Available online at www.CivileJournal.org

Civil Engineering Journal

Vol. 2, No. 10, October, 2016

Analyzing Parameters Influencing Scour Bed in Confluence Channels Using Flow3D Numerical Model

Behnam Shamohamadi [a]*, Ali Mehboudi [a]

[a] *Department of Civil Engineering, Yasooj Branch, Islamic Azad University, Yasooj, Iran.*

Received 15 August 2016; Accepted 22 October 2016

Abstract

Channels junction is a phenomenon which is used in most of irrigating and drainage networks and in hydraulic engineering in general. In two channels junction, main and secondary channels encounter with each other and move to the channel downstream. Scour holes and sedimentation zones are created in channels bed. 3D simulation of scour hole created in these channels is influenced by various factors. The ratio of main channel width to secondary channel width is one of the most important influencing factors. This parameter is the main focus of the present research. In the present study, a model calibrated with laboratory results has been simulated. The numerical model results have revealed that decreasing the ratio of main channel width to secondary channel width causes the secondary channel flow encounters to the front wall of the secondary channel. Also, it leads to creating scour near the front wall and the main hole is drawn towards the wall. Furthermore, in the present research, topographical changes of the bed with running time of the numerical model for the middle channel axis has been extracted and presented.

Keywords: Scour Hole; Numerical Simulation; Secondary Channel; Open-Channel Junction.

1. Introduction

As one of the complex phenomenon considered in open channels related studies, it can be referred to the movement of deposition s with water flow and deposition or erosion of channel. This phenomenon is more complex in channels junction. Such that, flow behavior depends on some variables such as discharge ratios, branches width, the junction angel of two branches of channel, bed balance variation at junction, flow characteristic in the upstream of each branches as well as flow Froude number. As evident, the maximum depth of scour occurs in particle movement threshold conditions, i.e. the border between clear water and water containing deposition. Under clear water conditions, the scour depth is increased a result of flow velocity increase; also, the maximum scour depth occurs under clear water conditions and at deposition particle movement [1, 2].

Paphitis (2000) investigated the movement of deposition particles under one dimensional flows using movement threshold curves. Using these curves, movement threshold of deposition particles was predicted. To define movement threshold states, there are no definite stochastic theories under which deposition particles can be replaced. Nevertheless, movement threshold responses range always exists [3]. Arumugam and Mason (1985) performed many studies on various relations resulted by turbulences using 47 laboratory models. According to analyzing the laboratory resulted data, they found the factor of d50 more appropriate than d90 and d85 to investigate the phenomenon of deposition by water [4]. Silviaand Gerardo (1999) explored scour hole in tidal channels junction. To identify the origin and gradual growth pattern, this study was conducted on the hole created in Baha-Blanca Estuary located in Argentina. The experiments were performed on two large holes located at Baha-Blanca Estuary. Based on the researcher observations, determining net value of carried deposition and gradual growth of holes created depends on the range of active flooding and tidal range. Moreover, it was observed that holes in the selected species moves towards upstream just like holes created in rivers [5]. Using 2D model of Boltzmann, Liu et al. (2008) simulated sub-critical flow in

---

[*] Corresponding author: b1986sh@yahoo.com

confluence channels. They also presented the characteristics of flow in junction with velocity vectors and flow lines [6]. In the study conducted by Biswal et al. (2016), hydraulic parameters changes of flow were explored at depth direction. To this end, they used turbulence models to investigate various velocity values in flow depth and water surface profile under different geometrical conditions of channel [7].

2. Methodology

To numerically model local scour at open-channel junction using Flow3D, based on the setting applied by user, the four stages including navigator, model setup, simulate, analyze, and display should be performed. Of course, it should be noted that all information necessary for simulation can be entered into the text file. For ease of ding, there is the possibility of graphically entering information.

The variables investigated in the present study are as following:

Q_d: Total discharge or downstream discharge of main channel

$Q_{bd} = Q_b/Q_d$: Discharge ratio of secondary channel to main channel

$B_{bd} = B_b/B_d$: Width ratio of secondary channel to main channel

In this study, modeling was performed in to sections including movement threshold investigation and local erosion pattern in channels junction. The following tables show the variables range in each part of the experiments.

