

# A Comparison on Discharge Coefficients of Side and Normal Weirs with Suspended Flow Load using FLOW3D

M. Taghavi<sup>1\*</sup> and H. Ghodousi<sup>2</sup>

<sup>1</sup>Department of Civil Engineering, Islamic Azad University, Kish Branch, Kish Island, Iran; mehditaghaavi@gmail.com

<sup>2</sup>Department of Water Engineering, University of Zanjan, Zanjan, Iran

## Abstract

Weirs and gates are structures which are used to measure water flow in open channels. Entering rivers' sediments into channels is considered as one of the main problems in irrigation and water transmission networks. The relations presented based on laboratory results for gates, weirs and apertures' discharge coefficients have been extracted and developed based on the conditions of sediment free water flow. Therefore, applying these relations in natural conditions requires investigating the effect of suspended load on hydraulic coefficients which are highly effective to technically and economically design and select these structures. The purpose of the present paper was to determine the amount of flow discharge and the type of effective parameters using different geometrical and hydraulic conditions such as overflow height. It was also attempted to present the relation of discharge coefficient at the presence of suspended load. To this end, three-dimensional FLOW3D model was employed. This numerical model can simulate the flow containing suspended sediments and sediments' deposition in various parts of channel. In numerical simulation, based on landing number of flow,  $C_d$  has a determination coefficient ( $R^2$ ) relatively better and higher than laboratory results. It can be attributed to the lack of suspended load thickness distribution in laboratory and difficult conditions of such action in laboratory modeling. In laboratory model, suspended load cannot be fully injected with appropriate ratio to the entire fluid. However, in numerical model, flow thickness amount can be easily added an injected to the model from input boundary on cubic meter unit of flow. This significant point is considered as one of the advantages of numerical model to simulate such problems. In the present study, total relation of flow discharge ( $C_d$ ) was presented as a combination of values of flow landing ( $F_r$ ), flow thickness amount ( $S$ ) and relative low depth ( $y_d/P$ ) in side weirs.

**Keywords:** Discharge Coefficients, Flow3D, Side Weir, Suspended Sediment

## 1. Introduction

Weirs and gates are structures which are used to measure water flow in open channels. Entering rivers' sediments into channels is considered as one of the main problems in irrigation network. Flow measurement structures are structures which are used for volumetric measurement in irrigation networks, water transmission lines and water tanks. Generally, based on discharge measurement method, flow measurement structures are divided in two classes:

- Flow measurement structures which measure flow discharge through creating critical depth such as various overflows and flumes;
- Structures which compute flow discharge by measuring velocity and using crossover geometrical characteristics such as molinet and ultrasonic flow meters.

In quick wash (hydraulic descaling) of dams' reservoirs which is performed through flushing operation to maintain and revive dams' reservoir capacity, flow passing through the related channel is affected by suspended

\* Author for correspondence

load. This effect should be investigated on discharge coefficient of such channel. On the other hand, more precise relation or relations should be obtained between water discharge and sediment discharge based on which the amount of suspended sediment can be more precisely estimated. Such an issue is highly helpful for experts of river and coastal engineering as well as water resources' management planners.

In addition to review related studies, to compare discharge coefficient of side and normal weirs with suspended load flow through FLOW3D software, the present study set the following objectives:

- Observing the effect of suspended load on discharge coefficient in side and normal weirs with various concentrations of suspended load
- Comparing discharge coefficient of side and normal weirs through numerical and laboratory methods
- Numerically simulating side and normal weirs at the presence of suspended load and computing discharge coefficients with pure water
- Investigating hydraulic parameters such as pressure, flow depth and flow velocity
- Determining the density of flow with suspended load

Studies on the presence of suspended load in the performance of sharp crested weirs have a long history. As the most important studies, we can refer to the researches performed by Moste and Etna who investigated the effect of dike length on rotating area behind the dike with respect to the effect of scale through FLOW3D Software. Simulating flow around cylinder in erosive bed through FLOW3D, Smith and Foster found out that the effect of bed wash, velocity profile and created vortices forms are consistent with laboratory model. Samani et al.<sup>17</sup> investigated discharge coefficient of side and normal weirs with suspended load in laboratory piece. They found that flow intensity coefficient in these two overflows is influenced by geometric parameters of structure and flow hydraulic. To investigate this effect, during two series of experiments, the effect of flow suspended load on flow intensity coefficient of side and normal weirs revealed that in both structures, flow intensity coefficient is influenced by flow suspended loads. Also, flow intensity coefficient is increased as a result of increasing flow suspended load. Brethour and Burnham simulated sediment corrosion in dam downstream using FLOW 3D Software. In their study, to compute each of characteristic coefficients of sediment through FLOW 3-D, a formula was presented and an area was determined for each coefficient.

## 2. FLOW3D and Governing Equations of Fluid Dynamics

FLOW 3D 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, FLOW 3D 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. In FLOW 3D, two methods have been simultaneously used for geometrical simulation. The first method is Volume of Fluid (VOF) which is used to show the behavior 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. A flow continuity equation is obtained from the law of conservation of mass and writing balance equation for a fluid element. General continuity equation is presented as Equation (1).

$$V_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + \frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) = 0 \quad (1)$$

Where  $V_f$  indicates the fraction of open volume to flow;  $\rho$  indicates fluid density; (u, v, w) indicate velocity components in the directions of (x, y, z);  $A_x$  indicates the fraction of open level in x direction;  $A_y$  and  $A_z$  indicate the fraction of open level in y and z directions.

