Applied Mathematical Modelling 39 (2015) 6285–6306

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

3D numerical simulation of free surface flows over hydraulic structures in natural channels and rivers

Van Thinh Nguyen *

Department of Civil and Environmental Engineering, Seoul National University, Republic of Korea

ARTICLE INFO

Article history:
Received 21 March 2014
Received in revised form 18 December 2014
Accepted 19 January 2015
Available online 7 February 2015

Keywords:
Free surface flow
Front tracking method
Front capturing method
Finite Volume Method
Turbulent flows
Reynolds-averaged Navier–Stokes (RANS)

ABSTRACT

In this paper, a three-dimensional numerical model is developed, in which an Euler implicit method for the temporal discretization and the Finite Volume Method for the spatial discretization are applied to solve the Reynolds-averaged Navier–Stokes (RANS) equations for free surface flows over hydraulic structures in natural channels and rivers. At the free surface two different methods, the front-tracking and the front-capturing, are applied to calculate free surface profiles. The model has been validated against typical benchmarking experiments, and applied to a number of practical applications for natural rivers in Germany. Due to the limitation of the observation data in some application cases, in order to verify our numerical model, we besides have modified and applied the well-known open source CFD toolbox OpenFOAM to the same applications, then compared the results obtained from the OpenFOAM with our model results.

1. Introduction

The flow over hydraulic structures in open channels and natural rivers, such as weirs, barrages, sluice gates, spillways, etc. is very complicated. It is characterized by highly turbulent and three dimensional flows including secondary currents, flow separation, and reattachment. For such flows it is very difficult to accurately calculate water surface profiles, turbulence structures, and secondary currents. Simulating the rapidly varied flow fields that involve a breakup of a free surface is still an important component and an active area of research. The secondary currents and turbulent structures also play an important role in nautical problems, bed and bank erosions and sediment transports. To simulate these motions, a special class of 3D non-hydrostatic models must be used instead of the rigid-lid approximation, and should be capable of simulating transient oscillations and breakup of a free surface (hydraulic jump), the turbulent structures and secondary motion occurred in the regions near to the surface and hydraulic structures.

Flow over spillways has been studied by many authors. Olsen and Kejellesvig [1] and Guo et al. [2] used numerical models to calculate the discharges through spillways. Song and Zhou [3] have coupled 1D and 3D models to simulate the flow over a spillway. Using the commercial CFD package Flow-3D, Savage et al. [4] have carried out numerical calculations for the flow over Ogee spillway, or using the open source CFD toolbox OpenFOAM to study dam break flows is carried out by Biscarini et al. [5]. Recently, Kang and Sotiropoulos [6] have developed a 3D model for turbulent free surface flow in natural waterways, however their model has been validated only against the observation data from the laboratory. Particularly, Nguyen and Nestmann [7] have used two commercial CFD software packages, FIDAP and Comet, to calculate the turbulent flows over

* Address: 1 Gwanak-ro, Gwanak-gu, Seoul 151-744, Republic of Korea. Tel.: +82 2 880 7355; fax: +82 2 873 2684.
E-mail address: vnguyen@snu.ac.kr

http://dx.doi.org/10.1016/j.apm.2015.01.046
0307-904X/

hydraulic structures in natural rivers, however FIDAP and Comet are not well designed for river flows, and no longer existed. Therefore the development of a numerical model for a specific simulation of three dimensional turbulent flows in natural channels and rivers over different man-made hydraulic structures, such as weirs, sluices, barrages, etc. are still in active and challenging research.

