International Symposium on Advances in Science and Technology
Bandar-Abbas | Iran
7-8 March 2013
7th SASTech
Persian Gulf
Hosted by:
Hormozgan University
Organised by:
Khavaran Institute of Higher Education

Effect of bed roughness on flow dynamics at Open Channel Confluences

P. Yavari, Msc Student of Hydraulic Engineering, Parastoo_yavari@yahoo.com
Department of Civil Engineering, Razi University, Kermanshah, Iran

A.A. Akhtari, Assistant Professor, Akhtari63aaa@yahoo.com
Department of Civil Engineering, Razi University, Kermanshah, Iran

Paper Reference Number: 07-98-5656
Name of the Presenter: parastoo yavari

Abstract

Open-channel confluence is a common occurrence in many hydraulic structures. Flow dynamics in the junction field is too complex. Compared to experimental findings , three dimensional numerical simulation can be useful to obtain more comprehensive insight of dynamics of the flow and will be useful in the hydraulic engineering projects. This paper numerically investigates the three dimensional flow pattern at the 90 °, sharp edges, rectangular open-channel confluences, using Navier-Stokes equations and $RNG\ k-\epsilon$ turbulence model. The equations are solved by finite volume method and unsteady flow conditions. The flow is analyzed in two-phase conditions. In the beginning , the numerical model has been validated by comparing with experimental data of shumate (1998). Then , the effect of bed roughness on flow characteristics is investigated. The numerical results show that the width of the separation zone is reduced by bed roughness.

Key words: 90^0 confluence angle, bed shear stress, numerical modeling, turbulent flow, bed roughness

1. Introduction

Channel confluences occur in many hydraulic structures such as rivers . A separation zone would be formed immediately downstream of the junction because of incoming flow from side channel to main channel flow. Figure 1 shows the flow pattern at the junction. As can be seen from Figure 1, there are six distinct regions in the junction:

1- stagnation zone , immediately upstream of junction 2- The flow diversion zone 3- flow separation zone 4- maximum velocity zone 5- flow recovery zone 6- shear plane zone

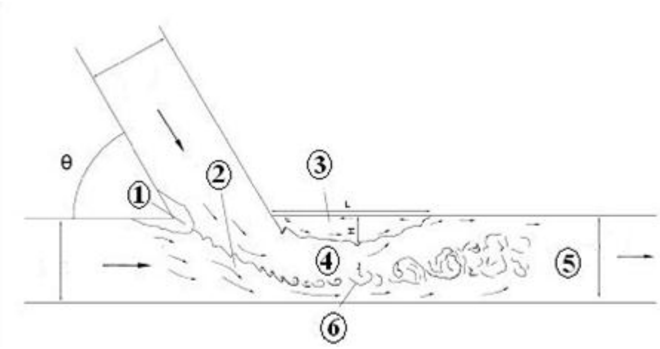


Fig. 1. Flow pattern at open-channel confluences [Goudarzizadeh et al (2011)]

Many parameters affect the flow dynamics of open channel junction. Several studies have been carried out in this respect. One of them is Taylor's research that could analytically predict the water depth in the upstream of the junction. Rodi & Weiss (1980) investigated the influence of bed roughness on the separation zone. They found that both the width and length of the separation zone were reduced by the bed roughness. Demuren & Rodi (1983) carried out a three-dimensional simulation of side channel into smooth-bed and rough-bed open channels using the standard $k-\epsilon$ model. Best & Reid (1984) conducted an experimental study of open channel confluences with various junction angles and flow ratios. Shumate et al (1998) performed an extensive experimental study of 90 degree open channel confluences , with objective of providing the comprehensive data set that can be used to numerical models validation. Wang & Cheng (2000) simulated a three-dimensional side discharge into a cross channel flow by a RNG based $k-\epsilon$ model. Effects of the jet to the cross channel width ratios on the size of the recirculation zone also investigated in detail, and correlation formulas of recirculation length and width depending on velocity ratio and width ratio proposed. The free surface boundary condition was considered by a rigid-lid assumption.

