Simulation Analysis of Combined Flow over and under Semi-Cylindrical Structure

Shaker Abdulatif Jalil

Water Resources Engineering Department, College of Engineering, University of Duhok, Iraq

shaker.abdulatif@uod.ac

Submission date:- 27/2/2019	Acceptance date:- 28/3/2019	Publication date:-5/5/2019

Abstract

Combined flow over and under structure may solve the problem of the deposit of suspension materials in channels. Semi-cylindrical shape reduces the curvature of streamlines which reflected on its performance. To study how this shape performs, experimental and simulation has been done. The laboratory models were of four different diameters and four different gate openings. The same physical structures have been modeled in commercial software, FLOW-3D®, by employing RNG k- ε turbulence model. The verification has been based on measured flow profile and discharge. Simulation outputs indicate that a separation zone located at a distance from the structure became farther when the diameter and gate opening decreases, also the separation portions and their thickness are related to the incoming discharge. The location height of separation zone tends to be lower when there is an increase in flow discharge and it is located at half the total depth when two flow portions are equal. The weir flow in this system shows a better performance than traditional weir by at least 33%, while the gate out flow is less than free flow of the same total head by 70% to 90%. Within the limitation of this work, two mathematical models for predicting discharge coefficient have been proposed for the weir and gate respectively, moreover a model for predicting relative discharge of weir to gate, and one mathematical model for the dimensionless total discharge.

Keywords: Semi-cylindrical structure, Combined flow, Streamlines separation, Flow portions, Weir and Gate;

1- Introduction

Flow Simulation for some motion phenomenon may help to understand and analysis flow process particularly when there are difficulties of measurements. As simulations is based on Computational fluid dynamics, May commercial and open software are produced, which they include many turbulence models which they can fit the actual state of flow. The computational methods may help to predict discharge and free surface profile of flow where there is a complication of measurements, monitoring and the hydraulic operation such as complexly combined sewer overflow in waste water treatment plant in Lyon, France [1]. The numerical simulations can make the internal flow currents are visualized, Jia et al [2] study the effect of submerged weirs located in river bends on the helical secondary current and then on navigation conditions. The study noted that the helical secondary current is distributed by the weir towards the inner bank which is comfortable for navigation. Modeling also is applied in the simulation of velocity currents in fish ways such as Fujihara et al [3] investigation which prove that there is a good agreement between the velocity of the current measured in a large scale physical model and related values obtained from numerical simulation. Simulating the flow over sharp crested weir using K-e turbulence model built in Flow-3D package shows a good prediction with a profile of flow experimental data, the results show the major parameter which affects the discharge coefficient is the head over the crest relative to the weir height (h/P)[4]. Finite difference codes based on Boussinesq-type equation models is generated to simulate non-hydrostatic pressure flow, where streamlines curvature occurs due to short length transitions, such as in short-crested weir, Parshall flume, hump and in free over fall structures [5]. A comparison between 2D and 3D numerical models of flow over broad crested weir shows that the results of free surface profile are close and there are no such significant differences although 3D model consumes time [6]. A piano key weir had been simulated by means of Flow 3D software, the models were verified by experimental data which show that the discharge increases with the increase of the inlet width to outlet width ratio of the weir key, and the performance of the weir reduced with increase of water depth to the weir height and so does the coefficient of discharge [7]. Four different turbulence models built in commercial software Fluent has been employed to study a turbulent flow over a circular spillway, a comparison between experimental water surface elevation and the results of these models shows the best agreement is produced by RSM turbulence model [8]. The best prediction similarity of free surface profile over broad crested weirs has been achieved by employing the standard k–ε turbulence model among four other different turbulence models existed in "Fluent" software [9]. Free code source software "OpenFOAM" is used to simulate flow over broad crested weir with inclined upstream face, the comparison between the results of two turbulence models, the standard k-ε model; and the SST k-w show good prediction of flow profile. The predicted area of separation zone by SST k-w model is larger than that predicted by standard k-e model and the coefficient of discharge decreases as the angle of the upstream face increases [10]. Two-dimensional finite element model (CCHE2D) is validated on the physical model of river bend with and without weir structure, the two-dimensional model is imposed to evaluate the roughness of the bed which is employed in three-dimensional model (CCHE3D).The models show that the flow in river bend is typical with a superelevation in the bend without weir structure and also there is a high water surface zone built up at a centre part of the upstream face of the weir [11]. Abdulatif & Rhmaan [12] simulated a flow over semicircle crested weir by using HEC-RAS software and validated by experimental data, a good agreement has been achieved and higher values of discharge coefficient are noted that the value of broad crested weir coefficient have the same dimensions.

There is a combined advantage when weir and gate operate in one structure such as silt-flushing, diversion, regulation and flow measurement. This investigation aims to lighting the physical processes of such flow phenomenon caused by a semi-cylindrical structure by simulating the conditions of flow after model verification by experimental work and analyze two portions of combined flow and their measure.

