Journal of Civil Engineering and Urbanism

Volume 4, Issue 4: 397-401 (2014) ISSN-2252-0430

Three Dimensional Simulation of Flow for Semi Cylindrical Weirs Using Fluent Software

Aylar Samadi, Sasan Nejati, Sajedeh Haji Azizi, Behrooz Oghati Bakhshayesh

MSc Student of Hydraulic Structures, Department of Water Sciences Engineering, Agriculture Faculty, Tabriz University, Tabriz, Iran

*Corresponding author's Email: behroozob@gmail.com

ABSTRACT: The weirs are one of the important and prevalent items to measure discharge in open channels and they are used in most of the water distribution networks. Because of the higher discharge coefficient, simplicity in design, more stability of the flow on it in bigger scales, and lower costs rather to the other types of weirs is more superior. In the present study the pattern of the flow on semi cylindrical weirs simulated in three dimension via Fluent software. For simulating the multiphase flow, the Volume Of Fluid (VOF) model is used and in order to simulate the turbulence of flow, k-ε turbulence model is used, and extracted results of the numerical models are evaluated to experimental data extracted from physical models. In general considering the error percentage it could be said that the result extracted from Fluent software has a good adaptation to the experimental data for the same model. It is worth noting simulating the flow has done for three different radiuses of semi cylindrical weirs.

Keywords: Flow pattern, Semi Cylindrical Weir, 3D simulation, K-ε Turbulence Model, Volume of Fluid Model, Fluent software

ORIGINAL ARTICLE

Received 09 Jun. 2014
Accepted 16 Jul. 2014

INTRODUCTION

The recent developments in computer software has advanced the use of computational fluid dynamics (CFD) in analyzing flow over spillways. Some recent works are due to Unami et al. (1999), Savage and Johnson (2001), and Ho et al. (2003). Unami et al. (1999) developed a two-dimensional numerical simulation for spillway flows. They found a reasonable agreement with experimental data. Savage and Johnson (2001) did a two-dimensional simulation of flow over an ogee spillway using a commercial CFD code (Flow-3D). They found a good agreement with experiments for both pressures and discharge. In comparison with two-dimensional models, there are fewer applications of three dimensional (3D) models for free overflow spillways. Ho et al. (2003) did two- and three-dimensional CFD modeling of spillway behavior under rising flood levels and validated the results using published data. The model was also applied to study several spillways in Australia.

CFD complements experimental and theoretical fluid dynamics by providing an alternative cost-effective means of simulating real flows. However, the usefulness of a mathematical model depends on the validity of the governing equations and the numerical methods. In order for CFD to become more reliable and acceptable as a design tool, numerical studies must be carefully validated with experimental results. Because hydraulic design of spillways is a new application area for CFD, it requires especially careful validation.

In general any obstacle in flow direction causing upstream head levels increment and simultaneously increasing flow velocity on the obstacle called weir. In general it could be said that weirs are one of the important items in hydraulic and dam construction projects for different aims such as discharge measurements, adjusting upstream head levels for intakes and turnouts, and so on.

Weirs have different types including semi cylindrical type. Semi cylindrical weir has many application in irrigation water, supply water and waste water, conveying systems, and discharge or distribution networks. The superiority of semi cylindrical weirs to the other types of weirs or other controlling structure are higher discharge coefficient, simplicity in design and construction, and the most important one lower costs in big scales (Chanson H. and Montes JS., 1998).

Semicircular weir is a new type of construction developed in 1990's. It has many advantages compared with other types of weirs. So this type of weir is used in hydraulic engineering recently. It is very important to investigate the behaviors of the flow over the weir, the bottom shear stress and the local scour around the semicircular weir.

There are a number of numerical studies on water flow over obstacles, most of which are based on the inviscid model Lamb (1945), Forbes and Schwartz (1982), Dias and Broeck (1989). for the condition of small deformed free surface, the motion of the free surface can be simulated by an inviscid model. When the free surface undergoes a large deformation with turbulence, the inviscid model is invalid. The method of Volume of Fluid (VOF) (Hirt and Nichols, 1981) developed in 1980's is suitable to simulate the condition of large deformed free surface. This method is used to investigate the hydraulic jump and breaking waves (Liu and Chen, 1993) and (Bradford, 2000). There are also some experimental studies on the flow over the weir (Fadda and Raad, 1997).

