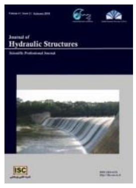
Journal of Hydraulic Structures
J. Hydraul. Struct., 2021;7(3):42-58
DOI: 10.22055/jhs.2021.37933.1180



OPEN ACCESS

Application of numerical model in determining the discharge coefficient containing suspended sediments passing through side weirs

Mousa Rasaei [1]
Amirreza Salemian[2]

Abstract

Side weirs are one of the most common water structures that are used to transfer or pass flood and excess water from headwater to downstream in channels and dams. One of the factors, which is often less considered in the design of such weirs, is the amount of suspended load along with the flow. Basically, suspended sediments along with flow, in addition to changes in the density of the passing water, can change most of the assumptions in the design of weirs. Due to the high cost and time-consuming nature of physical modeling, the powerful Flow-3D numerical model was used to simulate the flow of suspended sediments in this study. A channel with side weir was modeled according to laboratory conditions and the discharge coefficient passing through the side weir at different concentrations of suspended load was calculated. The results, while confirming the ability of the Flow-3D numerical model to simulate the flow containing sediment passing through the side weirs, showed that with increasing the concentration of suspended flow load, the discharge coefficient passing through the side weir increases. Also, using linear regression, experimental relationships were proposed to calculate the discharge coefficient through the side weir, taking into account the suspended load concentration.

Keywords: Side Weir, Suspended Sediments, Discharge Coefficient, Flow-3D Numerical Model.

Received: 10 July 2021; Accepted: 24 September 2021

1. Introduction

Side weir is one of the oldest hydraulic structures that is widely used in urban sewerage and drainage systems, water supply and irrigation, flow control, flood diversion and excess discharge of rivers and channels. This type of weir is installed on the side of the channel and when the water level reaches the level of the weir crown, the excess water is automatically diverted from the main channel. Figure 1 shows an example of a side weir constructed on a channel with a

[1] Department of Civil Engineering, Islamshahr Branch ,Islamic Azad university , Islamshahr ,Iran.Email: rasaei.iau@gmail.com **(Corresponding Author)**
[2] Department of Civil Engineering, Dehdasht Branch ,Islamic Azad University, Dehdasht ,Iran.

flow free surface. Due to the numerous applications of side weirs, these types of weirs must have characteristics in order to be used properly. One of the factors, which is often less considered in the design of such weirs, is the amount of suspended load along with the flow. Because physical modeling of sediment load with flood flow is a very costly, difficult and time-consuming process. Basically, suspended sediments along with flow, in addition to changes in the density of the passing water, can change most of the assumptions in the design of weirs [1].

Figure 1. An example of side weir made on a channel with free flow surface [1]

In general, in calculating the theory of the amount of water passing over the weir, which is a function of water height on the weir crown, gravity acceleration and length of the weir crown, a coefficient less than one called the discharge coefficient (Cd) must always be multiplied by the above parameters to equal the theoretical value to the actual value. In other words, the discharge coefficient is a numerical dimensionless that is determined in the laboratory and is unique to each weir. In general, the accuracy in determining the discharge rate passing the weir depends on the discharge coefficient and the accuracy of measuring the water head on the weir crown. Also, the lateral compression of the weir or the ratio of the weir crown length to the channel width is effective in transferring the flow turbulence to the water jet on the weir and the amount of flow rate and discharge coefficient.

In recent decades, extensive research has been conducted on side weirs with different geometric shapes and with the aim of studying and presenting relationships to determine the amount of discharge coefficient, which have been mainly laboratory [2, 3, 4, 5, 6, 7, 8, 9, and 10]. But, with the growth and development of the science of computational fluid dynamics (CFD), the use of numerical models expanded significantly so that in addition to accuracy and speed in calculations, the ability to model in various conditions was provided.

Perhaps, Cassidy [11] can be named as the initiator of numerical studies in the field of flow simulation over the side weirs. Using the relaxation technique, he obtained flow coefficients, pressure distributions, and free surface profiles for a side weir with a limited height and a specific crown shape. He concluded that the role of viscosity in determining the free flow surface is negligible. Ikegawa and Washizu [12] reviewed the cascade analyzes using the finite element method and obtained better answers. Betts [13] and Li et al. [14] using the linear finite element method and the principle of instability, obtain acceptable results for free surface and the pressures on crown compared with laboratory results. Savage and Johnson [15] built a physical model of a Plexiglas and placed it in a laboratory flume. They installed pressure control valves along the entire length of the weir and recorded ten types of flow under different conditions. They also used Flow-3D software for simulation in this study. The results of the numerical model could well cover the laboratory results. Lee and Holly [16] in 2002 offered a more complete relationship by conducting more extensive studies on side weirs. Vasquez and Walsh [17] showed that the Flow-3D numerical model has sufficient potential in designing and simulating the flow around the piers. In 2010, Ferrari [18] conducted a study with the aim of numerically modeling the free water surface in sharp-edged weirs, using the Smoothed Particle Hydrodynamics (SPH) technique to achieve the optimal model, natural aeration to water flow in successive weirs, and investigating the composition of weirs and optimal discharge in multiple. Ghanadan et al. [19] with the help of Flow-3D software, numerically simulated the flow from the side weir of the wide edge and compared the results of this software with laboratory data. The results showed that among the turbulence models in the software, RNG and K-Ɛ models have higher accuracy for simulating side weir. In 2012, Anderson and Tullis [20] examined the effect of different depths upstream of the dam, side slopes, and their effect on the discharge coefficient. Zahiri et al. [21] Studying side weirs of sharp rectangular composite edges. By defining the weight height of the crown, they provided relationships for estimating the discharge coefficient. Namaee and Shadpoorian [22] examined the flow pattern on the side weir both experimentally and numerically. In 2015, Parsaie and Haghiabi [23] investigated the application of neural networks to increase the accuracy of the numerical model in predicting the discharge coefficient passing over the side weirs. The results showed that the use of neural network to estimate the discharge coefficient increases the accuracy of the final model by about 16%. Karimi et al. [24] analyzed the discharge coefficient of 9 side weirs of the piano key with different geometries. They found that discharge coefficient of the piano key side weir was significantly higher than the discharge coefficient of rectangular side weir. In 2020, Gharib et al. [25] Used a new artificial intelligence model called the Self-Adaptive Extreme Learning Machine (SAELM) to simulate the discharge coefficient of side weirs in rectangular channels. They also used Monte Carlo simulations to evaluate the ability of numerical models. Examination of numerical models showed that such models simulate the discharge coefficient with acceptable accuracy.

