International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249 – 8958, Volume-2, Issue-6, August 2013

Experimental and 3-D Numerical Simulation of Flow over a Rectangular Broad- Crested Weir

Hossein Afshar, Seyed Hooman Hoseini

Abstract- A broad-crested weir is a flat-crested structure with alarge crest length compared to the flow thickness which is widely used in open channels, rivers, irrigation and drainage systems. New experiments were conducted on a rectangular broad-crested weir with a rounded corner. In this study, Computational Fluid Dynamics (CFD) model together with laboratory model were used in order to determining the free-surface profile of rectangular broad-crested weir. Simulations were performed using the volume of fluid (VOF) free surface model and 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.The structured mesh with high concentration near the solid regions was used in te numerical procedure. The computational results showed a close agreement with experimental data obtained in the laboratory. Also, results indicate that RNG model has the minimum level.

Index Terms—Broad-crested weir, CFD, Laboratory model, Free surface profile

I. INTRODUCTION

Broad-crested weirs are defined as structureswhere the streamlines run parallel to each otherover the weir crown, and the crest of the weir ishorizontal [4]. In most layouts of broad-crested weirs also the hydrostatic pressure is fullyaccomplished in the middle of the crest. However,in cases where the weir length is too small it might be that the hydrostatic pressure is not fully accomplished.The broad-crested weir is in addition to irrigation systems used for highways, railroads and for hydropower structures.Also an application as a simple dischargemeasurement structure is possible. The discharge above the weir equals:

$$Q = C_d\sqrt{g\left(\frac{2}{3}h_1\right)^3}\ .\ B \quad (1)$$

Where Q is the discharge, B is the channel breadth, g is the gravity acceleration, h_1 is the upstream total head above the crest, and C_d is the dimensionless discharge coefficient. C_d is unity for an ideal fluid flow above the broad-crest.

Study of broad-crested weirs has attracted the attention of many investigators [1, 7, 27] performed experiments on broad crested weirs. Woodburn [40] showed that the discharge coefficient increases up to 8% if the upstream corner of the weirs is curved [1].

Chow [6] developed a relationship for discharge coefficient, using momentum theorem. Ippen [19], using the Bernoulli and boundary layer equations, developed a relationship for discharge coefficient as a function of boundary layer thickness. Lewith [25] introduced the discharge coefficient as a function of the head of water over the weir, the length and width of the weir, and the flow viscosity [38].

Manuscript received on August, 2013.

Hossein Afshar, Mechanical EngineeringDepartment, Islamic Azad University, East Tehran Branch, Tehran, Iran.

Seyed Hooman Hoseini, Department of Water Engineering, Islamic Azad University, Central Tehran Branch, Iran.

Henderson [15] developed an equation to determine the discharge coefficient for round corner weirs with critical flow condition, assuming Y_b=0.715Y_c (Y_c is the critical depth and Y_b is the depth of water at the edge of the weir). In these studies it is also mentioned that separation of flow can be eliminated, if a broad crested weir with upstream round corner is used. Also the critical depth will move downstream, resulting in a subcritical flow dominating all over the weir.

Successful physical studies included Bazin [3], Woodburn [40], Tison [39] and Serre [36]. Hall [13] and Isaacs [20] studied the effects of developing boundary layer on the overflow. Ramamurthy et al. [31] investigated systematically the discharge characteristics of round-edged and square-edged weirs and Sargison and Percy [34] showed the influence of the weir inflow design on the bottom pressure distributions and discharge coefficient. Gonzalez and Chanson [11], Azimi and Rajaratnam [2], Salmasi et.al [33] conducted experiments on broad-crested weirs. Felder and Chanson [9] investigated the free-surface profiles, velocity and pressure distributions on a broad-crested weir.