To cite this paper: Samadi A., Nejati S., Haji Azizi S., Oghati Bakhshayesh B. 2.14. Three Dimensional Simulation of Flow for Semi Cylindrical Weirs Using Fluent Software. *J. Civil Eng. Urban.*, 4 (4): 397-401.
Journal homepage: http://www.ojceu.ir/main/

Most of them considered a simple geometrical shape and rectangle meshes were employed. There is little investigation on the flow over a semicircular weir, and scarcely any detailed and systematical work that studied the flow patterns over a semicircular weir, the separation and reattachment of the flow over the weir and the distribution of the bottom shear stress.

Based on studies of Bazin (1898) progresses in order to improve discharge volume has made which at last resulted in circular crested weirs design (Bazin, 1898). Creager developed Bazin's studies in order to define ogee spillway's profile. He also carried out experiments about circular crested weirs which later on used in design of *Pont* dam in Burgundy-France (Savage and Johnson, 2001).

Liu et al. (2002) modeled circular crested weirs in two dimension. The considered interval's range for them was 50D to 2.2 D (D is the weir diameter). The method to solve the equation was control volume method and in order to simulate multiphase flow they used volume of fluid (VOF) model and Geo-Reconstructed Model. In order to analysis of turbulence of flow standard k-ε model used and the numerical results had a good correlation with experimental data.

Dargahi (2004) simulated ogee spillway in three dimension using Fluent software and evaluated extracted data to the experimental data. He built ogee spillway model in flume with length of 4 meters, width of 0.403 meter and depth of 0.6 meter and he checked water surface profile for different points. It is worth to mention that Dargahi (2002) used standard k- ε model and ReNormalization Group model (RNG) for simulating turbulent flow and considering software the relative error he figured out that Fluent software has the ability to simulate ogee spillway in three dimension.

Esmaili et al. (2010) conducted experiments on different models of cylindrical weirs and measured pressure and velocity on these models and evaluated to the extracted data from Fluent software. The results of Esmaili et al. (2010) studies showed that the Fluent software has the ability to simulate cylindrical weir models, because the extracted data from numerical model for flow pattern on cylindrical weir had a good correlation with experimental data extracted from physical model.

Despite the studies conducted, there is not any comprehensive study token place on numerical simulation of semi cylindrical weirs, so that this study tries to calculate discharge coefficient by use of Fluent software for this type of weirs. In order to simulate and evaluate numerical model, experimental data of Abdul-latif and Abdul-mohsen (2010) is used.

MATERIAL AND METHODS

In present study the Fluent software based on continuity equation, momentum equation, and application of turbulence model is used to simulate flow for semi cylindrical weirs in three dimension and the results obtained from modeling compared to the experimental data. After selecting the optimum mesh for simulating the flow in multiphase form of the fluid model, the k-ε, RNG turbulence model used to simulate the flow and the results of the numerical solutions are evaluated to the experimental data extracted from physical model. It is noteworthy to mention that for selecting the optimum mesh, standard Relative Average Error (RAE) relation is used.

$$(\mathrm{RAE})_y\% = \frac{\sum_{i=1}^{N}|(y_{Pi}-y_{ni})|}{\sum_{i=1}^{N} y_{Pi}} \times 100 \qquad [1]$$

Fluent software

The Fluent software is one of the most perfect and applied Computational Fluid Dynamics (CFD) models which written in C programming language and converts the governing equations to algebraic equations using finite volume method and solve them. This software simulate both in two dimension and three dimension and within different modelling method and advanced turbulence modelling methods, it is possible to simulate almost all of the flow models or any kinds of hydraulic phenomena.

Governing equations of flow

The continuity equation and momentum equation are governing equations of the flow which for turbulent flow with incompressible fluid and constant viscosity and density are mentions in following (Navier-Stokes equations)(Versteeg, H.K., Malalasekera, W., 1995).

$$\rho \frac{\partial}{\partial x_j}(u_i u_j) = -\frac{\partial P}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \qquad [2]$$

$$\frac{\partial u_i}{\partial x_j} = 0 \qquad [3]$$

$$\tau_{ij} = \left[\rho(\nu+\nu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] - [\tfrac{2}{3}\rho(k+\nu_t)\frac{\partial u_i}{\partial x_j}\delta_{ij}] \qquad [4]$$

In these equations u_i is the velocity factor in x_i direction, P is total pressure, ρ is fluid density, g_i is the gravity acceleration in x_i direction, τ_{ij} is the stress tensor, ν is the cinematic viscosity, v_t is the turbulence viscosity, k is the kinematic energy of the turbulence, and δ_{ij} is the Kronecker delta.

