INTERNATIONAL JOURNAL OF OPTIMIZATION IN CIVIL ENGINEERING
Int. J. Optim. Civil Eng., 2014; 4(4):549-560

CFD SIMULATION OF FLOW OVER CONTRACTED COMPOUND ARCHED RECTANGULAR SHARP CRESTED WEIRS

A. Samadi*,† and H. Arvanaghi
Department of Water Engineering, Faculty of Agriculture, University of Tabriz

ABSTRACT

Measurement of discharge in open channels is one of the main concerns in hydraulic engineering. The structures used for this aim should be accurate, economical and easy to use. Weirs are among the oldest and most convenient hydraulic structures that have been used in both laboratory and field for flow measurement in open channels. Due to limitations of simple sharp crested weirs, recently compound sharp crested weirs have attracted great attention of civil engineers. In general, use of compound sharp crested weirs can be an appropriate solution when the discharge should be measure accurately with a reasonable sensitivity over a wide range of flows. The aim of this research is three-dimensional simulation of flow on contracted compound arched rectangular sharp crested weirs by using FLUENT software. For multiphase flow simulation, VOF method is used and for simulation of turbulent flow, RNG k-ε turbulence model is used and the result of numerical model is compared with experimental data. The results of this study indicate that; FLUENT simulate flow on contracted compound arched rectangular sharp crested weirs with high accuracy and we can use this software for determine the discharge coefficient on contracted compound weirs.

Received: 12 May 2014; Accepted: 25 November 2014

KEY WORDS: contracted compound weir; discharge coefficient; FLUENT; RNG k-ε; 3D simulation; VOF.

1. INTRODUCTION

Water measurements play a pivotal role in hydraulic and environmental management. There

---

*Corresponding author: Department of Water Engineering, Faculty of Agriculture, University of Tabriz
†E-mail address: a_samadi10@yahoo.com (A. Samadi)

are different types of structures available for flow measurements which the weirs are the most important of them. Weirs allow water to be routed through a structure of known dimensions, permitting flow rates to be measured as a function of depth of flow through the structure. Among different types of weirs, sharp crested weir has been widely used for discharge measurements in open channels. For a weir to be considered sharp crest, the top thickness of the crest and side plates should be between *1* and *2 (mm)*. If the plates are thicker than specified, the plate edges need to be beveled to an angle of at least *45°* to *60°* [3]. Sharp crested weirs are commonly used in hydraulic laboratories, industrial applications and irrigation systems.

Sharp crested weirs comprise the following two types: suppressed sharp crested and contracted sharp crested weirs. In suppressed sharp crested weirs; crest width is equal to bottom width of the channel, while contracted sharp crested weirs; reduces the channel width in the direction perpendicular to the water flow [3].

In many situations such as irrigation canals with wide range of variations in flow discharges, selecting of either rectangular or circular weir is erroneous and unsafe work. So researchers in order to increase the discharge coefficient and flow rate through the channel, have been designed compound sharp crested weirs. Compound sharp crested weirs have been widely used for measuring discharges with a reasonable sensitivity over a wide flow range. These types of weirs are combination of a lower part for handling the low or normal flow discharges and an upper part for measuring bigger flow discharges [6]. The compound weirs have various shapes.

In this study Mousavi's [5] experimental data and *FLUENT* software are used for simulation of flow on contracted compound arched rectangular sharp crested weirs. Arched rectangular sharp crested weir section; composed of a circle sharp crested weir at the bottom and a rectangular sharp crested weir in the upper part. Therefore in this section, descriptions about rectangular and circular sharp crested weirs would be provided:

1.2 Rectangular sharp crested weir

A rectangular notch, symmetrically located in a vertical thin plate, which is placed perpendicular to the sides and bottom of a straight channel, is defined as a rectangular sharp crested weir [3]. Researchers have conducted many studies to develop the flow rate equations in rectangular sharp crested weirs. Flow rate equations in sharp crested weirs usually are achieved by the integration of the flow elements strips over the weir crest. The conventional weir equation is considered in the form [3]:

$$Q = \frac{2}{3} C_d \sqrt{2g}.b.h^{\frac{3}{2}} \tag{1}$$

In which Q is the weir discharge, C_d is named as the discharge coefficient assumed to represent all effects that are not taken into consideration in the derivation such as viscous effects, streamline curvature due to weir contraction, three-dimensional flow structures in the vicinity of the weir plate and surface tension, g is gravitational acceleration, b is the weir width and h is the head over the weir.