Several numerical studies were carried out to find the flow pattern around weirs and porous media. Dias et al. [8] simulated the flow over rectangular weirs using a 2D model to find the corresponding discharge coefficient. Sarker and Rhodes [35] measured the free surface profile over a laboratory scale, rectangular broad-crested weir and compared their results with numerical calculations using CFD. Results showed that for a given flow rate, the prediction of the upstream water depth was excellent and the rapidly varied flow profile over the crest was reproduced quite well. In the supercritical flow downstream, a stationary wave profile was observed and reproduced in form by the calculations. Jia et al. [21] presented a numerical simulation to study the helical secondary flow and the near-field flow distribution around a submerged weir. Xia and Jin [41] improved depth-averaged model using the multilayer model to obtain the velocity and pressure distributions over broad-crest weirs. Karim et al. [22] simulated wave transformation in porous structures to calculate the hydraulic performance of a vertical porous structure. The model was developed using the VOF and two-phase flow (water and air). Yazdi et al. [42] simulated a spur dike with free-surface flow using fully three-dimensional and Reynolds averaged Navier–Stokes (RANS) equations. The volume of fluid (VOF) and standard k-εmodels were employed to simulate free surface and turbulence, respectively. Chan et al. [5] used numerical modeling to study the turbulent flow in a flume with different porous rib on one wall. The computed resultshowed that the flow characteristics are dependent on rib geometry and porous property. Shahamiri and Wierzba [37] used a one-dimensional modeling approach to simulate reactive process within porous medium. Hieu and Vinh [16] utilized numerical modeling to simulate wave overtopping from a seawall supported by porous structures. The verification results show that the VOF based two-phase flow model can be considered as a sufficient method to predict free surface and velocity distribution over porous structures.

II. EXPERIMENT LAYOUT

New experiments were conducted at the laboratory of Islamic Azad University of East Tehran branch, Iran, in a research flume that was made of glass with a cross section 0.30 m wide, 0.50 m deep and 4.8 m long (Fig. 1). Water was supplied from a large 1.5 m deep feeding basin leading to a sidewall convergent enabling a very smooth and wave less inflow. The weir is consisted of a 0.038 m height, 0.30 m width, with an upstream rounded corner (0.0134 m radius) and 0.187 m long. A pump controlled with an adjustable frequency AC motor drive delivered the flow rate, enabling an accurate discharge adjustment in a closed-circuit system. Clear-water flow depths were measured on the channel centerline with a point gauge and using photographs through the sidewalls. The accuracy of point gauge and photographic data yielded the same results within 1 mm.

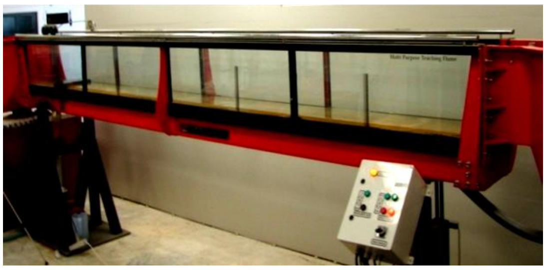

Fig. 1- View of laboratory flume

III. NUMERICAL MODELING

A. Flow-3D

The Flow-3D model developed by Flow Science Incorporated of Los Alamos, New Mexico, USA, is a Computational Fluid Dynamics (CFD) tool capable of simulating the dynamic and steady state behavior of liquids and gases in one, two or three dimensions. It does so through solution of the complete Navier Stokes equations of fluid dynamics. It is applicable to almost any type of the flow process and capable of simulating free surface flow, and the program utilizes specialized algorithms to track the location of the water surface over large and small spatial and temporal variations. These capabilities make the model well suited for simulating the varied and complex flow conditions, which typically occur in a variety of hydraulic design and analysis problems [17, 18, 24].

The continuity and Reynolds-averagedNavier-Stokes equations are for this casesolved in a vertical plane (two dimensions), inorder to compute the water motion for turbulentflow.

$$\frac{\delta u_i}{\delta x_i} = 0 \quad (2)$$

$$\frac{\delta u_i}{\delta t} + u_j \frac{\delta u_i}{\delta x_j} = \frac{1}{\rho} \frac{\delta}{\delta x_j} \left(-P\delta_{ij} + \rho v_T \left(\frac{\delta u_i}{\delta x_j} + \frac{\delta u_j}{\delta x_i}\right)\right) \quad (3)$$

Where U is the Reynolds-averaged velocity overtime t, x is the spatial geometrical scale, ρ is thewater density, P is the Reynolds-averagedpressure, δ is the Kronecker delta and ν_T is theturbulent eddy-viscosity.The turbulence is predictedby the RNG k-ε model (turbulent kineticenergy k and its dissipation ε) using the constantempirical values by Launder and Spalding [23].The pressure is calculated according to the SIMPLE method (Semi-Implicit Method for Pressure-Linked Equations, Patankar [30].

$$\frac{U}{u^*} = \frac{1}{k} \left(\frac{30y}{k_s}\right) \quad (4)$$

Where U is the velocity, u* is the shear velocity, kis the Karman constant (typically the value 0.4 isused), y is the water depth and k_s is the boundaryroughness.