2- Theoretical Background

Fundamentals and theory of weir and gate flow are very well stated in authenticity and original books such as Henderson [13], Chow [14] and Subramanya [15]. Empirical formulas for estimating discharge over sharp crested weir are many the well-known are Rehbock formula and British Standards [16]. The flow rate equation according to British Standards is:

$$Q_{weir} = \frac{2}{3}\left(0.602 + 0.083\frac{h}{P}\right)\sqrt{2g}B(h + 0.000012)^{3/2} \ldots\ldots\ldots\ldots(1)$$

Where: h= upstream gauged head above the weir crest, P= crest weir height from channel bed, Cd =the coefficient of flow rate, g= acceleration due to gravity, B=width of the weir. The gate flow equation also stated in the original open channel books, the coefficient of discharge has been determined by many researches such as Lozano et al [17] Shayan et al [18]. The general equation is:

$$Q_{gate} = C_d\ d.\ B\sqrt{2gH_t} \ldots\ldots\ldots\ldots(2)$$

Where: Cd is the coefficient of discharge and is the total head. Physical quantities and dimensions can be related by using dimensional analysis. The dimensionless parameters for the flow over and under cylindrical structure are expressed by [19] as in equation (3).

$$\frac{\mathrm{Q_{total}}}{\mathrm{bd}\sqrt{2\mathrm{gd}}} = \oint\left(\frac{h}{d},\frac{H_t}{\mathrm{d}},\frac{\mathrm{H_t}}{P},\frac{\mathrm{h}}{\mathrm{H_t}}\right)\ldots\ldots\ldots(3)$$

Where: Qtotal is the total discharge and P = (D+d) as the height of weir crest.

The two France and England scientists, Navier and Stokes, have formulated Euler equations that include internal shear forces. Navier-Stokes equations describe fluids motion based on conservation laws of real fluids, these equations are a partial differential equation of mass, energy and momentum; they characterize how velocity, pressure, temperature, and density are related in the movement of fluid. They contend the inertial and viscous forces which reflect the degree of turbulence generated in a flow. Turbulences cause random fluctuations of velocity; the mathematical solution of this matter state is based on Reynolds decomposition by substituting in mean steady value of variables [20]. The conservation equations lead to the following equations:

$$\frac{\partial}{\partial x_i}(\rho\,\bar{U}_i) = 0 \ \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ (4)$$

$$\frac{\partial}{\partial x_j}\left(\rho\bar{U}_i\bar{U}_j - \bar{\tau}_{i\,j}\right) + \frac{\partial\bar{P}}{\partial x_i} - \bar{S}_M = \frac{\partial}{\partial x_j}\left(-\rho\bar{\acute{u}}_i\bar{\acute{u}}_j\right)\ \ldots\ldots\ldots(5)$$

Where $\bar{U}$ the mean steady value of velocity, ρ is fluid density, $\bar{\tau}$ is molecular stress tensor, $\bar{P}$ is the pressure and $\bar{S}_M$ is source term and the last term is Reynolds stress. The numerical solution for these equations depends on the number of stress terms included in the models. In this study, the renormalization-group turbulence model (RNG k-ε) is employed. According to the package manual o Flow-3D 2012 the RNG k-ε model is recommended for unstable flow with chaos due to appearance of eddies of various sizes [21], also this model presents good predictions for the horizontal velocity field and flow profile in study of flow over weirs [22], [23] and [24], while by [25] shows that the standard k–ε turbulence model is also forecasting well flow over broad weir, so the solution of these equations using RNG k-ε model leads to describing the flow near reality in simulation, as a part of flow domain is turbulent with eddies. Figure (1) shows a definition sketch for the main measured experimental parameters and streamlines of flow for simulation output.

