RIGID-LID LES PREDICTIONS OF THE 3D-FLOW IN AN OPEN-CHANNEL CONFLUENCE WITH CONCORDANT AND DISCORDANT BEDS

PEDRO XAVIER RAMOS(1), LAURENT SCHINDFESSEL(2), JOÃO PEDRO PÊGO(3) & TOM DE MULDER(4)

(1,2,4) Hydraulics Laboratory, Dep. Civil Engineering, Ghent University, Ghent, Belgium
Pedro.Ramos@UGent.be, Laurent.Schindfessel@UGent.be, TomFO.DeMulder@UGent.be
(3) Faculty of Engineering, University of Porto, Porto, Portugal
jppego@fe.up.pt

ABSTRACT

Right-angled confluences of open-channels with rectangular cross-sections are common configurations used to study the complex confluence hydrodynamics by means of lab experiments and numerical simulations. Large-Eddy simulations (LES) of confluences have traditionally been validated using bulk quantities such as water levels and streamwise mean velocities. Given the cost of LES, simplifications are usually adopted (wall-models, rigid-lids as water surface, etc.). However, recent studies have shown that the predicted secondary flow turns out to be sensitive to the curvature of the top boundary of the numerical domain when it comes to secondary motions and, therefore, the secondary flow shall be used within the validation of the numerical simulations. Weber et al. (2001) provide an experimental dataset in a confluence with a main channel and a tributary having equal widths and concordant beds, at a moderately low downstream Froude number (Fr = 0.37) and at different discharge ratios, between the upstream main channel and the downstream main channel, q. In his paper, the authors found that the sensitivity of secondary flow to the rigid-lid treatment reported in Ramos et al. (2019) for the case with q = 0.25, does not show up for the concordant bed case of Weber et al. (2001) with the more moderate q = 0.58, despite the two cases sharing the same Froude number. Moreover, it has been shown, for the present case, that the discordance between the bed levels of the tributary and the main channel, which often occurs in natural confluences, alters the secondary flow patterns significantly, namely, distorting the mixing layer.

Keywords: open-channel confluences, Large-Eddy Simulation, secondary flow, upwelling.

1 INTRODUCTION

Open-channel confluences are a common occurrence in many natural watercourses as well as in networks of artificial canals. These locations are widely acknowledged as important nodes that control the downstream routing of water, nutrients, pollutants and sediments, having also a significant impact on the upstream water levels (Rice et al., 2008).

The complexity of open channel confluences accommodates flow mixing, secondary cells, flow separation, fluid upwelling, flow contraction and backwater effects. The aforementioned phenomena are influenced by the junction angle (σ), the discharge ratio (q), the channels width ratio (Ω) and channels beds elevation discordance (α), etc. (Rice et al., 2008). The influence of those factors on the complex hydrodynamics of confluences has been investigated intensively, in the field (Ashmore et al., 1992; Biron et al., 1993, 1996a; Rhoads and Kenworthy, 1995; De Serres et al., 1999; Rhoads and Sukhodolov, 2001, 2004; Konsoer et al., 2014; Sukhodolov et al., 2017, Gualtieri et al., 2018) as well as in laboratory tests, often in schematized confluences of sharp angled flumes with rectangular cross-sections (Best and Reid, 1984; Best, 1987; Best and Roy, 1991; Biron et al., 1996a, 1996b; Guillon-Ludena et al., 2015; Schindfessel et al., 2015, 2017; Creëlle et al., 2017). Numerical modelling has been increasingly helpful to understand the highly three-dimensional flow in the confluence hydrodynamics zone (CHZ), providing insights from higher (spatial and temporal) resolution data and obtained in a non-intrusive way (Bradbrook et al., 1998, 2000, 2001; Lane et al., 2000; Huang et al., 2002; Constantinescu et al., 2011, 2012; Schindfessel et al., 2015; Penna et al., 2018; Ramos et al., 2019). Those studies have shown how Computational Fluid Dynamics (CFD) tools can be useful and powerful means to understand the flow patterns and their interaction with the boundaries of the flow (bed, walls, air, etc.).

