Abstract- A broad-crested weir is a flat-crested structure with large 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 $k$-$\varepsilon$, standard $k$-$\varepsilon$ 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 the 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 structures where the streamlines run parallel to each other over the weir crown, and the crest of the weir is horizontal [4]. In most layouts of broad-crested weirs also the hydrostatic pressure is fully accomplished 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 discharge measurement structure is possible. The discharge above the weir equals:

$$ Q = C_d \left( g \frac{h_0^2}{2} \right)^{1/2} \cdot B \ (1) $$

Where $Q$ is the discharge, $B$ is the channel breadth, $g$ is the gravity acceleration, $h_0$ 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].

Henderson [15] developed an equation to determine the discharge coefficient for round corner weirs with critical flow condition, assuming $Y_c=0.715Y_0$ ($Y_c$ is the critical depth and $Y_0$ 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-crested 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-$\varepsilon$-models 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 results showed 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

Manuscript received on August, 2013.

Hossein Afshar, Mechanical Engineering Department, Islamic Azad University, East Tehran Branch, Tehran, Iran.
Seyed Hooman Hoseini, Department of Water Engineering, Islamic Azad University, Central Tehran Branch, Iran.
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 waveless 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.

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-averaged Navier-Stokes equations are for this casesolved in a vertical plane (two dimensions), inorder to compute the water motion for turbulentflow.

\[ \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = 0 \]

(2)

\[ \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = 0 \]

(3)

Where U is the Reynolds-averaged velocity over time t, x is the spatial geometrical scale, \( \rho \) is the water density, P is the Reynolds-averaged pressure, \( \delta \) is the Kronecker delta and \( \nu_r \) is the turbulent eddy viscosity. The turbulence is predicted by the RNG k-\( \varepsilon \) model (turbulent kineticenergy k and its dissipation \( \varepsilon \)) 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]).

\[ \frac{U}{u^*} = \frac{1}{k} \left( \frac{30\nu}{k} \right) \]

(4)

Where U is the velocity, \( u^* \) is the shear velocity, k is the Karman constant (typically the value 0.4 is used), \( \nu \) is the water depth and \( k_\text{s} \) is the boundaryroughness.

B. Computation of the free water surface

The free water surface represents a particular challenge in 3D numerical models. The selected computer programs uses different methods. Flow-3D uses the Volume of Fluid Method (VOF). This is a two-phase approach where both the water and the air are modeled in each computational cell. The method is based on the concept that each cell has fraction of water (F), which is 1 when the element is totally filled with water and 0 when the element is filled with air. If the value is between 1 and 0, the element contains the free water surface. Therefore an additional transport equation is added.

\[ \frac{\partial F}{\partial t} + U \frac{\partial F}{\partial x} + V \frac{\partial F}{\partial y} = 0 \]

(5)

Where u and v are fluid velocity components in the x and y coordinates respectively. The local height of the water surface, the surfaceslope and the curvature are calculated for the local column and its neighbor. The VOF method requires a fixed grid. The number of grid points in the curvature is on the waterdepth and the initial settings. The effective vertical water surface movement, \( \Delta z \), is thereby given by:

\[ \Delta z = \frac{1}{2} \left( \frac{\partial F}{\partial x} + \frac{\partial F}{\partial y} \right) \Delta t \]

(6)

Where \( U \), is the velocity in the surface cell, with component in n direction. \( A_n \) is the areacomponent on the top of the surface cell, also in direction n. The horizontal direction is 1, while 2 is the vertical direction. At is the time step and \( \Delta z \) is the volume flux deficit in the cell surface. In this study, a width-averaged approach is used, so the components in the direction normal to the main flow direction are zero. The formula can also be used for three dimensions.

C. Grid types and generation

Flow-3D uses a structured and orthogonal grid with rectangular (2D) and hexahedral cells (3D). The non-adaptive grid is fixed and does not move during the calculation. The border between the geometry and the water is defined by the Fractional Area Volume Obstacle Representation (FAVOR) method. Figure 2 shows a longitudinal profile 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 during the computation with a varying number of grid cells over the depth. The number of cells in the vertical direction is a function of the water depth and an empirical value (p):

\[ \Omega = \Omega_{\text{max}} \times \left( \frac{\text{depth}}{\text{depth}_{\text{max}}} \right)^p \]

(7)

Where \( \Omega \) is the number of grid cells in the vertical direction, \( \Omega_{\text{max}} \) is the maximum number of gridcells in the vertical direction, p is a parameter for the number of grid cells (where p = 0.5 gives the best result for this case).
The weir set-up in Flow-3D was performed by inserting an STL file. In STL files solid objects surfaces 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 straightforward 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. In the 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.

**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.

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

Fig. 9- Comparison between experimental data and simulated water surface for Q=3 Lit/s

Fig. 10- Comparison between experimental data and simulated water surface for Q=4 Lit/s

Fig. 11- Comparison between experimental data and 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:

\[
RE = \frac{100}{N} \sum_{i=1}^{N} \frac{|V_{\text{Experiment}}-V_{\text{CFD}}|}{V_{\text{Experiment}}}
\]

\[
RMSE = \frac{1}{N} \sum_{i=1}^{N} (V_{\text{Experiment}} - V_{\text{CFD}})^2
\]

\[
MAE = \frac{1}{N} \sum_{i=1}^{N} |V_{\text{Experiment}} - V_{\text{CFD}}|
\]

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 \(k-\varepsilon\) was better than other models.

<table>
<thead>
<tr>
<th>Model</th>
<th>RE</th>
<th>RMSE</th>
<th>MAE</th>
</tr>
</thead>
<tbody>
<tr>
<td>RNG</td>
<td>2.809</td>
<td>0.0247</td>
<td>0.0143</td>
</tr>
<tr>
<td>(k-\varepsilon)</td>
<td>3.647</td>
<td>0.0362</td>
<td>0.0232</td>
</tr>
<tr>
<td>LES</td>
<td>4.797</td>
<td>0.0457</td>
<td>0.0299</td>
</tr>
</tbody>
</table>

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 different turbulence models was used to predict the water surface and streamlines. The computed values were then compared with the experimental values. The computational results showed a good agreement with experimental data obtained in the laboratory. Also, results indicate that RNG \(k-\varepsilon\) model has the lowest error. The presented results in this study can encourage the researchers further in making new different designs of broad-crested weirs using CFD.

ACKNOWLEDGMENT

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

REFERENCES


