Numerical investigation of flow field and flowmeter accuracy in open-channel junctions

Minoo Sharifipour\textsuperscript{a}, Hossein Bonakdari\textsuperscript{b*}, Amir Hossein Zaji\textsuperscript{b} and Shahaboddin Shamshirband\textsuperscript{b}

\textsuperscript{a}Department of Civil Engineering, Razi University, Kermanshah, Iran; \textsuperscript{b}Department of Computer Science, Faculty of Computer Science and Information Technology, University of Malaya, Kuala Lumpur, Malaysia

(Received 31 July 2014; final version received 14 January 2015)

The confluence of two natural rivers or two artificial channels is of special importance due to the complexity of flow behavior. Such complexity has considerable effects on the accuracy of flow characteristics measured by instruments. The main objective of this study is to investigate the effect of width and discharge ratio change on open-channel junction velocity distribution and flowmeter mean velocity measurement accuracy. To accomplish this, a CFD model of an open-channel junction was simulated and validated with an experimental study. After validation, the open-channel junction was modeled with various width and discharge ratios. The results showed that increasing the width ratio from $W^* = 0.5$ to $W^* = 2.0$ decreases the effect of tributary flow on main flow, which in turn results in a decrease in flowmeter measurement error from 73\% to 35\%, respectively. Also, the increment of discharge ratio from 0.25 to 0.75 reduces flowmeter measurement error from 59.5\% to 13.65\%.

\textbf{Keywords:} open-channel junction; flowmeter; velocity field; CFD modeling

1. Introduction

Open-channel junctions are widely applicable in hydraulic structures such as irrigation and drainage, water distribution, and wastewater treatment systems. Due to the complex, three-dimensional hydrodynamic flow in the postjunction region, which is caused by the collision of two different flows with varying velocities and directions, extensive studies are necessary on this topic. A simple junction is formed when two tributary branches collide, or a tributary stream flows into the main channel. All geometrical parameters (channel shape and dimensions, junction angle, channel roughness) and hydraulic parameters (main channel and tributary discharge, Froude number) have significant effects on several areas that develop in a junction. Figure 1 shows a simple schematic representation of an open channel confluence flow and different areas developed in the horizontal plane. Tributary flow diversion is caused by the removal of wall pressure downstream of the tributary flow. The separation zone is susceptible to sediment deposition due to low velocity magnitudes in those areas. A contraction zone is the result of flow diversion and is the location where peak velocity occurs. An increase in deviation leads to an increase in flow contraction, which consequently leads to an increase in velocity in this region. Therefore, in this area, shear stress increases and provides favorable conditions for erosion in the channel floor and walls. In Figure 1, the velocity profiles of separation and contraction zones as well locations of flowmeters placed in the middle of the channel cross section are shown schematically.

Literature addressing effective parameters on flow behavior in open-channel confluence dates back to the 1940s when researchers mainly focused on analytical and experimental models. The first research on open-channel junction flow was done by Taylor (1944). The author used momentum analysis to develop an equation to estimate the junction's upstream-to-downstream depth ratio. Subsequently, several theoretical and experimental studies were done to analyze the post-junction regions developed, particularly the separation zone (Best & Reid, 1984; Hager, 1987; Kumar Gurram, Karki, & Hager, 1997; Modi, Ariel, & Dandekar, 1981; Webber & Greated, 1966). Weber, Schumate, and Mawer (2001) used an experimental setup to collect a complete, three-dimensional data set that could serve as a basis for the validation of numerical models used in various research works (Huang, Weber, & Lai, 2002; Shakibainia, Tabatabai, & Zarrati, 2010; Yang & Chang, 2005). Their physical model consisted of a sharp-edged rectangular channel with a junction angle of 90\° and identical widths. The authors measured the velocity, depth and turbulence intensities of flow over a wide range of discharge ratios. They also analyzed the size of the separation zone at various discharge ratios. Wang, Wang, Lu, and Liu (2007) experimentally studied the three-dimensional flow behavior in a junction flume. They concluded that it is possible to divide channel confluence flow into several zones consisting of a separation zone, maximum and minimum

---

\*Corresponding author. Email: bonakdari@yahoo.com

\(©\) 2015 The Author(s). Published by Taylor & Francis.

This is an Open Access article distributed under the terms of the Creative Commons Attribution License (http://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.
velocity regions and a shear plane developed between the two flows joining downstream of the confluent channel.

