Shahid Chamran University of Ahvaz

Irrigation Sciences and Engineering (JISE)
Vol. 44, No. 2, Summer 2021, p. 1-9

Original Paper

Journal homepage: https://jise.scu.ac.ir

Modeling Longitudinal and Transverse Velocity Profiles Upstream of an Orifice Using the FLOW-3D Model

S. Naderi[1], M. Daryaee[2*], S. M. Kashefipour[3] and M. R. Zayeri[4]

1- MS Student, Department of Water Structures, Faculty of Water and Environmental Engineering, Shahid Chamran University of Ahvaz, Ahvaz, Iran.
2- Corresponding Author, Assistant Professor, Department of Water Structures, Faculty of Water and Environmental Engineering, Shahid Chamran University of Ahvaz, Ahvaz, Iran. (M.Daryaee@scu.ac.ir).*
3- Professor, Department of Water Structures, Faculty of Water and Environmental Engineering, Shahid Chamran University of Ahvaz, Ahvaz, Iran.
4- Assistant Professor, Department of Water Structures, Faculty of Water and Environmental Engineering, Shahid Chamran University of Ahvaz, Ahvaz, Iran.

ARTICLE INFO

Artlcle history:
Received: 3 January 2021
Revised: 20 January 2021
Accepted: 25 January 2021

Keywords:
Flushing, Orifice, Turbulence Model, Shear Stress.

TO CITE THIS ARTICLE:

Naderi, S., Daryaee, M., Kashefipour, S., Zayeri, M. (2021). 'Modeling Longitudinal and Transverse Velocity Profiles Upstream of an Orifice Using the FLOW-3D Model', Irrigation Sciences and Engineering, 44(2), pp. 1-9. doi: 10.22055/jise.2021.36268.1943.

Abstract

The current study investigated the longitudinal flow velocity profile upstream of an orifice for different water depths using the FLOW-3D model. Experimental design was used along with LES, Laminar, and $k-\varepsilon$ turbulence models to calibrate the model. The obtained results indicated that turbulence models had almost high and equal accuracy for predicting longitudinal velocity profiles. For various depths upstream of the orifice, the general form of the longitudinal velocity profile followed an exponential function with high accuracy. Moreover, at larger-distance upstream of the orifice, the transverse velocity profile became uniform. Eventually, it was found that with the rise in the depth upstream of the orifice by eight times, the shear stress created on the bed increased by 148%.

Introduction

Due to the crisis of water scarcity, water resources management has become vital in Iran. Dam reservoirs are among the most important water resources. Construction of a dam on a river reduces the flow velocity in the reservoir, consequently resulting in the deposit of sediments in it. The depositing of sediments in the dam reservoir reduces the amount of useful volume and disturbs the dam's performance in terms of water storage. Therefore, proposals have always been suggested to manage and discharge sediments in the reservoir during the service period. In this regard, pressurized flushing is a common solution for eliminating sediments. In this method, by opening the bottom gates, the upstream water pressure discharges the sediments through the orifice. The volume of the displaced sediments is a function of a number of factors, such as gate diameter, sediments type and size, water height upstream the gate, and outflow discharge. Numerous studies have been conducted on the effect of these factors on the volume of sediments displaced from an orifice. Shahmirzadi et al.

(2010) experimentally evaluated the effect of the diameter of bottom dischargers on the dimensions of the flushing cone. Powell and Khan (2015) conducted tests to investigate the flow pattern upstream of a dam orifice under the fixed bed and equilibrium scour (mobile bed) conditions. Their results demonstrated that the velocity horizontal component was almost equal for both fixed and equilibrium scour conditions. The same conditions were also constant for the vertical component of the velocity.

