Applied Mathematical Modelling 45 (2017) 653–683

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

A two-dimensional artificial viscosity technique for modelling discontinuity in shallow water flows

Bobby Minola Ginting

Department of Civil Engineering, Faculty of Engineering, Parahyangan Catholic University, Jalan Ciumbuleuit No. 94, Bandung, West Java 40141, Indonesia

ARTICLE INFO

Article history:
Received 8 October 2015
Revised 29 November 2016
Accepted 5 January 2017
Available online 12 January 2017

Keywords:
Discontinuous flows
Hybrid formulation
Modified artificial viscosity
Runge–Kutta method
Shallow water equations

ABSTRACT

In this study, a two-dimensional cell-centred finite volume scheme is used to simulate discontinuity in shallow water flows. Instead of using a Riemann solver, an artificial viscosity technique is developed to minimise unphysical oscillations. This is constructed from a combination of a Laplacian and a biharmonic operator using a maximum eigenvalue of the Jacobian matrix. In order to achieve high-order accuracy in time, we use the fourth-order Runge–Kutta method. A hybrid formulation is then proposed to reduce computational time, in which the artificial viscosity technique is only performed once per time step. The convective flux of the shallow water equations is still re-evaluated four times, but only by averaging left and right states, thus making the computation much cheaper. A comparison of analytical and laboratory results shows that this method is highly accurate for dealing with discontinuous flows. As such, this artificial viscosity technique could become a promising method for solving the shallow water equations.

1. Introduction

Nowadays, numerical models of the shallow water equations (SWEs) play an important role in simulating hydrodynamics in many real applications, e.g. in rivers, lakes, floodplains and coastal areas. These numerical models should be efficient and accurate, even when simulating complex flow problems. However, when they encounter discontinuous flows, these models may become unstable because of significant oscillations, and therefore require special treatment. Finding the best way to treat these difficulties remains an outstanding research challenge.

Within the framework of the finite volume scheme, some significant developments have occurred in the past two decades, particularly for Riemann solvers such as the Roe, HLL or HLLC schemes, which have been proven to be robust. We refer to [1–16]. A Riemann solver can capture the sharp gradients of a hyperbolic system with low-level diffusion and oscillations. Therefore, this solver has become very popular for solving the SWEs.

In a cell-centred finite volume (CCFV)[1] scheme, a Riemann solver is used to re-evaluate the convective flux of the SWEs. In the Riemann solver approximation of Roe, the convective flux is re-evaluated based on a linearisation consisting of eigenvalues and eigenvectors of the approximate Jacobian matrix. In the Harten–Lax–van Leer-contact (HLLC) scheme, the left, middle (contact) and right wave speeds of every face should be computed to re-evaluate the convective flux. The main variables used in these schemes (such as depth and velocities or unit discharges) are usually reconstructed using a gradient function, which is based on the cell-average value, in order to achieve second-order accuracy in space. To extend

E-mail addresses: bobbyminola.g@unpar.ac.id, bobbyminola.g@gmail.com

[1] CCFV = cell-centred finite volume.

http://dx.doi.org/10.1016/j.apm.2017.01.013
0307-904X/

this to high-order accuracy in time without introducing unphysical oscillations, the Runge–Kutta scheme is generally used. However, computation using a Riemann solver may become very expensive when using a high-order scheme such as the Runge–Kutta fourth-order scheme. This is because the solver has to be computed four times per time step in order to re-evaluate the convective flux. For example, a domain that is to be simulated for 10 s with a 1-s time step requires this solver to be computed four times per second. This would mean performing 40 computations of the Riemann solver for 10 s, which would create an unavoidably high computational cost.

The present idea emerged from the above problem. We hypothesise that a cheaper method can be found that avoids the need to execute the Riemann solver at all when using the Runge–Kutta fourth-order scheme. Instead, we develop an artificial viscosity technique that replaces the role of the Riemann solver. This technique is cheaper because it is only computed once per time step. The convective flux is still re-evaluated four times, but only by averaging the left (L) and right (R) states. We will show that this artificial viscosity technique can also minimise unphysical oscillations and is highly accurate when dealing with discontinuous flows. However, we would like to emphasise that we are not claiming that our present method is better than the Riemann solver, but rather that it could be used as a cheaper alternative. Further explanation of the comparison between the Riemann solver and our present method is given in Section 2.5.

