Wasit Journal of Engineering Sciences
VOLUME: (6), NO. : (2) 2018

Comparison of Flow over Broad Crested Weir in Laboratory and by a Numerical Method

Laith Hamdan Hawal , Thameen Nazar Nahi

Environment Engineering Department , Al-Mustansiriayah University

Corresponding author Laith_hamdaen@yahoo.com , thamen92@yahoo.com

Submitted:15/6/2017
Accepted: 15/11/2017

Abstract: The main purpose of this study is to test the ability of competition fluid dynamic (CFD) model which (FLUENT) program to simulate the flow over broad crested weir with lower cost and lesser time. This purpose is done by operating an experimental model in the laboratory and then validating this model in the GAMBIT program and FLUENT (ANSYS R 15.0), by comparing the results. The FLUENT can solve Navier- Stokes equations of the flow numerically, the volume of fluid method (VOF) and Standard k– ε turbulence equation is depended to simulate water level. Generally, it can be established that the experimental results of flow have a good agreement with the numerical results. The percentage of error (RE %) between numerical and experimental discharge is 2.94%, while the maximum RE % for water level is 6.25%. The numerical results of flow over broad crested weir showed in counter and vector results, and the streamlines of flow was clear than that of the experimental model. Finally, the FLUENT proved that can it be relied upon in future by designing weirs without needing to work experimentally.

Keyword: competition fluid dynamic, program, simulate, FLUENT, volume of fluid.

مقارنة الجريان فوق الهدارات العريضة الحافة مختبريا مع طريقة التحليل العددي

ليث حمدان حول ، ثمين نزار ناهي

الخلاصة:
ان الغرض الرئيسي من هذه الدراسة هو لاختبار قدرة محاكاة ديناميكية الموائع CFD والمتمثل ببرنامج FLUENT لتمثيل الجريان فوق الهدارة العريضة الحافة بكلفة منخفضة ووقت قليل. وهذا الغرض عمل من خلال تشغيل موديل تجريبي في المختبر واثبات هذا الموديل في برنامج GAMBIT وبرنامج FLUENT (ANSYS) اصدار ١٥) من خلال مقارنة النتائج . إن برنامج FLUENT يحل معادلات Navier-Stokes لِلجريان عدديا، وطريقة حجم السائل VOF ومعادلة الاضطراب k– ε القياسية قد اعتمدت لتمثيل سطح الماء. عموما، يمكن الاعتماد على ان النتائج المختبرية للجريان كانت متقاربة بشكل جيد مع النتائج العددية. النسبة المئوية للخطأ RE% بين التصريف العددي والمختبري ٢.٩٤%، بينما كانت اقصى نسبة خطأ للسطح الماء كانت ٦.٢٥%. النتائج العددية للجريان فوق الهدارة العريضة الحافة قد بينت في نتائج الكنتور والمتجهات ، وخطوط انسياب ألجريان قد وضحت اكثر من الموديل المختبري. أخيرا FLUENT اثبت انه يمكن الاعتماد عليه مستقبلا عند تصميم الهدارات بدون الحاجة للعمل مختبريا.

1. Introduction

In the last 10 years ago, many researchers have studied the flow over weirs of different shapes by the numerical method**.** Simulation was made for the free surface profile over a weir with a flume by a CFD for the smaller flow rate, hopping to be used for higher flow rates and to study more complex structures than the weir **[1]**. When the velocity and pressure on broad-crested weir was investigated**,** the results showed a rapid redistributions of both velocity and pressure field at the U/S end of the weir crest.

These were hardly studied in a large size and under controlled flow conditions **[2]**. The validity of CFD for free surface flow over weir was conducted by using the experimental dataset of Hager and Schwalt (1994) in a series of CFD simulations, and then compared against experimental data. The agreement between the results has been an acceptable as level of accuracy **[3]**. Another study on a rectangular broad crested weir was presented to investigate the flow profile of water. The CFD results showed that there is a good agreement with experimental results obtained in the laboratory. 3D simulations were performed and three turbulence models of the RNG k–ε, standard k–ε and the large eddy simulation (LES) to find the water level profile **[4]**.

Laboratory measurements were carried out on rectangular broad crested weir with different geometries located on a rectangular channel to predict the equation for C_d **[5]**. The effect of inclination from 23º to 90º weir in the U/S face of broad crested weirs on flow characteristic was studied by using CFD model with laboratory model to reduce the effect of flow separation. The results showed that values C_d at a 23^o slope is about 22% higher than for a weir with a 90º U/S face slope **[6].** There is a study deal with explanation of flow structure in front of broad crested weir. The result has been simulated by using many models and based on comparing the results of the suitable model of turbulence. The error of its comparing in all cases up to 3 % by using ANSYS-CFX (2D and 3D), FLOW 3D (2D) and ANSYS-Flotran (2D) [7**]**.

The objective of this work is to study the flow over broad crested by comparing the water level by both the experimental and numerical models. The work will also try to validate the laboratory results by using a numerical program (FLUENT v.15) to show its possibility in the applications of the hydraulic problem and can be relied upon it in future when designing weirs without needs to make laboratory tests.

2. Theory

Weirs are the best hydraulic structures that can be easily used to measure the flow rate in the field due to their ability in measuring a wide range of flow, provide more accurate and floating material passes over the structure. Weirs are well-defined as constructions where the streamline parallel to each other and the hydrostatic pressure occurs in the middle of the horizontal crest of the weir. The types of weirs are built in a wide range of shapes and sizes and the choice of type depends on the purpose of the weir. The most common types of weirs are illustrated in figure (1) **[8].** In this study the broad crested weir is used.

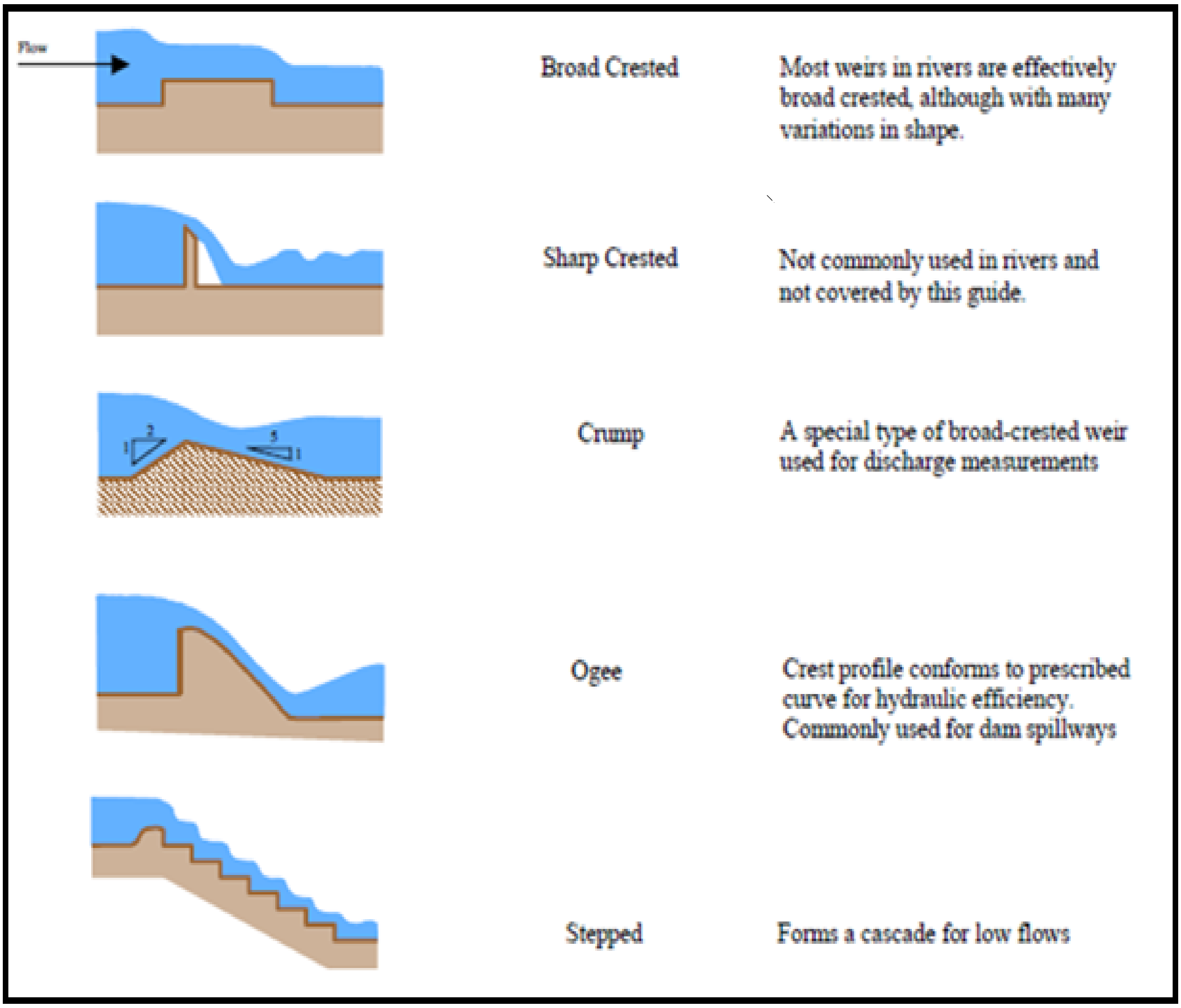

Figure (1) Types of Weir According to Purpose

Broad crested weir was a structure with a horizontal crest where the critical depth occurred along the crest and streamline was parallel. The condition of parallel streamlines flow over broad crested weir can be achieved if the equation below was satisfied;

$$2 \leq \frac{L}{h_0} \leq 12 \qquad (1)$$

Where; h_0 is the head over the weir U/S crest to free surface and L is length of the weir **[9],** as shown in figure (2)**.**

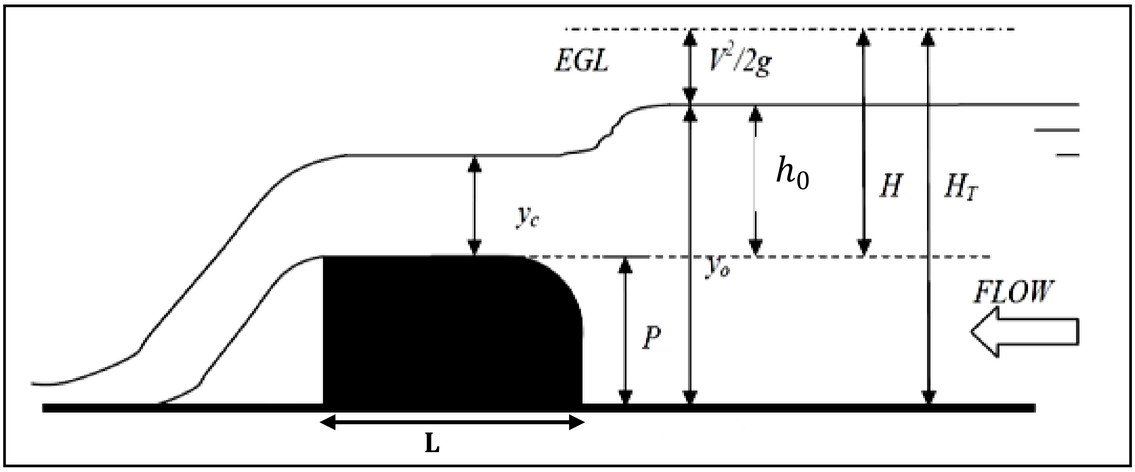

Figure (2) Designation of a Broad Crested Weir

The equation that used to measure the discharge is represented as a function of the head of the water and the weir geometrical properties. The relation between weir geometrical and head over the crest can be expressed by Bernoulli equation, as in equation below;

$$\mathbf{Q} = \frac{2}{3}\sqrt{\frac{2}{3}\mathbf{g}}\,(\mathbf{h_o})^{3/2}.\,\mathbf{B} \qquad (2)$$

Where; the Q is the discharge (m^3/s), g is the gravitational acceleration (m/s^2), B is the width of the weir (the perpendicular direction to the flow) (m) **[10]**.

The square edge U/S corner of weir has many advantages such as the constant coefficient of discharge (C_d), simple construction and low cost construction; but a separation zone appears. Round U/S face of weir leads to a decrease in the separation zone and an increase in C_d, the C_d increases to 8% if the U/S corner is round **[11].**

There is an obvious derivative for the free and submerged flow conditions of square edge of broad crested weirs, then applied the boundary layer theory and founded separation at crest zone **[12]**. The equation of C_d for round corner of broad crest weir was found at Y_b=0.715Y_c, where; Y_b = is the depth of water at the D/S edge of the weir and Y_c = is the critical depth **[13]**. Weirs are Classified depending on the degree of rounding of the U/S corner, when the ratio of rounding is (R/P < 0.094) here the weir is "slightly rounded", when the ratio of rounding is (0.094 < R/P < 0.250) here the weir is "moderately rounded " and when the ratio of rounding is (0.250 < R/P < 1.0) here the weir is "well-rounded"**[14]**.

3. Experimental Method

The experimental model and the materials that were used in laboratory tests are all carried out in the Hydraulic Laboratory, Environmental Engineering Department, and Faculty of Engineering in Al-Mustansiriya University at Baghdad. The dimension of broad crested weir was designed according to the recommendations suggested by the limitation conditions of broad crested weirs **[10]**. The height (P) of the weir was 0.15 m. The width (B) of the weir was 0.3 m. The length (L) of the weir was 0.36 m. The weir is made of steel plate with a thickness of 0.05 m with a round (R) U/S corner of 0.03 m. The walls of the flume were made of strong glass with 4.6 m length, 0.3 m width and 0.3 m height, as shown in figure (3). Three point gauges were installed along the centerline of the flume for head measurements.

Figure (3) Photographs Showing the Laboratory Flume with Weir

4. Numerical Method

The developments in computer science and numerical method produced the CFD as a controlling tool for this purpose**.** The computer is applied to solve the simple and small scale equations; this means the replacing of nonlinear differential equations with linear difference equations, then solving them until the calculations reaches to a fixed level **[15]**.

Furthermore, the aim of CFD is to simplify the governing continuity and momentum equations which called Navier-Stokes equations of the flow that can be solved numerically. The continuity and momentum equations are described by the differential equation written below **[16];**

$$\frac{\partial}{\partial t}\boldsymbol{\rho} + \frac{\partial}{\partial x_i}\boldsymbol{\rho u_i} = \mathbf{0} \qquad \textbf{(3)}$$

$$\frac{\partial}{\partial t}\boldsymbol{\rho u_i} + \frac{\partial}{\partial x_i}\boldsymbol{\rho u_i u_j} = -\frac{\partial}{\partial x_i}\boldsymbol{P} + \frac{\partial}{\partial x_i}\left[\boldsymbol{\varepsilon}\left(\frac{\partial u_i}{\partial u_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \boldsymbol{\rho g_i} + \vec{\boldsymbol{F}} \qquad \textbf{(4)}$$

Where; $u_{i,j}$ is velocity in x and y direction, ρ is fluid density, t is time, p is the pressure, ε is turbulence viscosity, g_i is gravity acceleration and $\vec{F}$ is the body force **[17].**

The GAMBIT program which it's one of the (ANSYS R 15.0) programs used to create a geometry modeling in the same dimension of the laboratory model and used to mesh generation. The mesh creation in this program contains also the coordinates of all the nodes and the factors deciding the best mesh to choose in time, accuracy and dimension, the size of mesh is 5 mm. The quadrilateral and triangle mesh are made together, because they continue in all shapes rather than triangular or quadrilateral only, as shown in figure (4).

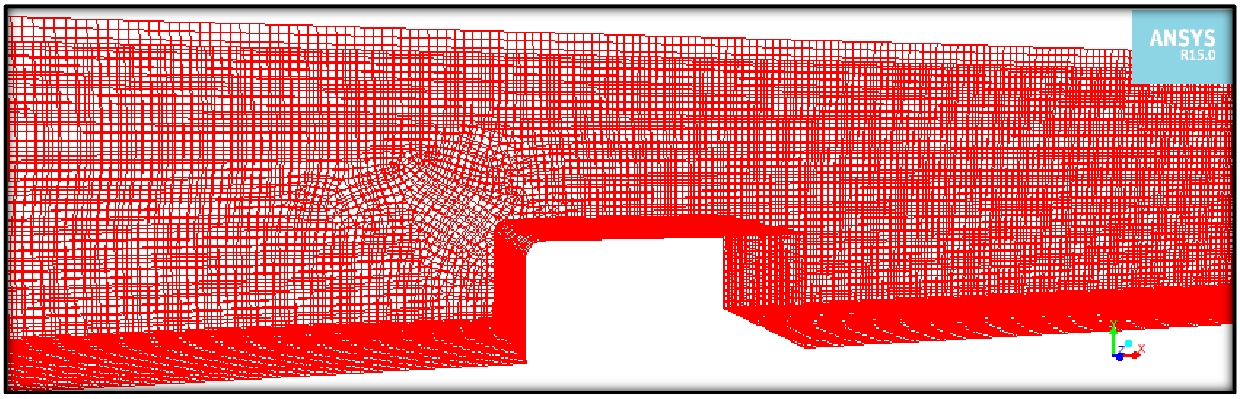

Figure (4) Procedure of Geometry and Mesh Generation Using Gambit (ANSYS R 15.0)

The FLUENT (version 15) program is used to solve the problem. FLUENT is one of the common CFD software which is able to simulate the steady and unsteady state by transforms the governing continuum and momentum equations to the numerical equations by Finite Volume Method (FVM). Depending on the Volume of Fluid (VOF), there is method which is one of many methods that can be used to determine the water surface profile between the air and water phases. The Standard k– ε turbulence model is more depended to solve the numerical model in this study because it gives more agreement with the laboratory model **[**18**].**

The boundary conditions of this study which are used in numerical model were the same conditions of the laboratory model. These can be classified as;

1. Inlet and outlet boundaries are pressure inlet and pressure outlet. Pressure inlet which is employed have two different inlets, the first is the pressure inlet for water and the second is pressure inlet for air in the model domain, where pressure outlet for air only.
2. Vertical boundary of weir and flume are walls.
3. The upper boundary condition is specified as symmetry. As shown in figure (5).

