Journal of Physics: Conference Series

PAPER • OPEN ACCESS

A finite volume method for solving the gravity wave-model equations

A finite volume method for solving the gravity wave-model equations

Cecilia Heru Purwitaningsih[1] and Sudi Mungkasi[2]

[1]Department of Mathematics Education, Faculty of Teacher Training and Education, Sanata Dharma University, Yogyakarta, Indonesia
[2]Department of Mathematics, Faculty of Science and Technology, Sanata Dharma University, Yogyakarta, Indonesia

E-mail: cecilia.purwita@gmail.com, sudi@usd.ac.id

Abstract. One of models for free surface flows is the gravity wave-model equations. In this paper, we propose a finite volume numerical method for solving the gravity wave-model equations. The numerical scheme is explicit, so it is easy to implement. The scheme is consistent with the gravity wave-model equations. Numerical results show that the proposed method is stable. As cell widths and time steps are taken smaller, the errors of numerical solutions get smaller too. Therefore, we infer that the numerical solutions are convergent to the exact solution.

1. Introduction

Real problems can be modelled mathematically. The Saint-Venant equations form a mathematical model for free-surface flows. The Saint-Venant equations are often simplified for practicality reasons and faster calculation. However, the simplification must still be able to capture the physical phenomena.

The gravity wave-model equations are some simplifications of the Saint-Venant equations by neglecting the convective term in the equation of momentum conservation [1-2]. This is valid as long as the gravity effect is much more dominant than the convective effect. This model can be used for simulation of natural disasters, such as tsunami, floods [3-6] and landslides [7]. The analytical exact solution to the gravity wave-model equations is available only for some specific type of problems, such as the dam break problem studied by Martins et al. [1]. The general analytical exact solution to this model is not yet available. Therefore, in order to solve this model for the general case, a numerical method should be developed. A fast and accurate but simple numerical method is still a research gap.

In this paper, we aim to fill the mentioned research gap by proposing a finite volume numerical method for solving the gravity wave-model equations. The numerical fluxes in our numerical method are calculated using the Lax-Friedrichs formulation. We provide the numerical scheme and some numerical results. We use the dam break problem for a numerical test case. However, note that the dam break problem in this paper is assumed to be governed by the gravity wave-model equations, instead of the usual Saint-Venant equations.

The rest of the paper is written as follows. We recall the gravity wave-model equations in Section 2. We provide the finite volume method that we propose in Section 3. Numerical results are explained in Section 4. Finally, we conclude the paper in Section 5.

2. Gravity wave-model equations

In this section we recall the mathematical model (the gravity wave-model equations) that we want to solve.

The gravity wave-model equations are [1, 8]

$$\frac{\partial}{\partial t}h + \frac{\partial}{\partial x}q = 0\,, \tag{1}$$

and

$$\frac{\partial}{\partial t}q \;+\; \frac{g}{2}\frac{\partial}{\partial x}h^2 = 0\,, \tag{2}$$

where $h = h(x,t)$ is the fluid depth, $q = q(x,t)$ denotes the unit-discharge, g is the gravitational acceleration, t is the time variable and x is the space variable. All quantities are assumed to be in SI units. Here, the conserved quantities are the water depth $h(x,t)$ for equation (1) and the unit-discharge $q(x,t)$ for equation (2). We assume the relation $q(x,t) = u(x,t)h(x,t)$, where $u(x,t)$ is water velocity.

The gravity wave-model equations are some simplifications of the shallow water equations (the Saint-Venant model). That is, the convective term in the momentum equation of the shallow water equations is neglected. This is valid as long as the convective term is assumed to be very much insignificant in comparison with the gravity term. More discussion on the gravity wave-model equations can be found in the work of Martins et al. [1-2].

Our focus in this paper is to solve the gravity wave-model equations numerically. The general exact solution to the gravity wave-model equations is not yet available. However, their exact solutions for some specific cases of the gravity wave-model equations have been identified, for example, see Martins et al. [1] and Mungkasi et al. [9]. We can use them as benchmarks for our numerical tests. In particular, we use the exact solution derived by Martins et al. [1] as the benchmark for the present paper.

3. Finite volume method

In this section, we present the Lax-Friedrichs finite volume method for solving the gravity wave-model equations.