Owing to the development of high-speed computers over the past decades numerical models have become widely employed in various engineering fields (Chau & Jiang, 2001, 2004; Liu & Yang, 2013; Wu & Chau, 2006). CFD methods have often been used for various engineering problems (Baghalian, Bonakdari, Nazari, & Fazli, 2012; Bonakdari, Baghalian, Nazari, & Fazli, 2011). The rapid expansion in the use of CFD is on account of advantages such as cost and time savings, detailed output information, performance prediction before installing the actual system, result reliability, visualization capability and handling a wide range of engineering problems. Bradbrook, Lane, and Richards (2000) used a three-dimensional, numerical model to simulate the confluence flow in a rectangular laboratory channel and the field confluence of the Kaskaskia River and Copper Slough. Huang et al. (2002) used a K-ω turbulence model to evaluate the size of the separation zones and surface elevation for various confluence angles of open-channel junctions. Zhao, Zhu, and Rajaratnam (2008) used a CFD model to simulate fully surcharged flow at a 90° combining sewer junction. Shakibainia et al. (2010) conducted a comprehensive analysis of various flow parameters at the junction. The authors showed that the size of the separation zone decreases at angles greater than 90° or smaller than 45°. Bonakdari, Lipeme-Kouyi, and Wang (2011) used a three-dimensional CFD model to examine the effect of discharge ratio on recirculation size and presented relationships for calculating the length and width of the separation zone in terms of discharge ratio for a junction angle of 30°. Momplot et al. (2012) investigated the effect of mesh refinement and various turbulence models on the performance of open-channel junction CFD modeling. Yang, Liu, Lu, and Wang (2013) indicated that the dynamic mesh technique performs better than rigid-lid and VOF in free-surface simulation. Xie, Lin, and Falconer (2013) predicted the turbulence structure of open channels using large eddy simulation and predicted the main velocity, secondary velocity, boundary shear stress and Reynolds stress.

Flow discharge in an open channel is calculated with Equation (1).

\[ Q = A(h) \times U_{\text{mean}} \]  

where \( U_{\text{mean}} \) is the mean velocity and \( A(h) \) is the wetted area calculated by using the depth of flow and geometrical conditions of the channel. Mean velocity is calculated using some sensors and assuming that the sensors represent the entire cross section area of fluid flow (Bonakdari & Zinatizadeh, 2011; Hughes, Longair, Ashley, & Kirby, 1996). The difference between real and measured mean velocity depends on the hydrodynamic characteristics of fluid flow (Bonakdari, Zinatizadeh, Tahershamsi, Shahrezaei, & Levacher, 2009; Koelling, 1996). In practice, flowmeter sensors only cover a limited volume of a specific section, and the measured mean velocity is obtained from this limited volume (Larrarte, Bernard Bardiaux, Battaglia, & Joannis, 2008). It is clear that due to three-dimensional, complex and non-developed characteristics of velocity fields in confluence zones, flowmeter measurement errors may increase in the confluence sections. The difference between the measured and real mean flow velocity could increase up to 60% (Mignot et al., 2012).

Computational Fluid Dynamic (CFD) methods can serve as powerful tools in sensor calibration (Bonakdari & Zinatizadeh, 2008). Koelling (1996) applied numerical modeling to find the correlation coefficient of flowmeter measurement in open-channel sewers and demonstrated that this coefficient is related to depth of flow. Hilgenstock and Ernst (1996) used numerical CFD modeling to investigate the velocity profile on flowmeter accuracy. The authors showed that ultrasonic flowmeters could be calibrated by CFD modeling.

The main goal of the present study is to investigate the velocity distribution and flowmeter accuracy of open-channel junctions for various widths and discharge ratios. To do this, open-channel junctions were simulated and verified with Weber et al.’s (2001) experimental study results. Then, by using CFD simulation and a k-ω turbulence model, open channel junctions were modeled with width ratios of 0.5, 0.75, 1.0, 1.25, 1.5, 1.75 and 2.0, and discharge ratios of 0.25, 0.5 and 0.75. To investigate the impact of width and discharge ratio on flowmeter accuracy in each section, the average velocity was compared to the line average velocity of an assumptive flowmeter in the middle of the cross section.

2. Experimental setup

The numerical model was validated using the experimental results of Weber et al. (2001). The channel’s geometry is shown in Figure 2. The origin of the coordinates was at the bottom corner of the tributary’s upstream. The direction of the coordinate axes is such that the positive direction of the \( x \)-axis was along the main channel’s upstream region, the positive direction of the \( y \)-axis was along the tributary channel’s downstream region and the positive direction of

