THE NUMERICAL STUDY OF OPEN CHANNEL JUNCTIONS WITH EXTREME CONFLUENCE ANGLES FOR SURFACE FLOW WITHOUT WALL ROUGHNESS

AHMED BILAL (1) & QIANCHENG XIE (2)

(1) College of Water Conservancy and Hydropower Engineering, Hohai University, Nanjing, China, ahmed.bilal@hhu.edu.cn
(2) Division of Fluid and Experimental Mechanics, Luleå University of Technology, Luleå, Sweden. qiancheng.xie@ltu.se

ABSTRACT

Confluences exist not only in nature, like in streams or rivers but also constructed environment like the structures for sewage and urban rainfall drainage canals. In the natural environment, two tributaries join at an angle from 30° to 135°. Existing related research focuses on tributary angels within this mentioned range. However, in the constructed environment, channel junctions with its angle beyond this range may occur. Hydrodynamic data from a 90-degree flume confluence is taken from experiments, available in the literature. The information is used to build a numerical model on Delft 3D. The similar flow parameters are used, with the comparable cross-sectional dimensions, with three confluence geometries. Firstly, two confluences with a junction-angle of 0°, and then with an angle of 180° are studied, each with three flow cases. Free slip condition is supplied to elude the effect of wall roughness on flow behavior. Interesting results have been found and presented. It is noticed that geometries affect the location of vertical velocity and velocity distribution in the confluence area. The 180° confluence shows a big stagnation zone in the angled tributary. This study improves the understanding of the interaction between flow dynamics and confluence angles better.

Keywords: Confluence, Extreme angle, Delft 3D, Hydraulic modeling, Surface flow patterns

1 INTRODUCTION

The joining of two streams, i.e., a confluence, is a focus of a large number of field, laboratory, and numerical modeling studies (Babarutsi and Chu, 1998; Bouchez et al., 2010; Shakibainia et al., 2010; Creelle et al., 2017). The angle, two tributaries are making at the intersection point, is called the confluence angle. In existing confluence studies, the angles of 30° to 135° were focused (Best, 1986, 1987; Riley et al., 2015; Guillein-Ludena et al., 2016). Generally, an angle of confluence beyond this range does not occur; studies for these extreme cases are rarely done.

Study of the confluences at their extremities give insight about the new flow features or give information about the ones which usually are inadequately hypothesized. For instance, Schindfessel et al. (2015) noticed the unique flow features during their study of a junction with flow ratio (ratio of the flow of the tributary aligned to post-confluence channel and total flow after the merger) less than 0.1. They noticed that for this case of extreme flow ratio, there are changes in mixing layer location along with the development of much stronger upwelling, which is not observed in typical confluence cases. This study takes the case of two extreme angles of confluences, i.e., 0° and 180°. Some of the known cases where parallel confluences, i.e., with an angle = 0°, are dealt include the studies by Cushman-Roisin and Constantinescu (2019), Bradbrook (1999), and Best and Roy (1991). Bradbrook (1999) and Best and Roy (1991) investigated the parallel confluences with discordant beds through physical and numerical model studies, respectively. Another study of parallel streams on a laboratory flume is performed by Uijttewaal and Booij (2000), in which they examined the extent of the mixing zone.

In most of the stated works, the width of the post-confluence channel is equal to the sum of widths of tributaries, which is usually not the case in the existing studies. So far, no study has been done, to the best of authors’ knowledge, in which parallel streams are studied with a post-confluence width less than the sum of its tributaries. Besides, a junction with a 180°angle has not been studied yet.

This study takes the dimensions of the channels, and flow values, which are comparable to the flume experiment of Yuan et al. (2017). The objectives are to examine the depth-averaged flow features, to identify the location of channels with enhanced vertical velocity, and to show how far a stable streamwise velocity is achieved for following geometries of a junction.

1. 0° confluence and centerline of post-confluence channels aligned to the shared wall of tributaries
2. 0° confluence and one wall of post-confluence channels aligned to the wall of one tributary.
3. 180° confluence and one wall of post-confluence channels aligned to the wall of one tributary.

