International Journal of
Scientific Engineering and Technology Research

ISSN 2319-8885
Vol.02,Issue.15,
November-2013,
Pages:1682-1687

www.semargroups.org,
www.ijsetr.com

CFD Modeling of Flow over Ogee Spillway by Using Different Turbulence Models

DR. SHAYMAA ABDUL MUTTALEB ALHASHIMI
Lecturer in Highway Engineering Department, College of Engineering, University of Al-Mustansiriya, Iraq,
E-mail: shyabed@yahoo.com.

Abstract: Ogee spillway are the most important structures used to control flood water and built at the same time concrete or masonry dams are constructed. Due to such an importance, they shall be studied. The development of computer science and different types of computational fluid dynamics (CFD) software, the behavior of ogee spillways can be studied in a short time and without paying high expenses. In this study, FLUENT software has been used to simulate flow over ogee spillway and results were compared with experimental data. As flow over ogee spillway is turbulent and has a free surface, its characteristics are complex and often difficult to be predicted. This study assesses the performance of different turbulence models to predict the hydraulic condition of flow over ogee spillway. The volume of fluid (VOF) method is applied to obtain the free surface in each case. The results indicate that in the case the RNG k-ε turbulent model is used, the accuracy of the results obtained from the flow over ogee spillway is increased.

Keywords: Ogee Spillway, Numerical CFD Model, Turbulent Models, Fluent, VOF.

1. INTRODUCTION

Spillway is a hydraulic structure that usually is used in detention and storage dams to release extra water and flood in emergency situations. Up to now many dams are destroyed because of inefficiency and non-convenient design of their spillways. Therefore, the analysis of water flow over a spillway is an important engineering problem (Chatila and Tabbara, 2004). In recent years, computational fluid dynamics (CFD) has been used to compute flow over an ogee crest. The results have been compared and evaluated against physical model data. In (1998), Olsen and Kjellesvig used Reynolds equations and standard k - ε equation in finite volume method to analyze the flow over ogee spillway in 3D and 2D spaces. Burgisser and Rutschmann (1999) used finite element method to solve the Navier-Stokes equations considering eddy viscosity. It used both Galerkin and least-squares methods for integration. Savage and Johnson (2001) simulated flow over an ogee spillway using Flow-3D software in 2D.

The results of the numerical model including pressure on the spillway crest, water surface profile, and discharge coefficient of the spillway were in very good agreement with the experimental values. Bouhadji (2002) conducted a numerical study on turbulent flow over spillways in a 3D space. In 2004, Jean and Mazen modeled flow over ogee spillway numerically. Kim and Park (2005) used the commercial numerical model of computational fluid dynamics of Flow-3D software to study the properties of flow including flow rate, water surface profile, pressure imposed on the crest of ogee spillway, pressure vertical distribution and speed based on the scale of model, the impact of surface roughness and details. Bhajantri et. al. (2006) studied the hydraulic model of flow over ogee spillway numerically considering downstream and the information obtained from two physical models have been compared with the results obtained from numerical study of two crested ogee spillways. Ferrari (2010) used a mesh less method called SPH1 to analyze the flow over ogee spillway. The results of numerical model compared to physical model were in good agreement. In this study, numerical analyses of different turbulent models are compared to calculate properly the profile of flow over ogee spillway using finite volume method.

2. RESEARCH METHODOLOGY

In the present study, FLUENT software was employed in order to study the physical properties of the field of flow over ogee spillways. For this purpose, it is required to plot the geometry of model and meshing using GAMBIT software. FLUENT is one of the most popular and suitable software of CFD that provides a wide range of advanced physical models for fluid flow and heat transfer including multiphase flow. It can exchange 2D and 3D dominant differential equations to algebraic equations by using the finite volume method. This software uses volume of fluid