---

**Figure 1.** Simple model of flow characteristics in open-channel junctions.
The z-axis was upward from the x-y plane. The channel bed was horizontal and the channel walls were smooth. Using ADV with accuracy of ±1% of the average velocity, three-dimensional velocity components in various longitudinal and transverse sections were measured. A total of 2850 measurements were recorded for each discharge ratio. \( X^* \), \( Y^* \) and \( Z^* \) denote the dimensionless coordinates of the measurement points obtained from \( x/W_d \), \( y/W_d \) and \( z/W_d \), respectively, where \( W_d = 0.91 \text{ m} \) is the width of the main channel. Each of the three-dimensional velocity components was divided by \( U_d \), which is the average downstream velocity with a value of 0.628 m/s to yield the dimensionless velocity components \( u^* \), \( v^* \) and \( w^* \), which are the dimensionless velocity components in the \( x \), \( y \) and \( z \) directions, respectively. \( w^* \) and \( q^* \) are the width ratio and discharge ratio parameters, respectively. These parameters represent the ratio of tributary width to main channel width, and ratio of discharge upstream of the junction to the discharge downstream of the junction in the main channel. In Weber et al.’s (2001) experiment, \( W^* = 1 \) and \( q^* \) had different values. The different discharge ratios applied to the model are shown in Table 1. In this table, \( Q_{\text{main}} \) is the discharge upstream of the main channel and \( Q_{\text{tributary}} \) is the tributary discharge.

### Table 1. Experimental flow conditions.

| \( Q_{\text{main}} \) (m$^3$/s) | \( Q_{\text{tributary}} \) (m$^3$/s) | *q |
|----------------|----------------|---|
| 0.014          | 0.156          | 0.083 |
| 0.042          | 0.127          | 0.250 |
| 0.071          | 0.099          | 0.417 |
| 0.099          | 0.071          | 0.583 |
| 0.127          | 0.042          | 0.750 |
| 0.156          | 0.014          | 0.914 |

3. **Numerical model**

As a result of the limitation of experimental data (Weber et al., 2001) for channel widths and discharge ratios, the channel CFD model was simulated with Ansys-CFX. After comparing the numerical model results with experimental data for discharge ratios of 0.25 and 0.75 and validating them, the channel flow was simulated and analyzed for width ratios of 0.5, 0.75, 1.0, 1.25, 1.5, 1.75 and 2.0, and discharge ratios of 0.25, 0.5 and 0.75.

Fluid flow is governed by the following equations:

1. The continuity equation for incompressible fluid and steady state is:

   \[
   \frac{\partial}{\partial x} u_i = 0
   \]

2. The three Reynolds time-averaged Navier-Stokes momentum equations for steady condition and an incompressible fluid are:

   \[
   \frac{\partial}{\partial t} u_i + u_j \frac{\partial}{\partial x_j} u_i = -\frac{1}{\rho} \frac{\partial}{\partial x_i} P + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} - \frac{1}{2} \frac{\partial}{\partial x_j} \left( \rho \nu \frac{\partial u_j}{\partial x_i} \right)
   \]

   where \( i \) and \( j \) are 1, 2 and 3, \( u_i \) is the average flow velocity along the \( x \), \( y \) and \( z \) directions, \( \rho \) is fluid density, \( P \) is flow pressure and \( \nu \) is fluid kinematic viscosity. \( u_i' \) is the Reynolds stress tensors, which is unknown and therefore a turbulence model was used.

To find an appropriated turbulence model in open-channel junction modeling, three models, namely RNG \( k-\epsilon \), \( k-\epsilon \) and \( k-\omega \) were investigated and compared for \( q^* = 0.25 \). The results show that the RNG \( k-\epsilon \) and \( k-\omega \) models with \( RMSE \) of 0.187 and 0.182, respectively, were more accurate than the \( k-\epsilon \) model with \( RMSE \) of 0.233. Because of the close performance between \( k-\omega \) and RNG \( k-\epsilon \) as well as lower time consumption by the \( k-\omega \) model, Wilcox’s (1998) \( k-\omega \) model is used in this research.

The \( k-\omega \) turbulence model has been employed in various open-channel confluence studies (Huang et al., 2002; Neary, Sotiropoulos, & Odgaard, 1999; Ramamurthy, Qu, & Vo, 2007; Zhang, Xu, & Wu, 2009). In the present research, Wilcox’s (1998) \( k-\omega \) model is used. The relevant equations of this model include a transport equation on the turbulent kinetic energy \( k = u_i' u_i'/2 \) and specific dissipation \( \omega = \epsilon/k \), where \( \epsilon \) is turbulent dissipation to calculate
the modified eddy viscosity \( \nu_t \) and close the system through the Boussinesq hypothesis for turbulent flow.

\[
\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} - 2 \nu_t \delta_{ij}
\] (4)

where \( \delta_{ij} = (\partial_i U_j + \partial_j U_i)/2 \) is the mean rate of strain tensors. The \( \omega \) variable determines the scale of the turbulence, whereas the first variable, \( k \), determines the energy in the turbulence.