There are several ways of numerically treating the aforementioned boundaries. Many studies have been improving the boundary conditions of their eddy-resolving models (such as Large Eddy Simulations, LES) or models based on the Reynolds-averaged Navier-Stokes (RANS) equations. As a result, a variety of functions and utilities are available for the treatment of the numerical domain boundaries. The free-surface is an especially complex interaction. The knowledge about the free-surface is important to understand the complicated flow patterns in confluences (Lane et al., 2000) but it has also practical applications, like flood control, or bank
protection purposes. For the treatment of the free surface, different modelling strategies are available (McSherry et al., 2017). A popular (“inexpensive”) approach is to use a rigid-lid as the upper boundary of the computational domain. The lid is a fixed and impermeable boundary, at some estimated position of the actual free-surface. To prevent the development of spurious boundary layers, the lid is treated as a frictionless wall. The pressure field onto the rigid-lid, computed either with a RANS-based or a LES-based model, can then be translated into a virtual field of free-surface elevations, by means of the hydrostatic pressure law assumption (McSherry et al., 2017; Ramos et al., 2019). To simulate open-channel flows which are characterized by small surface deviations compared to a characteristic mean water depth in the domain (approximately 10% of the depth is the suggested threshold value see, for example, Constantinescu et al., (2011), Rodriguez et al., (2004) and McSherry et al., (2017)), the rigid-lid is most often a flat surface. Since this rigid-lid approximation suppresses the actual surface deformations, a certain error is introduced in the continuity equation. Typically, a constant discharge is imposed, leading to overestimations or underestimations of the bulk velocity in the zones where the rigid-lid is, respectively, lower or higher than the real water level. A discussion on this issue is carried out, for example, in Constantinescu et al. (2011) and Ramos et al. (2019). Such an approximation not only brings uncertainty in the assessment of the water levels, but (consequently) compromises the linkage between the turbulent structures and the free-surface dynamics. This linkage has been studied in the past for open-channel flows (e.g. Kara et al., 2015; McSherry et al., 2017). Lane et al. (2000) and Yang et al. (2013) devoted efforts to the study of the aforementioned relation in the specificities of an open-channel confluence. Ramos et al. (2019) have shown that the oversimplification of the free-surface by means of the application of a flat rigid-lid could sacrifice the accuracy in secondary flow prediction of the specific case of the open-channel confluence of a discharge-ratio of dominant tributary inflow (q=0.25) of Weber et al. (2001), even though this case has a moderately low downstream Froude number (Fr=0.37).

Besides the flow ratio, the flow hydrodynamics in open-channel confluences is deeply affected by the bed elevation discordance ratio. In nature, open-channel confluences have generally a discordant bed elevation (Kennedy, 1984). Best and Roy (1991) have shown that the discordance of bed elevations causes the distortion of the mixing layer and strong fluid upwelling motion, which contributes for a rapid mixing (Gaudet and Roy, 1995). The mixing layer has been well documented over the last two decades due to the advancement in the experimental equipment with a higher sampling rate (Biron et al., 1993a; Rhoads and Sukhodolov, 2004) and numerical simulation methods (Bradbrook et al., 2000a, 2000b, 2001; Shakibainia et al., 2010; Yuan et al., 2016). Intense turbulence within the shear layer has been the subject of many studies. Roy et al. (1999) linked high turbulence observed in the shear layer at the same field site to higher bed load transport rates. De Serres et al. (1999) observed very high intensity of turbulence (peaks up to 50%) and turbulent kinetic energy in the shear layer. Best (1987) revealed that a shear layer was created between the two convergent flows along which powerful, vertical vortices were generated. Rhoads and Sukhodolov (2004) showed the process of eddy development in the shear layer in the horizontal plane appeared to be similar to traditional vortex pairing. These quasi two-dimensional vortices with near vertical axes may lead to substantial lateral transfer of fluid between two combining flows (Rhoads and Sukhodolov, 2001). Constantinescu et al. (2011) used an eddy-resolving numerical model to simulate the large-scale turbulence structure in an asymmetrical river confluence. Constantinescu et al. (2012) concluded that the types of coherent structures were greatly affected by the discharge ratio. Sukhodolov et al. (2010) adjusted the length of a splitter plate placed in a straight channel to change the lateral velocity gradient between the flows downstream, and to determine the influence of different gradients on shear layer dynamics. Bradbrook et al. (2000a) explain that the distortion on the mixing layer results in a fluid upwelling motion near the downstream corner of the confluence, which interferes with the recirculation zone (RZ) dimensions. Upwelling has been often suggested in the literature (see Biron et al., 1996b) to be a result of bed elevation discordance and its existence may have implications regarding the mixing of sediments, nutrients or pollutants in open-channel confluences. Best and Roy (1991) report a study on a zero degrees confluence angle, and those authors suggest that the fluid upwelling also subsist in higher angles confluences, but its location may vary depending on the bed discordance and discharge ratios.

