Three-dimensional modelling of flow, over the cylindrical weir using Flow-3D software

Saeed farzin [1], Hojat karami [1], Elham zamiri [2], Shahab nayyer [2]*
1. *Assistant professor, Department of civil engineering, Semnan university, Semnam, Iran*
2. *M.S. student, Department of civil engineering, Semnan university, Semnam, Iran*
** Corresponding author E-mail:* saeed.farzin@semnan.ac.ir

Abstract

Cylindrical weir is one of the hydraulic structures model that due to simple design, ease of floaters passage and sustainable pattern of flow is widely applied. In this article, flow over the cylindrical weir with 7.5 (cm) diameter on flume by 6 (m) length was modelled in Flow-3D software as three-dimensional and the results were compared with the physical model, also the results of three turbulence models LES, RNG and k-Ɛ were compared and it was found that the results of K-ε turbulence model best fit the results of the physical model. Root Mean square Error of comparing the numerical discharge coefficient with physical quantities was obtained 0.3427, which represents a good agreement hydraulic characteristics obtained from numerical and physical model, it was also observed, cylindrical weir discharge coefficient values is greater than 1.

Keywords: Discharge coefficient, cylindrical weir modelling, Flow-3D, k-ε Turbulence model.

Introduction

weirs are one of important hydraulic structures that generally are used to control the level of water in the upstream or accurate measurement of flow. Overpass or weir structures,depending on local conditions and hydraulic properties can be designed in different ways. ogee weir and broad crested weir that second type use in short structures, usually in dam body are used.Use of the cylindrical weir end of the 19th century before creation of ogee weir became common [1]. Convexity of cylindrical weir wall causes formation of curved flow lines, and therefore makes it more economical, easier to design and build this type of overflow,and higher discharge coefficient in compare with rectangular, triangular and other overflows [2]. other uses of this weir types are rolling gates and rubber dam which at high flows is more economical [3]. Flow3D is one of very powerful models in the field of fluid dynamics models, which includes physical patterns such as turbulence, porous environments, scouring and so on.this software is able to analyses 2D and 3D fluid field in the form of volume, and uses three-dimensional orthogonal elements.

Johnson and Montes (1998) examined the behavior of the flow on cylindrical weir,Their test results showed that by increasing total load of upstream, discharge coefficient increases [4].

Chonrong, L., A. Huhe, and M. Wenju, [2002] studied the flow on circular weir numerically and experimentally. Modeling was done by using k-Ɛ turbulence and VOF method [5].

In our country, Iran, Farsadi zadeh and associates (1387) studied the status of flow on the physical model of the cylindrical weir and compared the results with the results of Fluent numerical model simulation and found out that by increasing the load of water in upstream, discharge coefficient increases [6]. zahiri and associates (1391), simulated the flow on water channel and rectangular broad crest side channel spillway by Flow3D, and compared it with experimental results. This simulation was done by two turbulence models such as standard k-Ɛ and RNG k-Ɛ. The results indicated that RNG k-Ɛ model has best compliance with the experimental flow rate and weir hydraulic conductivity [7]. Heidar pour and associates (1391) simulated the passing flow over the trapezoidal gated over flow in circular channel with Flow3D, and results show that Flow3D softwares, estimated the surface profile without surface tension in good accuracy[8].

According to the tests, few studies have been done on weir cylindrical modeling with Flow3D softwares. so in this study try to compare discharge coefficient of flow on cylindrical weir by using k-Ɛ turbulence model in Flow3D with physical model to be determined ability of this model in simulation of flows.

Materials and Methods

Theory of flow in the cylindrical weir is under the influence of kinematic, dynamic and geometric parameters and characteristics,Which is defined by equation (1).

$$f\left(H, H_w, B, \mu, g, s, Q, R, \rho, \sigma, \delta\right) = 0 \tag{1}$$

Which in that, (B) is Channel width, (g) gravity, (μ) Dynamic viscosity, (s) Slope of the channel bottom, (ρ) fluid density, (σ) Surface tension, (δ) Compressibility factor, (R) Radius of the cylinder, (H) Water depth of upstream and (H_w) Water depth on weir.

Solutions network in Flow3D models consists of a rectangular cube cells. using this solutions network because of the easy production of network, the proper order and less memory requires in that and secondly due to the use of two useful tools like VOF[1] and Favor[2] in Flow3D models, will be very useful. VOF and Favor are example of volume of fluid methods. VOF method is used when the flow surface is free [9]. All the cells aren't full of fluid, and some of them in the fluid surface are half-full. Favor method is one of the volume of fluid ways,that using for simulate of surface and rigid volume such as geometric borders [10].