1.2 Circular sharp crested weir

A circular (flow control) section placed in a vertical thin (metal) plate, which is located at right angle to the sides and bottom of a straight approach channel is defined as a circular thin plate weir [3]. When the driving head over the weir crest exceeds the diameter of the weir, the weir is expected to behave like an orifice. Circular weirs have the advantage that the crest can be turned and beveled precisely in a lathe [11].

Vatankhah [12], presented flow equation in circular sharp crested weirs by the integration of the flow elements strips over the weir crest. This equation is as follows:

$$Q = C_d .2\sqrt{2g}.\varphi(\eta).D.h^{\frac{3}{2}} \tag{2}$$

In Equation (2) D is diameter of the circular sharp crested weir, h is the head over the weir and C_d is the discharge coefficient. Value of $\varphi(\eta)$ is calculated from equation (3):

$$\varphi(\eta) = 0.1963\,\eta^{\frac{1}{2}}\,(\sqrt{1-0.2200\eta} + \sqrt{1-0.7730\eta}) \tag{3}$$

η is filling ratio of the weir and is expressed as $\eta = \dfrac{h}{D}$.

High cost and being time consuming laboratory studies, includes factors that force researchers to seek for ways which using them could simulate the flow in open channels with high accuracy. In recent years, CFD[1] becomes a powerful tool being used in areas like fluid flow, heat/mass transfer, chemical reactions and related phenomena by solving mathematical equations that govern these processes using a numerical algorithm on a computer.

Ramamurthy et al. [9] were among the researchers that simulated flow pattern in a simple rectangular sharp crested weir in the form of Three-dimensional by using *Flow3D* software. They used the volume of fluid method for simulation multiphase flow and used k- ε method to simulate the turbulence model. Their study results showed that the values obtained from *Flow3D* software has a high degree of accuracy, because numerical and experimental data match well with each other.

Rady [8] simulated free flow over the rectangular sharp crested weir using *Flow3D* software and in two-dimensional and three-dimensional modes. In numerical models; he used volume of fluid method and *RNG* k- ε turbulence model. Study results of Radi [8] showed that for different values of the weir height, H_t / P parameter (in which H_t is specific energy upstream of the weir and P is weir height), has the greatest impact on the flow rate. In addition, discharge coefficient which is calculated for rectangular sharp crested weirs is equal to constant value of 0.64.

Haun et al. [4] calculated water flow over a trapezoidal broad crested weir by two different CFD codes of *Flow3D* and *SSIM 2*, where first one uses volume of fluid method

---

[1] Computational Fluid Dynamics

with a fixed grid, while last one uses an algorithm based on the continuity equation and the Marker-and-Cell method. They compared the results with measurements from a physical model study using different discharges and they state that the deviation between the computed and measured upstream water level was between 1.0 and 3.5 %.

Arvanaghi and Nasehi Oskuei [2] at first measured water surface profiles over the rectangular sharp crested weir in laboratory and then modeled the flow over the weir by using *FLUENT* software and as two-dimensional. They have done studies on the suppressed rectangular sharp crested weir, with a crest width of 25 (cm) for three weir heights of 10, 15 and 20 (cm). Comparison of experimental and numerical results showed that; there is a good agreement between them and *FLUENT* software has the ability to simulated flow over the sharp crested weir with high accuracy. Arvanaghi and Nasehi Oskuei [2] have reached the conclusion that in the range of $h/P > 0.6$, $Fr > 0.2$ and $Re > 20000$ discharge coefficient for rectangular sharp crested weir is equal to a constant value of 0.7. Beyond these boundaries, C_d is not constant and it is not recommended to use a unique C_d for different flow conditions.

Namaee et al. [6] simulated the 3 dimensional flows on a broad crested side spillway in a numerical way. *RNG* k-ε model was used to simulate turbulent and fluid volume model to find free water surface. The simulation results were evaluated using experimental data and showed that existing numerical methods using RANS are useful in designing side weirs.

Samadi et al. [10] simulated flow on semi cylindrical weirs via *FLUENT* software. They used the volume of fluid method for simulation multiphase flow and used k- ε method to simulate the turbulence of flow. Study results of Samadi et al. [10] showed that the results extracted from *FLUENT* software has a good adaptation with experimental data.

