 32] and Bryan and Kuhn [33] investigated structural differences between flows with symmetrical and asymmetrical confluence using numerical simulation and flume test, respectively. Biron et al. [34] studied the patterns of water surface topography at a river confluence. In recent years, some numerical models have been proposed for the study of the properties of confluence flow. Due to the vertical nonuniformity of the mean-flow quantities, such as secondary circulation and the scale of separation zone varying along channel depth, three-dimensional numerical models are preferable in this case. Huang et al. [35] performed three-dimensional simulation of right-angle confluence flow and verified the model with experimental data. Based on the model, they further investigated the flow structure under different confluence angles. Biron et al. [36] carried out a three-dimensional simulation of flow in confluence channel with a discordant bed, and the simulation results agreed well with the experiments. With a three-dimensional model, Shakibainia et al. [37] investigated the properties of separation zone and secondary circulation in the confluence region under different angles. By comparing the predictions of both three-dimensional and two-dimensional models with high-quality field data, Lane et al. [38] concluded that the three-dimensional model has a higher predictive ability, particularly if the two-dimensional model is not able to catch the effects on flow structure of secondary circulation. But in these three-dimensional simulations, the free water surface is usually treated as an assumed rigid-lid or captured by multi-phase modes. The position of the free surface cannot be pinpointed accurately, which will further ruin the simulation accuracy of velocity distribution.

The main difficulties in 3D simulation of confluence flow are rapid-change water surface and choice of turbulence models. For the first problem, the simplest way is to treat the free surface as a fixed lid [37]. The simulation of confluence flow is nearly the same as the simulation of flow in combined tubes [39]. By variable porosities in the surface layer of cells and RNG k-e turbulence models, Bradbrook et al. [16, 31, 32] investigated the controls on secondary circulation and came up with the proposition that velocity ratio is the prime determinant of cross-stream pressure gradient, which initiates cross-stream velocities. They also confirmed that RNG k-e model can improve model predictions in situations where separation zones form. Biron et al. [36] adopted the same method and turbulence model to investigate the mixing at river confluences. By utilizing the VOF method and RNG k-e model, Wang et al. [40] carried out three-dimensional simulations to study the effects of bed discordance on the structure of confluence flow. Huang et al. [35] simulated flow structures in a right angle open-channel junction and obtained good agreements between the simulation and experimental measurements by means of the model and a free-surface modification method. Zeng et al. [41] used the sigma coordinate to map the physical domain to a uniform transformed space and Spalart-Allmaras turbulence model and LES, respectively, to consider the effects of turbulence and found that the LES approach is more accurate than the RANS approach.

1.2. Dynamic Meshes Techniques. For most numerical simulations, the first step is meshing the simulation region. When the object to be simulated is solid or fluid in pipes, meshing is not difficult. But when it comes to water in open channels, especially when the surface of water varies, figuring out the position of water surface is not an easy job. The position of the surface is known or assumed at the start, but later its location has to be found out as part of the solution.

Generally, the ways of dealing with the free surface fall into two categories, interface capturing and interface tracking. In interface-capturing methods, computation is carried out in a solution domain that extends over regions occupied by both water and air (Figure 2). During the simulation, variables about meshes with or without water need to be stored and calculated. An additional equation is solved for concentration or volume fraction of one fluid. This costs a lot of computational resources and is very time consuming. Methods, like MAC (Marker and Cell) and VOF (Volume of Fluid), belong to this category. This kind of methods is good at treating complex phenomena like wave breaking but may generate large errors near the interface. For example, the location got by VOF usually has an error about three times the mesh size.
Table 1: Flow conditions in the experiment.

| Runs | \( Q_b \) (m\(^3\)/s) | \( Q_m \) (m\(^3\)/s) | \( Q_d \) (m\(^3\)/s) | \( q_b \) (m\(^2\)/s) | \( q_d \) (m\(^2\)/s) | \( R_q \) \((q_b/q_d)\) |
|------|----------------|----------------|----------------|----------------|----------------|----------------|
| 1    | 0.127          | 0.042          | 0.169          | 0.139          | 0.185          | 0.750          |

