ORIGINAL ARTICLE

A study on spatial variation of water flow at confluence connected to non-orthogonal channels

Yong-Wook Jeong1 · Woochang Jeong2

Received: 28 December 2020 / Accepted: 14 March 2022 / Published online: 15 April 2022
© The Author(s) 2022

Abstract
Most studies on the flood flow characteristics at a confluence focus on channels connected orthogonally or at right angle, but studies on non-orthogonally connected channels remain limited. In this study, hydraulic-model experiments and numerical simulations are conducted to analyze the spatial variation of water flow in and around a confluence connected to non-orthogonal channels. Comparison of the measured and simulated water depth distributions in and around the confluence indicates that the results are in relatively good agreement. In the experiment where the angle between two upstream channels is 45°, the water flow pattern in and around the confluence corresponds approximately to Type I proposed by (Mignot et al. J Hydraul Res 46:723–738, 2008). However, it was found that there is no any flow type to correspond to the water flow pattern measured in the case of the angle of 135°. For analyzing the variation of the water depth in and around the confluence with inflow, numerical simulation is performed by setting the inflow ratio of the two inlet channels to one, three, and six, respectively.

Keywords Hydraulic-model experiment · Non-orthogonal channels · Confluence · Water flow · Numerical simulation

Introduction

In most cities, crossroads connect three or four roads either orthogonally or non-orthogonally, as shown in Fig. 1a. In general, when the flood entering a crossroad is relatively less and the road gradient is not steep, the flow in a crossroad is characteristically subcritical. However, complex flood flow involving subcritical as well as supercritical flows can also be observed, as shown in Fig. 1b.

Most of the existing studies on urban flooding have adopted experimental or numerical methods that mainly consider road networks as water channels, instead of observing the actual flood flows moving through roads distributed across urban areas. For example, Best and Reid (1984), Weber et al. (2001), and Neary et al. (1999) experimentally studied the flow characteristics around a confluence connected to three channels under the subcritical conditions. These studies primarily analyzed the flow structure at a confluence, such as re-circulation flow, two-dimensional flow, and flow separation and contraction. Bowers (1966) suggested that hydraulic jumps develop in the inflow channels of a confluence zone under supercritical conditions depending on the geometric shape of the confluence and the water inflow. Schwalt and Hager (1995) conducted a study to identify the major characteristics of surface profiles formed when supercritical flows develop at a confluence connected to three channels. Rivière and Perkins (2004) examined the characteristics of supercritical flows at a confluence to which three channels are connected at 90°, and Rivière et al. (2011) investigated experimentally subcritical flow in an intersection formed by four similar orthogonal channels with two inflows and two outflows for a wide range of experimental conditions. Nania et al. (2004) investigated the characteristics of supercritical flows at a confluence with two inflowing channels and two outflowing channels connected at a right angle, and divided the flows into two types according to the location of hydraulic jumps (Type I for a normal hydraulic jump at each inflowing channel; Type II for a normal hydraulic jump at either inflowing channel and an oblique hydraulic jump within the confluence). Mignot et al. (2008) conducted an experimental and numerical studies on the supercritical flow...
flow characteristics in and around a confluence where four oblique channels are connected to each other at a right angle, and accordingly divided the flows into four types (Types I, II-1, II-2, and III) based on the location of normal hydraulic jumps formed at inflowing channels, and oblique hydraulic jumps formed within the intersection (Fig. 2).

The study’s applicability was also verified through a comparison of the experimental results and a simulation in and around the confluence with Rubar 20 (Paquier 1995; Mignot 2005), a two-dimensional finite volume model based on the shallow water equation and second-order MUSCL technique. Abderrezzak et al. (2011) conducted an experimental study to...
investigate the characteristics of dividing critical flows in a 90° open-channel junction formed by three horizontal equal-width channels, and found a relationship between the discharge division ratio and the tailwater Froude number. Recently, Rivière et al. (2014) examined experimentally trans-critical flows in three- and four-channel intersections and proposed empirical correlations derived from the experimental data for the flow distribution in three-channel intersections and four-channels intersections with one or two critical sections.