Zakwan and Khan [26] studied the discharge coefficient through side weirs. In their research, they used the general reduced gradient (GRG) technique as a powerful optimization method. The results showed that the estimation of the discharge coefficient through GRG method had more accurate results. Lindermuth et al. [27] analyzed the effect of channel and weir characteristics on the behavior of flow through side weirs based on a comprehensive parametric study and using three-dimensional FLOW-3D numerical model. They were able to introduce a revised formula for estimating the discharge coefficient through side weirs using multiple regression analysis. Also, the validation results of numerical simulations with physical model showed that there is a good agreement between the results.

Recent research records show that the determination of discharge coefficient through side weirs has been studied by various researchers. A noteworthy point in previous research is the lack of a comprehensive study to determine the discharge coefficient through side weirs by considering the amount of suspended sediment numerically. Therefore, this study was conducted with the aim of numerically investigating the discharge coefficient through side weirs in the conditions of flow with the clear water and flow with the suspended sediments using Flow-3D software.

2. Material and Methods
2.1. Flow-3d numerical model

The use of CFD as a powerful tool for fluid flow analysis in systems with asymmetric geometry and complex governing equations has expanded dramatically in recent years by researchers and engineers. One of the most powerful software in this field is the FLOW-3D numerical model, which was introduced by Tony Hirt in 1980. FLOW-3D software is dedicated to three-dimensional flows and has a special ability in analyzing flows with free surface. This software uses the finite volume method in calculations and uses various turbulence models such as LES, k-ε and RNG, etc. The software also uses two numerical techniques of the fluid volume method (VOF) which is to show the behavior of the fluid on the free surface and the Fractional Area/Volume Obstacle Representation (FAVOR) method which is used to simulate solid surfaces and volumes such as geometric boundaries. Modeling flows containing suspended sediments can also be considered as special capabilities of this software [1]. The governing equations of this model include continuity and momentum equations. On the other hand, the continuity equation can be represented as Equation (1) by ignoring the fluid compaction capability in the form of Cartesian coordinates (x, y, z).

$$V_F \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(uA_x) + \frac{\partial}{\partial y}(vA_y) + \frac{\partial}{\partial z}(\rho w A_z) = \frac{R_{SOR}}{\rho} \tag{1}$$

Here u, v, and w, are respectively the velocity components in the x, y and z directions. A_x, A_y, and A_z, are also surface fractions for flow in the x, y and z directions, respectively. Also in Equation (1), V_F is the volume fraction of the flow, ρ is the fluid density and R_{SOR} is the mass source.

$$\begin{aligned}
\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right\} &= -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x \\
\frac{\partial v}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial v}{\partial x} + vA_y\frac{\partial v}{\partial y} + wA_z\frac{\partial v}{\partial z}\right\} &= -\frac{1}{\rho}\frac{\partial P}{\partial y} + G_y + f_y \\
\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial w}{\partial x} + vA_y\frac{\partial w}{\partial y} + wA_z\frac{\partial w}{\partial z}\right\} &= -\frac{1}{\rho}\frac{\partial P}{\partial z} + G_z + f_z
\end{aligned} \tag{2}$$

In these equations G_x, G_y and G_z are the terms of mass acceleration. f_x, f_y and f_z are the terms of viscosity acceleration. P is also defined the pressure at any point in the fluid.

2.2. Models and laboratory data used in numerical simulation

In this research, the data and experimental results of Ayyoubzadeh et al. [9] have been used in order to simulate and validate the FLOW-3D numerical model. Figure (2) shows a schematic of the channel plan and side weir. The flume is a Plexiglas channel with a length of 10 meters, height 46 and width 30 centimeters, which a side weir is installed in the wall in the last third of the channel.

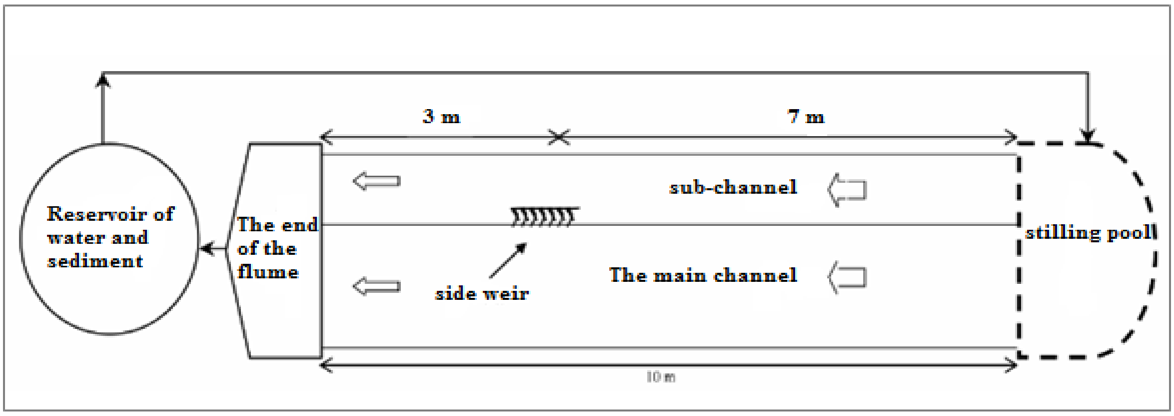


Figure 2. Channel plan and side weir in laboratory studies

2.3. Meshing

As mentioned earlier, in this study, numerical simulations have been performed based on the laboratory model of ayyoubzadeh et al. [9]. One of the weaknesses of the Flow-3D model is the inability to draw and create the model geometry. For this purpose, the first, using Solid works software, the field geometry was drawn in three dimensions according to Figure (3). After drawing the field geometry, the networking and geometry conditions for the model must be defined. One of the most important advantages of the Flow-3D numerical model compared to other simulation models is in determining and estimating the best solution field networking based on the studied geometric model. For this purpose, the size of computational cells in the longitudinal and transverse directions and height (X × Y × Z) with dimensions of 4.7 cm was considered. After validating the dimensions of the networking cells by the FAVOR section of the numerical model, it was determined that the dimensions of the selected cells are not suitable for channel and weir geometry. In other words, the whole geometry of the field is not covered by 4.7 cm cells, so that part of the channel and weir were not observed in the model. Finally, by shrinking and changing the dimensions of the networking cells by 2.7 cm in the longitudinal, transverse and depth of flow, the most optimal and appropriate dimensions of the networking cells were obtained. At the same time, as the cells of the network became smaller, the accuracy of the calculations increased dramatically. In total, about 118000 computational networks were considered for the above model. The different conditions of computational cell dimensions are shown in Figure (3) below.

