Water Surface Profile along Different Side Weirs in Subcritical Flow Regime

*Bruska S. Mamand¹, Adil M. Raheem²

¹Assistant Lect., College of engineering, Salahaddin University-Erbil
²Assistant Prof., College of engineering, Salahaddin University-Erbil

ARTICLE INFO

Article History:
Received: 14/05/2017
Accepted: 12/04/2018
Published: 01/06/2018

Keywords:
Side weir.
Fluent.
k-ε Turbulent Model.
Water Surface Profiles.

*Corresponding Author:
Email:
bruska.sardar@gmail.com

ABSTRACT

Side weir was hydraulic structure used to divert flow from the main channel into the side channel whenever the water surface rise above the side weir crest. In the present study FLUENT code that interfaced on ANSYS (V.14.0) was used to examine the computational domain in subcritical flow condition. Five different types of side weirs installed inside the horizontal prismatic rectangular main channel. Models were normal rectangular with (θ = 90°), oblique rectangular in which its crest is inclined with the flow direction by (θ = 75°, 60° and 45°), triangular labyrinth with apex angles (δ=90° and 120°), semi-hexagonal labyrinth with wall angles (Z=30° and 64.7°) and semi-circular labyrinth side weirs. The volume of fluid method (VOF) provided to truck the free surface profiles. The computational code is validated against the experimental models. Three different k-ε turbulent models were tested namely re-normalized group (RNG k-ε), standard k-ε and realizable k-ε. Free surface flow profile for three dimensional (3D) subcritical flow condition at the centerline of the main channel and near side weir bank were measured and plotted for different side weir models, the longitudinal water surface profile in the main channel was found to be increased from upstream end towards the downstream end of the side weirs. Result for aforementioned turbulent models was compared with the experimental data and it was found that the RNG k-ε turbulent model provide more accurate results compared with other turbulent model examined in the present study.

INTRODUCTION:

The flow condition in the main channel at the upstream and downstream end of a side weir (i.e. Fr₁, Fr₂ respectively), have a major influence on the behavior of the flow over the weir itself. (May et al., 2003) stated that the flow conditions at side weir was very complex issue due to many principal factors which were difficult to be analyse. Flow at the side weir section was strongly considered to be three dimensional flow passing over the weir crest crosses at an inclined angle rather than at right angle and the water level in the main channel does not remains constant but may rise and fall along the length of the side weir. Generally subcritical flow occur in the main channel with the existence of the side weirs. Using the principle of the energy in which developed by (De Marchi, 1934) considering the specific energy to be constant between upstream and downstream ends of the side weirs, the dynamic equation of spatially varied flow (SVF) can be obtained as:
\[
\frac{dy}{dx} = \frac{S_o - S_f - \alpha Q \frac{dQ}{dA} - \frac{\alpha Q^2 T}{gA^2}}{1 - \frac{\alpha Q^2 T}{gA^2}}
\]

where: \(y\) is depth of flow, \(x\) is the distance along the side weir from upstream end, \(S_o\) is the main channel slope, \(S_f\) is friction slope, \(\alpha\) is kinetic energy correction factor, \(Q\) is discharge in channel, \(dQ/dx\) is discharge per unit length of side weir, \(g\) is the acceleration due to gravity, \(A\) is cross-sectional area of flow and \(T\) is the top width of the channel section.

Most of investigations carried out on the flow over side weirs, (Swamee et al., 1994) analyzed the discharge coefficient using fourth order range kutta method. (Uyumaz, 2007) numerically investigated the flow over the side weirs in triangular main channel for both subcritical and supercritical flow regimes. Investigation both experimentally and numerically the flow over the rectangular side weir with zero sill height located in a horizontal rectangular channel was done by (Mangarulkar, 2010). Simulation experimentally water surface profile along 1m length of side weir using the ANSYS CFX code by (Martins, 2011). (Aydin, 2011) used ANSYS FLUENT code for model runs to simulate the free surface flow over triangular labyrinth side weir under subcritical flow condition. (Mahmodinia et al., 2012) were investigated the effects of Froude number on the upstream side of the weir on the free surface flow over the side weir. (Namaee, 2014) applied ANSYS FLUENT code to investigate the flow over broad crested side weir having a longitudinal inclined ramp and water surface profiles of k-\(\varepsilon\) turbulence model was compared with an experimental profiles. (Hoseini et al., 2013) investigated the discharge coefficient of rectangular broad crested side weir located on trapezoidal channel. One can notice that the flow over side weir was complex hence understanding its behavior and characteristics is essential to improve the design of such kind of hydraulic structure. For this purpose the computational FLUENT code is applied to predict the water surface profile along the side weirs for different flow conditions and flow rates in the main channel.