Table 2: Runs for the three-dimensional simulation of confluence flow.

| Runs | Cases | Turbulence model | Surface treating | Meshes adopted | Computing time (hours) |
|------|-------|------------------|------------------|----------------|-----------------------|
| 1    |       | Standard \( k-\varepsilon \) | Dynamic meshes | 165,456        | 118                   |
| 2    |       | Realizable \( k-\varepsilon \) | Dynamic meshes | 165,456        | 124                   |
| 3    | \( R_q = 0.75 \) | \( k-\omega \) | Dynamic meshes | 165,456        | 130                   |
| 4    | \( k-\omega \) | Rigid lid | 165,456        | 50             |
| 5    | \( k-\omega \) | VOF | 232,200        | 245            |

Interface-tracking methods treat the free surface as a sharp interface, and grids are advanced each time the free surface is moved (Figure 3). Only one fluid, water, is considered during the simulation. This type of methods saves a lot of storage and calculation capacity but needs efficient and stable algorithms to treat the intersections of free surface with other boundaries. These methods can track the position of free surface accurately, without considering the impact of fluid at the other side of the surface.

Interface-tracking methods can be further divided into two branches: pressure-based interface tracking and flux-based surface tracking. Before the two tracking methods are introduced here, the boundary conditions at the water surface need to be listed.

The kinematic condition requires no mass flux through the interface. Usually it is expressed as

\[
\left[ (\overline{\mathbf{u}} - \overline{\mathbf{u}}_b) \cdot \hat{n} \right]_{fs} = 0,
\]

where \( \overline{\mathbf{u}}_b \) is the velocity at the surface and \( \hat{n} \) is the normal direction of surface. The other boundary condition is the dynamic condition, which requires the force to be acting on the free surface in equilibrium. For water in open channels, the force should be equal to the pressure of the atmosphere. Usually the atmosphere is taken as the reference pressure, so the force will be zero, so

\[
p_{fs} = 0.
\]

If the two conditions are fulfilled at the boundary, the position of the boundary will be where the surface is. Surface tracking can be accomplished in two ways.

The first way is, during calculation, keeping the pressure at the surface boundary zero all the time (Figure 4(a)). Then, after updating velocity, there is a fictitious flux at the surface boundary. According to the magnitude of the flux, we modify the position of surface meshes until the flux is sufficiently small. This is how the flux-based surface tracking works.

The other way is, during the iteration, keeping the mass flux through the surface boundary zero (Figure 4(b)). So after updating of the flow field, there is nonzero pressure at the surface boundary. By the magnitude of pressure, we adjust the position of the mesh until the pressure is small. This is how the pressure-based surface tracking is accomplished.

Self-programmed computer codes adopting the latter way have been embedded into the commercial software ANSYS FLUENT for tracking the position of the free surface. The major advantage of using the pressure-based surface tracking is that the dominant force that water suffers is gravity. For any point in the water, its depth has a nearly linear connection to the pressure. Pinpointing the position of surface based on pressure is more reliable than the flux.

2. Study Methods

The simulation was conducted by ANSYS FLUENT; the mainstream and branch intake are set as mass flow inlets. Since the velocity distribution at the intakes is usually set as uniform, actually which is not true, the distance from the intakes to the junction corner is extended to eight times the channel width to make the uniform boundary conditions applicable. The outlet is set as pressure outlet with constant level 0.305 m. The free surface is treated as a symmetric plane and self-proposed surface tracking codes are used to update its position. When the VOF method is adopted, the top of simulation region is treated as pressure inlet.

The data of the experiment by Weber et al. [4] are used for verification. The experiments were carried out in a right
Figure 4: Surface-tracking methods.

(a) \[ P = 0 \] Flux \( \neq 0 \)

(b) \[ P \neq 0 \] Flux = 0

Figure 5: Measuring points arrangement in the experiments.

Figure 6: Meshes adopted in the 3D simulations.

Figure 7: Three-dimensional view of the meshes for simulation of confluence flow.
Figure 8: Contour of surface levels.