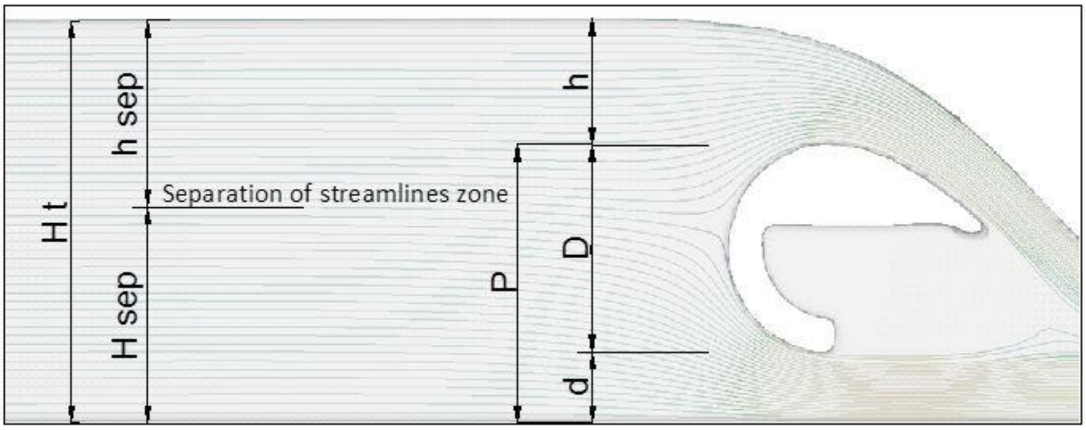

Figure (1): Simulation snap with definition sketch of geometric parameters

3- Experimental Work

Validation of numerical model is based on experimental data carried on laboratory flume; the flume has 5 m length, 0.3 m width and 0.4 m height. The semi-cylindrical structure has been made from metal plate. The diameters of physical models are 12, 9, 7 and 6 cm; they were installed to channel sides by silicone sealant at five heights from the channel bed equal to 5, 4, 3, and 2 cm, these heights are the gate openings. All the 16 models were tested by about 12 different flow rates. Water surface profile at the middle was measured by point gauge having accuracy of 0.0001m. The depth of water was measured at distances equal to 1 cm in structure zone where the maximum curvatures of water surface happen over and behind the crest.

4- Numerical Model Validation

The semi-cylindrical structure has been drawn in AutoCAD software and imported by the geometric tool of FLOW-3D® package to the main domain as a solid. The setup of the physical phenomenon was; air entrainment, sharp interface, acceleration of gravitational in the vertical direction, the flow is viscous, Nonslip wall shear boundary and the turbulence model is Renormalized group turbulence model (RNG k- ε). The discretization of the computational domain has been based on earlier studies [22], [26] and the mesh size was set equal to 5 mm and the aspect ratio equals one. The two sides and bottom of the channel were walls as conditions of flow domain boundary, while the top of channel was symmetry. The boundary condition upstream was a pressure head its value is equal to the value experimental flow depth, while the downstream boundary condition was outflow. Four flux baffles were generated, two of them placed over and under the semi-cylindrical structure to predict the discharge over the weir and to predict the discharge of gate underneath, and the other two flux baffles were placed in upstream and in downstream of the flow domain to predict the total discharge. FLOW-3D® package includes two algorithms for solving pressure velocity coupling, they are based on semi-implicit numerical techniques, one of them is successive over relaxation (SOR) and other is generalized minimum residual (GMRES), the two solvers are able to predict similar results as noted in the package manual [21].

5- Validation Result

Validation of the designed numerical model was based on the description of average error which was examined by two measures. The first measure of error was a comparison between the laboratory measurement of water level profile and that profile which predicted by numerical model. The second measure of error was a comparison between the predicted summation of flow discharges over and under the structure with that total actually measured in the laboratory. Statistical homogeneity analysis of the two samples was done with the prediction of error to verify the numerical model outputs, and with confidence 95% a T-test on the two group samples was done. The statistic descriptive of groups in the two comparisons test is shown in table (1), which includes the values of mean, standard deviation and standard error. The values in the table show how closed are samples group to each other in the two comparisons tests.

Table (1) Descriptive Statistics for Two Comparisons

Group Statistics

Group A		N	Mean	Std. Deviation	Std. Error Mean
Qweir+Qgate	1	208	20.933389	5.8123360	.4030130
	2	208	20.911004	5.8299942	.4042374
Flow Profile	1	1284	17.35	3.108	.087
	2	1284	17.16	2.930	.082

The actual data and the numerical ones were subjected to homogeneity of variance test to study whether the variances of two samples groups are significantly equal. Table (2) show result of the Independent T- test in which it can be noted that the P-value of Levene's test is much higher than 0.05, the variances are not significantly different across samples. The test also shows that P-value of 2-tailed is bigger than 0.05, indicates that there are no significant differences between the values of the mean of the tested groups. Moreover the zero value is in between the lower and upper interval for the difference between mean values.

Table (2) Independent T- test

Independent Samples Test

		Equality of		t-test for Equality of Means						
									of the Difference	
		F	Sig.	t	df	Sig. (2-tailed)	Mean Difference	Std. Error Difference	Lower	Upper
Qweir+Qgate	Equal variances assumed	.004	.953	.039	414	.969	.0223853	.5708129	-1.0996676	1.1444382
	Equal variances not assumed			.039	413.996	.969	.0223853	.5708129	-1.0996676	1.1444382
Flow Profile	Equal variances assumed	3.411	.065	1.601	2566	.110	.191	.119	-.043	.425
	Equal variances not assumed			1.601	2557.097	.110	.191	.119	-.043	.425

There are many methods for measuring error, as a fact that the methods can lead to Inaccurate forecasting results, it is better to choose more than one [27]. The prediction accuracy of model performance can be measured by well known statistical methods such as Mean Absolute Percent Error (MAPE) and Root Mean Square Error (RMSE). The MAPE and RMSE are widely used to represent natural measure of average model-performance error [28]. The error tests results by two methods show that they are reasonable as presented in table (3). The overall comparison results between actual measured values of discharge and flow profile with those predicted by simulation outputs lead to accept them for further analysis.

