Periodicals of Engineering and Natural Sciences ISSN 2303-4521
Vol. 8, No. 2, June 2020, pp.809-820

CFD simulation model and experimental study to implement a new flowrate formula for a rounded broad crested weir considering the end depth as control section

Sadiq S. Muhsun[1], Shaymaa Abdul Muttaleb Al-Hashimi[2], Sanaa A. Talab Al-Osmy[3]
[1,2,3] Department of Water Resources Engineering, College of Engineering, Mustansiriyah University

ABSTRACT

Weirs can be considered as the major types of flow measurement structures which are implemented along open channels to represent a controlling section to estimate the quantity of flowrate. This study depended on considered the control section at the end edge of weir and relate the depth of water at this edge ***(Ye)*** as a function of the critical depth flow ***(D_c).*** Consequently, a laboratory study was conducted for ten experiments tests of open canal flow with ten different longitudinal slopes ranged from (0 to 0.0495) in order to estimate such a relationship. The statistical regression analysis results illustrated that the relationship for (***D_c with Y_e***) for all experiments is about 1.45831 as an average. As consequence, a new formula for predicting flowrate over weir was derived. Different statistical indexes were used to investigate the precision of the suggested formula where it appeared a very good agreement with all experimental data. A commotional fluid dynamic simulated model CFD with volume of fluid (VOF) method and (k-ε) turbulent models was also applied to verify the formula using FLUENT ANSYS ver. 16. The results indicated that the CFD techniques are able to simulate the flow over the weir and satisfactory the results of the suggested formula with less than 10% percentage error for all experimental tests.

Keywords: Broad crested Weir, critical depth, CFD, flowrate, ANSYS

Corresponding Author:

Sadiq S. Muhsun,
Department of Water Resources Engineering, College of Engineering, Mustansiriyah University
14150, Badhdad, Iraq
E-mail: dr_sadiq71@yahoo.com & sadiq.aljilizy@uomustansiriyah.edu.iq

1. Introduction

Weir is a common type of hydraulic structure constructed in order to regulate the water levels and it is a best device for flowrate measurement in the field or in the laboratory, [1]. It has a varied cross section design, such as broad or sharp-crested weirs including rectangular, triangular, and trapezoidal weirs, [2]. The broad-crested weir is a flat-crested structure has a length larger than the flow thickness, [3], [4], see Fig.1. The flow along the broad weir has parallel and horizontal streamlines and a hydrostatic pressure distribution. Based on the head in the control section upstream of the weir, the flow over the weir may be estimated as, [5]:

$$\boldsymbol{Q = B\, C_d \frac{2}{3}\sqrt{\frac{2}{3}g}\;\; H_1{}^{3/2}} \qquad (1)$$

Where Q is the flowrate $(\mathrm{m^3/s})$, $\mathrm{C_d}$ is the dimensionless coefficient of discharge, g is the gravitational acceleration$(\mathrm{m/s^2})$, H_1 is the head at the U/S of the weir and b is the width of the weir (m).
Several researchers investigated the properties of the flow across broad-crested weirs. Many researches considered the models of the CFD technique to predict the water profile over the broad crested weir comparing the results with that of experimental works, [6]. The results showed that the prediction of the U/S profile was excellent for smaller flowrate in contrast for the case of the supercritical flow, where a stationary wave profile was observed at D/S. The effect of flow streamlines with vertical curvature resulting from change crest height of weir on discharge coefficient (Cd) are examined experimentally. The study was applied on four types of weirs under different flow conditions. The results indicated that the relative (H/Y) crest height was affected on (Cd) in both broad crested and sharp weirs and can correlated by a formula using the regression analysis. A good agreement obtained from the comparative values of discharge coefficient resulted from estimated formulas and those formulas developed by other researchers, [7].