**Navier-Stokes Equation:**

Navier–Stokes equations are the governing equations for the fluid motion which numerically can be solve for the models of flow through different boundary conditions for a known geometry. These equations in three dimensional form for unsteady viscous fluid were presented by (Desai and Patil, 2015) as follows:

\[
\begin{align*}
\frac{\partial u}{\partial t} + \nabla \cdot (uA_x u) + vA_y u + wA_z u &= -\frac{\partial p}{\rho \partial x} + G_x + f_x \\
\frac{\partial v}{\partial t} + \nabla \cdot (uA_x v) + vA_y v + wA_z v &= -\frac{\partial p}{\rho \partial y} + G_y + f_y \\
\frac{\partial w}{\partial t} + \nabla \cdot (uA_x w) + vA_y w + wA_z w &= -\frac{\partial p}{\rho \partial z} + G_z + f_z
\end{align*}
\]

It was supplemented by the mass conservation equation:

\[
\frac{\partial u}{\partial x} A_x + \frac{\partial v}{\partial y} A_y + \frac{\partial w}{\partial z} A_z = 0
\]

Where, \(V_F\) is the fractional volume open to flow, \(\rho\) is fluid density, \((u,v,w)\) are velocities in \((x,y,z)\) direction respectively, \(t\) is time, \((A_x,A_y,A_z)\) are fractional areas open to flow in direction \((x,y,z)\), \((G_x,G_y,G_z)\) are body accelerations and \((f_x,f_y,f_z)\) are viscous accelerations. These are the simplified equations for incompressible free surface flow with constant viscosity. Fluid configuration was defined in terms of volume of fluid (VOF).

**MATERIALS AND METHODS**

Experimental models were constructed for different types of side weirs then their geometries numerically were created by using FLUENT workbench. It was consisted of a solid domain of horizontal 3D rectangular main channel. The dimensions of the channel was 0.2m width, 0.3 m deep, and 2.2m downstream length which starts from the end of the side weir crest toward the downstream end of the
channel and the upstream length 1.4 m starts from the inflow boundary (inlet) of the main channel to the upstream end of the side weir. For diverting a part of the flow laterally, side weirs was placed at one side of the main channel as shown in Figure (1). This figure describes the definition sketch of different side weir models tested in the current study.

The geometric domain for triangular and semi-circular side weirs was shown in Figure (2). The computational area for all models was divided into a collection of cells which addressed as grids through set of points distributed on the geometry region. In the present study, the automatic mesh method used to calculate mesh grids throughout a solid domain as shown Figure (3). Mesh size should be fine enough to ensure the flow features spread out through all fields. Therefore the suitability of the mesh cells for all side weir models was checked from mesh metric option through a main key measurement named as skewness, the skewness should not exceed 0.85 for hexahedra, quadrilateral and triangular cells, while 0.90 for tetrahedral cells.

Boundary conditions are usually set to identify the physics of the flow passed over the side weirs. Their locations was divided into; Inlet: which water flows only into the channel with a known inlet water depth and the magnitude of inlet velocity that measured from the experiments. Side weir outlet: where fluid from the domain is exited out and Cartesian coordinate system (x, y, and z) for direction vectors was used. Ambient boundary: Top surface of the channel and walls were set as impermeable to flow, smooth and slipping did not occur. k-ε was selected as a specification method for inlet and outlet boundaries. Also second-order upwind scheme were set for mass, momentum and turbulence models. Mass imbalance for assessing the convergence criteria was checked via solver flux report with the actual experimental data.

RESULTS AND DISCUSSION

To produce more accurate computational results in term of water surface profiles along the channel centreline, the mesh dependence
was checked for several element sizes as shown in table (1). During all runs for meshes the RNG k-ε turbulence model was used.

| Table (1) Four mesh sizes examined in the study. |
|------------------------------------------------|
| Grid Name | No. of Elements | No. of Nodes |
|-----------|----------------|-------------|
| Mesh (A4) | 62400          | 70854       |
| Mesh (A3) | 218400         | 237615      |
| Mesh (A2) | 427500         | 457314      |
| Mesh (A1) | 519480         | 553336      |

Mesh quality plays a significant role in the accuracy and stability of the numerical tests. The longitudinal water surface profiles for each mesh grid was measured at the centreline of the main channel along the side weir section. Figure (4) illustrated the quite difference between mesh four mesh grids in comparing with experimental water surface profile. In other words, Mesh (A1) was in agreement with experimental data therefore this mesh size was allowed in the present work.