Table (3) Percentage Error Test

Measure	MAPE	RMSE	R^2
2- $Q_{weir} + Q_{gate}$	0.241505	0.137704	0.9994
3- Flow profile	1.914284	0.420172	0.9836

6-Results and Discussion

The flow passing Semi-Circular structure separates into two parts, measuring the discharge of each part practically is difficult. The simulation outputs allow predict the discharge of each part. The increase of incoming upstream discharge is reflected in flow depth. This increase in flow depth upstream causes rapid increase in weir discharge than it does to the gate discharge. The rapid discharge increase of weir is a whole Physical phenomenon process in which the separation zone of the two parts of flow is changed. The height of the separation zone (H sep) reduced as the incoming discharge increases. Figure (2) illustrate streamline directions and show how the height of the separation zone from the channel bed became less although the geometry is constant.

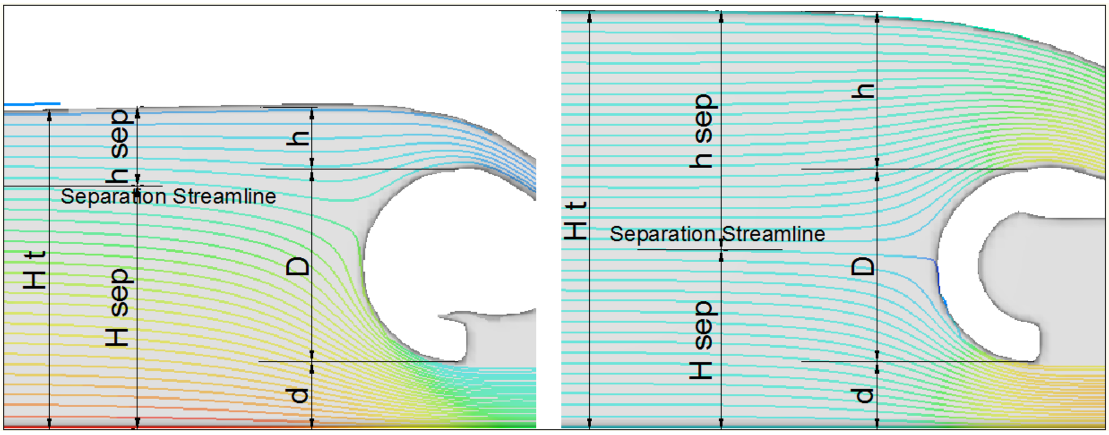

Figure (2) Streamlines visualization for combined flow and constant geometry

The process of separation and stabilization of streamlines into two different flow portions is state of matter to fit the conditions of weir flow and gate flow. The percentage of weir flow related to gate flow (Q weir/Q gate) is reflected on the depth of separation to depth of flow upstream (H sep/ H t). Figure (3) shows the reduction in separation depth to fit the condition of increasing flow of weir, in opposite way the height of upper portion (h sep) increases. It is worth noting that the equal flow rate of weir and gate reflect equal portion depths (h sep =H sep =0.5 Ht) as in figure (3 a and b).

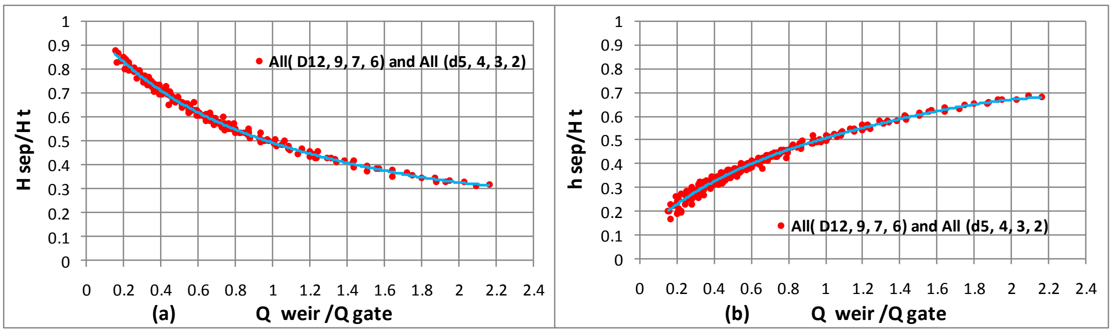

Figure (3) Variation of separation depths to the flow portions

The formation of streamlines in such combined flow is not as that in sold traditional weir. This flow starts to separate at a distance varies between 70 and 85 cm upstream to satisfy flow condition. The separation position is related to total head (Ht), structure diameter (D) and gate opening (d), the decrease in the values of last two geometrical parameters (D and d) shifts the separation position farther about 19% and 9% respectively. Semi-cylindrical weir subjected to combined flow performs better than the performance of sharp crested weir affected by the same head, this weir passes at list 33% more discharge than does traditional weir as it presented in figure (4 a and b). Experimental studies of openings in weir body located under weir crest increase the performance of the weir and so the total channel conveyance. [29], [30]