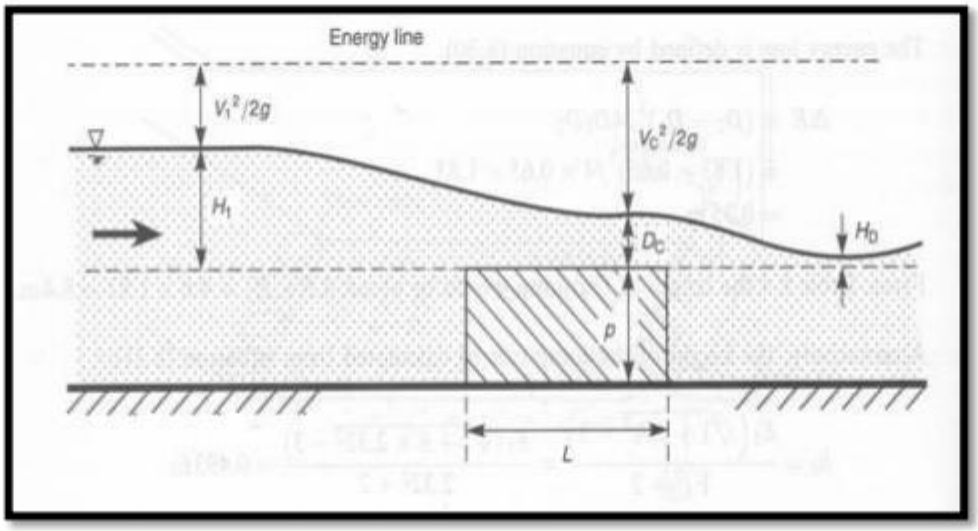


Figure 1. Rounded broad crested weir

Others presented the study on a huge-scale rounded corner of broad-crested weir experimentally. Using different flow range conditions, velocity, pressure measurements, and free surface were recorded. The results indicated that the rapid flow distribution occurred at large rates of flow at the next to the weir brink and at upstream end of weir. Also, they concluded that the critical flow conditions were appeared when x=L crest between 0.1 and 1 of weir crest while the velocity measurements showed a developing boundary layer, [8]. 3D simulated model were designed and operated to estimate the water level and streamlines over the broad crested weir, based on the volume of fluid (VOF) and three turbulent models of the standard k–ε, RNG k–ε and the large eddy simulation (LES). The results of the experimental test and the CFD model, indicted a good agreement in most cases, [9]. The overflow characteristics of semi-circular crested weir, was also investigated using twelve models fabricated and tested. The models had constant weir height and had the crest radius varies of 5, 7.5 and 10 (cm). The results were shown that the discharge coefficient (Cd) increased with increasing head to crest height ratio (h/P). There were increases in Cd by 3.75%, 3.11% and 3.12% for (5, 7.5 and 10 cm) respectively, [10]**.** A laboratory and CFD models were implemented on rectangular channel with broad crested weir to predict the discharge coefficient Cd. It was shown a good combatable between the laboratory and the CFD results, [11]. Many other researchers studied the description of flow structure over weir with broad crest. The numerical model results were simulated and the results of turbulence suitable models were recommended, error of it comparing in cases up to 3 % by using ANSYS-CFX (2D and 3D), FLOW 3D (2D) and ANSYS-Flotran (2D), were tested and carried out on one discharge, [12]. A 1D laser Doppler anemometry (LDA) is considered to find the velocity weir over a curvilinear broad-crested weir. Fluent model based on finite volume method with different turbulent models are employed with similar experimental conditions. The volume of fluid (VOF) method was applied to estimate the water level. The numerical and experimental results indicated that the RNG *k-ε* turbulence model provides the best results with respect to the other CFD models, [13]. Experimental and CFD models were also employed to study the flow over rounded corner broad crested weir. The numerical model was employed using Standard k–ε, RNG k–ε, Realizable k–ε and Standard k–ω turbulence models with the volume of fluid (VOF) to simulate the water surface levels. The results compared with experimental data to indicate that the standard k–ε model has the best results with experimental data than the standard k–ω model, [14]. The hydraulic parameters of wooden weirs that shaped and prepared with five different values of rounding curvature were measured with different value of discharge. The results indicated that the discharge coefficient CD increases up to 8 percent when the front weir top edge is curved while for both front and behind weir top edges are curved, the CD coefficient increases up to 14 percent, [15]. The critical depth Dc and flowrate relationship with the head over the crest of ogee spillway (Yo) and its relation with the longitudinal slope of the channel (S_o) was also simulated and studied. A ten laboratory experimental tests with a longitudinal value of slope ranging from zero to 0.02 was carried out to estimate the relationship between depths. The linear regression analysis indicated that the relationship between the (Yo) and Dc depths is about 1.2 regardless the effect of the S_o values. A new formula for flowrate over an ogee spillway was determined using this relationship giving a very good fitting with the results of CFD and experimental models, [16]. Four suggested design of submerged sharp weir with 30° and 120° inclined slopes were designed and considered to investigate the problems of flow properties and scouring using CFD model and experimentally. According to the statistical tests, the results of the CFD model for local scour were significantly varied from the results provided by the physical model of the weir of a 120° inclined angle, [17].

Referring to the above literatures, it was not found any trial to find a new formula to estimate the flowrate a cross the weir dependent on a control section at some where point on the weir. The aim of this study is to derive a new formula relating the flow over the weir with the depth at end edge section by considering it as a control section.

2. Theoretical assessment

The traditional relationships for assessing the flow characteristics over broad crested weirs are depended mainly on the relation between flow and upstream water level Eq. 1 as shown in Fig.1. The flowrate over the weir is related with the critical depth ***Dc*** by the following equation:

$$\boldsymbol{Q = C\sqrt{g}\, b\, D_C{}^{3/2}} \tag{2}$$

Where $\boldsymbol{Q}$ is the flowrate (m^3/s), ***Dc*** is critical depth (m), *C* = coefficient of discharge, *b* is the width of the weir (m) and *g* is the gravity acceleration (m/s^2).
However, Eq. 2 does not provide a very practical means of calculating $\boldsymbol{Q}$. This is because the location of the critical depth ***Dc*** is neither fixed nor is it easy to guess. The target of our work in this study is to relate the critical depth ***Dc*** to the depth at the end section of the weir and modify Eq.2 in order to provide the practical means of calculating $\boldsymbol{Q}$ based on the $\boldsymbol{Y_e}$ depth as a controlling parameter. This will be done through several experimental tests and a comprehensive statistical investigation with a CFD simulated model for verifications.

3. Experimental set-up

The experimental set-up was consisted of a hydraulic flume with a length of 1.5 (m) and a width of 0.051 (m) constructed on a Hydraulic Bench with a steel model of rounded broad crested weir, Figs. 2 & 3

Figure 2. Channel flume and hydraulic bench

Figure 3. Steel rounded broad crested weir

3.1 Experimental procedure

The tests of experimental work were conducted with different values of a longitudinal slope So. The first test was starting by setting a zero slope $\boldsymbol{S_o=0}$. For each flowrate $\boldsymbol{Q}$, the water depth at the end edge of the weir was measured and the critical depth is computed with Eq.3.

$$\boldsymbol{Q = \sqrt[3]{\frac{Q^2}{b^2 g}}} \tag{3}$$

Where $\boldsymbol{Q}$ is the actual flowrate (m^3/s), and the other variables as already defined. With each test, the flowrate was changed by certain steps to provide 8 runs with each value of the $\boldsymbol{S_o}$. After that, the value of $\boldsymbol{S_o}$ for the flume was increased and the same procedure of the previous test was repeated again to achieve another test.

The procedure was done for ten values of the longitudinal slope ranged from zero to 0.0495.

3.2 Experimental results

For the first run of the horizontal longitudinal slope, Fig.4 explains the relationship between the actual end depth (Y_e) and the theoretical critical depth ***(Dc)***. It is very clear that this relation takes a linear form a as:

$$Dc = a\,Y_e \tag{4}$$

Where ***a*** is gradient of the best fit line provided by the statistical regression analysis, [18].

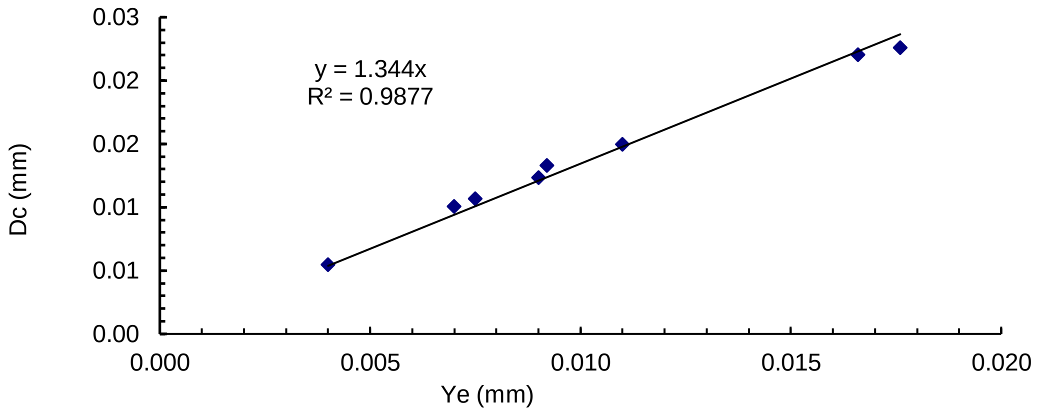


Figure 4. Dc and Y_e Relationship for $S_o = 0$

3.3 S_o, *Dc* and *Ye* relationship effect

The results for the relation between the ***Dc*** and ***Ye*** for the other different values of ***So*** are represented in **Table** 1. Comparing the results, it could be noted that the depth Y_e does not exactly equal to the critical depth ***Dc***. It can also be noted that when the weir has a curvature edge, the longitudinal slope S_o has an insignificant effect on the values of the constant **(a)** or the relationship between critical and end depth**.** Consequently, it can be considered that the average values of the all runs (1.45831) is a best estimated value for ***(a)***. In other words, regardless the longitudinal slope S_o, the relation between ***Dc*** and Y_e can be written as:

$$Dc = 1.45831\,Y_e \tag{5}$$

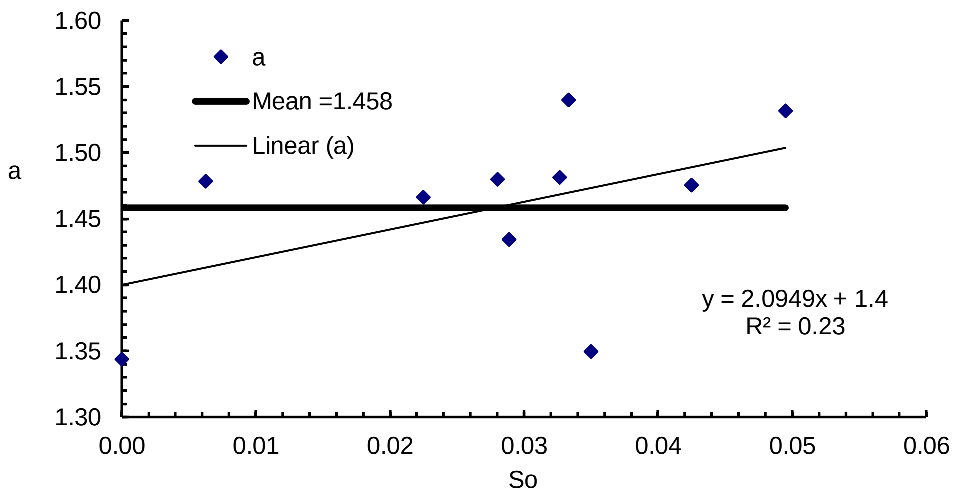


Figure 5. The effect of S_o on the values of *constant* (a) for rounded broad crested weir

Table 1. Relation of a verse S_o values for round broad crested weir

No.	Longitudinal Slope So	*Dc = a * Ye*
1	0	1.344
2	0.0063	1.4785
3	0.0225	1.4663
4	0.028	1.4801
5	0.0289	1.4347
6	0.0326	1.4817
7	0.0333	1.5403
8	0.035	1.3499
9	0.0425	1.4759
10	0.0495	1.5317
	Average	1.45831

3.4 Flowrate prediction

The flow over a weir can be estimated based on the critical condition formula of Eq.2, [19]. Considering the general relationship of $\boldsymbol{Dc = a\ Y_e}$ (Eq.5) with Eq.2, the general estimated flowrate formula will be obtained.

$$\boldsymbol{Q} = \sqrt{\boldsymbol{g}}\ \boldsymbol{b}\ (\mathbf{1.45831}\ \boldsymbol{Y_e})^{3/2} \tag{6}$$

Where all variable as were already defined above. As has been indicated, the value of the longitudinal slope $\boldsymbol{S_o}$ has an insignificant effect on the flowrate estimation. As the Fig. 6 shown, the formulae indicate a very good verification with the results of the corresponding actual flowrate for all the ten runs

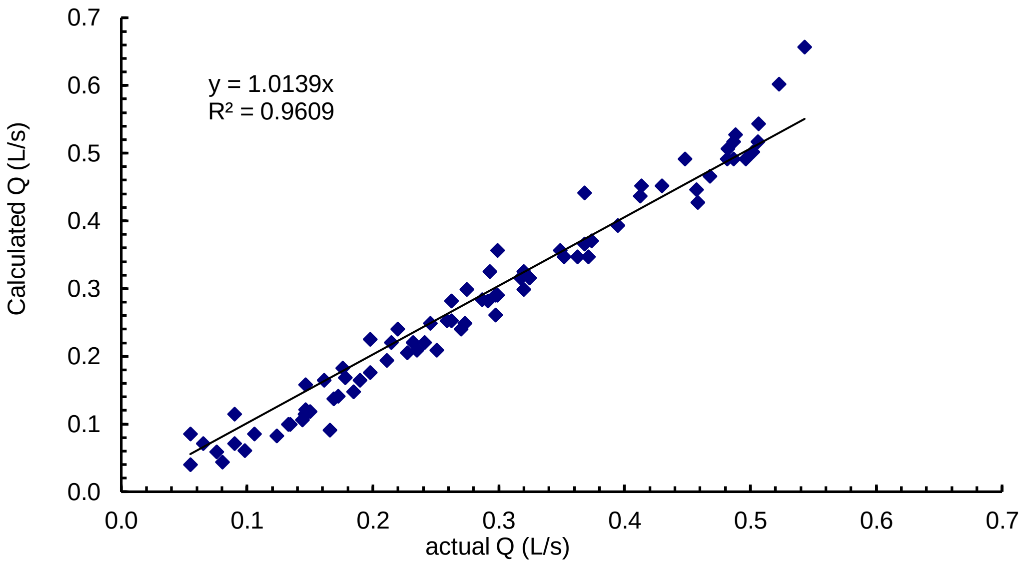
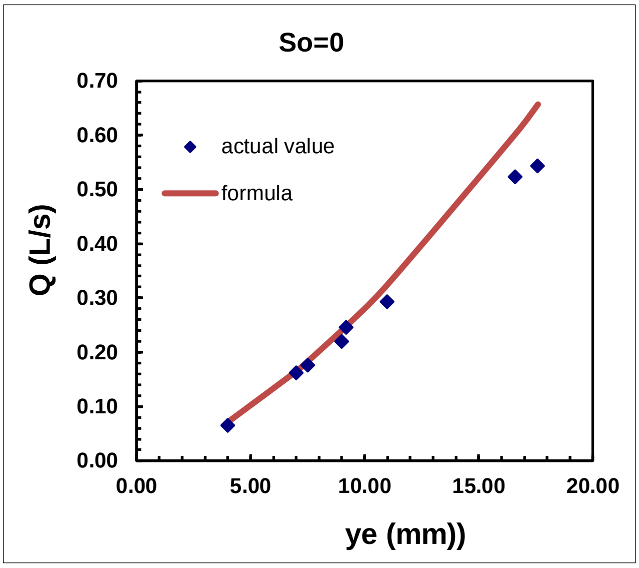
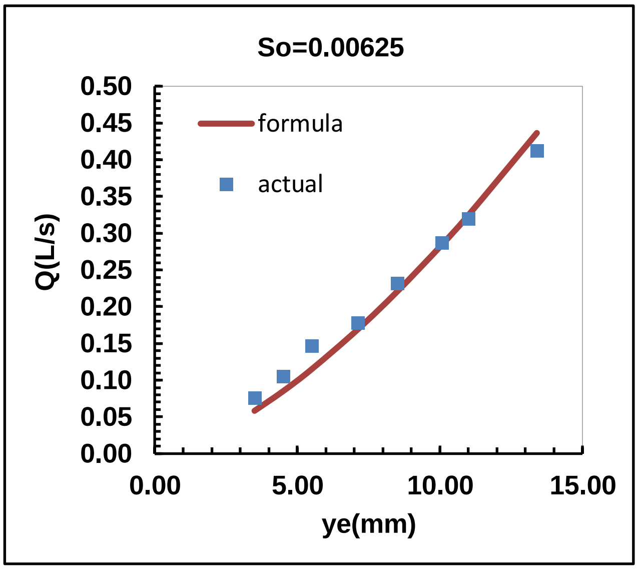