(VOF) model to determine the free surface of flow. Nichols and Hirt (1981) provided a model to determine the common surface of two-phase fluid, which is used in many hydraulic problems. In this study, 90% of the volume of cell is water and the remaining part consists of air. The equations governing flow over spillway in this software include Stokes, Reynolds averaging method (RANS) has been used in this study in such a way that the fluctuations of turbulent flow are inserted into the equations indirectly (Fluent Inc., 2006). To solve the field of turbulent flow based on continuity equations Eq. (1) and Reynolds Averaged Navier – Stokes equation Eq. (2), it is required Reynolds Stress $(-\rho\overline{\acute{u}_i\acute{u}_j})$ equations to be modeled using specified methods.

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. These models include zero equation models, one equation model (Spalart – allmaras model), two equation models, algebraic stress model, Reynolds stress model, Reynolds stress models (five equation model). Among these models, two-equation model has been studied due to the satisfactory results and simplicity of application for ogee spillways, Bruce et. al. (2006). This model has also been divided into the following classes: standard k – ω model, (shear stress transport (sst) k –ω model) and k – ε model (standard k – ε model, realizable k – ε model, renormalization group (RNG) k – ε model), Wilcox, (1993). These models are able to solve two differential equations. Equation of ε or ω has been added to the equation of k. The equation of turbulence kinetic energy, k, expresses the scale of velocity and the equation of turbulence kinetic damping rate, ε or ω, has also the scale of length.

$$\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] \quad (2)$$

Where;

$$(-\rho\overline{\acute{u}_i\acute{u}_j})=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\left(\rho k+\ \mu_t\frac{\partial u_i}{\partial x_i}\right)\delta_{ij} \quad (3)$$

Where u_i represent the velocities in the x_i directions which are x, y, z-directions; t is time, ρ is the volume-fraction-averaged density, p is revised pressure, μ_t turbulent viscosity, μ is molecular viscosity, k turbulent kinetic energy and δ_{ij} Kronecher delta.

2.1. Two Equations Turbulent Models

2.1.1. k – ω Turbulent Model k - ω equation:

$$\frac{\partial k}{\partial t}+u_j\frac{\partial k}{\partial x_j}=\frac{1}{\rho}\tau_{ij}\frac{\partial u_i}{\partial x_j}-\beta^* k\omega \quad (4)$$

$$\frac{\partial\omega}{\partial t}+u_j\frac{\partial\omega}{\partial x_j}=\frac{\omega}{\rho k}\tau_{ij}\frac{\partial u_i}{\partial x_j}-\beta k\omega^2+\frac{1}{\rho}\frac{\partial}{\partial x_j}\left[(\mu+\sigma\mu_t)\frac{\partial\omega}{\partial x_j}\right] \quad (5)$$

2.1.2. k - ε Turbulence Model

2.1.2.1. Standard k - ε turbulence model

Launder and Spalding (1974) proposed the standard *k* - *ε* model that was a semi-empirical model based on the model transport equations of the turbulent kinetic energy (*k*) and its dissipation rate (*ε*).

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial k}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k-\rho\varepsilon \quad (6)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial k}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]+c_{\varepsilon 1}\frac{\varepsilon}{k}G_k-c_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \quad (7)$$

The eddy viscosity μ_t is written as follows:

$$\mu_t=\rho c_\mu\frac{k^2}{\varepsilon} \quad (8)$$

σ_k and σ_ε are turbulence Prandtl numbers for k and ε respectively. The model constants as:

c_μ	$c_{\varepsilon 1}$	$c_{\varepsilon 2}$	σ_k	σ_ε
0.09	**1.44**	**1.92**	**1.00**	**1.30**

2.1.2.2. RNG k - ε Turbulence Model

The RNG *k* - *ε* model was derived from the instantaneous Navier-Stokes equations, using a mathematical technique called “renormalization group” (RNG) method. The model constants, additional terms and functions of the RNG *k* - *ε* model are different from the standard *k* - *ε* model. The more description of RNG theory can be found in Choudhury (1993).

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial k}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[(\alpha_k\mu_{eff})\frac{\partial k}{\partial x_i}\right]+G_k-\rho\varepsilon \quad (9)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial k}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[(\alpha_k\mu_{eff})\frac{\partial\varepsilon}{\partial x_i}\right]+c_{\varepsilon 1}\frac{\varepsilon}{k}G_k-c^*_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \quad (10)$$

$$c^*_{\varepsilon 2}=c_{\varepsilon 2}+\frac{c_\mu\rho\eta^3\left(1-\frac{\eta}{\eta_o}\right)}{1+\beta\eta^3}$$

$$\eta=\frac{sk}{\varepsilon} \quad (11)$$

The scale elimination procedure in RNG theory results in a differential equation for turbulent viscosity. In high Reynolds number $\mu_{eff}\approx 1$. Model constant is:

c_μ	$c_{\varepsilon 1}$	$c_{\varepsilon 2}$	$\alpha_k=\alpha_\varepsilon$	η_o	β
0.0845	**1.42**	**1.68**	**1.393**	**4.38**	**0.012**

2.1.2.3. Realizable k - ε Turbulence Model

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial k}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k-\rho\varepsilon \qquad (12)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial k}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]+\rho c_1 s_\varepsilon-\rho c_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}} \qquad (13)$$

$$c_1=max.\left[0.43,\frac{\eta}{\eta+5}\right] \quad , \quad \eta=\frac{sk}{\varepsilon} \qquad (14)$$

$$\mu_t=\rho c_\mu\frac{k^2}{\varepsilon}$$

$c_{\varepsilon 1}$	c_2	σ_k	σ_ε
1.44	**1.9**	**1.0**	**1.2**

The term G_k, representing the production of turbulent kinetic energy, is modeled identically for the standard, RNG, and realizable *k* - *ε* models. From the exact equation for the transport of *k*, this term may be defined as:

$$G_k=-\rho\overline{\acute{u}_i\acute{u}_j}\frac{\partial u_i}{\partial x_j} \qquad (15)$$

3. CASE STUDY

3.1. Physical model

In the present study, experimental findings of Mandali dam ogee spillway were used in order to calibrate the numerical models. A scheme of the ogee spillway is depicted in Figure (1). The geometric parameters of the investigated ogee spillway are as follow, (Rafidain State Company for Dams and Construction, 2008):
Design head (Hd) = 2.5 m, width = 250 m, height of spillway crest (P) = 10 m, crest level at elevation of 180 m.a.m.s.l. and maximum design discharge is 1724 m^3/s.

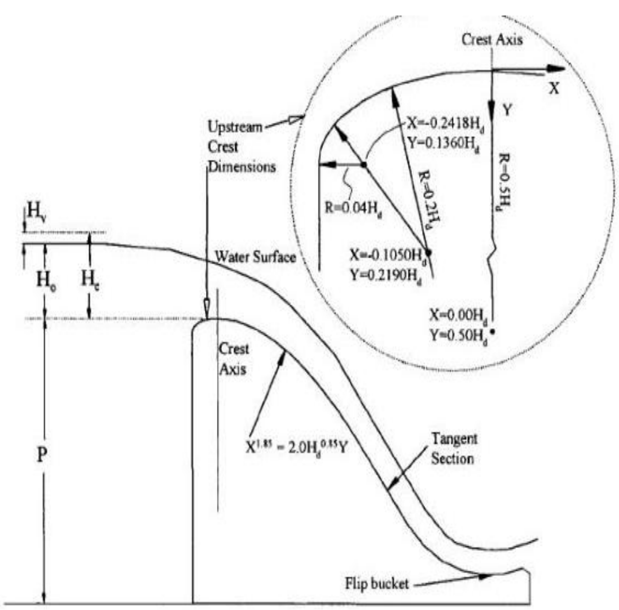

Figure1. The Dimensions of Ogee Spillway and Flow Parameters

3.2. Boundary Conditions

In general, one of the most important phases of the numerical analysis of flow field is to determine proper boundary conditions, which are matched appropriately with the physical conditions of the problem. In this study for numerical modeling of ogee spillway, the boundary conditions of the Figure (2) have been provided. According to Fig.(2), the original boundary conditions of flow over ogee spillway include inlet (velocity inlet for water, and pressure inlet for air), outlet (pressure outlet), stationary wall (wall) and free surface (pressure inlet).

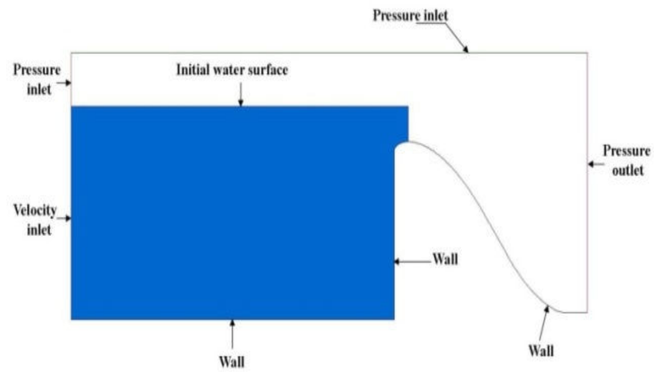

Figure2. Original Boundary Conditions of the Field of Flow over Ogee Spillway

3.3. Analysis Method

At first, it is required to verify the accuracy of meshing, correctness of the choice of turbulence model and ensure that they have no effect on the results. Considering that ogee spillway is curved in form, it is recommended that quad-

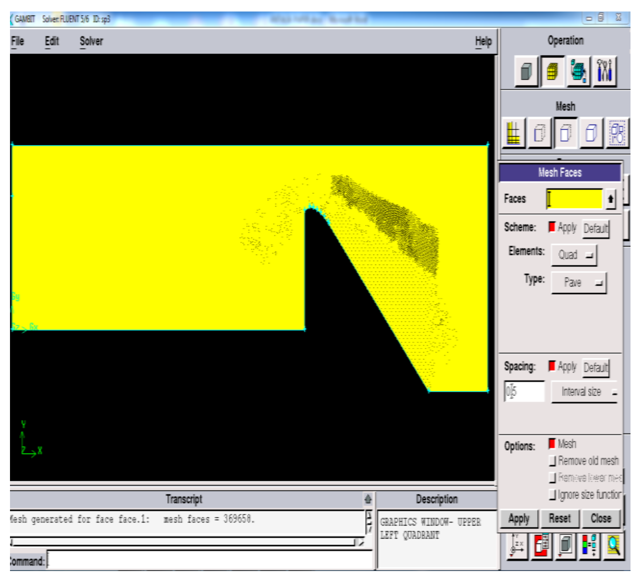

Figure3. Meshing and its Distribution in the Model of Ogee Spillway

pave meshing to be used to minimize the relative error of discharge parameter and optimize the time of numerical solution, (Daneshkhah, Vosoughifar 2011). The measure of meshing, time and step time has been chosen equal to 0.05

m, 50 seconds, and 0.01 seconds, respectively. The mesh used in this study as shown in Fig. (3). The governing equations are discretized using the finite volume method with the PRESTO pressure discretization scheme. The coupling of the pressure and the velocity is achieved through the PISO algorithm which used purely because it is designed specifically for transient simulations. The calculation domain is divided into discrete control volumes by the unstructured grid, which has a high flexibility to fit the complex geometry of spillways, Isaa (1986).

To have access to an appropriate turbulence model for calculating the parameters of flow over ogee spillway, numerical model is applied with different models of turbulence under the same conditions. Then the best and worst simulated flow profile can be accessed in comparison with laboratory model, which have been shown in Figures (4, 5, 6 & 7).

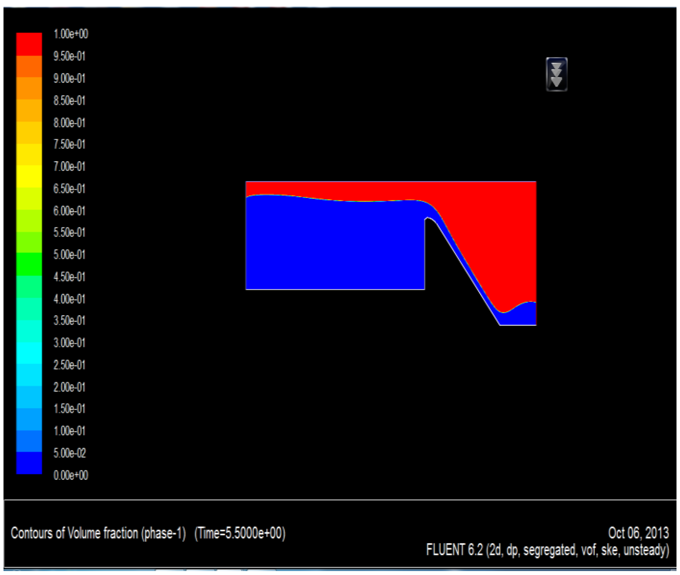

Figure4. Flow Surface Profile for Standard (k- ε) Turbulent Models

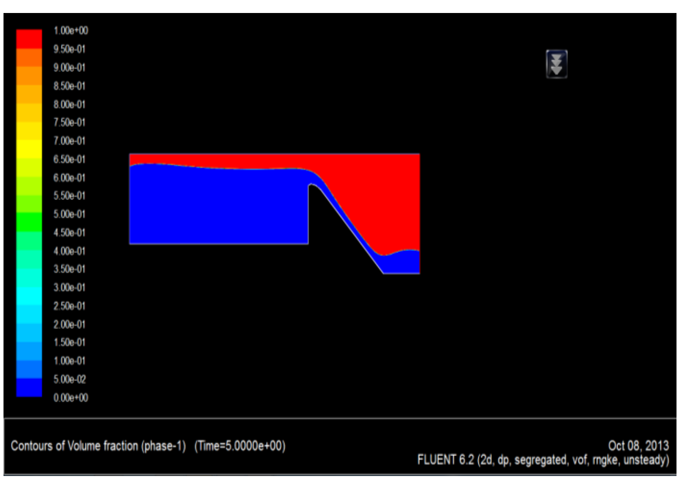

Figure5. Flow Surface Profile for RNG (k-ε) Turbulent Models

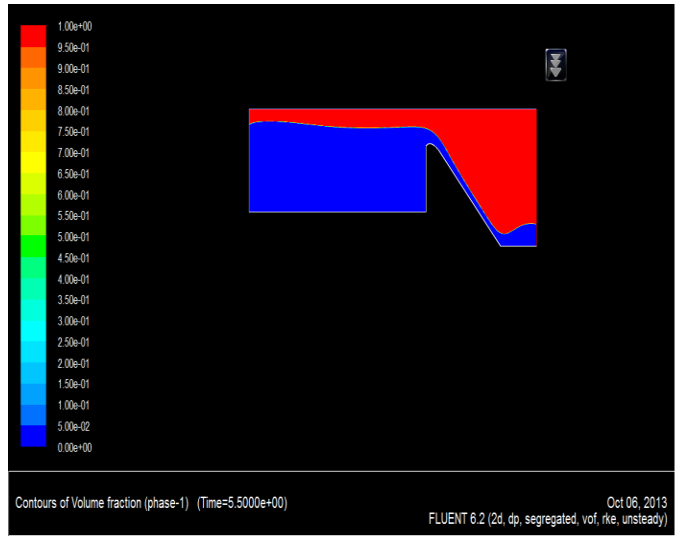

Figure6. Flow Surface Profile for Realizable (k- ε) Turbulent Models

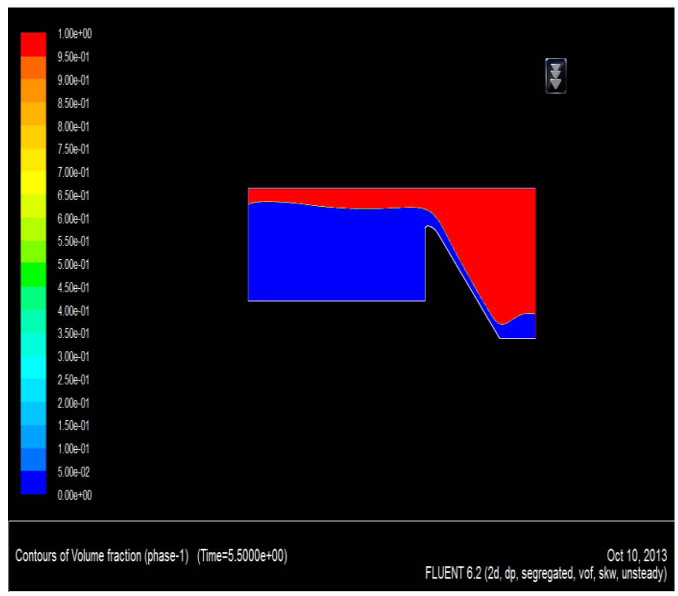

Figure7. Flow Surface Profile for Standard (k- ω) Turbulent Models

4. RESULTS AND DISCUSSION

For each turbulent models applying, the ratio of flowrate per unit width (q) over design flowrate (q_d = 6.896 m^3/s) was calculated as shown in table (1). The error (the difference between results obtained from the numerical and experimental models) are shown in Fig. (8). It can be seen that the results of numerical model for flowrate are greater than those obtained by experiments. The minimum and maximum error values were 3.7% and 7.4% respectively. The study of the flow profile of different turbulent models based on table (1) indicates that the flow profile obtained from RNG k – ε model is more similar to laboratory model, and in contrary the standard k–ω model has the minimum similarity with laboratory model.

Table1: Dimensionless Flowrates Obtained by the Numerical and Experimental Models

Turbulent Model	Standard k- ε	RNG k- ε	Realiza-ble k- ε	Standard k- ω
$q_{Exp.}/q_d$	0.312	0.312	0.312	0.312
$q_{Num.}/q_d$	0.326	0.428	0.332	0.335
Error %	4.5	3.7	6.4	7.4

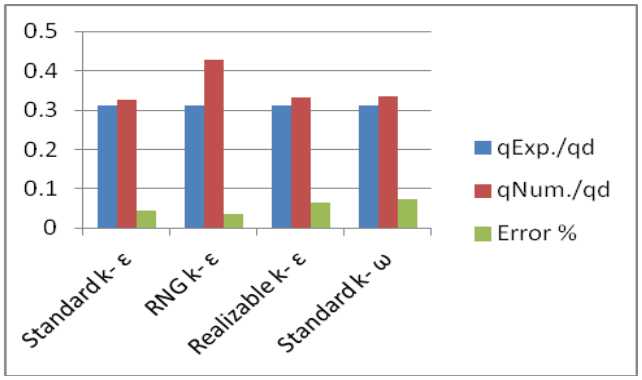

Figure8. Errors for Flowrate Resulting from Applying Different Turbulent Models

By determining the flow depth and knowing the flowrate the mean velocity at any section can be calculated. Dividing the mean velocity by the design velocity as $v_d = [2g (Z-(H_d/2))]^{1/2}$ gives the dimensionless ratio of velocities. H_d is the design head and Z is the vertical distance between the upstream flow surface and the bottom of the toe. These value are presented in table (2). Figure (9) shows the errors (difference between values obtained from the numerical and experimental models) for the mean velocity at crest of spillway. The minimum error of 3.2% result from applying RNG k- ε turbulent model while the maximum error of 11% occurred for applying standard k- ε model under the same conditions. consequently, it can be conculuded that the RNG k- ε turbulent model is the best for modeling flow over ogee spillway.