The \( k \)-equation is:

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{1}{\alpha} \left[ (\nu + \sigma \nu_T) \frac{\partial k}{\partial x_j} \right] - \beta k \omega + \tau_{ij} \frac{\partial U_i}{\partial x_j}
\] (5)

The \( \omega \)-equation is:

\[
\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \frac{1}{\alpha} \left[ (\nu + \sigma \nu_T) \frac{\partial \omega}{\partial x_j} \right] + \frac{\omega}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - \beta \omega^2
\] (6)

The five constants assume standard values of: \( \beta^* = 0.09 \), \( \alpha = 5/9 \), \( \beta = 3/40 \), \( \gamma = 1/2 \), \( \sigma^* = 1/2 \) (Wilcox, 1998).

The computational mesh was generated in such a way that the length, width and height of the channel were divided into 10, 5 and 2.5 cm, respectively. To focus on postjunction flow and investigate its effect on upstream and downstream flow, a finer mesh was used. To accomplish this, the area around the confluence of the two streams had 2.5 cm grid size in the longitudinal direction. Figure 3 shows a two-dimensional view of the final mesh. Since rectangular elements were used, the element size, especially in the interface, was changed gradually. The elements were also checked for overlapping. Table 2 presents the details of mesh generation in different channel conditions.

Table 2. Details of the generated mesh in the numerical models.

| CFD models | Number of nods | Number of elements |
|------------|----------------|--------------------|
| \( q^* = 0.25 \) | 9,76,277 | 10,46,707 |
| \( q^* = 0.5 \) | 9,76,277 | 10,46,707 |
| \( q^* = 0.75 \) | 9,76,277 | 10,46,707 |
| \( W^* = 0.5 \) | 11,24,206 | 20,87,674 |
| \( W^* = 0.75 \) | 9,06,195 | 10,71,579 |
| \( W^* = 1 \) | 9,76,277 | 10,46,707 |
| \( W^* = 1.25 \) | 9,98,421 | 12,11,690 |
| \( W^* = 1.5 \) | 14,48,090 | 28,74,732 |
| \( W^* = 1.75 \) | 15,79,156 | 30,55,242 |
| \( W^* = 2 \) | 16,15,318 | 30,52,653 |

Mean velocity and flow depth were imposed as channel inlet boundary conditions. Input velocity had a similar value in all models, but input depth had different values depending on the width and discharge ratios. Static pressure was imposed as an outlet boundary condition. Downstream depth had a constant value in all models, which was similar to the experimental model. No-slip conditions and smooth roughness were imposed on the walls, which were similar to the experimental channel. Finally, to govern the free surface condition, an opening boundary condition was imposed on the model. The Volume of Fluid (VOF) method was used to calculate the water level in the process of iteration (Chen, Dai, & Liu, 2002a; Chen, Chau, & Kouh, 2002b; Hirt & Nichols, 1981; Liu & García, 2008; Yang et al., 2013).

After specifying the type of mesh and boundary condition, the numerical model was initiated to solve the two phases of flow variables (water + air) by extrapolating from the interior solution domain. The solution process stopped automatically when all normalized residuals of the governing equation reached \( 10^{-7} \).
4. Validation

To verify the numerical results and evaluate the reliability of using them where experimental data were not available, it was necessary to validate the proposed numerical model. To do this, the simulated and experimental results were compared and the simulated error was estimated. The experimental results of Weber et al. (2001) served to validate the numerical simulation.

Figure 4 illustrates a comparison between the measured and simulated $u^*$ values. The horizontal axis represents the dimensionless width of the channel, $Y^*$, and the vertical axis represents the dimensionless height of the channel, $Z^*$. In this figure, the velocity patterns of flow were compared for two discharge ratios of $q^* = 0.25$ and 0.75. The comparisons were made for $X^* = -1, -2, -3$ and $-5$ and longitudinal cross sections of $Y^* = 0.25, 0.5$ and 0.75. Also according to this figure, by increasing the discharge ratio the simulated accuracy increased because of the reduction in tributary channel flow effect and reduction in the separation zone as well as recirculation flow. Thus, compared to $q^* = 0.25$, the simulation was more accurate for $q^* = 0.75$.

After the confluence of the main and tributary flows, due to having eliminated the downstream wall pressure of the tributary flow and the difference between the momentums of the two flows, the tributary flow tended to deviate towards the inner wall of the main channel. With increasing tributary flow velocity, the deviation increased too. The separation zone developed in the vicinity of the inner wall of the main channel downstream of the junction, reducing...
Figure 5. Experimental (left column), numerical (middle column) and prediction error % (right column) contours of dimensionless velocity component along the flow direction for a discharge ratio of 0.25.

the effective width of the channel which increased the flow velocity. Therefore, by increasing the size of the separation zone, the maximum velocity increased as well (Figures 5 and 6).