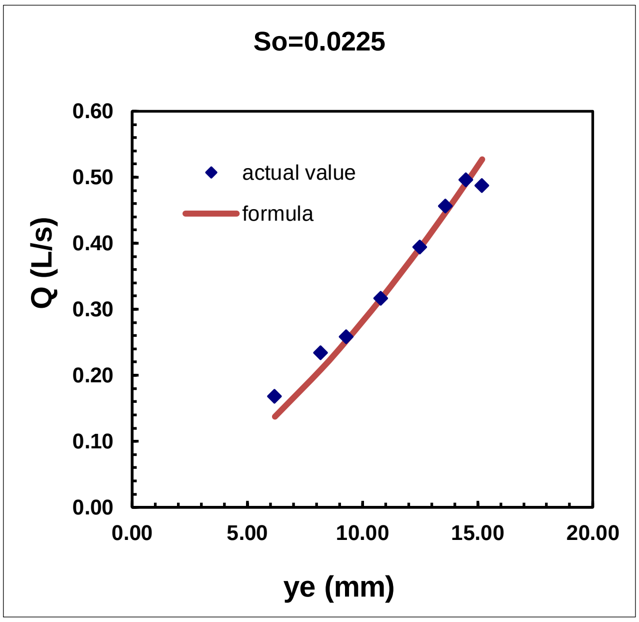
Figure 6. Verification of the weir formula corresponding to the actual values of ten slopes

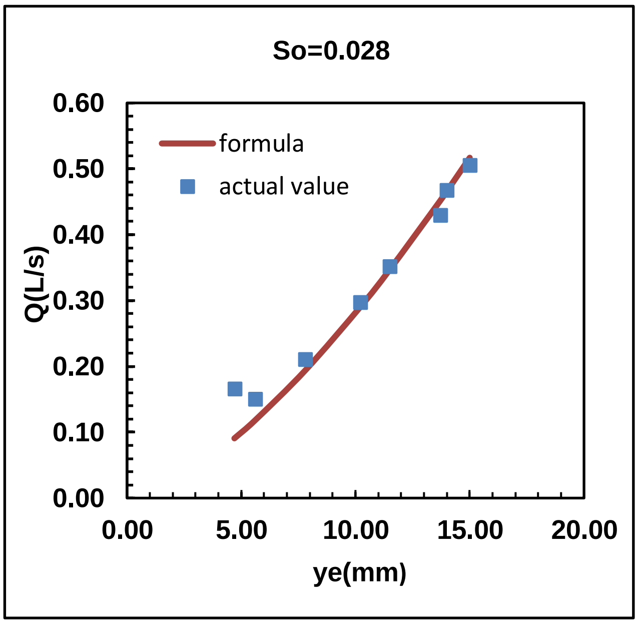
Fig.7 illustrates the formula verification of each value of slope individually, where the formula appeared a very good agreement with all the tests of the ten slopes.

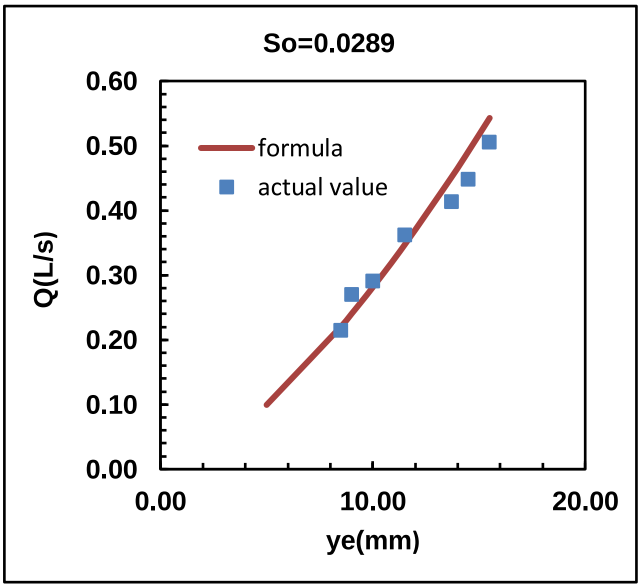
For the purpose of examining the confidence of that formula it was also statistically tested by computing the relative errors of the formula, Table **2**. The results indicated that the new formula satisfied an excellent accuracy and prove a well dependence for flowrate estimation.

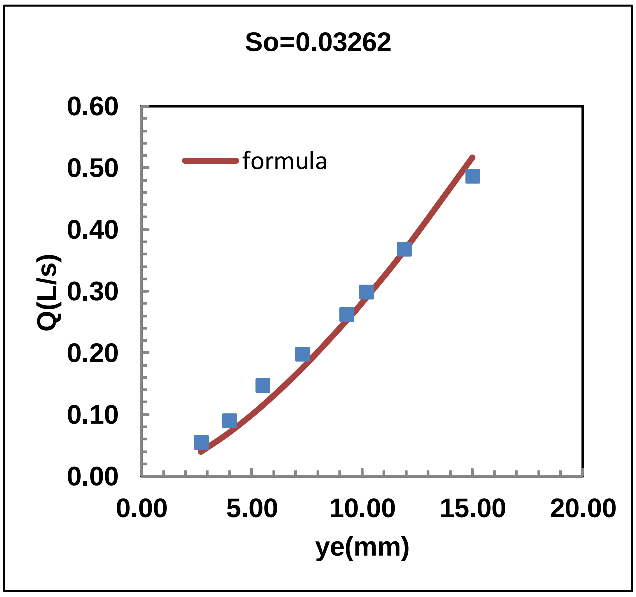

So=0
actual value
formula
Q (L/s)
ye (mm))


So=0.00625
formula
actual
Q(L/s)
ye(mm)


So=0.0225
actual value
formula
Q (L/s)
ye (mm)


So=0.028
formula
actual value
Q(L/s)
ye(mm)


So=0.0289
formula
actual value
Q(L/s)
ye(mm)


So=0.03262
formula
Q(L/s)
ye(mm)