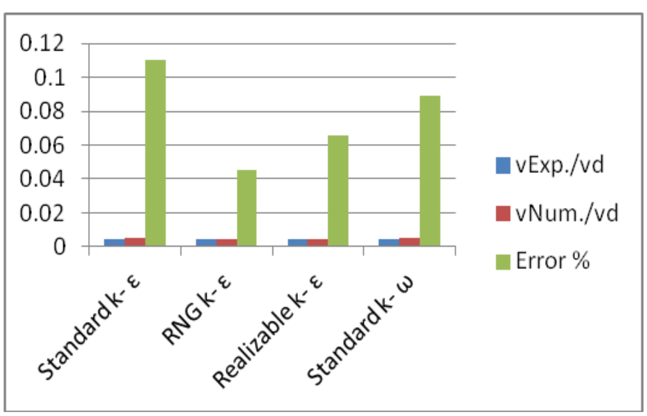

Figure9. Errors for the Mean Velocity Resulting from Applying Different Turbulent Models

Table 2: Dimensionless Flowrates Results from Numerical and Experimental Models

Turbulent Model	Standard k- ε	RNG k- ε	Realizable k- ε	Standard k- ω
$v_{Exp.}/v_d$	0.0045	0.0045	0.0045	0.0045
$v_{Num.}/v_d$	0.005	0.0047	0.0048	0.0049
Error %	11	4.5	6.6	8.9

Also, it can be seen the distribution of pressure over ogee spillway by using RNG k- ε turbulent model as shown in Fig. (10).

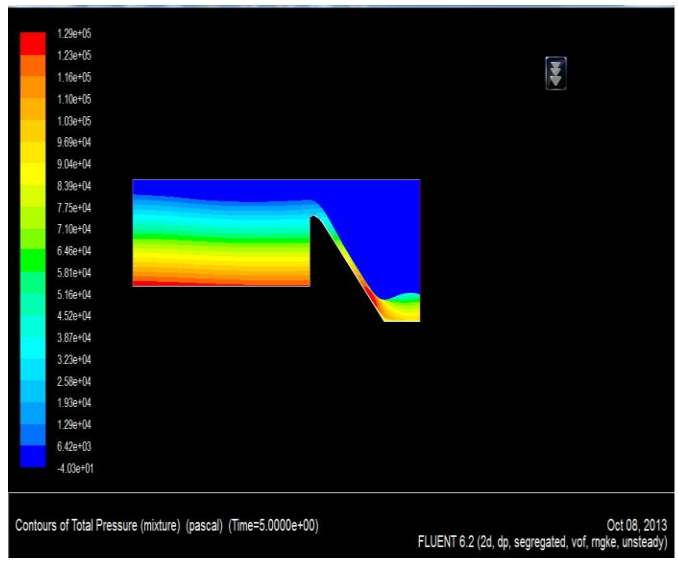

Figure10. Distribution of Total Pressure over Ogee Spillway Using RNG k- ε Model

5. CONCLUSION

In the present numerical study, flow over ogee spillway was simulated by using 2D code (FLUENT software). The different k-ε turbulence models with the VOF method and unstructured grid can simulate the ogee spillway overflow successfully. The VOF model of the air and water can well trace the free water surface based on the time-dependent simulation. The difficult to treat complex boundary of the ogee spillway was overcome by using unstructured quad.-pave grid of 0.05 * 0.05 m^2 to discrete the domain using Gambit. The solution of the free water surface has significance in the engineering. Results obtained in this study showed that the flow parameters such as the flowrate and mean velocity obtained by the FLUENT software numerical model for turbulent RNG k- ε model were in good agreement with the experimental results than other turbulent models for flow over ogee spillway and can be used for analysis or design of such hydraulic structures. The predicted values for flowrate by the numerical model were greater than those obtained by the experimental results with minimum error result from applying RNG k- ε turbulent model.

