Advances in Environmental Biology, 8(21) October 2014, Pages: 483-488

AENSI Journals

Advances in Environmental Biology

ISSN-1995-0756 EISSN-1998-1066

Journal home page: http://www.aensiweb.com/AEB/

Numerical Modeling of Lateral Intake Flow Pattern using Flow 3D (Case study: Eastern intake of Shahid Al Ishaq diversion dam)

[1]Froude Tghinasab, [2]Ebrahim Nohani, [3]Behrouz Dahanzadeh

[1,3]Department of Irrigation Structures, Shoushtar Branch, Islamic Azad University, Shoushtar, Iran
[2]Department of hydraulic Structures, Dezful Branch, Islamic Azad University, Dezful, Iran

ARTICLE INFO

Article history:
Received 4 September 2014
Received in revised form 24 November 2014
Accepted 1 December 2014
Available online 25 December 2014

Keywords:
Flow pattern, numerical simulation, discharge, intake.

ABSTRACT

The purpose of this study is numerical simulation of flow pattern in Shahid Al Eshaq diversion dam intake (Kanjancham) using the mathematical model Flow 3D in terms of gate different orifices with 40, 70 and 100 cm sizes and a discharge design of 5 cubic meters per second. In this study, the mathematical model was calibrated using the Manning equation with an accuracy of approximately 8.5 percent. The results showed that, in general, by increasing the gate opening height, flow depth as well as the amount of water level turbulence in the intake upstream decreases. Froude number changes in the intake in the three orifice altitudes indicates that in all three orifices in the upstream and downstream, Froude number is less than 1 and no hydraulic jump occur. Generally by increasing the amount of gate orifice, Froude number in downstream decreases.

To Cite This Article: Froude Tghinasab, Ebrahim Nohani, Behrouz Dahanzadeh, Numerical Modeling of Lateral Intake Flow Pattern using Flow 3D (Case study: Eastern intake of Shahid Al Ishaq diversion dam). ***Adv. Environ. Biol., 8(21),*** *483-488, 2014*

INTRODUCTION

Rivers are one of the cheapest and most available water resources for human which impoundment for different applications is important. With regard to the importance of the river as one of the most important sources of water supply, river intake and branching from it is a subject that we always face in Hydraulic and River Engineering. Water diversion from the mainstream for various purposes such as agriculture, urban water supply, power generation, etc., is done with the help of intakes. Flow in an impoundment branching is inherently a three-dimensional flow and has complexities. River impounding by gravity method is the most common impounding method. Despite the numerous intakes design and implementation due to the three-dimensional nature of the intakes, using mathematical models flow is inevitable to study the hydraulic, especially for intakes with high capacity. Knowledge about the structure or pattern of flow in many of the hydraulic phenomena especially intake structures is essential since the flow discharge rate and sediment entering the intake largely depend on the characteristics of the pattern. Prediction of flow pattern terms will help design engineers to design intakes, in similar hydraulic load condition, with highest discharge and lowest diversion sediment or to consider appropriate methods of controlling sediment entering to the intakes. Few studies are performed on three-dimensional flow pattern in lateral intakes and by the advance of mathematical models and development of measurement tools, these studies are required to be carefully done. Characterization of flow and sediment-flow interaction is a complex phenomenon and sometimes with a high cost. Providing a physical model, using expert experiences, applying mathematical models in flow modeling in the case of one-dimensional, two-dimensional and three-dimensional are among methods that can be used in examining flow pattern. Given the three-dimensional nature of the flows, using three-dimensional mathematical models in river engineering can be a key to technical challenges. Christodoulou [1] examined the formation of a hydraulic jump in the combination of the three branches flow. His experiments were done for three branches with 90 and 17 degrees. In his experiments, the flow in the sub-branches is sub-critical and in main branch weak supercritical with $F_r = 1.5 - 2$. The purpose of his researches was to find a criterion in the formation of hydraulic jump of three rectangular prongs. Laeey *et al*. [5] examined different flow parameters at the junction for three sharp edge branches of the same width and angle of obliquity 90 ° in three-dimensional case. He gathered the speed values at different depths in the networked points at the junction by Acoustic Doppler Velocimeter and then he calculated the average turbulent speed and intensity from the time series in different places. Shazy *et al*. [8]