Each of the gravity wave-model equations (1)-(2) is a conservation law in the form of

$$\frac{\partial}{\partial t}Q(x,t) + \frac{\partial}{\partial x}F(Q(x,t)) = 0\,, \tag{3}$$

where $Q(x,t)$ denotes the conserved quantity and $F(x,t)$ represents the corresponding flux function. Using the standard finite volume method for conservation law (3), we have the first order finite volume scheme in the explicit form, as follows:

$$Q_i^{n+1} = Q_i^n - \frac{\Delta t}{\Delta x}\left(F^n_{i+\frac{1}{2}} - F^n_{i-\frac{1}{2}}\right), \tag{4}$$

where $Q_i^n \approx Q(x_i, t^n)$, $F^n_{i+\frac{1}{2}} \approx F\big((x_{i+1/2}, t^n)\big)$, Δt is the time step, and Δx is the space step. The Lax-Friedrichs fluxes for equation (1) are

$$F^n_{i+\frac{1}{2}} = \frac{1}{2}(q^n_{i+1} + q^n_i) - \frac{\Delta x}{2\Delta t}(h^n_{i+1} - h^n_i), \tag{5}$$

and

$$F^n_{i-\frac{1}{2}} = \frac{1}{2}(q^n_i + q^n_{i-1}) - \frac{\Delta x}{2\Delta t}(h^n_i - h^n_{i-1}). \tag{6}$$

The Lax-Friedrichs fluxes for equation (2) are

$$F^n_{i+\frac{1}{2}} = \frac{1}{2}\left(\left(\frac{g}{2}h^2\right)^n_{i+1} + \left(\frac{g}{2}h^2\right)^n_i\right) - \frac{\Delta x}{2\Delta t}(q^n_{i+1} - q^n_i), \tag{7}$$

and

$$F^n_{i-\frac{1}{2}} = \frac{1}{2}\left(\left(\frac{g}{2}h^2\right)^n_i + \left(\frac{g}{2}h^2\right)^n_{i-1}\right) - \frac{\Delta x}{2\Delta t}(q^n_i - q^n_{i-1}). \tag{8}$$

The finite volume scheme (4) with numerical fluxes (5)-(8) can be iterated with respect to the space index i and the time index n, in order to obtain the numerical solution to the gravity wave-model equations. The finite volume method (4)-(8) is consistent with the conservation law (3), as explained by LeVeque [10-11].

4. Numerical results

We present our numerical results. Numerical solutions shall be judged based on its accuracy with respect to the curve matching with the analytical exact solution. All quantities are assumed to have SI units with the MKS system.

In our simulations, we use $g = 1$, $\Delta x = 0.05$, and $\Delta t = 0.25\Delta x$. This time step Δt is set to this value, in order that the numerical method is stable. Our simulations are stopped at time $t = 1$. We assume that the initial fluid depth is $h_1 = 10$ and $h_0 = 5$, as shown in Figure 1. The initial condition for fluid velocity is $u(x, 0) = 0$, for all x. The boundary condition at $x = -L$ is

$$h(-L, t) = 10, \quad u(-L, t) = 0, \tag{9}$$

and at $x = L$ is

$$h(L, t) = 5, \quad u(L, t) = 0. \tag{10}$$

This is called the dam break problem. In this problem, we want to investigate how water flows for time $t > 0$ if the dam wall is removed at time instant $t = 0$. In our simulation, we take $L = 10$.

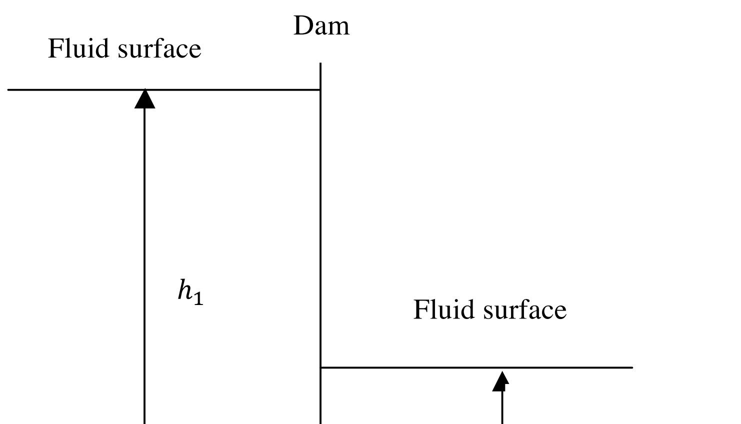

Figure 1. Initial condition of the dam break problem.