Shetter and Murthy (1996) numerically analyzed the flow characteristics at a confluence connected to three channels under the subcritical conditions using a two-dimensional numerical model based on the $k-\varepsilon$ turbulence technique and shallow water equation. In addition, Khan et al. (2000) applied a simple turbulence model, based on the mixing length formula, and the two-dimensional model, based on the shallow water equation, for numerically analyzing the flow characteristics at a bifurcation and a confluence of three channels under the subcritical condition. To verify applicability to subcritical conditions at a confluence, Huang et al. (2002) used the three-dimensional model based on the $k-\varepsilon$ turbulence technique for comparative analysis with the experimental results of Shumate (1998). Ghostine et al. (2009) utilized the experimental results reported by Mignot et al. (2006) to verify the applicability of the two-dimensional finite element model based on the Runge–Kutta Discontinuous Galerkin (RKDG) technique by comparing the simulation results from Rubar 20 and FLUENT, a three-dimensional model. Jeong et al. (2010) recently numerically analyzed on the flood flow characteristics at a confluence symmetrically connected to four channels using the two-dimensional well-balanced HLLC finite volume model.

Very recently, Mignot et al. (2019) reviewed the 45 existing studies available on urban flooding based on laboratory experiments to help computational and laboratory modelers.

As apparent, most studies on the flood flow characteristics at a confluence focus on channels connected orthogonally or at right angle, but studies on non-orthogonally connected channels remains limited. In this study, a hydraulic-model experiment and a numerical simulation using ANSYS CFX (ver. 14) (2013), which is the commercial three-dimensional CFD model, are performed for investigating the characteristics of flood flow in and around a confluence connected non-orthogonally to four channels. The simulated results are verified by comparison with the results of the hydraulic-model experiment, and the flow characteristics at a confluence are analyzed under various inflow scenarios.

**Hydraulic-model experiment**

**Experimental methods and conditions**

A hydraulic model was used to analyze the flood flow characteristics of channels and a confluence composed of acrylic as shown in Fig. 3, in order to observe the water flow and includes a pump to provide water flow and an electronic flowmeter installed between the water tanks connected to the end of each channel for controlling the flow.

Flow control valves and electronic flowmeters (WTM-1000, range: 0.03 – 10 m$^3$/s, accuracy: $\pm 0.5\%$ and repeatability: $\leq 0.1\%$) were used to control the water flow and generate flow into the channels. Then, after the flows were stabilized (within approximately 50 s in this experiment), the ultrasound water level meter (UC500-20GM, range: 0.838 to 30.480 cm, accuracy: better than 0.2% of range) shown in Fig. 4a was used to measure the water depth within a series of the grids around the confluence. The grid system included 1447 (3 $\times$ 3 cm) diamond shape grids, as illustrated in Fig. 4b. After the water flow in the confluence and channels reaches steady state, the water depth was measured 5 times for each grid and its values presented hereafter are the average of 5 measurements.

The widths of the four channels connected to the confluence were identical 0.3 m each. The lengths of the two horizontally connected channels were 2.0 m each, and those of the two channels connected non-orthogonally at an inclination of 45° to the horizontal channels were 2.3 m each.

In Fig. 5, ①, ②, ③ and ④ represent the channels connected in the confluence. Two cases were considered in the experiment. In Case I, channels ① and ② were the inflow channels and the angle between two upstream channels was 45°; channels ③ and ④ were the outflow channels (Fig. 5a). In Case II, channels ① and ④ were the inflow channels and the angle

![Fig. 3 Set-up of the hydraulic-experiment model](image-url)
between two upstream channels was 135°; channels ② and ③ are the outflow channels (Fig. 5b).

$Q_1$ and $Q_2$ are the inflows at the inflow channel boundaries, and $h_1$ and $h_2$ refer to the water depths corresponding to each inflow channel boundary. $Q_3$ and $Q_4$ are the outflows through the outflow channel boundaries, and $h_3$ and $h_4$ are the water depth corresponding to each outflow channel boundary.

At the beginning of the experiment, there is no water flow in all channels. When the water-surface in the water tank connected to the inflow channel reaches the bottom of the inflow channel, the water starts to flow in the inflow channel. The inflow was controlled by the electronic flowmeter connected to the water tank, and the water depth at the inflow channel boundary was measured by an attached ruler. The outflow was measured by considering the volume of water filled ($\Delta V = l \times w \times h_2 - l \times w \times h_1$) ($l$ and $w$ are the length and width of water tank, respectively) in the water tanks for a given time interval ($t_2 - t_1$) (Fig. 6).

Table 1 summarizes the inflow conditions in the two inflow channels and the corresponding water depths for each case. In this study, the experiment was conducted by setting the inflows ratio of the two inflow channels to one (flow ratio = 1) and three (flow ratio = 3).