Fluid movement equations with velocity components of (u, v, w) in three different directions, i.e. Navier-Stokes equations are presented as following:

$$\begin{aligned} \frac{\partial u}{\partial t} + \frac{1}{V_F} \left( u A_x \frac{\partial u}{\partial x} + v A_y \frac{\partial u}{\partial y} + w A_z \frac{\partial u}{\partial z} \right) &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + G_x + f_x \\ \frac{\partial v}{\partial t} + \frac{1}{V_F} \left( u A_x \frac{\partial v}{\partial x} + v A_y \frac{\partial v}{\partial y} + w A_z \frac{\partial v}{\partial z} \right) &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + G_y + f_y \\ \frac{\partial w}{\partial t} + \frac{1}{V_F} \left( u A_x \frac{\partial w}{\partial x} + v A_y \frac{\partial w}{\partial y} + w A_z \frac{\partial w}{\partial z} \right) &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + G_z + f_z \end{aligned} \quad (2)$$

In these equations,  $(G_x, G_y, G_z)$  indicates mass acceleration and  $(f_x, f_y, f_z)$  indicate viscosity accelerations.

In FLOW 3D, to simulate transfer, erosion, deposition, and change of sediments establishment status is due to fluid flow. Sediment model of this numerical model uses two concentration fields including suspended sediments and bed sediments. Displacement of suspended sediments with fluid is due to local pressure's gradient changes. These suspended sediments may be created due to input flow containing suspended particles or due to bed erosion. Since bed erosions have been limited by neighboring particles and are not easily displaced, they can move only in case of changing into suspended load in shared level of bed and fluid. Suspended load can be changed into load when the velocity of depositing is higher than the velocity of bed erosion. A part of control volume occupied by solid particles of sediment is  $(f_s)$  and the rest is defined from accumulated fluid of  $(f_L)$  such that:

$$f_s + f_L = 1 \quad (3)$$

Suspended load causes to the increase of real fluid viscosity. This increase continues until solid particles' volumetric element reaches to volumetric cohesion element. After that, increasing suspended load does not lead to the increase of viscosity but it causes that particles start to behave in solid manner. In this state, average viscosity of fluid is computed from the following relation:

$$\mu^* = \mu_f \left( 1 - \frac{\text{Min}(f_s, f_{sco})}{f_{sCR}} \right) \quad (4)$$

Where  $\mu_f$  indicates fluid's viscosity and  $\mu^*$  indicates average viscosity of sediment particles' critical element. Apparent density of  $\bar{\rho}$  is assumed as linear function of sediments volume where  $\rho_s$  and  $\rho_L$  indicate apparent density of sediment and fluid, respectively.

Drift refers to the deposition of sediment particles under the impact of floating forces affecting sediment

particle. In the model of sediment washing in FLOW 3-D, sediment particles are assumed in spherical shape such that it is influenced by fluid's viscosity effect. Therefore, deposition coefficient is automatically computed according to the following relation:

$$\frac{50 \times (\rho_s - \rho_L)}{18} \quad (5)$$

Therefore, deposition velocity is computed through the following relation:

$$u_{\text{drift}} = D_f \times f_L \frac{\nabla P}{\rho} = \frac{f_L \times d_{50}^2}{18\mu} \frac{\nabla P}{\rho} (\rho_s - \rho_L) \quad (6)$$

Where in the above equation,  $\left(\frac{\nabla P}{\rho}\right)$  indicates mechanical potential of gradient or acceleration and is limited to 10 times more than particle's weight and causes to omit numerical fluctuations in pressure value. Near to fluid's free level, the value of  $\left(\frac{\nabla P}{\rho}\right)$  is replaced with acceleration ( $g$ ). The coefficient of  $f_L$  has been employed in the above equation since sedimentation is possible only at the presence of solid particles (sediment). Therefore, if control volume is full of sediments,  $f_L = 0$  and then,  $u_{\text{drift}} = 0$ .

At the level of bed sediments, shearing stress is active and causes to erosion and displacement of sediment at bed sediment level. This erosion is a function of fluid's shear stress at surface, critical shearing stress and sediment and fluid's density. The parameter of critical shields indicates the minimum shear stress required to lift sediment particles from the shared surface of fluid and active bed.

$$\theta_{\text{cirt}} = \frac{\tau_{\text{crit}}}{g(\rho_L - \rho_s)d} \quad (7)$$

Where  $\theta_{\text{cirt}}$  indicates critical shields;  $\Gamma_{\text{cirt}}$  indicates the minimum shear stress necessary for bed length to lift sediment particles. The purpose of developing and explaining this model is to estimate and predict the magnitude of sedimental flow which has been worn out. To this end, the parameter of shear velocity is defined to measure removal power of flow. So, the velocity of removing sediments from bed can be presented through the following equation:

$$U_{\text{lift}} = \alpha n_s \sqrt{\frac{\tau - \tau_{\text{crit}}}{\bar{\rho}}} \quad (8)$$

Where  $n_s$  indicates normal vector of bed surface;  $\alpha$  indicates dimensionless parameter indicating the probability of sediment particles removal from bed

(usually 1). In static fluid, internal friction angle of sediment particles determine the minimum slope through which sediments' walls can be steady. Internal friction angle above sediments indicates steady wall slope in steep slopes such as clay. In low angles of walls, there is a strong intention to collapse and move forwards such as sand.