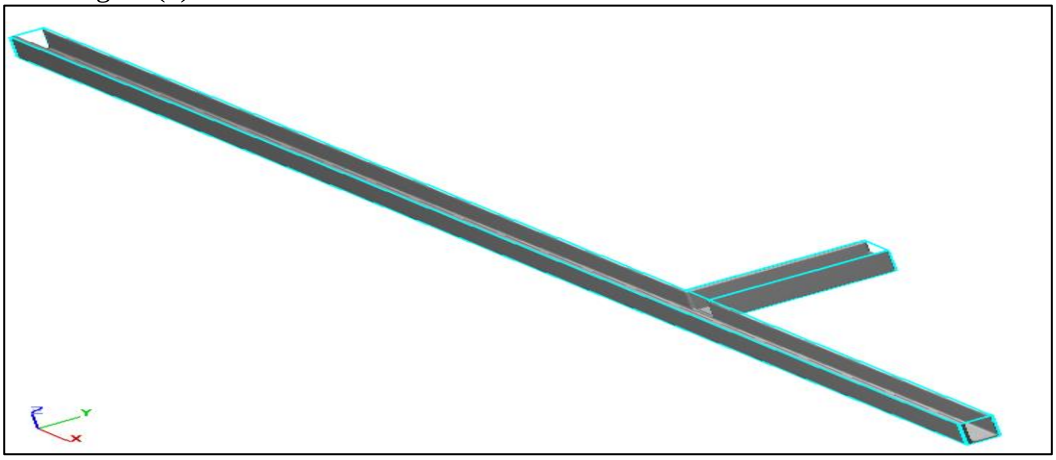


Figure 3. Calibrated networking for model geometry

2.4. Boundary conditions

Applying appropriate boundary conditions in the numerical model is one of the most important parameters in software simulation. In this research, in order to be stable and increase the accuracy of calculations, the boundary conditions appropriate to the laboratory conditions in the numerical model have been used. For this purpose, the boundary conditions of the channel floor (Z_{min}) were considered as the walls of the laboratory model of rigid material. Since in the experimental model the flow has a free surface and is in direct contact with air, the symmetry boundary conditions were used at the model surface (Z_{max}). It should be noted that in this case the rotation of air pressure changes inside and outside is considered zero and since in Flow-3D model there is no need to define the air phase as a new phase in hydraulic problems, the free surface flow conditions and direct contact with the relative air pressure is fully established. Regarding the correct selection and application of flow input and output boundary conditions in (X_{min}) and (X_{max}) sections, based on laboratory studies, a stable flow with a certain height from the fluid should be introduced to the numerical model. Therefore, using the boundary conditions in the Flow-3D numerical model in this section, different values of fluid height were applied for the X_{min} section with the fluid height boundary conditions, which is exactly consistent with laboratory studies. On the other hand, after passing the side weir, which is installed in the rectangular channel, it should be directed to the output boundary, where the output flow is applied downstream as Outflow. For the boundary condition between two blocks, the Symmetry boundary condition was used so that creating such a boundary does not change the results between the two blocks. Figure (4) shows all the applied boundary conditions.

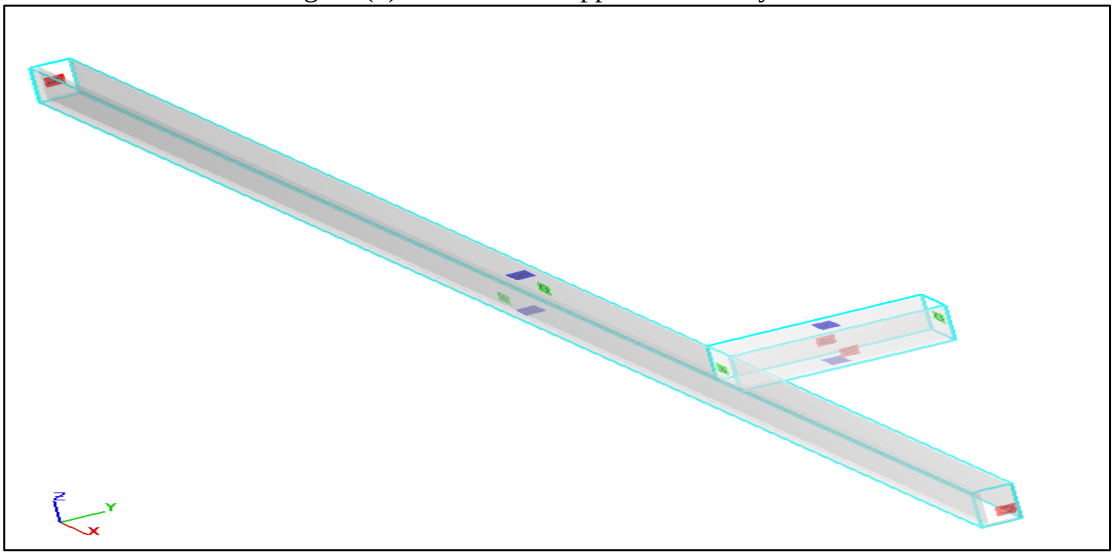
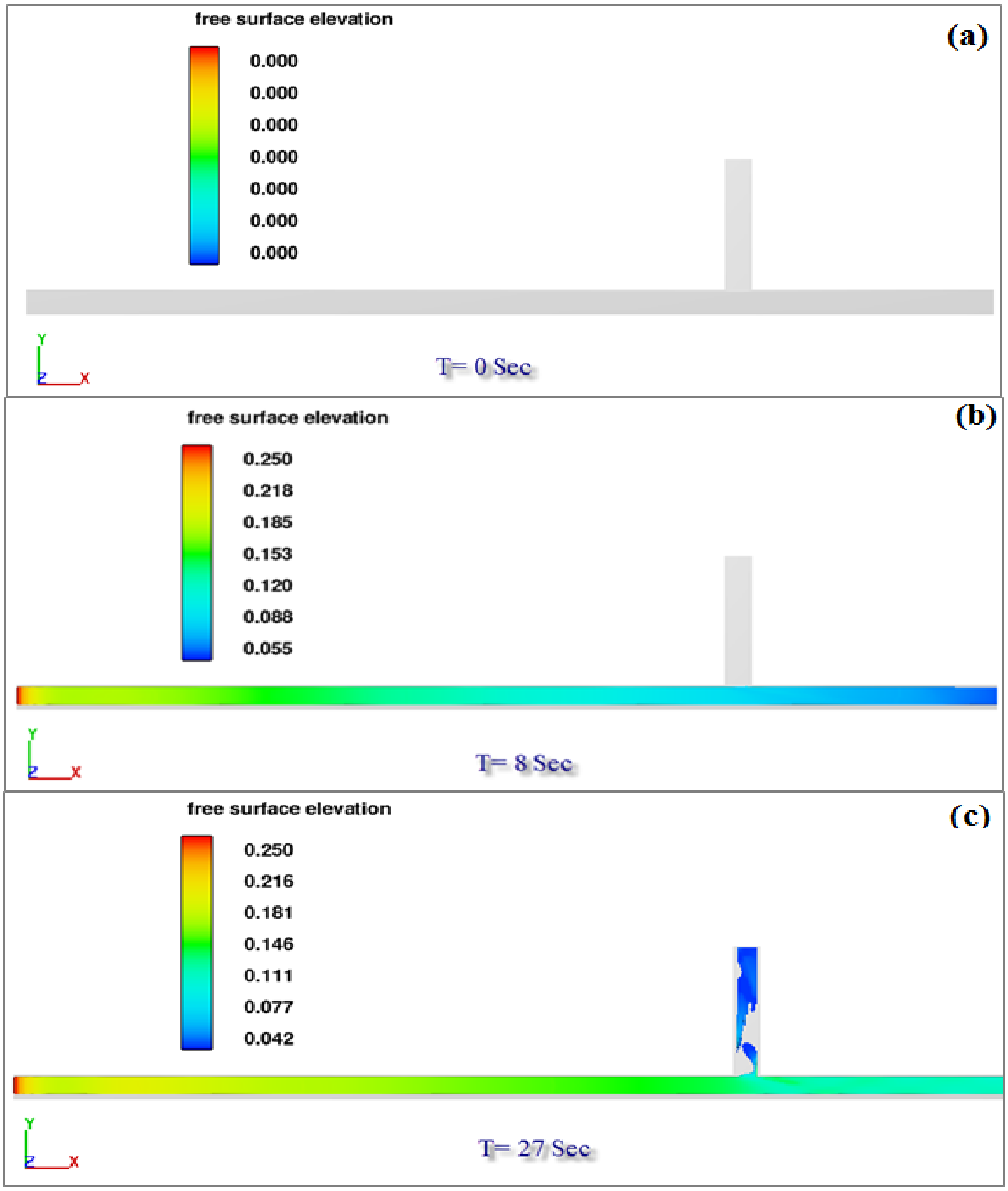