The artificial viscosity technique was very popular in the 1980s and 1990s, particularly in aeronautics. Jameson et al. [17] pioneered the development of the artificial viscosity technique in the form of the well-known pseudo-second-order Jameson–Schmidt–Turkel (JST) scheme. They solved the Euler equations to simulate inviscid transonic flows associated with shock waves on structured curvilinear meshes. Jameson and Mavriplis [18] and Mavriplis [19] then developed this model on unstructured triangular meshes, achieving stable and accurate results. Van der Burg et al. [20] improved the JST scheme by replacing the Lax–Friedrichs dissipative flux with Roe's one. This new scheme was more accurate but also more expensive, since in Roe's scheme the Jacobian matrix must be accounted for. Swanson et al. [21] presented a modified JST scheme in a form similar to a total variation diminishing (TVD) scheme. They proved that the scheme could properly minimise oscillations due to strong shock waves.

In hydrodynamic and free-surface flow simulations, we are not aware of any significant developments of the artificial viscosity technique in relation to the Riemann solver. We note some works here that are based on the finite volume method and also on the finite difference method. Fennema and Chaudhry [22] used explicit second-order finite difference schemes (the MacCormack, Lambda and Gabutti schemes) with an artificial viscosity technique to simulate unsteady free-surface flows associated with shocks and bores. Hino et al. [23] applied an artificial viscosity technique to a finite volume scheme for simulating flows around submerged hydrofoils on an unstructured grid. Younus and Chaudry [24] used an artificial viscosity technique with a finite difference scheme to simulate a hydraulic jump over a transition channel; they also included a k–ε turbulent scheme in their model. Meselhe et al. [25] and Fiedler and Ramirez [26] simulated respectively transcritical flows in an open channel and two-dimensional overland flows using an artificial viscosity technique in a finite difference scheme. Sabbagh-Yazdi et al. [27] solved a time-dependent concentration diffusion using an artificial viscosity technique in a finite volume scheme. Previously, we also developed an artificial viscosity technique using a CCFV scheme to simulate dam-break flows on curvilinear meshes. Interested readers are referred to [28–30]. In the present study, we modify our previous artificial viscosity technique specifically for the determination of the adaptive coefficient. Our intention is to investigate the capability of this new scheme when dealing with discontinuous flows. This paper is organised as follows. The governing equations and numerical model are explained in Section 2. The verifications of our numerical model against some theoretical and experimental benchmark tests are given in Section 3. Finally, conclusions are given in Section 4.

2. Mathematical formulation

In this section, the mathematical formulations of both the governing equations and our numerical model are presented. A CCFV scheme and the Runge–Kutta fourth-order method are used for space and temporal discretisation respectively.

2.1. Governing equations

The integration of the three-dimensional Navier–Stokes equations in the vertical direction with the assumptions of uniformly distributed vertical velocity, negligible hydrostatic pressure distribution, free surface, small bed inclination and incompressibility yields the SWEs. These can be written as

$$\frac{\partial \mathbf{W}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} = \mathbf{S}, \tag{1}$$

where **W, F** and **G** are the vectors described in Eq. (2). This consists of three main variables, H, u and v, which respectively are the water depth and the velocity components in the x and y directions. The variable g is the acceleration due to gravity, and Sx and Sy are the bed slopes in the x and y directions respectively:

$$\mathbf{W} = \begin{bmatrix} H \\ uH \\ vH \end{bmatrix}; \quad \mathbf{F} = \begin{bmatrix} uH \\ u^2H + \frac{1}{2}gH^2 \\ uvH \end{bmatrix}; \quad \mathbf{G} = \begin{bmatrix} vH \\ uvH \\ v^2H + \frac{1}{2}gH^2 \end{bmatrix}; \quad \mathbf{S} = \begin{bmatrix} 0 \\ gH(Sx - Sfx) \\ gH(Sy - Sfy) \end{bmatrix}. \tag{2}$$