A Finite Volume Method for Water Hammer Problems

Sudi Mungkasi[1,a,*], Andreas Prasetyadi[2,b], and F. A. R. Sambada[2,c]

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

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

[a]sudi@usd.ac.id, [b]pras@usd.ac.id, [c]sambada@usd.ac.id

*Corresponding author

Keywords: central-upwind flux, finite volume method, pipe flow, water hammer problem

Abstract. Water hammer problems occur in pipe systems. The pressure in pipe flow can be very excessive that makes the system to be broken. One way to minimize the breaking is by conducting simulations so that we know the characteristics of the system. However, numerical simulations need a good numerical solver of the problem. In this paper, we propose a finite volume numerical method to solve water hammer problems. The method is very accurate, gives a sharp resolution at discontinuity, yet simple to implement in the computation.

Introduction

Water hammer problems occur in daily life, such as at water taps. Solutions to water hammer problems are useful, as they can be used to predict if pipe and/or valve breaks for a certain pressure hitting the valve.

A number of authors have attempted to solve water hammer problems. Some of them are Markendahl [1] and Mungkasi, *et al.* [2]. They studied a particular water hammer problem using the linear acoustic equations. The theory of these acoustic equations is available in the work of LeVeque [3]. However shock waves in the solutions of Markendahl [1] and Mungkasi, *et al.* [2] might not be resolved sharply.

In the present paper we propose a finite volume method with the central-upwind flux formulation to solve water hammer problems. The central-upwind flux was developed by Kurganov, *et al.* [4] for hyperbolic conservation laws and Hamilton-Jacobi equations. With a special choice of the time stepping, we are able to simulate the shock wave sharply, so accurate solution is obtained.

Governing Equations

Water hammer problems are governed by the acoustic equations [1]

$$p_t + \rho c^2 u_x = 0, \tag{1}$$

$$u_t + \frac{1}{\rho} p_x = 0. \tag{2}$$

Here x is the space variable, t is the time variable. The notation $p(x,t)$ represents the pressure, $u(x,t)$ the velocity, ρ the density, and c the propagation speed of pressure wave.

Let us review briefly the water hammer problem considered by Markendahl [1] and Mungkasi, *et al.* [2]. We are given a one-dimensional space $[0, L]$. Suppose that water flows from a tank to a valve, as shown in Figure 1. The position $x = 0$ is the joint between the tank and the pipe. The position $x = L$ is the location of the valve on the right end. The boundary condition at $x = 0$ is $p(0,t) = p_{\text{tank}}$, where p_{tank} is the tank pressure on the left end. The boundary condition at $x = L$ satisfies

$$\frac{1}{2}\rho u(L,t)^2 = \alpha(t)^2[p(L,t) - p_{\text{right}}], \tag{3}$$

where p_{right} is the environment pressure on the right end. The variable α is the opening coefficient of the valve and is defined as

$$\alpha(t) = 1 - \frac{t - t_{\text{start}}}{t_{\text{stop}}}. \tag{4}$$

The loss coefficient $K = 1/\alpha^2$ satisfies $p - p_{\text{right}} = \frac{1}{2}K\rho u^2$. The initial condition is a constant pipe pressure with a constant velocity. In this model, the effect of pipe bending is assumed to be negligible and frictions to the pipe wall is neglected.

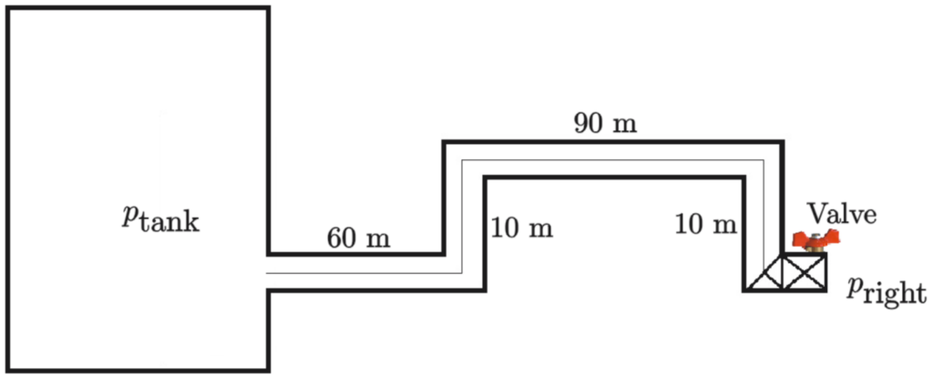


Figure 1: Illustration of a pipe system (see Markendahl [1] and Mungkasi, *et al.* [2]).

Numerical Method

A finite volume method is used to solve the acoustic equations (Eq. 1 and Eq. 2) in order to find the solution to the problem. Let us consider the following vectors

$$\mathbf{q}(x,t) = \begin{pmatrix} p \\ u \end{pmatrix}, \quad \mathbf{f}(\mathbf{q}(x,t)) = \begin{pmatrix} \rho c^2 u \\ \frac{1}{\rho} p \end{pmatrix}. \tag{5}$$

The acoustic equations can be written as

$$\mathbf{q}_t + \mathbf{f}(\mathbf{q})_x = 0, \tag{6}$$

which is a system of hyperbolic conservation laws [3].

