The Effects of the Upstream Froude Number on the Free Surface Flow over the Side Weirs

Sharareh Mahmodinia\textsuperscript{a}, Mitra Javan\textsuperscript{b,c}, Afshin Eghbalzadeh\textsuperscript{b,c, c*}

\textsuperscript{a}M.S. Student, Department of civil Engineering, Razi University, Iran
\textsuperscript{b}Assistant Professor, Department of civil Engineering, Razi University, Iran
\textsuperscript{c}Water and Wastewater Research Center, Razi University, Kermanshah, Iran

Abstract

Side weirs are flow diversion devices widely used in irrigation as a head regulator of distributaries and escapes, land drainage, and urban sewage systems. In this paper, free surface flow over side weir in different Froude number is simulated by using FLUENT 6.30 program. Comparisons with the experimental measurements show that the predictions of Longitudinal water surface profiles along the centreline of the channel and Channel discharge rates in side-weir Section are with reasonable accuracy. It was observed that the separation zone area move toward the downstream end of the side-weir by increasing the upstream Froude number.

© 2012 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Society for Resources, Environment and Engineering Open access under CC BY-NC-ND license.

Keywords: Side- weir, Numerical simulation, Separation zone, Froude number, FLUENT;

1. Introduction

Side weirs are also known as a lateral intake structure, which is widely used in irrigation, land drainage, and urban sewerage systems for flow diversion or intake purposes. Many researchers such as Subramanya and Awasthy (1972), Khashab (1975), Agaccıoglu and Yuksel (1998), Cosar and Agaccıoglu (2004) and Agaccıoglu and Onen (2005) also observed a separation zone and the reverse flow at the downstream end of the side-weir for subcritical flow [1-5]. These researchers noted that the location and size of the separation zone and reverse flow area depend on the Froude number on the upstream side of the weir and also on the length of the side-weir. A complete analytical solution of the equations governing the weir

\* Corresponding author: Mitra Javan. Tel.: +98-831-4280093; fax: +98-831-4280094.
\textit{E-mail address: javanmi@gmail.com.}
discharge is not possible as there are many parameters influencing the flow phenomenon and computing methods and high speed computers are strong tools for engineers, now days. Therefore, one does not need to perform time consuming and expensive experimental test procedures to obtain the mean characteristics of flows in hydraulic practices. Numerical methods (Computational Fluid Dynamics, CFD) with their advantages of lower cost and greater flexibility can reasonably predict the mean characteristics of flows such as velocity distributions, pressure distributions, and water surface profiles of complex problems in hydraulic engineering. Karizi and Honar (2007) simulated flow pattern over board crest side weir by using Fluent [6]. Mangarulkar (2010) used ANSYS ICEM CFD 12.0.1. for the numerical simulation of the sharp crest side-weir[7]. In this paper by using FLUENT program, free surface and flow pattern over the sharp crest side weir is simulated and the effects of the upstream Froude number on them are investigated.

2. Governing equations

The flow field is determined by the following incompressible fluid Reynolds-averaged continuity and momentum equations. In equations, the turbulent stresses are calculated with Reynolds Stress Model (RSM). The volume of fluid (VOF) scheme is used for simulating free surface.

3. Numerical Model Description

FLUENT is the CFD solver for choice for complex flow ranging from incompressible (transonic) to highly compressible (supersonic and hypersonic) flows. In paper used QUICK discretization scheme for Momentum, First order upwind discretization scheme for turbulent kinetic energy and turbulent dissipation rate; used Presto d discretization scheme for Pressure and PISO algorithm for Pressure-Velocity Coupling Method. The under-relaxation factors are chosen between 0.2 and 0.5. The small values of the under-relaxation factors are required for the stability of the solution of this interpolation scheme. In the iterative solutions, it must be ensured that iterative convergence is achieved with at least three orders (1e-3) of magnitude decrease in the normalized residuals for each equation solved. For time-dependent problems, iterative convergences at every time step are checked and all residuals are dropped below four orders (1e-4) in about 10,000 iterations. The time steps size are selected as t=0.01s.

4. Boundary Conditions

In 3D simulations performed in the present study, the boundaries depending on the nature of the flow are solid walls, inlet, outlet and free-surface. Wall boundary is used to specify the solid surfaces. Two separate inlets for air and water are specified. At each inlet, uniform distributions are given for all of dependent variables. Open channel and pressure outlet boundary condition is used for two outlets for all of runs.