In downstream hole where sediments are compiled together and create a mass of sediments, sediments establishment status makes an angle with horizon surface which indicates internal friction angle. In the model, this angle is signified by  $\xi$ . Natural establishment angle of sediments in various temporal and spatial conditions is computed through the following relation:

$$\varphi = \frac{n_{\text{interface}} \cdot g}{|g|} \quad (9)$$

Where  $n_{\text{interface}}$  equals normal vector of surface and  $g$  indicates gravity acceleration. After scour or transferring sediments suspended in the surface, critical shear stress occurring in sloped surface for each surface is computed through the following relation:

$$\tau_{\text{crit}} = \tau_{\text{crit}}^0 \sqrt{1 - \frac{\sin^2 \varphi}{\sin^2 \xi}} \quad (10)$$

According to the above equations, when natural slope of sediments equals their internal friction ( $\varphi = \xi$ ), critical shear stress equals zero, indicating that bed surface undergoes erosion due to every kind of imposed shearing stress. Also, when ( $\varphi > \xi$ ), we have ( $\tau_{\text{crit}} < 0$ ), indicating that higher internal friction angel of sediment particles leads to higher wall slope ( $\varphi$ ) since the wall of scour or sediment washing hole undergoes erosion without shear stress ( $\tau_{\text{crit}} = 0$ ). The movement of suspended sediments in system is expressed through convection-diffusion equation.

$$U_j \frac{\partial c_s}{\partial x_j} - \omega_s \left( \frac{\partial c_s}{\partial z} \right) = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial c_s}{\partial x_j} \right) \quad (11)$$

Where  $C_s$  indicates sediments' concentration,  $\Gamma$  indicates diffusion coefficient and  $\omega_s$  indicates particles' collapse velocity which equals:

$$\omega_s = u_{\text{lift}} - u_{\text{drift}} \quad (12)$$

Therefore, the above equation is entered into problem solving as following:

$$\frac{\partial c_s}{\partial t} + u \cdot \nabla c_s = \Gamma \nabla^2 c_s - u_{\text{lift}} \cdot \nabla c_s - u_{\text{drift}} \cdot \nabla c_s \quad (13)$$

The concentration of sediments suspended in shared surface of sediments' bed and water before starting sediment washing ( $t = 0$ ) equals:

$$C_{s0} = f_s \cdot \rho \quad (14)$$

The above computational equations and algorithm in FLOW 3D are used to discharge bed sediments.

### 3. Geometrical Simulation of Channel and Side and Normal Weirs

To simulate the geometry of hydraulic model of channel and normal and side weirs in FLOW 3D, geometrical simulation software such as AutoCAD, CATIA or SolidWorks should be constructed. To this end, SolidWorks 2011 software has been used in this research.

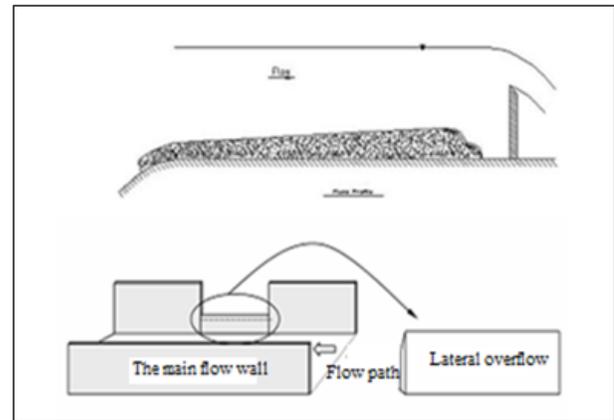
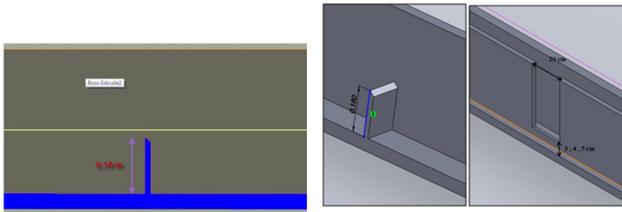


Figure 1. A schematic of channel and normal and side weirs in laboratory studies.

SolidWorks software is powerful software of design, particularly three-dimensional design. In this software, there are separate parts for modeling, sheet work, welding, casting, and molding operations as well as analyzing stress and modeling the behavior and resistance of piece under various loadings. Compared to similar soft wares, the outstanding feature of this software is to possess a simple and easy environment. In spites of simple and easy application, this software has a considerable ability in designing. Accordingly, SolidWorks 2011 software has been used to simulate channel and side and normal weirs. The model constructed in numerical software is based on laboratory; at all stages, the model simulation is also based on laboratory results.

The model investigated in the present study has been depicted in SolidWorks software (Figure 2).



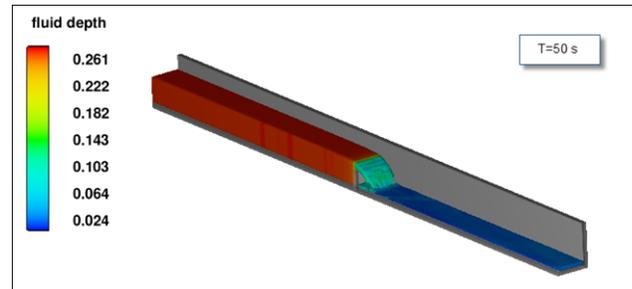
**Figure 2.** Geometrical simulation of overflow using SolidWorks.

#### 4. Numerical Modeling of Flow in Normal Overflows with/without Suspended Load using FLOW 3D

The present numerical model is performed in two separate stages with side and normal weirs. At the first stage with normal overflow, in a condition with a tilting laboratory flume with the width of 30cm, the height of 45 cm, the length of 10 cm, the maximum discharge of 12.5 l/s and the slope of 0.001 has been used. Sharp-edge rectangular overflow is placed parallel with the width of flume without compression with flume in the middle part of the flume. In previous section, the status of the sharp-edge rectangular overflow has been observed. At the first stage, to investigate the amount of discharge coefficient of normal overflow, a sedimental substance such as laboratory sediment substance is used. For normal overflow, sawdust with  $D_{50} = 0.192$  mm and  $G_s = 1.3$  is used. According to laboratory report, the most appropriate substance which can supply flow suspended load is sawdust which is accumulated in normal overflow of sediments behind the overflow.