As mentioned before; in this study Mousavi's [5] experimental data and *FLUENT* software are used, for simulation of flow on contracted compound arched rectangular sharp crested weirs within various values of the circular arc height; as three-dimensional. Finally numerical results have been compared with the experimental values. Mousavi [5] has carried out experiments in a laboratory flume made of glass walls. The flume length, width and height are 10, 0.25 and 0.5 (m), respectively.

2. MATERIALS AND METHODS

In this study, numerical modeling is carried out by *FLUENT* and variations of water depth at upstream of the contracted compound arched rectangular sharp crested weir is simulated in 3D environment. Then within data which obtained from numerical models, discharge coefficient value is calculated for different circular arc heights. In order to simulate multiphase flow, VOF method has been used and RNG k- ε turbulence model has been used for simulation of turbulent flow. Fig. 1 shows the weirs cross sections which used in this study. As it can be seen, weirs crest width (b) is 20 (cm), their height (P) is 15 (cm) and circular arc height (h_0) are respectively 10, 7.5, and 5 (cm).

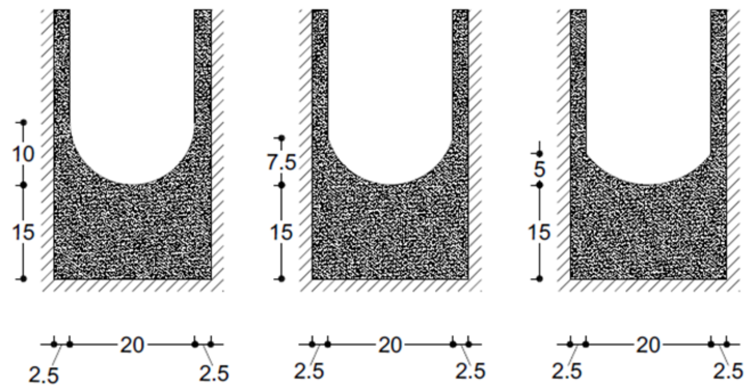

Figure 1. Sections of arched rectangular sharp crested weirs (b= 20 cm)

If the head over the weir (h) were less than circular arc height (h_0), weir acts as a circle sharp crested weir with a radius of R and the flow rate is calculated using equation (2). But if head over the weir got more than circular arc height ($h>h_0$), in this conditioned weir acts as compound and the flow rate is calculated using equation (4):

$$Q = C_d\,[4\sqrt{2g}.\varphi(\eta).R.h_0^{\frac{3}{2}} + \frac{2}{3}\sqrt{2g}.b_e.h_{1e}^{\frac{3}{2}}] \tag{4}$$

In above equation C_d is total discharge coefficient, R is radius of the semicircle weir, h_0 is circular arc height, b_e is effective width and h_{1e} is effective water head for apply viscosity and surface tension effects in a rectangular weir.

It should be mentioned that in this research, for assessing the accuracy of numerical results with experimental data Relative Average Error (RAE) method is used. The equation is as follows:

$$RAE(\%) = \frac{\sum_{i=1}^{N} | h_{EXP} - h_{CFD} |}{\sum_{i=1}^{N} h_{EXP}} \times 100 \tag{5}$$

At equation (5), h_{EXP} is head over the weir in physical model, h_{CFD} is head over the weir in numerical model and N is total number of data.

2.1 CFD simulation

The *FLUENT* is general-purpose CFD code, which is used by several researchers worldwide. This software by converting the governing equations to algebraic equations using finite volume method; solves 2D and 3D problems in open channel flow [1].

The continuity equation and momentum equation (Navier-Stokes equations) are governing equations of flow. For turbulent flow with incompressible fluid and constant viscosity and density Reynolds-averaged Novier-Stockes equations (RANS) are mentioned in following [1]:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \tag{6}$$

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}[\mu(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i})]+\frac{\partial}{\partial x_j}(-\rho\overline{u'_i u'_j}) \tag{7}$$

In these equations ρ is fluid density, u is velocity components, x is space dimensions, t is time, p is hydrostatic pressure, μ is dynamic viscosity, $\overline{u'_i u'_j}$ is Reynolds stress tensor and δ_{ij} is crooner delta.