Performing a number of tests and proposing some numerical relationships, Bryant et al. (2008) studied the flow pattern upstream of an orifice in different sizes. They used the polar coordinate system to evaluate the transverse and depth velocity profiles and found that this method provided a better prediction compared to the velocity profile in the vicinity of the orifice. Wei et al. (2014) simulated the pressurized flushing cone using the FLOW-3D model. They developed a 3D model capable of simulating the scour hole, bed load, and suspended sediment load. Dargahi (2010) simulated the discharge properties of a bottom outlet using the FLOW-3D model and compared the results with the experimental results of the scaled model of Aswan Dam, Egypt. The obtained results showed that the RNG turbulence model had higher accuracy compared to the $k-\varepsilon$ turbulence model. Chapokpour et al. (2012) studied vortex flow behavior using the FLOW-3D model. Evaluating the velocity components, they found several clockwise and counterclockwise vortexes at some points. Furthermore, the comparison of the obtained results with the previous ones indicated that the FLOW-3D numerical model is capable of modeling vortex flows. Chanson et al. (2002) investigated the behavior of an unsteady flow upstream of an orifice experimentally, while water elevation was lowered. Performing experimental studies in the same line, Powell and Khan (2012) showed that by increasing the relative distance from an orifice (x/D, x = distance from upstream of the orifice), the relative velocity (V_C/U_O, V_C = velocity along the orifice center at each distance from the orifice and U_O = mean velocity of the water outflow from the orifice) decreased, so that for $x/D > 2$, the relative velocity tended towards zero. Shammaa et al. (2009) defined the flow pattern behind the gate and orifices using the flow potential, and analytically solved it according to various scenarios.

Since the movement of sediments and their discharge volume during the flushing are a function of the flow pattern upstream of the orifice, it is necessary to predict flow patterns in different operation conditions. Because designing an experimental model and surveying velocity information upstream of an orifice are very costly, using a numerical model for this purpose can be very helpful. Accordingly, the current paper aims at a feasibility study on using the FLOW-3D model to predict the flow pattern and shear stress formed on the bed upstream of the orifice under different water depths.

Materials and methods

FLOW-3D software

The FLOW-3D is a commercial CFD software for solving Navier-Stokes equations within the medium Reynolds number range using the finite volume method. It allows both 2D and 3D analyses of the flow field in the finite volume mode. The software uses orthogonal 3D elements. Given the possibility of using it for various fluids, the FLOW-3D provides acceptable results for hydraulic uses in particular. Due to the increased number of users, and recent debugging, the software is now increasingly being used in different fields of fluid mechanics, especially the hydraulics of open channels and hydraulic structures.

Governing equations

Basic continuity and momentum equations for an incompressible fluid are as follows:

$$\frac{\partial}{\partial x}(uA_x) + \frac{\partial}{\partial y}(vA_y) + \frac{\partial}{\partial z}(wA_z) = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + \frac{1}{V_F}\left(uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + F_x \tag{2}$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial P}{\partial y}+G_y+F_y \qquad (3)$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial P}{\partial z}+G_z+F_z \qquad (4)$$

In these equations, V_F is the fraction of open volume with the flow F; $A_x. A_y. A_z$ are respectively the fractions of open flow surfaces in the directions x, y, and z; P denotes the pressure; $G_x.G_y.G_z$ are the terms of gravity acceleration in three coordinates directions, and the terms $F_x.F_y.F_z$ indicate the viscous pressure in three coordinates directions (x, y, and z).

Turbulence models

Researchers have developed various turbulence models for the simulation of turbulent flows. The major and common goal of all these models is to pave the way for determining the turbulence effects on different parameters of a flow. Depending on the method and the number of differential equations used, the models are divided into zero-equation, single-equation, and two-equation models, along with equations containing stress and simulation models of large vortexes. The $k-\varepsilon$ equations are among the commonest and the most useful two-equation turbulence models. This turbulence model uses two transport equations in order to solve the turbulent kinetic energy, k, and the vortex dissipation rate, ε. The Laminar, $k-\varepsilon$, and LES turbulence models were employed in the present study.

The Experimental Model

The model was calibrated according to the findings of Bryant et al. (2008). The experimented flume in this research was 122 cm wide, 91 cm high, and 914 cm long (Fig. 1). The discharge of the intended flow varied by the pump velocity alterations. Two orifices with diameters (D) of 7.62 cm and 15.24 cm were used in this study. In order to eliminate the effects of lateral walls, the centers of the whole orifices were located in the middle of the channel width. Moreover, the water depth upstream of the orifice center (H) was considered fixed, being equal to 5.92 cm. The flow pattern was measured using an ADV (Acoustic Doppler Velocity Meter) with an accuracy of ±1%.