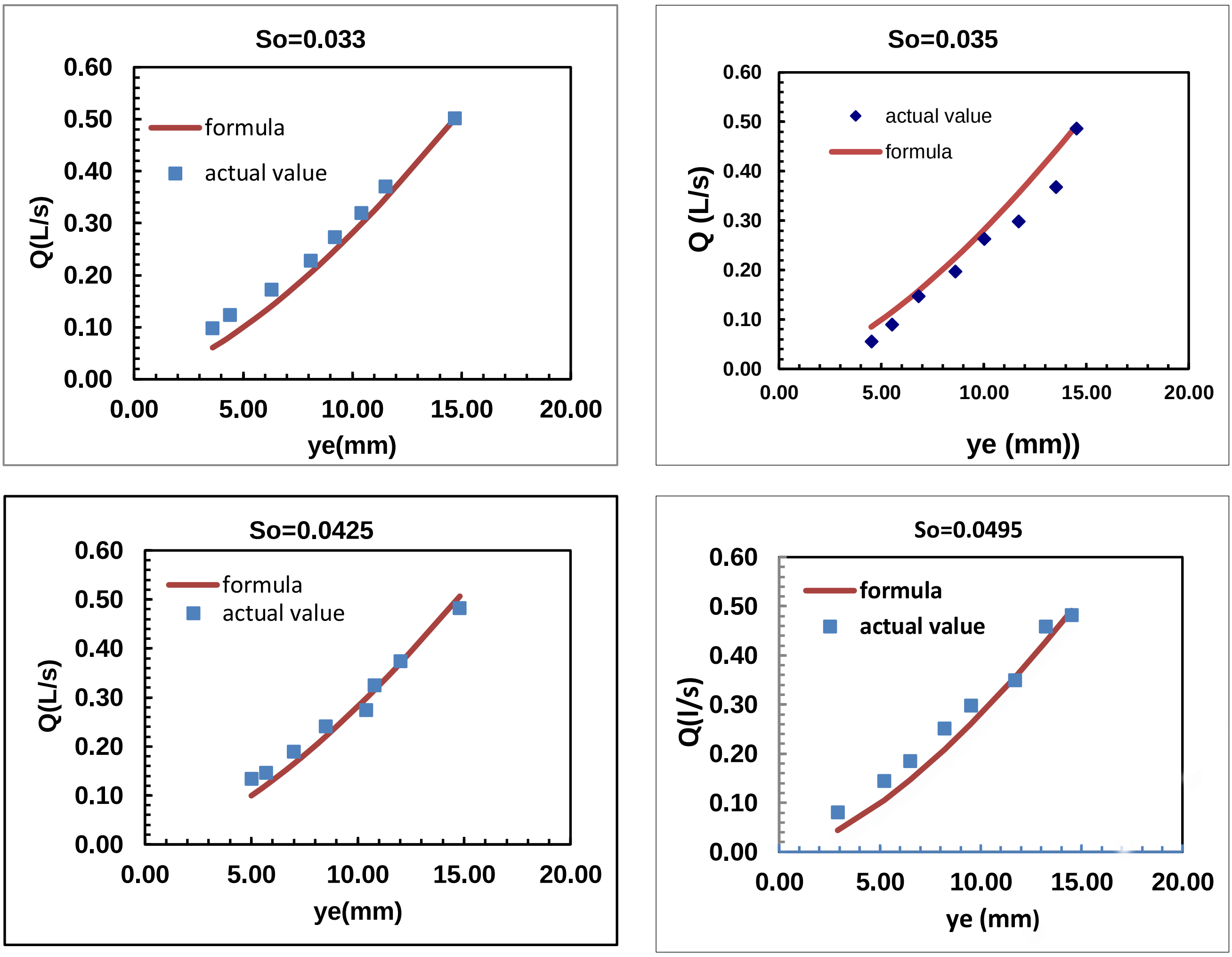


Figure 7. Individual verification of (Eq.6)

Table 2. Statistical analysis test for Eq. 6

Slope	RMSE	MSE	MAE	MAPE %	RSE
0	0.050864	0.002587	0.033004	9.117493	0.101299
0.0225	0.020524	0.000421	0.015151	5.526314	0.030807
0.035	0.037699	0.001421	0.030806	18.81995	0.076544
0.0495	0.032469	0.001054	0.029909	16.52773	0.057895
0.00625	0.01785	0.002549	0.015355	10.37297	0.027524
0.028	0.030916	0.000956	0.02143	10.81445	0.056947
0.0289	0.029565	0.000874	0.026494	9.072548	0.063383
0.033	0.027959	0.000782	0.009556	15.25999	0.048815
0.0326	0.019945	0.000398	0.017521	11.96526	0.021425
0.0425	0.022717	0.000516	0.020756	10.26662	0.041356

$$RMSE = \sqrt{\frac{1}{n} \sum_{i=1}^{n} e_i^2} \quad (7)$$

$$MSE = \frac{1}{n} \sum_{i=1}^{n} e_i^2 \quad (8)$$

$$MAE = \frac{1}{n} \sum_{i=1}^{n} |e_i| \quad (9)$$

$$MAPE = \frac{100\%}{n} \sum_{i=1}^{n} \left|\frac{e_i}{Q_a}\right| \quad (10)$$

$$RSE = \frac{\sum_{i=1}^{n} e_i^2}{\sum_{i=1}^{n} (Q_a - \bar{Q_a})^2} \quad (11)$$

Where,
RMSE = The Root Mean Squared Error, MSE = Mean Squared Error, MAE = Mean Absolute Error, MAPE is the Mean Absolute Percentage Error. n is the total observed number, e = error = Q actual – Q formula = Q_a – Q_f and $\overline{Q_a}$ is the mean of actual flowrates

4. Numerical solution

4.1. Governing equations and the model of turbulence

ANSYS-Fluent, which solve the 3D Reynolds-averaged continuity with Navier-Stokes equations (RANS) using finite volume method, was used to perform the numerical analysis of flow over the weir, [20]. Also, the simulated model was employed with the help of the experimental results to verify and check the precise of accuracy of the target formula for flowrate estimation.
The problem of flow across broad crested weir could be formulated considering the Reynolds-averaged Navier–Stokes (RANS) equations as,

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (12)$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial u_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i + \vec{F} \quad (13)$$

Where: ρ = fluid density, $u_{i,j}$= velocity vector due to time, x = space, t = time, p = the pressure, $\mu = \mu_0 + \mu_t$, μ_t is turbulence viscosity and μ_0 is dynamic viscosity, $\vec{F}$ = the body force and g_i = acceleration due to gravity.