Mass continuity equation,generally is in the form of equation (2).

$$v_f \frac{\partial p}{\partial t} + \frac{\partial}{\partial x}\left(\rho u A_x\right) + R\frac{\partial}{\partial y}\left(\rho v A_y\right) + \frac{\partial}{\partial z}\left(\rho w A_z\right) + \zeta \frac{\rho u A_x}{x} = R_{DIF} + R_{SOR} \tag{2}$$

Which in that, (v_f) volume fraction of fluid,(ρ) Fluid density,(R_{DIF})turbulence distribution phrase,(R_{SOR})mass source. Speed components(u,v,w) are in (x,y,z) directions. A_x , A_y و A_z are Surface fraction of fluid in x,y,z directions and R Coefficient is related to Coordinate system. Corrected continuity equation comes in the form of equation (3).

---

[1] Volume of Fluid

[2] Fractional Area-Volume Obstacle Representation

$$\frac{v_f}{\rho c^2}\frac{\partial uA_x}{\partial t}+\frac{\partial uA_x}{\partial x}+R\frac{\partial vA_y}{\partial y}+\frac{\partial wA_z}{\partial z}+\xi\frac{UA_x}{x}=\frac{R_{sor}}{\rho} \quad (3)$$

K-ε turbulence equation including transport equation for kinetic energy and its dissipation. Transport equation for the dissipation of turbulence, ƐT according to Equation (4) is defined.

$$\frac{\partial \varepsilon_T}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial \varepsilon_T}{\partial x}+vA_y\frac{\partial \varepsilon_T}{\partial y}+wA_z\frac{\partial \varepsilon_T}{\partial z}\right)= \frac{CDIS1.\varepsilon_T}{K_T}\left(P_T+CDIS3.G_T\right)+Diff_\varepsilon-CDIS2\frac{\varepsilon^2{}_T}{K_T} \quad (4)$$

In equation (4), CDIS1, CDIS2 and CDIS3 are dimensionless parameters that their values are assigned by the user and for $k-\varepsilon$ model in respectively have 1.44, 1.92, 0.2 default values. Cylindrical weir is a kind of weir, so coefficient discharge in this weir as same as broad crested overflow can be achieved from Boss (1978) equation (5).

$$C_d=\frac{q}{g^{0.5}\left(\frac{2H_1}{3}\right)^{1.5}} \quad (5)$$

In equations (5), (Cd) is flow discharge coefficient, (g) Acceleration of gravity, (q) flow discharge of overflow unit width and (H1) water level in upstream.

Root Mean square Error can be derived from the equation (6) and Mean Absolute Error can be derived form the equation (7).

$$RMSE=\sqrt{\frac{\sum_{i=1}^{N}(O-P)^2}{N}} \quad (6)$$

$$MAE=\frac{1}{N}\sum_{i=1}^{N}\left|O-P\right| \quad (7)$$

In the equations (6) and (7), (N) is the number of data, (O) experimental values and (P) numerical values.

Results and Discussion

The requirement for building a structure across a river to provide necessary head for adequate delivery of water has been visualized by engineers for many centuries. A weir structure is an essential feature of many hydraulic structures such as dames, barrages, canal drops or falls regulators, cross regulators, etc.one of the hydraulic structures model that due to simple design, ease of floaters passage and sustainable pattern of flow is widely applied is Cylindrical weir.

In this research, flow numerical models on cylindrical weir was simulated and results were compared with experimental results of Ghareh Gezlo (1391) [11]. Laboratory model for flow level in upstream weir and for free surface flow have been done.

Simulations have been done in 7 different discharges and with k-ε turbulence model by Flow3D. For numerical model used a flume with 6 m length and 7.5 cm width and 14 cm height. Cylinder with 75 mm diameter was placed in the channel and modeling was done for different discharge in the range of 0.5 to 2 liters per seconds. Figure 1 is showing the experimental model schematically.