The meshing of the flow zone, boundary conditions, and calibration of the numerical model

The AutoCAD software was used to create the flume geometry based on the given dimensions. It should be noted that in order to shorten the simulation period, the dimensions of the solution meshing were limited to a length of 150 cm, a width of 122 cm, and a height of 60 cm. The fluid was considered incompressible in the simulation. The meshes had a uniform dimension of 0.01 m, while finer meshes were used adjacent to the orifice to increase the calculation accuracy. A period of 30s was allocated for the modeling, which was found suitable by obtaining the numerical model results and comparing them with those of the experimental model; the numerical model became steady in this period. Fig. (2). illustrates the boundary conditions defined by the software. The water depth in the flume inlet was assumed fixed, being equal to 40.64 cm. The outflow boundary condition was defined for the orifice, the walls and floor had rigid boundaries (wall), and symmetry conditions were defined for the top boundary.

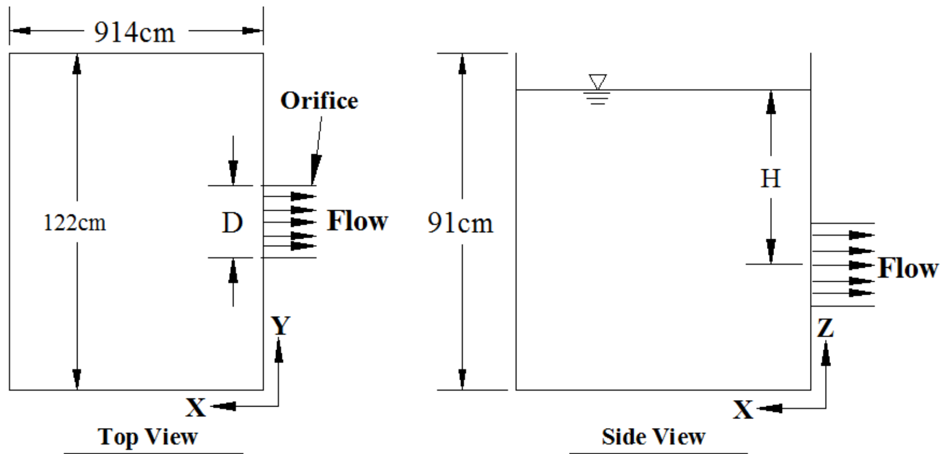

Fig. 1- The dimensions of the experimental flume used by Bryant et al. (2008)

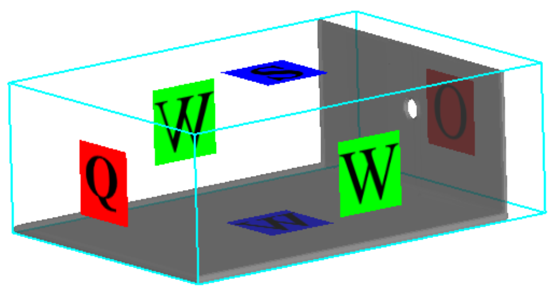

Fig. 2- The boundary conditions defined in the FLOW-3D

It is worth mentioning that, in order to determine the shear stress, another geometry was plotted as is shown by Fig. 2, in which the floor was positioned below the orifice.

The statistical parameters of mean absolute error (MAE), root-mean-square error (RMSE), and coefficient of determination (R^2) were employed to evaluate the accuracy and to choose the optimum turbulence model. The parameters are defined as the following:

$$MAE = \frac{\sum_{i=1}^{N}|X_{mi} - X_{ci}|}{N} \tag{5}$$

$$RMSE = \sqrt{\frac{\sum_{i=1}^{N}(X_{mi} - X_{ci})^2}{N}} \tag{6}$$

$$R^2 = 1 - \frac{\sum_{i=1}^{N}(X_{mi} - X_{ci})^2}{\sum_{i=1}^{N}(X_{mi} - \bar{X})^2} \tag{7}$$

In these equations, X_{mi} is the measured data, X_{ci} is the calculated data, and $\bar{X}$ is the average of the measured data. After calibrating the model and determining the optimum turbulence model, the water level upstream of the orifice was defined in terms of four submergence (H/D) ratios, being equal to 0.5, 0.77, 2, and 4 m, to evaluate the effect of water level upstream of the orifice on the general form of the longitudinal velocity profile and the shear stress formed on the bed. Under the same conditions, the transverse velocity profile upstream of the orifice was also investigated at three relative distances of $x/D = 1.2.3$.

Results and discussion

Model calibration

As was mentioned earlier, the results of the study by Bryant et al. (2008) were used to calibrate the model. The submergence ratio used in this study was 0.77. Fig. (3) makes a comparison between the results obtained from the executed model using the experimental data and those of the presumed turbulence models. Furthermore, Table (1) provides the results obtained from the statistical analysis to determine the accuracy of each of the models. Accordingly, all turbulence models had reasonable accuracy in predicting the results, and the LES turbulence model had a relatively higher accuracy. Therefore, it was used to execute other scenarios.

According to Fig. (3), like what was already reported by Powell and Khan (2012), the relative velocity tended towards zero for $x/D > 2$.

Modeling of the longitudinal velocity profile

After calibrating the model, in order to evaluate the effect of water depth upstream of the orifice on the form of the longitudinal velocity profile, three other submergence ratios ($\frac{H}{D} = 0.5, 2, 4$) were defined in the model (in addition to the one used by Bryant et al. (2008) ($\frac{H}{D} = 0.77$)). Fig. (4) compares the longitudinal velocity profiles at various submergence ratios.

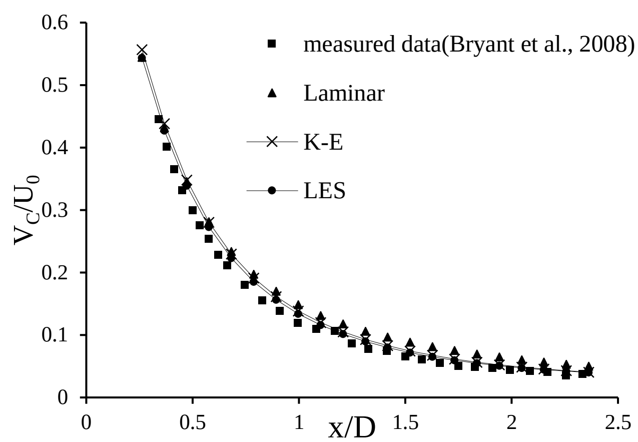

Fig. 3- A comparison of the results of the longitudinal dimensionless water velocity profile along the orifice axis using the three turbulence models

Table 1- A comparison of the accuracy of the turbulence models

	Laminar	$k-\varepsilon$	LES
R^2	0.97	0.98	0.99
MAE	6.50	7.30	7.10
RMSE	15.90	15.00	15.30

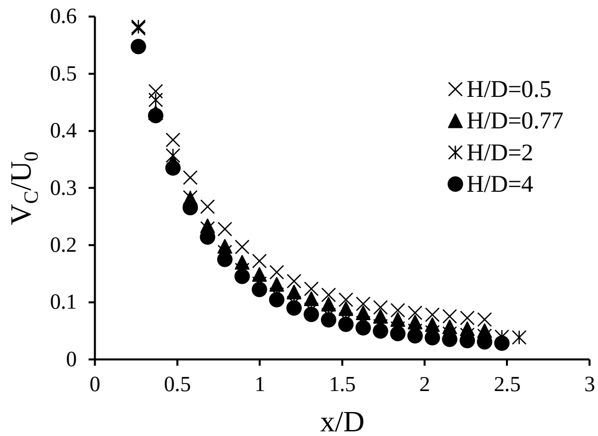

Fig. 4- A comparison of the results of the longitudinal dimensionless water velocity profile along the orifice axis for the intended submergence ratios

Table 2- Constant coefficients of Eq. 8 for the intended submergence ratios

	$H/D = 0.5$	$H/D = 0.77$	$H/D = 2$	$H/D = 4$
a	0.25	0.35	0.15	0.29
b	1.32	1.59	1.51	1.85
c	0.92	1.15	1.15	0.51
R^2	0.99	0.99	0.99	0.99

As it is observable, the general form of the longitudinal velocity profile remained almost constant after the changes in the submergence ratio upstream of the orifice. Moreover, at all submergence ratios, the changes in the longitudinal velocity started to reduce at a relative distance of $x/D = 1$; and at distances more than the relative distance of $x/D = 1.5$, the velocity remained almost constant at all submergence ratios.