Corresponding Author: Ebrahim Nohani, Department of hydraulic Structures, Dezful Branch, Islamic Azad University, Dezful, Iran.
Tel: +989166148901 E-mail: nohani_e@yahoo.com

presented a dynamic model for subcritical flow analysis in junction of channels. The model is able to calculate and estimate the upstream depth using the momentum equation of motion and the law of conservation of mass and with information such as downstream discharge and depth. Chung-Chieh *et al*. [2], concluded that the energy heads upstream and downstream of the division in the main channel are found to be almost equal. They gave the energy-loss coefficient of a division is expressed in terms of discharge ratio, upstream Froude number and depth ratio. Kiamanesh and Dahanzadeh [4] due to problems in the Dez Diversion Dam studied the reservoir area using two-dimensional model TASE aimed at determining the flow pattern, water surface profile and velocity values. They performed their model for two levels of 116.5 and 177 meters (dam crown level) and examined the model outputs including water surface longitudinal and transverse sections, water depth, Froude number and bed shear stress. They introduced the gradient adjustment between the eastern and western branches as the only solution to the problem. Rashwan [6] developed a one-dimensional theoretical new model for subcritical flows in dividing open channel junction. The new model is derived with the aid of the overall mass conservation together with the momentum principle in the streamwise direction to two control volumes through the junction. Given the inflow discharge, depth and a downstream boundary condition the proposed model solves for each discharges and depth in main channel extension and branch channel. Sabbgagh Yazdi *et al*. [7] performed the numerical study of flow in the hydraulic jump stilling basin using VOF method. They examined the hydraulic jump which is one of the most important ways to the energy dissipation in hydraulic engineering using FLOW 3D. in first step, to verify the software in predicting the flow parameters in hydraulic jump, a sample was used and results obtained from numerical simulation was compared using turbulence and RNG models. Emami [3] examined the hydraulic performance of the wastewater discharge system and energy dissipation of an embankment dam using FLOW 3D.

This study examined the flow pattern through eastern intake of Shahid Al Eshaq diversion dam (Kanjacham) in design discharge of 5 cubic meters per second in three gate opening 40, 70 and 100 cm using the numerical model of Flow 3D.

Methodology:

In the present study Flow 3D is used to simulate the flow conditions using numerical methods. This software solves the flow equations using Finite Volume Method. In this software, the continuity equations (Equation 1) and momentum (Equation 2) simulate fluid flow using the finite volume method.

$$V_F\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x}(\rho uA_x)+R\frac{\partial}{\partial y}(\rho vA_y)+\frac{\partial}{\partial z}(\rho wA_z)+\varepsilon\frac{\rho uA_x}{x}=RDIF+RSOR \quad (1)$$

Where ρ is fluid density, $RDIF$ turbulence diffusion term, $RSOR$ mass spring, V_F is the volume fraction to flow, (u,v,w) velocity components along (x,y,z) and A_x, A_y, A_z are flow level in x, y and z directions, respectively.

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial u}{\partial x}+vA_yR\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial x}\right\}-\xi\frac{A_yv^2}{xV_F}=-\frac{1}{\rho}\frac{\partial P}{\partial x}+G_x+f_x-b_x-\frac{RSOR}{\rho V_F}u \quad (2)$$

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial v}{\partial x}+vA_yR\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial x}\right\}+\xi\frac{A_yuv}{xV_F}=-\frac{1}{\rho}\left(R\frac{\partial p}{\partial y}\right)+G_y+f_y-b_y-\frac{RSOR}{\rho V_F}v \quad (3)$$

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial w}{\partial x}+vA_yR\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial x}\right\}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+G_z+f_z-b_z-\frac{RSOR}{\rho V_F}w \quad (4)$$