Figure 4. Boundary conditions applied in numerical modeling according to laboratory conditions

2.5. Calibrate the number model

What has been discussed so far in the previous sections is the calibration of the numerical model in terms of networking and boundary conditions. However, in order to extract the correct and accurate values of the data of a numerical model, in addition to networking and appropriate boundary conditions, it is necessary to consider the number of computational steps in order to reach the stable condition of the model.

After reviewing and implementing several models with multiple time steps, finally the appropriate time to extract the results from the numerical model in end of calculations was considered equal to 50 computational steps. Figure 5 shows the output diagrams of the numerical model calibration process and how the flow passes through the main and lateral channels in different time steps. As shown in Figure (5b), for up to 8 seconds, the current flows only in the main channel to reach the required height of the side weir. Then, in 27 seconds, the flow passes through the side weir and enters the side channel (Figure (5c)). Finally, after a period of 50 seconds, the flow is formed on both the main and side channels and reaches a steady state (Figure 5d). Figure 6 also shows the final schematic of the input flow pattern to the side channel.



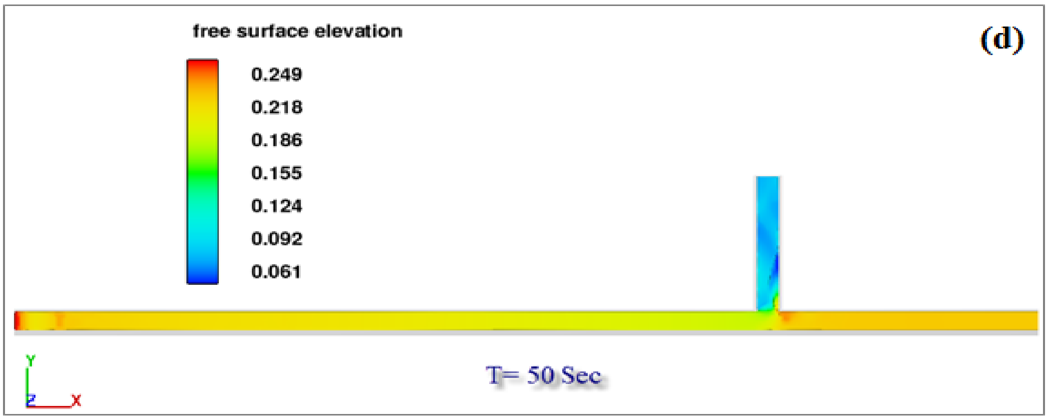


Figure 5. Different time steps of flow stability in the main and lateral channel a) 0 seconds b) 8 seconds c) 27 seconds d) 50 seconds

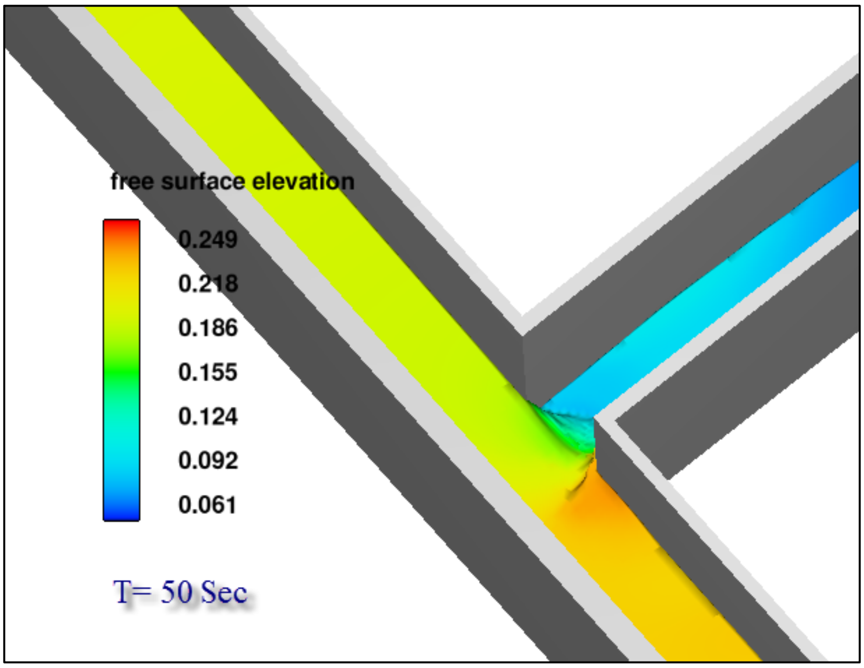


Figure 6. The final shape of the input flow pattern to the output side channel of the FLOW-3D numerical model

Figure 7 also shows the results of changes in the output flow from the boundaries of the end and beginning of the main and side channels. As can be seen, after an approximate time of 50 seconds, the total volume of the output flow from the boundary of the end of the main and side channel is almost constant and the so-called flow has reached a steady state.