**Table 1.** Comparing the laboratory and numerical model results in clear-water flow and flow with suspended load states

yd/p	Cd Exp		Cd Flow 3D		Error %	
	Clear-Water Flow	Flow With Suspended Load	Clear-Water Flow	Flow With Suspended Load	Clear-Water Flow	Flow With Suspended Load
0.150	0.651	0.667	0.642	0.672	-1.3	0.7
0.200	0.657	0.668	0.658	0.680	0.2	1.8
0.250	0.645	0.666	0.649	0.662	0.6	-0.6
0.300	0.655	0.664	0.656	0.670	0.2	0.9
0.350	0.652	0.672	0.667	0.647	2.3	-3.7
0.400	0.658	0.660	0.651	0.656	-1.0	-0.6
0.450	0.659	0.650	0.663	0.661	0.6	1.7



**Figure 3.** Steady-state conditions of flow in sharp-edge overflow channel at  $T = 50$ s.

To extract accurate and precise values of a numerical or laboratory model data, it is necessary to reach steady-state conditions. In the studied numerical model, fifty-step computational model has been considered after investigating several models, appropriate time to extract results (the end of computations with numerical model).

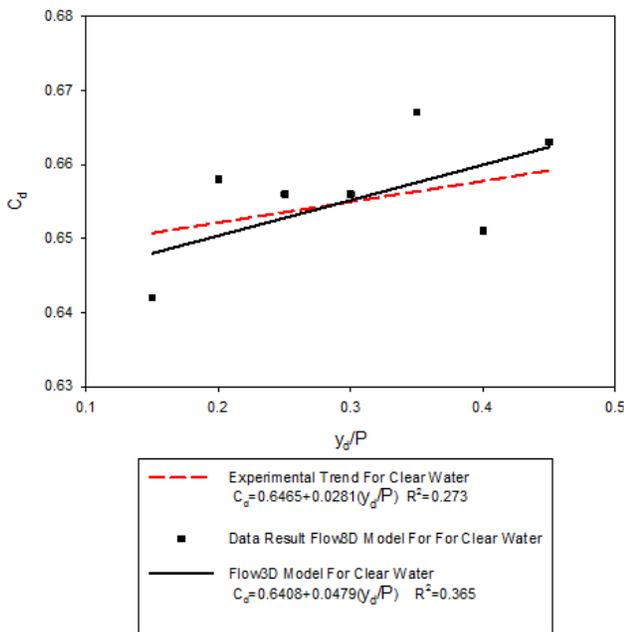
In the following table, the values of  $C_d$  presented in the study conducted by Gohari Asadi et al.<sup>17</sup> have been compared and evaluated using the numerical model results. Table 1 shows the conditions of two states of clear-water flow and flow with suspended load.

To investigate the trend of discharge coefficient changes in sharp-edge weirs under clear-water flow and flow with suspended load, the changes of discharge coefficient have been shown based on the change soft water height over overflow relative to overflow height. In the following figure, values of flow discharge coefficient for sharp-edge rectangular overflows for clear-water flow and flow with suspended load resulted by FLOW 3-D were compared with the laboratory results. In these diagrams, the dotted lines indicate the results pertained to the laboratory model and the fitness of data with the numerical model results has been shown with solid line. As shown in Table

1, the maximum simulation error for flow with suspended load is about 4%. With respect to the applied conditions for the numerical model and the calibration performed in various tests of the numerical model, simulation error between the numerical model and the physical model is acceptable. Based on the obtained fitness, flow discharge coefficient of  $C_d$  for clear-water flow with determination coefficient of  $R^2=0.365$  is presented as following:

$$c_d = 0.648 + 0.0479 \left( \frac{y_d}{P} \right) \quad R^2 = 0.365 \quad (15)$$

In this equation, discharge coefficient has been presented based on the height of water over overflow to weir height (P).



**Figure 4.** The changes of flow discharge coefficient in clear-water flow and comparing the results with numerical model.

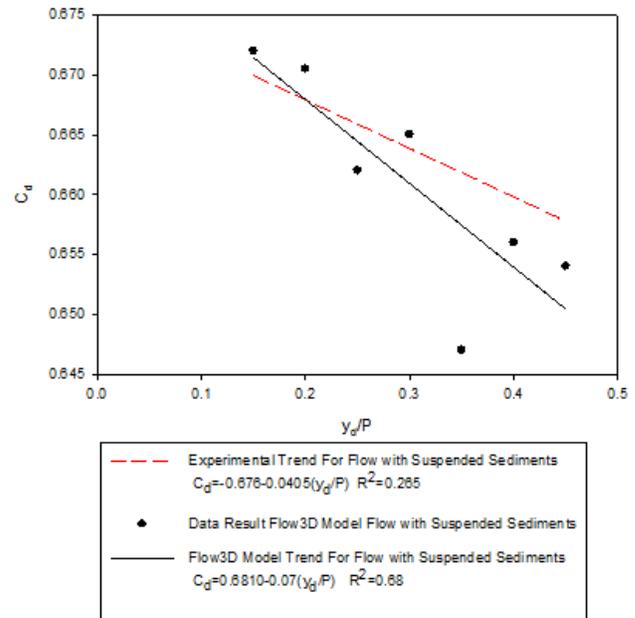
Based on the obtained fitness, flow discharge coefficient of  $C_d$  for flow with suspended load with determination coefficient of  $R^2 = 0.68$  is presented as following:

$$i_1(0), i_1(1), i_1(2), i_1(3), i_1(4), i_1(5), i_1(6), i_1(7,) \quad (16)$$

In this equation, discharge coefficient has been presented based on the height of water over overflow to overflow height (P).