Where fx, fy and fz increase the velocity, bx, by and bz flow loss in medium porous and μ is the dynamic viscosity. Model geometry in Cartesian coordinates was drawn using the gate specifications in the AutoCAD environment and three-dimensional modeling was carried out. In this study, the flow is considered as permanent, turbulence and incompressible fluid. Turbulence model used in this study is $k-\varepsilon(RNG)$. The reason for applying this model in addition to the recommendations of previous researches is the model well consistency with the rotating and curved flows with large strains. In Flow 3D model, meshing is done by using perpendicular cubic elements that in places with high gradient, a finer mesh is used. Boundary conditions of model include input level, output level, wall and flow free surface. Flow discharge is used in input boundary condition; the outlet boundary condition is the flow outlet model. Using the results of the numerical model, velocity profile, the Froude number, water depth and other hydraulic flow parameters can be extracted. Eastern intake of martyr Al Eshaq diversion dam has two gates with a width of 1.7 m and a height of 1.5 m. In this study, 33 meters if intake length is modeled.

Advances in Environmental Biology, 8(21) October 2014, Pages: 483-488

RESULTS AND DISCUSSIONS

In this research, to calibrate the numerical model, the flow rate at a distance of 20 meters from intake downstream was calculated using the Manning equation. The amount to the speed was obtained 2.3 meters per second. To calibrate the numerical model with changing the calibrated model bottom roughness and velocity at the point corresponding to the intake, in the numerical model with roughness equivalent to Manning roughness 0.028 is equal to 2.5 meters per second that comparing with the calculated rate of Manning equation, it has about 8.5 percent error and it is acceptable. In this section, in separate charts, changes in water surface profiles, the Froude number and the mean flow velocity are presented for different opening heights. Following, to compare the effect of various openings on various parameters, these parameters are plotted on a graph and compared. Figure 1 (a) to (c) show the changes of the water surface profile in the opening 40 and 100 cm, respectively. According to the figure in different openings, the flow after passing through the gate continues without hydraulic jump, and in the downstream, flow gate exits from under the gate as a wave. At the opening, 100 cm of gate has a trivial impact on the flow and almost the flow gets out of the gate freely. Also in Figures 2 to 4 output of numerical modeling in different openings are shown.

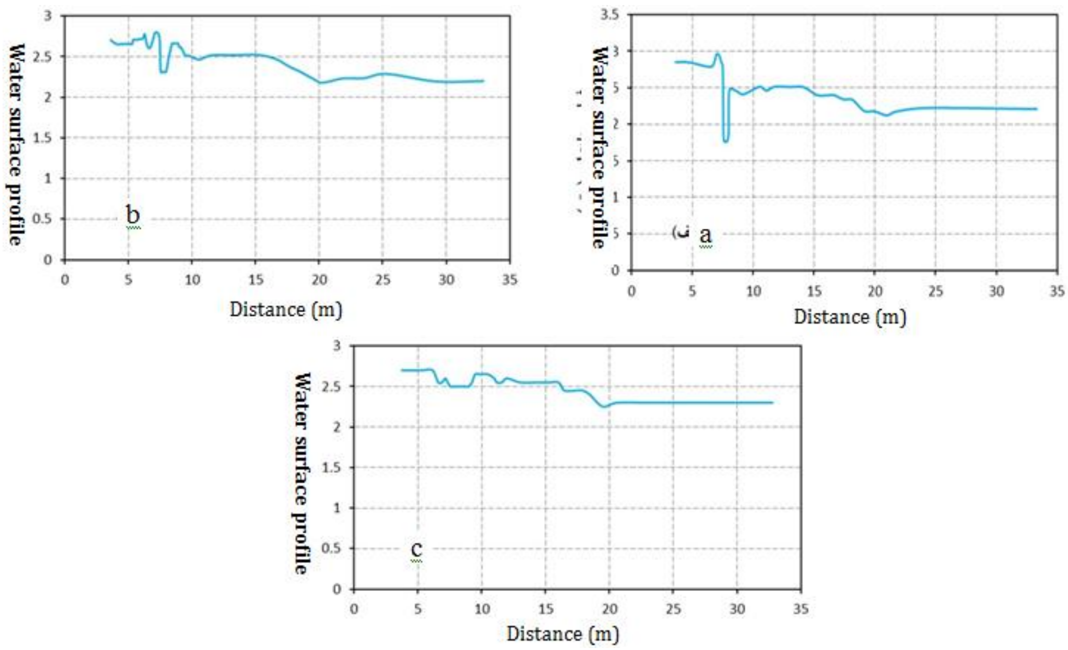


Fig. 1: Changes in water level profile in the intake opening a) 40 cm b) 70 cm c) 100 cm