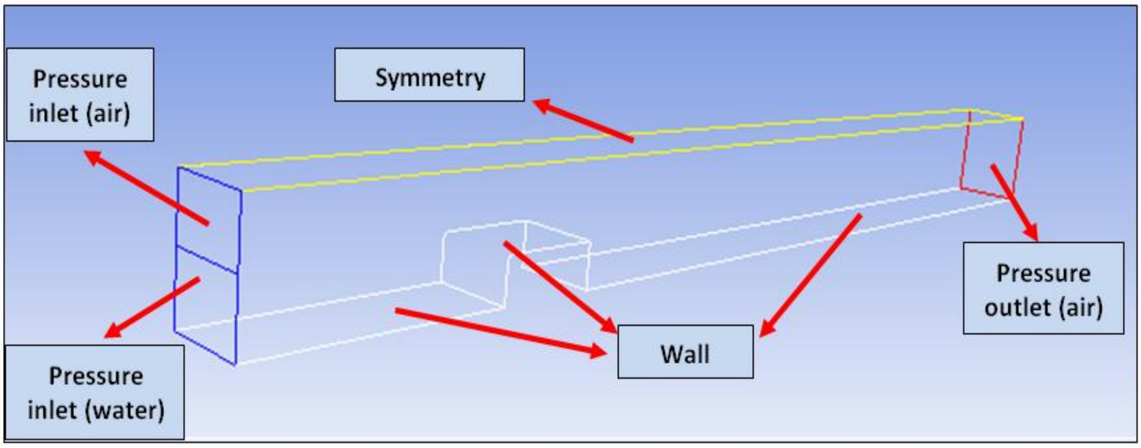

Figure (5) Boundary Condition of Numerical Model

4. Results and discussion

Verification and validation of this program are necessary to discover its accuracy and to provide a level of confidence in applying CFD method in future studies. The results of three dimension (3D) model in this software are faster and are more economical than experimental results. The results of discharge in the numerical method and experimental results are given in table (1)**.** The table includes the percentage of error between the two methods (as RE %) as defined by the following equation:

$$RE\ \% = \left|\frac{Q_{exp}-Q_{num}}{Q_{exp}}\right| * 100 \qquad \textbf{(5)}$$

Where; Q_{exp} is the experimental discharge, Q_{num} is the numerical discharge obtained. Generally, it can be established that the experimental results for the broad crested weir are of a good agreement with the numerical simulation.

Table (1) Percentage of error of the Discharge over Broad Crested Weir

Title	Q_{exp} m3/hr	Q_{num} m3/hr	RE%
Value	44.2	42.9	2.94

The numerical results of water level FLUENT program and experimental work broad crested weir was drawn and compared by XY plot. From figure (6), which shows that the subcritical flow occurs before the U/S corner of weir and exactly before x=90 cm, the water level accelerates in decreasing till the D/S of the weir and the critical depth was occurred over the weir crest. While, the flow rapidly varies with highly steep decreases after the D/S corner of weir exactly after x=126 cm, where the flow becomes supercritical in this region.

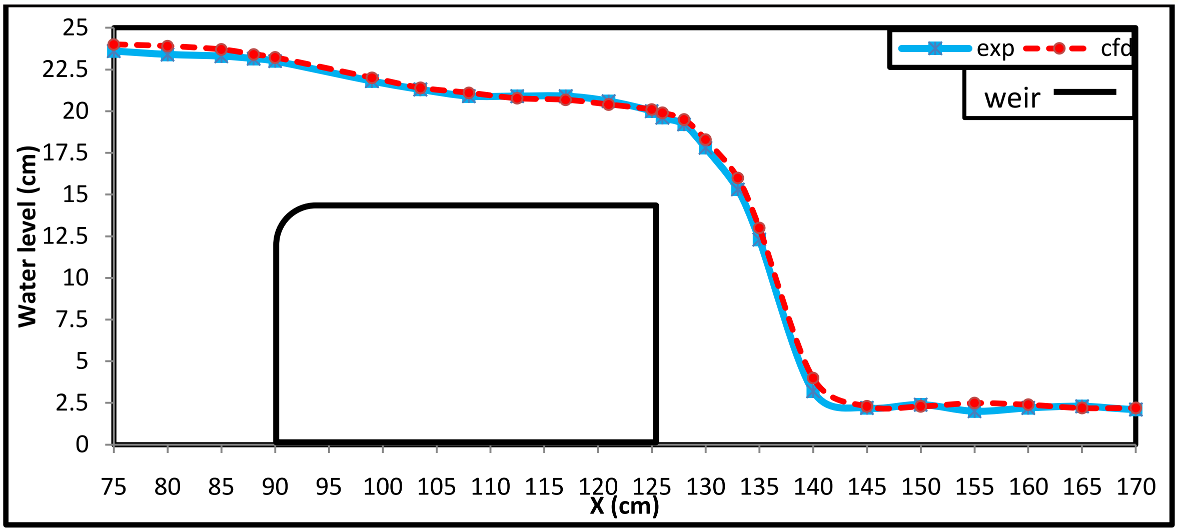

Figure (6) Comparison of Water level between Experimental and Numerical Results

The percentage or error (RE %) of water level between two methods at different locations (different points) of point gauge are determined. A good agreement between the experimental and the numerical results for the water level along the flow direction, as can be seen in table **(2)** and the equation of RE% is;