Applied turbulence models

k-ε turbulence model is one of the most effective methods in simulating. This model has three methods which are standard, RNG, and realizable. The RNG method is mostly used in vertical or rotational flows and unlike the standard method it use the analytical or algebraic equations. In order to define the turbulence in curved flow fields or in complex flow fields it is preferred to use RNG model, because it has high accuracy in low Reynolds's number. In flows which the flow lines are curved, RNG models and realizable model would have better performance rather than the standard model (Chen et al., 2002).

Fluid volume model

One of the best methods to define accurate water surface levels is Volume of Fluid (VOF) method. This method designed for two types of fluid which could not get mixed and the interface between the fluid are considered.

Experimental studies

Abdul-latif and Abdul-mohsen (2010) experimentally studied on semi cylindrical weirs in a flume with 5 meters length, 0.076 meter width, and 0.15 meter depth in Baghdad University's hydraulic laboratory.

In this experiments semi cylindrical weirs with radius of 4, 6 and 7 centimetres were used.

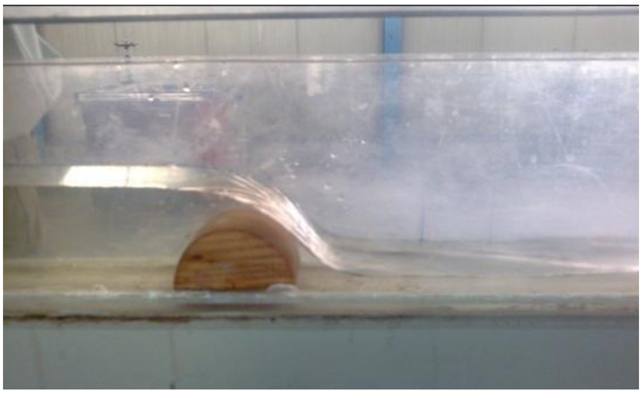


Figure 1. Flow on semi cylindrical weir in experimental flume (Abdul-latif and Abdul-mohsen, 2010)

The steps of working with Fluent software

In first step the geometrical model designed using a modelling software and after using a proper meshing for model, the suitable and proper equations to simulate the model are selected and in next step the boundary conditions are defined to software. Then within a proper solution algorithm such as algorithms of velocity, pressure, and momentum equations, the software ran.

In this study models are meshed via gambit software. The Figure 2 shows the netting of the models. The gambit software has different patterns for meshing the models including triangular, quadratic, and triangular/quadratic simultaneously. In this study for meshing the model quadratic pattern with map type is used and also for netting the volume the Hex pattern with map type is used. At last in order to choose the best meshing, multiple meshing are evaluated. The result of this evaluation is mentioned as Table 1. It is observable that within increasing the meshing count, the results would be close to the reality and after that increasing the mesh count would not be efficient and it would only make the process last long. So that in this study the meshing count of 15400 chosen for the optimum meshing.

Table 1. Error percentage in selecting optimum mesh

Mesh count	7500	10300	15400	23000
$(RAE)_y$%	9.6	7.4	3.04	3.5

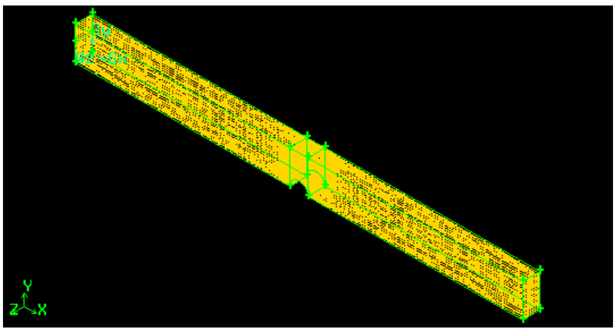


Figure 2. Weir model and meshes in gambit software (radius of the semi cylindrical weir is 4cm)

In present study the data extracted from three dimension simulation of flow on semi cylindrical weirs is evaluated to experimental data extracted from a physical model and also evaluated to the relation provided by Heydarpour et al (1383) and Esmaili et al (1388).

RESULTS AND DISCUSSION

Water surface profiles

Figure 3 is one of the water surface profiles simulated via Fluent software for semi cylindrical weirs within 4 centimetres of radius.

Evaluating the data extracted from numerical simulation and physical model the relative error of different discharges and different radiuses of semi cylindrical weirs would be calculated. Table 2 shows the results of calculations.