Table 1. The variables range in modelling movement threshold of bed materials for scour in two channels junction

B_{bd} (B_b/B_d)	Q_{bd} (Q_b/Q_d)	Q_d (Lit/s)
1	0.8, 0.6, 0.4, 0.3	15.5 , 10, 8.5, 7.5

Table 2. The variables range in modelling local erosion pattern in t two channels junction

B_{bd} (B_b/B_d)	Q_{bd} (Q_b/Q_d)	V/V_c	Q_d (Lit/s)
1, 0.5	0.8, 0.5, 0.25	0.5, 0.8	7.5

To obtain the best answer from Flow3D Software and comparing the possible alternatives, appropriate boundary initial conditions as well as the study of the sensitivity of the model relative to various parameters affecting the modelling, different tests were performed. The sensitivity of numerical models to meshing and solving field disconnection has been always one of the important issues in numerical models. In this regard, various meshing patterns were also presented and each of them had their own weaknesses and strengths. Compared to solid boundaries and meshing, Flow3D has a relatively different manner with other computational fluids dynamic packs. To specify the range of meshing, some blocks are determined in which all dimensions of the considered structure and free space are defined. All the considered details can be put in one or more blocks. In the following figure, numerical modelling and Favor method in various meshing have been shown. According to the above mentioned, it can be concluded that creating appropriate solid boundaries in numerical model meshing. Particularly in models with relatively complex geometry and conditions (like the studied model) is the biggest limitation. Also, its appropriate selection can increase the accuracy of computations. In the present work, meshing with two blocks was investigated by different tests and the best meshing was selected for performed simulations.

One of the most important advantages of Flow3D, compared to other flow field simulation models, is determining and estimating the best solution field meshing based on the studied geometrical model. To determine the size of computational field meshing of flow in lateral overflows, computational cells were firstly considered in length, width and height of 0.08 m. In this state, after validating the dimensions of meshing cells by Favor part of Flow3D, it was found that the dimensions of the cells selected for the channel geometry and its lateral walls are not appropriate. In other words, the entire of the exiting accurate geometry is not covered by the 0.08 m- cells. Therefore, computational cells were considered for numerical simulation. In this state, the dimension of the length, width and height was considered 0.05 m. After investigating and validating the dimensions is observed that in this stage, a part of the introduced geometrical model cells, particularly in the channel diagonally encountering to the main channel is not appropriate by the numerical model. In the same way, changing the dimensions of meshing cells in the most optimal numerical model and the most appropriate dimensions of meshing cells with the length of 0.02 m, the width of 0.02 m and the flow depth of 0.02 m were considered. Various conditions of computational cells dimensions have been presented in the following figure. Notably, in this simulation, totally, 1027200 computational cells have been considered for each two blocks.

