Applied Mathematical Modelling 37 (2013) 6661–6669

Contents lists available at SciVerse ScienceDirect

Applied Mathematical Modelling

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

2D numerical simulation of submerged hydraulic jumps

M. Javan *, A. Eghbalzadeh

Department of Civil Engineering, Razi University, Kermanshah, Iran

ARTICLE INFO

Article history:
Received 19 November 2011
Received in revised form 14 November 2012
Accepted 21 December 2012
Available online 2 January 2013

Keywords:
Free surface
Hydraulic jump
Turbulent flow
Numerical simulation
Water jets

ABSTRACT

Hydraulic jumps are usually used to dissipate energy in hydraulic engineering. In this paper, the turbulent submerged hydraulic jumps are simulated by solving the unsteady Reynolds averaged Navier–Stokes equations along with the continuity equation and the standard k–ε equations for turbulence modeling. The Lagrangian moving grid method is employed for the simulation of the free surface. In the developed model, kinematic free-surface boundary condition is solved simultaneously with the momentum and continuity equations, so that the water elevation can be obtained along with velocity and pressure fields as part of the solution. Computational results are presented for Froude numbers ranging from 3.2 to 8.2 and submergence factors ranging from 0.24 to 0.85. Comparisons with experimental measurements show that numerical model can simulate the velocity field, variation of free surface, maximum velocity, Reynolds shear and normal stresses at various stations with reasonable accuracy.

1. Introduction

Hydraulic jumps have been studied extensively because of their importance in energy dissipation in hydraulic structures. Several laboratory experiments have been performed to study turbulent Hydraulic jumps. Rouse et al. [1] studied the turbulence characteristics using an air model despite of the differences between air and water flows. Viewing them as wall jets, Rajaratnam [2,3], Eriksson et al. [4] and Ead and Rajaratnam [5] studied the jumps and conducted many experiments. Many others also conducted experimental research but because of the complexity in nature of the turbulence production and diffusion process numerous investigations have been made mainly towards the understanding of their macroscopic structures [6–8]. Long et al. [9] have provided valuable experimental data about the turbulence characteristics of submerged hydraulic jumps with Laser Doppler Anemometry (LDA). Zare and Baddour [10] studied 3D characteristics of submerged hydraulic jumps in a symmetric sudden expansion.

Field and laboratory experiments can provide valuable information on flow characteristics by measurements and flow visualization, but the cost to conduct these experiments is expensive. With the rapid development of numerical methods and advancements in computer technology, computational fluid dynamics (CFD) has been widely used to study submerged hydraulic jumps. Numerical techniques available for the computation of free-surface flows can be divided into two categories, i.e. the fixed-grid (Eulerian) and the moving-grid (Lagrangian) methods. The Eulerian methods, such as volume of fluid (VOF) method, are based on the concept of only fractional volume of the surface cell being occupied by fluid [11–14]. In Lagrangian methods the free-surface is located at one boundary of the mesh, and the mesh deforms as the free surface moves.

* Corresponding author. Address: Department of Civil Engineering, Razi University, Baghe Abrisham, Kermanshah, Iran. Tel.: +98 (831) 4274535 9; fax: +98 (831) 4283264.

E-mail addresses: javanmi@gmail.com, mjavan@razi.ac.ir (M. Javan).

0307-904X/$ - see front matter © 2013 Elsevier Inc. All rights reserved.
http://dx.doi.org/10.1016/j.apm.2012.12.016

There are still only a few numerical studies available in literature which investigate the characteristics of hydraulic jumps. Long et al. [15] made numerical simulations with the standard k–ε turbulence model in Cartesian coordinates. An offset control volume method for discretizing the variable domain, coupled with the hybrid scheme and the SIMPLE algorithm is implemented in the numerical simulation by Long et al. [15]. However they used the governing equations for steady flow, and the approach used for the treatment of free surface was limited to flows with small surface slopes. Qingchao and Drewes [16] also studied the turbulence characteristics in free and forced hydraulic jumps with a numerical method. Ma et al. [17] investigated numerically the turbulence characteristics of submerged hydraulic jumps by means of the standard k–ε turbulence model. Rostami et al. [18] simulated 2D undular hydraulic jumps using the FLOW-3D software. The concept of a fractional volume of fluid (VOF) is employed to track the moving free surface by Qingchao and Drewes [16], Ma et al. [17] and Rostami et al. [18].