It is well-known that DNS (Direct Numerical Simulation) and LES (Large Eddy Simulation) methods will provide more accuracy and detail information on turbulence structures, however DNS and LES are still very expensive in river engineering. DNS is to numerically solve unsteady Navier–Stokes equation directly to calculate the mean flow and all turbulent velocity fluctuations from smallest to largest scales on spatial grids that are sufficiently fine to resolve the Kolmogorov microscales ($\mathrm{Re}_\eta \sim 1$) at which energy dissipation takes place, and with time steps sufficiently small to resolve the period of the fastest fluctuations. These calculations require very strong computer resources, and would exceed the available capacity of most powerful High Performance Computers for a large Reynolds number simulation. So this method is not used for river engineering as a design tool. Instead of solving directly Navier–Stokes equations, LES uses a spatial filtering operation to separate the large and small eddies by a filter with certain cutoff width. All eddies with length scale greater than the cutoff width will be calculated directly as in DNS method, the smaller unresolved ones should be easier to capture with a compact model; a sub-grid-scale (SGS) model. To perform an accurate LES, it requires very fine grids with near-wall grid point $y^+ \leqslant 1$, this requirement is again tackled in highly computing cost. Particularly in river engineering, we usually need to simulate the flows in very large and complicated bathymetries of a natural river stretch with the length in kilometers. In these applications, for high Reynolds number flows included thin boundary layers it is necessary to reduce a number of grid cells by means of graded non-uniform refine meshes in the near-wall region. Consequently, this would require different filter cutoff widths in the core flow and near-wall regions. In Finite Volume Method, the filter cutoff width $\Delta = \sqrt[3]{\Delta x \Delta y \Delta z}$ is necessary close to the grid size, therefore the cutoff width would vary along with the control volume size for the non-uniform grids. This choice of the cutoff width is inherent a price to pay for, because it can blur the distinction between the effects of the SGS eddies and the numerical errors associated with the discretization of the equations on the grid [8]. In addition, setting up inlet boundary conditions for LES is very challenging since the inlet flow properties are convected downstream, therefore inaccurate specification of the inlet boundary condition can strongly affect simulation quality. There are some alternatives to deal with this issue suggested by Lund et al. [9], Klein et al. [10], Ferrante and Elgobashi [11], Versteeg et al. [12], etc., however most of suggested methods are consequently increased significant computational cost and/or impossible to implement in natural river flows. Furthermore, the validation of the results from LES is somehow not possible due to limitation of observation data from field surveys. Therefore, in this research, we are applying the Finite Volume Method, and still combined with RANS approach (two-equation k–ε model) to simulate free surface flows over hydraulic structures in natural channels and rivers. Numerical results were validated against typical benchmarking experiments such as Rozovskii's and Koshizuka's experiments; and field survey data from the Federal Waterways Engineering and Research Institute of Germany (BAW) at Karlsruhe. Since some parameters (turbulence characteristics, three-dimensional free surface profile, etc.) cannot be obtained from field surveys, we tried to compare our numerical results with the results obtained from the well-known open source CFD toolbox OpenFOAM. In fact, the OpenFOAM by itself cannot solve directly our problems, therefore we had to build a new solver with specific boundary conditions, then immersed it into the OpenFOAM, which is capable of dealing with our applications.

Agreement between our numerical computations, observed data and OpenFOAM's results is judged to be satisfactory on all major comparisons.

2. Basic equations

The computations were based on the solution of the Reynolds-averaged Navier–Stokes (RANS) equations for the mean flow field together with a two-equation turbulence model (k–ε model). All the conservation equations can be re-written in the form of the following generic transport equation for an arbitrary part of the continuum of volume V bounded by surface S

$$\frac{\partial}{\partial t}\int_V \rho\phi dV + \int_S \rho\phi(\mathbf{v} - \mathbf{v}_S)\cdot ds = \int_S \Gamma_\phi\, grad\,\phi \cdot ds + \int_S q_{\phi S}\cdot ds + \int_V q_{\phi V} dV, \tag{1}$$

where $\phi,\ \Gamma_\phi,\ q_{\phi S}$ and $q_{\phi V}$ are given in Table 1, $\mathbf{v}$ is fluid velocity, $\mathbf{v}_s$ is surface velocity, ρ is fluid density, and s is outward pointing surface vector.

Particularly, for $\phi = 1,\ q_{\phi S} = q_{\phi V} = 0$, Eq. (1) becomes the continuity equation,

$$\frac{\partial}{\partial t}\int_V \rho dV + \int_S \rho(\mathbf{v} - \mathbf{v}_S)\cdot ds = 0. \tag{2}$$

3. Initial conditions

At the initial condition at an initial time $t = t_0$ the values of dependent variables $\phi = \mathbf{v}$, P, k and ε should be provided at all points of the computational domain V as follows: