Flow Measurement and Instrumentation 46 (2015) 170–178

Contents lists available at ScienceDirect

Flow Measurement and Instrumentation

journal homepage: www.elsevier.com/locate/flowmeasinst

The flow pattern in triangular channels along the side weir for subcritical flow regime

Hamed Azimi [a,b], Saeid Shabanlou [c,*]

[a] Department of Civil Engineering, Razi University, Kermanshah, Iran
[b] Water and Wastewater Research Center, Razi University, Kermanshah, Iran
[c] Ph.D. of hydrology and water resource, Department of water engineering, college of agriculture, Kermanshah branch, Islamic azad university, Kermanshah, Iran

ARTICLE INFO

Article history:
Received 20 November 2014
Received in revised form
24 March 2015
Accepted 3 April 2015
Available online 16 April 2015

Keywords:
Triangular channel
Side weir
Numerical simulation
Free surface
Froude number effects

ABSTRACT

Side weirs in triangular channels are used for control and measurement of flow in irrigation networks and drainage systems. The main purpose of this study is investigation of the effects of side weirs on the subcritical flow conditions parameters in the triangular channels that is not 3D simulated numerically, so far. The turbulent RNG $k-\varepsilon$ model and the volume of fluid (VOF) scheme were used to simulate turbulent flow field and flow free surface in a triangular channel along the side weir, respectively. Comparison of numerical and experimental results shows that the numerical model simulates the free surface and flow field characteristics with acceptable accuracy. First, the flow pattern within triangular channel along the side weir in subcritical flow regime is investigated numerically. Due to the side weir inlet effects, a sudden drop occurred at the first quarter of the side weir length. Then, the effects of subcritical regime Froude numbers in range of 0.406–0.793 are investigated. At all the Froude numbers, a surface jump occurred along the end quarter of the weir length. The length of this surface jump increased with increasing the Froude number. The angle of spilling jet pattern (φ) was similar for all the Froude numbers. However, the minimum φ decreased upon increasing Froude number. For all the Froude numbers, a secondary flow cell was produced after the side weir, which developed as it moved downstream in the main channel.

1. Introduction

When a side weir is installed on a main side channel, excess water flows over the side weir into the side channel. Side weirs are structures used for controlling and directing flow of water in irrigation–drainage systems, urban sewage disposal networks, and flood protection plans. The flow over a side weir is considered as a spatial variable flow (SVF) with decreasing discharge. Various studies as well as experiments have been conducted on the hydraulic behavior of rectangular channels fitted with the side weirs. Emiroglu et al. [1] conducted a series of experiments to study the longitudinal profile of the flow along a side weir. They assumed that the side weir flow rate coefficient of a rectangular channel was a function of some variables and obtained a formula for calculating discharge coefficient [1]. Bagheri and Heidarpour [2] conducted laboratory experiments to study velocity field characteristics of the flow over a rectangular side weir at various Froude numbers [2]. By using fluid visualization techniques, Novak et al. [3] conducted experiments to study flow field characteristics over a rectangular side weir in subcritical flow regime [3]. U-shaped channels are used as transitional sections from circular to rectangular channels in sewage manholes. By conducting an experiment and presenting an analytic method, Uyumaz [4] investigated the flow in a U-shape channel [4]. Vatankhah [6] used a specific energy principle and an incomplete elliptic integrals method to predict the free surface profile of flow in the U-shaped channel along a side weir [5]. Also, Vatankhah [6] applied the specific energy principle and the incomplete elliptic integrals method to solve the free surface profile of flow over a rectangular weir in a parabolic channel [6]. Triangular channels are widely used in irrigation and drainage systems. Uyumaz [7] conducted an analytic study on triangular channels along the side weirs [7]. Using the energy principles as well as the finite element method, Uyumaz proposed a method for designing side weirs in triangular channels. Vatankhah [8] used the direct integration method theorized on the basis of energy principles to present an analytic method for triangular channels along the side weirs [8]. Saving time as well as laboratory costs is among the greatest advantages of numerical simulation. Some of the most important numerical studies related to simulation of flow over side weirs are as follows: using the VOF method and the turbulent $k-\omega$ model, Qu (2005) simulated the free surface profile of flow over a side weir at a weir length to channel width (L/B) ratio of 0.4, and predicted the stagnation zone location at the lower end of the side

* Corresponding author.
E-mail addresses: Azimi86Hamed@gmail.com (H. Azimi), saeid.shabanlou@gmail.com (S. Shabanlou).