In the present study, in order to model the Reynolds stress term, it is depended on the standard k-ε model to provide the solution. It is a two-equation turbulence model depend on the eddy viscosity concept. These transport equations are the turbulence kinetic energy (k) and the turbulence dissipation rate (ε) equations. The model neglects the effect of molecular viscosity and assumes that the flow is fully turbulent. In this model, the equations of the turbulent energy rate of dissipation and the turbulent kinetic energy derived as follows, **[21].**

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial k}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k - \rho\varepsilon \quad (14)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial \varepsilon}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + G_{1\varepsilon}\frac{\varepsilon}{k}G_k - G_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (15)$$

Where the eddy viscosity μ_t, is considered in equation (6);

$$\mu_t = C_\mu\,\rho\frac{k^2}{\varepsilon} \quad (16)$$

In above Equations, the term *Gk*, represents the production of turbulent kinetic energy. $G_{1\varepsilon}$, $G_{2\varepsilon}$, and C_μ are constants and equal to 1.44, 1.92, and 0.09, respectively. σ_k and σ_ε are the turbulent Prandtl Numbers for *k* and ε equal to 1.0, 1.3, respectively.

4.2. Free-Surface Modeling

The problem is a type of air-water two-phase flow. The method of volume of fluid VOF is used here to form the free surface at the interface between the air and the water which developed by Hirt and Nichols, [22]. Where $\boldsymbol{\alpha}$ is a volume fraction that applied to estimate the percentage of fluid which occupied by each mesh cell. The value of $\boldsymbol{\alpha}$ ranges from zero to one. Eq.17 considered as an approximate convection transport that could be employed to evaluate the value of $\boldsymbol{\alpha}$ along the domain

$$\frac{\partial \alpha_w}{\partial t} + \frac{\partial}{\partial x_i}(\alpha_w u_i) = 0 \quad (17)$$

$$\boldsymbol{\alpha_a = 1 - \alpha_w} \quad (18)$$

Where α_w is volume fraction of water and α_a is the volume fraction of air.

Since the VOF of the air phase can be inferred from the Eq. 18; it is just need to solve one transport equation. The first setup in simulation model is built the weir geometry of problem at the same dimension of the experimental work and generated meshing model. Therefore, program of mesh geometry is applied to do all these orders. The other most important steps in numerical models is defined boundary conditions which should be the same as that of the physical model. The boundary conditions of this study are included the velocity and pressure for water and air at inlet, pressure at outlet, wall and free surface. Also, the other an important part of the numerical simulation is the size of mesh because of its effect on the result accuracy and time simulation.
The boundary condition at the inlet of the channel is:
Inlet velocity V (0,0) = Uo, water depth Yo and VOF= 1. at point of p (0,0)
In the outlet channel, the boundary condition is as follow:
Outlet pressure = 0 and VOF (water ratio fraction) = 0 at point of P (L, y)
The adopted pressure condition is applied value of pressure (1 atm). The no-slip wall boundary condition is set to the walls of side and bed of the weir channel. The mesh of the CFD Model was designed to be as Proximity and Curv ature type with at total number elements of 162494 and 32281 nodes have a minimum and maximum sizes of 2.0537e-4 m and 4.1073e-2 m respectively, Fig. 8.

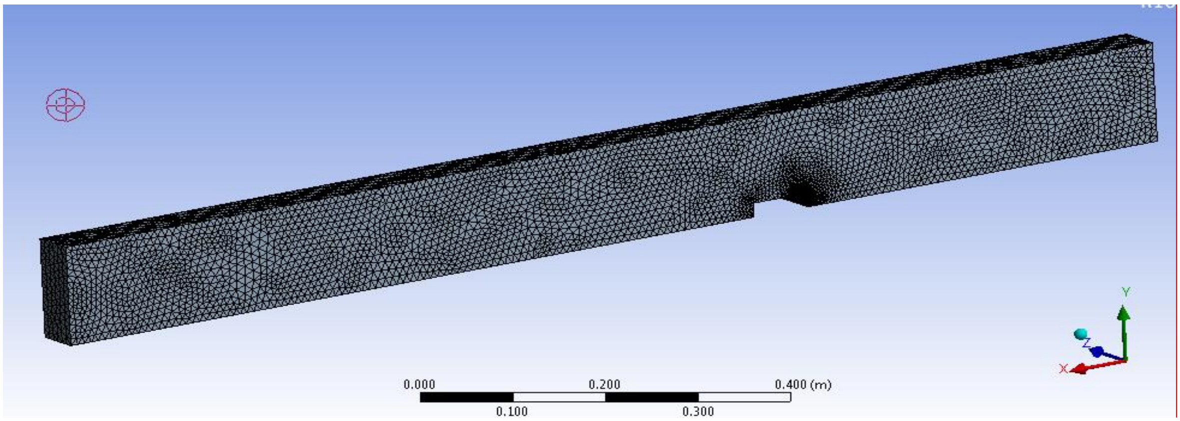


Figure 8. Model meshing.

A simulation results of a case study with a laboratory values of $\boldsymbol{Q_a}$ = 0.3268 l/s, ***Yo*** = 43.2 (mm), ***Ye*** = 11.8 (mm) and ***So*** = 0 were analyzed and represented in Figs. (9 to 12). Figs. (9 & 10) show the water volume fraction and the stream flow pattern, while the Figs. (11 & 12) show the velocity and pressure distributions at the inlet channel and end edge sections of the weir. As the figures indicted, the results of the CFD simulated model provides a very good agreements with the all cases. Also, Table 3 shows the results of the CFD model for several elective experimental tests. However, the formula indicates a very good match with the results of the CFD model and experimental tests with a maximum percentage error less than 10% in a worst case.