As shown in the above figure, in clear water flow state, increasing the ratio of  $y_d/p$  leads to the increase of flow discharge coefficient; however, in flow with suspended

load, increasing the ratio of  $y_d/p$  leads to the decrease of flow discharge coefficient.

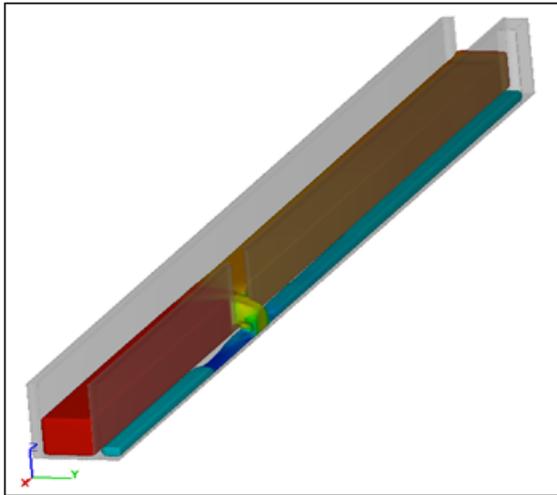


**Figure 5.** The changes of flow discharge coefficient in the state of flow with suspended load and comparing with the numerical model results (the average steady concentration in the numerical model 1115 ppm).

## 5. Numerical Modeling of Flow Discharge Coefficient Determination in Side Weirs with Various Concentrations of Suspended Load

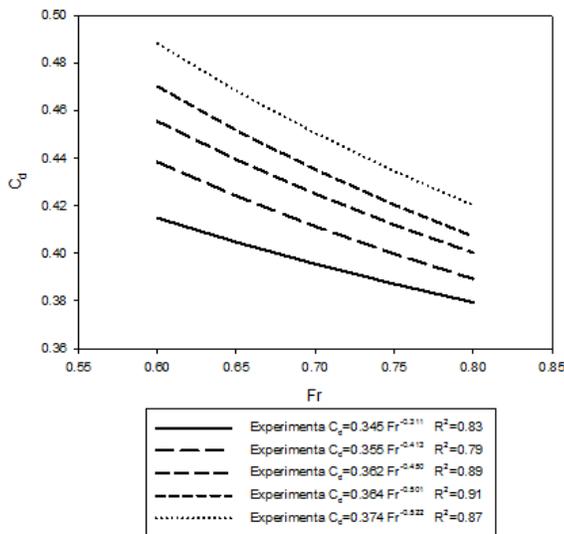
In flow state, in side weirs, in the employed laboratory model, flow channel has been divided into two parts of 20 cm and 10 cm and three overflows with the heights of 4, 5 and 6 cm have been installed at the one-third end part of flume. In this state, sediment substance is silica powder with  $G_s = 0.06$  and  $D_{50} = 2.65$  mm.

In this study, the concentrations required for the performed simulations for side weirs with various geometrical conditions equal 3000, 5700, 10500, and 19500 ppm. To precisely investigate flow discharge coefficient changes, Figure 6 shows the flow formed in side weirs. Accordingly, in the flow numerical model, before entering into overflow with various landing numbers, their discharge coefficient has been simulated for each of the three overflows with the heights of 4, 5 and 6 cm.