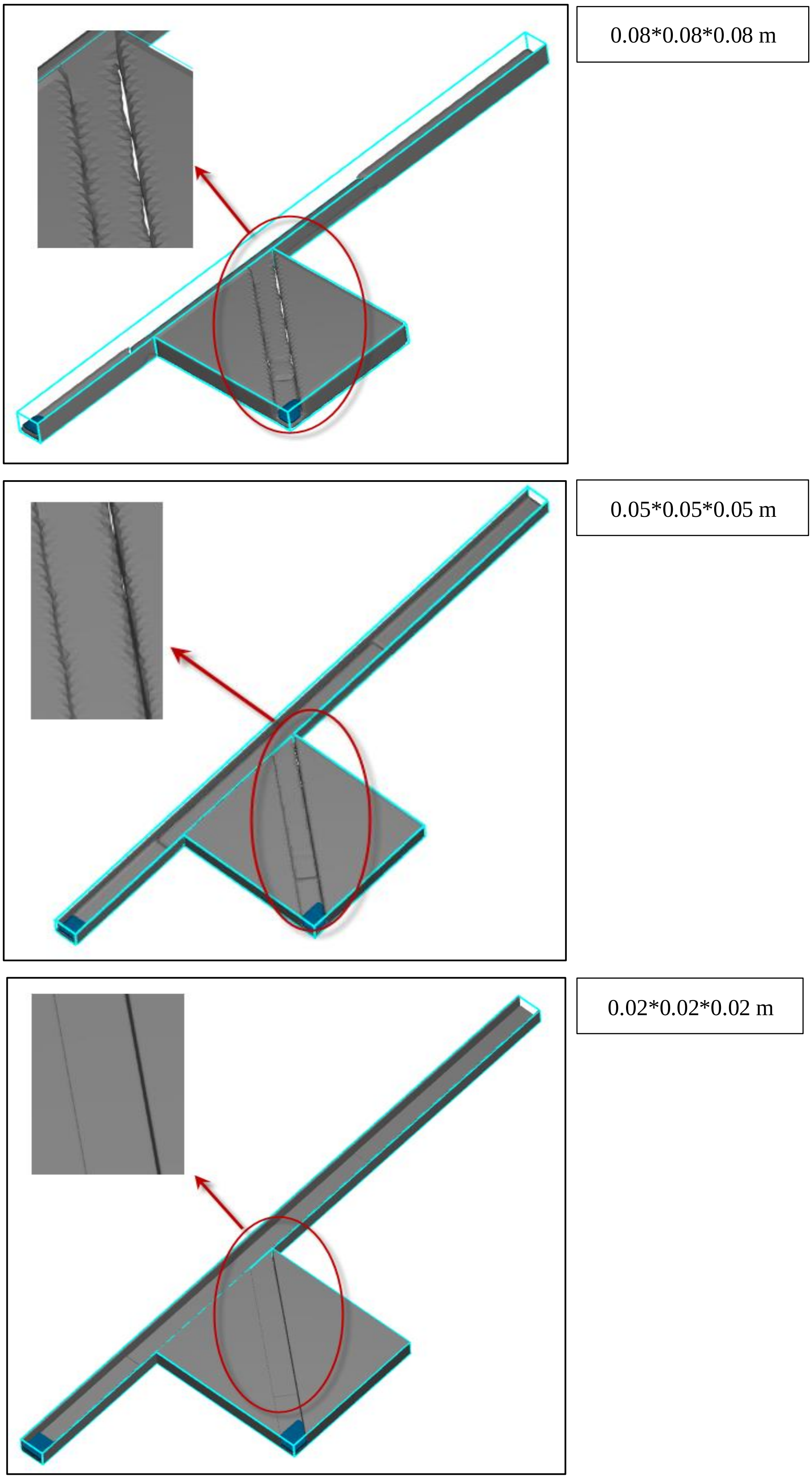

Figure 1. The performance of Favor method with rigid boundaries with different meshing and calibration of confluence channels in Flow3D

3. Numerically Simulating Scour Using Flow3D

Flow3D is an appropriate model for complex fluids problems. This numerical model is widely used, particularly for unsteady 3-dimensional flows with free level and complex geometry. In this model, finite volume method is used in regular rectangular grid generation. Due to using finite volume method in a regular grid, the form of the employed discrete equations is similar to discrete equations in finite difference method. Accordingly, FLOW3D enjoys first and second-order reliability methods which are explained in the following. Also, this software uses five turbulence models such as k-ε and RNG [8]. In Flow3D, two methods have been simultaneously used for geometrical simulation. The first method is volume of fluid (VOF) which is used to show the behaviour of fluid at free level. The second method is fractional area-volume obstacle representation (FAVOR) which is used to simulate solid levels and volumes such as geometrical boundaries.

Equations governing fluid flow are obtained from the law of conservation of mass and the law of conservation of momentum. These equations are in the form of partial differential equations. In general, to obtain flow equations, three steps should be considered: selecting accurate base laws, applying laws by an appropriate model and adopting mathematical equations showing the above physical laws. The main equations to simulate 3-dimensional flow are three differential equations including continuity relations and movement size in x, y and z directions [9].

For numerical analysis of the local scour at the junction of open channels, the results of laboratory studies performed by Borghei and Jabbari Sahebari in the year 2010 are utilized [10]. In these studies the main channel length is 12 m and that of the sub main channel is 2.5 m and the junction angle (θ) in this research is taken equal to 50 degrees. The width of both channels is 40 cm. Based on the laboratory studies in this research the longitudinal slope is ignored. In the following figure different parts of the model are shown. The erodible bed has a depth of 13cm and its length is 2m at upstream of the main and sub main channels, before the junction, and 2.5 downstream of the main channel after the junction. The dimensions of this zone which are depicted in the figure are chosen empirically. At the beginning of this zone in the main and sub main channels, to create the laminar flow use has been made of flow smoother which is a gravel zone of 0.5 m length with a grading ranging from the course to fine grains. To eliminate the effect of non-uniformity and cohesion of the grains on the erosion pattern and sedimentation, the used bed materials have a uniform grading without cohesion. As is shown in the figure below, the center of coordinates is taken at downstream corner of the junction and the positive direction of the x-axis is taken towards upstream flow and the positive direction of the Y-axis is taken towards inside of the main channel.