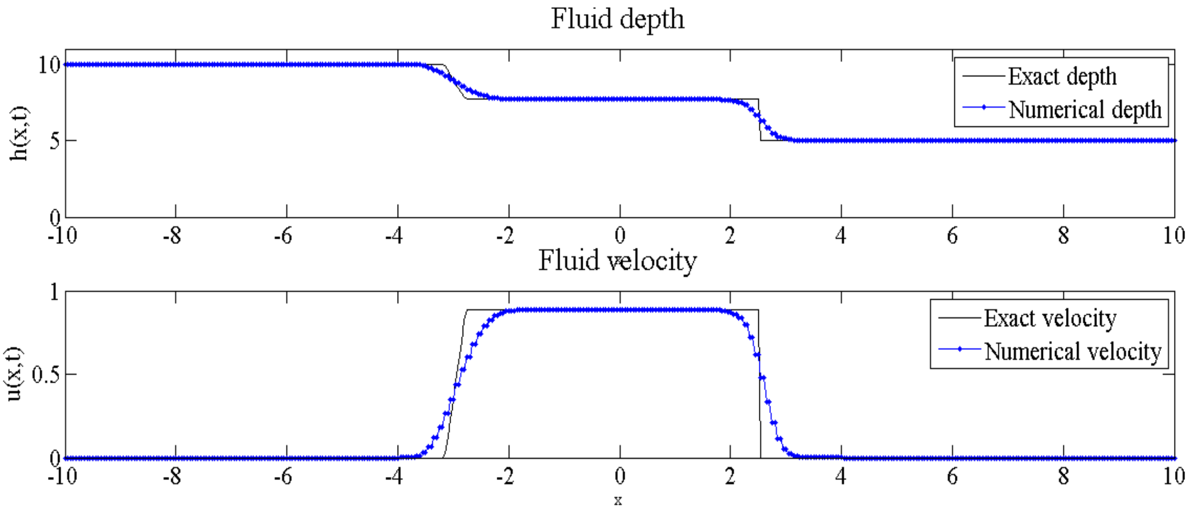

Figure 2. Graph of fluid depth and velocity. Here we compare our numerical solutions with the exact solution. The number of cells is $N = 400$, the uniform cell width is $\Delta x = 0.05$, and the uniform time step is $\Delta t = 0.25\ \Delta x$.

The exact solution to this dam break problem governed by the gravity wave-model equations was derived by Martins et al. [1]. The exact solution for the fluid depth is

$$h(x,t) = \begin{cases} h_1 & \text{if} \quad x \leq -c_1 t \\ h_3 = x^2/gt^2 & \text{if} \quad -c_1 t \leq x \leq -c_2 t \\ h_2 = c_2^2/g & \text{if} \quad -c_2 t \leq x \leq \xi t \\ h_0 & \text{if} \quad x \leq \xi t \end{cases} \tag{11}$$

The exact solution for the fluid velocity is

$$u(x,t) = \begin{cases} 0 & \text{if} \quad x \leq -c_1 t \\ u_3 = 2(c_1^3 t^3 + x^3)/3x^2 t & \text{if} \quad -c_1 t \leq x \leq -c_2 t \\ u_2 = 2/3\,[(c_1^3/c_2^2) - c_2] & \text{if} \quad -c_2 t \leq x \leq \xi t \\ 0 & \text{if} \quad x \leq \xi t \end{cases} \tag{12}$$

where c is the wave propagation speed. In addition,

$$c_1 = \sqrt{gh_1}\,, \qquad c_3 = -x/t\,, \qquad c_2^3 = c_1^3 + 3/2\,(c_0^2 - c_2^2)\sqrt{(c_0^2 + c_2^2)/2}\,, \qquad c_0 = \sqrt{gh_0}\,. \tag{13}$$

Furthermore, ξ is the shock speed given by

$$\xi = \sqrt{(c_0^2 + c_2^2)/2}\,. \tag{14}$$

A representative of numerical results is given in Figure 2. This figure shows a comparison between the numerical solution with the exact solution. The numerical solution is produced using the number of cells $N = 400$, the uniform cell width $\Delta x = 0.05$, and the uniform time step $\Delta t = 0.25\ \Delta x$. We notice that numerical solutions are very close to the exact solution. We also notice that the significant source of errors are at non-smooth regions.