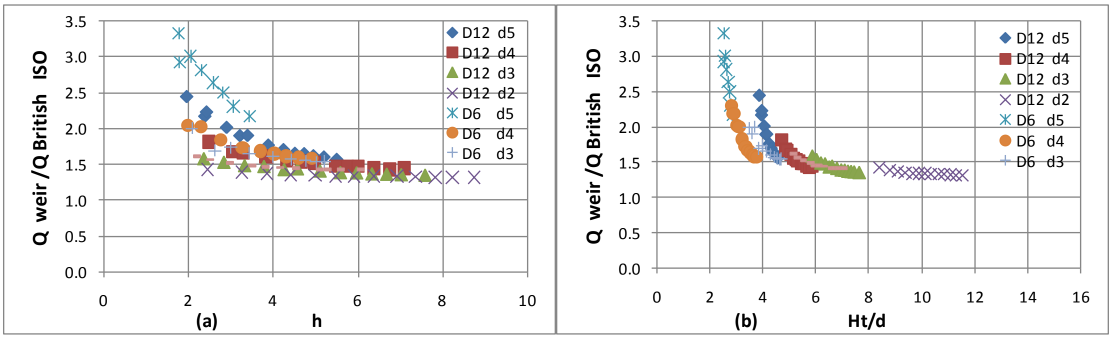

Figure (4) Semi-cylindrical weir performance subjected to combined flow

The flow from the gate is not only subjected to flow separation put it is also subjected to submergence directly behind the gate due to water falling from the weir. This state condition makes gate discharge less than free low. Free flow has been calculated on the base of Shayan et al [18] equation for radial gate. A comparison with the discharge of gate in combined flow is less about 70% to 90% within the range of this experiment, figure (5) presents this reduction.

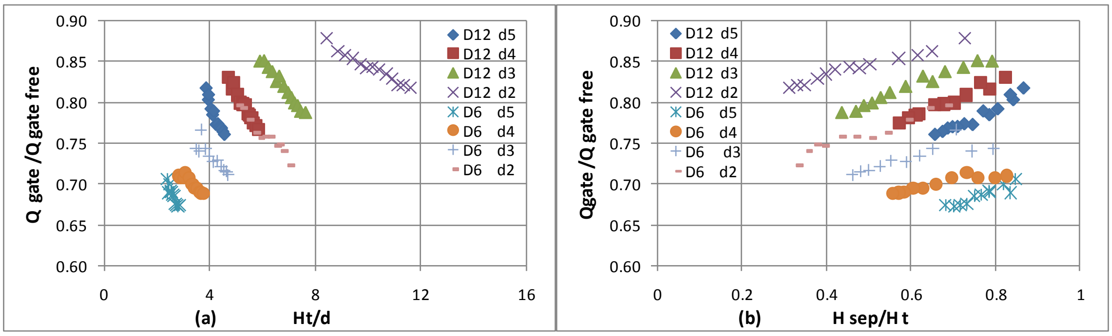

Figure (5) Variation of relative gate discharge

The calculated values of discharge coefficients for each weir and gate are presented in figure (6 a, b) respectively, the coefficient for the weir increases with increase of h/(D+d) and varies between 0.6 and 1.15, while for the gate decreases with increase of (Ht/d), it value varies between 0.6 and 0.77. This is within the range of 0.52 and 0.8 which investigated by Shayan et al [18], while (Salmasi et al. [31] found the coefficient of discharge varies between 0.58 to 0.67 for radial gate without a sill.