**Experimental results and analyses**

Figure 7 depicts the spatial variation of water depth measured around the confluence for Case I after the flow reaches steady-state. In Case I-1, the water flowing through the two inflow channels approaches the confluence with a depth of 3.1 cm. After flowing into the confluence, the water depth begins to decrease, with the depth around the mouth of channels ③ and ④ connected to the confluence decreasing to an average of 1.0 cm. When the water into the confluence...
reaches at the point adjoining channels ③ and ④ meet (●), the flow is separated and an oblique hydraulic jump is appeared. This type of water flow around the confluence corresponds approximately to Type I proposed by Mignot et al. (2008).

For Case I-2, after flowing through channel ① at a depth of 5.0 cm and channel ② at a depth of 3.2 cm, the water flows into the confluence with a significantly lower depths of 1.0 and 1.4 cm at the exits of the channels ③ and ④, respectively (Fig. 6.I-1). After the flow is stabilized, channels ① and ② have a similar depth distribution (5.0 cm on an average). This result can be attributed to the backwater effect: the larger quantity of water flowing into channel ② moves faster than that flowing into the channel ①, causing the water flowing through channel ① to move in a direction opposite to the flow without appropriately passing through the confluence (Fig. 6.I-2).

Figure 8 displays the spatial variation of the water depth measured around the confluence for Case II. In Case II-1, the water quantity flowing into channels ① and ④ is the same at 100 l/min, and the depth of water moving through these two inflowing channels is approximately 3.1 cm on reaching the confluence. However, the water depth distribution at the confluence in Case II-1 exhibits a considerably different trend from that of Case I-1. After the water flows into the confluence via the two channels, the water depth does not decrease; instead, it maintains the water depth before inflow. The water flow into the confluence is divided after reaching the point (●) adjoining channels ② and ③, and moves into each channel with oblique hydraulic jumps. This type of water flow around the confluence does not correspond to any flow type proposed by Mignot et al. (2008). The rapid decrease in the water depth (from 3.1 to 1.7 cm) in channels ② and ③ after passing the confluence adopted a tilted shape because of the continual supply of water from the point of divided flow.

In Case II-2, because of the larger quantity of water in channel ④ flowing toward the confluence at a fast pace, the

| Case I-       | $Q_1$ (①) (l/min) | $h_1$ (m) | $Q_2$ (②) (l/min) | $h_2$ (m) | $Q_2/Q_1$ |
|--------------|-------------------|-----------|-------------------|-----------|------------|
| 1            | 100               | 0.0209    | 100               | 0.0209    | 1.0        |
| 2            | 100               | 0.0209    | 300               | 0.0404    | 3.0        |

| Case II-     | $Q_1$ (①) (l/min) | $h_1$ (m) | $Q_4$ (④) (l/min) | $h_4$ (m) | $Q_4/Q_1$ |
|--------------|-------------------|-----------|-------------------|-----------|------------|
| 1            | 100               | 0.0209    | 100               | 0.0209    | 1.0        |
| 2            | 100               | 0.0209    | 300               | 0.0404    | 3.0        |
water flowing into the channel ① fails to pass the confluence and demonstrates the backwater effect, as observed in Case I-2, with the water moving toward the flow direction. The water depth distribution is observed to be greater in channel ① with the reduced inflow compared to channel ④ with a greater inflow at the initial stage. In Case I-2, the inflow to channels ① and ④ is from the same direction (45°), but in Case II-2, the direction of inflow into channels ① and ④ is almost opposite directions (135° apart). This causes greater disturbance in the flows at the confluence, and thereby, a greater water depth distribution. The flow at the confluence heads toward channel ① with a relatively lesser water quantity, contributing to greater water depth. In addition, in Case I-2, an area with rapidly increasing water depth (the area marked with an oval) in channel ② in the immediate vicinity of the confluence is not observed.

Further, the results of the hydraulic-model experiments for Cases I and II are verified through numerical simulation. Table 2 shows the comparison between inflows and outflows based on the results of the hydraulic-model experiment for Cases I and II; the inflow and outflow agree relatively well in both cases.