In this study, *k-ε* turbulence model is used to solve RANS equations. In this model, two independent equations for k and ε are solved simultaneously. The aim of solving these equations is, calculate k and ε and finally obtain the turbulence velocity and incorporation length. Parameter k is turbulent kinetic energy which depends on the compressibility and buoyancy force. Parameter ε is also dissipation rate of turbulent kinetic energy. k-ε turbulence model is based on three types: Standard, RNG and Realizable. *RNG k-ε* method is used analytical or algebraic equations. So, this turbulence model is preferred to use in curved flow fields or in complex flow fields and it has high accuracy even in low Reynolds's number [1].

FLUENT implement the surface capturing approach; by using the VOF[2] scheme for general multiphase flow modeling. The VOF model is ideally suited to applications involving free surface flows. Volume of Fluid method is based on that, two or more fluids are not mix together. For each phase which is added to the model a variable is added, that is the phase volume ratio on computational cells. In each control volume the total volume fractions of all phases is equal to 1. So specification and amount of each cell represents one phase or mixture of them that depends on values volume ratio is variable between 0 and 1, Ref. [1].

It should be noted that, in this study the geometry design of models and their meshing was done using *GAMBIT* software. Fig. 2 shows the meshing of models. Also, in order to choose the best meshing, multiple meshing is evaluated. The result of this evaluation is mentioned as Fig. 3. According to this figure it can be said that, within increasing the number of meshes from 5280 to 15200 the relative error is reduced. But for larger amounts of 15200 relative error percent have a little change and the use of more meshes cause longer convergence time. So that in this study the meshing count of 15200 chosen for the optimum meshing.

[2] Volume of Fluid

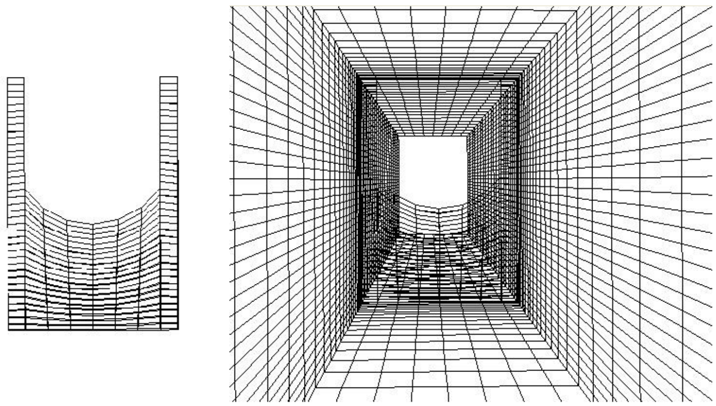

Figure 2. weir model and meshes in gambit software (b= 20 cm)

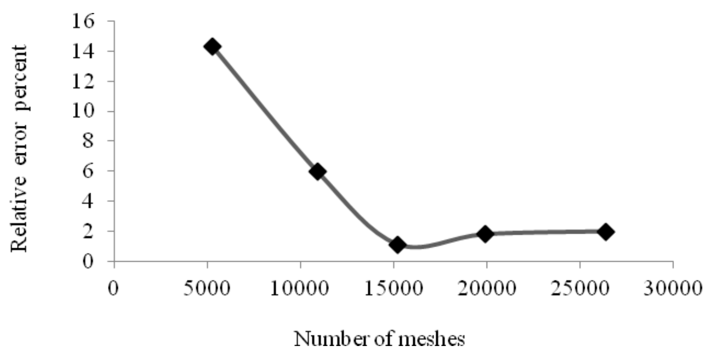

Figure 3. Relative error ratio of water depth in upstream of the weir crest for different amounts of meshes

After meshing, boundary condition is defined for the model. Fig. 4 shows the boundary conditions for the models. In this paper the walls of the channel and combined weir defined by Wall boundary condition, water depth at the entrance and freeboard of channel defined by Velocity inlet condition and output stream page is defined by Pressure outlet condition.