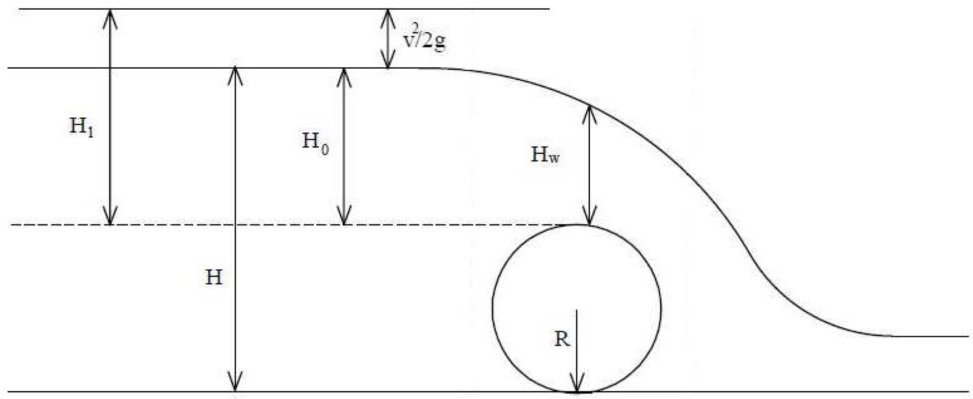

Fig. 1. Model simple shape.

Volume flow rate in boundary condition for x_{min} in simulation were used, and for z_{max} surface is the free surface of fluid and for the side wall and channels floor is Wall and for surface in x_{max} is outflow. the initial value of fluid depth at beginning of the modelling is in full mode. In Figure 2, three-dimensional view of the channel is displayed.

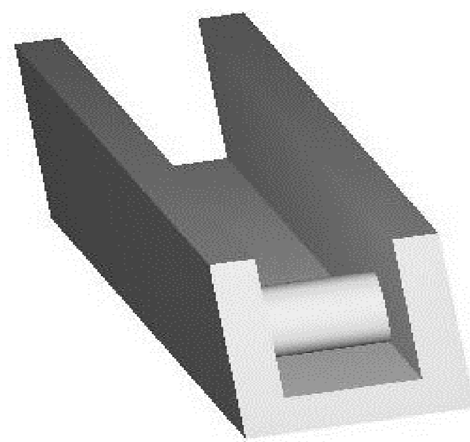

Fig. 2. Three-dimensional view of experimental channel.

In the simulation, were used 2 models with 300850 and 344448 meshes, According to the results and compared them with physical modelling, by reducing the number of mesh to 300850, error increases, so for simulation used 344448 number of mesh as optimal mesh.because of the importance of hydraulic characteristics in the z direction compare to the other directions, smaller mesh were used in this direction.the error values of discharge , discharge coefficient and (H_w/R) are calculated with RMSE, MAE and correlation coefficient (R^2) , the results for discharge is RMSE= 0.0755, MAE=0.05133 and R^2 =0.9872 and for discharge coefficient is RMSE=0.3427, MAE=0.3006 and R^2 =0.9367 and also for (H_w/R) RMSE=0.2417, MAE=0.2269 and R^2 =0.9053 is resulted.

In the cylindrical weir modelling used three model, turbulence models k-Ɛ, RNG, LES. The error obtained from the simulation with this turbulence models show that k-Ɛ turbulence

model, leads to better results and the output of numerical simulations with this model is more consistent with the physical results.
The results of the physical model compared with flow simulation in Flow3D model according to dimensionless parameters. Changes in discharge shows in figure 3 in physical model and the Flow3D model, which represents a high compliance of flow in both condition with R^2 = 0.9872 (Regression).
Then, in figure (4) changes of the (H_w/R) is shown in physical model and Flow3D model. Regression number in this situation is 0.9053, that it is near to number one, so it represents high compliance between physical model and Flow3D models. the relationship between the entrance discharge to the flume and the discharge coefficient shown in figure (5) and indicates that the linear relationship between these two parameters.