For the experimental case of Weber et al. (2001) with q=0.58, several numerical studies (e.g. Đorđević 2013; Ramos et al., 2018) also have shown that discordance in bed elevation between the confluent open-channel rectangular cross-sections at a 90° angle increases turbulence intensity and changes the structure of the secondary motion to the streamwise flow. Note that these two last aforementioned studies have used a flat rigid-lid. Given Pouchoulin et al. (2018) and Ramos et al. (2019) remarks on the sensitivity of the secondary flow predictions in open-channel confluences, it is necessary to study the sensitivity of the secondary flow to the curvature of the top boundary of the numerical domain. Consequently, the present study follows the work of Ramos et al. (2018, 2019), focusing now on a more balanced discharge ratio between the two incoming flows. Like in Ramos et al. (2019), the motivation of this study resides on the fact that simulations performed with a flat and frictionless, rigid-lid implementation may not be valid very close to the recirculation zone and the contracted section because, in the latter, the flow undergoes an acceleration that causes the water surface to drop appreciably (±20%, according to Rice et al. (2008) and Crébille et al. (2018)). Therefore, the first objective (O1) of the present study is to assess the influence of the rigid-lid approximation onto the wall-modelled LES of secondary flow for the same case (Weber et al., 2001) for a different discharge ratio (q=0.58 instead of q=0.25). The time-averaged flow predicted in the simulations will be analyzed. In the second part of this study,
maintaining the same flow ratio of $q=0.58$, the numerical model will be applied to one more bed elevation discordance ratio (with a ratio of step height to downstream flow depth $\sigma=0.25$), with the objective (O2) of concisely describing the influence of the discordance bed elevation onto the flow, especially regarding the position of the mixing layer and its distortion, and investigate if the upwelling motions near the downstream corner of confluence reported by Biron et al. (1996b) are also present for this specific confluence geometry.

The flow case that will be studied in this paper is presented briefly following this introductory chapter. The succeeding section describes the adopted numerical methodology. The time-averaged results from the two numerical simulations ($\sigma=0$ and $\sigma=0.25$) are presented in the next section. Wherever possible, i.e. for the concordant case, results will be compared with the available experimental data (Weber et al., 2001), together with insights about the flow patterns gained from the simulations for each bed discordance ratio.

2 FLOW CASE

In this paper, only attention will be given to a right-angled, open-channel confluence of channels with a rectangular cross-section of equal width ($W$) and with concordant and horizontal beds. For the latter type of confluence, an important parameter is the discharge ratio:

$$q = \frac{Q_u}{Q_d} = \frac{Q_u}{(Q_u + Q_t)}$$  \hspace{1cm} [1]

where $Q_u$ and $Q_t$ are the incoming discharge of the main channel and the tributary, respectively, and $Q_d$ is the downstream discharge.