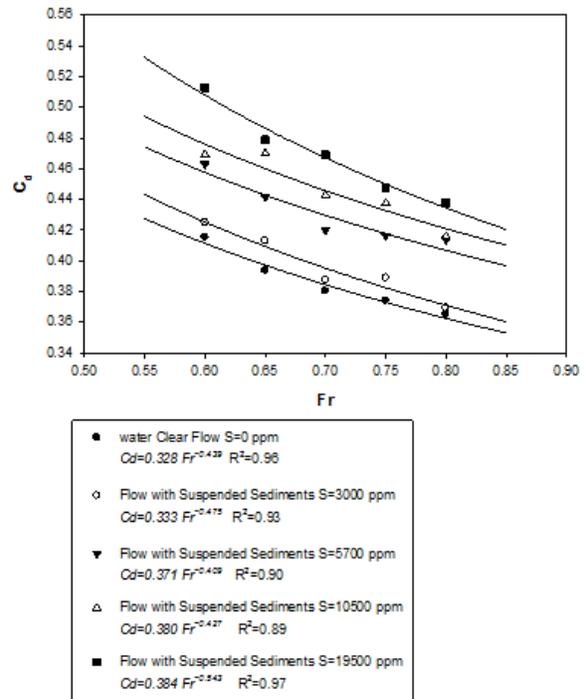


**Figure 6.** Flow formation in 3-D model of side weirs.

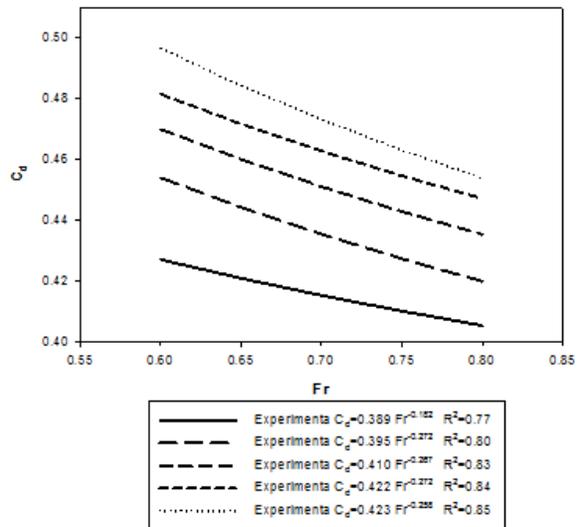
To precisely investigate flow discharge coefficient changes for each of overflows, the following figures show flow discharge coefficient changes based on flow landing number. In Figure 7, a clear-water flow state and flow with various suspended load (cm) have been depicted. At the first stage, the laboratory results have been shown for 4-cm overflow. Afterwards, the results related to the numerical model have been shown. In the numerical model results, determination coefficient has been simulated for each of concentrations under various concentrations (Figure 7).



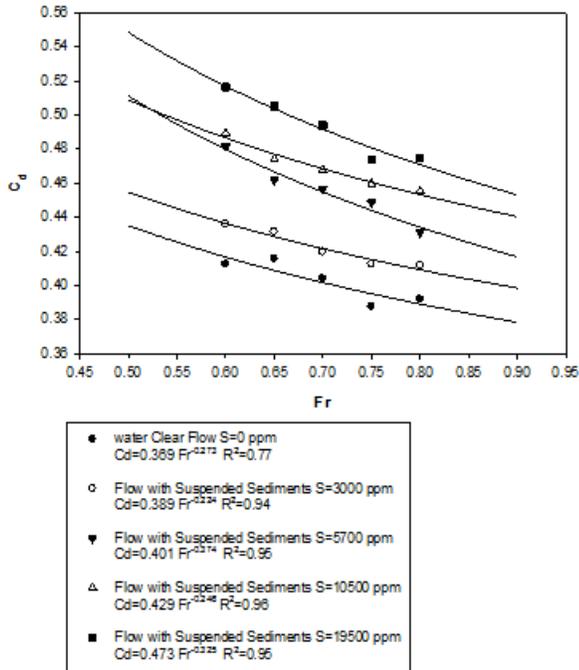
**Figure 7.** Laboratory results of flow discharge coefficient changes of  $C_d$  based on flow landing number changes for 4-cm overflow.



**Figure 8.** Numerical modeling results of flow discharge coefficient changes of  $C_d$  based on flow landing number changes for 4-cm overflow.



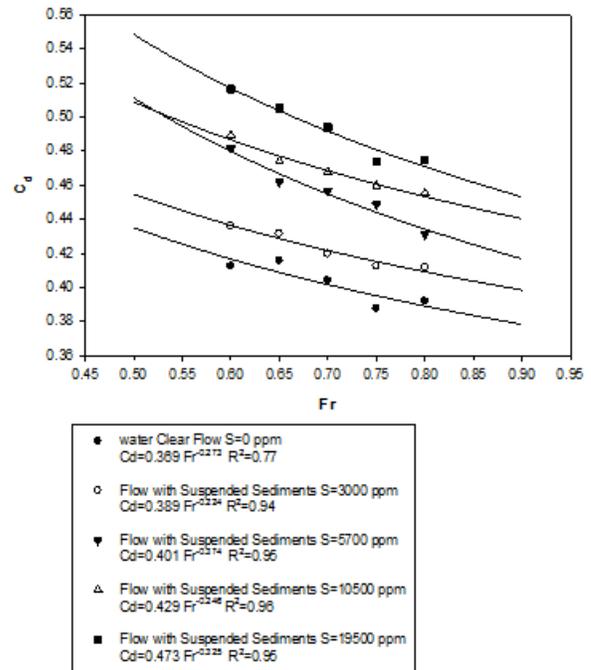
**Figure 9.** Laboratory results of flow discharge coefficient changes of  $C_d$  based on flow landing number changes for 5-cm overflow.



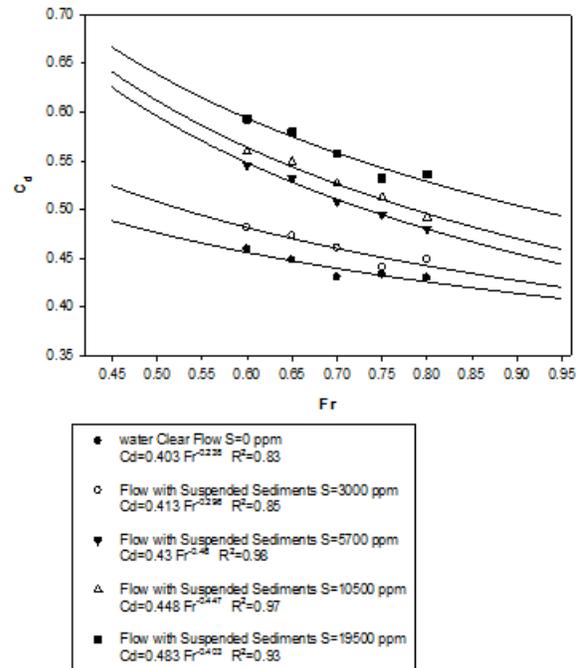
**Figure 10.** Numerical modeling results of flow discharge coefficient changes of  $C_d$  based on flow landing number changes for 5-cm overflow.

As shown in the figures, leading landing number in all the performed simulations leads to the decrease of  $C_d$  coefficient according to geometrical conditions of side weirs and various concentrations. On the other hand, in side weirs with various heights, it was proved that increasing the concentration of suspended load,  $C_d$  coefficient is increased. As observed in the numerical model results, based on flow landing number, the relation of  $C_d$  has a relatively better and higher determination coefficient compared to the laboratory results. It can be attributed to the lack of suspended load's concentration distribution in laboratory and difficult conditions of this action in laboratory modeling.