Three different k-ε turbulence models as (Standard, Realizable and RNG) were applied in this study to identify the numerical model that increase the accuracy of the numerical results. Figure (5) shows the error percentage for the flow passing over the side weirs ($Q_w$) using experimental flowrate ($Q_w^{exp.}$). Average error obtained for three models (standard, realizable and RNG) models were (8.75%, 6.99% and 6.85%) respectively. This result demonstrate that the standard k-ε turbulence model provide higher percentage of error compared with the other two models. This may be due to the fact that the flow condition near the wall regions cannot be solved by this type of turbulence model. It can also be observed that the percentage of error produced by the other two models was identical. RNG model was more responsive to the effects of streamline curvature and treating sensitively near wall regions, and the profiles of RNG k-ε model at both positions (at centreline of the channel and near the side weir) was in agreement with the experimental profiles.

The aforementioned errors may be due to surface waves especially near the weir bank which was experimentally difficult to be measured exactly.

The longitudinal water surface profile for different side weir models as mentioned above were drawn experimentally and numerically. Two positions for profiles was selected, one at centerline of the main channel and the other near the side weir bank by a distance 0.05m. For this purpose a plane of the water-volume fraction was set as shown in Figure (6).
**Fig. 6.** Longitudinal water surface profiles along centreline and near side weir bank.

Figure (7-A) indicates that for subcritical flow condition inside the main channel, the longitudinal water surface levels at both positions increase from the side weir entry towards the weir exit, due to the increase of the water depth in the downstream side of the weir ($y_2 > y_1$). Water levels near the weir bank drop slightly at side weir entry with a small distance ($x_1$) and this phenomenon has been attributed to the side weir entrance effect at the upstream end near weir crest. Afterwards a gradual increase of water levels can be observed towards the middle of the crest. Finally, the water level rapidly rises by a distance ($x_2$) reaching its maximum level near the weir exit and parallel flow to the channel bed was observed at the downstream of the side weir.

On the other side, the water profiles at the centreline of the channel was found to be smoother than the profiles along the side weir due to the absent of the side weir entrance effect at the channel centreline.

**Fig. 7-A.** Water surface profiles along the centerline and banks of rectangular side weir ($\theta=90^\circ$).

Similar profiles was drawn for other types of the side weirs as shown in Figure (7-B). The tendency of surface profiles was slightly changed for different side weir geometric shape, dimensions, and channel inflow discharge. For furthermore on the free surface profiles the (SURFER V.08) software was used to plot their 3 dimensional view of elevations inside the main channel. Different types of side weirs were installed and their crests was taken as a reference datum for measuring the depth of water. For this purpose the same inlet discharge was tested for all type of the side weirs and a slight differences in $Fr_1$ values noted. Figure (8) revealed that the dropped flow toward the side weir was clearly noted.
The presence of lateral flow created by side weir overflow in the main channel causes the flow to separate from inner channel wall which forms a separation flow region as shown in Figure (9). The location of separation region, its length ($S_L$) and width ($S_w$) depend on the upstream Froude number, as well as on the shape and overflow length of the side weir. Results shows that length of separation flow
region decreases with increasing in Fr₁ value, the same result was obtained by Mangarulkar (2010), Rao (1968) and Emiroglu et al. (2010). While maximum width of this zone(Sw) was measured approximately to be less than 0.25 times the width of the main channel in comparing with Mangarulkar (2010) who showed Sw = 0.3 B.

![Fig.9](image9) Top view of separation flow streamlines for Normal rectangular side weir.

Another main feature that occurred in labyrinth and an oblique rectangular side weirs was the formation of depression zone. This phenomenon was appeared near the upstream crest of the inner face of side weirs, it was due to contacting bed streamlines with side weir wall which causing the flow to re-circulate then either reversed to main channel (especially bed streamlines) or passed over the side weir into the side channel. As shown in Figure (10), the velocity streamlines at the inner side of upstream semi-circular side weir face developed an approximate elliptical rotation of streamlines due to weir circular bend. While at the downstream half of weir face larger amount of water was diverted to the side channel over its crest as their upstream portion is blocked by the return flow zone.

![Fig.10](image10) Depression zone of circular labyrinth side weir.