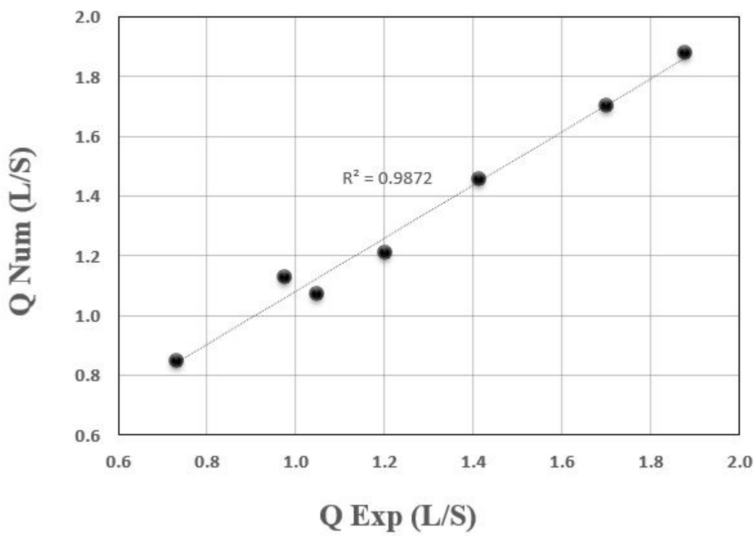

Fig. 3. Experimental and numerical discharge comparison

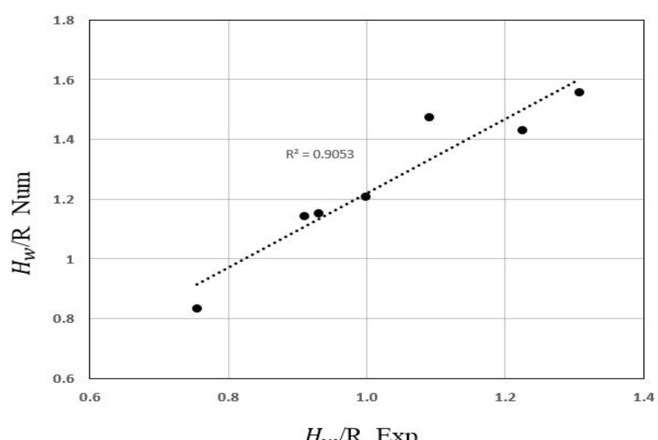

Fig. 4. Experimental and numerical (H_w/R) comparison.

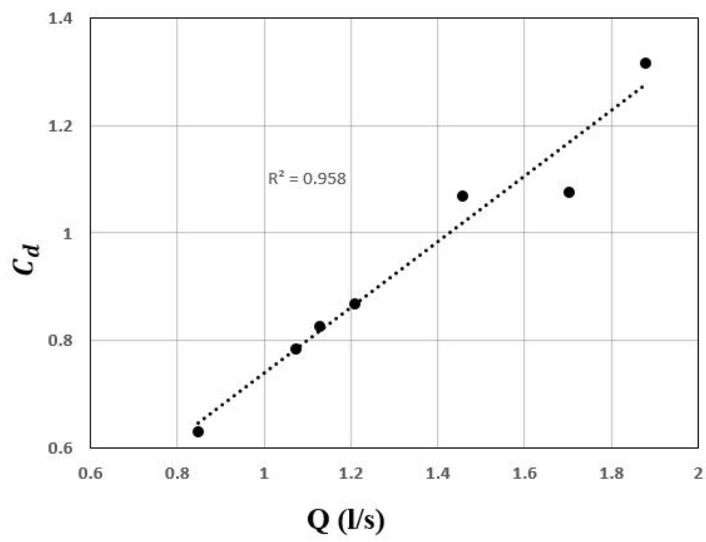

Fig. 5. Linear relationship between the entrance flow and flow coefficient.

Flow coefficient of physical model was compared with numerical model in figure(6), and the regression number is 0.9367 that is shown there are high conformity between numerical and experimental data, also the discharge coefficient progressively becomes greater than one. Figure (7) shows changes in discharge coefficient in comparison with H_W/R, by observing this shape, it becomes clear that, with increasing water depth on weir ،discharge coefficient is increases too.

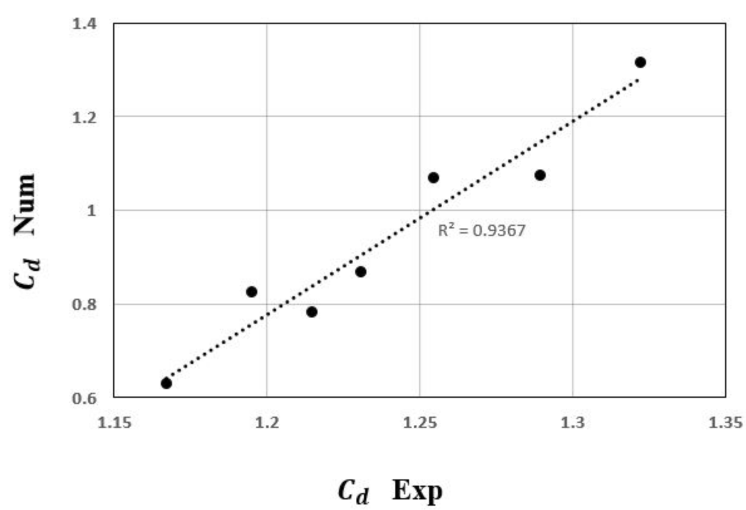

Fig. 6. Experimental and numerical flow coefficient comparison.