According to Shammaa et al. (2009) and Bryant et al. (2008), the longitudinal velocity along the orifice center is obtained based on the potential flow theory, as Eq. (8):

$$\frac{V_C}{U_O} = 1 - (1 + \frac{a}{(x/D)^b})^{-c} \qquad (8)$$

In this equation: a = 0.25, b = 2, and c = 0.5 (the other parameters were previously introduced. The coefficients for Eq. 8 in the current study were determined using the nonlinear regression for the submergence ratios of the orifice, as presented in Table (2). Furthermore, the coefficient of determination (R^2) was 0.99 for all cases. It can be said that for different submergence ratios of the orifice, the longitudinal velocity profile followed the exponential equation of Eq. 8 with sufficient accuracy.

Modeling of the transverse velocity profile

The results obtained from the transverse velocity modeling for the intended submergence ratios at the three relative distances $(x/D = 1.2.3)$ upstream of the orifice are shown in Fig. (5). The figure is plotted in a dimensionless form using the mean velocity of the water outflow from the orifice (U_O) and the orifice diameter (D). As can be observed, at larger distances upstream of the orifice, the velocity variations decreased, tending towards zero. Besides, for each relative distance, the maximum velocity was along the orifice, while the velocity decreased, and the transverse velocity profile became uniform with the increased distance from the orifice center along the y-axis.

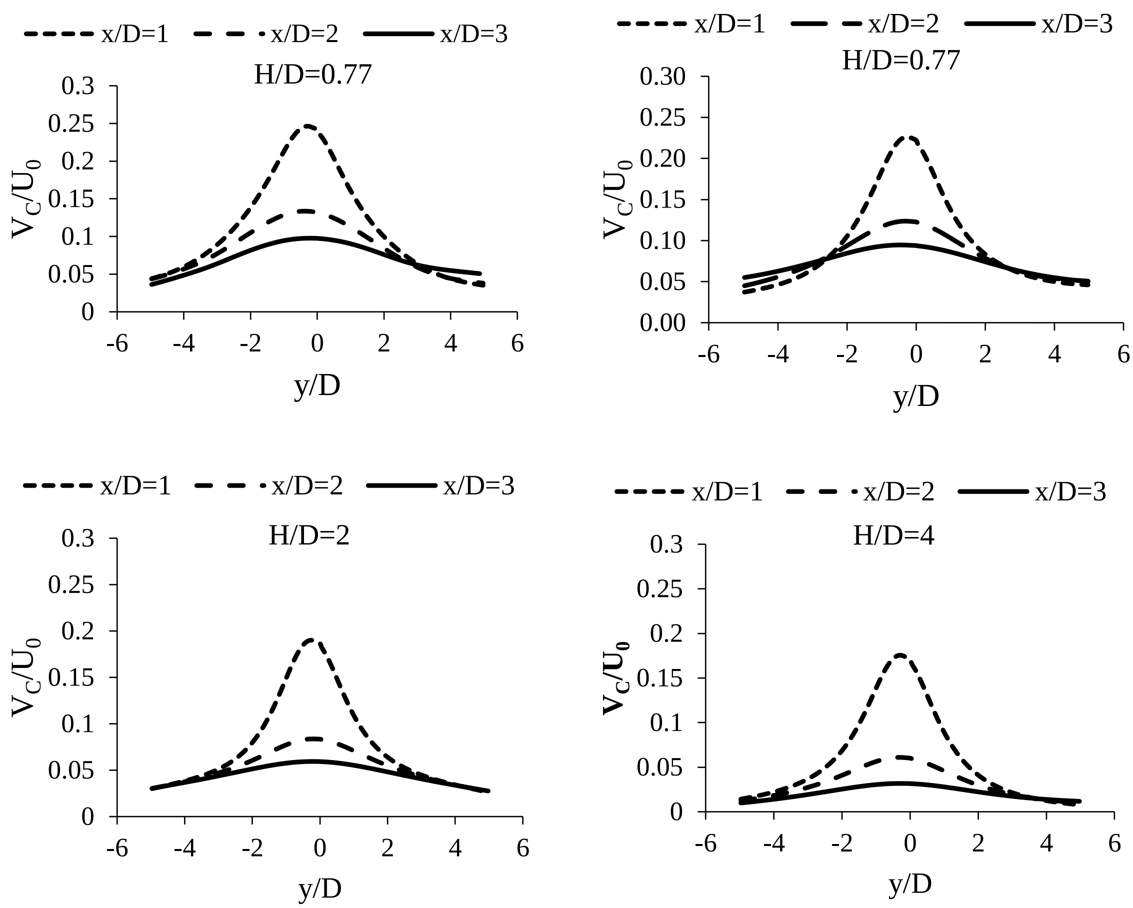

Fig. 5. A comparison of the results of the transverse dimensionless water velocity profile along the y-axis for the intended submergence ratios and relative distances