B. Computation of the free water surface

The free water surface represents a particularchallenge in 3D numericalmodels. The selected computer programs usedifferent methods. Flow-3Duses the Volume ofFluid Method (VOF).This is a two-phase approach where both thewater and the air are modeled in each computational cell. Themethod is based on the concept that each cell hasa fraction of water (F), which is 1 when theelement is totally filled with water and 0 when theelement is filled with air. If the value is between 1and 0, the element contains the free water surface.Therefore an additional transport equation isadded.

$$\frac{\delta F}{\delta t} + U\frac{\delta F}{\delta x} + V\frac{\delta F}{\delta y} = 0 \quad (5)$$

Where u and v are fluid velocity components inthe x and y coordinates respectively.The local height of the water surface, the surfaceslope and the curvature are calculated for the localcolumn and its neighbor. The VOFmethod requires a fixed grid. The number of gridpoints in the vertical direction depends on the waterdepth and the initial settings. The effective vertical water surface movement, Δz,is thereby given by:

$$\Delta z = \frac{1}{2}\left(\frac{\Delta v}{A_2} + \frac{\sum_{n=1}^{2} u_n A_n}{A_2}\right)\Delta t \quad (6)$$

Where U_n is the velocity in the surface cell, withcomponent in n direction. A_n is the areacomponent on the top of the surface cell, also indirection n. The horizontal direction is 1, while 2is the vertical direction. Δt is the time step and ΔVis the volume flux deficit in the cell surface. Inthis study, a width-averaged approach isused, so the components in the direction normalto the main flow direction are zero. The formulacan also be used for three dimensions.

C. Grid types and generation

Fllow-3D uses a structured and orthogonal gridwith rectangular (2D) and hexahedral cells (3D).The non-adaptive grid is fixed and does not moveduring the calculation. The border between thegeometry and the water is defined by theFractional Area Volume Obstacle Representation(FAVOR) method. Figure 2 shows a longitudinalprofile of the grid used in Flow-3D. When the watersurface moves, the grid will also move vertically.Hence, only the water phase will be calculated,not the air phase. The grid is regenerated duringthe computation with a varying number of gridcells over the depth. The number of cells in thevertical direction is a function of the water depthand an empirical value (p):

$$\square = \square_{max} \times \left(\frac{depth}{depth_{max}}\right)^P \quad (7)$$

Where ∏ is the number of grid cells in the verticaldirection, $∏_{max}$ is the maximum number of gridcells in the vertical direction, p is a parameter fornumber of grid cells (where p = 0.5 gives the bestresult for this case.

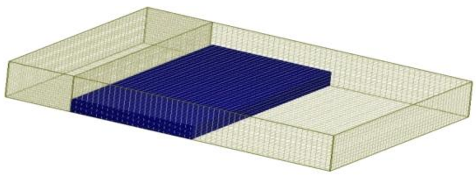

Fig. 2- View of mesh

ISSN: 2249 – 8958, Volume-2, Issue-6, August 2013

The weir set-up in Flow-3D was performed by inserting an STL file. In STL files solid objectsurfaces are approximated by triangles. A grid sensitivity analysis was made with respect to the computational time. This was chosen as criteria, because it is not straight forward to compare different types of grids. The cell size was 1 cm in the x-direction.

D. Boundary conditions

To have an accurate result, an appropriate condition should be selected for boundary based on the nature of the flow. Inthe present study,the computational domain is divided into two sequential zones with overlapping boundaries, which enable the calculations to efficiently make use of meshes as well as to avoid the time step limitations due to high infiltration at the module. Inlet condition of the weir is set as specified pressure. Also, outflow Boundary condition is used for outlet. Boundary condition for both inlet and outlet has set as continuative because to allow water infiltrate. For the top, specified pressure condition selected as to visualize experiment model. The Figure 3 shows conditions applied for each boundary.

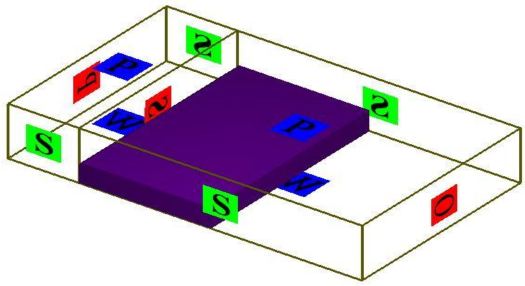

Fig. 3- Domain and boundary conditions

IV. RESULTS AND DISCUSSION

Fig. 4 shows the 3-D simulated water surface profile over a rectangular broad-crested weir. Fig. 5 Shows the pressure contours over a broad-crested weir for Q=2 Lit/s. Fluid depth and fraction of fluid contours are shown in Figs. 6 and 7 respectively. Three different turbulence models were selected to evaluate accuracy of model for simulation of water level. Four different discharges were used to model broad-crested weir. All turbulence methods were utilized to model water surface profile over a weir, a symmetric plan was chosen to compare observation and simulation results.