http://dx.doi.org/10.1016/j.flowmeasinst.2015.04.003
0955-5986/

Nomenclature

A_X, A_Y, A_Z	fractional areas open to flow (–)
A	cross-sectional area of flow (–)
C_u	constant coefficient (–)
E	specific energy (m)
F	fluid volume fraction in a cell (–)
F_1	Froude number at beginning of side weir on axis of main channel (-)
f_X, f_Y, f_Z	viscous accelerations (–)
G_X, G_Y, G_Z	body accelerations (–)
g	acceleration gravity ($m\ s^{-2}$)
k_t	turbulence kinetic energy ($m^2\ s^{-2}$)
L	side weir length (m)
P	side weir height (m)
p	pressure ($N\ m^{-2}$)
Q	discharge in the main channel ($m^3\ s^{-1}$)
Q_1	discharge at section 1 in the main channel ($m^3\ s^{-1}$)
R_{SOR}	mass source (–)
S_0	bed slope of the main channel (–)
T_{len}	turbulent length scale (m)
t	time (s)
u, v, w	velocity components ($m\ s^{-1}$)
u_*	wall shear velocity ($m\ s^{-1}$)
V_F	fractional volume open to flow (–)
V_X	longitudinal velocity component ($m\ s^{-1}$)
V_Y	lateral velocity component ($m\ s^{-1}$)
V_Z	vertical velocity component ($m\ s^{-1}$)
X, Y, Z	Cartesian coordinate directions (m)
y_1	distance of the cell center from the solid wall (m)
y^+	non-dimensional parameter (–)
Z_1	depth of the flow at section 1 in the main channel (m)
Z_2	depth of the flow at section 2 in the main channel (m)
α	bottom angle of the triangular cross section (–)
δ	secondary flow strength (m/s)
ε_t	turbulence dissipation rate ($m^2\ s^{-3}$)
ν	kinematic viscosity ($m^2\ s^{-1}$)
ν_t	turbulent kinematic viscosity ($m^2\ s^{-1}$)
ρ	fluid density ($kg\ m^{-3}$)
φ	angle of the spilling jet (degree)
Ω	streamwise vorticity (s^{-1})

weir [9]. By use of the turbulent RNG $k-\omega$ model and the VOF method, Mangarulkar [10] simulated free surface variations of flow through a rectangular channel along a side weir at zero crest height [10]. Aydin and Emiroglu [11] used the FLUENT-ANSYS software to simulate discharge capacity in the triangular labyrinth side weir of a rectangular channel [11]. Azimi et al. [12] simulated flow free surface and field of velocity in a circular channel along the side weir with RNG $k-\varepsilon$turbulence model and VOF scheme [12].

Review of the previous studies on side weirs in channels with different shapes shows that the 3D flow pattern in triangular channels with the side weir has not been studied numerically yet. The main objective of this research is the study of the side weir's effects on the flow pattern in triangular channels in subcritical flow conditions, numerically.

In the numerical study, the free surface variation and flow field turbulence were simulated by the VOF method and RNG $k-\varepsilon$ turbulence model respectively. In continuation, the effects of the side weir upstream Froude number on flow pattern in the main channel were investigated.

1.1. Governing equations

To solve the flow field in this numerical study the continuity equation and Reynolds-averaged Navier–Stokes equations are used as follows:

$$V_F\frac{\partial\rho}{\partial t}+\frac{\partial(\rho uA_x)}{\partial x}+\frac{\partial(\rho vA_y)}{\partial y}+\frac{\partial(\rho wA_z)}{\partial z}=R_{SOR} \tag{1}$$

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left(uA_X\frac{\partial u}{\partial X}+vA_Y\frac{\partial u}{\partial Y}+wA_Z\frac{\partial u}{\partial Z}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial X}+G_X+f_X \tag{2}$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial y}+G_y+f_y \tag{3}$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left(uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial z}+G_z+f_z \tag{4}$$

where$(u,\ v,\ w)$,$(A_x,\ A_y,\ A_z)$, $(G_x,\ G_y,\ G_z)$, and $\left(f_x,\ f_y,\ f_z\right)$ are the velocity components, fractional area opens to flow, gravitational forces and accelerations due to viscosity in x, y and z directions, respectively. Also t, ρ, R_{SOR}, p and V_F are time, density, term of the source, pressure and fractional volume open to flow, respectively. In present study for simulation of flow turbulence the RNG $k-\varepsilon$ model has been used. This model simulates the regions with high stress and turbulent flows with high accuracy. Also, the RNG $k-\varepsilon$ turbulence model is more applicable than the two-equation turbulence models (for example, the standard $k-\varepsilon$ turbulence model) and needs less experimental proof and shows better performance for simulation of areas with flow separation. In numerical modeling of the flow field, simulation of the free surface is very important. One of the most important methods that simulate the flow free surface using the interface capturing method is volume of fluid (VOF) scheme. This scheme was provided for the first time by Hirt and Nichols [13]. In this numerical simulation, the free surface variations have been modeled using VOF model. In the VOF method the following continuity equation is used for simulating the computational field changes:

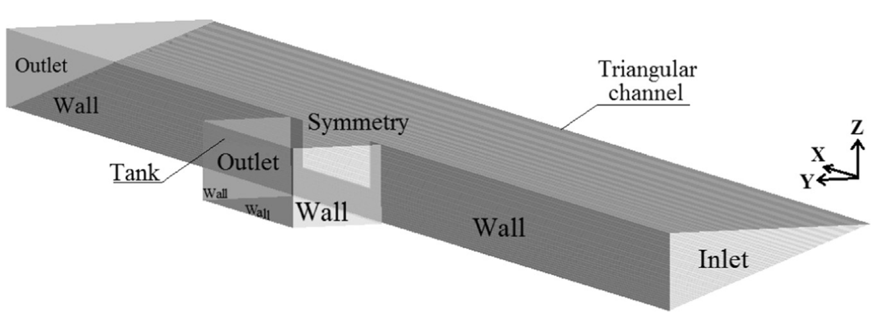

Fig. 1. The used boundary conditions in the numerical simulation.