Original research

Numerical simulation of effect of inflow Froude number on flow separation zone in junctions

ABSTRACT:
Junctions are generally used in water distribution networks, irrigation channels, sewage networks, water/wastewater treatment facilities, input to power generation facilities, and etc. In this study, the flow numerical simulation has been performed in a 90° junction using RSM turbulence model. Water surface and flow velocity profiles at different sections of the main and lateral channels were compared with the experimental results and a good agreement has been found between them. Then, considering the good results obtained from previous sections, Froude number effect of the main and lateral channels entrance on flow separation zone was studied. The results showed that for a ratio of constant discharge, with approaching to channel bed, length of circulation zone is also decreased. Moreover, stream tube dimensionless width in water surface and bed were increased and decreased, respectively.

Keywords:
T-Junction, Flow Pattern, Inflow Froude number, Separation Zone, RSM turbulence model

Authors:
Farhad Majidian and Maaroof Siosemarde

Institution:
Department of Civil Engineering, Mahabad Branch, Islamic Azad University, Mahabad, Iran.

Corresponding author:
Maaroof Siosemarde

Email Id:
maaroof-33m@yahoo.com

Web Address:

Article Citation:
Farhad Majidian and Maaroof Siosemarde
Numerical simulation of effect of inflow Froude number on flow separation zone in junctions

Dates:
Received: 20 Jan 2017    Accepted: 21 Jan 2017    Published: 25 Feb 2017

This article is governed by the Creative Commons Attribution License (http://creativecommons.org/licenses/by/4.0), which gives permission for unrestricted use, non-commercial, distribution and reproduction in all medium, provided the original work is properly cited.
INTRODUCTION

Open-channel junction flow is important in environmental and hydraulic engineering. It occurs in many hydraulic structures, such as irrigation systems, urban water supply and wastewater treatment facilities. The flow entering the lateral channel has complex specifications and leads to the creation of flow separation zones at the downstream corner of the junction. Different parameters effect in hydrodynamics of junction, flow that characterize the flow pattern. These include the shape, slope, angle between lateral and main channels and Froude numbers. In the past, study of junction flows has been performed using physical models, 1-D theoretical analysis, or 1-D numerical solutions. Taylor (1944) was probably the first one who applied an analytical model to open-channel junction flows. This model was able to predict the tributary channel depth upstream of the junction. Modi et al. (1981) calculated the size of the separation zone using a conformal mapping method under the assumption of in viscid and irrotational flow. A shape index of flow separation zone was obtained in the T-Junction with deviation different angles and discharge ratios by Best and Reid (1984). Chen and Lian (1992) simulated the geometry of T-junction using standard K-ε model with above Reynolds numbers. The results obtained for the ratio of small discharges had good agreement with experimental results, but in the higher discharges ratio, were significantly different from that of the measurements. Issa and Oliveira (1994) performed 3-D simulation of turbulent flow for T- shaped geometries. They solved Reynolds time-averaged Navier–Stokes equations (RANS) through K-ε-Standard model with wall functions. Ramamurthy et al. (2007) studied the transfer of lateral momentum from the branch channel to the main channel by measuring the pressure difference between the side banks of the branch channel. Biron et al. (1996) investigated the previously neglected effects of bed discordance on the channel junction flow. Hsu et al. (1998a) described a one-dimensional approach to predict the water depth upstream of the junction for a subcritical open channel and they were able to estimate the contraction coefficient at the maximum flow constriction. Their results correlated well with the available experimental data. Later, Hsu et al. (1998b) extended their previous model to investigate the flows in channels with different junction geometries, including of 30°, 40°, and 50° angles. Huang et al. (2002) numerically investigated water surface and flow velocity profiles at different sections of the main and lateral channel using k-ω turbulence model. Then, an empirical relation between the junction losses, the junction angle and discharge ratio were suggested. Ting et al. (2009) simulated the 90° equal-width open-channel junction flow using the 3-D k-ω model. Then applied to investigate the effect of the discharge ratio on the shape of separation zone cross-sectional mean flow angle and the contraction coefficient. The studies mentioned above provided useful information for understanding of the flow and engineering design of open-channel junctions. However, theoretical or numerical models aren’t capable of taking some complex flow conditions into account such as effect of inflow Froude number on secondary flow and separation. In the present study, Froude number effects of the main and lateral channels entrance on flow separation zone are studied.

MATERIAL AND METHODS

For numerical simulation of the turbulent flow field, Fluent software with RSM turbulent model was used. Reynolds-averaged Navier–Stokes (RANS) equations are solved in addition to continuity equation. Quick and second Order Upwind schemes were used for solving momentums and turbulence equations, respectively. Moreover, standard method was used for discretization of pressure equation. Flow analysis occurring at persistent mode and PISO algorithm is used for velocity-pressure coupling. Method of discretization
of momentum equation, loss and turbulence kinetic energy and reynolds stress is done in second order upwind method. Also, standard method is used for discretization of pressure equation. The small values of the under-relaxation factors are required for the solution stability of the mentioned method, so the under-relaxation factors are chosen between 0.2 and 0.5. It should be noted that for calculating an unsteady free surface profile, a small initial time steps are required. Therefore, a time step between 0.01 and 0.1 are selected. Iterative convergences at every time step are checked and all residuals are dropped below three orders (1e-3).