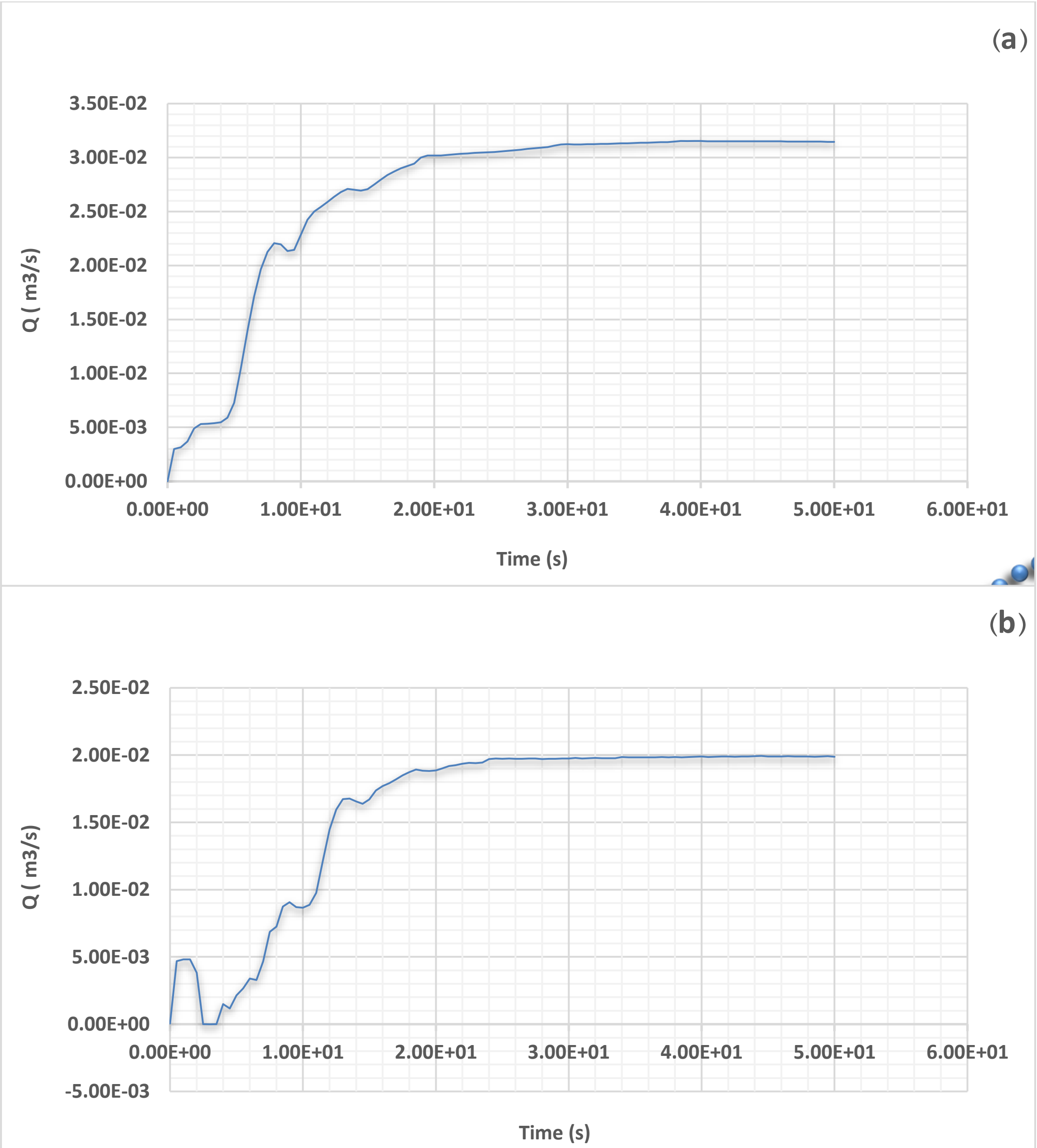


Figure 7. Flow rate across the output boundary a) Main channel and flow stabilization in the main channel b) Side channel and flow stabilization in the side channel

For more accurate calibration of the numerical model, the discharge confident over the side weirs was calculated using the output values of the numerical model and placement in Equation (1) and with laboratory data from the results of Ayyoubzadeh et al. [9], Comparison and validation were performed.

$$Q=\frac{2}{3}\sqrt{2g}\,C_d\,BH^{\frac{3}{2}} \tag{3}$$

In the above relation, Q, g, B, H, and Cd are flow rate, gravity acceleration, weir width and water flow height before weir and discharge coefficient respectively. In Table (1), the values of the discharge coefficient obtained in the laboratory studies of Ayyoubzadeh et al. [9] and the results of the numerical model of this research are shown in two modes of flow with clear water and flow with suspended load and are compared and validated.

Table 1. Comparison of laboratory and numerical results of passage discharge coefficient in flow with clear water and flow with suspended load

Y_d/P	Discharge coefficient obtained from laboratory model		Discharge coefficient obtained from numerical model		Relative error percentage%	
	Flow with Clear water	Flow with suspended load	Flow with Clear water	Flow with suspended load	Flow with Clear water	Flow with suspended load
0.150	0.651	0.667	0.642	0.672	1.32%	0.71%
0.200	0.657	0.668	0.658	0.680	0.23%	1.82%
0.250	0.645	0.666	0.649	0.662	0.69%	0.61%
0.300	0.655	0.664	0.656	0.670	0.21%	0.94%
0.350	0.652	0.672	0.667	0.647	2.32%	3.73%
0.400	0.658	0.660	0.651	0.656	1.07%	0.68%
0.450	0.659	0.650	0.663	0.661	0.64%	1.71%

As can be seen in the table above, the maximum simulation error for the flow with clear water and the flow with suspended load is estimated to be about 2.3 and 3.7 percent, respectively. Therefore, it can be inferred that the results of the discharge coefficient passing over the side weirs in flow with the clear water and flow with suspended load in both numerical and laboratory models have a good agreement and the present numerical model is a calibrated and validated numerical model.

3. Discussion and results

Since the purpose of this study is to investigate the flow field containing suspended sediments in order to determine the discharge coefficient through the side weir, so by changing different concentrations of suspended load and different hydraulic and geometric conditions according to Table (2), the discharge coefficient passing through the side weirs were more comprehensively analyzed numerically.

Table 2. Hydraulic and geometric conditions

Variable	Range
Froude number	0.6, 0.7 and 0.8
Suspended load concentration.	3000, 5700 and 19500 mg/l
Weir height	40, 50 and 60 mm
Suspended load specific gravity.	2.65
Average diameter of sediment particles.	0.06 mm

3.1. Flow discharge coefficient

Figures 8, 9 and 10 show the diagrams of discharge coefficient changes through the weir with different heights according to the Froude number of the flow and in the flow conditions with clear water and flow with suspended load separately. The numerical model outputs show that by increasing the Froude number of the flow in all cases, including flow with clear water and flow with suspended load, and in all geometric conditions of the side weir, the discharge coefficient passing through the weir decreases. On the other hand, it is observed that in side weirs with different heights with increasing the concentration of suspended load, the discharge coefficient through the weir has also increased. Also, in all cases, the discharge coefficient through the weirs in the suspended load conditions has increased compared to the flow with clear water. The results of numerical analysis and calculations performed on the output graphs of the numerical model show that, for example, by increasing the flow rate by 33%, the flow rate through the weir in the state of flow with clear water and flow with suspended load, has decreased by an average of about 13 and 15 percent respectively. Also, the calculations showed that by increasing the concentration of suspended flow rate of the flow by 3000 ml compared to the flow with clear water, the discharge coefficient through the weir has increased by an average of about 3%. For other values and simulation conditions, these results were true in the same percentage range. On the other hand, the results showed that with a 550% increase in the suspended load concentration, the flow discharge coefficient has increased by an average of about 21%, which indicates the significant effect of the suspended load concentration on the weir.