Huang et al (2002) developed a three-dimensional multi-blocks CFD model with capability of prediction of the water surface variations for 90 degree open channel junction simulation. Shakibainia et al (2010) used the SSIIM 2.0 to simulating the three-dimensional 90 degree open-channel junction flow.

Gudarzizadeh et al (2011) analytically examined the flow separation zone and the quality of the bed shear stress distribution.

In most of the studies conducted by previous researchers, effect of the bed roughness is neglected , while in the nature , the open channels bed are not perfectly smooth. Also , since the two- phased modeling is costly , in many cases , the variations in the water surface are ignored. Therefore, the aim of this paper is to provide a three-dimensional numerical method to simulate the open channel junction flow that can be useful in many engineering projects. Then, the effect of the bed roughness on flow dynamics at open channel confluences is investigated. FLUENT 6.3.26 is used to simulating the model.

2.Data and Material

A. The Laboratory Model Specifications

The experimental study conducted by shumate (1998), is used for validation of the numerical model. Figure 2 shows the characteristics of the experimental model. As can be seen from Figure 2, the channel consists of a 21.946m main channel and a 3.658m side channel. The side channel is located 5.486m downstream of the main channel input. The coordinate origin is located in the bed at upstream corner of the junction. The positive *x*-axis oriented in the upstream direction of the main channel. The positive *y*-direction points to the main channel wall opposite of the channel junction. Thus the positive z-axis is upward in the vertical direction. Width of both channels is 0.914 m that define as W. depth of channel is 0.509m. The bed of the channel at all locations is horizontal.

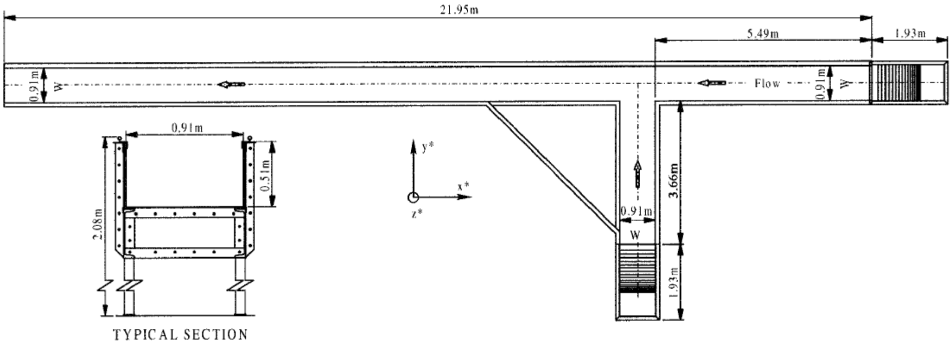


Fig. 2. Layout of experimental flume [Huang et al (2002)]

The total flow rate (combination of the upstream main channel flow and side channel flow), Q $=0.17\, {}^{m^3}\!/_{s}$ and tailwater depth, $H_0 = 0.296$ m ,are held constant. Therefore, this constant Q and H_0 produced the tailwater flow velocity $U_0 = 0.628\, {}^{m}\!/_{s}$ and Froude number Fr=0.37 . Experiments were performed at six RUNS with various flow rate. In this article discharge ratios $q^* = 0.25$ and $q^* = 0.75$ (q^* is ratio of upstream main channel flow to combined flow) are applied to validation. Important parameters are showed in table 1

Table1
Hydraulic characteristics of 90 degree channel junction

q^*	$H_0(m)$	$U_0(m/s)$	Fr	Re
RUN5 0.75	0.296 m	0.628	0.37	186,000
RUN2 0.25	0.296 m	0.628	0.37	186,000

B. Governing Equations

The governing equations for open-channel flows are the Reynolds-averaged Navier–Stokes (RANS) equations with assuming the steady-state and incompressible flow. The Continuity and Navier-Stokes equations can be written as: [Rodi (1979)]

Continuity equation :

$$\frac{\partial u_i}{\partial x_i} = 0 \qquad (1)$$

Momentum equation :

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j}(u_i u_j) = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + g_i + \frac{\partial}{\partial x_j}[\tau_{ij}] \qquad (2)$$

The present study also has used the finite-volume method (FVM) to discretize the equations.