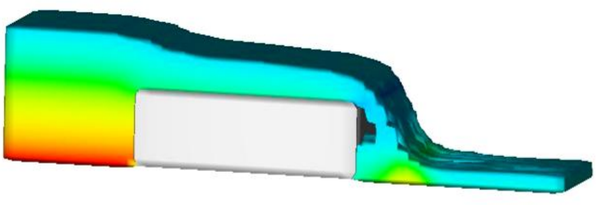

Fig. 4- Simulated water surface profile over a broad-crested weir

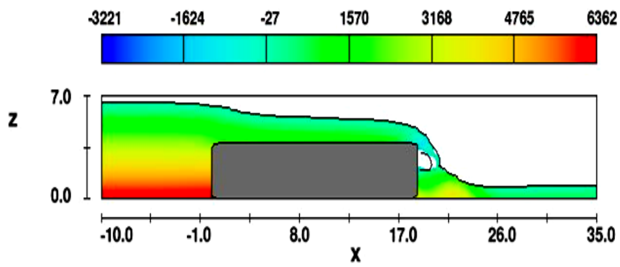

Fig. 5- Pressure contours over a broad-crested weir (Q=2Lit/s)

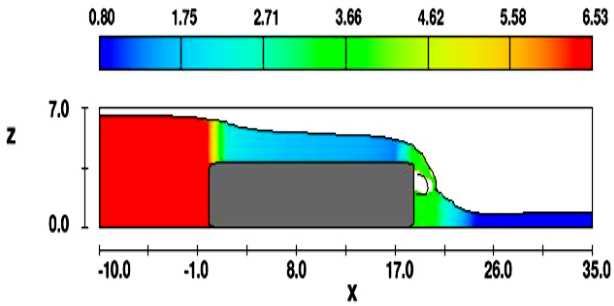

Fig. 6- Fluid depth contours over a broad-crested weir (Q=2Lit/s)

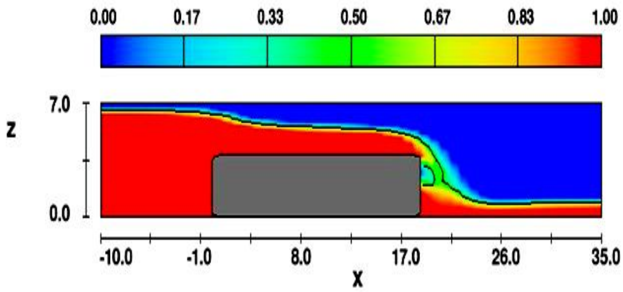

Fig. 7- Fraction of Fluid contours over a broad-crested weir (Q=2Lit/s)

Figs. 8 to 11 indicate a comparison of water surface profile between experimental data and predicted value obtained from the RNG k–ε, k–ε and the large eddy simulation model (LES). Upstream of the weir, the measured water depths were under-predicted by the turbulence models used in this study. Above the weir crest there were an over-prediction, but downstream of the weir in the supercritical flow region the average water depths compared well. The comparison of velocity value according to different discharges is given in Figs. 12 to 15. As is shown in these figures, RNG model has the best similarity with experimental data.

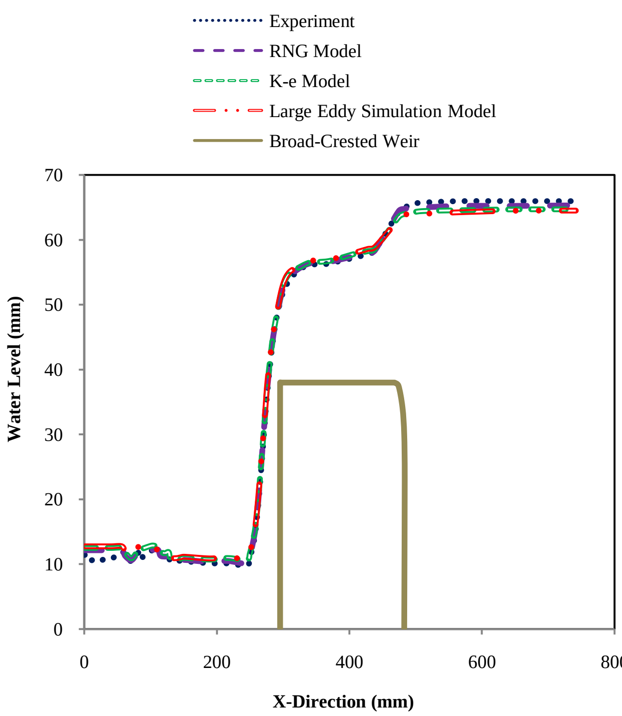

Fig. 8- Compassion between experimental dataand simulated water surface for Q=2 Lit/s