Figure 9: Comparisons of surface levels.
angle confluence flume with equal width (0.914 m). All the coordinates were normalized by the width, denoted by \( x^*, y^*, \) and \( z^* \). The flow conditions of each run of the experiment are listed in Table 1. Flow rates downstream of the confluence in each run are nearly the same. Figure 5(a) shows the plane arrangement of measuring points, and Figure 5(b) shows the points along the vertical measuring line. There are 17 points in each measuring line. Due to limitations of space, only results of the simulation under flow rate ratio of 0.75 are presented (Table 2). The computational costs are also listed in Table 2. All the simulations are carried out on a computer with an Intel core 2 E7500 CPU and 2G memory. For simulations utilizing the dynamic meshes techniques or rigid-lid method, the same meshes are adopted at the initial state. As the meshes for rigid-lid method are not adjusted during the calculation, much shorter computation time is needed. For simulations utilizing VOF method, more meshes are used to increase the accuracy in capturing the free surface.

Three two-equation models, Standard \( k-\varepsilon \), and Realizable \( k-\varepsilon \) and \( k-\omega \) are utilized to consider the effects of turbulence (Pope, 2000 [42]; Wilcox, 1994 [43]). In these models, transport equations of kinetic energy \( (k) \), turbulent

**Figure 10: Contour of \( U^* \) at cross section \( X^* = -3 \).**
dissipation rate ($\varepsilon$), and specific dissipation rate ($\omega = \varepsilon/k$) are solved to obtain the turbulent viscosity ($\nu_t = k^2/\varepsilon$ or $\nu_t = k/\omega$). The Standard $k-\varepsilon$ model shows good performance in simulating fully turbulent flows and has been found to work fairly well for a wide range of wall-bounded and free shear flows. The Realizable $k-\varepsilon$ model is a modified version of Standard $k-\varepsilon$ model and provides the best performance for several validations of separated flows and flows with complex secondary flow features. Studies show that the $k-\omega$ model has better accuracy for predicting free shear flows. In confluence flow, free shear flow, separated flow, and secondary flow occur at the same time, and it is a good chance to verify the performance of the three turbulence models.

Figure 6 shows the mesh used in the simulation, and Figure 7 shows the three-dimensional view of the mesh at the beginning and the end of each simulation. For convenient observation, the $Z$ to $X$ and $Y$ coordinates ratio has been changed. The prismatic mesh has a triangular projection in the horizontal plane and rectangular projection in the vertical plane. This makes the moving nodes of the same face at the free surface be always in one plane, and no negative volumes come out during the calculation.

3. Results and Discussion

3.1. Water Levels. Figure 8 shows the contour of free surface levels obtained by experiment and different turbulence models. Comparisons among them at three sections are presented in Figure 9. Figure 9 shows clearly that as the dynamic meshes techniques are adopted, the predicted water levels are in very satisfactory agreement with the observed ones. Turbulence models present little impacts on the accuracy of surface tracking, although the errors are a bit larger in the confluence zone ($X^* = -4$). However, when the VOF method is
3.2. Longitude Velocity. Figure 10 presents the distribution of velocity component $U^*$ at section $X^* = -3$. From the scale of separation zone (positive $U^*$) point of view, the result from the realizable $k$-$\epsilon$ model shows best agreement with the experimental data. The results from the standard $k$-$\epsilon$ model and $k$-$\omega$ model exhibit larger sizes. From the magnitude of velocity point of view, the peak region ($U^* < -1.4$) obtained by the realizable $k$-$\epsilon$ model and $k$-$\omega$ model coupled with VOF method is smaller than the experiment results, while standard $k$-$\epsilon$ model predicts a larger region. The rigid-lid treatment for the surface is by no way correct. The $k$-$\omega$ model shows best agreement among the three models.

3.3. Section Circulation. Figure 11 shows the circulation at section $X^* = -3$. It can be seen that all the three turbulence models, either coupled with dynamic meshes or VOF
method, can predict the circulation cells correctly except the result got by treating the surface as rigid lid. But the predicted magnitude of the circulation (denoted by the length of vector) is a bit smaller than the experimental measurements.