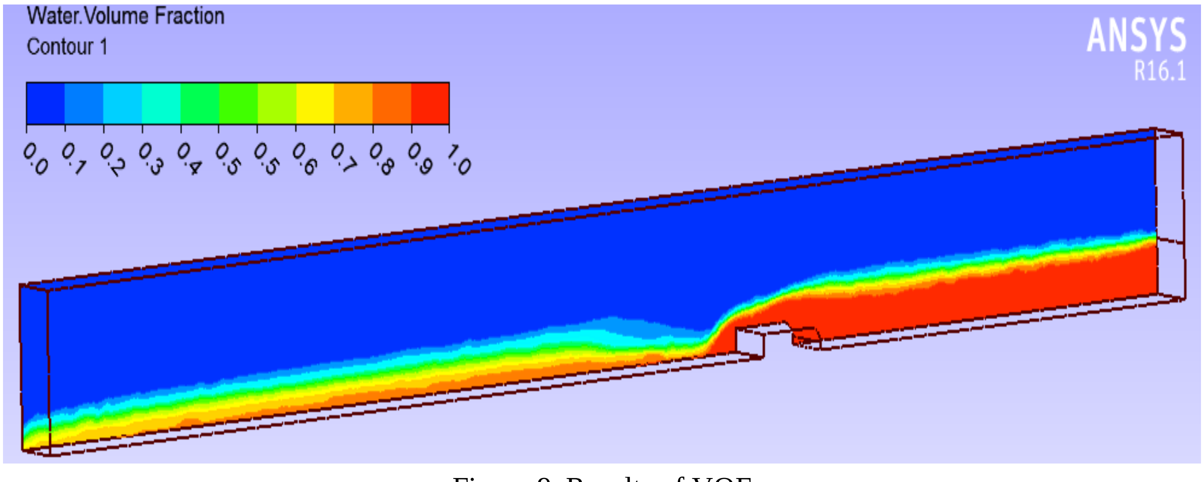


Figure 9. Results of VOF

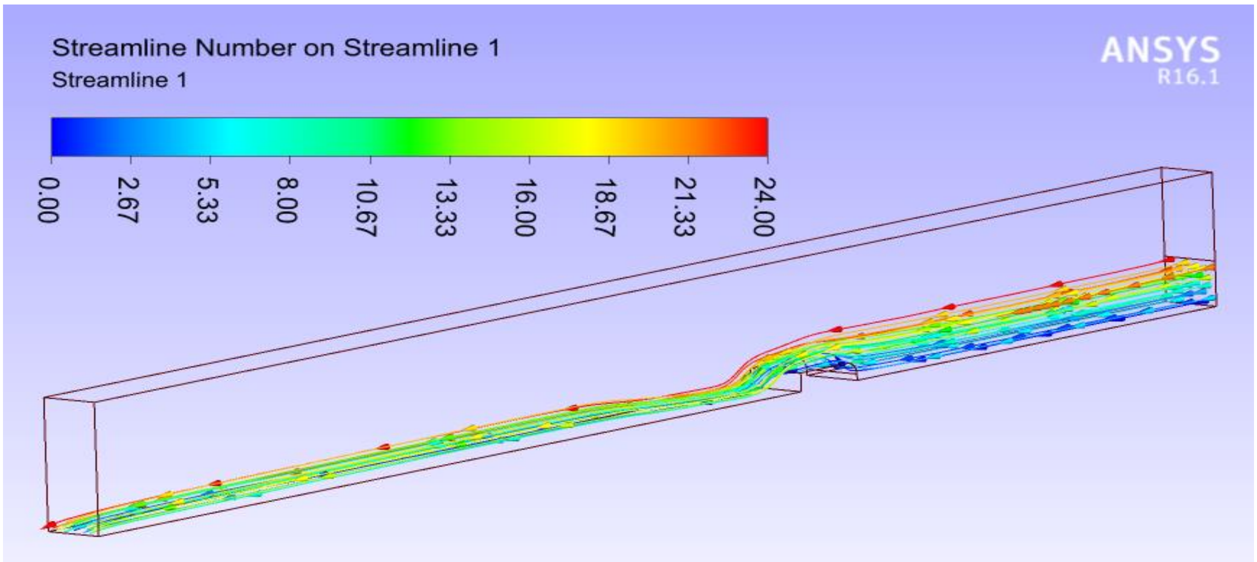


Figure 10. Results stream flow pattern

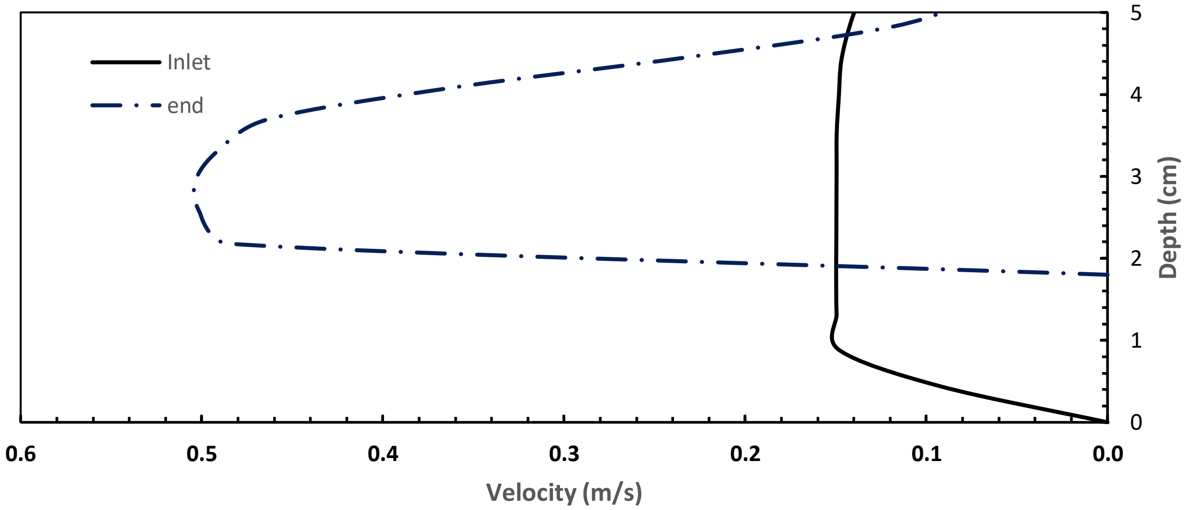


Figure 11. Velocity distribution resulted from the CFD model

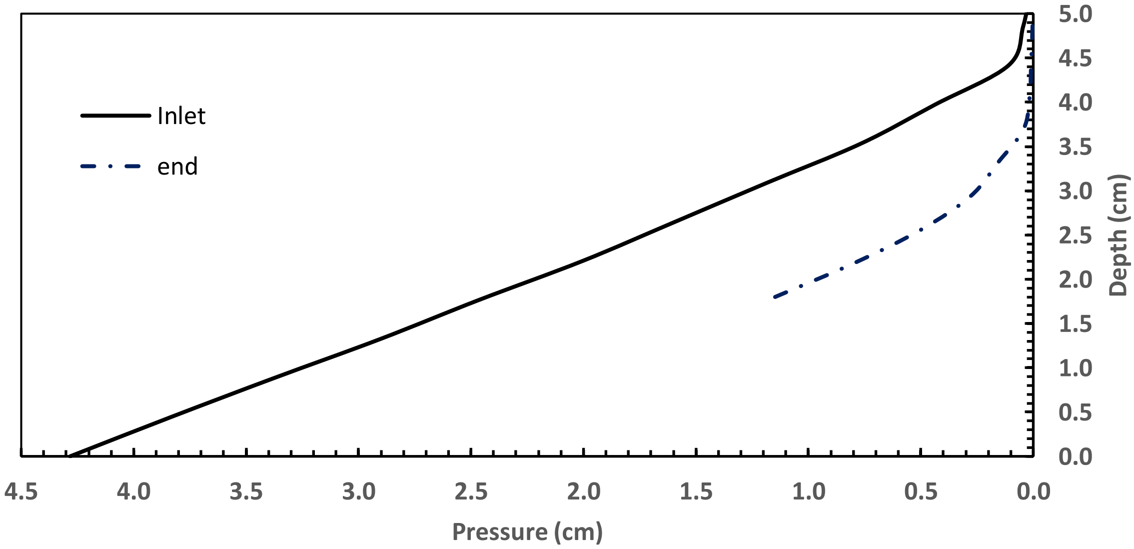


Figure 12. Pressure distribution resulted from the CFD model

Table 3. verification of Eq. 6 for experimental tests and CFD model

So	Actual Q L/s	Q of Eq.6 L/s	Q of Simulation model) L/s	Error of % Eq.6 based on actual Q	% Error of Eq.6 based on Simulated Q
0.0000	0.3286	0.3600	0.3284	9.56	9.62
0.0063	0.2316	0.2204	0.2355	4.84	6.41
0.0225	0.3171	0.3157	0.3098	0.44	1.90
0.0280	0.4676	0.4660	0.4676	0.34	0.34
0.0289	0.2912	0.2813	0.2913	3.40	3.43
0.0326	0.2988	0.2898	0.2980	3.01	2.75
0.0333	0.3709	0.3469	0.3705	6.47	6.37
0.0350	0.1465	0.1577	0.1464	7.65	7.72
0.0425	0.3241	0.3157	0.3235	2.59	2.41
0.0495	0.3489	0.3560	0.3487	2.03	2.09

5. Conclusions

The study depended on considering the end section of the curved broad crested weir as a control section and relating the critical depth ***(Dc)*** with water depth ***(Ye)*** at this section. Ten experiments tests with different longitudinal slopes ranged from (zero to 0.0495) were done to find the relationship between the ***Dc*** and ***Ye*** depths. The results of linear regression analysis showed that the concluded relationship for ***Dc*** and ***Ye*** depths is equal to 1.45381 as an average. Consequently, a new formula for predicting the flow over weir was derived. Different statistical indexes were applied to verify the accuracy of the predicted formula and the results appeared a well match with the all experimental results. The results of CFD models showed that the CFD technique is able to simulate the flow over the weir and satisfied the results of the new formula with a maximum percentage error less than 10% in worst case for the all laboratory tests. It was also shown that the CFD technique can be strongly used as a useful tool to simulate and accuracy predict the pressure and velocity distribution of flow over the weirs.

6. Acknowledgments

Deep appreciation to Mustansiriyah University (www.uomustansiriyah.edu.iq) Baghdad-Iraq for the helps and supports to achieve this study and to the staff of the Laboratory of Hydraulics in the College of Engineering for their efforts.

7. References

[1] S. Kumar, Z. Ahmad and T. Mansoor, "A new approach to improve the discharging capacity of sharp-crested triangular plan form weirs", *Journal of flow measurement and instrumentation*, 22, pp.175-180, 2011.

[2] M. G. BOS, "Discharge Measurement Structures", *International Institute for Land Reclamation and Improvement/ILRI, Wageningen*, The Netherlands, 1989.

[3] A. J. M. Harrison, "The streamlined broad-crested weir". *ICE Proc.*, 38, pp.657–678, 1967.

[4] J. S. Montes, "The streamlined broad-crested weir", *Discussion, ICE Proc.*, 42, pp.576–578, 1969.

[5] J. S. Montes, "Hydraulics of open channel flow", *ASCE*, New York, 1998.