6. REFERENCES

[1] Chatila J, Tabbara M. (2004). "Computational Modeling of Flow over an Ogee Spillway". Journal of computers and structures, 82: 1805-1812.

[2] Olsen, N.R.B. and Kjellesvig, H.M., (1998),“Three Dimensional Numerical Flow Modeling for Estimation of Spillway Capacity,” J. Hydr. Res, 36(5), pp 775-784.

[3] Burgisser, M. F., and Rutschmann, P., (1999), "Numerical Solution of Viscous 2DV Free Surface Flows: Flow over Spillway Crests", Proceedings of 28th IAHR Cong., Technical University Graz, Graz., Austria.

[4] Savage B.M, Johnson C. (2001), "Flow over Ogee Spillway: Physical and Numerical Model Case Study". Journal of Hydraulic Engineering, ASCE, 127(8): 640-649.

[5] Bouhadji, L., (2002), ” Three Dimensional Numerical Simulation of Turbulent Flow Over Spillways,”, Asl-Aqflow Inc., Sidney, British Columbia, Canada.

[6] Jean, C. and Mazen, T., (2004), ”Computational Modeling of Flow Over an Ogee Spillway,” Department of Civil Engineering, School of Engineering and Architecture, Lebanese American University, P.O. Box.36, Byblos, Lebanon.

[7] Kim, Dae Geun. and Park, Jae Hyun., (2005), ”Analysis of Flow Structure Over Ogee-Spillway in Consideration of Scale and Roughness Effects by Using CFD Model,” Ksce Journal of Civil Engineering, Vol. 9, No.2, pp 161-169.

[8] Bhajantri, M.R. and Eldho, T.I. and Deolalikar, P.B., (2006), “Hydrodynamic Modeling of Flow Over a Spillway Using a Two Dimensional Finite Volume Based Numerical Model,” Sadhana, pp 743-754.

[9] Ferrari A. (2010). "SPH simulation of free surface flow over a sharp-crested weir". Advances in Water Resources, 33: 270-276.

[10] Hirt, C. W. and Nichols, B. D., (1981), ‘‘Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries’’ J. of Computational Physics, 39, 201–225.

[11] Fluent Inc, (2006), “User’s guide,” ver. 6.3.26.

[12] Bruce, M., Johnson, Michael C. and Savage, (2006),” Physical and Numerical Comparison of Flow Over Ogee Spillway in the Presence of Tailwater,” Journal of Hydraulic Engineering, Vol.132, No.12.

[13] Wilcox DC (1993), "Turbulence Modeling for CFD". Dcw Industries, Inc, La canada, California Launder, B.E. and D.B. Spalding., (1974). "The Numerical Computation of Turbulent Flows". Computer Methods in Applied Mechanics and Engineering 269-289.

[14] Choudhury, D., 1993. "Introductuon to the Renormalization Group Method and Turbulence Modeling". Fluent Inc. Technical Memorandum TM-107.

[15] Rafidain State Company for Dams Construction, (2008), "Hydraulic Model Study of Mandali Dam Spillway", Final Report, Ministry of Water Resources.

[16] Daneshkhah, AR, Vosoughifar, HR (2011), "Numerical Investigation of Passed Flow Different Parameters Over a Standard Ogee Spillway to Satisfy Flow Profile in CFD Method". 5th SAS Tech. Khavaran Higher-Education Institute Mashhad Iran.

[17] Issa, R. I., (1986), "Solution of Implicitly Discretized Fluid Flow Equations by Operator Splitting", J. Comput. Phys. 62: 40-65.