The velocity contour at \( q^* = 0.25 \) and at the cross-sections of \( X^* = -2, -3, -5 \) are demonstrated in Figure 5. This figure shows that the maximum separation zone occurred at close to \( X^* = -2 \). The negative velocity near the inner wall represents backflow and the development of recirculation flow in this area while the maximum velocity in the opposite wall represents a contraction zone that formed. The reduced velocity near the inner wall and increased velocity near the outer wall had a gradual trend towards average velocity along the channel. It can be understood from 5 and 6 that by moving in the longitudinal main channel direction, the difference between the velocity magnitudes of the separation and contraction zones was reduced, and finally for \( X^* = -5 \), the confluence effect was very low and the flow nearly reached the developed flow.

Investigating Figures 4 to 6 makes it clear that because of the separation zone effect on both discharge ratios of 0.25 and 0.75, the maximum difference between measured and simulated results occurred at about \( Y^* = 0.25 \) and the maximum error was calculated as 33%. For both discharge ratios, the error was maximum at about \( X^* = -2 \) where the separation zone had the largest width. The maximum simulation error decreased by moving up along the channel and reduced to about 20% for \( X^* = -5 \). In general, considering the fact that the approximate error was about 7%, the numerical model was found to be suitable.

5. General flow pattern for various width ratios (W*)

5.1. \( u^* - v^* \) vector field

In Figure 7 the velocity vectors in free surface with different width ratios are compared. Increasing the width ratio while keeping the discharge constant, reduced the velocity in the tributary compared to the main channel. A decrease in tributary momentum decreased the size of the separation zone. According to the figures, the length of the separation zone was maximum at \( W^* = 0.5 \) and minimum at \( W^* = 2.0 \). The angle of tributary flow entry also reflects the effect of width ratio on the momentum difference between the main and tributary flows. At lower width ratios, due to the effect of high-velocity tributary flow, the main flow deviation from its direct path increased. As a result, the tributary flow pushed the main flow towards the outer wall of the main channel. Figure 7 shows that increasing the width ratio led to a decrease in the velocity magnitude in the tributary channel. A decrease in tributary velocity led to a decrease in the main channel’s separation zone. The contraction zone velocity reduced as the separation zone became smaller. At lower width ratios, the flow became uniform at farther points from the beginning of the main channel’s downstream area.
Figure 6. Experimental (left column), numerical (middle column) and prediction error % (right column) contours of dimensionless velocity component along the flow direction for a discharge ratio of 0.75.

Figure 7. Plan view of vector fields in numerical models near the surface ($Z^* = 0.22$) with $q^* = 0.5$. 
5.2. $u^*-w^*$ vector field

Figure 8 shows the behavior of flow near the inner and outer walls downstream of the main channel. The tributary’s location is represented by a dashed line in each figure. Circulating flow developed at $W^* = 0.5$ and $W^* = 1.0$, and disappeared at $W^* = 2.0$ (left column). Near the inner wall, just at the beginning of the main channel’s downstream area, the vector size decreased, which represented a decrease in flow velocity in the separation zone. Comparing the velocity vectors near the inner wall showed that the length of the separation zone decreased with an increase in tributary width. Figure 8 demonstrates that the tributary flow’s entry into the main channel caused the main flow to move to the bed. This effect was intensified near the inner wall. The flow was more uniform near the outer wall compared to the inner wall. Also, the flow velocity was higher near the outer wall. This is because the formation of a separation zone reduced the effective width of the channel, which in turn led to an increase in velocity in the region opposite the separation zone. A decrease in the size of the separation zone following an increase in width ratio decreased the maximum velocity in the contraction zone near the outer wall. In all cases, after the tributary flow entered the main flow, the depth of the mixed flow decreased at the beginning of the main channel’s downstream region. By increasing the width ratio and decreasing the tributary flow velocity, the effect of the tributary channel decreased.