[6] M.A. Sarkar, and D.G. Rhodes, CFD and Physical Modeling of Free Surface over Broad-Crested Weir, Cranfield University, Cranfield, 2004.

[7] K. S. EL-ALFY, "Effect of Vertical Curvature of Flow at Weir Crest", *Ninth International Water Technology Conference, IWTC9*, Sharm El-Sheikh, Egypt, pp. 249–262, 2005.

[8] S. Felder and H. Chanson, "Free-Surface Profiles, Velocity and Pressure Distributions on a Broad-Crested Weir: A Physical Study", *Journal of Irrigation and Drainage Engineering*, Vol. 138, No. 12, December 1, 2012.

[9] H. Afshar and H. Hooman, "Experimental and 3-D Numerical Simulation of Flow over a Rectangular Broad-Crested Weir", *International Journal Engineering and Advanced Technology*, 2, (2013), pp. 214-219.

[10] ABDULLAHI, U., Experimental Investigation of Flow Characteristics over Semi-Circular Broad Crested Weir Models, PhD thesis, Ahmadu Bello University, ZARIA NIGERIA, 2014.

[11] H. Seyed and A. Hossein, "Flow over a Broad-Crested Weir in Subcritical Flow Conditions", *Physical Study, Journal of River Engineering,* Vol. 2, No. 1, 2014.

[12] Z. Zbynk and R. Ladislav, "Flow structure in front of the broad-crested weir", *Czech Republic, European Physical Journal (EPJ) Conferences,* Vol. 92, 2015.

[13] O. Simsek, S. Akoz and G. Soydan, "Numerical validation of open channel flow over a curvilinear broad-crested weir", *Progress in Computational Fluid Dynamics*, Vol. 16, No. 6, 2016.

[14] S. A. Al-Hashimi, M. Huda and T. N. Nahi, "Experimental and Numerical Simulation of Flow over Broad Crested Weir and Steeped Weir using Different Turbulence Models", *Journal of Engineering and Sustainable Development*, Vol. 21, No. 02, March 2017.

[15] M. Ashour, T. Abu-Zeid and R. Hassan, "The rule of the front and behind top edges of weirs on their hydraulic performance and working efficiency", *Limnol. Rev.* 17, No. 3, pp.113-122, 2017.

[16] S. M. Sadiq, and Z. T. AL-SHARIFY, "Experimental Work and CFD model for Flowrate Estimating Over OGEE Spillway under Longitudinal Slope Effect", *International Journal of Civil Engineering and Technology* (IJCIET), Vol. 9, Issue 13, pp. 430–439, 2018.

[17] T. R. Al-Husseini, A. S. T. Al-Madhhachi and Z. A. Naser, "Laboratory experiments and numerical model of local scour around submerged sharp crested weirs", *Journal of King Saud University–Engineering Sciences*, 2019.

[18] J. Wiley and Sons, Fundamentals of Probability and Statistics for Engineers, T.T. Soong, New York, U.S.A. 2004.

[19] R. S. Khurmi, A textbook of hydraulic, fluid mechanics and hydraulic machines. Ram nagar, New dalhi, 2009.

[20] ANSYS *Fluent Theory Guide*, ANSYS Inc. USA. 2016.

[21] B. Launder and B. D. Spalding, "The Numerical Computation of Turbulent Flows", *Computer Methods in Applied Mechanics and Engineering*, 3, pp.269-289, 1974.

[22] C.W. Hirt and B. D. Nichols, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", *Journal of Computational Physics*, 39, pp.201-225, 1981.