We discretize Eq. 6 using the finite volume method with uniform space step Δx and uniform time step Δt. The space $[0, L]$ is discretized into $C_i = [x_{i-1/2}, x_{i+1/2}]$, $i = 1, 2, 3, \ldots, N$, where N is the total number of cells. The time is discretized as $t^n = n\Delta t$, $n = 0, 2, 3, \ldots$. The finite volume scheme for Eq. 6 is

$$\mathbf{Q}_i^{n+1} = \mathbf{Q}_i^n - \frac{\Delta t}{\Delta x}(\mathbf{F}_{i+1/2}^n - \mathbf{F}_{i-1/2}^n). \tag{7}$$

Here the numerical quantity $\mathbf{Q}_i^n$ is the averaged quantity over the i th cell at time t^n. The numerical flux $\mathbf{F}_{i-1/2}^n$ is the averaged flux flowing through the interface $x_{i-1/2}$ during one time step. In this paper, we use the central-upwind formulation to compute the flux $\mathbf{F}_{i-1/2}^n$. We refer to the work of Kurganov, *et al.* [4] for this flux formulation. Therefore, for non-boundary fluxes we obtain

$$\mathbf{F}_{i+1/2} = \frac{1}{2}(\mathbf{F}_i + \mathbf{F}_{i+1}) - \frac{c}{2}(\mathbf{Q}_{i+1} - \mathbf{Q}_i). \tag{8}$$

The boundary condition is numerically treated as follows [1, 2]. At the left end ($x_{1/2} = 0$),

$$\mathbf{F}_{1/2} = \begin{pmatrix} \rho c^2[u_1 + \frac{1}{\rho c}(p_{\text{tank}} - p_1)] \\ \frac{1}{\rho} p_{\text{tank}} \end{pmatrix}. \tag{9}$$

for every time step. At the right end ($x_{N+1/2} = L$),

$$\mathbf{F}_{N+1/2} = \begin{pmatrix} \rho c^2 u_{N+1/2} \\ \frac{1}{\rho} p_{N+1/2} \end{pmatrix}. \tag{10}$$

The velocity at the right end of the space is

$$u_{N+1/2} = -\frac{c}{K} + \frac{c}{K}\sqrt{1 + \frac{2K}{\rho c^2}(p_N - p_{\text{right}} + \rho c u_N)}\,. \tag{11}$$

The pressure at the right end of the space is

$$p_{N+1/2} = p_{\text{right}} + \frac{1}{2} K \rho u_{N+1/2}^2\,. \tag{12}$$

After the valve is completely closed, the velocity at the right end is zero. This means that for $t \ge t_{\text{stop}}$ we need to enforce $u_{N+1/2} = 0$. Therefore for $t \ge t_{\text{stop}}$, we take

$$\mathbf{F}_{N+1/2} = \begin{pmatrix} 0 \\ \frac{1}{\rho} p_N + c u_N \end{pmatrix}. \tag{13}$$

Numerical Results

We present our results of the water hammer problem with the model shown in Figure 1. If units of quantities are omitted, they should be assumed to have SI units. The space domain is $[0, 170]$, where the value 170 is the total of three segments 60, 10, 90 and 10 m.

We take $p_{\text{tank}} = 10^5$ Pa and $p_{\text{right}} = 0$. The propagation speed of pressure wave is $c = 1.5\times 10^3$ m/s. The water density is $\rho = 10^3$ kg/m^3. The initial pressure in the pipe is $p(x,0) = p_{\text{tank}}$. The initial velocity of water flowing in the pipe is $u(x,0) = 8$ m/s. The valve closes with $t_{\text{stop}} = 0.1$ s. The domain is discretized into $N = 10^3$ uniform cells. The pressure and velocity hitting the valve are illustrated in Figures 2-5. In these figures the computational domain is $[0, 170]$, but we extend the plots to be $[0, 180]$ to make the figures clearer at around the valve. For any time $t < t_{\text{stop}}$ it is important to take the time step to be $\Delta t = \Delta x / c$ in order to get an accurate solution, especially at around the shock position.

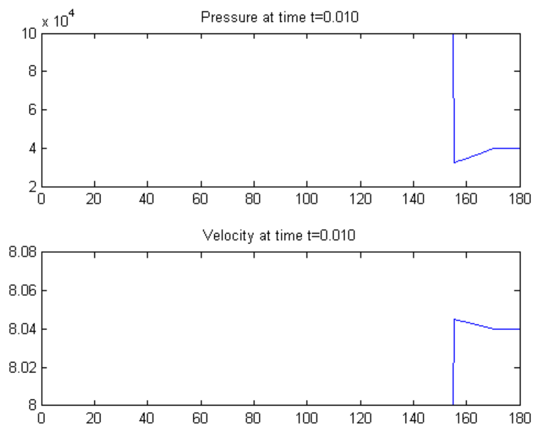


Figure 2: The pressure and velocity at $t = 0.01$.

