

Conservative formulation of unsteady pipe-flow model for water in its liquid form

Bambang Supriyadi¹, I Gusti Ketut Puja², F A Rusdi Sambada² and Sudi Mungkasi³

¹ Department of Civil and Environmental Engineering, Faculty of Engineering, Gadjah Mada University, Jalan Grafika No. 2, Yogyakarta 55281, Indonesia

² Department of Mechanical Engineering, Faculty of Science and Technology, Sanata Dharma University, Mrican, Tromol Pos 29, Yogyakarta 55002, Indonesia

³ Department of Mathematics, Faculty of Science and Technology, Sanata Dharma University, Mrican, Tromol Pos 29, Yogyakarta 55002, Indonesia

E-mail: bbsupri@tsipil.ugm.ac.id, ketut@usd.ac.id, sambada@usd.ac.id, sudi@usd.ac.id

Abstract. We consider flows in a pipe system. The problem relating to the pipe system usually occurs in real life as well as in the laboratory, such as in the distillation process. This unsteady problem can be modeled into a hyperbolic system of partial differential equations. Our goal is to propose the use of a conservative finite-volume numerical method for solving the unsteady pipe-flow model. We obtain that the method is able to solve the pipe-flow problem successfully.

1. Introduction

Pipe flows occur in daily life as well as in the laboratory. In the laboratory, pipe flows can be found in the water distillation process. Distillation itself includes two types of water forms, gas and liquid.

Mathematical models for pipe flows have been available in the literature [1]. In the gaseous form, Mungkasi [2] used a conservative finite-volume numerical method for solving the Euler equations of gas dynamics. In the liquid form, Škifić *et al* [3] proposed a nonconservative formulation of unsteady pipe-flow problems. The conservative formulation for water in its liquid form was not given in the work of Škifić *et al* [3].

This paper complements the work of Škifić *et al* [3]. We propose the use of a conservative formulation of unsteady pipe-flow model for water in its liquid form. Because the model is hyperbolic, we choose a finite-volume method as our conservative formulation to solve the model. Finite-volume methods have been well known as powerful methods to reconstruct smooth and nonsmooth functions as solutions to problems [4].

This paper is organized as follows. We recall the mathematical model governing the problem in Section 2. We write the finite-volume scheme to solve the model in Section 3. Numerical results are provided and discussed in Section 4. Some remarks conclude the paper in Section 5.



2. Mathematical model

In this section, we write the mathematical model and the general formulation for solving the model. We assume that the model is one-dimensional.

The mathematical model governing conservation laws of water flowing in a pipe [3] is

$$H_t + \frac{c^2}{gA} Q_x = 0, \quad (1)$$

$$Q_t + gAH_x = 0. \quad (2)$$

Here, the involved variables and parameters are as follows: t is the time variable, x is the space variable for coordinate along the conduit length, $H = H(x, t)$ denotes the piezometric head, $Q = Q(x, t)$ represents the discharge, A means the pipe cross-sectional area, c is the wave speed, and g is the acceleration due to gravity. In this paper, we assume that there is no head losses in the system.

Equations (1) and (2) can be written in the vector form of conservation laws

$$\mathbf{u}_t + \mathbf{f}(\mathbf{u})_x = \mathbf{0}, \quad (3)$$

where the vector of conserved quantities is

$$\mathbf{u} = \begin{pmatrix} H \\ Q \end{pmatrix} \quad (4)$$

and the vector of flux functions is

$$\mathbf{f}(\mathbf{u}) = \begin{pmatrix} \frac{c^2}{gA} Q \\ gAH \end{pmatrix}. \quad (5)$$

The Jacobian matrix of the vector of flux functions is defined as

$$\mathbf{A}(\mathbf{u}) = \frac{\partial \mathbf{f}}{\partial \mathbf{u}} = \begin{pmatrix} 0 & \frac{c^2}{gA} \\ gA & 0 \end{pmatrix}, \quad (6)$$

with eigenvalues and right eigenvalue vectors are

$$\lambda^{(1),(2)} = \mp c \quad (7)$$

and

$$\mathbf{r}^{(1),(2)} = \begin{pmatrix} \frac{\lambda^{(1),(2)}}{Ag} \\ 1 \end{pmatrix}. \quad (8)$$