In Figure 6 (a) to (c) Froude number changes throughout the gate in 3 different openings are displayed. According to the figure, the value of Froude number in the intake in all openings is less than 1 and increases from the approximate amount of 0.15 in up to the approximate amount of 0.38 in downstream. This is confirming that in all three openings hydraulic jump is not formed. Also in Figure 7 (a) and (c) the average rate changes over the intake are shown in three different opening, according to figure, the speed increases from the approximate amount of 0.65 meters per second in upstream to the approximate amount of 1.3 meters per second in downstream.

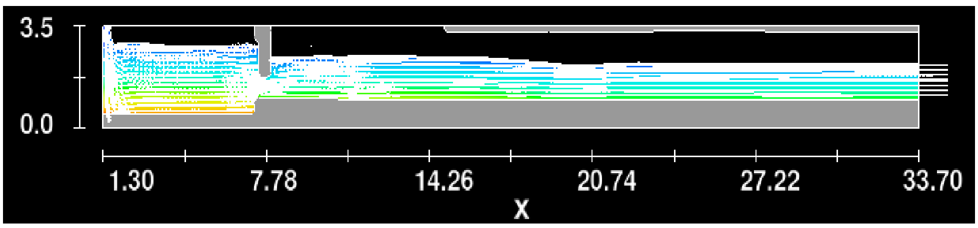


Fig. 2: Displaying the water depth in software modeling in opening 40 cm of gate .

Advances in Environmental Biology, 8(21) October 2014, Pages: 483-488

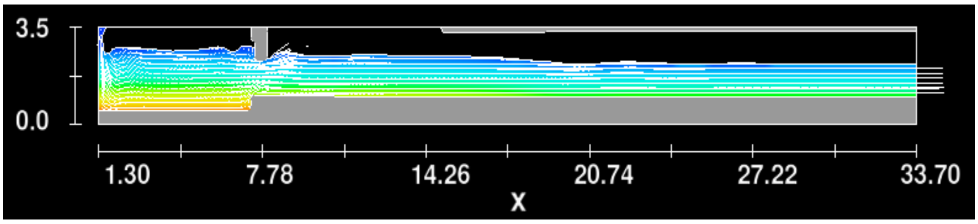


Fig. 3: Displaying the water depth in software modeling in opening of 70 cm of gate .

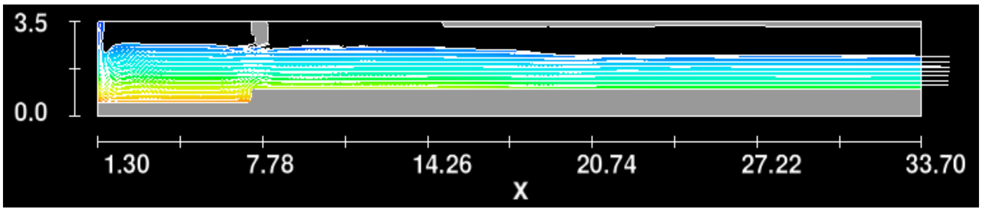


Fig. 4: Displaying the water depth in software modeling in opening 100-cm of gate