### Numerical simulations

**ANSYS CFX model**

For the homogenous mixture of multiphase flows, such as a flow moving through an open channel with a mixture of air and water in a rectangular coordinate system, ANSYS CFX commercial model (ANSYS Inc., 2013) was applied in this study. This model has been widely applied to simulate various types of water flow in open channels, such as flow around a half-submerged cylinder in an open channel (Alamayreh et al. 2021), energy loss in steep open channels (Thappeta et al. 2021), design of contractions in supercritical flow open channels (Martinez and Niño 2020). The numerical scheme of this model is the finite volume method, which divides the computational domain into small cells for obtaining solutions by assigning a boundary condition to each cell, and the complete set of governing equations for an incompressible multiphase flow model is as follows:

#### Continuity equation

\[
\frac{\partial \rho_m}{\partial t} + \nabla \cdot \left( \rho_m \vec{V}_m \right) = 0
\]  

(1)

#### Momentum equation

\[
\frac{\partial (\rho_m \vec{V}_m)}{\partial t} + \nabla \cdot \left( \rho_m \vec{V}_m \otimes \vec{V}_m \right) = -\nabla p_m + \nabla \cdot \left( \vec{t} + \vec{f} \right)
\]  

(2)

#### Volume fraction transport equation

\[
\frac{\partial (\alpha_i \rho_i)}{\partial t} + \nabla \cdot \left( \alpha_i \rho_i \vec{V}_m \right) = 0
\]  

(3)

where \( \rho_m \) is the mixture density of the fluid, \( \vec{V}_m = [u, v, w] \) is the velocity vector, \( t \) is the time, \( p_m \) is the pressure, \( \vec{t} \) is the viscous stress tensor, \( \vec{f} \) is an external force such as buoyancy, \( \rho_i \) is the density of each fluid in the mixture and \( \alpha_i \) is the liquid-phase volume fraction ranging between [0, 1] for each liquid phase. If water and air are mixed, the value of \( \alpha_i \) becomes 0.5 at the boundary between the two fluids. If two fluids, such as water and air, are mixed with each other, the mixture density \( \rho_i = \sum_{n=1}^{2} \alpha_i \rho_n. \)

The numerical analysis methods provided by the ANSYS CFX model for analyzing the turbulence elements include the Reynolds-Averaged Navier–Stokes (RANS), Eddy-Viscosity method, RANS Reynolds-Stress method, and Eddy Simulation method (ANSYS Inc., 2013). In this study, the standard \( k - \varepsilon \) model, which is a type of RANS Eddy-Viscosity method, was chosen due to its simplicity in terms of its empirical parameters and its wide use in engineering applications (Matthews et al. 1998).

### Table 2 Comparison of inflow and outflow measured in the experiment

| Case I- | Inflow (l/min) | Outflow (l/min) | Absolute error (l/min) |
|--------|---------------|-----------------|------------------------|
|        | \( Q_1 \) | \( Q_2 \) | \( Q_1 + Q_2 \) | \( Q_3 \) | \( Q_4 \) | \( Q_1 + Q_2 \) | \( Q_3 \) | \( Q_4 \) | \( Q_1 + Q_2 \) |
| 1      | 100          | 100            | 200                    | 99.6         | 100.2         | 199.8      | 0.2        |
| 2      | 100          | 300            | 400                    | 189.0        | 211.2         | 400.2      | 0.2        |
| Case II- | Inflow (l/min) | Outflow (l/min) | Absolute error (l/min) |
|        | \( Q_1 \) | \( Q_4 \) | \( Q_1 + Q_4 \) | \( Q_5 \) | \( Q_2 \) | \( Q_1 + Q_4 \) | \( Q_5 \) | \( Q_2 \) | \( Q_1 + Q_4 \) |
| 1      | 100          | 100            | 200                    | 99.6         | 100.2         | 199.8      | 0.2        |
| 2      | 100          | 300            | 400                    | 161.4        | 237.6         | 399.0      | 0.1        |
Table 3  Comparison of outflows measured and simulated at two downstream channels boundaries

| Case I- | Measured (l/min) | Simulated (l/min) |
|--------|------------------|-------------------|
|        | $Q_{3M}$          | $Q_{4M}$          |
|        | $Q_{3S}$          | $Q_{4S}$          |
| 1      | 99.6             | 100.2             |
|        | 99.6             | 99.6              |
| 2      | 186.0            | 211.2             |
|        | 189.6            | 209.4             |
| Case II- | Measured (l/min) | Simulated (l/min) |
|        | $Q_{2M}$          | $Q_{3M}$          |
|        | $Q_{2S}$          | $Q_{3S}$          |
| 1      | 99.6             | 100.2             |
|        | 100.2            | 100.2             |
| 2      | 161.4            | 237.6             |
|        | 163.8            | 240.6             |

