Advances in Water Resources 83 (2015) 263–276

Contents lists available at ScienceDirect

Advances in Water Resources

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

A 3-D implicit finite-volume model of shallow water flows

Weiming Wu [a,*], Qianru Lin [b]

[a] Department of Civil and Environmental Engineering, Clarkson University, Potsdam, NY 13699, USA
[b] National Center for Computational Hydroscience and Engineering, The University of Mississippi, 327 Brevard Hall, University, MS 38677, USA

ARTICLE INFO

Article history:
Received 26 October 2014
Revised 10 June 2015
Accepted 19 June 2015
Available online 25 June 2015

Keywords:
Shallow water flows
Finite volume method
Three-dimensional
Quadtree rectangular mesh
Implicit
Non-staggered grid

ABSTRACT

A three-dimensional (3-D) model has been developed to simulate shallow water flows in large water bodies, such as coastal and estuarine waters. The eddy viscosity is determined using a newly modified mixing length model that uses different mixing length functions for the horizontal and vertical shear strain rates. The 3-D shallow water flow equations with the hydrostatic pressure assumption are solved using an implicit finite-volume method based on a quadtree (telescoping) rectangular mesh on the horizontal plane and the sigma coordinate in the vertical direction. The quadtree technique can locally refine the mesh around structures or in high-gradient regions by splitting a coarse cell into four child cells. The grid nodes are numbered with a one-dimensional index system that has unstructured grid feature for better grid flexibility. All the primary variables are arranged in a non-staggered grid system. Fluxes at cell faces are determined using a Rhie and Chow-type momentum interpolation, to avoid the possible spurious checkerboard oscillations caused by linear interpolation. Each of the discretized governing equations is solved iteratively using the flexible GMRES method with ILUT preconditioning, and coupling of water level and velocity among these equations is achieved by using the SIMPLEC algorithm with under-relaxation. The model has been tested in four cases, including steady flow near a spur-dyke, tidal flows in San Francisco Bay and Gironde Estuary, and wind-induced current in a flume. The calculated water levels and velocities are in good agreement with the measured values.

1. Introduction

Three-dimensional flow features significantly affect the mass transport and morphology in surface water systems. Even though many one-dimensional (1-D) and depth-averaged two-dimensional (2-D) models have been widely used with certain success, realistic simulation of these complex features preferably needs a three-dimensional (3-D) model, which can be based on the full 3-D Navier–Stokes equations or the 3-D shallow water equations that are simplified from the Navier–Stokes equations by assuming hydrostatic pressure or gradually-varied flow. Solving the 3-D Navier–Stokes equations to compute flows in large water bodies such as coastal and estuarine waters is still very time-consuming, even with the ever-increasing capacity of computing technologies. The cheaper approach is using the 3-D shallow water flow equations in cases where the hydrostatic pressure assumption is approximately valid.

Efficiently solving the 3-D shallow water equations has been a great challenge. Several early 3-D shallow water flow models, such as CH3D (Curvilinear-grid Hydrodynamics 3D model) originally developed by Sheng [29] and POM (Princeton Ocean Model) by Blumberg and Mellor [2], split the governing equations into internal and external modes. The internal mode handles the slower vertical baroclinic flow structures, while the external mode computes the depth-integrated quantities that are governed by the fast barotropic dynamics. Different time steps can be used in the internal and external modes accordingly, but there may be a tendency of velocity mismatch between internal and external modes that needs to be corrected during the time marching. This problem has been addressed in latter models FVCOM [4] and ROMS (Regional Ocean Modeling System; [7]), which also adopt the mode-splitting approach. Leendertse [11] developed an alternating implicit method to solve the continuity and momentum equations, which later evolves to the Delft3D model system [12]. Casulli and Cheng [3] proposed a semi-implicit projection-type finite-difference method to solve the 3-D shallow water flow equations without splitting the internal and external modes. This semi-implicit approach has been found to be efficient and adopted by several models. In addition, the 3-D models developed by Lin and Falconer [14] and Shanhar et al. [28] divide the flow depth into several layers, integrate the 3-D shallow water flow equations in each layer, and solve the layer-integrated continuity and momentum equations using 2-D solution methods.

The 3-D shallow water equations inherit the weakness of the Navier–Stokes equations, in which the velocity is linked to the water

* Corresponding author. Tel.: +1 315 2686550.
E-mail address: wwu@clarkson.edu (W. Wu).

http://dx.doi.org/10.1016/j.advwatres.2015.06.008
0309-1708/

level (pressure) gradient by the momentum equations but the continuity equation is just an additional constraint on the velocity field without directly linking to the water level. Because of such a weak linkage, coupling of velocity and water level is an important issue in solving the 3-D shallow water equations. The key aspect is the arrangement of the grid system and primitive variables. Staggered grid arrangement, often used with MAC (Marker and Cell) method [8], projection method [5], and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm family [17,18] for the Navier–Stokes equations, can eliminate potential oscillations. The majority of the aforementioned existing 3-D shallow water models thus use staggered grids (e.g., [3,4,11]). However, the staggered grid is somehow inconvenient in 3-D models with non-rectilinear mesh systems because it uses different grids and control volumes for the velocity components and water level (pressure). The advantage of non-staggered (collocated) grid arrangement is its simplicity in arrangement of primitive variables in a single grid, but it was criticized because potential checkerboard oscillations may appear when the linear interpolation method is used to evaluate the interfacial fluxes [17]. Such oscillations can be eliminated by using the momentum interpolation method proposed by Rhie and Chow [23]. Therefore, the non-staggered grid has been used more and more in 3-D CFD models, especially in curvilinear or other complicated meshes [6,19,34].