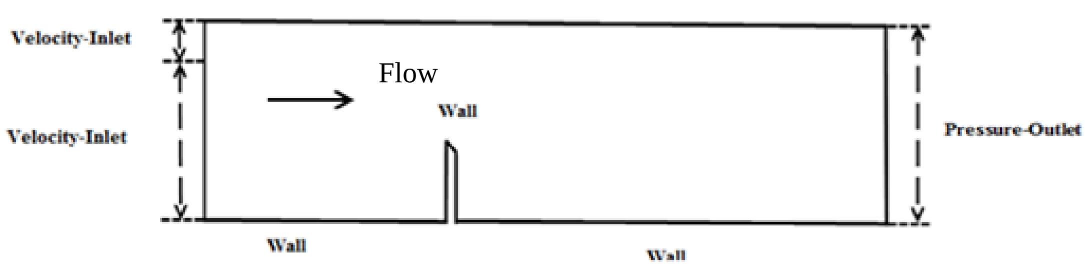

Figure 4. Boundary conditions imposed on the model

3. RESULTS AND DISCUSSION

3.1 Water surface profile

At Figs. 5 to 7, ratio of flow rate - water head for contracted compound arched rectangular sharp crested weir for various values of the circular arc height (h_0) is presented. Table 1 also represents relative error percent of water depth in upstream of the weir crest.

According to Figs (5) to (7) and Table 1 it can be said that; for various values of the circular arc height, if $h/b > 0.41$ ($h>8.2$ cm) results of numerical model had a good agreement with the experimental data and water depth relative error in upstream of the weir was less than 1%.

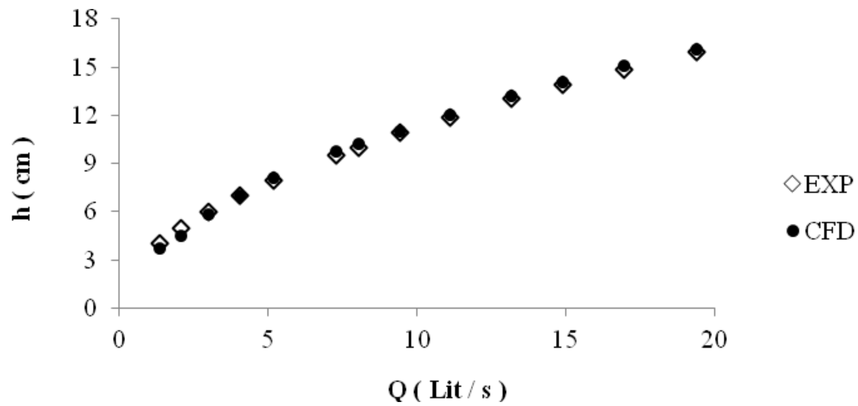

Figure 5. Ratio of flow rate variations - water head for contracted compound arched rectangular sharp crested weir b=20 (cm), h_0=10 (cm)

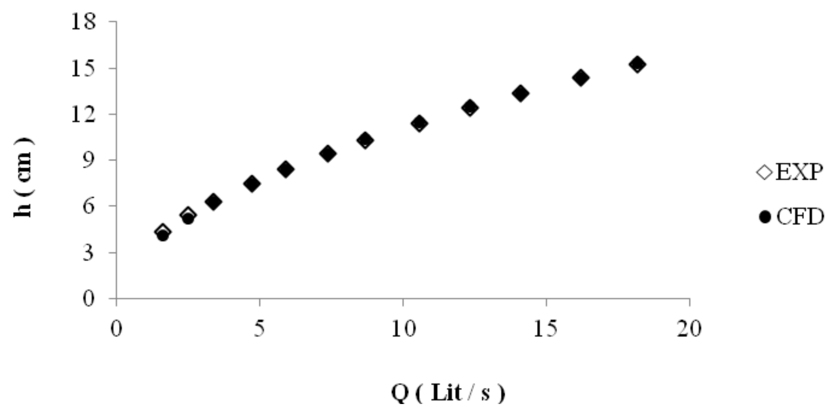

Figure 6. Ratio of flow rate variations - water head for contracted compound arched rectangular sharp crested weir b=20 (cm), h_0=7.5 (cm)

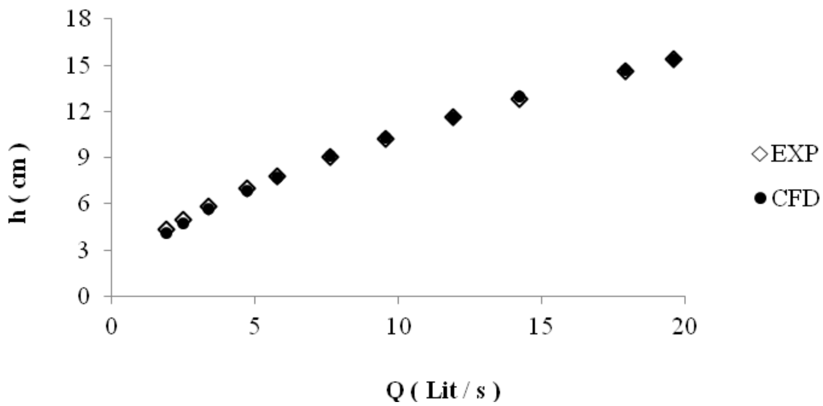

Figure 7. Ratio of flow rate variations - water head for contracted compound arched rectangular sharp crested weir b=20 (cm), h_0=5 (cm)