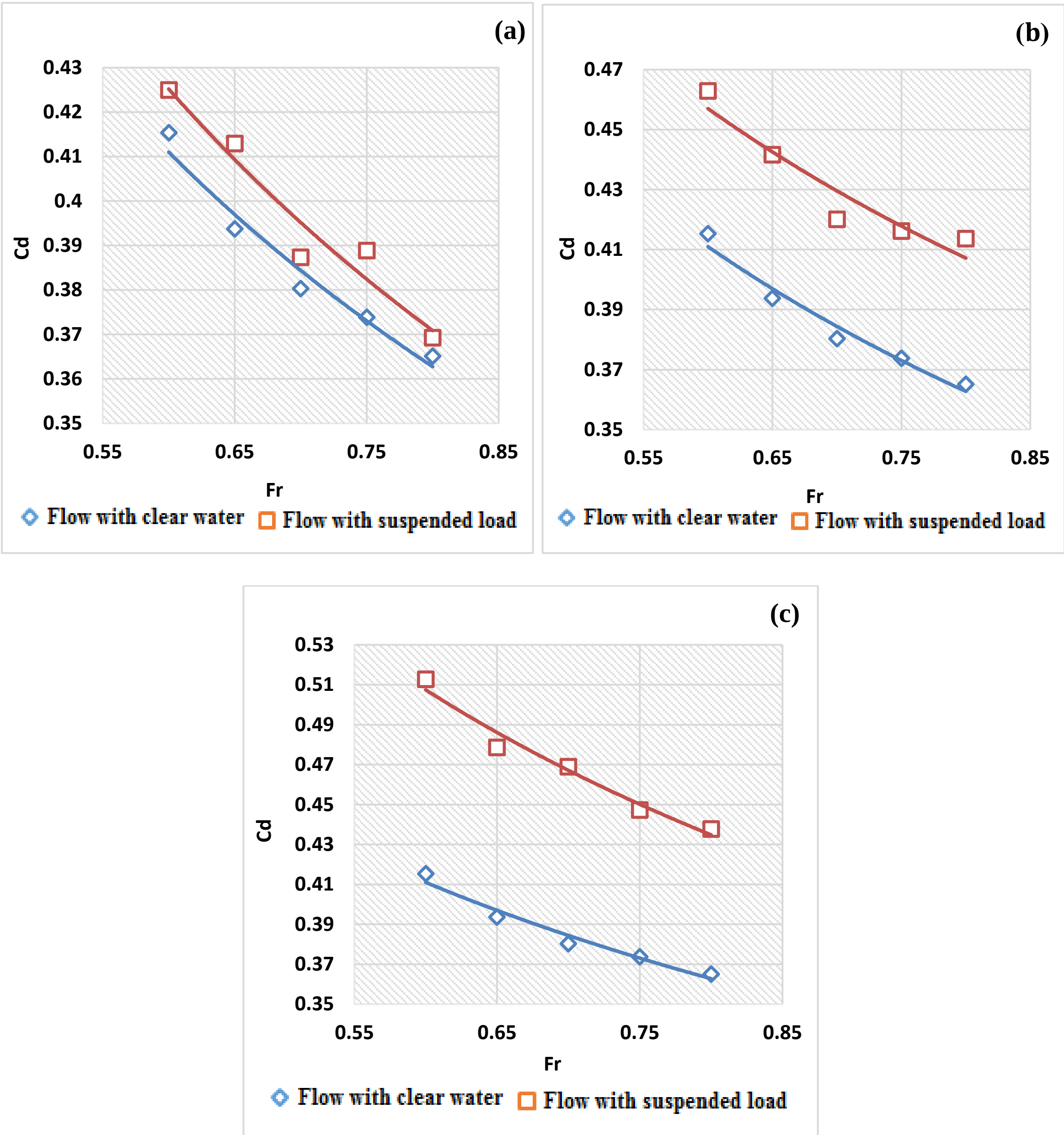


Figure 8. Comparison of discharge coefficient changes passing over the side weir with a height of 40 mm in state of clear water and water with suspended load by the amount. A) 3000 ml, B) 10500 ml, C) 19500 ml