6. The effects of width and discharge ratio on flowmeter accuracy

Flowmeters are accurate and practical instruments used to measure the average velocity in channels. They are usually installed in the middle of a channel’s cross section. Flowmeter measurement points are located on a line passing through the center of the channel’s cross section perpendicular to the bed. A flowmeter calculates the average of the measured values on its body line to obtain the average velocity in the flow direction, $U_f$, $U_f$ is multiplied by the wet area of the cross section to give the discharge ($Q_f$). Flowmeters are usually erroneous in channels with junctions. High turbulence, flow deviation and rapid changes in velocity at the confluence of two streams, lead to considerable differences between the discharge measurement by the flowmeter ($Q_f$) and real-average discharge of the channel ($Q_m$) in different locations. In this study, flowmeter accuracy in an open channel junction was investigated for various widths and discharge ratios. It should be mentioned that the average velocity and flowmeter velocity extracted from an open-channel junction was used for model simulation. To do this, the following two steps were taken. First, the average velocity in various sections was obtained ($Q_m$). Second, by assuming a flowmeter in the middle of the channel, the line average velocity was obtained ($Q_f$). The measurement error of a flowmeter installed in the middle of the cross section of an open channel with a $90^\circ$ junction was investigated in the following conditions: (i)
width ratios of 0.5, 0.75, 1.0, 1.25, 1.5, 1.75 and 2.0 with a discharge ratio of 0.5, (ii) discharge ratios of 0.25, 0.5 and 0.75 with $W^* = 1$.

Figure 9 shows the differences between $Q_m$ and $Q_f$ for various width ratios. Here, the horizontal axis indicates the nondimensional longitudinal direction of the main channel ($X^*$) and the vertical axis indicates the flow discharge. An increase in the tributary channel’s width decreased tributary flow momentum. As a result, the flow entered the confluence zone with less force, the turbulence decreased in the postjunction zone, and the flowmeter accuracy increased. It was observed that when $W^*$ was greater than 1.5, the difference between $Q_m$ and $Q_f$ became negligible over a significant length of the channel.

As seen in Figure 9, at low width ratios ($W^* < 1$), the tributary flow momentum was higher due to the lower tributary width compared to that of the main channel. Therefore, on account of the wider separation zone, a significant difference existed between flowmeter discharge measurements and real average discharge. Figure 9 demonstrates that the difference between the flowmeter and mean discharges for $W^*$ of 0.5, 0.75 and 1 was negligible at $X^*$ of $-10$, $-7$ and $-5$, respectively. The maximum difference between the two values of $Q_m$ and $Q_f$ was observed at $W^* = 0.5$, which was equal to 73%, and the minimum difference was observed at $W^* = 2.0$, which was equal to 5%. Considering Figure 9, it can be concluded that in the range of width ratios of 0.5 to 2 investigated in this study, flowmeter measurement error is negligible for about $X^* < -10$.

Another important factor in evaluating flowmeter accuracy in open channel junctions is discharge ratio (Figure 10). Higher discharge ratio values led to lower tributary channel discharges and more accurate flowmeter measurements. The maximum error decreased from 59.5% to 13.7% when $q^*$ varied from 0.25 to 0.75. By increasing the discharge ratio, the velocity in the tributary decreased. Therefore, due to the low tributary flow momentum, the main flow was dominated by the tributary flow and did not deviate much from its direct path, which led to a decrease in separation zone size. At $q^* = 0.75$, the flowmeter error was nearly constant along the flow direction in the main channel. The average error of the flowmeter measurements was minimal at a discharge ratio of 0.75, which equals about 6%.

Table 3 shows the location of maximum flowmeter error at various width ratios ($W^*$). Evidently, with an increase in $W^*$, the location of peak error approached the confluence zone.

Table 4 shows the locations of maximum flowmeter error for various discharge ratios. At discharge ratios of 0.25 and 0.5, the location of flowmeter maximum error was $X^* = -1.65$. Therefore, it can be concluded that the area with more turbulence and a wider separation zone was similar for those two discharge ratios. By increasing $q^*$ from 0.25 to 0.75, the location of peak error approached the confluence zone due to severe decline in flowmeter error from $q^* = 0.25$ to $q^* = 0.75$.

7. Conclusion
In this study, to investigate the width and discharge ratio effect on the flow pattern and flowmeter measurement accuracy, a CFD model was simulated and verified with the experimental study results of Weber et al. (2001). To accomplish this, a CFD model was simulated in a 90°
open-channel junction with a width ratio of 1 and discharge ratios of 0.25 and 0.75. The model was verified empirically. Following CFD verification, the numerical model was simulated at width ratios of 0.5, 0.75, 1, 1.5, 1.75, and 2 and discharge ratios of 0.25, 0.5 and 0.75. The width and discharge ratio effects on flow pattern in an open-channel junction were also investigated. Another objective of this study was to examine the effect of width and discharge ratio on flowmeter accuracy. To do this, using CFD modeling an assumptive flowmeter was considered in the middle of the main channel and the line average velocity of the flowmeter was compared with the real average velocity of the section considered. The results showed that the numerical model was appropriate to simulate open-channel junction flow. The velocity vector results for various width ratios indicated that with increasing the width ratio, the entry angle of tributary flow to the main flow decreased. In addition, the size of the separation zone decreased with an increase in width ratio.