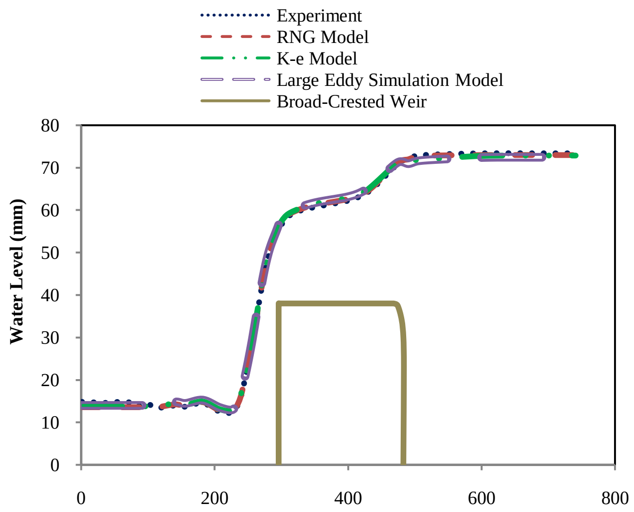

Fig.9- Compassion between experimental dataand simulated water surface for Q=3 Lit/s

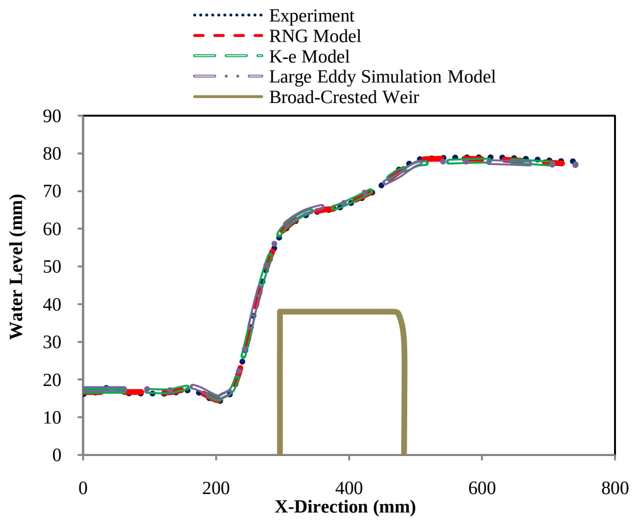

Fig.10- Compassion between experimental dataand simulated water surface for Q=4 Lit/s

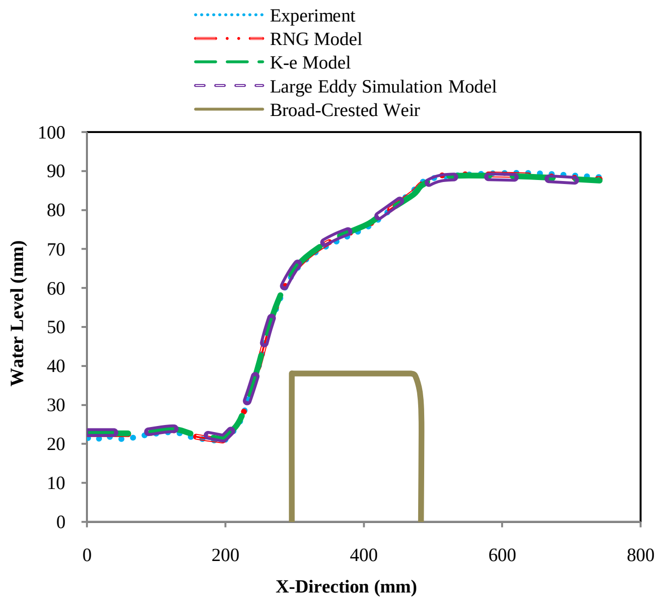

Fig.11- Compassion between experimental dataand simulated water surface for Q=6 Lit/s