Figure 12 presents the distribution of TKE at section \( X^* = -3 \). It can be seen that standard \( k-\varepsilon \) model, realizable \( k-\varepsilon \) model, and \( k-\omega \) model coupled with VOF method all overpredicted the magnitude of TKE. The results got by \( k-\omega \) model show best agreement while the simulation with rigid-lid surface is much inaccurate.

The above analyses do not lead to a sound conclusion about which turbulence model performs the best in all respects. Either realizable \( k-\varepsilon \) or \( k-\omega \) model shows better accuracy at different occasions, and the standard \( k-\varepsilon \) model shows poor agreement under most conditions. The \( k-\omega \) model is preferable for simulation of confluence flow.

4. Conclusions

In the three-dimensional simulation of confluence flow, three types of surface treating methods: rigid lid, VOF, and self-proposed codes with dynamic meshes techniques, are used to track the surface position. Several two-equation turbulence models are also adopted to consider the influence of turbulence. The simulation results show that different surface treating methods impact the simulation accuracy greatly, although the same turbulence model is adopted. The rigid-lid method treats the free surface as an assumed surface, which generates great error when the actual surface varies largely. The VOF method captures free surface by a multi-phase model, which shows better accuracy than that of rigid-lid method, but still presents poor accuracy when the water flow is shallow. When the dynamic meshes techniques are utilized, good agreement with the experimental data is obtained and turbulence models show little impact on the tracking of water levels. For the velocity distribution, \( k-\omega \) model is preferable for simulation of confluence flow.

Acknowledgments

This work was partially supported by the central level scientific research institutes fund project “Study on numerical methods of large slope surface tracking in simulation of open channel flow,” National Natural Science Foundation of China (Grant nos. 41171016, 51209016), Hong Kong RGC-GRF Grant (CityU 118212), and Strategic research grant, City University of Hong Kong (Project no. CityU-SRG 7002718).

References

[1] J. L. Best, “Flow dynamics at river channel confluences: implications for sediment transport and bed morphology,” in Recent Developments in Fluvial Sedimentology, vol. 39 of SEPM Special Publication, pp. 27–35, 1987.

[2] B. L. Rhoads, J. D. Riley, and D. R. Mayer, “Response of bed morphology and bed material texture to hydrological conditions at an asymmetrical stream confluence,” Geomorphology, vol. 109, no. 3-4, pp. 161–173, 2009.

[3] B. De Serres, A. G. Roy, P. M. Biron, and J. L. Best, “Three-dimensional structure of flow at a confluence of river channels with discordant beds,” Geomorphology, vol. 26, no. 4, pp. 313–335, 1999.

[4] L. J. Weber, E. D. Schumate, and N. Mawer, “Experiments on flow at a 90° open-channel junction,” Journal of Hydraulic Engineering, vol. 127, no. 5, pp. 340–350, 2001.

[5] X.-K. Wang, Q.-Y. Yang, W.-Z. Lu, and Y. Sun, “Study on characteristics of separation zone in confluence river,” Journal of Sichuan University (Engineering Science Edition), vol. 40, no. 6, pp. 1–6, 2008 (Chinese).

[6] Y. Qing-Yuan, W. Xian-Ye, L. Wei-Zhen, and W. Xie-Kang, “Experimental study on characteristics of separation zone in confluence zones in rivers,” Journal of Hydrologic Engineering, vol. 14, no. 2, pp. 166–171, 2009.

[7] Q. Yang, Y. Sun, X. Wang, W. Lu, and X. Wang, “Effects of turbulence models on the numerical simulation of flow in open channel junction,” Mechanics of Advanced Materials and Structures, vol. 18, no. 8, pp. 566–571, 2011.

[8] M. G. Unde and S. Dhakal, “Sediment characteristics at river confluences: a case study of the Mula-Kas confluence, Maharaashtra, India,” Progress in Physical Geography, vol. 33, no. 2, pp. 208–223, 2009.