For different side weir models, Figure (11) demonstrate the size and shape of the return flow area for different weir geometry, upstream Froude number (Fr₁) and dimensionless (L/B) term.

![Fig.11](image11) Return flow area for other side weir models.

**CONCLUSION:**

1. Experimental and numerical water surface profile inside the main channel was investigated for different types of side weirs. For subcritical flow condition the water depth at upstream end of the side weir was greater than the depth at downstream end (y₂>y₁).

2. Different mesh sizes were used and examined to determine the size that produces more accurate computational results compared with experimental data. Three difference k-ε turbulent models (Standard, realizable and RNG) were applied to identify the accuracy of the numerical models, the percentage of errors between the experimental and the numerical results for the flow passing over the side weirs (Qₘ) for (standard, realizable and RNG model) were found to be (8.75%, 6.99% and 6.85%) respectively. RNG model was more responsive to the effects
of streamline curvature and treating sensitively near walls regions, the profiles of RNG k-ε model at both positions (at centreline of the channel and near the side weir) was in agreement with the experimental profiles.

3. The presence of lateral flow created by side weir overflow in the main channel causes the flow to separate from inner channel wall which forms a separation flow region, location of separation region, its length (S_L) and width (S_w) depend on the upstream Froude number. The length of separation flow region decreases with increasing in Fr_1 value the maximum width of this zone (S_w) was measured approximately to be less than 0.25 times the width of the main channel. Formation of depression zone appeared near the upstream crest of the inner face of the side weirs, which cause the flow to be re-circulate. Size and shape of the return flow area for different weir geometry, upstream Froude No (Fr_1) and dimensionless (L/B) term were also investigated.

REFERENCES:

ANSYS FLUENT 12.0, tutorial guide (2009). Certified to ISO 9001: 2008 and ANSYS FLUENT 14.0, Solver and theory helps.
Aydin, M. (2011). 'CFD simulation of free-surface flow over triangular labyrinth side weir ', Department of Civil Engineering, BitlisEren University.
De Marchi, G. (1934). 'Saggio di teoria de funzionametedeglistramazziLaterali' Energ. Elettr, 11(11), 849–860.
Desai, R. and Patil, L. (2015). 'Performance Comparison of Various Labyrinth Side Weirs ', International Journal of Application or Innovation in Engineering & Management (IJAEM), 4 (6), p.68-73.
Hoseini, S., Jahromi, S. and Vahid, M. (2013). 'Determination of Discharge Coefficient of Rectangular Broad-Crested Side Weir in Trapezoidal Channel by CFD ', J. of Hydraulic Engineering, 2 (4), p. 64-70.

Emiroglu, M., Kaya, N. and Agaccioglu, H. (2010). 'Discharge capacity of labyrinth side weir located on a straight channel', J. of irrigation and drainage engineering, ASCE, 136(1), P. 37-46.
Mahmodinia, S., Javan, M. and Eghbalzadeh, A. (2012). 'The Effects of the Upstream Froude Number on the Free Surface Flow over the Side Weirs ', International Conference on Modern Hydraulic Engineering, Vol.28, p. 644-647.
Margarulikar, K. (2010). 'Experimental and numerical study of the characteristics of side weir flows ', MSc thesis, University of Concordia, Quebec, Canada.
Martins, J. (2011). 'Analysis of hydraulic performance of a side weir by CFD ', MSc thesis, Polytechnic school, University of Sao Paulo, Dept. of civil engineering, Sao Paulo, Portugal.
May, R., Bromwich, B., Gasowski, Y. and Richard, C. (2003). 'Hydraulic design of side weirs ', Thomas telford publishing, heron quay, London E14 4JD.
Namaee, M. (2014). 'Numerical Investigation of a Side Weir with an Inclined Ramp ', J. of World Applied Sciences, 31 (10), p. 1759-1766.
Rao N., Sridharan K., and Baig M. (1968). 'Experimental study of the division of flow in an open channel', India institute of science, p.139-142.
Subramanya, K. and Awasthy, S.C, (1972). 'Flow over side weirs ', Indian institute of technology, p. 328-334.
Swamee, P., Pathak, T. and AIP, M. (1994). 'Side-weir analysis using elementary discharge coefficient ', J. of irrigation and drainage engineering, ASCE, 120 (3), p. 742-755.
Uyumaz, A., 2007. 'Discharge control by a side weir in a triangular main channel ' University of technical Istanbul, p.12451-1258.