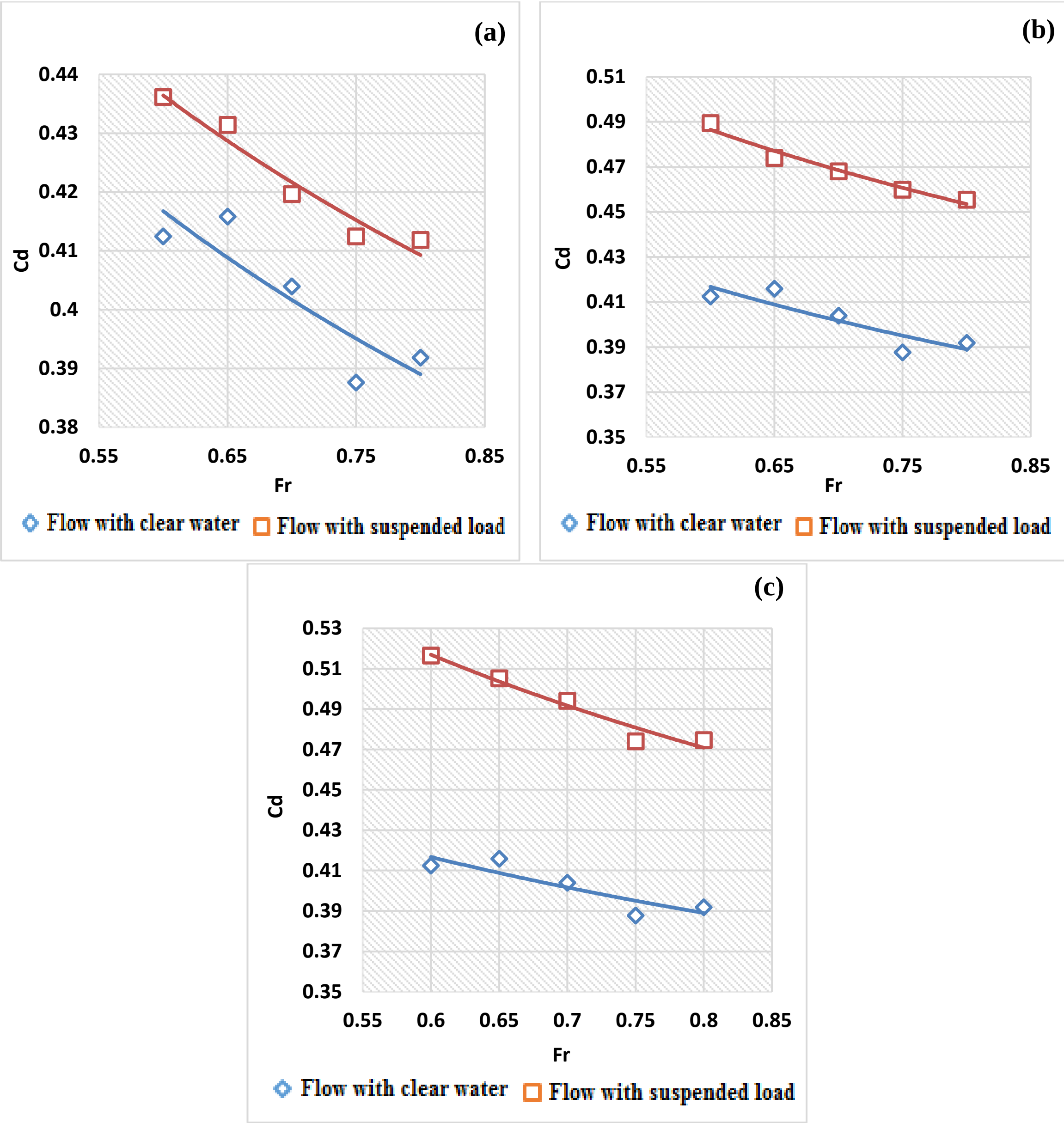


Figure 9. Comparison of discharge coefficient changes passing over the side weir with a height of 50 mm in state of clear water and water with suspended load by the amount. A) 3000 ml, B) 10500 ml, C) 19500 ml

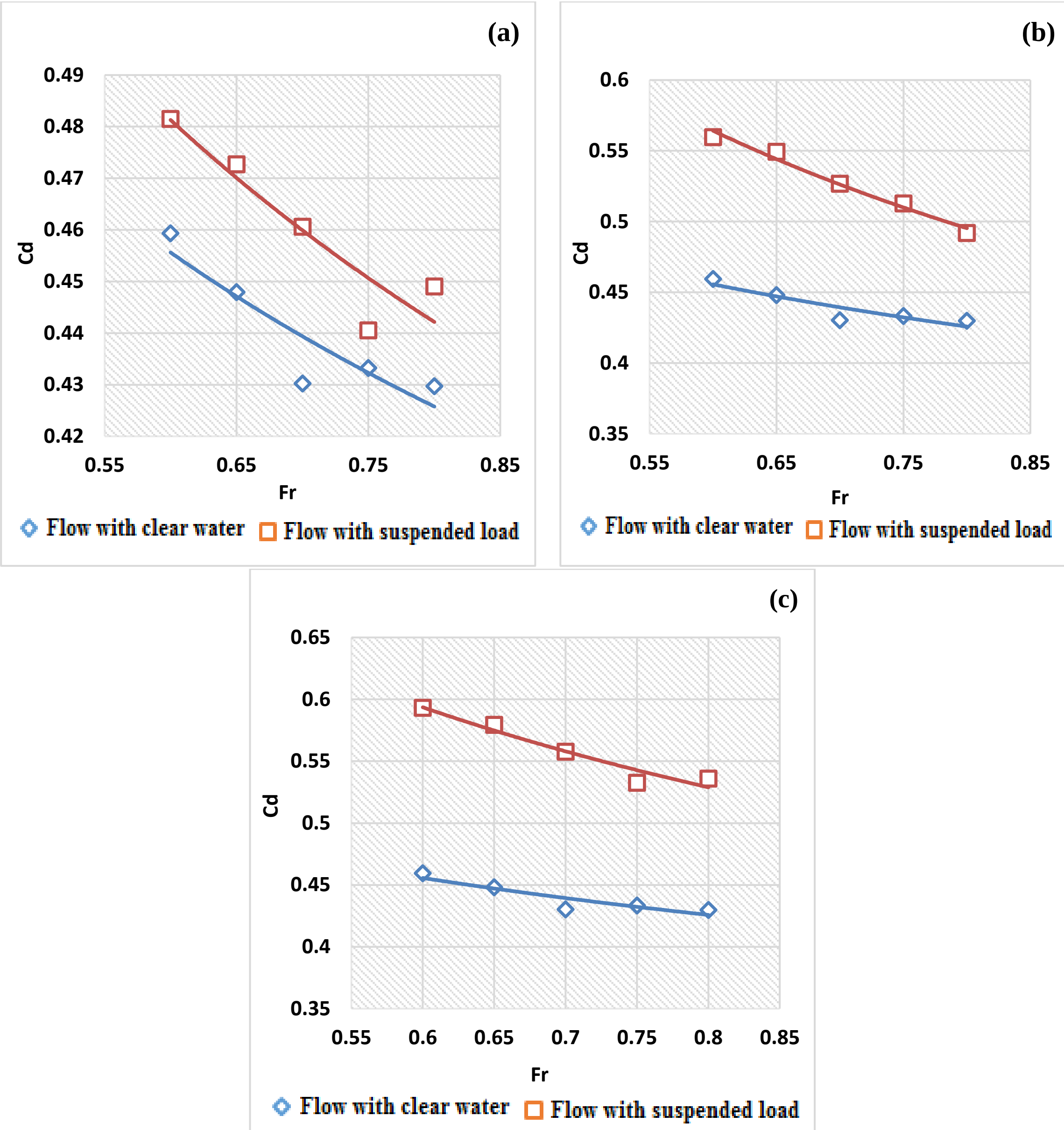


Figure 10. Comparison of discharge coefficient changes passing over the side weir with a height of 60 mm in state of clear water and water with suspended load by the amount. A) 3000 ml, B) 10500 ml, C) 19500 ml

3.2. Relation of passing discharge coefficient

Determining the discharge coefficient through the side weir is always a function of several parameters. Therefore, in this study, using the available data from different modeling conditions, the most optimal fitted relationships were introduced to determine the values of discharge coefficient passing through side weir with suspended load. One of the general and comprehensive relationships to determine the discharge coefficient based on the suspended load concentration and Froude number with regression coefficient R2 = 0.55 is:

$$C_d=0.6-(0.25\times Fr)+\left(5.0\times 10^{-6}.ppm\right) \tag{4}$$

Which, Cd: discharge coefficient passing through the side weir, Fr: Froude number and ppm: is the amount of suspended load concentration in the flow.

