ZANCO Journal of Pure and Applied Sciences

The official scientific journal of Salahaddin University-Erbil

ZJPAS (2018),30 (5); 131-139

http://dx.doi.org/10.21271/ZJPAS.30.5.11

CFD Modeling of Simultaneous Flow Over Broad Crested Weir and Through Pipe Culvert using Different Turbulence Models

Othman K. Mohammed[1], Yaseen W. Aziz[2]

1- Department of Civil Engineering, College of Engineering, Salahaddin University-Erbil

2- Department of Dams and Water Resources Engineering, College of Engineering, Salahaddin University-Erbil

A R T I C L E I N F O

Article History:
Received: 05/04/2018
Accepted: 10/09/2018
Published:28/10/2018

Keywords:
Broad Crested Weir
CFD Modeling
Simultaneous Flow
Turbulence Models
ANSYS CFX

Corresponding Author:
Othman K. Mohammed
othman.mohammed@su.edu.krd

A B S T R A C T

Culverts and broad crested weirs are hydraulic structures that could be used for measuring the flow rate in open channels. In this research, a simultaneous flow over broad crested weir and through pipe culvert was simulated using the numerical software ANSYS CFX. Three turbulence models were used for the modeling of flow turbulent to determine the best turbulence model for combined flow simulation. To achieve this purposes, results of simulations have been compared with data gathered from lab experiments from literature. The computational results showed a close agreement with obtained experimental data, but that of the (RNG) k – ε model provides more accurate results compared with other two turbulence models used in this study. Therefore, it can be concluded that this turbulence model (RNG) k – ε model can be used for simulation of simultaneous flow over broad crested weir and below through culverts.

1. INTRODUCTION

In the case of culvert overtopping with water, where the culvert cross sectional area is insufficient to drain the incoming flood, the ordinary solution is either to replace the old culvert with a bigger capacity one or to add new vents to the original one. An alternative solution is to use a hydraulic structure (broad crested weir with circular opening). In this case a part of the flow will go through the culvert vents (circular openings) and the rest will overtop it. (Mahmoud S. M. 2002) and (Negm A. M. 2002) conducted an experimental investigation on simultaneous flow through box culvert and over contracted broad crested weir. The flow at the culvert outlet is considered as submerged flow, a discharge prediction models have been developed by means of multiple linear regression techniques. (Othman K. Mohammed 2010) simulated experimentally the combined flow through pipe culvert and over broad crested weirs of different side slopes, he developed empirical relations between the discharge coefficient and geometrical parameters of the combined weir culvert model. Due to high cost in construction of the physical models (Kositgittiwong, 2012). . In addition, because of the difficulties in solving the high order partial differential equations of many fluid flow (Aziz, 2016). Nowadays, most researchers turn to the use of numerical methods. (Sarker and Rhodes 2004) compared the free surface profile over a laboratory rectangular broad crested weir with numerical CFD model using commercial software FLUENT applying both slandered k – ε model and (RNG) k – ε model, they reported that the uncertainties in predicting the water levels above and D/S the crest were higher compared to the wave-like profile observed in the laboratory experiments. (Hargreaves D. M. 2007) conducted a serious of CFD simulations using version 6.2 of FLUENT, for predicting free surface profiles over broad crested weir, they used the experimental data of (Hager W. and Schwalt M. 1994) to verify the validity of the computational code in prediction the position of free surface profile, velocity and pressure distributions for different flow rates. (Afshar, H. and Hoseini, H. 2013) used CFD together with laboratory model in order to simulate the flow over rectangular broad-crested weir. Simulations were performed using three turbulence models of the RNG k–ε, standard k–ε and the large eddy simulation (LES) to find the water level profile and streamlines. Their results indicated that RNG model has lowest error compared with the other models. (S. Hoseini, S. Jahromi and M. Vahid 2013) used ANSYS FLUENT V.14 together with laboratory model for determining the discharge coefficient of the rectangular broad-crested side weir located on the trapezoidal channel, they found that both results of CFD and physical model showed that Cd coefficient decreases with increasing values of Fr and Cd coefficient increases with increasing values of Re. (Hoseini S. H. 2014) simulated the free surface flow over the triangular broad-crested weir using FLOW 3D. The simulation results were found in reasonable agreement with experimental observations. (Jalil, Shaker and Qasim, Jihan 2016) used FLOW-3D and HEC-RAS software's to predict the free surface profile of Flow over Single-Step Broad- Crested Weir, they found that HEC-RAS has limited ability to produce curved profiles past vertical faces, while FLOW-3D produced more accurate results. (Al-Hashimi A. S. 2017) used Fluent Software to compare four different turbulence models accuracy in computing free surface flow over broad crested weir and stepped weir with rounded corner. Results are compared with the experimental data and showed that the predictions provided by the standard k–ε model are closer to the experimental data, whereas those obtained from the standard k–ω model deviate the most. As found from literature survey that the characteristics of flow over broad crested weir along with the development of CFD codes have attracted the attention of many investigators. In this study, the flow characteristics through Combined Pipe Culvert and Broad Crested Weir were investigated using ANSYS-CFX 14. The results of the numerical model were compared with the experimental data of (Othman K. Mohammed 2010).

2. THEORETICAL ANALYSIS

Computational Fluid Dynamics (CFD) involves the solution of the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region. The set of the governing equations of fluid flow which are solved by ANSYS-CFX are the Reynold average Navier-Stokes equations. The governing equations of continuity and momentum for incompressible flow can be written as:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad \ldots\ldots\ldots (1)$$

$$\frac{\partial}{\partial t}(\rho\, u_i) + \frac{\partial}{\partial x_i}(\rho\overline{u_i u_j}) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i + \vec{F} \qquad \ldots\ldots\ldots (2)$$

Where:

ρ = fluid density, $\overline{u_i u_j}$= average velocity in x and y directions, x and y = space dimensions, t = time, P = the pressure, μ =μo + μt , μo is dynamic viscosity and μt is turbulence

viscosity, gi = acceleration due to gravity and $\vec{F}$ = the body force.
The suffices i and j indicate that the stress component acts in the j-direction on a surface normal to the i-direction. (Versteeg H. K. and Malalasekera W. 2007)

ANSYS-CFX code uses finite volume method to convert governing equations to algebraic equation in order to be solved numerically. The Naiver - Stokes equations with time average velocity called Reynolds averaged Naiver – Stokes (RANS) equations. since the Navier-Stokes equations are non-linear, it is difficult to solve them analytically especially for turbulent flow. Because the size of the computational cells should be smaller than the length scale of the smallest turbulent this is impossible which cannot be achieved in many cases (Versteeg H. K. and Malalasekera W. 2007).

Turbulent models have been classified based on the application of their design and number of differential equations to create relation between turbulence stresses and averaged rates or their gradients. Among these models, two-equations model for modeling turbulence with RANS equations have been used, one-layer model such as k – ε and (RNG) k – ε and two-layer model such as shear stress transport (SST).

2.1. Standard k – ε model:

This model expresses the turbulent viscosity in terms of turbulent kinetic energy (k) and its dissipation rate (ε). The following two transport partial differential equations are solved for the values of k and ε (Launder and Spalding 1974)**:**

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial k}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k - \rho\varepsilon \qquad \ldots\ldots\ldots(3)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial \varepsilon}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad \ldots\ldots\ldots(4)$$

The eddy viscosity μt is written as follows

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad \ldots\ldots\ldots(5)$$

where $G_k = \mu_t S^2$

$$S = \sqrt{2S_{ij}S_{ij}} \quad \text{and} \quad S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

Model constants: C1ε = 1.44, C2ε = 1.92, Cμ =0.09, σk =1.0, and σε =1.3.

2.2. Renormalization Group (RNG) k – ε model (Choudhury D. 1993):

The (RNG) k-ε turbulence model is derived from the instantaneous Navier-Stokes equations, from using a mathematical technique called, “renormalization group" (RNG) methods. The analytical derivation results in a RNG model with constants different from those in the standard k-ε model and additional terms and functions in the transport equations for k and ε**.**

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial k}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(a_k\mu_{\varepsilon ff}\right)\frac{\partial k}{\partial x_i}\right]+G_k - \rho\varepsilon \qquad \ldots\ldots\ldots(6)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial\varepsilon}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(a_k \mu_{\varepsilon ff}\right)\frac{\partial\varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}^{*}\rho\frac{\varepsilon^2}{k} \qquad \ldots\ldots\ldots(7)$$

$$G_{2\varepsilon}^{*} = G_{1\varepsilon} + \frac{C_\mu \rho \eta^3 \left(1 - \frac{\eta}{\eta_o}\right)}{1 + \beta\eta^3} \qquad \ldots\ldots\ldots(8)$$

in which $\eta = \frac{S\,k}{\varepsilon}$

In above Equations, C1ε, C2ε, and Cμ are constants and equal to 1.42, 1.68, and 0.0845, respectively. ak and aε equal to 1.393, ηo equal to 4.38, μεff equal to 1 and β equal to 0.012**.**

2.3. Shear Stress Transport (SST) model (Menter FR, 1994):

Menter (1994) developed the SST turbulence model to blend effectively the robust and accurate formulation of the k- ω model in the near-wall region with the free stream independence of the k- ω model in the far field. It is an eddy-viscosity model which includes two main novelties:

- It is combination of a k-ω model (in the inner boundary layer) and k-ε model (in the outer region of and outside of the boundary layer);
- A limitation of the shear stress in adverse pressure gradient regions is introduced.

The transport equations and effective viscosity are modelled in SST k-ω model, by the following equations**:**

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - Y_k \qquad \ldots\ldots\ldots(9)$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\rho\omega u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial\omega}{\partial x_j}\right] + G_\omega - Y_\omega + D_\omega \qquad \ldots\ldots\ldots(10)$$

$$\mu_t = \frac{\rho k}{\omega}\frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{\sqrt{2\Omega_{ij}\Omega_{ij}}F_2}{\alpha_1\omega}\right]}$$

in which $\Omega_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right)$

α* damps the turbulent viscosity causing a low Reynolds number correction

G_k, G_ω is the generation of k and ω

and $D_\omega = 2(1 - F_1)\ \frac{\rho\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j}$

$$\sigma_k = \frac{1}{F_1/\sigma_{k1} + (1 - F_1)/\sigma_{k2}}$$

$$\sigma_\omega = \frac{1}{F_1/\sigma_{\omega 1} + (1 - F_1)/\sigma_{\omega 2}}$$

F1, F2 are the blending functions

Y_k, Y_ω represent the dissipation of k and ω due to turbulence

3. EXPERIMENTAL DATA

The experimental data (Table 1) used for the comparisons were taken from laboratory tests conducted by (Othman K. Mohammed 2010). The geometry and dimensions of the combined broad crested weir and pipe culvert model are stated in Fig. (1). The experiments were conducted in a horizontal research flume with a width of 0.5 m, a height of 0.5 m and a total length of 12 m. the laboratory model was made of concrete box shape like of dimensions (50 x 50 x 13.1 cm), containing a plastic pipe 10.6 cm diameter. The notations in this paper are kept identical to those defined by (Othman K. Mohammed 2010).

Table 1, Experimental data.

Run	H/P	Q l/sec	Cd
1	0.180	10.780	0.526
2	0.240	12.360	0.521
3	0.310	15.140	0.542
4	0.400	17.950	0.527
5	0.480	21.620	0.531
6	0.510	22.720	0.535
7	0.560	24.900	0.53
8	0.590	28.270	0.569
9	0.760	37.010	0.566
10	0.930	47.350	0.575

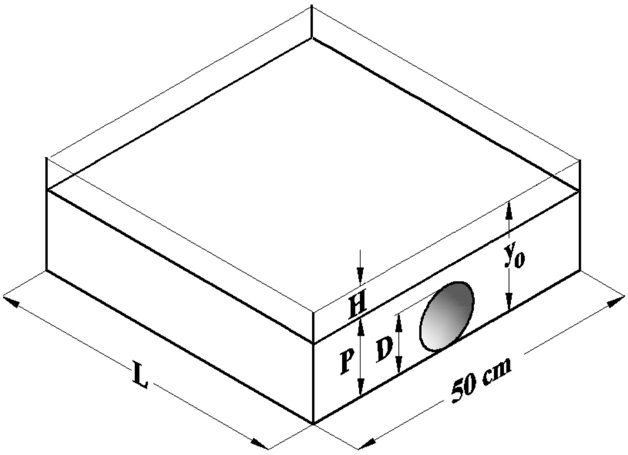

Fig. (1) Geometry of the tested Model D = 10.6 cm, P = 13. 1cm, L = 50 cm

4. NUMERICAL MODELLING

The numerical model was constructed at the same dimensions as the physical model. This allows direct comparison of the predicted results with physical model results.

4.1. Mesh Design

Meshing is an important step to solve the hydraulic systems in numerical modelling. According to earlier studies, the smaller the mesh size the greater is the accuracy and the more is the computational time (Aziz, Y. W. 2016). ANSYS ICEM was used for mesh generation as it contains many methods for mesh generation. In this study the Multi-zone method was used with the maximum and minimum mesh size of 0.05 m and 0.000196 m respectively such as mesh sizes used by (Aziz, Y. W. 2016), and the hexahedron mesh type was provided as shown in Fig. (2).

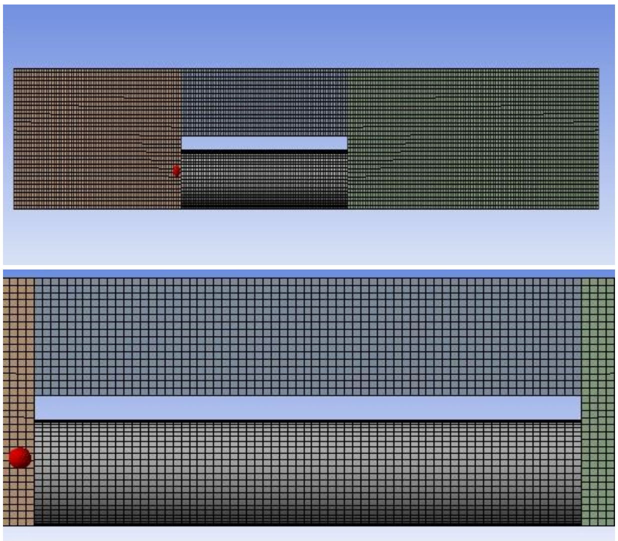

Fig. (2) Meshing and its Distribution

4.2. Boundary conditions

ANSYS-CFX contains several boundary conditions including inlet, outlet, opening and wall. Fig. (3). Inlet boundary condition imposed at inlet section with the average velocity, water and air volume fraction. Static pressure used at the outlet and opening boundary condition was specified for the top of the fluid domain. On the walls, the no slip wall boundary condition was applied; that is the fluid velocity next to the wall immediately is equal to zero. Walls were assumed to be smooth, since the channel sides were made from glass.

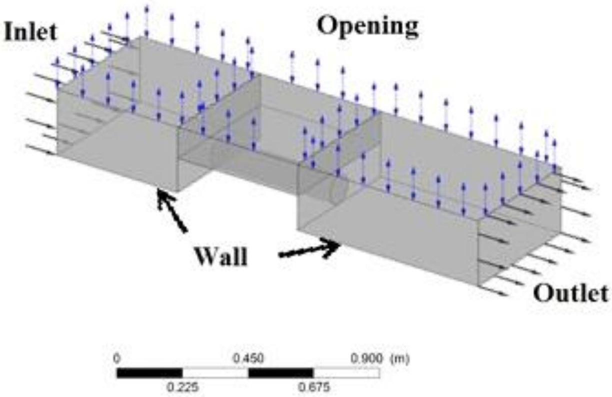

Fig. (3) Boundary conditions

5. RESULTS AND DISCUSSION

To have access to an appropriate turbulence model for the simulation, the numerical model is examined with different models of turbulence such as standard k – ε model, RNG k – ε model and SST model under the same conditions (boundary condition, material, mesh and so on). Then the results of these turbulence models are compared with those provided by the experimental data

The experimental and numerical results of discharge through combined broad crested weir and pipe culvert were plotted as shown in Fig. (4). The results for all turbulence models with the experimental data are very close to each other, but some of them are in closer agreement to the experimental data as presented in table 1.

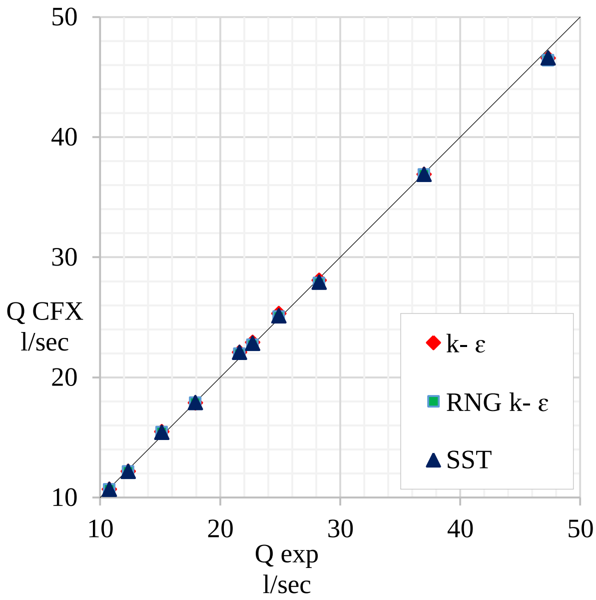

Fig. (4) Comparison of Q_{CFD} with Q_{Exp} for Combined Weir and Culvert

Slight deviations are observed between the predicted by numerical model and the measured values from Fig. (4)

Fig. (5) shows the 3D and longitudinal section at the center line of the simulated velocity distribution through the culvert and over the weir predicted by the k- ε turbulence model for discharge flow rate of 28.27 l/s. Since from the experiment there is no any measurement of velocity, so the comparion with the CFD modelling can not be done.

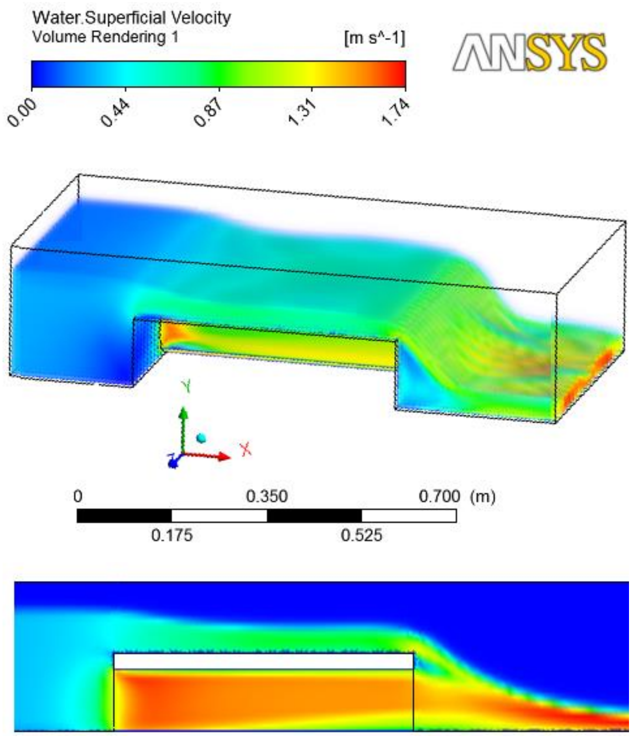

Fig. (5) Velocity distribution through combined structure for discharge 28.27 (l/sec) using k-ε turbulence model.

Table (2) shows that discharge coefficient results obtained from SST turbulence are mostly closer than the other models to the experimental data. Further, it can be observed that the RNG k- ε turbulence model performs better than k- ε model. In addition, k- ε has lesser agreement with the experimental data as it has higher error percentage.

Table (2) Discharge coefficient and relative error of the numerical simulations

Exp.	CFX					
	Standard k- ε		RNG k- ε		SST	
Cd	Cd	Error %	Cd	Error %	Cd	Error %
0.526	0.521	1.018	0.519	1.267	0.521	0.939
0.521	0.514	1.433	0.512	1.742	0.513	1.541
0.542	0.554	2.163	0.552	1.849	0.552	1.934
0.527	0.525	0.391	0.525	0.468	0.525	0.288
0.531	0.542	2.079	0.539	1.468	0.542	2.064
0.535	0.539	0.840	0.535	0.023	0.538	0.564
0.53	0.539	1.658	0.533	0.554	0.535	0.888
0.569	0.565	0.675	0.560	1.536	0.562	1.280
0.566	0.565	0.202	0.565	0.263	0.565	0.188
0.575	0.566	1.625	0.563	2.064	0.566	1.582

Discharge coefficients Resulted from Applying turbulent models together with experimental values plotted against relative upstream water depth (H/P) are shown in Fig. (6). In this figure H and P stand for the head of water above the weir and weir height respectively. It can be seen that, for all cases Cd increased with (H/P) increasing this is due to increase the ratio of (Flow cross sectional Area/Contracted parameter).

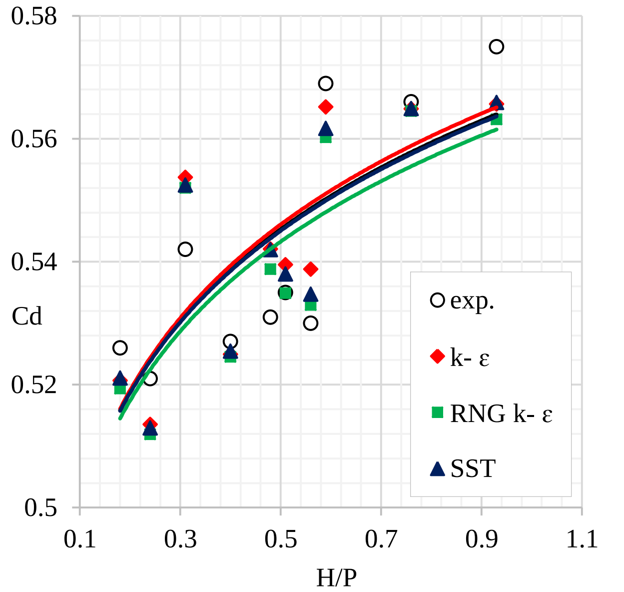

Fig. (6) Head-discharge coefficient of the numerical simulations and Experimental data

In order to determine the accuracy of the simulation results, the Relative Error percent (E %) of the experimental and the numerical discharge results are calculated using the equation:

$$E\ \% = 100\frac{1}{n}\sum_{i=1}^{n}\left|\frac{Cd_{CFX}-Cd_{Exp}}{Cd_{Exp}}\right|$$

Table (3) Relative Errors of the average discharge coefficient

Exp.	CFX					
	Standard k- ε		RNG k- ε		SST	
Cd	Cd	Error %	Cd	Error %	Cd	Error %
0.542	0.543	1.208	0.540	1.123	0.542	1.127

Errors for discharge coefficient Resulting from applying different turbulent models are shown in Fig. (7).

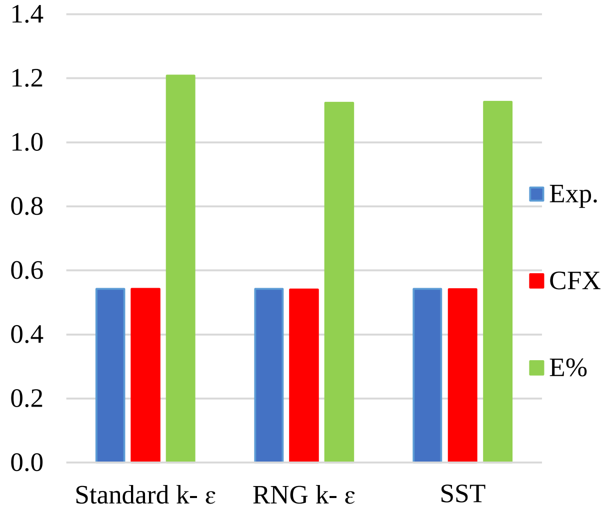

Fig. (7) Discharge coefficient and their Relative Errors from Applying Different Turbulent Models

It can be seen that (Fig 6 and Fig 7) the results of k- ε turbulence model for discharge coefficient are greater than those obtained by experiments, but those of both (RNG) k – ε and SST are lower than experimental results. From table (3) it is clear that higher percentage error obtained by using k – ε turbulence model, while using RNG k – ε has lower average percentage error with the experimental data. These differences can be clearly seen in charts shown in Fig. (7).

6. CONCLUSIONS

In the present study, flow over broad crested weir combined with the circular culvert is simulated using ANSYS – CFX. The sensitivity of the results obtained from CFD modeling of different turbulence models is examined, to determine the turbulence model that gives accurate predict of flow through the combined structure. The summery of the results of this study can be defined as follows:

1) SST model perform much better has the maximum accuracy in comparison with other turbulence models for most discharges values, but for average relative percentage error, (RNG) k- ε has a bit greater accuracy than SST.
2) The results of k- ε have less accuracy compared with other turbulence models.
3) The discharge coefficient in all methods slightly increased with upstream relative head (H/P) which was (from 0.526 to 0.575) such variation may be considered as constant, same conclusion indicated by (Hager W. and Schwalt M. 1994) for broad crested weir flow.
4) It is also concluded that, such type of software is useful to study the number of fluid flow problems without going for expensive and time consuming experiments.

REFERENCES

Afshar, H. and Hoseini, H. "Experimental and 3D Numerical Simulation of Flow over a Rectangular Broad-Crested Weir". International Journal of Engineering and Advanced Technology (IJEAT) August 2013, Volume-2, Issue-6, 214-219.

Al-Hashimi A. S., Madhloom M. H., and Nahi N. T. "Experimental and Numerical Simulation of Flow Over Broad Crested Weir and Stepped Weir using Different Turbulence Models", Journal of Engineering and Sustainable Development Vol. 21, No. 02, March 2017.

Aziz, Y. W. "Evaluation of Hydraulic Performance of Nazanin Dam Side Channel Spillway" MSc Thesis, College of Engineering Salahaddin University Erbil 2016.

Choudhury D., (1993). "Introduction to the Renormalization Group Method and Turbulence Modeling", Fluent Inc. Technical Memorandum TM-107.

Duangrudee and Kositgittiwong "Validation of Numerical Model of the Flow Behavior through Smooth and Stepped Spillways using Large-scale Physical Model" PhD Thesis, Faculty of Engineering, King Mongkut's University of Technology Thonburi 2012.

Duru, Aysel "Numerical Modelling of Contracted Sharp Crested Weirs" MSc Thesis, School of Natural and Applied Sciences of Middle East Technical University, 2014

Hager W. H. and Schwalt M. "Broad-crested weir", Journal of Irrigation and Drainage Engineering, Vol. 120, No. 1, (1994), pp.13-26.

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

Hoseini, S.H. "3D Simulation of Flow over a Triangular Broad-Crested Weir", Journal of River Engineering, 2(2), 1-7 (2014).

Jalil, Shaker & Qasim, Jihan. (2016). “Numerical Modelling of Flow over Single-Step Broad- Crested Weir Using FLOW-3D and HEC-RAS”. Polytechnic General Sciences Journal/ Erbil Polytechnic University. 6. 435-448.

Launder, B. E., and Spalding, D. B. “The numerical computation of turbulent flows.” computer methods in applied mechanics and engineering 3 (1974) 269-289.

Mahmoud S. M. "Characteristics and Prediction of Simultaneous Flow Over Broad-Crested Weirs and Through Culverts", EJEST, Vol, 6, No.1, January 2002

Menter FR (1994). "Two-equation eddy viscosity turbulence models for engineering applications". AIAA Journal 32(8):1598-1605.

Negm A. M. “Analysis and modeling of simultaneous flow through box culverts and over contracted broad-crested weirs” Proc. of 5th International Conference on Hydro-science and Engineering. ICHE2002. Sept. 18- 21. Warsaw, Poland.

Othman K. Mohammed "Flow Characteristics through Pipe Culvert Combined with Broad Crested Weir" MSc Thesis, College of Engineering Salahaddin University Erbil 2010.

S. Hoseini, S. Jahromi, M. Vahid “Determination of Discharge Coefficient of Rectangular Broad-Crested Side Weir in Trapezoidal Channel by CFD” IJHE 2013, 2(4): 64-70

Sarker, M.A and Rhodes, D.G," Calculation of free surface profile over a rectangular broad-crested weir". Flow Measurement and Instrumentation, 2004, 15(4) 215-219.

Versteeg H. K. and Malalasekera W. "An Introduction to Computational Fluid Dynamics" Second Edition, Pearson Education Limited 2007, pp 14, 66.

Wilcox D. C., "Turbulence Modeling for CFD", DCW Industries Inc., La Canada, California (1993).