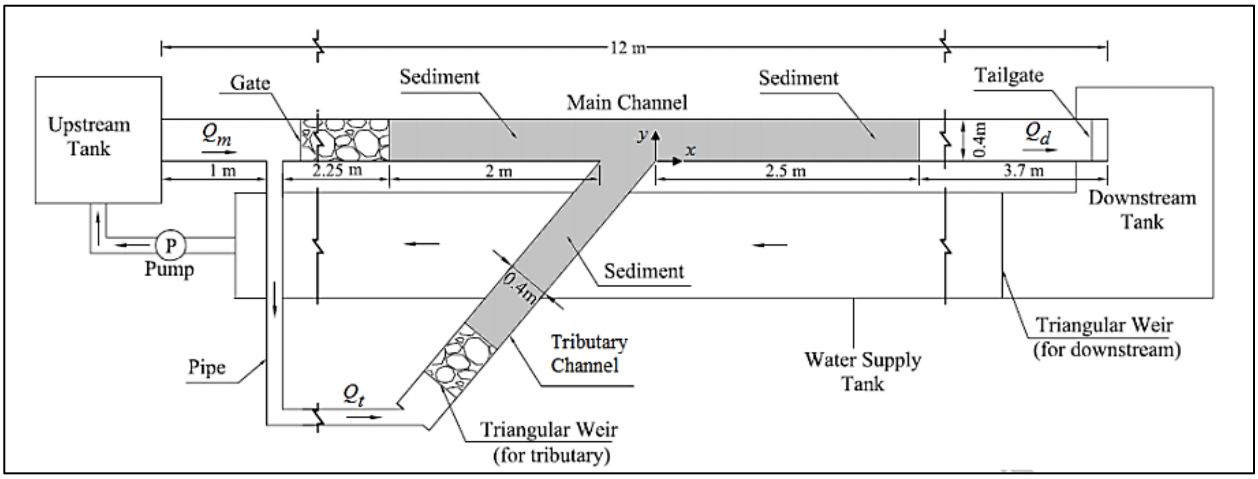

Figure 2. Channels geometries and range of bed materials [11]

After modelling and calibrating the numerical models under boundary and initial conditions of the required geometrical and hydraulic parameters of the study have been changed and the results have been presented.

One of the most important parameters investigated in the present study is changing the discharge ratio of main channel to secondary cannel. Changing the discharge ratio of main channel to secondary cannel between 0.25- 0.8 and keeping other flow conditions constant, the effect of the discharge ratio of main channel to secondary cannel on scour junction pattern has been analysed. In the following figures, the effect of increasing the discharge ratio of main channel to secondary cannel has been shown. Comparing these figures reveals that increasing Q_{bd} from 0.25 to 0.8 in B_{bd}=1 and V/V_c=0.8 causes that the maximum scour depth moves towards the junction about 7cm in the width of the main channel. Further, it causes that scour hole is drawn from the main channel to the inside of the secondary channel and the maximum erosion occurs in the corner of junction downstream. The depth of scour hole in the deepest point has been increased to more than 10 mm by increasing discharge ratio from 0.25 to 0.8 m. It is due to the fact that increasing the discharge ratio of main channel to secondary cannel causes that the secondary channel connects to the main channel with more intense flow. It leads to strengthening vortex flow in junction. Due to the high volume of particles washed from the channel bed, the length of deposition zone has been increased to 35 cm.