In the laboratory model, suspended load cannot be totally injected to the entire fluid with appropriate ratio; while in the numerical model, flow concentration can be easily added to the model ( $M^3$ ) through input boundary. This point is one of the advantages of the numerical model to simulate such problems.



**Figure 11.** Laboratory results of flow discharge coefficient changes of  $C_d$  based on flow landing number changes for 6-cm overflow.



**Figure 12.** Numerical modeling results of flow discharge coefficient changes of  $C_d$  based on flow landing number changes for 6-cm overflow.

At this stage, a general relation can be presented to change flow intensity coefficient with landing number and flow concentration based on ppm:

$$C_d = 0.590 - (0.243Fr) + (4.95 \times 10^{-6}.S) \quad R^2 = 0.55 \quad (17)$$

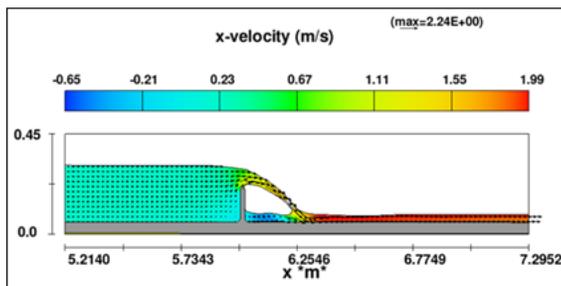
As shown in the above equations, the general relation of discharge coefficient is presented in numerical modeling based on concentration and landing number with determination coefficient of 0.55. Comparing the above relations with the relations of the laboratory results presented by Gohari<sup>17</sup> indicates a 4% better determination coefficient (determination coefficient of the laboratory studies is 0.32).

Flow discharge coefficient can be stated based on relative depth of flow ( $y_d/P$ ). Accordingly, computing relative depth values relative to side weirs height as well as flow landing number and flow concentration, a general relation can be written as following:

$$c_d = 0.525 - \left( \frac{0.0706y_d}{P} \right) + (4.95 \times 10^{-6}.S) \quad R^2 = 0.905 \quad (18)$$

As observed in the above equation, based on flow concentration and relative height of flow entering channel, discharge coefficient relation has an appropriate determination coefficient and its magnitude is higher than laboratory magnitude.

Additionally, flow discharge coefficient ( $C_d$ ) can be presented in a combination form of landing number value ( $Fr$ ), flow concentration ( $S$ ) and relative flow depth ( $Y_d/P$ ) in side weirs as following:

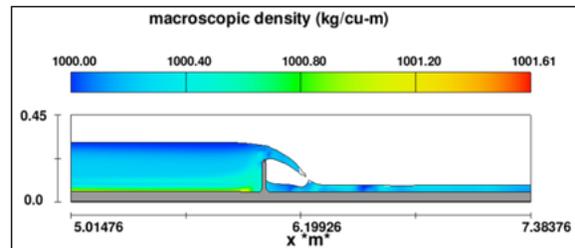


**Figure 13.** Flow velocity values with velocity vectors in longitudinal direction of channel in normal overflows.

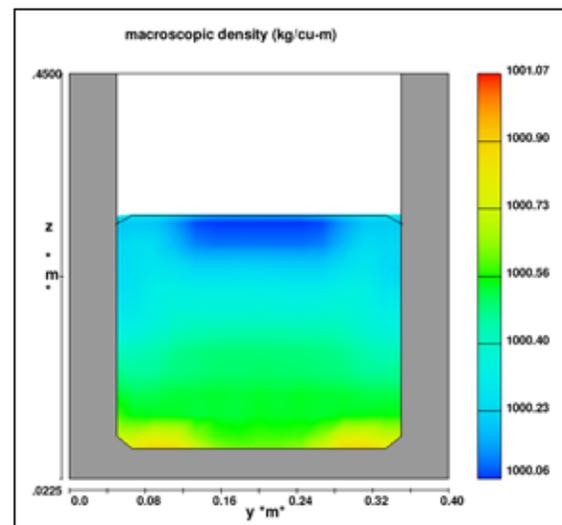
$$c_d = 0.555 - (0.11Fr) - \left( \frac{0.04y_d}{P} \right) + (4.95 \times 10^{-6}.S) \quad R^2 = 0.94 \quad (19)$$

The above equation covers those variables considered in the laboratory model. As presented Equation 19, determination coefficient of this relation equals 0.94 which is a better value compared to the laboratory results.

In the following figures, flow velocity has been shown in longitudinal direction. Negative velocity values are normal due to the formation of rotating flow in normal overflow downstream.



**Figure 14.** Flow density changes behind normal overflow and suspended load values in the numerical model.



**Figure 15.** Density changes in cross-section before normal overflow.

According to the laboratory report, flow suspended load has been accumulated behind normal overflow. Similar to the laboratory model, in the numerical model, flow suspended load is accumulated behind normal overflow and appeared as a result of its density change. Notably, the maximum amount of flow density for sawdust suspended load equals 1001.07 kg/m<sup>3</sup>, which has been extracted from the numerical model.

## 6. Conclusion

Weirs and gates are structures which are used to measure water flow in open channels. Entering rivers' sediments into channels is considered as one of the main problems in irrigation and water transmission networks. The relations presented based on laboratory results for gates, overflows and apertures' discharge coefficients have been extracted and developed based on the conditions of sediment free water flow. Therefore, applying these relations in natural conditions requires investigating the effect of suspended load on hydraulic coefficients which are highly effective to technically and economically design and select these structures. The purpose of the present paper was to investigate the effect of suspended load in hydraulic conditions of rectangular sharp-edge, normal and side weirs. Using numerical simulations designed and performed according to the laboratory model, overflow's discharge coefficients were measured and compared in clear-water flow and flow with suspended load states.