[9] R. I. Ferguson, J. R. Cudden, T. B. Hoey, and S. P. Rice, “River system discontinuities due to lateral inputs: generic styles and controls,” Earth Surface Processes and Landforms, vol. 31, no. 9, pp. 1149–1166, 2006.

[10] J. L. Best and A. G. Roy, “Mixing-layer distortion at the confluence of channels of different depth,” Nature, vol. 350, no. 6317, pp. 411–413, 1991.

[11] R. Smith and N. C. Daish, “Dispersion far downstream of a river junction,” Physics of Fluids A, vol. 3, no. 5, pp. 1102–1109, 1991.

[12] J. M. Gaudet and A. G. Roy, “Effect of bed morphology on flow mixing length at river confluences,” Nature, vol. 373, no. 6510, pp. 138–139, 1995.

[13] B. L. Rhoads and S. T. Kenworthy, “Time-averaged flow structure in the central region of a stream confluence,” Earth Surface Processes and Landforms, vol. 23, no. 2, pp. 171–191, 1998.

[14] A. N. Sukhodolov and B. L. Rhoads, “Field investigation of three-dimensional flow structure at stream confluences 2. Turbulence,” Water Resources Research, vol. 37, no. 9, pp. 2411–2424, 2001.

[15] B. L. Rhoads and A. N. Sukhodolov, “Spatial and temporal structure of shear layer turbulence at a stream confluence,” Water Resources Research, vol. 40, no. 6, 2004.

[16] K. F. Bradbrook, P. M. Biron, and K. S. Richards, “Investigation of controls on secondary circulation in a simple confluence geometry using a three-dimensional numerical model,” Hydrological Processes, vol. 12, no. 8, pp. 1371–1396, 1998.

[17] L. Maurice-Bourgoin, B. Quemerais, P. Moreira-Turcq, and P. Seyler, “Transport, distribution and speciation of mercury in the Amazon River at the confluence of black and white waters of the Negro and Solimões Rivers,” Hydrological Processes, vol. 17, no. 7, pp. 1405–1417, 2003.

[18] R. E. Rathbun and C. E. Rostad, “Lateral mixing in the Mississippi River below the confluence with the Ohio River,” Water Resources Research, vol. 40, no. 5, 2004.

[19] S. P. Rice, R. I. Ferguson, and T. B. Hoey, “Tributary control of physical heterogeneity and biological diversity at river confluences,” Canadian Journal of Fisheries and Aquatic Sciences, vol. 63, no. 11, pp. 2553–2566, 2006.
[20] L. E. Schemel, M. H. Cox, R. L. Runkel, and B. A. Kimball, “Multiple injected and natural conservative tracers quantify mixing in a stream confluence affected by acid mine drainage near Silverton, Colorado,” Hydrological Processes, vol. 20, no. 13, pp. 2727–2743, 2006.

[21] A. Laraque, J. L. Guyot, and N. Filizola, “Mixing processes in the amazon river at the confluences of the negro and solimões rivers, encontro das águas, manaus, Brazil,” Hydrological Processes, vol. 23, no. 22, pp. 3131–3140, 2009.

[22] J. L. Best and I. Reid, “Separation zone at open-channel junctions,” Journal of Hydraulic Engineering, vol. 110, no. 11, pp. 1588–1594, 1984.

[23] J. L. Best, “Sediment transport and bed morphology at river channel confluences,” Sedimentology, vol. 35, no. 3, pp. 481–498, 1988.

[24] P. Ashmore and G. Parker, “Confluence scour in coarse braided streams,” Water Resources Research, vol. 19, no. 2, pp. 392–402, 1983.

[25] S. M. Borghei and A. J. Sahebari, “Local scour at open-channel junctions,” Journal of Hydraulic Research, vol. 48, no. 4, pp. 538–542, 2010.

[26] P. Biron, J. L. Best, and A. G. Roy, “Effects of bed discordance on flow dynamics at open channel confluences,” Journal of Hydraulic Engineering, vol. 122, no. 12, pp. 676–682, 1996.