The bed discordance ratio (or relative step height) is defined as the ratio between the step height, $s$, and the downstream flow depth, $h_d$:

$$\sigma = \frac{s}{h_d}$$  \hspace{1cm} [2]

The associated water level elevations, are also influenced by the downstream Froude number:

$$Fr_d = \frac{U_d}{\sqrt{gh_d}}$$  \hspace{1cm} [3]

where $U_d=Q_d/(h_dW)$ is the cross-sectionally averaged downstream velocity, $h_d$ the downstream flow depth and $g$ the gravitational acceleration.

The confluence flow at $Fr_d=0.37$ which was studied experimentally by Weber et al. (2001) at different discharge ratios, will be used in this paper. In particular, the case with the flow ratio $q=0.58$ (a more balanced discharge ratio, compared to the dominant tributary case of Ramos et al., 2019) is selected. The aforementioned flow case has been used several times in the literature for validation of numerical models (Huang et al., 2002; Đorđević, 2013; Yang et al., 2013). The selected flow case corresponds to a downstream Reynolds number $Re_d$ of approximately 11,200, which is sufficiently high to ensure turbulent flow:

$$Re_d = \frac{U_dR_d}{\nu}$$  \hspace{1cm} [4]

where $R_d=(W \cdot h_d)/(W+2h_d)$ is the hydraulic radius of the downstream section and $\nu$ is the kinematic viscosity of water.

<p>| Table 1. Hydraulic conditions (Weber et al., 2001). |</p>
<table>
<thead>
<tr>
<th>Q_u (m$^3$/s)</th>
<th>Q_d (m$^3$/s)</th>
<th>q (-)</th>
<th>$\sigma$ (-)</th>
<th>$\Omega$ (-)</th>
<th>$\alpha$ (º)</th>
<th>$h_d$ (m)</th>
<th>W (m)</th>
<th>$h_d/W$ (-)</th>
<th>$U_d$ (m/s)</th>
<th>$Fr_d$ [-]</th>
<th>$Re_d$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.042</td>
<td>0.169</td>
<td>0.58</td>
<td>0</td>
<td>1</td>
<td>90</td>
<td>0.296</td>
<td>0.914</td>
<td>0.324</td>
<td>0.628</td>
<td>0.37</td>
<td>11,200</td>
</tr>
</tbody>
</table>

3 NUMERICAL METHODOLOGY

3.1 Large Eddy Simulations

The numerical simulations in the present contribution are conducted within the computational fluid dynamics (CFD) software OpenFOAM, version 5.0. In this open-source toolbox, the LES equations (i.e. the spatially averaged three-dimensional Navier-Stokes and continuity equations) are discretized using the Finite Volume Method. Therefore, the LES equations are written in function of the cell-averaged quantities and fluxes through the cell boundaries (Rodi et al., 2013). The equations are coupled and solved using the PISO algorithm. The standard Smagorinsky model is used with a constant $C_s$ of 0.158. This SGS model is used in the simulations in combination with a van Driest damping function, which guarantees that $\nu_{GDS}$ tends towards zero in the vicinity of the wall, as it is physically required (Rodi et al., 2013; Schindfessel, 2017). Further details about the choice of numerical parameters in the present model are found in Schindfessel (2017) and Ramos et al. (2019). Verification steps follow the guidelines of Pope (2004) (amount of turbulent kinetic energy resolved on the mesh) and Rodi et al. (2013) (spatial and temporal mesh convergence).
3.2 Free-surface treatment

The flat rigid-lid LES are conducted by prescribing the height of the flow domain equal to the mean-flow depth downstream, \( z_{\text{rigid lid}} = h_s = 0.324W \) (see Table 1) throughout the confluence. For the curved rigid-lid simulations, the time-averaged pressures on the flat rigid lid, \( P \), obtained in the simulation with a flat lid, at height \( z_{\text{rigid lid}} \) is used to deform the rigid lid, by means of Equation 5.

\[
h(x, y) = z_{\text{rigid lid}} + \frac{p(x,y,z_{\text{rigid lid}})}{\rho g}
\]

The structured mesh of the flat rigid-lid LES is then deformed to fit the curved rigid-lid, in a similar way as adopted by Rameshwaran and Naden (2004). As one can infer from Figure 1, the mesh deformation consists of displacing the nodes in the vertical (z) direction. However, the first nodes next to the beds do not suffer any displacement (Figure 1.b), in order to maintain the dimensionless wall-normal distance of the first cell, \( z^+ \), values constant and apply the wall model always under the same circumstances.

This approach relies on the principle that the vorticity field may be improved indirectly by a slightly changed velocity field due to the deformation of the free surface (van Balen, 2009). On both the flat and curved rigid lids, zero shear stress and zero normal velocity conditions are imposed.

![Figure 1. Schematic view of the mesh deformation (a. symbolizes the flat lid; b. symbolizes the curved lid)](image)

3.3 Other boundary conditions

At the bed and sidewalls, the same wall function approach is used as in Ramos et al. (2019), which was described and validated earlier in Schindfessel et al. (2015, 2017). Since a LES resolves (a large part) of the turbulence on the mesh, and for the sake of implementing a realistic boundary condition, the inlet velocity should also be turbulent and fully developed. In the present numerical set up, this is achieved by use of a precursor simulation. In this subdomain, a periodic channel is simulated and its turbulent velocity is used as an inlet condition. In order to prevent the need to store the results of precursor simulations, the precursor channel is enclosed into the computational domain. In the present simulations, this means that the flow simulated at 2W downstream of the main channel and tributary domain inlets is emulated to those inlets, preserving the specified mass flux. To initiate and preserve turbulent fluctuations, some random variation is added to the inlet velocity of the precursor channel. This variation makes up 15% of the total signal, and has a standard deviation of 25% of the mean value (Schindfessel, 2017).

3.4 Matrix of simulations

Analogous to what was done by Biron et al. (1996b) and Đorđević (2013), the flow conditions were slightly different for the concordant and discordant bed cases (Table 2) since the introduction of the step in the interface between the two channels results in modifications of mean velocities and depth. Total discharge entering the confluence was, however, equal for all cases and also the discharge ratio was kept constant (\( q \approx 0.58 \)).

<table>
<thead>
<tr>
<th>( F_\text{or} )</th>
<th>( C_\text{or} )</th>
<th>( \sigma )</th>
<th>( \text{RIGID-LID SHAPE} )</th>
<th>( U_t/U_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>flat</td>
<td>0.402</td>
</tr>
<tr>
<td>0.25</td>
<td>0</td>
<td>0.25</td>
<td>curved</td>
<td>0.536</td>
</tr>
</tbody>
</table>

3.5 Computational meshes

For the concordant bed case, the same fine resolution is adopted as in Table 3 of Ramos et al. (2019) (around 6.6 million cells). Also the mesh for the discordant bed case has a similar resolution (basically defined by removing the horizontal cell layers below the elevation of the step). Note that these meshes were the result of a mesh sensitivity analysis, which will not be fully reported here for the sake of brevity. One should be aware, however, that the coarser mesh results reported in Ramos et al. (2018) for the present case produced streamwise velocities identical to the ones in the present paper, but different secondary flows.
3.6 Computational parameters

All simulations are run on a high-performance computer, using 4 x 8-core Intel E5-2670 (Sandy Bridge @ 2.6 GHz) processors. The total computational cost of the simulation is approximately 4600 CPU hours. The code is parallelized via domain decomposition, in 32 sub-domains, meaning that the real computation time is about 144 hours for each simulation.

4 RESULTS AND DISCUSSION

4.1 Water levels

The validation and quantitative assessment of the predictive capabilities of the method described in section 3.2 for the water surface treatment are provided in Figure 3, where the measured and numerically predicted water surface are compared for the case with concordant beds ($\sigma=0.0$). Additionally, the numerical results for the discordant case ($\sigma=0.25$) are also included, but the experimental dataset does not include such a configuration. For the concordant case ($\sigma=0.0$), near the recirculation zone ($y/W=0.167W$) and in the longitudinal centerline of the main channel ($y/W=0.50$), the CFD model gives results reasonably close to the experiments and their agreement is better than the one in $y/W\approx 0.833$. In the region near the outer bank ($y/W\approx 0.833$), the flow accelerates, creating a water level depression, which is not accurately captured in this (or others, to the best of our knowledge) numerical model. That discrepancy can also be attributed to the measurement uncertainties and the underresolved flow field as such free surface features are not easily simulated. In fact, other studies reported the same lack of agreement when there are similar dips in the water levels, even when using more sophisticated approaches that fully resolve the water surface (e.g. level-set method in Kara et al. (2015) for a flow around an abutment; dynamic mesh in Huang et al. (2002) or Volume of Fluid (VOF) in Yang et al. (2013) for the exact same open-channel confluence).

4.2 Streamwise mean flow ($O_1$)

In Figure 4, the vertical profiles of the measured and computed streamwise velocity are compared. As shown in these figures, the simulation with the curved rigid-lid produces a better agreement with the experiments in most locations. The flat rigid-lid LES does not predict the increase in water depth after the CHZ, which has led to an overprediction ($\approx 20\%$) of the streamwise velocity for $x/W=5$. Yet, the lack of a clear advantage and the slight discrepancies found in both simulations arguably do not motivate the (more expensive) use of the curved rigid-lid. Both simulations have trouble in capturing the velocity near bed, which might be due to the implementation of the wall model. This is, however, a quite convincing agreement, given the fact that the location with the worst agreement (differences of $\approx 25\%$) is $x/W=1$, a location which is quite complex in terms of turbulence, and for all the other locations the agreement is equal or superior to the analogue results found in the literature for the present case.
4.3 Secondary flow (O1 and O2)

In Figure 5, the multicellular pattern of the secondary flow at the different cross-sections of the main channel, as seen in the experiment (Weber et al., 2001) and in the simulations with concordant bed ($C_{o=0.0}$), is visualized by time-averaged velocity vectors, suggesting the existence and persistence, both in time and along the main channel, of several secondary flow cells. Due to its low resolution compared to simulations and the fact they do not reach the full extent of the cross-section; the experiments do not enable a full comparison with the simulations in terms of number and extent of secondary flow cells. Nevertheless, the agreement in terms of direction of the secondary flow is quite satisfactory, although, like in Ramos et al. (2019), the upper-central zone ($y/W=0.5 ; z/W>0.15$) of the secondary flow at x/W=2 (left panel) does not show the same direction in the simulations and in the experiments. The magnitude of the secondary flow vectors, however, seems to be underpredicted by the numerical simulations.

Figure 6 compared the predictions of the secondary flow, in the same three cross-sections downstream the CHZ, obtained by the LES with a flat and a curved rigid-lid ($F_{o=0.0}$ vs $C_{o=0.0}$). Similarly to what happens with the streamwise component of the time-average velocity, the curvature of the rigid-lid does not have a strong influence on the secondary cells downstream the CHZ. This is the opposite of what happens, according to Ramos et al. (2019), for the case with a dominant tributary discharge ($q=0.25$). This means that the conclusions reached in that study, for that specific discharge ratio, cannot be directly generalized to other discharge ratios. Some details might be speculatively suggested as the cause for such a difference, namely the peculiarity of the case $q=0.25$, as it is reported in Huang et al. (2002), Shakibaeinia et al. (2010), Sharifipour et al. (2015), Zaji and Bonakdari (2015) and (Chen et al., 2017), who also suggest that particular discharge ratio as the most challenging (they reported a higher discrepancy while comparing the $q=0.25$ with the experimental data of Weber et al. (2001)) and as a tipping point of the secondary flow patterns in terms of number and orientation of cells. All those studies have point out the greater water-surface elevation/depression for the specific case of $q=0.25$, while the present case ($q=0.58$) has a less pronounced variation in the water levels. The separation zone is also smaller in the present case, leading to a less contracted flow.