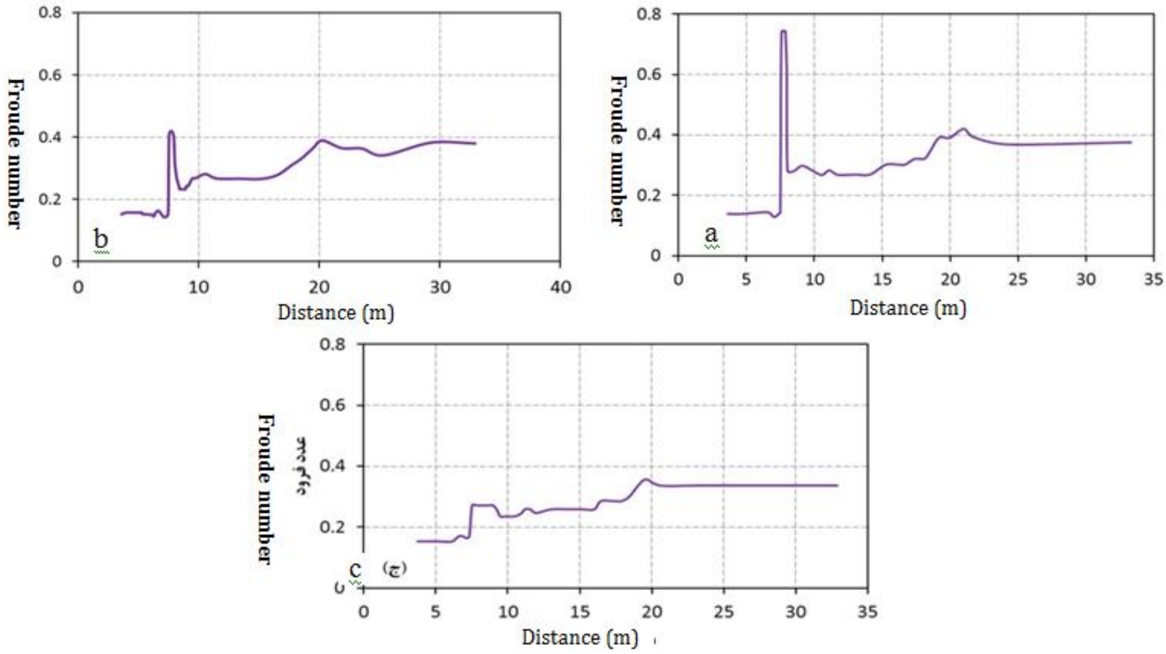
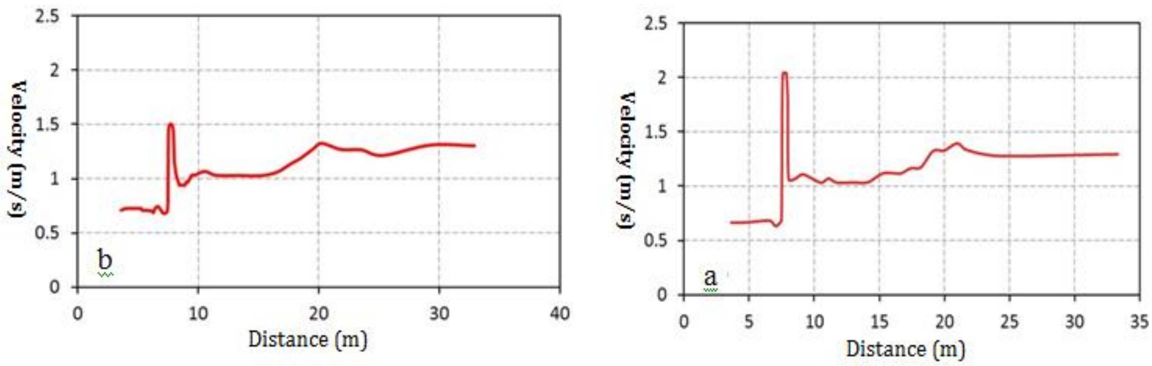


Fig. 5: Changes in the Froude number through intake in the opening a) 40 cm b) 70 cm c) 100 cm.