3. Research Methodology

A. Boundary Conditions

In the main and side channel inlet, average velocity is adapted. Since the water depth is determined numerically and is not known before the computation, the length of the main and side channels are lenghtened so that the uniform velocity distributions are prescribed at the inflow boundaries. The constant water surface elevation ($H_0 = 0.296$ m) is adapted for the outlet boundary. Stationary wall and no slip are used for wall and bottom boundaries. The standard wall function is applied here in order to establish a relation between the sub-layer and the fully turbulent layer. The variations in the water surface are considered (two-phased model) .

B. Turbulent Model

$RNG\ k-\epsilon$ turbulence model uses the below transport equations [Yakhot et al (1992)]

$$\frac{\partial k}{\partial t}+\frac{\partial}{\partial x_j}(ku_i)=P-\epsilon+\frac{\partial}{\partial x_j}\left[\alpha_k(\nu+\nu_t)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

$$\frac{\partial \epsilon}{\partial t}+\frac{\partial}{\partial x_j}(\epsilon u_i)=C_{\epsilon 1}^{*}\frac{\epsilon}{k}P-C_{\epsilon 2}\frac{\epsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\alpha_\epsilon(\nu+\nu_t)\frac{\partial \epsilon}{\partial x_j}\right] \tag{4}$$

$$P=2\nu_t\overline{S_{ij}S_{ij}} \quad ; \quad \bar{S}_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad ; \quad \nu_t=C_\mu\frac{k^2}{\epsilon} \tag{5}$$

$$C_\mu=0.0845\ ;\ \alpha_k=1.39\ ;\ \alpha_\epsilon=1.39\ ;\ C_{\epsilon 1}=1.42\ ;\ C_{\epsilon 2}=1.68$$

$$C_{\epsilon 1}^{*}=C_{\epsilon 1}-\frac{\eta(1-\eta/\eta_0)}{1+\beta\eta^3} \qquad \eta=sk/\epsilon\ ,s=\sqrt{2\overline{S_{ij}S_{ij}}}\,,\eta_0=4.377,\beta=0.012$$

4. Results and Analysis

Figure 3 shows the comparison of streamwise velocities (u) with experimental findings at different cross sections for RUN5. The velocity components are nondimensionalized by downstream velocity and x , y and z components are nondimensionalized by width of the channel. Figure 4 shows the comparison of main channel water surface elevation (h) with experimental data, at different profiles along the flow direction for RUN2. It is seen that a good agreement between predicted and measured data is obtained. Figure 5 shows the average Root-Mean-Squrae Error of streamwise velocities at different cross sections for RUN5. Figure 6 shows the Root-Mean-Squrae Error of water surface elevation at different profiles along the flow direction for RUN2

The Root Mean Square Error (RMSE) is a frequently used measure of the difference between values predicted by a model and the values actually observed from the experiment that is being modeled.

The RMSE of a model prediction with respect to the estimated variable X_{model} is defined as the square root of the mean squared error:

$$\mathbf{RMSE}(\%) = \sqrt{\frac{1}{n}\sum_{i=1}^{n}\left(\frac{X_{obs,i}-X_{mod,i}}{X_{obs,i}}\right)^2} \qquad (6)$$

Where X_{obs} is observed values and X_{model} is modeled values at time/place i.