Table 1: The relative error of water depth in upstream of the weir

h_0 (cm)	RAE of water depth in upstream of the weir (%)
10	1.92
7.5	0.68
5	1.34

3.2 Discharge coefficient

By using data of water depth in upstream of the weir crest which obtained from numerical model, discharge coefficient for contracted compound arched rectangular sharp crested weirs for multiple values of the circular arc height is calculated. As mentioned before; due to flow conditions, amount of C_d is determined using equations (2) and (4).

In Figs (8) to (10) calculated amounts for discharge coefficient from numerical and experimental data on contracted compound arched rectangular sharp crested weirs, were compared. Table (2) also shows relative error of C_d for various values of the circular arc height. According to Figs. (8) to (10) and Table 2 it can be said that:

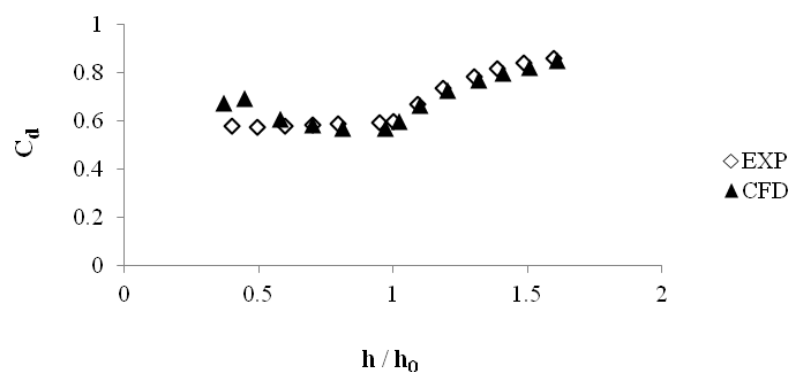

Figure 8. C_d changes in contracted compound arched rectangular sharp crested weir b=20 (cm), h_0=10 (cm)

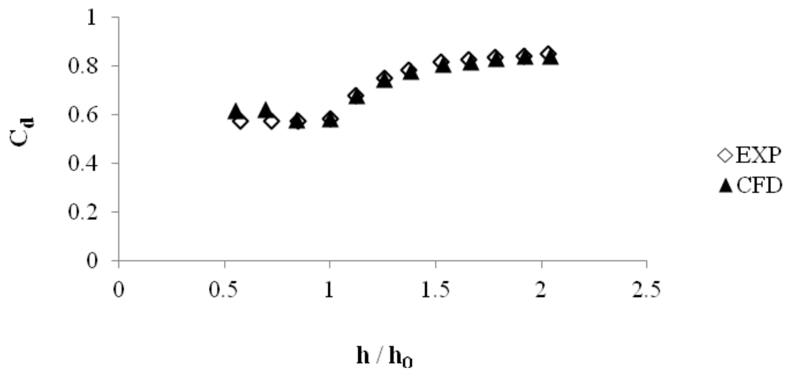

Figure 9. C_d changes in contracted compound arched rectangular sharp crested weir b=20 (cm), h_0=7.5 (cm)

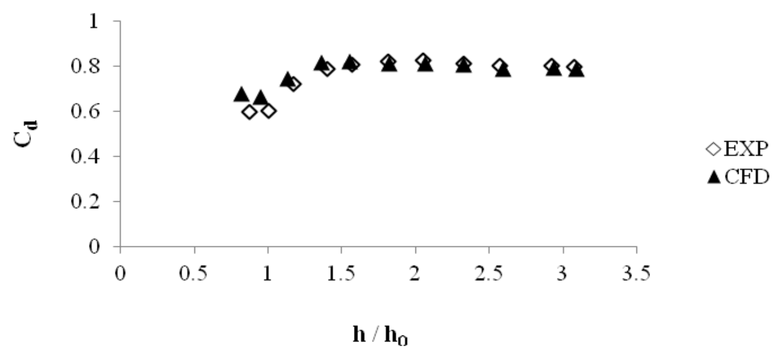

Figure 10. C_d changes in contracted compound arched rectangular sharp crested weir b=20(cm), h_0=5 (cm)