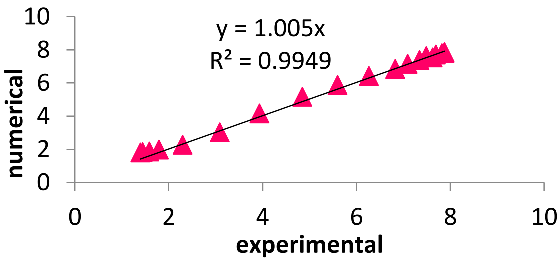
Graph 1 shows that evaluation of data extracted from numerical simulation via Fluent software and experimental data for semi cylindrical weir within 4 centimetres of radius and 1.098 (L/S) discharge rate. Focusing on graphs and relative error rate (Table 2) it could be expressed that there is a good correlation between numerical model data and experimental data.

Graph 2 and 3 respectively shows the water surface profile for weir within 6 centimetres of radius and 1.229 (L/S) discharge rate and for weir within 7 centimetres radius and 1.98 (L/S) discharge rate using both numerical model and physical model. The percentage of the error in Graph 2 is 2.2% and for Graph 3 is 3.1%.

Figure 3. Flow on the semi cylindrical weir with radius of 4 cm

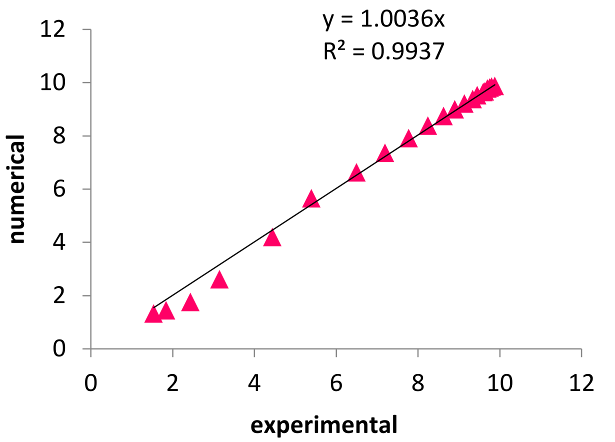
Table 2. The values of relative error for simulation results and experimental results

Radius (cm)	Discharge (L/S)	RAE (%)
4	1.098	2.4
4	0.972	3.2
4	0.780	2.8
4	0.714	2.2
4	0.620	2.8
6	1.229	2.2
6	1.209	3.3
6	1.083	2.8
6	0.980	2.2
6	0.933	3.9
7	1.25	4.1
7	1.9868	3.1
7	0.8282	3.3
7	0.7587	3.3
7	0.6622	3.3

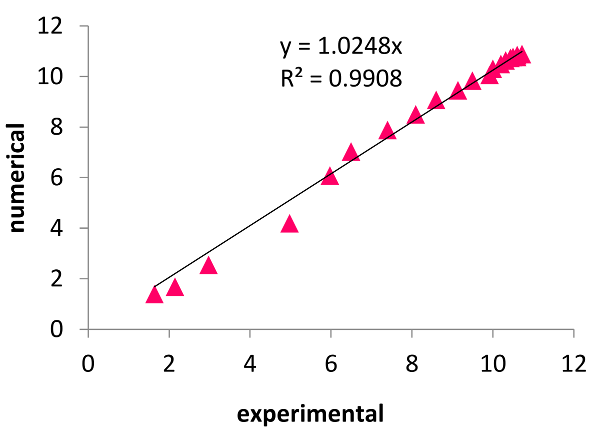


Graph 1. Water surface profile in both numerical an experimental method for semi cylindrical weir with 4 cm radius and 1.098 (L/S) discharge

To cite this paper: Samadi A., Nejati S., Haji Azizi S., Oghati Bakhshayesh B. 2.14. Three Dimensional Simulation of Flow for Semi Cylindrical Weirs Using Fluent Software. *J. Civil Eng. Urban.*, 4 (4): 397-401.
Journal homepage: http://www.ojceu.ir/main/



Graph 2. Water surface profile in both numerical an experimental method for semi cylindrical weir with 6 cm radius and 1.229 (L/S) discharge



Graph 3. Water surface profile in both numerical an experimental method for semi cylindrical weir with 7 cm radius and 1.25 (L/S) discharge

Table 3 shows the correlation coefficient (R^2) and regression equation for three different radiuses and different discharge rates.