Model verification

For verifying the model applied in this study, the four different inflow conditions in Case I and II of the hydraulic-model experiments were simulated under an unsteady state condition in order to investigate the spatial variation of the water depth with time. The channels were presumed to be flat and without inclination; the grid system comprised 153,304 nodes and 621,634 cells. The cell type was tetrahedron and the maximum, minimum, and mean cell sizes were $4.5 \times 10^{-7}$, $2.4 \times 10^{-7}$, and $3.8 \times 10^{-7}$ m$^3$, respectively. The upstream boundaries of the two inflow channels $\odot$ and $\odot$ for Case I, and channels $\odot$ and $\odot$ for Case II in Fig. 4 were considered as the inflow boundary conditions. The downstream boundaries of the two outflow channels $\odot$ and $\odot$ for Case I, and channels $\odot$ and $\odot$ for Case II in Fig. 4 were considered as the open boundary conditions. In addition, the channel bottoms and side-walls were presumed to be flat and smooth. The downstream boundaries of the outflow channels were treated as open boundary condition. The total simulation time was set to 60 s, which is sufficient to reach steady state, and the time interval was set to 0.01 s. In this verification, time was set to 60 s, which is sufficient to reach steady state, treated as open boundary condition. The total simulation involved $L_1$ (absolute mean error), $L_2$ (root mean square error), and $L_\infty$ (maximum error), respectively, were applied. The estimated results are depicted in Table 4. As the results from these three error equations show significantly small values, it can be concluded that the results of the numerical model applied in this study are in agreement with the measured values.

\[
\frac{1}{n} \sum_{i=1}^{n} |\epsilon_{ij}| \quad (4)
\]

\[
\left( \frac{1}{n} \sum_{i=1}^{n} |\epsilon_{ij}| \right)^{1/2} \quad (5)
\]

\[
\frac{1}{h_{\text{max}}} |\epsilon_{ij}| \quad (6)
\]

Changes in water depth at the confluence with increasing inflows

For analyzing the changes in the water depth in and around the confluence with increased inflow, the results of Cases I-1, I-2, II-1, and II-2 are compared with those of the numerical simulation by setting the ratio between the inflow into the confluence, and then begins to slowly increase.

For a more quantitative comparison of the measured and simulated water depths, the three error Eqs. (4), (5), and (6) involving $L_1$ (absolute mean error), $L_2$ (root mean square error), and $L_\infty$ (maximum error), respectively, were applied. The estimated results are depicted in Table 4. As the results from these three error equations show significantly small values, it can be concluded that the results of the numerical model applied in this study are in agreement with the measured values.

\[
\frac{1}{n} \sum_{i=1}^{n} |\epsilon_{ij}| \quad (4)
\]

\[
\left( \frac{1}{n} \sum_{i=1}^{n} |\epsilon_{ij}| \right)^{1/2} \quad (5)
\]

\[
\frac{1}{h_{\text{max}}} |\epsilon_{ij}| \quad (6)
\]
Fig. 9  Water-surface profiles along channels for Case I

Fig. 10  Water-surface profiles along channels for Case II
The inflow conditions for Cases I-3 and II-3 are shown in Table 5.

Figure 11 compares the spatial variation of the water depth in Cases I-3 and II-3 at 60 s with those in Cases I-1 and -2 and II-1 and -2, respectively. As the inflows increase, the backwater effect in channel ① is enhanced, and the water depth in Case I increases more in channel ④ than in channel ③. The result is the same for Case II-3: when the inflows increases, the backwater effect strengthens in channel ①, and the water depth increases in channel ②. However, the increase in water depth in channel ③ is relatively limited.

Based on the results shown in Fig. 12, the spatially changing water depths along the centerline of each of the four channels around the confluence in Cases I and II are compared, as shown in Fig. 11. The water flowing through channels ① and ③ in Case I enters the confluence with an increased water depth (approximately 3.10 cm for Case I-1, 4.85 cm for Case I-2, and 6.35 cm for Case I-3); a rapid decrease in all these water depths occurs in the confluence in Case I. After the water flows through the confluence, a hydraulic jump is observed in Case I-2 farther from the confluence compared to the Case I-1 location (1.93 m for Case I-1 and 1.63 m for Case I-2); the scale is greater in Case I-2 (1.0 cm for Case I-1 and 1.5 cm for Case I-2).