Advances in Environmental Biology, 8(21) October 2014, Pages: 483-488

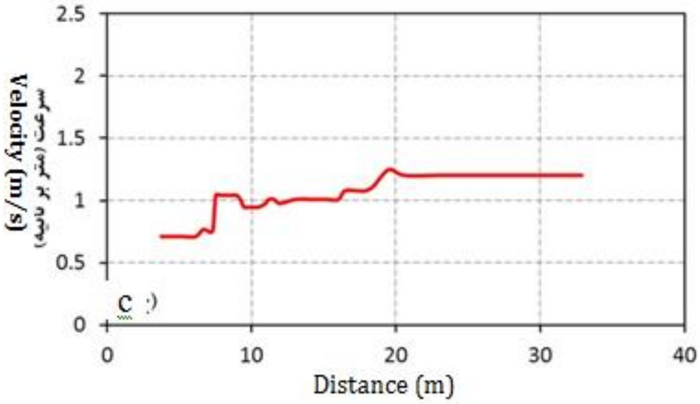


Fig. 6: Changes in average speed through the intake at the opening of a) 40 cm b) 70 cm c) 100 cm.

In this part to compare the effect of different openings of the spillway on the parameters of the water surface profile, the Froude number and the average velocity of flow in the upstream and downstream of intake gate, the results are compared in Figures 7to 9. According to Fig. 7 where the water surface profiles are drawn in different openings, generally with increasing gate opening height, flow depth and the amount of water surface turbulence in the upstream of the intake decreases with increasing opening. Also in intake downstream by increasing the opening height, flow depth increases. In Figure 8, Froude number changes through the intake are presented in three different opening heights. According to figure, in general by increasing the amount of gate opening Froude number decreases. Also in figure 9, speed changes through the intake are shown. According to the figure, speed changes have no significant difference in upstream and downstream of different openings 40, 70 and 100 cm but below the gate where the opening increased to 40 cm.

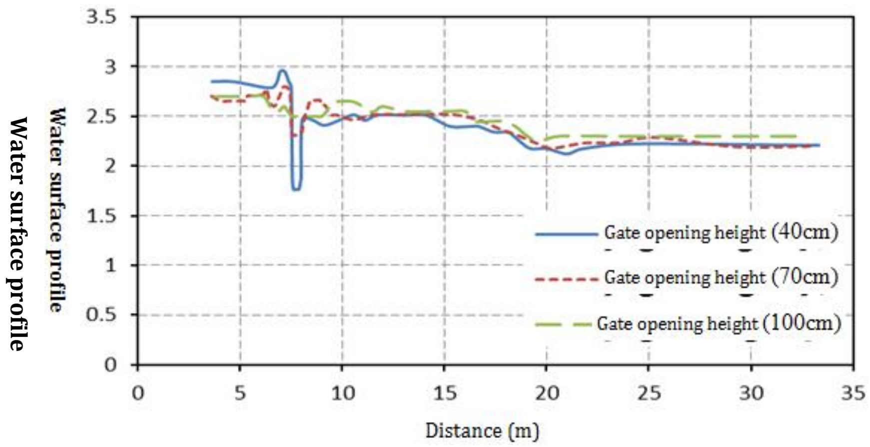


Fig. 7: Changes in water depth with interval in different openings.

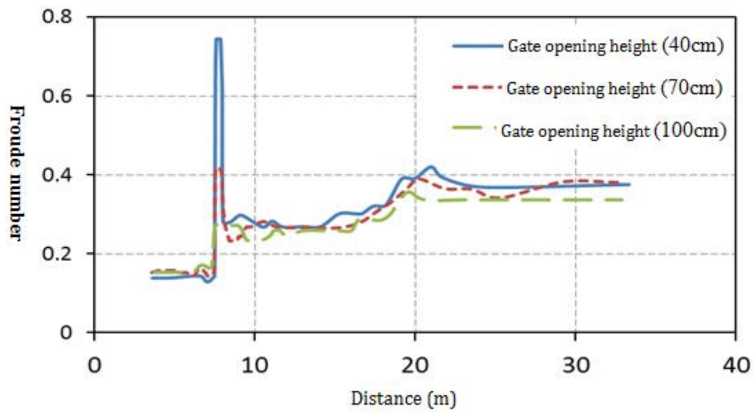


Fig. 8: Changes in the Froude number with interval in different openings.