To handle the complex computational domain, grid flexibility is often another issue in numerical modeling of surface water flows. A simple rectangular mesh is difficult to conform to the irregular boundaries and locally refine the mesh around the structures or high-gradient regions. A structured quadrilateral (curvilinear) grid can serve this purpose by stretching or shrinking the mesh sizes, but it is less flexible for a very large, complex domain than the unstructured triangular mesh (e.g., [4]). On the other hand, the rectangular or quadrilateral mesh is more convenient than the triangular mesh for establishing high-order (e.g., second and third) schemes or for discretizing second or higher order spatial derivatives. Therefore, quadtree (telescoping) rectangular or quadrilateral mesh has been recently used for local refinement of computational mesh [6,16,19,35]. On the quadtree mesh, a coarse cell is split into four child cells and thus the mesh can be easily refined locally.

Therefore, a 3-D shallow water flow model is developed in this study using the recently advanced technologies mentioned above. The model uses the quadtree rectangular mesh on the horizontal plane and the sigma coordinate in the vertical direction. The governing equations are solved using the finite volume method based on non-staggered grid. The SIMPLEC algorithm with under-relaxation and the Rhie and Chow's momentum interpolation method are used to handle the coupling of velocity and water level. Mathematical formulations, numerical solution methods and validations of the developed model are described below.

2. Model formulations

2.1. 3-D shallow water flow equations

The 3-D shallow water equations in the Cartesian coordinate system are written as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (vu)}{\partial y}+\frac{\partial (wu)}{\partial z} = -g\frac{\partial \eta}{\partial x}+\frac{\partial}{\partial x}\left(\nu_{tH}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\nu_{tH}\frac{\partial u}{\partial y}\right) + \frac{\partial}{\partial z}\left(\nu_{tV}\frac{\partial u}{\partial z}\right)+f_c v \tag{2}$$

$$\frac{\partial v}{\partial t}+\frac{\partial (uv)}{\partial x}+\frac{\partial (vv)}{\partial y}+\frac{\partial (wv)}{\partial z} = -g\frac{\partial \eta}{\partial y}+\frac{\partial}{\partial x}\left(\nu_{tH}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\nu_{tH}\frac{\partial v}{\partial y}\right) + \frac{\partial}{\partial z}\left(\nu_{tV}\frac{\partial v}{\partial z}\right)-f_c u \tag{3}$$

where t is time; x and y are the horizontal coordinates; z is the vertical coordinate pointing upward above the reference datum (e.g., the seal level or still water level); u, v, and w are the velocities in x-, y-, and z-directions; η is the water surface elevation; g is the gravitational acceleration; ν_{tH} and ν_{tV} are the horizontal and vertical eddy viscosities, respectively; and f_c is the Coriolis coefficient.

Note that because the main focus here is the numerical solution algorithm, Eqs. (1)–(3) consider only a constant water density. In general, the water density in coastal and estuarine waters varies in space and time due to salinity, temperature or sediment transport. These features are not considered here for simplicity.

2.2. Eddy viscosity

The eddy viscosity can be evaluated using many turbulent models, from zero- to second-order [24]. Because high-order turbulence models usually require fine meshes and are expensive to use for flows in large domains, only zero-order turbulence models, including the parabolic eddy viscosity model, subgrid model [30] and mixing length model, have been implemented here. Among these options, the mixing length model is found to be adequate for a relatively wide range of problems. The mixing length model was first proposed by Prandtl [21] for two-dimensional boundary layer flows. It is modified as follows for the eddy viscosity, ν_t, of the 3-D shallow water flow:

$$\nu_t=\sqrt{\left(l_{mV}^2\left|\bar{S}_V\right|\right)^2+\left(l_{mH}^2\left|\bar{S}_H\right|\right)^2} \tag{4}$$

where l_{mV} is the vertical mixing length, l_{mH} is the horizontal mixing length, $|\bar{S}|_V=[(\partial u/\partial z)^2+(\partial v/\partial z)^2]^{1/2}$, and $|\bar{S}|_H=[2(\partial u/\partial x)^2+2(\partial v/\partial y)^2+(\partial u/\partial y+\partial v/\partial x)^2]^{1/2}$. Eq. (4) is a combination of the horizontal and vertical mixing length models and ignores the contribution of vertical velocity that is assumed much smaller than the horizontal velocities in the case of shallow water flow. The vertical and horizontal mixing lengths are determined as

$$l_{mV}=\kappa z'\sqrt{1-z'/h} \tag{5}$$

$$l_{mH}=\kappa\ \min\,(l, c_m h) \tag{6}$$

where z' is the vertical coordinate above the bed, l is the horizontal distance to the nearest solid wall, h is the total flow depth, κ is the von Karman constant, and c_m is a coefficient which can be calibrated.

Eq. (5) is the mixing length for vertical two-dimensional open-channel flow proposed by Саткевич [37]. Eq. (6) was used by Wu [34] in a depth-averaged 2-D flow model. Both are modified from Prandtl's mixing length $l_m=\kappa z'$ of boundary layer flows. Eq. (5) can be applied in the entire depth of open-channel flow, whereas the mixing length of Prandtl is only for the log-law layer near wall boundary. Eq. (6) considers the constraint of the horizontal eddy scale by the local flow depth.

Eq. (4) is used to determine the horizontal and vertical eddy viscosities, ν_{tH} and ν_{tV}. Different values of c_m in Eq. (6) may be used for ν_{tH} and ν_{tV}, to consider different length scales of turbulence structures in the horizontal and vertical directions. Wu [34] reports c_m between 0.4 and 1.2 for a depth-averaged 2-D model, which can be a reference for c_m in the horizontal eddy viscosity. In the present study, an isotopic value of 0.3 for c_m is found to work well in the four test cases described later. This value is slightly out of the range recommended by Wu [34], perhaps due to different calculations of shear strain rates in 2-D and 3-D models.