Now, suppose that we have the scalar conservation law of (3), that is

$$u_t + f(u)_x = 0, \quad (9)$$

where u is a scalar (not a vector). The corresponding fully discrete finite-volume numerical scheme is

$$Q_i^{n+1} = Q_i^n - \frac{\Delta t}{\Delta x} (F_{i+1/2}^n - F_{i-1/2}^n), \quad (10)$$

where Δx is the cell width, and Δt is the time step in a finite-volume framework. Here, U_i^n is the finite-volume average approximation of the quantity at the i th cell at the n th time step, and $F_{i+1/2}^n$ is the finite-volume average approximation of the flux function at the $(i + 1/2)$ interface from the n th time step to the $(n + 1)$ th time step. The scalar finite-volume scheme (10) can be extended for the vector form (3) in order to solve (1) and (2). We discuss this in the next section.

3. Numerical method

As follows, we briefly describe the numerical method that we use to solve (1) and (2). The method is a finite-volume method, which is conservative.

Let us consider (3). Using the standard finite-volume framework, let Δx be the cell width of uniformly discretized space domain, and Δt be a given time step. In the semidiscrete form, the finite-volume method is

$$\frac{d}{dt}\mathbf{U}_j(t) = -\frac{1}{\Delta x}(\mathbf{F}_{j+\frac{1}{2}}(t) - \mathbf{F}_{j-\frac{1}{2}}(t)) \quad (11)$$

where \mathbf{U} is the vector of approximate conserved quantities and \mathbf{F} is the vector of approximate fluxes. This scheme is called semidiscrete because we have discretized the equations with respect to space, but the time variable is still continuous.

To evolve the conserved quantity, we need to integrate the semidiscrete form (11) with respect to time. We can use any standard method for solving ordinary differential equations (ODEs). As we have used a first-order method in space, we choose to use a first-order method in time. Note that a higher order method in time is not needed because we have used a first-order method in space. In this paper, we implement the first-order Runge-Kutta method (the Euler method) to integrate the semidiscrete form (11) with respect to time. The fully discrete finite-volume method is

$$\mathbf{U}_j^{n+1} = \mathbf{U}_j^n - \frac{\Delta t}{\Delta x}(\mathbf{F}_{j+\frac{1}{2}}^n - \mathbf{F}_{j-\frac{1}{2}}^n). \quad (12)$$

The flux is approximated using the Lax–Friedrichs formulation (see LeVeque [5] for more details):

$$\mathbf{F}_{j+1/2} = \frac{1}{2}(\mathbf{F}_j + \mathbf{F}_{j+1}) - \frac{\Delta x}{2\Delta t}(\mathbf{U}_{j+1} - \mathbf{U}_j). \quad (13)$$

Here we have dropped the superscript n for brevity of writing. Considering the Courant–Friedrichs–Lewy (CFL) condition, we do the numerical computation in such a way that the numerical method is stable.

4. Numerical results

In this section we present some numerical results to show that the numerical method works well for solving (1) and (2). All measured quantities are assumed to be in SI units with the MKS system.

Consider a circular pipe with $L = 1200$, $D = 2$, $c = 1000$, and $g = 9.81$. We assume to have the initial conditions

$$H(x, 0) = \begin{cases} 10.00, & \text{if } 0 < x \leq 1000, \\ 42.44, & \text{if } 1000 < x \leq 1200, \end{cases} \quad (14)$$

and

$$Q(x, 0) = \begin{cases} 1, & \text{if } 0 < x \leq 1000, \\ 0, & \text{if } 1000 < x \leq 1200. \end{cases} \quad (15)$$