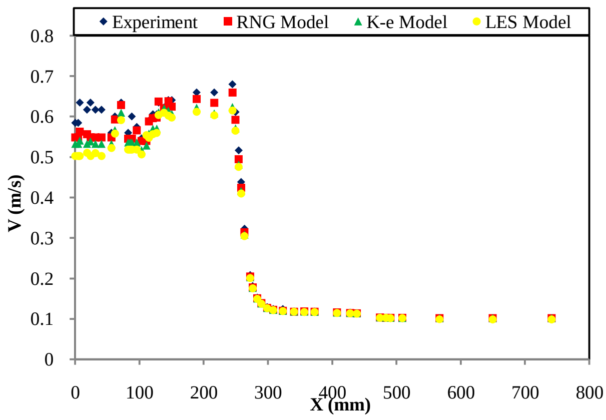

Fig. 12- Comparison between velocity streamlines values for Q=2 Lit/s

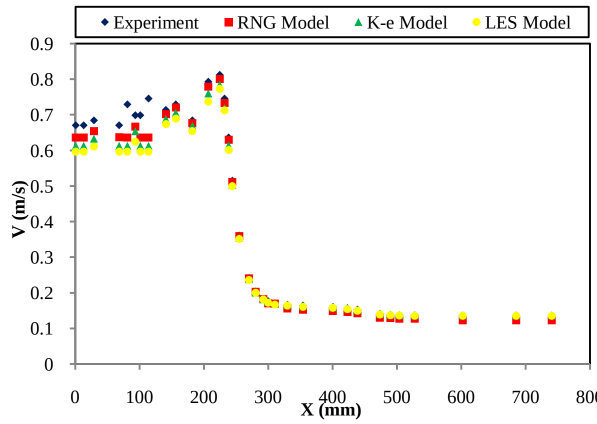

Fig. 13- Comparison between velocity streamlines values for Q=3 Lit/s

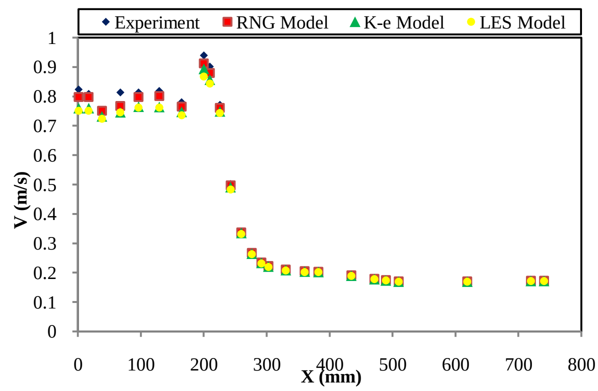

Fig. 14- Comparison between velocity streamlines values for Q=4 Lit/s

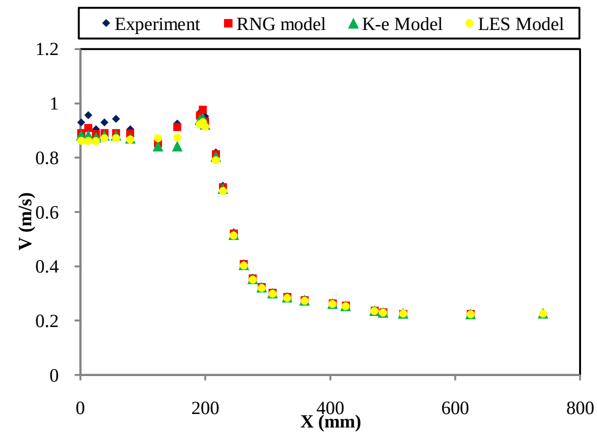

Fig. 15- Comparison between velocity streamlines values for Q=6 Lit/s

The mean relative error percentage (RE), root mean square errors (RMSE) and mean absolute errors (MAE) are used to evaluate the model accuracies. Different types of information about the predictive capabilities of the model are measured through RMSE and MAE. The RMSE sizes the goodness of the fit related to high discharge coefficient values whereas the MAE measures a more balanced perspective of the goodness of the fit at moderate discharge coefficients. The RE, RMSE and MAE are defined as:

$$RE = \frac{100}{N}\sum_{i=1}^{N}\left|\frac{V_{Experiment}-V_{CFD}}{V_{Experiment}}\right| \quad (8)$$

$$RMSE = \sqrt{\frac{1}{N}\sum_{i=1}^{N}(V_{Experiment} - V_{CFD})^2} \quad (9)$$

$$MAE = \frac{1}{N}\sum_{i=1}^{N}\left|V_{Experiment} - V_{CFD}\right| \quad (10)$$

N is the total number of runs which is 135 in this study and i is the ith run. Table 1 shows the value of RE,MAE and RMSE for present study. Although all turbulence models had an acceptable prediction for water surface profile, performance of the RNG k–ε was better than other models.