In Case I-3, a smaller hydraulic jump is observed at a location similar to that of the hydraulic jump in Case I-2; however, a location within 0.6 m is associated with a gradually increasing water depth. As the inflow increases, the water depth in channels ① and ③ exhibits a clear increasing tendency. The water depth before the arrival of the flow at the confluence and within the confluence increases to a level similar to those in channels ① and ③.

The decreasing tendency of the water depth differs in the confluence; as the inflow increases, the degree to which the water depth reduces also increases (3.10 cm to 1.00 cm in Case I-1, 4.85 cm to 1.50 cm in Case I-2, and 6.35 cm to 2.35 cm in Case I-3). After the flow passes the confluence, a hydraulic jump occurs at a location increasingly farther from the confluence when the inflow increases, and there is an increase in the scale as well (1.0 cm for Case I-1, 1.5 cm for Case I-2, and 2.0 cm for Case I-3). Unlike in Cases I-1 and I-2, two hydraulic jumps are observed in Case I-3, which may contribute to the smaller hydraulic jumps in channels ① and ③.

In Case II, water flows through channels ① and ③ reaches the confluence at an increased water depth (the same as in Case I) as the inflow increases. In Cases II-1 and II-2, the water depth in the confluence remains almost constant and rapidly decreases immediately before passing the confluence. Further, the water depth falls to approximately 1.1 cm before increasing again. Compared to Case I, the scale of the hydraulic jumps is smaller (1.5 cm for Case II-1 and 3.25 cm for Case II-2), and they are located farther from the confluence as the inflow increases (1.35 m for Case II-1 and 1.15 m for Case II-2).

In Case II-3, the water depth increases up to the midpoint of the confluence before rapidly decreasing, and falls to a level similar to those in Cases II-1 and II-2 after passing the confluence. Although two hydraulic jumps are observed after the flow passes the confluence in channels ① and ② in Case I-3, smaller ones are found in channels ① and ③ in Case II-3.

As the inflow increases in channels ① and ②, the water depth in the confluence remains almost constant in Case II-1, but rapidly falls immediately before passing the confluence. In Cases II-2 and II-3, the water depth decreases before the flow reaches the confluence, and the surface profiles are convex-shaped in the confluence area. This phenomenon becomes increasingly evident when the inflow increases, and the water depth distribution following the flow movement

| Case | Along channels ①-② | Along channels ②-④ |
|------|------------------|------------------|
| L₁   | L₂   | Lₘ          | L₁   | L₂   | Lₘ          |
| I-1  | 0.04293 | 0.00221 | 0.07745 | 0.04719 | 0.00306 | 0.12954 |
| I-2  | 0.01306 | 0.00260 | 0.04340 | 0.02678 | 0.00130 | 0.11947 |
| II-1 | 0.02520 | 0.00060 | 0.08472 | 0.04117 | 0.00214 | 0.07703 |
| II-2 | 0.02613 | 0.00085 | 0.07004 | 0.05752 | 0.00352 | 0.09026 |

Table 5 Inflow conditions for Cases I-3 and II-3 (①, ② and ④: channel number)

| Case | Q₁ (l/min) | Q₂ (l/min) | Q₁/Q₂ |
|------|------------|------------|--------|
| I-1  | 100.0      | 600.0      | 6.0    |
| I-2  | 100.0      | 600.0      | 6.0    |
| II-1 | 100.0      | 600.0      | 6.0    |
through the confluence is almost consistent (1.5 cm for Case II-1, 3 cm for Case II-2, and 4.3 cm for Case II-3).

Table 6 shows the comparison of the maximum and minimum water depths in channels ① and ③, and in channels ④ and ② for Cases I and II. The results suggest that the maximum depth in channels ① and ③ increases in Cases I-1 and I-3 by two and three times, respectively, compared to Case I-1, and increases by a similar ratio in Case II. The maximum water depth in channels ④ and ② increases in Cases I-2 and I-3 by 1.5 times and 2 times, respectively, compared to Case I-1, and increases in Cases II-2 and II-3 by 2.2 times and 3.8 times, respectively, compared to Case II-1.