Among these researchers, only Long et al. [15] and Ma et al. [17] studied submerged hydraulic jumps. Long et al. [15] used steady state of the Navier–Stokes equations. On the other hand, The VOF method employed by Ma et al. [17] has severe stability restriction and inhibited from being applied to problems with a large computational domain.

The main purpose of this study is to develop a numerical model to solve unsteady Reynolds-averaged Navier–Stokes equations. The numerical algorithm presented in [19] is firstly employed to simulate free surface of a submerged hydraulic jump. It is expected that this Lagrangian moving grid method may reduce run time of model compared to VOF method. A time splitting method on a non-staggered grid in curvilinear coordinates for simulation of two-dimensional (2D) turbulent submerged hydraulic jump is developed. In this model, kinematic free-surface boundary condition is solved simultaneously with the momentum and continuity equations, so that the water elevation can be obtained along with velocity and pressure fields as part of the solution. The non-staggered-grid method of Rhie and Chow [20] couples the volume flux on the face of the cell to the Cartesian velocity components at the cell centre. In this way, both momentum and the continuity equations are enforced in the same control volume.

2. Governing equations

The flow field is determined by the following incompressible fluid Reynolds-averaged continuity and momentum equations. The equations are written here in general form:

$$\partial u_i/\partial x_i = 0, \tag{1}$$

$$\partial u_i/\partial t + \partial u_i u_j/\partial x_j = -\partial\varphi/\partial x_i + 1/\rho\partial\tau_{ij}/\partial x_j, \tag{2}$$

where u_i ($i = 1,2$) are the velocity components; φ is the pressure divided by fluid density; ρ = fluid density. The turbulent stresses τ_{ij} are calculated with the standard k–ε turbulence model [21], which employs the eddy viscosity relation:

$$\tau_{ij} = \rho v_t(\partial u_i/\partial x_j + \partial u_j/\partial x_i) - 2/3\delta_{ij}k \quad \text{with } v_t = c_\mu k^2/\varepsilon, \tag{3}$$

where the turbulent kinetic energy k and its dissipation rate ε determining the eddy viscosity v_t are obtained from the following equations:

$$\partial k/\partial t + \partial u_j k/\partial x_j = \partial(v_t/\sigma_k\partial k/\partial x_j)/\partial x_j + G - \varepsilon, \tag{4}$$

$$\partial\varepsilon/\partial t + \partial u_j\varepsilon/\partial x_j = \partial(v_t/\sigma_\varepsilon\partial\varepsilon/\partial x_j)/\partial x_j + (c_{\varepsilon1}G - c_{\varepsilon2}\varepsilon)\varepsilon/k. \tag{5}$$

Here $G = v_t((\partial u_i/\partial x_j) + (\partial u_j/\partial x_i))(\partial u_i/\partial x_j)$ is the production of k. The standard values of the model coefficients are used: $c_\mu = 0.09$, $c_{\varepsilon1} = 1.44$, $c_{\varepsilon2} = 1.92$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$.

Governing equations are transformed into curvilinear coordinates in strong-conservation-law form:

$$\partial U_m/\partial\xi_m = 0, \tag{6}$$

$$\partial J^{-1}u_i/\partial t + \partial F_{im}/\partial\xi_m = 0, \tag{7}$$

where the flux is

$$F_{im} = U_m u_i + J^{-1}\partial\xi_m/\partial x_i\varphi - v_t GG^{mn}\partial u_i/\partial\xi_n, \tag{8}$$

J^{-1} is the inverse of Jacobian or volume of the cell; U_m is the volume flux (contravariant velocity by J^{-1}) normal to the surface of constant ξ_m; and GG^{mn} is called mesh the "skewness tensor". These quantities are expressed as:

$$U_m = J^{-1}\partial\xi_m/\partial x_j u_j, \tag{9}$$

$$J^{-1} = \det(\partial x_i/\partial\xi_j), \tag{10}$$

$$GG^{mn} = J^{-1}\partial\xi_m/\partial x_j\partial\xi_n/\partial x_j. \tag{11}$$