In another relation, the discharge coefficient passing over the weir was obtained based on the flow relative depth and with a regression coefficient of 0.91. In this equation, considering the values of the flow relative depth (Yd) to the height of the side weir (p), the Froude number of flow (Fr) and the flow concentration (p), the general relation (2) is presented as follows:

$$C_d=0.53-\left(\frac{0.07\times Y_d}{P}\right)+\left(5.0\times 10^{-6}.ppm\right) \tag{5}$$

As can be seen in the above relationship, the discharge coefficient relationship based on flow concentration rate and the relative height of the input flow to the channel also has a suitable regression coefficient.

In the following other relation based on the values of Froude number, flow concentration and relative flow depth in side weirs is presented as follows:

$$C_d=0.56-(0.12\times Fr)-\left(\frac{0.042\ Y_d}{P}\right)+\left(4.93*10^{-6}.ppm\right) \tag{6}$$

The results showed that the relationship (6) with regression coefficient of 0.96 and considering the most important parameters affecting the model is one of the most comprehensive relationships to determine the discharge coefficient through over the side weir in this study, which shows the accuracy of numerical modeling results.

4. Conclusion

- The maximum simulation error for the flow mode with clear water and the flow with suspended load was estimated to be about 2.3% and 3.7% compared to the laboratory data, respectively. This error rate indicates the accuracy of the Flow-3D numerical model in simulating the flow through side weirs compared to laboratory models.
- In all cases, the results of numerical modeling on side weirs show that the flow passing discharge coefficient in flow conditions with suspended load is higher than flow conditions with clear water.
- The results of numerical modeling show that in all cases, with increasing the Froude number, the discharge coefficient passing over the side weir has decreased.
- In all cases, with increasing the concentration of suspended flow load, the discharge coefficient passing over the side weir has increased.
- Considering the parameters affecting the discharge coefficient, the most important relationships extracted from the numerical model outputs were determined to determine the discharge coefficient passing over the side weirs.

References

1. Khani M, (2007). Numerical modeling of the flow field containing suspended sediments in the side weirs in order to determine the discharge coefficient. M.S.C Dissertation. University of Islamic Azad Rudehen. iran
2. De Marchi G, (1934). Essay on the performance of lateral weirs (in Italian). L'Energia Ellectrica Milan, 11(11): 849–860.
3. Subramanya K, Awasthy S C, (1972). Spatially varied flow over side-weir. J Hydr Engrg ASCE, 98(1): 1-10.
4. Nadesamoorthy T, Thomson A, (1972). Discussion of spatially varied flow over side weir. J Hydr Engrg ASCE, 98(2): 2234-2235.
5. Yu-Tech L, (1972). Discussion of spatially varied flow over side weirs. J Hydr Engrg ASCE, 98(11): 2046-2048.
6. Hager W H, (1987). Lateral outflow over side weirs. J Hydr Engrg ASCE, 113(4): 491-504.
7. Swamee P K, Pathak S K, Mohan M, Agrawal S K, Ali M S, (1994). Subcritical flow over rectangular side weir. J Hydr Engrg ASCE, 120(1): 212–217.
8. Borghei S M, Jalili M R, Ghodsian M, (1999). Discharge coefficient for sharp crested side weirs in subcritical flow. J Hydr Engrg ASCE, 125(10): 1051- 1056.
9. Ayyoubzadeh S A, Gohari-Asadi S, Vali-samani J M, (2006). The effect of suspended load on discharge coefficient of side weirs in rectangular channels. Journal of agricultural engineering research, 7(26).
10. Ameri A, Ahmadi A, Dehghan A A, (2015). Determination of discharge coefficient of compound triangular -rectangular sharp crested side weirs. Journal of Water and Soil Conservation, 22(3).
11. Cassidy J J, (1965). Irrigational flow over the spillways of finite height. J Mech Eng Div, ASCE, 91(6): 155–173.
12. Ikegawa M, Washizu K, (1973). Finite element method applied to analysis of flow over a spillway crest. J Numer. Methods Eng., 6: 179-189.
13. Betts, P, (1979). A variational principle in terms of stream function for free-surface flows and its application to the finite element method. Computers & Fluids – Compute fluids, 7:145-153.
14. Li W, Xie Q, Chen C J, (1989). Finite analytical solution of flow over spillway. Eng. Mech. ASCE,115: 2635-2647.
15. Savage B M, Johnson M C, (2001). Flow over ogee spillway: Physical and numerical model case study. J. Hydraulic Eng. ASCE, 127(8): 640-649.
16. Lee K L, Holly E R, (2002). Physical modeling for side channel weirs. Center of research for water resources. The university of Texas at Austin. Available on http:www.crwr.utexas.edu/online shtml.
17. Vasquez A, Walsh B W, (2009). CFD simulation of local scour in complex piers under tidal flow. 33rd IAHR Congress: Water Engineering for a Sustainable Environment. Vancouver, Canada, 913-920.
18. Ferrari A, (2010). SPH simulation of free surface flow over a sharp-crested weir. Advances in Water Resources, 33(3):270-276.
19. Ghanadan R, Zahiri A R, Kahe M, Jalalodin M S, (2012). Numerical simulation of wide edge -side weir, using flow-3d model. National Conference on Water and Wastewater Engineering. (In Persian).

20. Anderson R M, Tullis B P, (2012). Comparison of piano Key and rectangular Labyrinth Weir Hydraulics. Journal of Irrigation and Drainage Engineering, ASCE, 138(4), 358-361.
21. Zahiri A, Azamathulla H M, Bagheri S M, (2013). Discharge coefficient for compound sharp crested side weirs in subcritical flow conditions. J. Hydrol. 480: 162.
22. Namaee M R, Shadpoorian R, (2015). Numerical Modeling of Flow Over Two Side Weirs. ARABIAN JOURNAL FOR SCIENCE AND ENGINEERING. 41(4).
23. Parsaie A, Haghiabi A H, (2015). The Effect of Predicting Discharge Coefficient by Neural Network on Increasing the Numerical Modeling Accuracy of Flow Over Side Weir. Water Resources Management 29(4).
24. Karimi M, Attari J, Saneie M, Jalili M, (2017). Experimental study of discharge coefficient of a Piano Key Side Weir. Labyrinth and Piano Key Weirs III – PKW .109-116.
25. Gharib R, Heydari M, Kardar S, Shabanlou S, (2020). Simulation of discharge coefficient of side weirs placed on convergent canals using modern self-adaptive extreme learning machine. Appl Water Sci, 10(50) (2020).
26. Zakwan M, Khan I, (20200. Estimation of Discharge coefficient for side weirs. Water and Energy International, 62(11):71-74.
27. Lindermuth A, Ostrander P, Achleitner S, Gems B, Aufleger M, (2021). Discharge Calculation of Side Weirs with Several Weir Fields Considering the Undisturbed Normal Flow Depth in the Channel. Water, 13, 1717.