Table 6

| Case  | Channel ① | Channel ③ | Channel ④ | Channel ② |
|-------|------------|------------|------------|------------|
| I-1   | 1.5 cm     | 4.3 cm     | 2.5 cm     | 3.2 cm     |
| I-2   | 3 cm       | 4.8 cm     | 2.8 cm     | 3.4 cm     |
| I-3   | 4.3 cm     | 6 cm       | 3.5 cm     | 4 cm       |
| II-1  | 1.5 cm     | 4 cm       | 2.5 cm     | 3 cm       |
| II-2  | 3 cm       | 4.5 cm     | 2.8 cm     | 3.2 cm     |
| II-3  | 4.3 cm     | 5.5 cm     | 3.5 cm     | 4 cm       |

Fig. 11  Comparison of water-surface profiles along channels with increased inflow for Case I and II
Conclusions

In this study, the hydraulic-model experiment and the numerical simulation were performed in order to investigate the characteristics of flood flow in and around a confluence connected to channels aligned non-orthogonally, and the following results were obtained. The experimental and numerical results of water depth distribution in the confluence with increasing inflows were in relatively good agreement.

When the same inflows entered two channels, the water depth in the horizontal channel (channels ① and ③) immediately decreased after the flow entered the confluence (Case I); however, the water depth before the flow arrived at the confluence remained constant, when either of the two channels had an increasing inflow, and decreased rapidly before the flow passed the confluence (Case II). In addition, while a

![Fig. 12 Comparison of water-surface profiles along channels with increased inflow for Cases I and II](image)

![Table 6 Comparison of maximum and minimum water depths for Case I and II](image)

| Case     | Along channels ①-③ | Along channels ②-④ |
|----------|--------------------|--------------------|
|          | Max. water depth (m) | Min. water depth (m) | Max. water depth (m) | Min. water depth (m) |
| Case I   |                     |                    |                     |                    |
| 1        | 0.032               | 0.009              | 0.033               | 0.010              |
| 2        | 0.049               | 0.010              | 0.050               | 0.015              |
| 3        | 0.063               | 0.013              | 0.066               | 0.021              |
| Case II  |                     |                    |                     |                    |
| 1        | 0.029               | 0.013              | 0.029               | 0.013              |
| 2        | 0.047               | 0.013              | 0.046               | 0.028              |
| 3        | 0.062               | 0.012              | 0.059               | 0.049              |
clear hydraulic jump was observed with a rapidly increasing water depth after the flow passed the confluence in Case I, the water depth in Case II gradually increased, resulting in relatively smaller hydraulic jumps.

When the same quantity of water flowed into channels ⑦ and ⑨, the water depth rapidly decreased immediately after the flow reached the confluence. With increasing inflow, the water depth increased abruptly after the flow passes the confluence, producing increasingly large hydraulic jumps (Case I). In particular, two hydraulic jumps were observed in Case I-3. The water depth tended to be lower in Case II before the flow reached the confluence, and the surface profiles were convex shaped in the confluence area. This phenomenon was more evident when the inflow increased, and is associated with a consistent depth distribution after the flow passed the confluence.

The results of the hydraulic-model experiment and the numerical simulation of water flows at a channel confluence, where the channels are connected non-orthogonally at 45° and 135° have only limited applicability in terms of establishing flood prevention plans for urban areas. To resolve this limitation, it is necessary to perform hydraulic-model experiments and numerical analyses of structures having connecting channels with more diverse angles. Therefore, hydraulic-model experiments with connection angles of 22.5° and 67.5° are currently underway. In this study, the variations of flood flow around the confluence were only investigated physically and numerically. In the further study, the water velocity fields in and around the confluence area will be considered.

**References**

Abderrezak KEK, Lewicki L, Paquier A, Rivière N, Travin G (2011) Division of critical flow at three-branch open-channel intersection. J Hydraul Res 49(2):231–238. https://doi.org/10.1080/00221686.2011.558174

Alamayreh MI, Fenocchi A, Petaccia G, Sibilla S, Persi E (2021) Numerical analysis of fluid flow dynamics around a yawed half-submerged cylinder inside an open channel. J Hydrolyn. https://doi.org/10.1007/s42241-021-0005-0

ANSYS (2013) CFX-solver theory guide. Release 13.0. ANSYS Inc.; 275 Technology Drive, Canonsburg, PA 15317, USA, 390

Best JL, Reid I (1984) Separation zone at open-channel junctions. J Hydraul Eng 110:1588–1594. https://doi.org/10.1061/(ASCE)0733-9429(1984)110:11(1588)

Bowers CE (1966) Study of open channel junctions. Technical paper. Part V Hydraul Model Stud Whiting Field Naval Air Station 6:1–250