Solution procedure was continued until achieving 1e-4 for normalized residuals of each solved equation (Fluent Inc, 2006).

\[ U_k \frac{\partial u_i}{\partial x_k} = P_{ij} + \Phi_{ij} + D_{ij} - \varepsilon_{ij} \]  

(3)

Where, ‘\( \mu_{ij} \)’ is Reynolds stress, ‘\( P_{ij} \)’ is production tensor, ‘\( \Phi_{ij} \)’ is strain– pressure tensor, ‘\( D_{ij} \)’ is diffusion tensor and ‘\( \varepsilon_{ij} \)’ is dissipation tensor (Fluent Inc, 2006).

In this study, the experimental data of Weber et al. (2001) were selected for validation of the numerical simulation. In experimental study of Weber et al. (2001), the main channel with the length of 21.95 m and the lateral channel with length of 3.66 m located at 90° have been investigated. The inlet velocities of lateral and main channels are equal to 0.5 m/s and 0.166 m/s, respectively. The outlet discharge of the main channel downstream is 0.17 m³/s, ratio of diverted discharge (q*) is 0.25, depth of downstream flow is 0.296 m Reynolds number (Re) is 186000 and width of both the channels (W) is 0.91 m. The simulation domain has been carefully chosen to represent the inflow and outflow boundaries. It is noted that the free-surface elevation is determined numerically and is not known before the computation. Therefore, the main channel length upstream of the branch channel is lengthened to 12W=10.92 m and the branch channel length is prolonged to 10W=9.10 m; so that, the fully developed flows are obtained near the downstream of the inlet boundaries. The length of main channel downstream of the branch channel is determined with 10W, where the flow depth is nearly constant (Weber, 2001).

In 3-D simulations performed in the present study, the boundaries depending on the nature of the flow were solid walls, inlet, outlet and free-surface. Two separate inlets were specified for air and water that at each inlet, uniform distributions were given for all of dependent variables. The water velocity was defined at the inlet section with a known water depth entering the flow domain and inlet discharge and the applied air velocity at the flow inlet was equal to 1e-5 m/s. The
turbulence kinetic energy \( (k_0) \) and dissipation rate \( (\varepsilon_0) \) at the inlet boundary were estimated by the following equations (Fluent Inc, 2006):

\[
k_0 = \frac{3}{2} (T_u U_0)^2
\]

(4)

\[
\varepsilon_0 = C_\mu^{3/4} k^{3/2} l
\]

(5)

Here, \('U_0'\) refers the average inlet velocity, \('T_u'\) denotes the turbulence intensity which is typically between 1% and 5% and it depends on the upstream flow. \('C_\mu'\) represents constant and it is equal to 0.09 and \(l\) denotes \(0.07D_h\) in which \(D_h\) is the hydraulic diameter. At the air surface, symmetry conditions were applied which enforce a zero normal velocity and a zero shear stress there. To estimate the wall effects on the flow, empirical wall functions known as standard wall functions were used. The RSM model utilized the wall functions approach for linking the gaps between the logarithmic layer and the near-wall viscous sub layer.

Appropriate meshing of regional flow exists is also one of the important parameters in the run time model. Figure 1 shows the three-dimensional plan and view of computational domain meshing in the \(90^\circ\) junction. The dimension of cells number of computational domain meshing in the \(x\), \(y\) and \(z\) directions in zones 1, 2, 3 and 4 are \(47\times40\times22\), \(40\times40\times22\), \(47\times40\times22\) and \(40\times48\times22\), respectively. A finer mesh used did not show an appreciable difference in the results.

RESULTS

Evaluation of water surface profiles

In this section, water surface profiles in different cross-sections of the main channel are investigated for constant discharge ratio of 0.25. In Figure 2, the dimensionless water surface profiles \((Z^*)\), for different cross-sections of the main channel for discharge ratio of 0.25 are illustrated. \(X^*, Y^*\) and \(Z^*\) are distances at \(x\), \(y\) and \(z\) axes, respectively which have become dimensionless by the width of lateral and main channels.

According to Figure 2, water surface profiles at different sections of the main channel are compared with the experimental results and a good agreement is found between them. It is seen that similar results of Ting et al. (2009), the surface elevation nearer to the branch channel (with less \(Y^*\)) has more smart depression. Table 1 presents the mean error percentage obtained from the comparison of the numerical values with experimental results.

<table>
<thead>
<tr>
<th>Section</th>
<th>(Y^*=0.167)</th>
<th>(Y^*=0.333)</th>
<th>(Y^*=0.500)</th>
<th>(Y^*=0.667)</th>
<th>(Y^*=0.883)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present Study</td>
<td>0.72</td>
<td>1.25</td>
<td>1.36</td>
<td>0.52</td>
<td>0.63</td>
</tr>
<tr>
<td>Ting et al. (2009)</td>
<td>2.01</td>
<td>3.08</td>
<td>3.68</td>
<td>1.61</td>
<td>0.55</td>
</tr>
</tbody>
</table>

Table 1. The mean error percentage obtained from the comparison of the numerical values with experimental results.