The study is performed using numerical simulation of all the cases with a free slip condition to avoid the effect of wall roughness. It focuses exclusively on the geometry of the intersection without any feedback of shear stress of the channel wall.
2 METHODS

Based on the geometric arrangement described in the previous section, and with the details of dimensions used by Yuan et al. (2017), three confluences are defined and shown in Fig. 1. The $Q_1$, $Q_2$, and $Q_{\text{tot}}$ denote the flows for Tributary 1, Tributary 2, and post-confluence channel, respectively. Three cases of different flow ratios, $Q_r$ (where $Q_r = Q_2/Q_1$), are used for each confluence type (Table 1). Although the planform geometry is different, the cross-sectional shape of channels in all the three types is the same. Delft 3D numerical model is selected to simulate all the cases mentioned in Table 1.

**Confluence 1**: $0^\circ$ angle of confluence, with centerline of post-confluence channel aligned to the common wall of the tributary channels.

**Confluence 2**: $0^\circ$ angle of confluence, with one side of post-confluence channel aligned to the outer wall of Tributary 1.

**Confluence 3**: $180^\circ$ angle of confluence.

Fig. 1: The geometrical consideration of the confluence cases used in this study: Confluence 1 is a $0^\circ$ junction, and the post-confluence channel is centered in the middle of intersection; Confluence 2 is also a $0^\circ$ junction, but its post-confluence channel has its one bank (wall) in common with the bank of a Tributary 1; Confluence 3 is a $180^\circ$ junction with a smooth bend of Tributary 2. The units shown are in millimeters.
Table 1: Description of the three flow cases for each confluence type.

<table>
<thead>
<tr>
<th>Confluence Type</th>
<th>Case</th>
<th>Confluence Angle</th>
<th>$Q_1$ (m$^3$/s)</th>
<th>$Q_2$ (m$^3$/s)</th>
<th>$Q_{tot}$ (m$^3$/s)</th>
<th>$Q_r$</th>
<th>Bed Slope</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>A</td>
<td>0</td>
<td>0.39</td>
<td>0.26</td>
<td>0.65</td>
<td>0.67</td>
<td>0.001</td>
</tr>
<tr>
<td>1*</td>
<td>B*</td>
<td>0</td>
<td>0.26</td>
<td>0.39</td>
<td>0.65</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>C</td>
<td>0</td>
<td>0.325</td>
<td>0.325</td>
<td>0.65</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>A</td>
<td>0</td>
<td>0.39</td>
<td>0.26</td>
<td>0.65</td>
<td>0.67</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>B</td>
<td>0</td>
<td>0.26</td>
<td>0.39</td>
<td>0.65</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>C</td>
<td>0</td>
<td>0.325</td>
<td>0.325</td>
<td>0.65</td>
<td>1.00</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>A</td>
<td>180</td>
<td>0.39</td>
<td>0.26</td>
<td>0.65</td>
<td>0.67</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>B</td>
<td>180</td>
<td>0.26</td>
<td>0.39</td>
<td>0.65</td>
<td>1.50</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>C</td>
<td>180</td>
<td>0.325</td>
<td>0.325</td>
<td>0.65</td>
<td>1.00</td>
<td></td>
</tr>
</tbody>
</table>

Note: Confluence 1*, Case B is not modeled due to symmetric confluence geometry.

2.1 MODEL SETUP

The numerical simulation package, Delft 3D-Flow, is based on the finite difference method and solve the Navier-Stokes equation with shallow water and Boussinesq assumption (Deltares, 2014). For a detailed description of hydrodynamics, the readers are referred to works by Lesser et al. (2004) and Deltares (2014).

A grid cell of size 20 x 20 cm was chosen. A boundary fitted σ-layer model is used for vertical layering. A total of 11 equally spaced layers are used so that the secondary current is reasonably resolved for the 3D simulation. A manning’s $n$ value of 0.028251 is used; the roughness is based on the calculation of roughness from the study by Yuan et al. (2017). The hydrodynamic grids for all the three geometries are shown in Fig. 2.

Fig 2: The hydrodynamic grid for (a) Confluence 1, (b) Confluence 2, and (c) Confluence 3

The hydrodynamic simulation is performed for 25 minutes with a time step of 0.001 minute. First seven minutes are used as smoothing time. The results are taken at the end of simulation when the flow is sufficiently stable both numerically and hydraulically. The boundary condition for the open-boundaries is controlled by the total flow. Drogues are used in both the tributaries to trace the path of a free-floating particle. These particles are seeded at an equal distance across the cross-section at the same time, after eight minutes of simulation, in both the tributaries. Movement of drogue particles is governed by the horizontal velocity in the surface layer.