Table 2: Relative error of C_d for various values of the circular arc height

h_0 (cm)	RAE of C_d (%)
10	3.44
7.5	1.57
5	3.11

In condition $h/h_0>1.10$ there is a good agreement between experimental data and numerical data of discharge coefficient and the relative error of C_d is less than 1%. But at low flow rates ($h/h_0<1.10$) due to contact of the flow with weir wall, we cannot ignore effects of surface tension, viscosity and also impacts of channel walls onto flow. So there is a less agreement between C_d coefficients.

4. CONCLUSION

The CFD analyses which were performed by *FLUENT* software to simulate the free surface flow over contracted compound arched rectangular sharp crested weirs indicate:

- The relative error of water depth in upstream of the weirs crest are between 0.68 % and 1.92%, so *FLUENT* software has simulated the flow over contracted compound arched rectangular sharp crested weirs with high accuracy.
- In the range of $h/b> 0.41$ relative error of the water depth in upstream of the weir, is negligible and about 1%.
- Considering that the relative error of C_d for various values of the circular arc height on contracted compound weirs is less than 3.5%, we can use *FLUENT* software for determine the discharge coefficient on contracted compound weirs.
- In the range of $h/h_0>1.10$, there is good agreement between experimental and numerical values of discharge coefficient and the relative error of C_d is less than 1%.
- The most important factor that causes merging differences between numerical and experimental data of the discharge coefficient; is theoretical equation of C_d coefficient in compound mode. Equation (4) is obtained from linear combination assumption of flow equations on rectangular and circular sharp crested weirs. While in real terms, flow equation in contracted compound arched rectangular sharp crested weirs is not linear. That's because the characteristics of flow over a compound weir is completely different from simple sharp crested weirs.
- Relative error of C_d for various values of the circular arc height; does not have a certain ratio and by reducing the relative error of water depth in upstream of the weir decreases and vice versa.
- Effect of channel side walls (physical characteristics of model) and accuracy of experimental data, are among the factors that cause differences between laboratory and numerical data.

REFERENCES

1. Anonymous. *FLUENT 6.2 User's Guide*, 2005.
2. Arvanaghi H, Nasehi Oskuei N. Sharp crested weir discharge coefficient, *J Civil Eng Urban* 2013; **3**(3): 87-91.
3. Bos MG. *Discharge measurement structures, International Institute for Land Reclamation and Improvement (ILRI)*, Publication 20, Wageningen, the Netherlands, 1989.
4. Haun S, Reidar NBO, Feurich R. Numerical modeling of flow over trapezoidal broad-crested weir, *Eng Applics Comput Fluid Mech* 2011; **5**(3): 397-405.
5. Mousavi N. Assessing discharge coefficient of arched trapezoidal and rectangular sharp crested weirs, M.Sc. thesis, Hydraulic Structures, Agriculture Faculty, Tabriz University, 2012.
6. Namaee MR, Rostami M, Jalaledini S, Habibi M. A 3-Dimensional numerical simulation of flow over a broad-crested side weir, *Adv Hydroinform* 2014; 511-23.
7. Piratheepan M, Winston NEF, Pathirana KPP. Discharge measurements in open channels using compound sharp crested weirs, *J Institut Eng* 2006, Sri Lanka; **XXXX**(3): 31-8.
8. Rady R. 2D-3D modeling of flow over sharp crested weirs, *J Appl Sci Res* 2011; **7**(12):

2495-2505.
9. Ramamurthy AS, Tadayon R, Chen Z. Numerical simulation of sharp crested weir flows, *J Can Civil Eng* 2009; **36**(9): 1530-4.
10. Samadi A, Nejati S, Haji Azizi S, Oghati Bakhshayesh B. Three dimensional simulation of flow for semi cylindrical weirs using fluent software, *J Civil Eng Urban* 2014; **4**(4): 397-401.
11. Stevens JC. Flow through circular weirs, *J Hydraulic Eng*, *ASCE* 1959; **85**(1): 85-98.
12. Vatankhah AR. Flow measurement using circular sharp crested weirs, *Flow Measur Instrum* 2010; **21**(2): 118-22.