Table 1. Errors and convergence rates of numerical solutions

N	Error of h	Error of u	Convergence rate for h	Convergence rate for u
100	0.1150	0.0422	-	-
200	0.7777	0.0280	0.68	0.66
400	0.0510	0.0185	0.66	0.66
800	0.0330	0.0121	0.65	0.65
1600	0.0205	0.0075	0.62	0.62

Results of those if we vary the number of cells are quantitatively written in Table 1. As the number of cells gets larger, the errors becomes smaller. Note that a large number of cells means a small cell width. We observe that numerical solutions are convergent to the exact solution, as the number of cells gets larger. We have used a first order finite volume method. This means that the convergence rate should be one if the solution is smooth. However, our solution to the dam break problem contains non-smooth regions. This makes the convergence rate is less than one, as shown in Table 1.

These findings could have implications in the development of flood flow simulations when the gravity wave-model equations are used. The proposed finite volume method based on the Lax-Friedrichs formulations are simple and fast to compute yet it is accurate with relatively small errors. The proposed method could be implemented as an alternative solver in case studies, such as that in the work of Martins et al. [2].

5. Conclusion

We have presented the Lax-Friedrichs finite volume method used to solve the gravity wave-model equations. The method is explicit, so it is simple to implement. The first order method leads to less than one for its convergence rate when we solve the dam break problem. As long as the time step is chosen sufficiently small such that the method is stable, the numerical solutions are convergent to the exact solution. Our research is limited to frictionless model of the gravity wave-model equations. Future research direction could be investigating the numerical solver for the gravity wave-model equations involving friction.

Acknowledgements
This work was financially supported by Sanata Dharma University. Both authors are very thankful to Sanata Dharma University for the financial support.

References

[1] Martins R, Leandro J and Djordjevic S 2016 Analytical solution of the classical dam-break problem for the gravity wave-model equations *Journal of Hydraulic Engineering* **142** 06016003
[2] Martins R, Leandro J and Djordjevic S 2018 Influence of sewer network models on urban flood damage assessment based on coupled 1D/2D models *Journal of Flood Risk Management* **11** S717
[3] Aronica G, Tucciarelli T and Nasello C 1998 2D multilevel model for flood wave propagation in flood-affected areas *Journal of Water Resources Planning and Management* **124** 210
[4] Bates P D, Horritt M S, Fewtrell T J 2010 A simple inertial formulation of the shallow water equations for efficient two-dimensional flood inundation modelling *Journal of Hydrology* **387** 33
[5] De Almeida G A M, Bates P, Freer J E and Souvignet M 2012 Improving the stability of a simple formulation of the shallow water equations for 2-D flood modeling *Water Resources Research* **48** W05528
[6] Seyoum S D, Vojinovic Z, Price R K and Weesakul S 2012 Coupled 1D and noninertia 2D

flood inundation model for simulation of urban flooding *Journal of Hydraulic Engineering* **138** 23
[7] Mangeney A, Heinrich P and Roche R 2000 Analytical solution for testing debris avalanche numerical models *Pure and Applied Geophysics* **157** 1081
[8] Mungkasi S and Dheno M F S 2017 Adomian decomposition method used to solve the gravity wave equations *AIP Conference Proceedings* **1788** 030103
[9] Mungkasi S, Supriyadi B and Wiryanto L H 2016 Steady flow over an arbitrary obstruction based on the gravity wave equations *Journal of Physics: Conference Series* **776** 012080
[10] LeVeque R J 1992 *Numerical Methods for Conservation Laws* (Basel: Springer)
[11] LeVeque R J 2002 *Finite Volume Methods for Hyperbolic Problems* (Cambridge: Cambridge University Press)