[27] P. Biron, A. G. Roy, and J. L. Best, “Turbulent flow structure at concordant and discordant open-channel confluences,” Experiments in Fluids, vol. 21, no. 6, pp. 437–446, 1996.

[28] K. F. Bradbrook, S. N. Lane, K. S. Richards, P. M. Biron, and A. G. Roy, “Role of bed discordance at asymmetrical river confluences,” Journal of Hydraulic Engineering, vol. 127, no. 5, pp. 351–368, 2001.

[29] B. L. Rhoads and A. N. Sukhodolov, “Field investigation of three-dimensional flow structure at stream confluences: 1. Thermal mixing and time-averaged velocities,” Water Resources Research, vol. 37, no. 9, pp. 2393–2410, 2001.

[30] X. Wang, Q. Yang, W. Lu, and X. Wang, “Effects of bed load movement on mean flow characteristics in mobile gravel beds,” Water Resources Management, vol. 25, no. 11, pp. 2781–2795, 2011.

[31] K. F. Bradbrook, S. N. Lane, and K. S. Richards, “Numerical simulation of three-dimensional, time-averaged flow structure at river channel confluences,” Water Resources Research, vol. 36, no. 9, pp. 2731–2746, 2000.

[32] K. F. Bradbrook, S. N. Lane, K. S. Richards, P. M. Biron, and R. A. G., “Large Eddy Simulation of periodic flow characteristics at river channel confluences,” Journal of Hydraulic Research, vol. 38, no. 3, pp. 207–215, 2000.

[33] R. B. Bryan and N. J. Kuhn, “Hydraulic conditions in experimental rill confluences and scour in erodible soils,” Water Resources Research, vol. 38, no. 5, pp. 211–214, 2002.

[34] P. M. Biron, A. Richer, A. D. Kirkbride, A. G. Roy, and S. Han, “Spatial patterns of water surface topography at a river confluence,” Earth Surface Processes and Landforms, vol. 27, no. 9, pp. 913–928, 2002.

[35] J. Huang, L. J. Weber, and Y. G. Lai, “Three-dimensional numerical study of flows in open-channel junctions,” Journal of Hydraulic Engineering, vol. 128, no. 3, pp. 268–280, 2002.

[36] P. M. Biron, A. S. Ramamurthy, and S. Han, “Three-dimensional numerical modeling of mixing at River confluences,” Journal of Hydraulic Engineering, vol. 130, no. 3, pp. 243–253, 2004.

[37] A. Shakibainia, M. R. M. Tabatabai, and A. R. Zarrati, “Three-dimensional numerical study of flow structure in channel confluences,” Canadian Journal of Civil Engineering, vol. 37, no. 5, pp. 772–781, 2010.

[38] S. N. Lane, K. F. Bradbrook, K. S. Richards, P. A. Biron, and A. G. Roy, “The application of computational fluid dynamics to natural river channels: three-dimensional versus two-dimensional approaches,” Geomorphology, vol. 29, no. 1-2, pp. 1–20, 1999.

[39] A. S. Ramamurthy, J. Qu, and C. Zhai, “3D simulation of combining flows in 90° rectangular closed conduits,” Journal of Hydraulic Engineering, vol. 132, no. 2, pp. 214–218, 2006.

[40] X.-G. Wang, Z.-M. Yan, and W.-D. Guo, “Three-dimensional simulation for effects of bed discordance on flow dynamics at Y-shaped open channel confluences* Project supported by the National Natural Science Foundation of China (Grant No. 30490235),” Journal of Hydrodynamics, vol. 19, no. 5, pp. 587–593, 2007.

[41] J. Zeng, G. Constantinescu, and L. Weber, “3D calculations of equilibrium conditions in loose-bed open channels with significant suspended sediment load,” Journal of Hydraulic Engineering, vol. 136, no. 9, pp. 557–571, 2010.

[42] S. B. Pope, Turbulent Flows, Cambridge University Press, Cambridge, UK, 2000.

[43] D. C. Wilcox, Turbulence Modeling for CFD, DCW industries, 1994.