Shear stress distribution upstream of the orifice

Fig. (6) demonstrates the shear stress distribution upstream of the orifice for the intended submergence ratios. As it is shown, the shear stress on the bed is increased with the depth upstream of the orifice, since at larger depths, the velocity of the outflow from the orifice has increased, resulting in higher shear stresses at the floor. As a result, with the rise in the submergence ratio from 0.5 to 4, the shear stress grew by 148% from 4.31 Pa to 10.7 Pa. It is also observed that at higher submergence ratios, larger domains upstream of the orifice were affected by the shear stress. According to Powell and Khan (2011), after opening the bottom gate of a dam and starting the pressurized flushing, the movement of the sediments at the first stage starts due to the shear stress formed on the bed. Thus, according to the discussions, with a rise in the submergence ratio upstream of an orifice, more sediments are expected to be washed by the shear stress and discharged by the water flow.

 DOI: 10.22055/JISE.2021.36268.1943

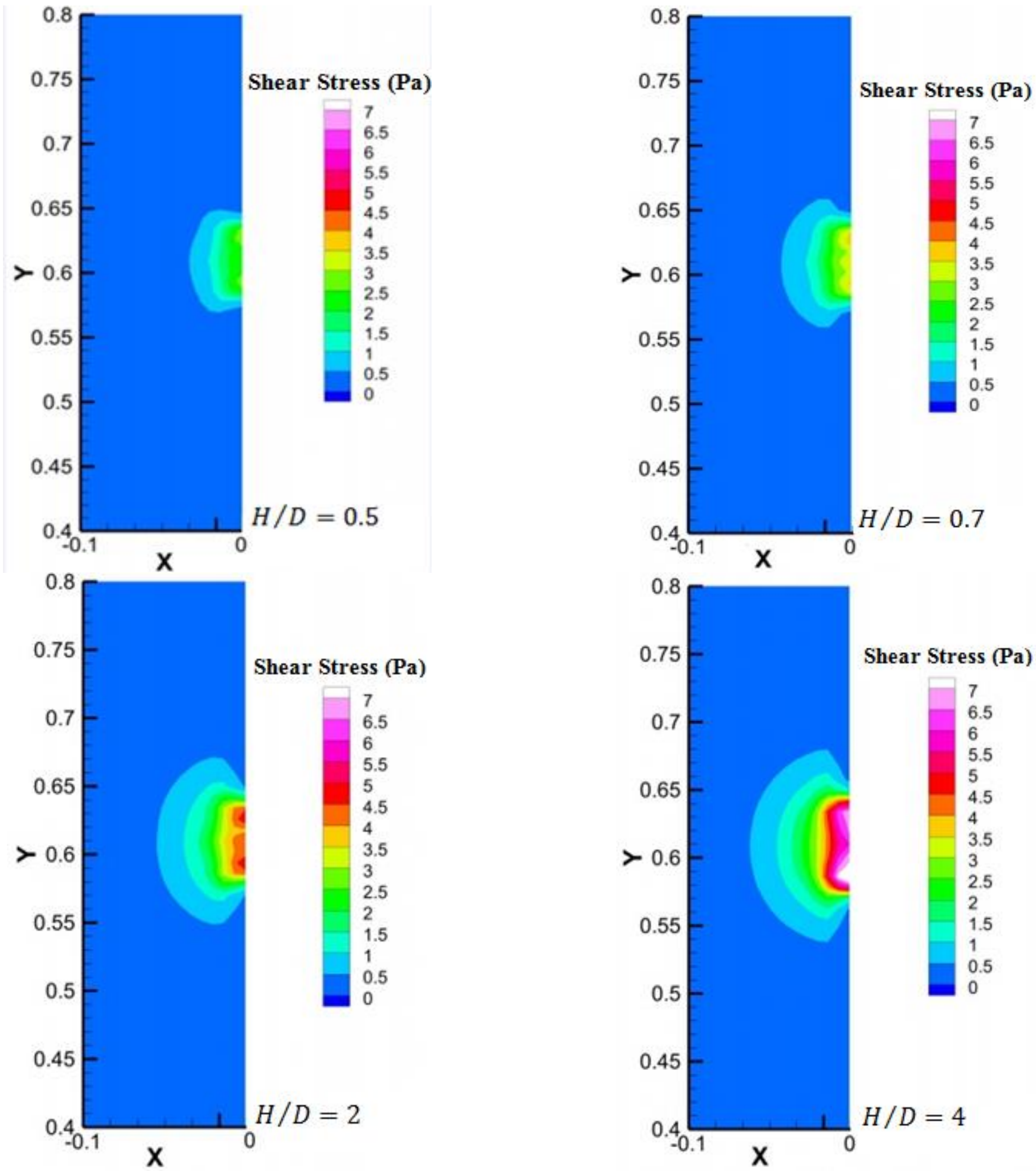

Fig. 6-Shear stress distribution upstream of the orifice for different depths

Conclusions

The current analysis undergone a feasibility study on using the FLOW-3D model in predicting the flow pattern upstream of an orifice. In this regard, some scenarios were defined in the model for different water depths upstream of the orifice, leading to the following results:

- The FLOW-3D software modeled the longitudinal velocity profile upstream of the orifice with sufficient accuracy.
- The Laminar, $k-\varepsilon$, and LES turbulence models had high and almost equal accuracy in predicting the longitudinal velocity profile upstream of the orifice.
- With the rise in the water depth upstream of the orifice, the general form of the longitudinal velocity profile became constant following an exponential equation.
- At larger distances upstream of the orifice, the velocity changes the width decreased, and the transverse velocity profile became uniform.