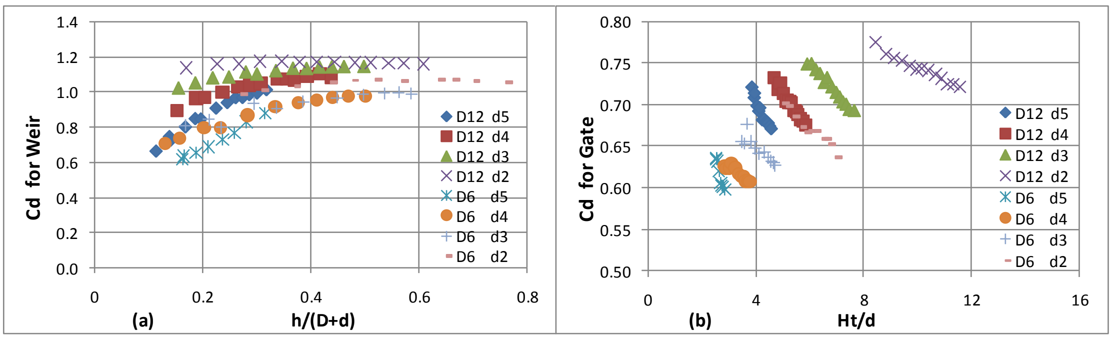

Figure (6) Variation of Cd for weir and gate with dimensionless major parameters

The dimensionless discharge is highly correlated with dimensionless parameters in equation (3), the highest Pearson correlation values are 0.988 and 0.938 for (h/d) and (Ht/d) respectively, other parameters are also have high correlation values exceed 0.723. The dimensionless discharge $\left(\frac{Q_{total}}{bd\sqrt{2gd}}\right)$ is presented with two majors dimensionless parameters in figure (7). It is useful to mention that semi cylindrical performs better when its diameter is larger for the same relative depths, this matter may be related to the gradual curvature of streamlines subjected to larger diameter. It also shows the increasing variation of dimensionless discharge with increasing values of (H/d), it is useful to note that for a constant value of (Ht/d) and for certain value of (d), then the smaller diameter of semi cylinder gives higher head over the weir and then the performance is better, as shown in figure (7 b).

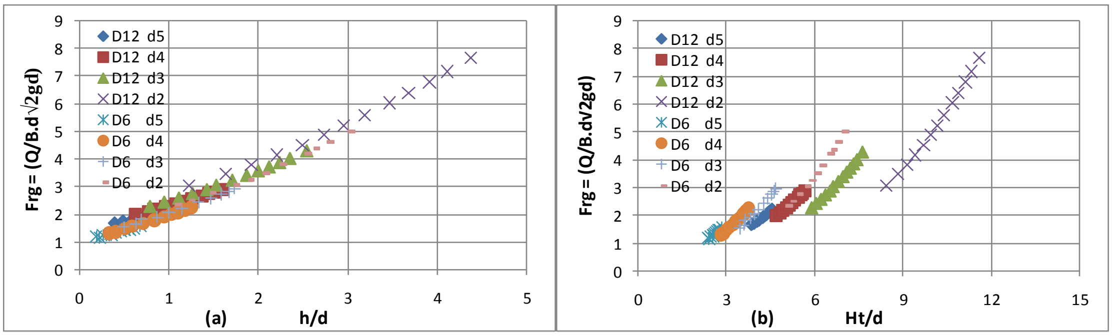

Figure (7) Variation dimensionless discharge with (h/d) and (Ht/d)

The statistical regression has been done to find a prediction mathematical models for some useful dimensionless quantities such as discharge coefficients of weir and quite separately when they acts in combine flow, also it is useful to forward the prediction models of relative discharge of weir to the gate and the model of prediction total dimensionless discharge which is listed as follows:-

$$Cd_{weir} = -0.183 + 0.167\frac{H_t}{d} + 4.715\frac{h}{H_t} - 0.407\frac{h}{d} - 1.009\frac{h}{(D+d)} \quad , R^2 = 0.911 \quad \ldots\ldots(9)$$

$$Cd_{gate} = 0.552 + 0.068\frac{(D+d)}{h} + 0.027\frac{H_t}{d} \quad , R^2 = 0.872 \quad \ldots\ldots(10)$$

$$\frac{Q_{weir}}{Q_{gate}} = 0.371 + 0.582\frac{h}{d} - 0.082\frac{H_t}{d} + 2.514\frac{h}{(D+d)} - 3.735\frac{h}{H_t} \quad , R^2 = 0.997 \ldots\ldots\ldots(11)$$

$$\frac{Q_{total}}{bd\sqrt{2gd}} = -1.441 + 1.576\frac{h}{d} - 8.134\frac{h}{H_t} + 2.802\frac{H_t}{(D+d)} + 0.046\frac{H_t}{d} \quad , R^2 = 0.998 \ldots(12)$$

7-Conclusion

The combined flow passing semi-cylindrical structure has been simulated and compared with experimental work. Even though this work has its limitations, within that the following findings can be listed:-

1- The location height of separation zone into two portions reduces as the incoming discharge increases and it is equal to half incoming discharge depth to satisfy two equal portions of flow.
2- The streamlines tend to take upward or downward directions at distance between 70cm and 85 cm from the structure, this distance increases with the decrease of each gate opening and structure diameter.
3- The overflowing flow from the weir in combined flow condition higher than that of traditional weir by at least 33%.
4- The outflow from the gate in combined flow condition is less that free flow about 70% to 90% for the same total head.
5- The discharge coefficient for the weir varies between 0.6 and 1.15, while for the gate between 0.6 and 0.77.

CONFLICT OF INTERESTS.

- There are no conflicts of interest.

References

[1] G. L. Kouyi, P. Bret, J. Didier, B. Chocat, and C. Billat, ‘The use of CFD modelling to optimise measurement of overflow rates in a downstream-controlled dual-overflow structure’, *Water Sci. Technol.*, vol. 64, no. 2, pp. 521–527, 2011.