Ghostine R, Kesserwani G, Vazquez J, Rivière N, Chenaim A (2009) Simulation of supercritical flow in crossroads: confrontation of a 2D and 3D numerical approaches to experimental results. Comput Fluids 38:425–432. https://doi.org/10.1016/j.compfluid.2008.05.003

Huang JC, Weber LJ, Lai YG (2002) Three-dimensional numerical study of flows in open-channel junctions. J Hydraul Eng 128:268–280. https://doi.org/10.1061/(ASCE)0733-9429(2002)128:3(268)

Jeong WC, Lee JW, Cho YS (2010) A numerical study on characteristics of flow in a crossroad: case of crossroad connected symmetrically with inclined four roads. J Korean Soc Hazard Mitigation 10:99–108

Khan A, Cadavid R, Wang S (2000) Simulation of channel confluence and bifurcation using the CCHE2D model. In: Proceedings of the Institution of Civil Engineers-Water and Maritime Engineering 142:97–102. https://doi.org/10.1680/wame.2000.142.2.97

Martinez M, Niño Y (2020) 3D numerical modeling to support the design of contractions in supercritical flow channels, 8th IAHR ISHS 2020

Matthews BW, Fletcher CAJ, Patridge AC (1998) Computational simulation of fluid and dilute particulate flows on spiral concentrators. Appl Math Model 22(12):965–979

Mignot E, Paquier A, Haider S (2006) Modeling floods in a dense urban area using 2D shallow water equations. J Hydrol 327:186–199. https://doi.org/10.1016/j.jhydrol.2005.11.026

Mignot E, Paquier A, Rivière N (2008) Experimental and numerical modeling of symmetrical four-branch supercritical cross-junction flow. J Hydraul Res 46:723–738. https://doi.org/10.1080/00221686.2008.9521918

Mignot E, Li X, Dewals B (2019) Experimental modelling of urban flooding: a review. J Hydrol 568:334–342. https://doi.org/10.1016/j.jhydrol.2018.11.001

Mignot E (2005) Étude expérimental et numérique de l’inondation d’une zone urbanisée: Cas des écoulements dans les carrefours en croix. Ph. D. thèse, École Centrale de Lyon, Lyon, France

Nania LS, Gomez M, Dolz J (2004) Experimental study of the dividing flow in steep street crossings. J Hydraul Res 42:406–412. https://doi.org/10.1080/00221686.2004.9728406
Neary V, Sotiropoulos F, Odgaard A (1999) Three-dimensional numerical model of lateral-intake inflows. J Hydraul Eng 125:126–140. https://doi.org/10.1061/(ASCE)0733-9429(1999)125:2(126)

Paquier A (1995) Modélisation et simulation de la propagation de l’onde de rupture de barrage. Ph.D. thesis, Université Jean Monnet de Saint Etienne, Saint Etienne, France

Rivièrè N, Travin G, Perkins RJ (2011) Subcritical open channel flows in four branch intersections. Water Resour Res 47:1–10. https://doi.org/10.1029/2011WR010504

Rivièrè N, Travin G, Perkins RJ (2014) Subcritical open channel flows in four branch intersections. J Hydraul Eng 140(4):04014003. https://doi.org/10.1061/(ASCE)HY.1943-7900.0000835

Rivièrè N, Perkins RJ (2004) Supercritical flow in channel intersections. In: Proceedings of the 2nd International Conference on Fluvial hydraulics, Grecco, Carravetta and Della Morte Eds, River Flow 2004, Napoli, Italy 1073–1077. https://doi.org/10.1201/b16998-138

Schwalt M, Hager WH (1995) Experiments to supercritical junction flow. Exp Fluids 18:429–437. https://doi.org/10.1007/BF00208465

Shetter AS, Murthy KK (1996) A numerical study of division of flow in open channels. J Hydraul Res 34:651–675. https://doi.org/10.1080/00221689609498464

Shumate ED (1998) Experimental description of combining flows at an open channel junction. Master thesis, University of Iowa, Iowa, USA, 127

Thappeta SK, Bhallamudi SM, Chandra V, Fiener P, Baki ABM (2021) Energy loss in steep open channels with step-pools. Water 13(1):72. https://sci-hub.do/https://doi.org/10.3390/w13010072

Weber LJ, Shumate ED, Mawer N (2001) Experiment on flow at a 90° open-channel junction. J Hydraul Eng 127:340–350. https://doi.org/10.1061/(ASCE)0733-9429(2001)127:5(340)

Publisher’s Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.