The boundary conditions at $x = 0$ are

$$H(0, t) = 10, \quad Q(0, t) = 1, \quad (16)$$

and at $x = 1200$ are

$$H(1200, t) = 42.44, \quad Q(1200, t) = 0. \quad (17)$$

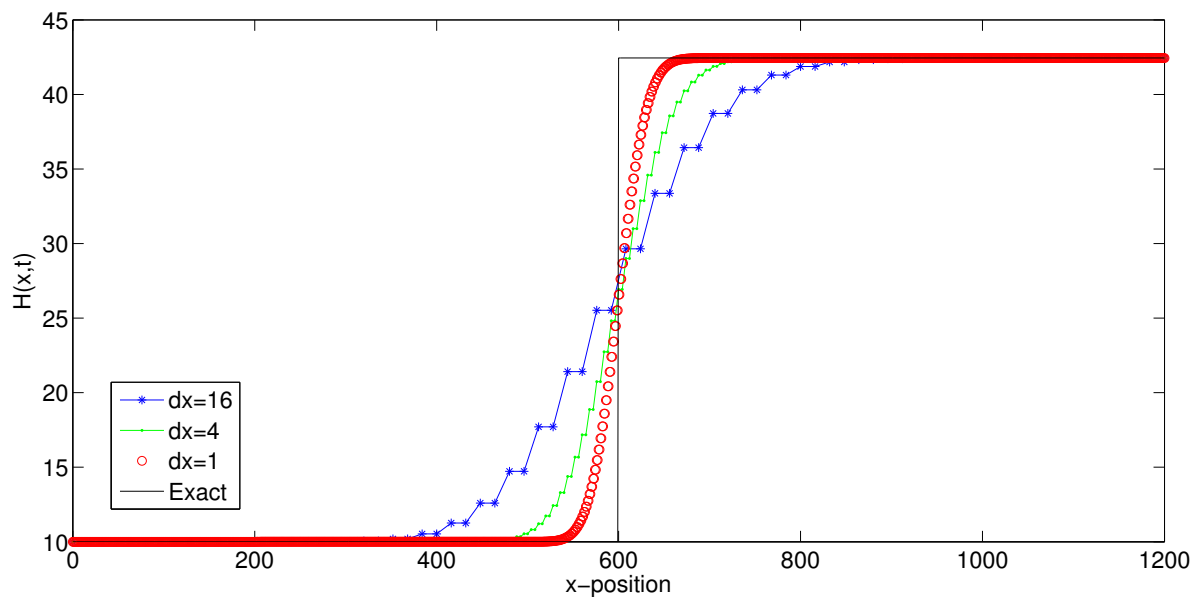


Figure 1. Results for varying cell width Δx at time $t = 0.4$ with dx meaning Δx . Here $\Delta t = \Delta x/2c$. The numerical solutions converge to the exact solution as Δx approaches zero.

With these settings, we simulate the problem to obtain numerical solutions at time $t = 0.4$ and analyze the results.

Based on simulation results, the presented numerical method is diffusive. However, the diffusion can be reduced by taking more computational cells, that is, taking a smaller cell width Δx , as shown in figure 1. (To generate this figure, we take $\Delta t = \Delta x/2c$ for our numerical simulations.) This could make the computation more expensive, but in fact it is not too expensive due to the nature of the first-order numerical method and the model. We observe that taking smaller Δx leads to more accurate numerical solution. Therefore, by numerical experiments our numerical method converges to the exact solution, as the cell width tends to zero. We observe that taking the time step $\Delta t > \Delta x/c$ results in an unstable method.

5. Conclusion

We have proposed the use of the conservative finite-volume method to solve the water flow problem in a pipe. The method is simple in structure, because it is explicit. The method is conditionally stable. Our results are limited to one-dimensional problems. Possible future direction may extend these results for multidimensional problems.

Acknowledgments

This research was financially supported by Sanata Dharma University and a research grant from *Direktorat Riset dan Pengabdian Masyarakat* (DRPM) of Ministry of Research, Technology and Higher Education of the Republic of Indonesia year 2016.

References

- [1] Chaudhry M H 1987 *Applied Hydraulic Transients* (New York: Van Nostrand Reinhold)
- [2] Mungkasi S 2016 *AIP Conf. Proc.* **1737** 040002
- [3] Škifić J, Maćešić S and Črnjarić-Žić N 2010 *J. Hydraul. Eng.* **136** 483

- [4] Mungkasi S 2016 *Adv. Math. Phys.* **2016** 7528625
- [5] LeVeque R J 2002 *Finite Volume Methods for Hyperbolic Problems* (Cambridge: Cambridge University Press)