[2] Y. Jia, S. Scott, Y. Xu, S. Huang, and S. S. Y. Wang, ‘Three-Dimensional Numerical Simulation and Analysis of Flows around a Submerged Weir in a Channel Bendway’, *J. Hydraul. Eng.*, vol. 131, no. 8, pp. 682–693, 2005.

[3] M. Fujihara, M. Akimoto, and M. Izumi, ‘3-D Numerical Simulations of Pool-and-Weir Fishways Using VOF’, *J. Rainwater Catchment Syst.*, vol. 14, no. 1, pp. 69–76, 2008.

[4] R. M. A. E. Rady, ‘2D-3D Modeling of Flow Over Sharp-Crested Weirs’, *J. Appl. Sci. Res.*, vol. 7, no. 12, pp. 2495–2505, 2011.

[5] Y. T. Zerihun and J. D. Fenton, ‘One-dimensional simulation model for steady transcritical free surface flows at short length transitions’, *Adv. Water Resour.*, vol. 29, pp. 1598–1607, 2006.

[6] S. A. M. Al-Hashimi, H. M. Madhloom, R. M. Khalaf, T. N. Nahi, and A. A.-A. Nadhir, ‘Flow over Broad Crested Weirs: Comparison of 2D and 3D Models’, *J. Civ. Eng. Archit.*, vol. 11, pp. 769–779, 2017.

[7] L. Abrari, N. Talebbeydokhti, and S. Sahraei, ‘Investigation of Hydraulic Performance of Piano Shaped Weirs Using Three Dimensional Numerical Modeling’, *Iran. J. Sci. Technol. Trans. Civ. Eng.*, vol. 39, no. C2+, pp. 539–558, 2015.

[8] H. Rahimzadeh, R. Maghsoodi, H. Sarkardeh, and S. Tavakkol, ‘Simulating Flow Over Circular Spillways by Using Different Turbulence Models’, *Eng. Appl. Comput. Fluid Mech.*, vol. 6, no. 1, pp. 100–109, 2012.

[9] S. A. M. Al-Hashimi, H. M. Madhloom, and T. N. Nahi, ‘Experimental and Numerical Simulation of Flow over Broad Crested Weir and Stepped Weir Using Different Turbulence Model’, *J. Eng. Sustain. Dev.*, vol. 21, no. 02, pp. 28–45, 2017.

[10]L. Jiang, M. Diao, H. Sun, and Y. Ren, ‘Numerical Modeling of Flow Over a Rectangular Broad-Crested Weir with a Sloped Upstream Face’, *Water (Switzerland)*, vol. 10, no. 11, pp. 1–12, 2018.

[11]S. L. Huang and C. Ng, ‘Hydraulics of a submerged weir and applicability in navigational channels : Basic flow structures’, *Int. J. Numer. Methods Eng.*, vol. 69, pp. 2264–2278, 2007.

[12]T. mohammed Abdulatif and R. R. Rhmaan, ‘Modeling of One Dimensional Flow over Semicircle Crested Weir Profile Using (Hec-Ras) Program’, *Diyala J. Eng. Sci.*, vol. 10, no. 2, pp. 48–59, 2017.

[13]F. M. Henderson, *Open Channel Flow*. New York: Macmilan Pubishing co., Inc., 1966.

[14]V. Te Chow, *Open-channel hydraulics*. Tokyo: International Student Edition, Kogakusha Company, LTD, 1959.

[15]K. Subramanya, *Flow in Open Channels*, Third Edit. New Delhi: Tata McGraw-Hill, 2009.

[16]British Standards Institute, *BS ISO 1438 : 2008 : Hydrometry-Open channel flow measurement using thin-plate weirs*. London, 2008.

[17]D. Lozano, L. Mateos, G. P. Merkley, and A. J. Clemmens, ‘Field Calibration of Submerged Sluice Gates in Irrigation Canals’, *J. Irrig. Drain. Eng.*, vol. 135, no. 6, pp. 763–772, 2009.

[18]H. K. Shayan, J. Farhoudi, and R. Roshan, ‘Estimation of flow discharge under the sluice and radial gates based on contraction coefficient’, *IJST, Trans. Civ. Eng.*, vol. 38, no. C2, pp. 449–463, 2014.

[19]G. Arvanaghi, H. and Mahtabi, ‘Hydraulic Characteristics of Rectangular Combined Sharp-Crest Weir-Gate’, *Adv. Environ. Biol.*, vol. 8, no. 21, pp. 32–38, 2014.

[20]H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, Second Edi. Edinburgh Gate: Pearson Education Limited England, 2007.

[21]Flow-3D, *Flow-3D Documentation, Release 10.1.0*. 2012.

[22]M. S. Akoz, V. Gumus, and M. S. Kirkgoz, ‘Numerical Simulation of Flow over a Semicylinder Weir’, *J. Irrig. Drain. Eng.*, vol. 140, no. 6, p. 04014016, Jun. 2014.