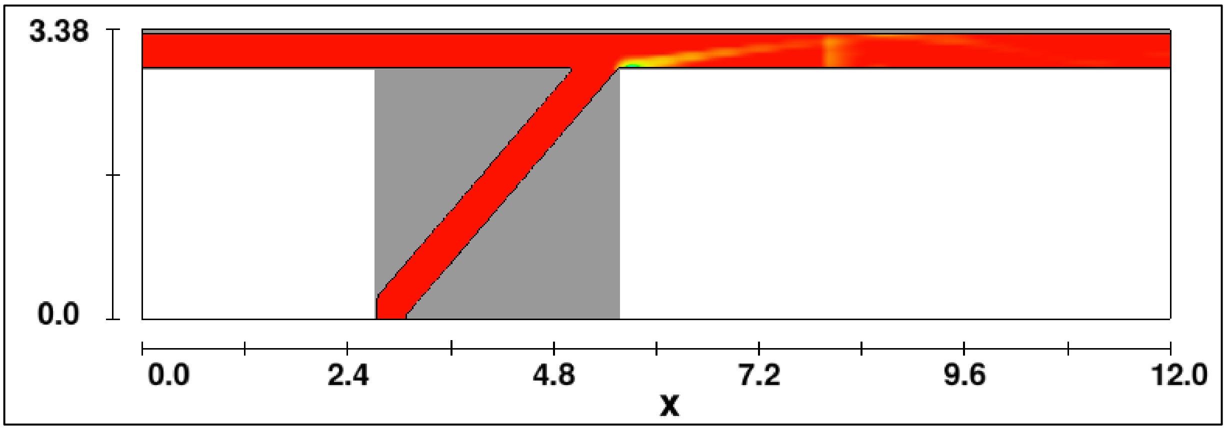

Figure 3. Local scour formation in Q_{bd}=0.25

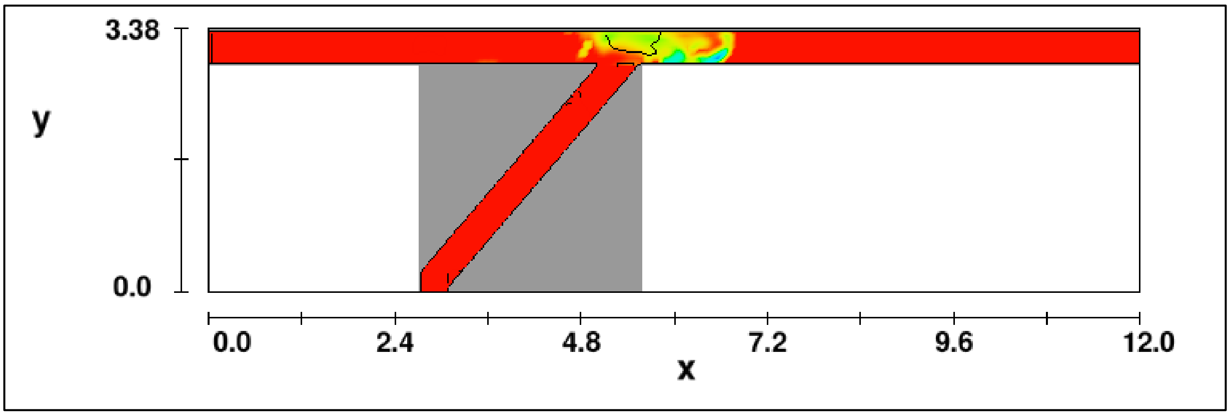

Figure 4. Local scour formation in Q_{bd}=0.8

One of the most important parameters investigated in the present numerical model is the width ratio of the secondary channel to the main channel. To investigate the effete of the width ratio of the secondary channel to the main channel on scour pattern and deposition of three models with various conditions already performed for the width ratio of the main channel to the secondary channel is repeated by halving the secondary channel width. In the following figure, the effect of decreasing the secondary channel width on the increase of scour depth has been shown through depicting the diagram of scour depth charges against the changes of the width ratio of the secondary channel to the main channel. Keeping the discharge ratio of the secondary channel to the main channel constant and decreasing the width ratio of the secondary channel to the main channel leads to the increase of discharge in the width unit of the secondary channel. This causes that the secondary channel flow enters into the main channel with higher velocity. Increasing the velocity of the secondary channel of the bed particles is subjected to higher shear stress and erosion depth is increased.

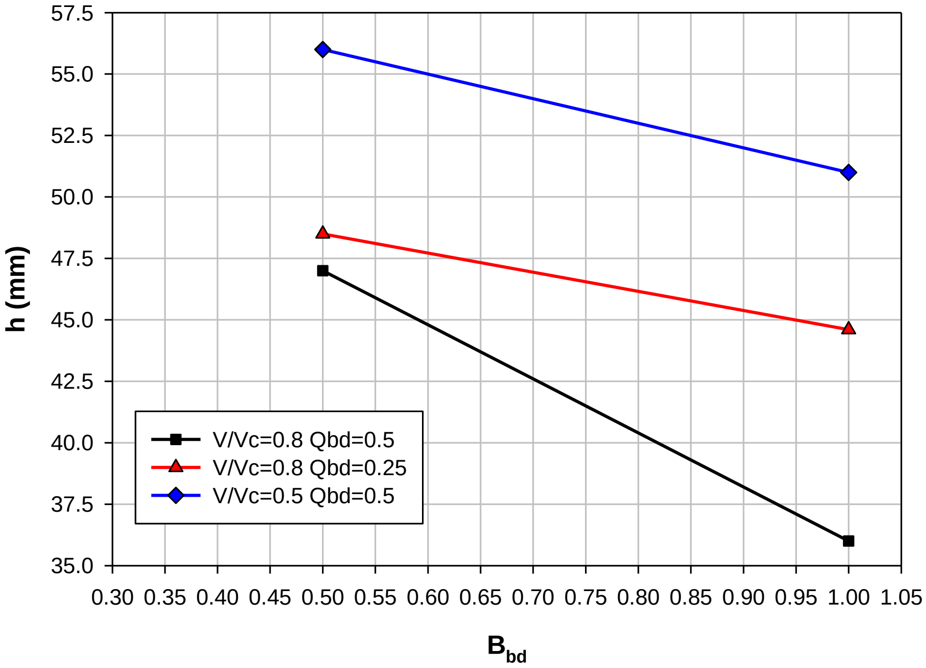

Figure 5. The changes of scour depth based on the changes of width ratio of the secondary channel to the main channel

As shown in the figure, in the equal width ratio, scour depth value in V/V_c=0.8 and Q_{bd}=0.25 is higher than V/V_c=0.5and Q_{bd}=0.5. In other words, keeping the width ratio of the secondary channel to the main channel constant, decreasing the discharge ratio of the secondary channel to the main channel as well as increasing the ratio of flow velocity to downstream movement threshold velocity had led to the increase of scour. It indicates the effect of V/V_c on scour depth. Due to the decrease of the secondary channel width and the subsequent increase of discharge in width unit and velocity of the secondary channel flow, this flow encounters to the wall in front of the secondary channel in the main channel. Also, creating secondary flows in this zone causes the creation of scour next to the main channel wall. Halving the secondary channel width leads that causes that scour hole is drawn towards the main channel wall in front of the secondary channel and its width is increased. Furthermore, increasing flow velocity in the channel bed and intensifying shear stress in the bed leads to increasing the size of the scour hole compared to the similar state and increasing the length of deposition zone in the main channel.