$$RE\ \% = \left|\frac{h_{exp}-h_{num}}{h_{exp}}\right| *100 \qquad (6)$$

Where; $\mathbf{h_{exp}}$ is the experimental head, $\mathbf{h_{num}}$ is the numerical head.

(2) Percentage of error of Water Level with Distance over Broad Crested Weir

Location at X (cm)	h_{exp} (cm)	h_{num} (cm)	$\left\|\frac{h_{exp}-h_{num}}{h_{exp}}\right\|$*100
75	23.6	24	1.7
80	23.4	23.9	2.14
85	23.3	23.7	1.7
88	23.15	23.4	1.1
90	23	23.22	0.95
99	21.8	22	0.92
103.5	21.3	21.4	0.47
108	20.9	21.1	0.96
112.5	20.9	20.77	0.62
117	20.9	20.68	1.05
121	20.6	20.4	0.97
125	20	20.1	0.5
126	19.6	19.9	1.53
128	19.2	19.5	1.56
130	17.8	18.3	2.81
133	15.3	16	4.58
135	12.3	13	5.7
140	3.2	3.4	6.25
145	2.2	2.3	4.5
150	2.4	2.3	4.1
155	2.4	2.5	4.1
160	2.3	2.4	4.35
165	2.2	2.3	4.34
170	2.1	2.2	4.8

The FLUENT program was selected because it provides a useful behavior of various flow profiles by vectors and contours which is considered of difficult feature to predict by experimental model. The results of FLUENT program are shown in figures (7), (8), (9) and figure (10).

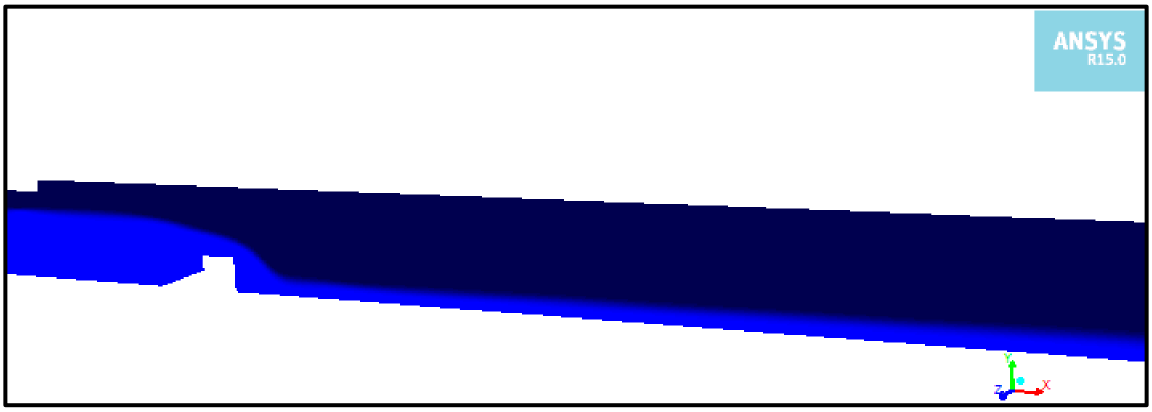

Figure (7) Water Flow by Contour Type over Broad Crested Weir at Q = 42.9 m^3/hr

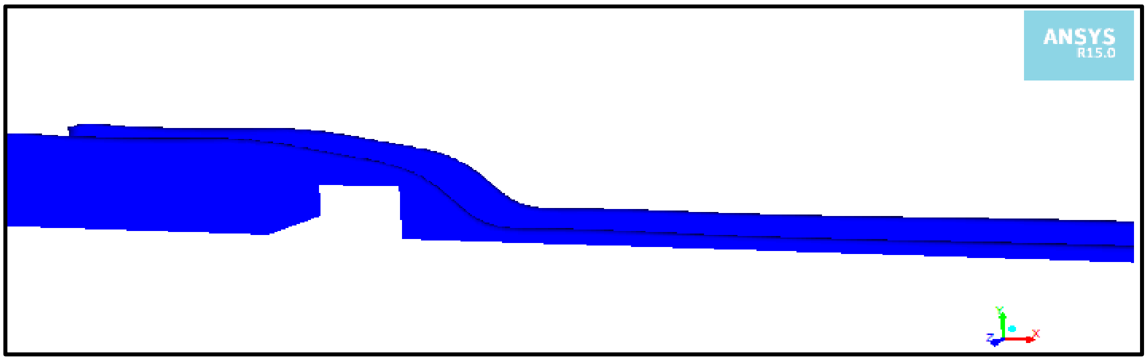

Figure (8) Flow Profile over Broad Crested Weir by Contour Type at Q = 42.9 m^3/hr

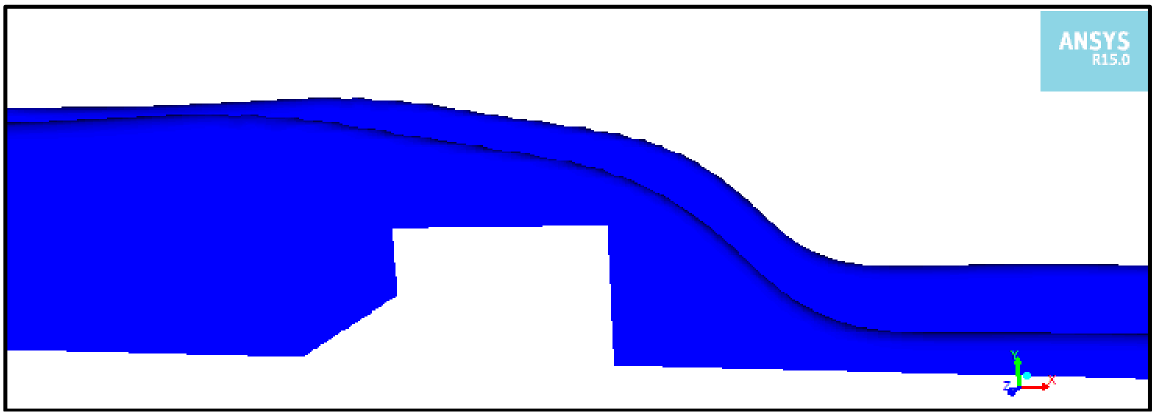

Figure (9) Section of Flow Profile near Broad Crested Weir by Contour Type at Q = 42.9 m^3/hr

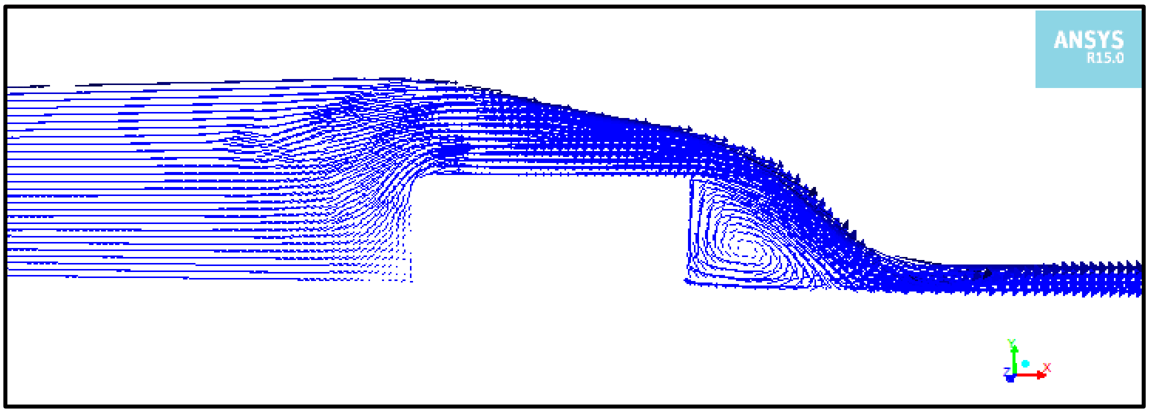

Figure (10) Flow Profile over Broad Crested Weir by Vector Type at Q = 42.9 m^3/hr

The FLUENT results are shown in figure (11), the streamlines are parallel to the crest everywhere along the crest which is not clearly shown in the experimental model.

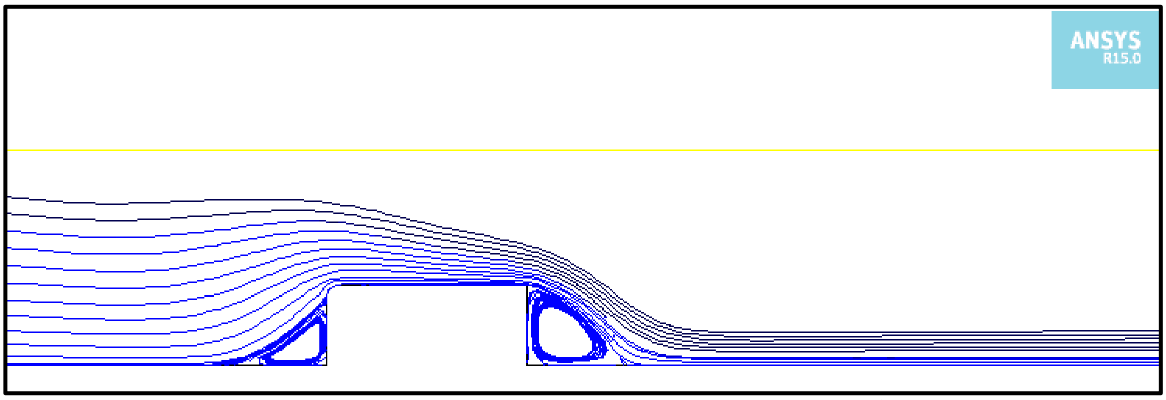

Figure (11) Streamlines of Flow over Broad Crested Weir at Q = 42.9 m^3/hr

4. Conclusions

Through this study, the following main conclusions can be drawn;

1) A good conformation results were obtained between the experimental and the numerical flow over broad crested weir.
2) The agreement between numerical and experimental results for water level is good, with a minimum percentage of error of 0.47% at a distance of 103.5 cm and a maximum percentage of error of 6.25% at a distance of 140 cm from up stream
3) The percentage of error for the discharge between numerical and experimental results is 2.94%.
4) The FLULENT program provide more details on the flow profile in many cases, such as counter and vector in good time and more economical cost.

References

[1] Sarkar M. A. and Rhodes. D. G., 2004. "CFD and physical modeling of free surface over.

[2] Gonzalez, C. A., and Chanson, H. 2007. "Experimental measurements of velocity and pressure distribution on a large broad-crested weir", Australia, Flow Measurement and Instrumentation, Vol. 18, pp. 107–113.

[3] Hargreaves D. M., Morvan H. P. and Wright N. G., 2007. "Validation of the Volume of Fluid Method for Free Surface Calculation: The Broad-Crested Weir", The University of Nottingham, Engineering Applications of Computational Fluid Mechanics, Vol. 1, No. 2, pp. 136–146.

[4] Afshar H. and Hooman H., 2013. "Experimental and 3-D numerical simulation of flow over a rectangular broad-crested weir", International Journal of Engineering and Advanced Technology, Vol. 2, No. 6, pp. 214-219.

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

[6] Al-Hashimi A. S., Sadeq A. and Huda M., 2015. "Determination of Discharge Coefficient of Rectangular Broad-Crested Weir by CFD", The 2nd International Conference of Construction, BCEE2.
[7] Zbynk Z. and Ladislav R., 2015. "Flow structure in front of the broad-crested weir", Czech Republic, European Physical Journal (EPJ) Conferences, Vol. 92.
[8] Charles R., Rodney D. and Jeremy P., 2003. "River Weirs – Good Practice Guide", University of Hertfordshire, United Kingdom.
[9] Chow, V. T., 1959. "Open channel hydraulics", New York, McGraw-Hill.
[10] ASTM International Designation, 2014. "Standard Guide for Selection of Weirs and Flumes for Open-Channel Flow Measurement of Water", United States.
[11] Woodburn J. G., 1932. "Tests of Broad-Crested Weirs", Transactions of the American Society of Civil Engineers, Vol. 96, No.1, pp. 417-453.
[12] Hall G. W., 1962. "Discharge Characteristics of Broad-Crested Weirs Using Boundary Layer Theory", London, England, Proceedings of the Institution of Civil Engineers, pp. 172-190.
[13] Henderson F. M., 1966. "Open Channel Flow", New York, MacMillan.
[14] Ramamurthy A. S., Tim U. S., and Rao M. J., 1988. "Characteristics of Square-Edged and Round-Nosed Broad-Crested Weirs", Journal of Irrigation and Drainage Engineering, Vol. 114, No. 1, pp. 61-73.
[15] Wilcox D. C., 1993. "Turbulence Modeling for CFD", DCW Industries Inc., La Canada, California.
[16] User’s FLUENT Manual, Version 15, ANSYS, Inc., 2009.
[17] Liu C, Hute A. and Wenju M., 2002. "Numerical and experimental investigation of flow over a semicircular weir", Acta Mechanica Sinica, Vol. 18, pp. 594-602.
[18] Hirt C. W. and Nichols B. D., 1981. "Volume of fluid (VOF) method for the dynamics of free boundaries", Journal of Computational Physics, Vol. 39, No. 1, pp. 201-225.