Figure 1. Computational geometry and grid, a) plan and b) three-dimensional view.
comparison of the numerical values resulted from the present study and Ting et al. (2009) study with experimental values at different sections of the main channel. For error percentage calculation, Root Mean
Square Error method (RMSE) is used. The RMSE of a model prediction with respect to the estimated variable is defined as the square root of the mean squared error:

\[ \text{RMSE} = \sqrt{\frac{\sum_{i=1}^{n}(X_{\text{obs},i} - X_{\text{model},i})^2}{n}} \]  

Where ‘\(X_{\text{obs}}\)’ is observed values and ‘\(X_{\text{model}}\)’ is modeled values at time/place ‘\(I\)’.

According to Table 1, in all sections except section \(Y^*=0.883\), present study results have better agreement with the experimental results than Ting et al. (2009) results. It indicates the high ability of this numerical model in predicting distribution of water surface elevation profiles.

In continue, flow velocity profiles in different cross-sections of the junction are investigated for constant discharge ratio 0.25. In Figures 3 to 6, the dimensionless velocity profiles \((U_x/U_0)\) near the water

<table>
<thead>
<tr>
<th>Section</th>
<th>(x^*=1)</th>
<th>(x^*=1)</th>
<th>(x^*=1)</th>
<th>(x^*=6)</th>
<th>(x^*=1)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(y^*=)</td>
<td>(y^*=)</td>
<td>(y^*=)</td>
<td>(y^*=)</td>
<td>(y^*=)</td>
</tr>
<tr>
<td>Present Study</td>
<td>1.54</td>
<td>1.89</td>
<td>1.58</td>
<td>0.78</td>
<td>1.90</td>
</tr>
<tr>
<td>Ting et al. (2009)</td>
<td>2.01</td>
<td>6.64</td>
<td>1.85</td>
<td>3.92</td>
<td>9.11</td>
</tr>
</tbody>
</table>

Table 2. The mean error percentage obtained from the comparison of the numerical values with experimental

Figure 3. Comparison of computational velocity profiles at \(X^*=1\) in main channel

Figure 4. Comparison of computational velocity profiles at \(X^*=-1\) in main channel
level, for different cross-sections of the lateral and main channels are shown. ‘U₀’ is the maximum velocity at the cross-section X*=-6 whose value is equal to 0.28 m/s.

Figure 3 shows the upstream regions of main channel (at X*=1) here, the velocity is evenly distributed across the section. Due to the momentum inflow of lateral channel and the influence of separation zone, the flow in the junction region shifts to the wall opposite to the junction and the flow distribution is quite non-uniform (Figure 6). At the location of Y*=0.25, it is apparent that higher velocities occur near the bed. Similar results of Ting et al. (2009) downstream of the separation zone (X*=-6), flow begins to recover from the effect of separation and to distribute more evenly. For completeness of the comparison, the transverse velocity in the lateral channel is also compared in Figure 8. The results indicated that transverse velocity is very small in
the upstream of the branch channel ($Y^*=-1$), indicating that the secondary flow is negligible at these cross sections. However, transverse velocity close to the junction starts to increase, indicating that the secondary flow is quite strong (Issa and Oliveira, 1994).

Table 2 shows the mean error percentage obtained from the comparison of the numerical values resulted from the present study with experimental values at different sections of the lateral and main channels. According to Table 2, in all sections, present study results have better agreement with the experimental results than Ting et al. (2009) results. Average RMSE obtained of the Ting et al. (2009) was 5.94% while in present study this amount was 1.97%.

**DISCUSSION**

![Figure 8. Streamlines in different depths for constant Froude number of main channel entrance 0.37](image1)

![Figure 9. Streamlines in different depths for constant Froude number of lateral channel entrance 0.37](image2)

![Figure 10. Changes of the flow separation zone dimensions in different depths for different Froude numbers of main channel entrance](image3)
Figure 7 shows circulation zone in main channel and different parameters. In this Figure, ‘L, W, and B’ are length and width of circulation zone and stream tube width, respectively.

Figures 8 and 9 show streamlines near the water surface, middle depth and near the bed for constant Froude number of main and lateral channels entrance (Fr).

According to the Figures 8 and 9, Figures 10 and 11 show changes of the flow separation zone dimensions in different depths for different Froude numbers of main and lateral channels entrance, respectively.

According to the Figures 10 and 11, Froude numbers of main and lateral channels entrance are important parameters on length and width of circulation zone in different depths and their increasing leads to the increase of length and width of circulation zone. For a ratio of constant discharge, with approaching to channel bed, length of circulation zone decreases. Also, width of circulation zone nearer to the water surface is larger than the bed. With increasing Froude number of the main and lateral channels entrance, stream tube dimensionless width in water surface and bed increases and decreases, respectively.

REFERENCES


Chen HB and Lian GS. (1992). The numerical...