Model	RE	RMSE	MAE
RNG	2.809	0.0247	0.0143
K-ε	3.647	0.0362	0.0232
LES	4.797	0.0457	0.0299

Table 1- Value error of present study

V. CONCLUSION

Basic experiments were conducted on a rectangular broad-crested weir to investigate the free surface profile of water. 3-D numerical scheme provided by differentturbulence models was used to predict the water surface and streamlines. The computed values were thencompared with the experimental values. The computational results showed a good agreement with experimental data obtained in the laboratory. Also, results indicate thatRNGk–ε model has the lowest error. The presented results in this study can encourage the researchersfurther in making new different designs of broad-crested weirsusing CFD.

ACKNOWLEDGMENT

The authors acknowledge the laboratory of Islamic Azad university of East Tehran branch in Iran for the technical assistance.

REFERENCES

[1] Ackers, P, "Weirs and Flumes for Flow Measurement". Jon Wiley and Sons, U.K, 1978.
[2] Azimi, A.H andRajaratnam, N,"Dischargecharacteristics of weirs of finite crest length". Journal of Hydraulic Engineering, 2009, 135(12):120-125.
[3] Bazin, H.. "ExpériencesNouvellessurl'Ecoulement par Déversoir." ('Recent Experiments on the Flow of Water over Weirs.')Mémoireset Documents, Annales des Ponts et Chaussées, Paris,1896.
[4] Bos, M.G, "Discharge Measurement Structures." Publication No. 161, Delft Hydraulic Laboratory, Delft, The Netherlands,1976 (also Publication No. 20, ILRI, Wageningen, The Netherlands).
[5] Chan, H.C., Zhang, Y., Leu, J.M., Chen, Y.S,"Numerical calculation of turbulent channel flow with porous ribs", J. Mech, 2010, 26:15–28.
[6] Chow V. T,"Open Channel Hydraulics". McGraw-Hill, Inc, 1959.
[7] Clemmens, A.J., Bos, M.G. and Replogle, J.A,"RBC broad-crested weirs for circular sewers and pipes". Journal of Hydrology,1984, 68: 349-368.
[8] Dias, F., Keller, J.B., Vanden-Broeck,J.M,"Flows over rectangular weirs", Phys. Fluids, 1988, 31:2071–2076.
[9] FELDER, S, and CHANSON, H, "Free-surface Profiles, Velocity and Pressure Distributions on a Broad-Crested Weir: a Physical study." Journal of Irrigation and Drainage Engineering, ASCE, 2012, Vol. 138, No.12, pp. 1068–1074.
[10] French, R.H,"Open-channel hydraulics". McGraw- Hill, New York, 1987.
[11] Gonzalez, C.A. and Chanson, H, "Experimental Measurements of Velocity and Pressure Distribution on a Large Broad-Crested Weir." Flow Measurement and Instrumentation, 2007, Vol. 18, No. 3-4, pp. 107-113.
[12] GovindaRao, N.S.,Muralidhard," Discharge characteristics of weirs of finite-crest width". La Houille Blanche, 1963, 5: 537–545.
[13] Hall, G.W, "Analytical Determination of the Discharge Characteristics of Broad-Crested Weirs using Boundary Layer Theory." Proc. Instn. Civ. Engrs., London, 1962, Vol. 22, June, paper No. 6607, pp. 177-190.
[14] Harrison, A.J.M, "The Streamlined Broad-Crested Weir." Proc. Instn. Civil Engrs., London, 1967, Vol. 38, Dec., pp. 657-678.
[15] Henderson, F.M, "Open Channel Flow." MacMillan Company, New York, USA, 1966.
[16] Hieu,P.D.,Vinh,P.N,"Numerical study of wave overtopping of a seawall supported by porous structures", Appl. Math. Model, 2012, 36:2803–2813.
[17] Hirt, C.W., and Sicilian, J.M,"A porosity technique for the definition of obstacles in rectangular cell meshes". InProceedings of the 4th International Conference on Ship Hydro-dynamics, Washington, D.C, NationalAcademy of Sciences, Washington, D.C, 1985.
[18] Hur D.S., Mizutani N,"Numerical estimation of the wave forces acting on a three dimensional body on submergedbreakwater". Coastal Engineering, 2003, 47, 329-345.