Figure 7 suggests that the cross-sectional vector fields are influenced the most with the introduction of the step in more upstream cross-sections (such as $x/W=2$). At the left corner ($0<y/W<0.25; 0<z/W<0.15$) of the cross-section $x/W=2$, major differences occur. According to Ramos et al. (2018), the RZ gets destroyed for
α=0.25 in that region. For x/W=3 and 5 the flow converges to somehow similar secondary patterns with vectors of the same magnitude.

![Figure 7](image)

**Figure 7.** Secondary flow as computed in the curved rigid-lid LES for the concordant (C_{α}=0.0, blue) and discordant (C_{α}=0.25, orange) bed case.

Most of the previous studies on the numerical simulation of turbulent open-channel flow of Weber et al. (2001) use the rigid-lid approximation and the validation is typically done based upon the streamwise velocity and water levels. Those studies (e.g. Zhang et al., 2009; Shakibaenia et al., 2010; Cheng et al., 2017; Ramos et al., 2019) present some inconsistencies in the number and/or orientation of the secondary cells between them and/or the experiments of Weber et al. (2001). As Ramos et al. (2019) discusses, the secondary flow of Weber et al. (2001) has been poorly modelled, despite the fact that the water levels and streamwise velocity profiles compare well with the experiments. The validation of the simulations shall, therefore, include the secondary flow field observed in the experiments.

4.4 Mixing layer (O2)

As the water levels shown in Figure 3 suggest, the fact that the discordant case has a higher tributary inlet cross-sectionally averaged velocity makes the main flow to be squeezed laterally, which, in its turn, to conserve its volumetric flowrate, forces the water level to remain higher than otherwise be. Thus, as Biron et al. (1996a, b) explains, the centerline of the mixing layer, near the bed, migrates towards the side of lower velocity. Figure 8 depicts the centerline of the mixing layer, for different cross-sections of the downstream main channel, obtained based on the gradient of the time-averaged velocity \( \partial u/\partial y \) (Creëlle, 2017). Although the complexity of the mixing layer, namely its oscillatory behavior, requires more than just this time-averaged results-based approach, these results already suggest that, similarly to what happens to the 30° confluence of Biron et al. (1996b, 2004), the step in the interface of the two channels leads to the distortion of the mixing layer.

![Figure 8](image)

**Figure 8.** Mixing layer centerline, calculated based on the velocity gradients (\( \partial u/\partial y \)) for the concordant (C_{α}=0.0, left) and discordant (C_{α}=0.25, right) bed case (with h being the local water depth as predicted by the time-averaged pressures on the rigid-lid).

4.5 Upwelling motions (O2)

Figure 9 shows the vector field based on the streamwise and vertical time-averaged velocity components at the longitudinal plane y/W=0.0125. According to the conceptual model of Best (1987) for open-channel confluences with discordant bed elevations (α=0), such longitudinal plane as y/W=0.0125 crosses the recirculation zone, which explains the backflow observed in Figure 9.a (streamwise flow is from the left to the right). In the discordant bed case, a very strong vertical motion exists near the downstream corner of the confluence (Figure 9.b), which greatly contrasts with the concordant case, where almost all the vectors remain parallel to the bed.

![Figure 9](image)

**Figure 9.** Vector field based on the streamwise and vertical component of the time-averaged velocity at the longitudinal plane y/W=0.0125 for the concordant (C_{α}=0, a, left) and discordant (C_{α}=0.25, b, right) bed cases.

Figure 10 shows 3D streamlines for the curved rigid-lid LES simulation of the discordant bed case (C_{α}=0.25). The streamlines are colored by the time-average streamwise velocity. u. In Figure 10.a, the streamlines are...
seeded in the tributary and in b. they are seeded in the main channel. In both cases, they are seeded at an elevation corresponding to 75% of the water depth downstream of the main channel. Also in both cases, they reveal upwelling motions, which means that the fluid that describes those motions is originated in both main and tributary channels. This suggests that the mixing of both fluids occurs already as upstream as x/W=1.