The entry of tributary streams to the main flow caused the main flow to deviate towards the bed, thus decreasing the depth of mixed flow just at the beginning of the main channel’s downstream region. This reduction in depth was intensified by increasing the width ratio. The flowmeter measurement accuracy investigation showed that due to a decreasing tributary flow effect on the main channel flow, increasing the width and discharge ratio led to a reduced effect of the separation and contraction zones, and flowmeter accuracy error decreased as well. In addition, by moving up along the main channel in the downstream direction, the tributary channel flow decreased and flow developed again. With the development of flow, flowmeter accuracy in mean flow velocity measurement augmented. Investigating flowmeter accuracy in open-channel junctions with various confluence angles and combining the results with this study (which investigated flowmeter accuracy for various width and discharge ratios) could be useful to gain comprehensive, practical information for using flowmeters in open-channel junction velocity measurements.

References

Baghalian, S., Bonakdari, H., Nazari, F., & Fazli, M. (2012). Closed-form solution for flow field in curved channels in comparison with experimental and numerical analyses and artificial neural network. Engineering Applications of Computational Fluid Mechanics, 6(4), 514–526.

Best, J. L., & Reid, I. (1984). Separation zone at open-channel junctions. Journal of Hydraulic Engineering – ASCE, 110(11), 1588–1594.

Bonakdari, H., Baghalian, S., Nazari, F., & Fazli, M. (2011). Numerical analysis and prediction of the velocity field in curved open channel using artificial neural network and genetic algorithm. Engineering Applications of Computational Fluid Mechanics, 5(3), 384–396.

Bonakdari, H., Larrarte, F., & Joannis, C. (2007). Effect of a bend on the velocity field in a circular sewer with free surface flow. Proceeding of 6th International Conference on Sustainable Techniques and Strategies in Urban Water Management, Lyon, France.

Bonakdari, H., Lipeme-Kouyi, G., & Wang, X. (2011). Experimental validation of CFD modeling of multiphase flow through open channel confluence. World Environmental and Water Resources Congress 2011: Bearing Knowledge for Sustainability. Palm Springs, CA.

Bonakdari, H., Zinatizadeh, A., Tahershamsi, A., Shahrrezaei, A., & Levacher, D. (2009). Effect of the angle of alignment deviation on the velocity field in down stream of the bends. 33nd Congress of IAHR, the International Association of Hydraulic Engineering & Research. Vanvouver, British Columbia, Canada.

Bonakdari, H., & Zinatizadeh, A. A. (2008). How can computational fluid dynamics improve measuring in real sewers? 6th International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering. Prague, Czech Republic, Technical University in Prague.

Bonakdari, H., & Zinatizadeh, A. A. (2011). Influence of position and type of Doppler flow meters on flow-rate measurement in sewers using computational fluid dynamic. Flow Measurement and Instrumentation, 22(3), 225–234.

Brabrook, K. F., Lane, S. N., & Richards, K. S. (2000). Numerical simulation of three-dimensional, time-averaged flow structure at river channel confluences. Water Resources Research, 36(9), 2731–2746.

Chau, K., & Jiang, Y. (2001). 3D numerical model for Pearl River Estuary. Journal of Hydraulic Engineering, 127(1), 72–82.

Chau, K., & Jiang, Y. (2004). A three-dimensional pollutant transport model in orthogonal curvilinear and sigma coordinate system for Pearl river estuary. International Journal of Environment and Pollution, 21(2), 188–198.

Chen, Q., Dai, G., & Liu, H. (2002a). Volume of fluid model for turbulence numerical simulation of stepped spillway over‐flow. Journal of Hydraulic Engineering, 128(7), 683–688.

Chen, Y. J., Chau, S. W., & Kouh, J. S. (2002b). Application of two-phase fluid approach for free-surface ship flow simulation. Journal of the Chinese Institute of Engineers, Transactions of the Chinese Institute of Engineers, Series A, 25(2), 179–188.

Hager, W. H. (1987). Discussion of “Separation Zone at Open-Channel Junctions” by James L. Best and Ian Reid (November, 1984). Journal of Hydraulic Engineering, 113(4), 539–543.

Hilgenstock, A., & Ernst, R. (1996). Analysis of installation effects by means of computational fluid dynamics - CFD vs experiments? Flow Measurement and Instrumentation, 7(3–4), 161–171.

Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics, 39(1), 201–225.

Huang, J., Weber, L. J., & Lai, Y. G. (2002). Three-dimensional numerical study of flows in open-channel junctions. Journal of Hydraulic Engineering, 128(3), 268–280.