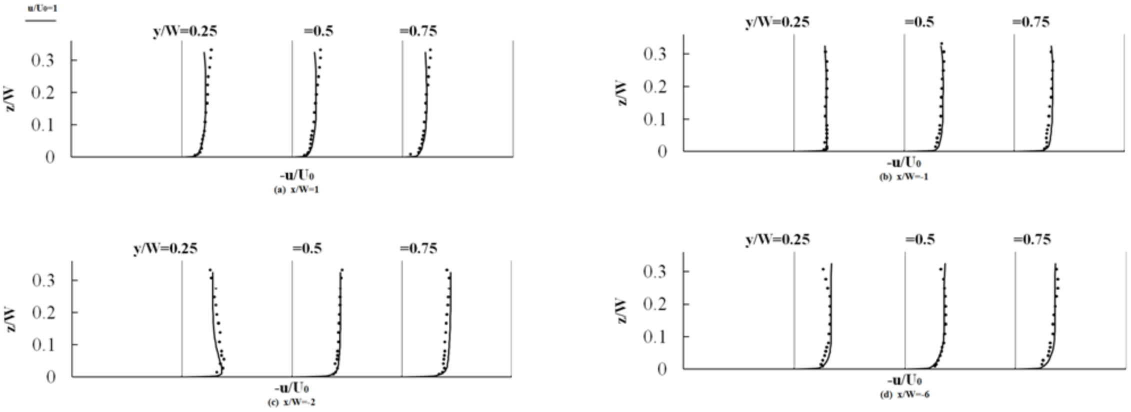
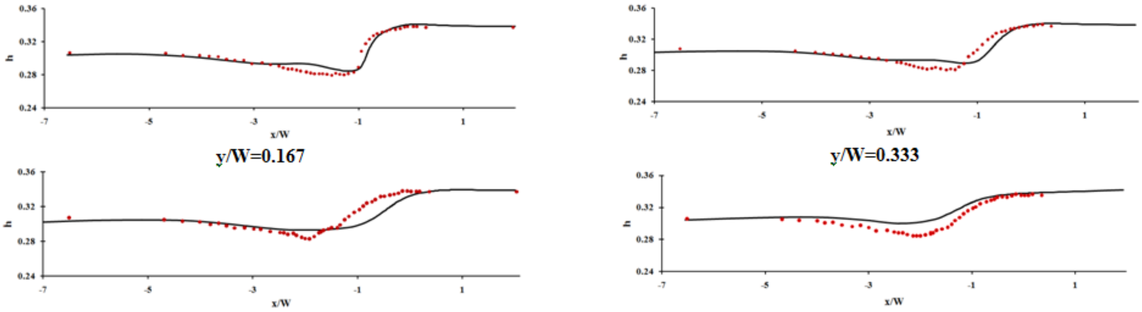


Fig .3. Comparison of streamwise velocities at selected stations for RUN5. Circles are experimental data by Shumate; solid lines are present calculations.



y/W=0.167

y/W=0.333

y/W=0.667

y/W=0.883

Fig .4. Comparison of main channel water surface elevation profiles along the flow direction for RUN2. Circles are experimental data by Shumate; solid lines are present calculations.

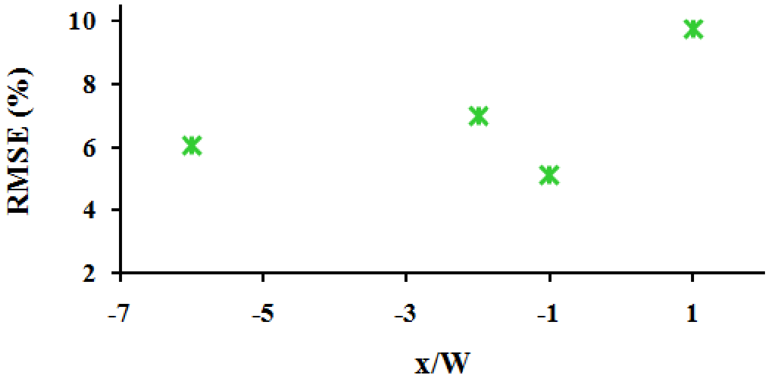


Fig .5. Average RMSE of streamwise velocities at different cross sections (u/U0) for RUN5

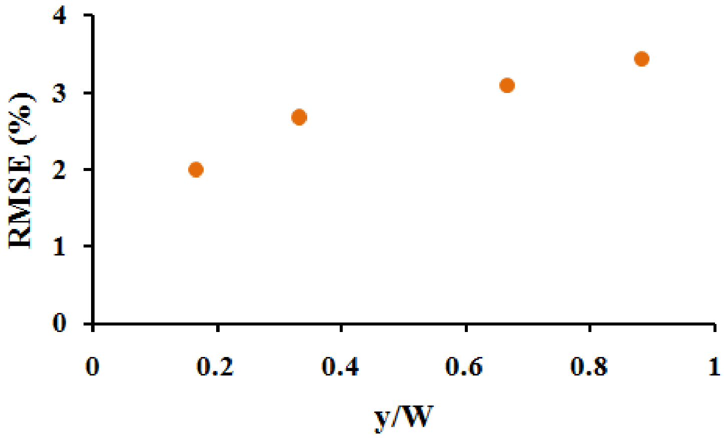
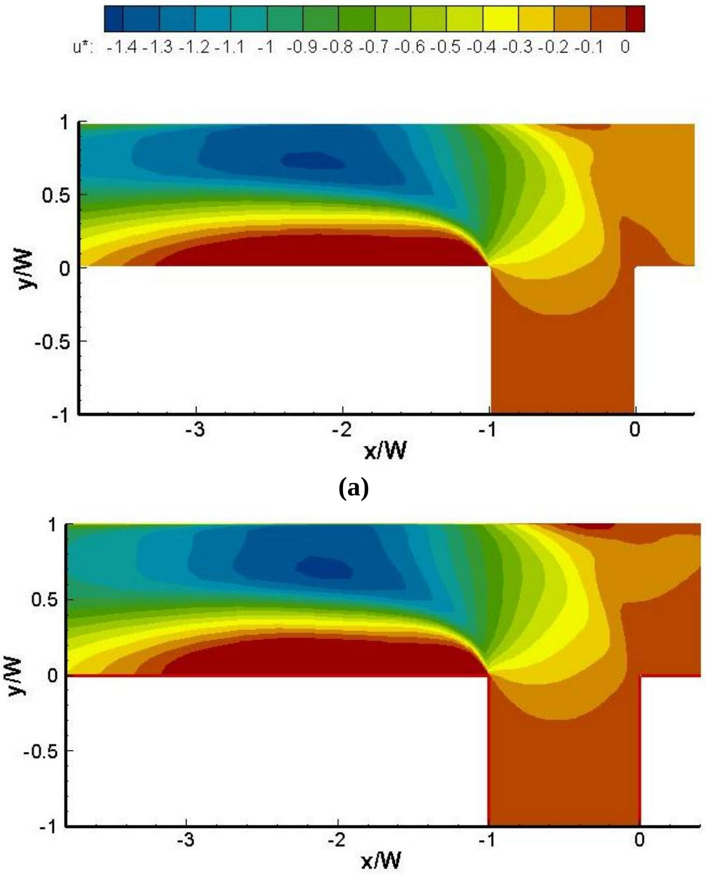


Fig .6. RMSE of water surface elevation at different profiles along the flow direction for RUN2