Topographical changes of the bed with running time of the numerical model have been another parameter considered in this research. Flow pattern in channels junction is very complex and follows different variables. Bed particles movement is a function of flow conditions and the presence of rotating cells and secondary flows in various points of this zone makes it difficult to predict erosion and deposition pattern and maximum scour zones. In the present study, it has been attempted to investigate the process of scour depth creation by extracting numerical model of bed topography in certain temporal distances and depicting longitudinal cross-sections. By the time pass, due to the bed erosion, flow depth is more observed in scour zone. Accordingly, shear stress imposed on the particles due to vortex flows and compression leads to the decrease of flow. In addition, decreasing the particles movement intensity and scour value leads to the balance of the bed profile.

In the following figures, the longitudinal cross-sections have been depicted in the local scour maximum depth zone in various results extraction times. Based on the numerical model results, over time, local scour hole is outgrown and the increase of erosion depth, its zone moves towards the external wall of the main channel and keep distance from the channel internal wall. Deposition zone and hill formation also moves towards the channel downstream and its volume is increased. Based on the numerical modelling results, when the intensity of vortexes is high and shear stress in the bed is more, only one large hole is created in junction and other holes are filed in the main channel downstream due to high deposition volume over time. In the first diagram where V/V_c is higher and Bbd is smaller, compared to the second diagram, the hole created in x=497 mm and during 120 minutes is filled by continuing erosion process and disappeared in 960 minutes. However, in the second figure, the second hole is gradually overgrown and moves towards downstream.

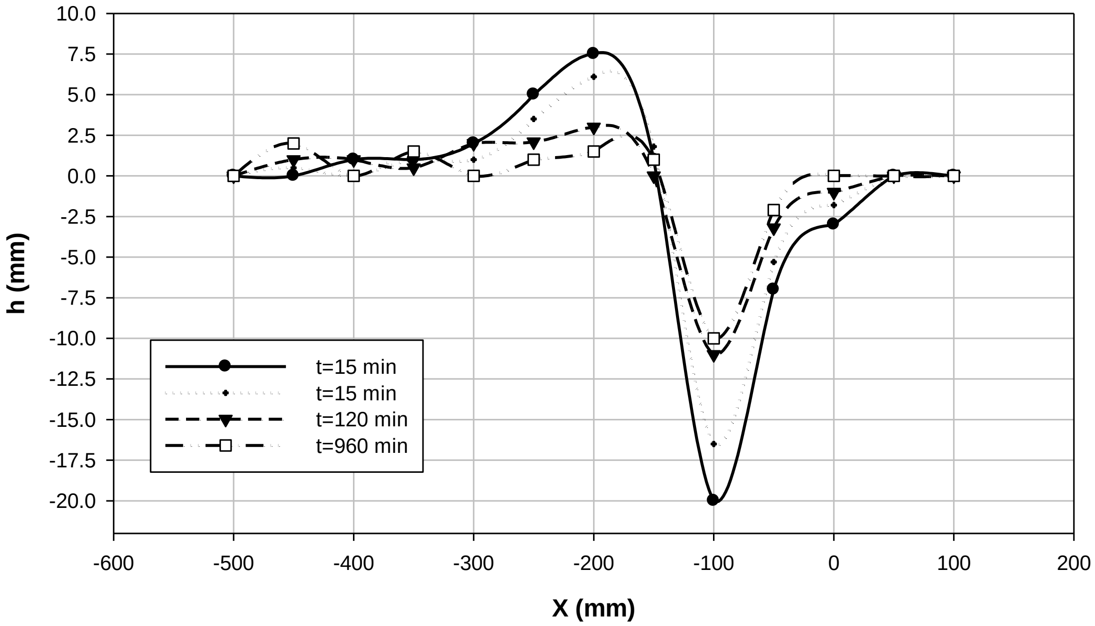

Figure 6. Longitudinal cross-sections in scour maximum depth zone in various times for (V/V_c = 0.8, B_{bd}=0.5, Q_{bd}=0.25)

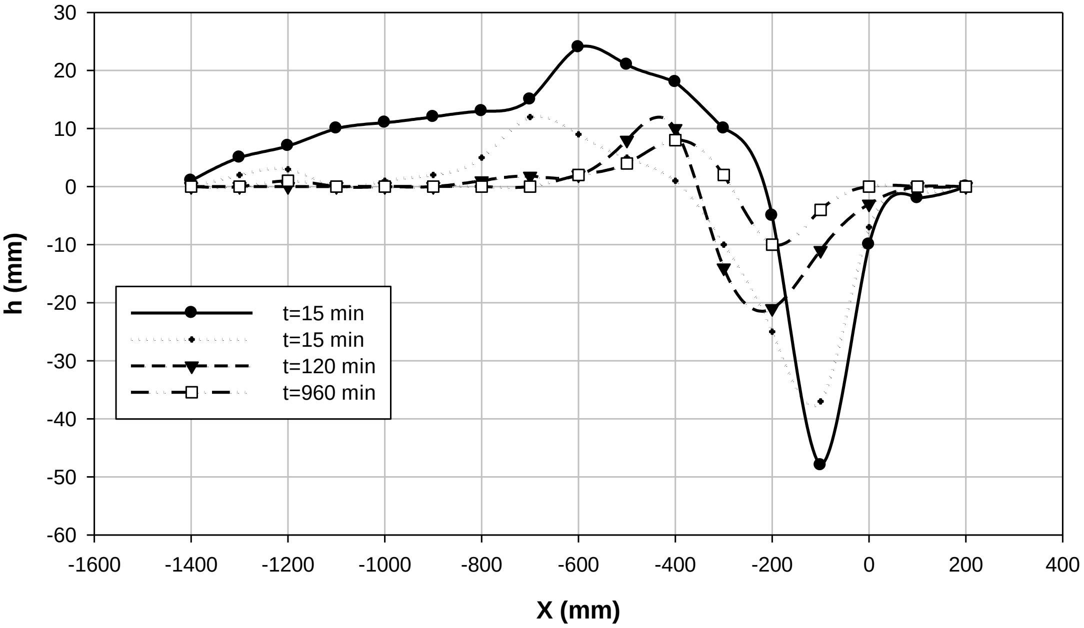

Figure 7. Longitudinal cross-sections in scour maximum depth zone in various times for (V/V_c = 0.5, B_{bd}=1, Q_{bd}=0.25)

In the following figure, flow velocity values have been shown in the average depth with velocity vectors as well as flow turbulence changes way in the main and secondary channels junction.

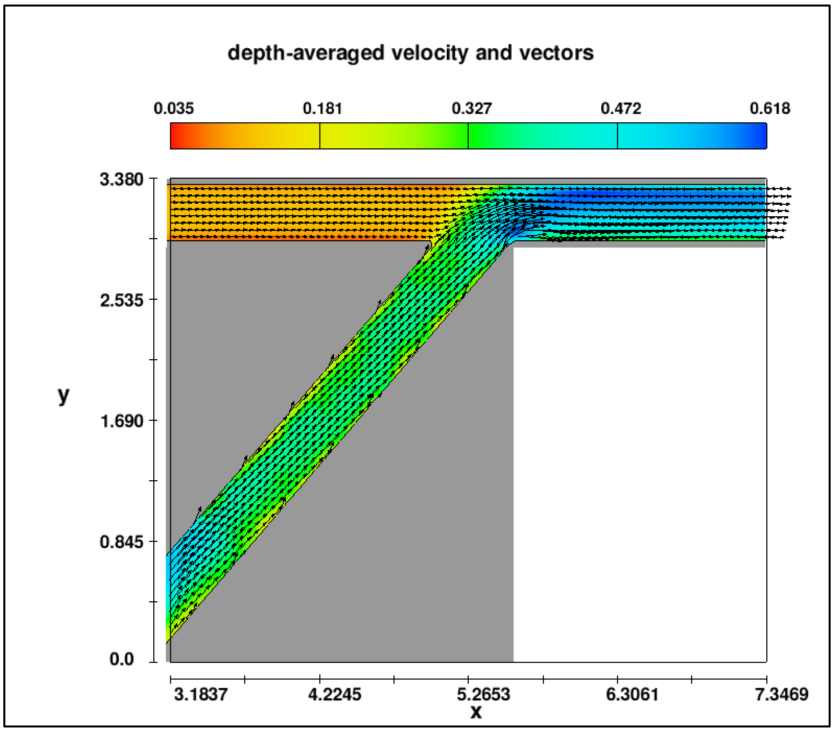

Figure 8. Flow velocity changes values with velocity vectors in the channels junction

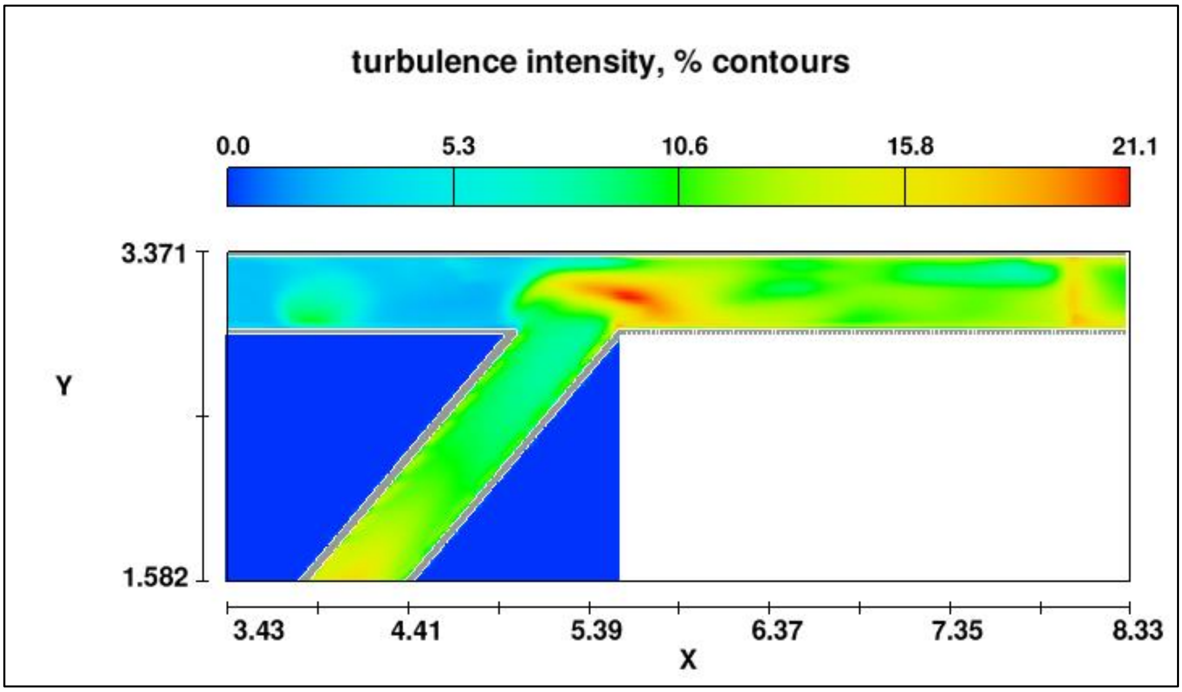

Figure 9. Flow turbulence intensity changes values in the channels junction

As shown in the above shown figures, the maximum velocity values in flow depth is created in the place of encountering two branches flow in the channels. Also, the flow maximum turbulence intensity values occur in the two flows junction and the main channel.

4. Conclusion

In channels junctions where the main and secondary channels flow are associated, the creation of vortex flows and the increase of velocity in flow narrowing place leads to the creation of erosion and deposition in this zone. This phenomenon in the two flows junction causes the decrease of water transfer efficacy and damages to coastal structures. In the present research, the effect of various variables such as the discharge ratio of the main channel to secondary channel and the width ratio of the main channel to secondary channel was investigated on movement threshold and scour hole depth was modelled as numerical calibration in a 50-degree junction. As the research results revealed, the increase of the discharge ratio of the secondary channel to the main channel can cause a movement in flow velocity to critical velocity ratio smaller than 0.5. Decreasing the secondary channel width, scour hole moves towards the wall in front of the main channel junction and in larger discharges, erosion hole is drawn to the secondary channel and the erosion depth approaches to the flow depth. In addition, in models with more intense vortexes and higher shear stress in bed, only one large hole is created in the channel junction and other holes in the main channel downstream are filled over time due to high volume of deposition.

5. References

[1] Yang, C.-T., “Sediment Transport; Theory and Practice”, New York, 1996.

[2] Yang, Chih Ted. "Incipient motion and sediment transport." Journal of the hydraulics division 99, no. 10 (1973): 1679-1704.

[3] Paphitis, D. "Sediment movement under unidirectional flows: an assessment of empirical threshold curves." Coastal Engineering 43, no. 3 (2001): 227-245.

[4] Mason, Peter J., and Kanapathypilly Arumugam. "Free jet scour below dams and flip buckets." Journal of Hydraulic Engineering 111, no. 2 (1985): 220-235.

[5] Ginsberg, Silvia S., and Gerardo ME Perillo. "Deep-scour holes at tidal channel junctions, Bahia Blanca Estuary, Argentina." Marine Geology 160, no. 1 (1999): 171-182.

[6] Liu, Haifei, Jian Guo Zhou, and Richard Burrows. "Multi-Block Lattice Boltzmann Simulations of Subcritical Flow in Open Channel Junctions." Computers & Fluids 38, No. 6 (2009): 1108-1117.

[7] Biswal, Sushant K., Pranab Mohapatra, and K. Muralidhar. "Hydraulics of Combining Flow in a Right-Angled Compound Open Channel Junction." Sadhana 41, No. 1 (2016): 97-110.

[8] Release, FLOW3D. "3.2: User Manual." AEA Industrial Technology, Harwell Laboratory, Oxfordshire, United Kingdom (1992).

[9] Hirt, C. W., and B. Nichols. "Flow-3D User’s Manual." Flow Science Inc (1988).

[10] Borghei, S. Mahmood, and Aidin Jabbari Sahebari. "Local Scour at Open-Channel Junctions." Journal of Hydraulic Research 48.4 (2010): 538-542.

[11] Nazari-Giglou, Abolfazl, Aidin Jabbari-Sahebari, Ahmad Shakibaeinia, and Seyyed Mahmood Borghei. "An Experimental Study of Sediment Transport in Channel Confluences." International Journal of Sediment Research 31, No. 1 (2016): 87-96.