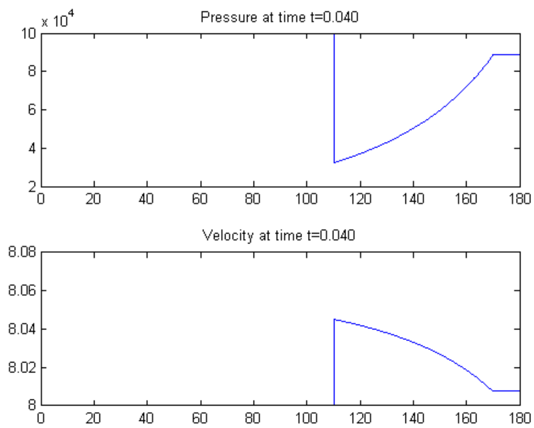
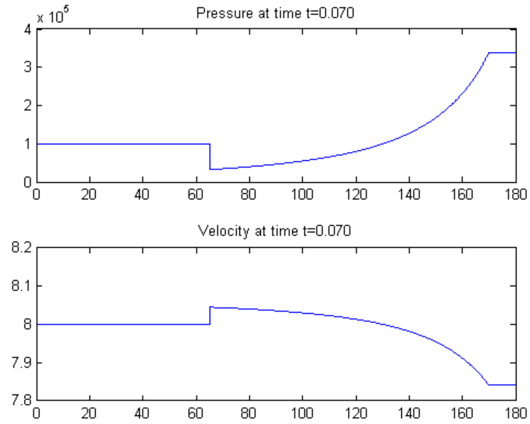


Figure 3: The pressure and velocity at $t = 0.04$.



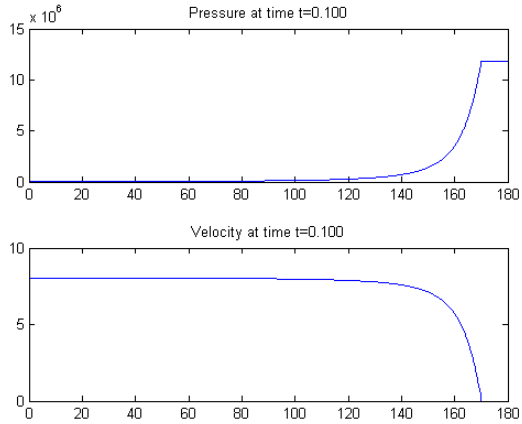


Figure 4: The pressure and velocity at $t = 0.07$.

Figure 5: The pressure and velocity at $t = 0.1$.

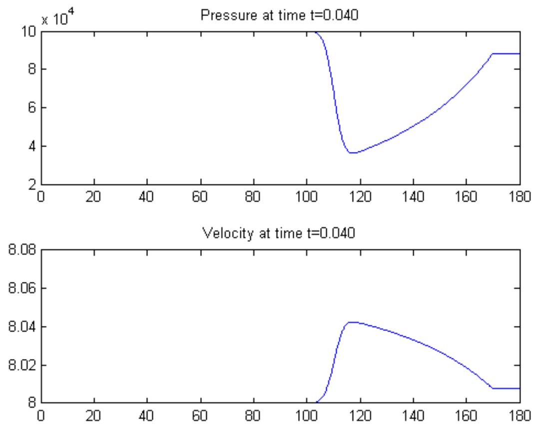


Figure 6: The pressure and velocity at $t = 0.04$ with a non-optimal time step; shock is not sharp.

From Figure 5, we obtain that the pressure hitting the valve at time $t = t_{\text{stop}} = 0.1$ is $p = 1.184 \times 10^7$ Pa, which is more than 100 time the initial pressure. Note that the higher the initial velocity, the higher the pressure hits the valve.

To confirm the accuracy of our proposed numerical strategy, we plot the results if we take the time step to be $\Delta t = 0.0001 \cdot \Delta x$. This time step is not optimal in terms of accuracy. The results is plotted in Figure 6. In this figure we almost do not see any shock, as the shock is smeared out. This can be compared with Figure 3, which shows the results for the same time $t = 0.04$.

Summary

A finite volume method has been proposed for solving water hammer problems. The method uses the central-upwind flux formulation. Shock wave can be sharply simulated using the proposed method. Future research could be conducted for higher dimensional water hammer problems.

Acknowledgements. This research was financially supported by DIKTI Research Grant in the Ministry of Research, Technology and Higher Education of The Republic of Indonesia.

References

[1] A. Markendahl: *Finite volume simulation of fast transients in a pipe system*, Uppsala University, Sweden (2010).

[2] S. Mungkasi, A. Prasetyadi and F.A.R. Sambada: *numerical solutions to fast transient pipe flow problems*, to appear in Advanced Materials Research (2015).

[3] R.J. LeVeque: *Finite volume methods for hyperbolic problems*, Cambridge Univ. Press (2002).

[4] A. Kurganov, S. Noelle and G. Petrova: *Semidiscrete central-upwind schemes for hyperbolic conservation laws and Hamilton-Jacobi equations*, SIAM Journal on Scientific Computing, Vol. 23, pp. 707–740 (2001).