Hughes, A. W., Longair, I. M., Ashley, R. M., & Kirby, K. (1996). Using an array of ultrasonic velocity transducers to improve the accuracy of large sewer mean velocity measurements. Water Science and Technology, 33, 1–12.

Koelling, C. (1996). SIMK - A new finite element model significantly improves the accuracy of flow measurements in sewers. 7th International Conference on Urban Storm Drainage, 2, 665–670.

Kumar Gurram, S., Karki, K. S., & Hager, W. H. (1997). Subcritical junction flow. Journal of Hydraulic Engineering, 123(5), 447–454.
Larrarte, F., Bernard Bardiaux, J., Battaglia, P., & Joannis, C. (2008). Acoustic Doppler flow-meters: A proposal to characterize their technical parameters. *Flow Measurement and Instrumentation, 19*(5), 261–267.

Liu, T., & Yang, J. (2013). Three-dimensional computations of water-air flow in a bottom spillway during gate opening. *Journal of Hydraulic Research, 8*(1), 104–115.

Liu, X., & Garcia, M. H. (2008). Three-Dimensional numerical model with free water surface and mesh deformation for local sediment scour. *Journal of Waterway, Port, Coastal and Ocean Engineering, 134*(4), 203–217.

Mignot, E., Bonakdari, H., Knothe, P., Lipeme Kouyi, G., Bessette, A., Rivière, N., & Bertrand-Krajewski, J.-L. (2012). Experiments and 3D simulations of flow structures in junctions and their influence on location of flowmeters. *Water Science and Technology, 66*(6), 1325–1332.

Modi, P. N., Ariel, P. D., & Dandekar, M. M. (1981). Conformal mapping for channel junction flow. *Journal of the Hydraulics Division, ASCE, 107*(HY12, Proc. Paper 16763), 1713–1733.

Momplot, A., Bonakdari, H., Mignot, E., Lipeme Kouyi, G., Rivière, N., & Bertrand-Krajewski, J.-L. (2012). *Effects of computational meshes on hydrodynamics of an open channel junction flow using CFD technique*. Proceedings of the 9th International Conference on Urban Drainage Modelling, Belgrade, Serbia.

Neary, V. S., Sotiropoulos, F., & Odgaard, A. J. (1999). Three-dimensional numerical model of lateral-intake inflows. *Journal of Hydraulic Engineering, 125*(2), 126–140.

Ramamurthy, A. S., Qu, J., & Vo, D. (2007). Numerical and experimental study of dividing open-channel flows. *Journal of Hydraulic Engineering, 133*(10), 1135–1144.

Shakibainia, A., Tabatabai, M. R. M., & Zarzati, A. R. (2010). Three-dimensional numerical study of flow structure in channel confluences. *Canadian Journal of Civil Engineering, 37*(5), 772–781.

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

Wang, X., Wang, X., Lu, W., & Liu, T. (2007). Experimental study on flow behavior at open channel confluences. *Frontiers of Architecture and Civil Engineering in China, 1*(2), 211–216.

Webber, N. B., & Greated, C. (1966). *An investigation of flow behaviour at the junction of rectangular channels*. ICE Proceedings. Thomas Telford.

Weber, L. J., Schumate, E. D., & Mawer, N. (2001). Experiments on flow at a 90° open-channel junction. *Journal of Hydraulic Engineering, 127*(5), 340–350.

Wilcox, D. C. (1998). *Turbulence modeling for CFD*. La Canada, CA: DCW Industries.

Wu, C., & Chau, K. (2006). Mathematical model of water quality rehabilitation with rainwater utilisation: A case study at Haigang. *International Journal of Environment and Pollution, 28*(3), 534–545.

Xie, Z., Lin, B., & Falconer, R. A. (2013). Large-eddy simulation of the turbulent structure in compound open-channel flows. *Advances in Water Resources, 53*, 66–75.

Yang, H. C., & Chang, F. J. (2005). Modelling combined open channel flow by artificial neural networks. *Hydrological Processes, 19*(18), 3747–3762.

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. *Advanced Mechanical Engineering, 1*–10. Retrieved from http://dx.doi.org/10.1155/2013/860431

Zhang, T., Xu, W. I., & Wu, C. (2009). Effect of discharge ratio on flow characteristics in 90° equal-width open-channel junction. *Journal of Hydrodynamics, 21*(4), 541–549.

Zhao, C. H., Zhu, D. Z., & Rajaratnam, N. (2008). Computational and experimental study of surcharged flow at a 90° combining sewer junction. *Journal of Hydraulic Engineering, 134*(6), 688–700.