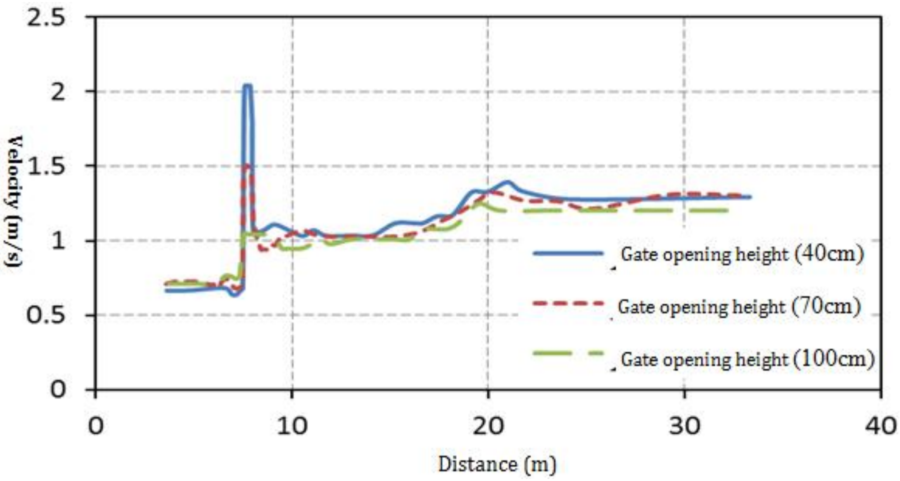


Fig. 9: Mean velocity changes with distance in different openings.

Conclusions:

The purpose of this study is the numerical simulation of flow pattern in diversion dam intake of the martyr Al Eshaq (Kajancham) using the mathematical model Flow 3D in terms of different openings with 40, 70 and 100 cm and a design discharge of 5 cubic meters per second. Turbulence model used in this study is $k-\varepsilon(RNG)$. In this study, the mathematical model was calibrated using Manning equation with accuracy of 8.5 percent. The results show that, by increasing the gate opening height, flow depth and the amount of water surface turbulence in the intake upstream decreases. Also in intake downstream, flow depth increases by increasing the opening height. In all gate openings, the flow exits the gate as a wave and no hydraulic jump is formed. Froude number changes in along the intake in the three heights of opening indicates that the Froude number in all three openings in the upstream and downstream is less than 1 and as a result confirms that no hydraulic jump occurs. Also the results suggest that the velocity changes in the upstream and downstream of the gate are about 0.65 and 1.3 meters per second, respectively, and for different openings height and it has no significant changes for different opening heights. Finally, the output parameters of the software confirm the design discharge accuracy.

REFERENCES

[1] Christodoulou, G.C., 1990. Incipient Hydraulic Jump at channel Junctions. Journal of Hydraulic Engineering, ASCE, 119(1): 409-420.
[2] Chung-Chieh, H., W. Feng-Shuai, L. Wen-Jung, 2002. Flow at a 90° Equal-Width Open-Channel Junction. Journal of Hydraulic Engineering, 124(2): 186-191.
[3] Emami, S., 2008. Evaluation of free-hydraulic overflows using a 3D mathematical model FLOW3D, 2th National Conference of dams and hydroelectric plants. Tehran, Iran. (In Persian).
[4] Kiamanesh, H., B. Dahanzadeh, 2002. Evaluation of the Dez diversion dam. 6th International River Engineering conference, Chamran University., Ahvaz, Iran.
[5] Laeey, J., E. Weber, D. Nicolamawer, 2001. Experiments on Flow at a 90 degree open channel junction . Journal of Hydraulic Engineering, ASCE, 127(5): 340-350.
[6] Rashwan. I., 2004. Dynamic Model for Subcritical Dividing Flows in Open Channel Junction. 18th International Egyptian Water Technology Conference, ALEXANDRIA, EGYPT, MARCH 26-28: 511–520.
[7] Sabbgagh Yazdi, S., H. Shamlou, F. Rogstami, 2008. Numerical study of flow in hydraulic jump stilling basin using VOF method. 2th National Conference of dams and hydroelectric plants. Tehran, Iran. (In Persian).
[8] Shazy, S., P. Stetfler, F. Hicks, 2002. Dynamic model of subcritical combining flows in the junction. Journal of Hydraulic Engineering, ASCE, 128(9): 821– 828.