3 RESULTS AND DISCUSSION

The results of simulation Confluence 1 Cases A and C are shown in Fig. 3. The results show that although the flow achieves a uniform velocity at a short distance in the post confluence channels, the streamlines (i.e., the path of drogue particles) of both the channels remain separate.

(a)  (b)
Fig. 3: Flow behavior of Confluence 1, Cases A and C; (a) Case A: Depth-averaged velocity magnitude is shown in color map and, path of tracer particles (drogues) of Tributary 1 and 2 are also shown; white arrows show the direction of the velocity vector; (b) Case A: Location and magnitude of vertical velocity at the junction; (c) Case C: Depth-averaged velocity; and (d) Case C: Location and magnitude of vertical velocity at the junction.

The simulation results of all cases for Confluences 2 and 3 are shown in Figs. 4 and 5. Similar to Confluence 1, a uniform velocity appears at a short distance in the post-confluence channel in the Confluence 2. This observation is also noticed in research by Cushman-Roisin and Constantinescu (2019), where they analyzed the flow features of an almost parallel confluence. There are small areas where vertical velocity is noticed near the surface. For Confluence 2, the area of vertical velocity reduces with increase in $Q_r$. By contrast to Confluence 1, where vertical velocity is more prominent near smooth cross-section of the post-confluence channel; it appears at the end of tributaries in Confluence 2. As expected, flow from the tributary with more momentum presses the flow from the second tributary.

For Confluence 3, there is a significant zone of stagnation visible in Tributary 2 near the upstream junction corner. It is noticed that the size of the stagnation zone (the blue area) reduces with the increase of $Q$. The six confluence hydrodynamic zones, as usually mentioned in related literature (Schindfessel et al., 2015; Yuan et al., 2017), does not appear in any of the type and cases. This is probably attributable to the free slip condition supplied at walls. It is noted that for 180° confluence, the streamlines from Tributary 1 first attract toward the Tributary 2 and then enter the post-confluence channel. This phenomenon is more prominent with low $Q_r$. A zone of low velocity is also observed, just before the post-confluence channel, which does not happen in conventional geometries. The flow bending and the zone of stagnation are almost together in Tributary 2 of Confluence 3. A zone of flow deflection is also clearly visible in this geometric arrangement.
Fig. 4: Flow behavior of Confluence 2, Cases A, B, and C; (a) Case A: Depth-averaged velocity magnitude is shown in color map and, path of tracer particles (drogues) of Tributary 1 and 2 are also shown; white arrows show the direction of the velocity vector; (b) Case A: Location and magnitude of vertical velocity at the junction; (c) Case B: Depth-averaged velocity; (d) Case B: Location and magnitude of vertical velocity at the junction; (e) Case C: Depth-averaged velocity; (f) Case C: Location and magnitude of vertical velocity at the junction.
CONCLUSION

In this paper, three extreme confluence geometries, in terms of angle of confluence are numerically simulated using Delft 3D. These extreme geometric configurations include two 0° and one 180° confluences. A free-slip condition is supplied to the numerical model to isolate the flow features from the effect of wall roughness. It is noticed that despite having the same hydraulic parameters and junction angle, the shape of flow features and location of vertical velocity is different in Confluences 1 and 2.

It is also noticed in first two geometric arrangement that although a uniform velocity is achieved after a short distance in the post-confluence channel, the flow of both tributaries remains separate for considerable long distance in parallel confluences.

In light of 180° confluence, a stagnant area with little or no velocity is very apparent near the upstream corner of the junction in Tributary 2; the size of this stagnant area reduces with the increase of flow in Tributary 2.

The study is based merely on the results of numerical simulations with no data from any physical model for calibration or validation. Due to this, it is emphasized that the results may be considered as qualitatively and used only to see the trend of flow behavior in a specific confluence geometry. More flume studies or numerical modeling with validation data from a real case are required to be carried out to strengthen the results.
ACKNOWLEDGMENTS

The authors are partially financed in the form of a Ph.D. scholarship from the Chinese Scholarship Council (CSC). Authors are also thankful to Dr. Saiyu Yuan at Hohai University, for giving some initial discussion about the open channel junctions and modeling possibilities.

REFERENCES