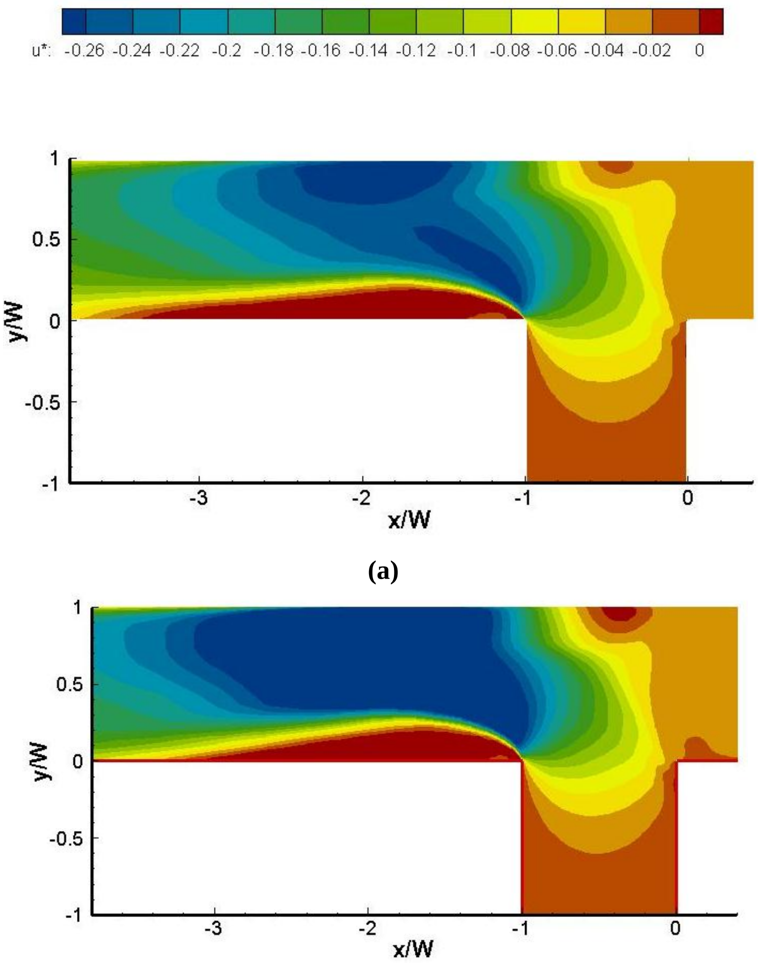
Then, the effect of bed roughness on dimensions of the separation zone is investigated. Figure 7 , 8 show the u velocity distribution for RUN2 , with and without bed roughness. In the case with bed roughness, the roughness height in the bed is considered 1.5 mm.



(a)

(b)

Fig .7. u velocity distribution for RUN2 , near the water surface (z/W=0.278) : a) rough bed , b) smooth bed



(a)

(b)

Fig .8. u velocity distribution for RUN2 , near the bed (z/W=0.014) : a) rough bed, b) smooth bed

As can be seen from Figures 7 and 8 , width of the separation zone is reduced by increasing the bed roughness. This causes the velocity decreases in the maximum velocity zone. This is particularly important in the vicinity of the channel bottom, So that the scouring is reduced by decreasing the maximum velocity and this can be useful in engineering design.

5. References

1. Demuren, A.O., Rodi, W. (1983). " Side discharge into open channels: mathematical model". J. Hydraul. Eng., 109(12):1707±22.

2. Goudarzizadeh, R., Mousavi Jahromi, S.H., Hedayat, N., and Naghshineh, M.H. (2011). "Numerical Investigation of the Flow Separation Zone and Quality of the Bed Shear Stress Distribution at Open-Channel Confluences" World Academy of Science, Engineering and Technology

3. Hager, W.H. (1989). " Transitional flow in channel junctions." J.Hydraul. Eng., 115(2), 243-259.

4. Hsu, C.C., Lee, W.J. and Chang, C.H. (1998b). "Subcritical Open-Channel Junction Flow" J. Hydraul. Eng., 124 (8), 847-855

5. Hsu, C.C., Wu, F.S., and Lee, W.J. (1998a). “ Flow at 90 degree Equal-Width Open-Channel Junction.” J. Hydraul. Eng., 124 (2), 186-191

6. Huang , J., Weber , L.J. , Lai , Y.G. (2002) “ Three-Dimensional Numerical Study of Flows in Open-Channel Junctions ” J. Hydraul. Eng. Vol. 128, No. 3 , March 1.

7. Rodi, W. (1979)."Turbulent models and their application in hydraulics-at state-of-the art review".IAHR, Delft, Netherlands.

8. Rodi W, Weiss K. Laboruntersuchungen des Nahfelds von warm-wassereinleitungen in rechteckgerinne. Research Report No. 574, Institute for Hydromechanics, University of Karlsruhe, W. Germany, 1980.

9. Shumate, E.D. (1998). “Experimental description of flow at an open channel junction”. Master thesis, Univ. of Iowa, Iowa

10. Weber, L. J., Schumate, E. D., and Mawer, N. (2001). “ Experiments on flow at a 90° open-channel junction.” J.Hydraul. Eng., 127(5), 340-350.

11. Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. & Speziale, C.G. (1992). "Development of turbulence models for shear flows by a double expansion technique". Physics of Fluids, Vol. 4, No. 7, and pp1510-1520