![Figure 10. Streamlines for the discordant bed case (C=0.25), colored by the time-averaged streamwise velocity, as injected at an elevation of 75% of the main channel water depth, in the tributary (a., left) and in the main channel (b., right).](image)

The previously described distortion of the mixing layer induces an upwelling motion near the downstream corner of the confluence. This upwelling contributes for the destruction of the RZ reported for the same case already in Ramos et al. (2018) and Đorđević (2013). For the exact same case, Đorđević (2013) and Ramos et al. (2018) mention that the RZ gets its dimensions reduced and RZ is destroyed near the bed. Although those studies make use of simulations with a flat rigid-lid, due to the reasons exposed in the section 4.1 and 4.2 of the present paper, the RZ dimensions should not alter significantly depending on the curvature of the rigid-lid. Moreover, Biron et. (1996b) have experimentally observed a similar reduction in the RZ dimensions, for another geometry (α=30º). Biron et al. (1996b) also find the upwelling in the same location, suggesting it as the root cause of the reduction of the RZ.

5 CONCLUSIONS

This study arises as a follow up to two previous numerical studies (Ramos et al., 2018, 2019) on experiments by Weber et al. (2001). It tries to answer questions arisen in those studies: (O1) Is the curvature of the rigid-lid significantly improving the secondary flow patterns predicted by LES for the case with a more balanced discharge ratio between the two incoming channels (q=0.58), as was found by Ramos et al. (2019) for the tributary dominant case (q≈0.25)? (O2) What is the influence of the bed elevation discordance at the interface between the tributary and the main channel onto the 3D flow features, more specifically onto the position and distortion of the mixing layer between the merging flows and the occurrence of upwelling motion near the downstream corner? In this scenario, what causes the reduction of the RZ dimensions (as Đorđević (2013) and Ramos et al. (2018) have shown) when the bed elevations of the confluence are discordant?

In view of objective O1, we have firstly assessed the effects of the curvature of the rigid-lid on the discordant bed open-channel confluence flow, by comparing the water level predictions, the streamwise velocity profiles and secondary flow vector fields between the LES simulation and the experimental dataset of Weber et al. (2001) at moderately low Froude number (Fr=0.37). Contrarily to what happened for a LES of a dominant tributary flow confluence (q=0.25, Ramos et al., 2019) for the same geometrical configuration, the flat and curved rigid-lid produced approximately the same predictions of secondary flow. More research is needed to understand the difference in sensitivity to the curvature of the top boundary between the case where the tributary inflow is dominant (q≈0.25) and the more balanced inflow scenario (q=0.58).

Concerning objective O2, a discordant bed case (α=0.25) has been compared to the associated concordant bed case (α=0.0). The distortion of the mixing layer due to the discordance of bed elevations is demonstrated for this specific case. This phenomenon has been reported consistently in earlier studies of open-channel confluences with discordant beds (Best and Roy (1991), Biron et al. (1996b)). The results seem to match with the presumptions of Biron et al. (1996b), where it was theorized that the pressure differential created by the separation zone in the lee of the step (avalanche face) draws fluid from the mixing layer on the main channel side in towards the bed. Interestingly, the secondary flow (at cross-sections x/W=2, 3 and 5) is substantially different from other flow ratio cases (see, for example, Shakibaeria et al., 2010; Chen et al., 2017; Ramos et al., 2019), but the distortion of the mixing layer is quite similar. Additionally, an upwelling motion occurs as in confluence cases with a different junction angle (see Biron et al., 1996a, b), at a similar location (downstream confluence corner) and with comparable impact on the recirculation zone dimensions (vanishing RZ near the bed).

Note that the present paper only considered time-averaged flow results. The oscillations of the mixing layer, a phenomenon reported earlier in the literature (Biron et al., 1996b; Bradbrook et al., 2004) will be studied in near future.
ACKNOWLEDGEMENTS

This work was performed using the computational facilities of the HPC infrastructure of Ghent University.

REFERENCES