[19] Ippen, A. T,"Engineering hydraulics." New York, John Wiley and Sons, Inc, 1950, p. 526-27.
[20] Isaacs, L.T, "Effects of Laminar Boundary Layer on a Model Broad-Crested Weir".Research Report No.CE28, Dept. of Civil Eng., Univ. of Queensland, Brisbane, Australia, 1981, 20 pages.
[21] Jia, Y., Scott,S.H.,Xu, Y., Huang, S., Wang,S.S.Y,"Three-dimensional numerical simulation and analysis of flows around a submerged weir in a channelbendway", J. Hydraul. Eng, 2005, 13: 682–693.
[22] Karim, M.F.,Tanimoto,K.,Hieu,P.D,"Modelling and simulation of wave transformation in porous structures using VOF based two-phase flow model". Appl. Math. Model. 2009, 33:343–360.
[23] Launder, B.E, Spalding, D.B,"Lectures in Mathematical Models of Turbulence". Academic Press, London, 1972.
[24] Lara J.L., Losada I.J., Guanche R., 2008, Wave interaction with low-mound breakwaters using a RANS model. OceanEngineering 35, 1388 1400.
[25] Lewittو E.H,"Hydraulics and Fluid Mechanics". 10th Edition, London, 1978.
[26] Mohammadpour, R., Ghani, A., Azamathulla, H.M,"Numerical modeling of 3-D flow on porous broad crested weirs". Journal of Applied Mathematical Modelling, 2013, In Press.
[27] Montes, J.S, "The Streamlined Broad-Crested Weir. Discussion." Proc. Instn. Civil Engrs., London, 1969, Vol. 42, Mar., pp. 576-578.
[28] Montes, J.S, "Hydraulics of Open Channel Flow." ASCE Press, New-York, USA, 1998, 697 pages
[29] Montes, J.S., and Chanson, H, "Characteristics of Undular Hydraulic Jumps. Results and Calculations." Journal of Hydraulic Engineering, ASCE, 1998, Vol. 124, No. 2, pp. 192-205.
[30] Patankar, S.V, "Numerical Heat Transfer and Fluid Flow". McGraw-hill Book Company, New York,1980.
[31] Ramamurthy, A.S., Tim, U.S., and Rao, M.V.J, "Characteristics of Square-Edged and Round-Nosed Broad-Crested Weirs."Jl of Irrig and Drainage.Engrg., ASCE, 1988, Vol. 114, No. 1, pp. 61-73.
[32] RangaRaju, K.G "Flow through open channel". McGraw-Hill, New York, 1993.
[33] Salmasi, F., Pooreskandar, S., HosseinzadeDalir, A., Farsadizade, D "Discharge Relations for Rectangular Broad-Crested Weirs." Journal of Agricultural Sciences, 2011, 17: 324-336.
[34] Sargison, J.E., and Percy, A, "Hydraulics of Broad-Crested Weirs with Varying Side Slopes." Jl of Irrigation and Drainage Engrg., ASCE, 2009,Vol. 135, No. 1, pp. 115-118.
[35] Sarker, M.A and Rhodes, D.G,"Calculation of freesurface profile over a rectangular broad-crested weir". Flow Measurement and Instrumentation, 2004, 15(4) 215-219.
[36] Serre, F, "Contribution à l'Etude des Ecoulements Permanents et Variables dans les Canaux." ('Contribution to the Study of Permanent

and Non-Permanent Flows in Channels.')Jl La Houille Blanche, Dec., pp. 1953, 830-872 (in French).
[37] Shahamiri,S.A.,Wierzba, I,"Modeling the reactive processes within a catalytic porous medium."Appl. Math. Model, 2011, 35 (4):1915–1925.
[38] Streeter, V.L. and Wylie, E.B,"Fluid Mechanics". McGraw Hill, New York, 1981.
[39] Tison, L.J, "Le DéversoirEpais.('Broad-Crested Weir.')"Jl La Houille Blanche, 1950, pp. 426-439 (in French).
[40] Woodburn, J.G.."Tests of Broad-Crested Weirs."Transactions, ASCE, 1932, Vol. 96, pp. 387- 416. Discussion: Vol. 96, pp. 417-453.
[41] Xia, C., Jin, Y.C,"Multilayer depth-averaged flow model with implicit interfaces."J. Hydraul. Eng. 2007, 13:1145–1154.
[42] Yazdi, J.,Sarkardeh,H.,Azamathulla, H.M.,Ghani, A.A, "3D simulation of flow around a single spur dike with freesurface flow." Int. J. River Basin Manage,2010,8:55–62.