- The rise in the submergence ratio upstream of the orifice resulted in higher shear stresses and larger bed areas affected by it.

Acknowledgement

We are grateful to the Research Council of Shahid Chamran University of Ahvaz for financial support (GN: SCU.WH98.31370). The authors are also grateful to the Center of Excellence for the Improvement and Maintenance of the Irrigation and Drainage Networks for provision a part of data.

References

1- Bryant, D.B., Khan, A.A. and Aziz, N.M., 2008. Investigation of flow upstream of orifices. *Journal of Hydraulic Engineering*, *134*(1), pp.98-104.

2- Chanson, H., Aoki, S.I. and Maruyama, M., 2002. Unsteady two-dimensional orifice flow: a large-size experimental investigation. *Journal of Hydraulic Research*, *40*(1), pp.63-71.

3- Chapokpour, J., Ghasemzadeh, F. and Farhoudi, J., 2012. The numerical investigation on vortex flow behavior using FLOW-3D. *Iranica Journal of Energy & Environment*, *3*(1), pp.88-96.-

4- Dargahi, B., 2010. Flow characteristics of bottom outlets with moving gates. *Journal of Hydraulic Research*, *48*(4), pp.476-482.

5- Powell, D.N. and Khan, A.A., 2011. Sediment transport mechanics upstream of an orifice. *Journal of visualization*, *14*(4), pp.315-320.

6- Powell, D.N. and Khan, A.A., 2012. Scour upstream of a circular orifice under constant head. *Journal of hydraulic research*, *50*(1), pp.28-34..

7- Powell, D.N. and Khan, A.A., 2015. Flow field upstream of an orifice under fixed bed and equilibrium scour conditions. *Journal of Hydraulic Engineering*, *141*(2), p.04014076.

8- Shahmirzadi, M.M., Dehghani, A.A., Meftahh, M. and Mosaedi, A., 2010. Experimental investigation of pressure flushing technique in reservoir storages. *Water Geosci*, *1*, pp.132-137.

9- Shammaa, Y., Zhu, D.Z. and Rajaratnam, N., 2009. Flow field in a rectangular basin with a line inlet and a circular outlet. *Journal of Hydraulic Engineering*, *135*(10), pp.857-864.

10- Wei G, Brethour J, Grünzner M, Burnham J. The sedimentation scour model in FLOW-3D®. Flow Sci. Rep. 2014 Jun:3-14.