Table 3. the relative errors of the experimental and numerical modelling results

Radius (cm)	Discharge (L/S)	R^2	Relation
4	1.098	0.995	y = 1.005 x
4	0.972	0.992	y = 1.008 x
4	0.780	0.993	y = 0.9896 x
4	0.714	0.994	y = 1.0115 x
4	0.620	0.992	y = 1.0045 x
6	1.229	0.993	y = 1.0036 x
6	1.209	0.996	y = 1.0268 x
6	1.083	0.996	y = 1.0151 x
6	0.980	0.996	y = 1.0105 x
6	0.933	0.995	y = 1.0336 x
7	1.25	0.991	y = 1.0248 x
7	1.9868	0.992	y = 1.0152 x
7	0.8282	0.984	y = 1.0133 x
7	0.7587	0.994	y = 1.023 x
7	0.6622	0.995	y = 1.0254 x

CONCLUSION

Considering the extracted results it could be expressed that:

1-The extracted relative error shows a good similarity and good correlation between simulated data and experimental data. So that in order to lower the costs of physical models, it could be replaced with numerical models.

2-Choosing the best meshing both in 2D and 3D is one of the most determinative parameters in modelling by Fluent software. Choosing improper meshing could result in bigger relative error and inaccurate numerical model. So that the optimization of the mesh sizes and mesh count should be taken seriously.

3-The numerical model should be long enough to include the backwater effect on the canal and in the other word to include water profile almost completely.

REFERENCES

Abdul-latif, T. and Abdul-mohsen, M. (2010), Calibrating the discharge coefficient of semicircular crested weir, Eng. & Tech. Journal, Vol. 28, No. 24, 2010.

Bazin H. (1898). Expériences Nouvelles sur l'Ecoulement par Déversoir ('Recent Experiments on theFlow of Water over Weirs.'). Mémoires et Documents, Annales des Ponts et Chaussées, Paris, France. Sér 7. No. 15. P. 151 to 264.

Chanson H. and Montes J.S. (1998). Over flow characteristics of circular weirs: Effects of in flow conditions. J Irrig and Drain Eng, ASCE. No. 124. P. 152 to 161.

Chen، Q., Dai, G.، and Liu, H. (2002)، "Volume of fluid for turbulence numerical simulation of stepped spillway overflow"، J of Hydraulic Eng. P. 683 to 688.

Creager WP. (1917). Engineering of Masonry Dams. John Wiley & Sons, New York, USA.

Dargahi B, 2004. Experimental study and 3D numerical simulations for a free over flow spillway. Hydraul Engin, ASCE. No. 132. P. 899 to 907.

Dins F, Vanden Broeck JM. (1989). Open channel flow with submerged obstacles. *J Fluid Mech*,. No. 206. P. 155 to 170

Esmaili K., Naghavi B., Kourosh vahid F., Yazdi J,. (2010) "experimental and numerical study of flow pattern on cylindrical weirs" soil and water Journal (agriculture science and industry) Vol. 24 No.1 P. 166 to 179.

Fadda D, Rand PE. (1997). Open channel flow over submerged obstructions: an experimental and numerical study. *J Fluids Eng*,. No. 119. P. 906 to 910

Forbes LK, Schwartz LW. (1982). Free-surface flow over a semicircular obstruction. *J Fluid Mech*,. No. 114. P. 299 to 314

Hirt CW, Nichols BD. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. J *Comput Phys*,, No 39. P. 201 to 225

Ho, H., Boyes, K., Donohoo, S., and Cooper, B. (2003). "Numerical flow analysis for spillways." *Proc., 43rd ANCOLD Conf.*, Hobart, Tasmania, P. 24 to 29.

Lamb H. Hydrodynamics. Dover, 1945

Liu QC, Chen CT. (1993). Numerical study of turbulence characteristics in hydraulic jumps. *Chinese J Hydraulics*,. No. 1. P. 1 to 10

Liu C, Huhe A and Wenju MA, (2002). Numerical and experimental investigation of flow over a semicircular weir. Acta Mechanica Sinica, No. 18. P. 594 to 602

Savage BM and Johnson C, (2001). Flow over ogee spillway: Physical and numerical model case study. J. Hydraul Engin, ASCE, No. 127. P. 640 to 649.

Unami, K., Kawachi, T., Babar, M. M., and Itagaki, H. (1999)."Twodimensional numerical model of spillway flow." *J. Hydraul. Eng.*, No. 125(4). P. 369 to 375.

Versteeg, H.K., Malalasekera, W., (1995), "An introduction to computational fluid dynamics, The Finite Volume Method", Longman.