[23]G. Mahtabi and H. Arvanaghi, ‘Experimental and numerical analysis of flow over a rectangular full-width sharp-crested weir’, *Water Sci. Eng.*, vol. 11, no. 1, pp. 75–80, 2018.

[24]S. Haun, N. R. B. Olsen, and R. Feurich, ‘Numerical Modeling of Flow over Trapezoidal Broad-Crested Weir’, *Eng. Appl. Comput. Fluid Mech.*, vol. 5, no. 3, pp. 397–405, 2011.

[25]R. Maghsoodi, M. S. Roozgar, H. Sarkardeh, and H. M. Azamathulla, ‘3D-Simulation of Flow Over Submerged Weirs’, *Int. J. Model. Simul.*, vol. 32, no. 4, pp. 237–243, 2012.

[26]L. Abrari, N. Talebbeydokhti, and S. Sahraei, ‘Investigation of Hydraulic Performance of Piano Shaped Weirs Using Three Dimensional Numerical Modeling’, *IJST, Trans. Civ. Eng.*, vol. 39, no. C2+, pp. 539–558, 2015.

[27]M. V. Shcherbakov, A. Brebels, N. L. Shcherbakova, A. P. Tyukov, T. A. Janovsky, and V. A. Kamaev, ‘A Survey of Forecast Error Measures’, *World Appl. Sci. J.*, vol. 24, pp. 171–176, 2013.

[28]C. J. Willmott and K. Matsuura, ‘Advantages of the mean absolute error (MAE) over the root mean square error (RMSE) in assessing average model performance’, *Clim. Res.*, vol. 30, pp. 79–82, 2005.

[29]H. I. Mohamed, G. Abozeid, and S. M. Shehata, ‘Hydraulics of clear and submerged overfall weirs with bottom circular-openings’, *Ain Shams Eng. J.*, vol. 1, no. 2, pp. 115–119, 2010.

[30]N. Y. Saad and E. M. Fattouh, ‘Hydraulic characteristics of flow over weirs with circular openings’, *Ain Shams Eng. J.*, vol. 8, no. 4, pp. 515–522, 2017.

[31]F. Salmasi, M. Nouri, and J. Abraham, ‘Laboratory Study of the Effect of Sills on Radial Gate Discharge Coefficient’, *KSCE J. Civ. Eng.*, pp. 1–9, 2019.

تحليل محاكاة التدفق المشترك فوق وتحت الهيكل شبه أسطواني

شاكر عبد اللطيف جليل

هندسة الموارد المائية، كلية الهندسة، جامعة دهوك، العراق

shaker.abdulatif@uod.ac

الخـــــلاصة:

يمكن للتدفق المشترك الناتج عن وضع منشأ نصف أسطواني يرتفع عن قعر القناة من حل مشكلة المواد المعلقة وترسبها. ويقلل الشكل شبه الأسطواني من انحناء خطوط التيار مما تنعكس ايجابيا على أدائه. لدراسة كيفية أداء هذا الشكل من المنشآت، تم إجراء التجارب ثم محاكاتها. تم اختبار اربعة نماذج المختبرية مختلفة الأقطار وضعت على أربعة ارتفاعات عن قعر النقنات لتشكيل فتحات بوابة. وقد تم انشاء نفس الهياكل المادية في البرمجيات التجارية، FLOW-3D®، واستخدام نموذج الاضطراب RNG k- ε. تم التحقق من صحة اداء النموذج بواسطة المقارنة بين سطح الحر للماء وكذلك التصريف. تشير نواتج المحاكاة إلى أن منطقة فصل بين خطوط التيار المتجوهة نحوالهدار تبتعد عن جسم المنشآت كلما صغرالقطر وقلت فتحة البوابة، كما أن سمك الماء المتوجه نحو الهدار مرتبط ايضا بمقدار التصريف القادم. يميل موقع منطقة الفصل إلى الانخفاض نحو القعرعندما يكون هناك زيادة في التدفق القادم، حيث تصبح عند نصف العمق الكلي عندما يتساوى التصريف الخارج من البوابة ومن الهدار. يظهر الهدار في هذا النظام أداء أفضل من الهدار التقليدي بنسبة ٣٣٪ على الأقل، في حين أن تدفق البوابة أقل من التدفق الحر لنفس العمق الكلي للماء وبنسبة ٧٠٪ إلى ٩٠٪. ضمن حدود هذا العمل، تم اقتراح نموذجين رياضيين للتنبؤ بمعامِل التفريغ للهدار والبوابة على كل على انفراد، بالإضافة إلى نموذج للتنبؤ بالتصريف النسبي للهدار إلى البوابة، كما اقترح نموذج رياضي واحد للتصريف الكلي عديم البعد.

الكلمات الدالة: منشآت شبه أسطوانية؛ تدفق مشترك؛ انفصال خطوط التيار؛ أجزاء التدفق؛ هدار والبوابة.