- Comparing the laboratory and numerical results in calibration and validation process of the numerical model revealed that the maximum simulation error for flow with suspended load is about 4% and the selected model has appropriate boundary, primary and grid generation conditions.
- Comparing discharge coefficient results in side and normal weirs revealed that these coefficients are different in clear-water flow and flow with suspended load states.
- According to the numerical modeling results, in side and normal weirs, discharge coefficient in suspension flow conditions under identical water load is higher than clear-water state.
- Increasing landing number in all the simulations leads to the decrease of  $C_d$  coefficient based on all geometrical conditions and various concentrations.
- According to the numerical modeling results, increasing flow concentration in side weirs leads to the increase of flow discharge coefficient.
- The maximum concentration of flow with suspended load for sediment substance of sawdust in normal overflows is  $1001.07 \text{ kg/m}^3$  and for side weirs with suspended load, silica powder is  $1024.3 \text{ kg/m}^3$ .
- Based on the laboratory results, flow intensity coefficient is decreased by increasing relative depth of flow intensity coefficient values.
- In the numerical simulation, the relation of  $C_d$  based on flow landing number has a relatively better and

higher determination coefficient compared to the laboratory results. It can be attributed to the lack of suspended load thickness distribution in laboratory and difficult conditions of such action in laboratory modeling. In laboratory model, suspended load cannot be fully injected with appropriate ratio to the entire fluid. However, in numerical model, flow thickness amount can be easily added and injected to the model from input boundary on cubic meter unit of flow. This significant point is considered as one of the advantages of numerical model to simulate such problems.

## 7. References

1. Razavian SH, Heydarpour M. Investigating discharge coefficient in combinational model of overflow-sharp-edge gate. The articles collection of the 6th hydraulic civil conference; Sharhrekord. 2007. p. 390–8.
2. Burton M, et al. Etna 2004–05: An archetype for geodynamically controlled effusive eruptions. *Geophysical Research Letters*. 2005 May; 32(9).
3. Prabhata SK, Pathak SK, Masoud Ali S. Side-weir analysis using elementary discharge coefficient. *Journal of Irrigation and Drainage Engineering*. 1994; 120(4):742–55.
4. Borghei SM, Jalili MR, Ghodsian MASOUD. Discharge coefficient for sharp-crested side weir in subcritical flow. *Journal of Hydraulic Engineering*. 1999; 125(10):1051–6.
5. Amir Asalani SH, Pirestani MR, Salehi Neyshabouri A. Numerical investigation of the effect of internal friction angle of sediments on the size of scour hole due to free falling jet. 2nd National Conference on Dam and Electro-Water Power Plants; Tehran. 2008. p. 1–9.
6. Soltani M, Rahimi Asl R. *Computational Fluids Dynamic through FLUENT Software*. 4th edition. Tehran: Tarah Publication; 2007.
7. Javareshkian MH. Secondary flow in a 90 degree bend having rectangular cross-section. Thesis of Master of Science (Engineering). The University of Liverpool;
8. Wersing HK, Malalaskra V. An Introduction on Computational Fluids Dynamic. 2nd edn. In editor: Shojaei Fard MH and Nourpour Hashtroudi ER. Tehran: Science and Industry University Publication;
9. Olsen NRB. CFD algorithms for hydraulic engineering, department of hydraulic and environmental engineering. The Norwegian University of Science and Technology; 2000 Dec.
10. Etesam Saeij E. Numerical investigation of the flow of water passing in basin-edge overflow using Volume of Fluid method (VOF). MA thesis. River engineering, Sistan and Balouchestan, Zahedan. Iran. 2009.
11. Wilcox DC. *Turbulence Modeling for CFD*. 1st edn. California: DCW Industries, Inc; 1993
12. Yakhot Y, Orszang SA. Renormalization Group Analysis of Turbulence. *Journal of Scientific Computing*. 1986.

13. Azimian A. Computational fluids dynamic. Isfahan University Publication; 1998.
14. Afshar MH. Approximation process and finite element method. Science and Industry University Publication; 1996.
15. Jafari A. Investigating permeability coefficient of labyrinth overflows using CFD and optimizing overflow geometry using fuzzy neural model and genetic algorithm. MA thesis of hydraulic structures. Sistan and Balouchestan. 2010.
16. Amir Asalani SH, Pirestani MR, Salehi Neyshabouri A. Numerical investigation of the effect of internal friction angel of sediments on the size of scour hole due to free falling jet. 4th national conference on dam and electro-water power plants; Tehran. 2008. p. 1–9.
17. Gohari Asadi S, Ayoubzade A, Samani MV, Aberi Foroutan SH. Comparing discharge coefficients of normal and lateral overflows with flow suspended load. 5th Hydraulic Conference of Iran; Kerman. Bahonar University; 2005.
18. Yakhot V, Orszag SA, Thangam S, Gatski TB, Speziale CG. Develpoment of Turbulence Model for Shear Flow by a Double Expansion Technique. *Physics of Fluids A*. 1992; 4(7): 1510–20.
19. Sabagh Yazdi SR, Shamlou H, Rostami F. Educational application of FLOW 3-D Software to investigate turbulence simulation of 3-d flow in hydraulic jump pool with concrete barriers. *Technology and Education Magazine*. 2008; 2(4).