5. Result and Discussions

In this paper, we used the experimental data of Hager (1982) for validation of numerical simulation [8]. The models of Hager (1982) were set up in a rectangular channel. The length of the side weir and the width and total length of the main channel were 1m, 30cm and 5.70m, respectively. The parameter s used for the numerical simulation of this case are the sill of the side weir (s), upstream flow depth (y₁), downstream flow depth (y₂), inlet discharge \( Q₁ \) and outlet discharge \( Q₂ \) which equal to, 0.15(m), 0.192(m), 0.209(m), 0.0385(m³/s) and 0.02003(m³/s) respectively. The simulations consider \( 140 \times 15 \times 35 \)
and \(100 \times 20 \times 35\) non uniform grid in the x-, z- and y- directions at the main channel and intake, respectively. Existing experimental results (Hager 1982) related to water surface profiles and channel discharge rates were used to validate the numerical simulation predictions. For water surface profiles along the centerline of the channel, Fig. 1 shows that there is a good agreement between numerical predictions and experimental results. Both experimental and predicted channel discharge rates are shown in Fig. 2. The simulation results agree well with the experimental. In order to study the effects of the upstream Froude number, only the velocity inlet of Hager's case has been changed. Fig. 3 shows the streamlines near the bottom of the main channel. It can be seen how the streamlines have been diverted to the side weir. In Fig. 3, a separation a zone at the front of the side weir and the stagnation zone at the downstream end of the side weir is observed. The location and size of the separation zone depend on the Froude number on the upstream side of the weir (Fig. 3). When the upstream Froude number is increased, the separation zone area moves toward the downstream end of the side weir (Fig. 3). Fig. 4 shows a typical set of vertical profiles of the x-velocities in weir section at the main channel. The velocities are normalized by dividing with maximum longitudinal velocity. Most noticeable is the gradual decrease in velocity at the downstream of side-weir. The velocity variation tends toward zero and negative at the downstream of side-weir. The gradual decrease in velocity results from the stagnation zone. The decrease in velocity is less by increasing the upstream Froude number (Fig. 4).

Fig. 1. Longitudinal water surface profiles along the centerline of the channel
Fig. 2. Channel discharge rates in side-weir Section
Fig. 3. Streamlines layout near the bottom of the main channel
Fig. 4. the x-velocity in the weir section at the main channel
Fig. 5 shows the water surface elevations for some subcritical flows. In Fig. 5, the difference of surface profiles along the channel axis and the sidewalls can be observed. By increasing the upstream Froude number, the water depth has decreased toward the downstream (Fig. 5).

![Fig 5](image_url)

Fig.5. the water surface elevations for different Froude numbers

6. Conclusion

In this paper, free surface flow over side weir in different Froude number is simulated by using FLUENT 6.30 program. Comparisons with the experimental measurements show that the predictions of Longitudinal water surface profiles along the centerline of the channel and Channel discharge rates in side-weir Section are with reasonable accuracy. The location and size of the separation zone depend on the Froude number on the upstream side of the weir. When the upstream Froude number is increased, the separation zone area moves toward the downstream end of the side-weir.

References

[1] Subramanya K, Awasthy SC. Spatially varied flow over side weirs. Journal of the Hydraulics Division 1972; 98:1–10.
[2] Khashab AEI, Smith KVH. Experimental investigation of flow over side weirs. Journal of the Hydraulics Division, 1976; 102: 1255-1268.
[3] Agaccioglu H, Yüksel Y. Side-weir flow in curved channels. Journal of Irrigation and Drainage 1998; 124:163–175.
[4] Cosar A, Agaccioglu H. Discharge coefficient of a triangular side-weir located on a curved channel. Journal of Irrigation and Drainage Engineering 2004; 130: 410–423.
[5] Agaccioglu H, Onen F. Clear-water scour at a side-weir intersection along the bend. Journal of Irrigation and Drainage 2005; 54: 553–569.
[6] Karizi A, Honar T. Investigation of flow Pattern and Shear stress distribution in Side Weirs with Round and Sharp corners Inlet. J Sci &Technol.Agric & Natur Resour 2008; 14: 15–25.
[7] Mangarulkar K. Experimental and numerical study of the characteristics of side weir flows, PhD thesis, Concordia University, Montreal, Quebec, Canada 2010.
[8] Hager WH. Lateral outflow over side weirs. Journal of Hydraulic Engineering 1987; 113: 491–504.