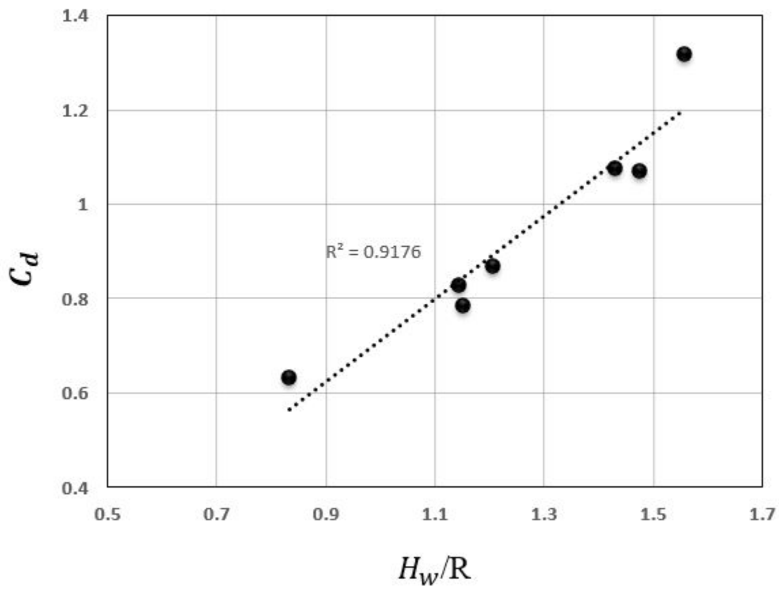

Fig. 7.Discharge coefficient changes compared to H_w/R.

Conclusion

Scrutiny the values of discharge coefficient shows that, C_d increased with increasing discharge and in cylindrical overflow its quantity tend to number one. between different models of turbulence [RNG, LES, k-ε] in the results of the k-ε turbulence model in Flow3D can be seen that there is a better conformity with the experimental model and with fewer errors. The results of the numerical model reveals that Discharge coefficient increases with increasing (H_W/R). therefore, discharge coefficint with increasing the overflow water depht increases. relative error of comparing the resultant discharge coefficient of the numerical model with physical quantities is 2.71 percent.the results indicate high conformity between numerical and physical model. Therefore, comparing the results of the physical model with Flow3D model, shows that the Flow3D has high accuracy.

References

[1] honar, t., and mazlom shahraki,S .(1393). "Cylindrical and half-cylindrical side weir discharge coefficient analysis in the under the crisis flow". Journal of Science and Technology of Agriculture and Natural Resources, Water and Soil Sciences, Eighteenth year, Number sixty-ninth,1393 Autumn.

[2] Severi,A., Masoudian, M,. kordi,A., and Ratcher, K. (1393). "experimental analysis of discharge coefficient changes and energy loss in the cylindrical overflow-valves with vertical movement". journal of civil and environmental engineering.vol 44,number 4.

[3] Abbaspour,A., and Hashemi kia, A .(1392). "flow numerical simulation on cylindrical overflow with considering the roughness component by using k-ε criterion" .Journal of water resources .sixth year.pages 87 to 97.

[4] Chason, H., and Montes, J. S. (1998). "Over flow characteristics of circular weirs: Effects of inflow conditions". J Irrig and Drain Engin, ASCE, 124: 152-161.

[5] Chonrong, L., Huhe, A., and Wenju, M. (2002). "Numerical and experimental investigation of flowover cylindrical weirs". Acta Mech. Sinica. 18: 23-31.

[6] varjavand, D., farsadizade, P,. khosravi, P., and rafeei, Z. (1389). "flow simulation on cylindrical overflow by using fluent model and compare the results with the physical model". journal of soil and water ,vol 20.1,number 2.
[7] ghanadan,R.,. zahiri, A,. And jalaledin M. S. (1391). "numerical simulation of wide edge side overflow with flow3D model".national conference of water and wastewater engineering.kerman, graduate university of industrial and advanced technology.
[8] pashazadeh, M., heidarpour, M., saghaeian nejad. H., and ghareh daghi,M. M. (1391). "overflow –sharp edged trapezoidal valve combined model to simulate flow at the bottom of the circular channel by flow3D" .Eleventh conference of hydraulics , urmia university, november.
[9] shojaeian, Z., and kashefi pour, S. M. (1390). "numerical simulation of hydraulic jump in Nimrod dam stilling basin".Journal of soil and water.vol 23,number 2,pages 283 to 295.
[10] shahrokhi, M. (1387). "Comparison of turbulence models in turbulent flows". Fourth National Congress of Civil Engineering, Tehran University.
[11] Ghareh gezlo, M. (1391). "Simultaneous laboratory evaluation of flow with overflow-cylindrical valve combined model". MS Thesis, Faculty of Agricultural Engineering, University of Agricultural Sciences and Natural Resources, Sari.