

# **GODUNOV-TYPE SOLUTIONS FOR TWO-PHASE WATER HAMMER FLOWS**

ARTURO S. LEON

*Dept. of Civil and Envir. Engng., Univ. of Illinois at Urbana-Champaign, 2519  
Hydrosystems Lab., MC-250. 205 North Mathews Av., Urbana, Illinois 61801, USA.*

MOHAMED S. GHIDAoui

*Dept. of Civil Engng., The Hong Kong University of Science and Technology, Room  
3569, Clear Water Bay, Kowloon, Hong Kong.*

ARTHUR R. SCHMIDT

*Dept. of Civil and Envir. Engng., Univ. of Illinois at Urbana-Champaign, 2535a  
Hydrosystems Lab., MC-250. 205 North Mathews Av., Urbana, Illinois 61801, USA.*

MARCELO H. GARCIA

*Dept. of Civil and Envir. Engng., Univ. of Illinois at Urbana-Champaign, 2535b  
Hydrosystems Lab., MC-250. 205 North Mathews Av., Urbana, Illinois 61801, USA.*

**Abstract.** This paper focuses on the formulation and assessment of a second-order accurate Finite Volume (FV) shock-capturing scheme for simulating two-phase water hammer flows. The two-phase flow model is based on the single-equivalent fluid concept. The accuracy and numerical efficiency of the proposed scheme is compared to the fixed-grid MOC scheme with space-line interpolation. The results show that the proposed scheme is much more efficient than the MOC scheme.

## **1. Introduction**

The study of two-phase water hammer flows has great significance in a wide range of industrial and municipal applications including power plants, petroleum industries, water distribution systems, etc. For Real-Time Control (RTC) of these systems, the numerical efficiency of transient flow models is a critical factor, since several simulations are required within a control loop in order to optimize the control strategy, and small simulation time steps are needed to reproduce the rapidly varying hydraulics (e.g., [1]). RTC is becoming increasingly indispensable for industrial and municipal applications in general. For instance, in the case of water distribution systems, RTC facilitates delivery of safe, clean and high-quality water in the most expedient and economical manner.

The partial differential equations that describe two-phase flows in closed conduits can be simplified to a great extent when the amount of gas in the conduit is small. In this case, the gas-liquid mixture can be treated as a single-equivalent fluid (e.g., [2], [3]). The governing equations when using the single-equivalent fluid approximation are identical to those for a one-phase flow. Due to this fact, similar techniques to those for a one-phase flow are used to solve the two-phase flow governing equations that are based on the single-equivalent fluid concept. However, since shocks may be produced during transient conditions in two-phase flows (e.g., [4]), only those methods that can handle shocks without special treatment are suitable for these applications.

Numerical schemes that have been proposed for modeling one-dimensional two-phase flows using the single-equivalent fluid approximation include MOC schemes, Lax-Wendroff schemes, a plethora of explicit schemes, and implicit methods (e.g., [2]). The MOC scheme requires isolation of shocks. The Lax-Wendroff scheme has the advantage that shock waves can be handled without special treatment (e.g., [2]). However, the solution produces an overshooting of the shock front, followed by damped oscillations. When using implicit methods, biasing or weighting problems may be encountered (e.g., [2]). Recently, [3] and [5] has applied GTS schemes to two-phase flows with good success. The first-order GTS presented by [3] showed that numerical diffusion leads to a very fast degradation of the solution quality after a few oscillation periods. The second-order scheme by [5] is largely superior to his first-order scheme, although an iterative process is required to solve the Riemann problem.

The present paper focuses on the formulation and numerical efficiency assessment of a second-order accurate FV shock-capturing scheme for simulating two-phase water hammer flows. In the proposed approach, no iteration is required for the solution of the Riemann problem. This paper is organized as follows: (1) the governing equations are presented in conservation-law form; (2) the corresponding FV discretization is described; (3) a Riemann solver for the flux computation at the cell interfaces is provided; and (4) results from testing the proposed model are presented.

## 2. Governing Equations

The mass and momentum conservation equations for the “single-equivalent fluid” assumptions can be written in their vector conservative form (e.g., [2]):

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = \mathbf{S} \quad (1)$$

where the vector variable  $\mathbf{U}$ , the flux vector  $\mathbf{F}$  and the source term vector  $\mathbf{S}$  may be written as:

$$\mathbf{U} = \begin{bmatrix} \Omega \\ Q_m \end{bmatrix}, \mathbf{F} = \begin{bmatrix} Q_m \\ \frac{Q_m^2}{\Omega} + A_f p \end{bmatrix} \text{ and } \mathbf{S} = \begin{bmatrix} 0 \\ (S_0 - S_f)\rho_f g A_f \end{bmatrix} \quad (2)$$

where  $\rho_f$  is the fluid density,  $A_f$  is the full cross-sectional area of the conduit,  $\Omega = \rho_f A_f$  is the mass of fluid per unit length of conduit,  $Q_m = \Omega u$  is the mass discharge,  $u$  is the water velocity,  $p$  is the pressure acting on the center of gravity of  $A_f$ ,  $g$  is the gravitational acceleration,  $S_0$  is the slope of the conduit, and  $S_f$  is the slope of the energy line. The vector Eq. (1) does not form a closed system in that the flow state is described using three variables:  $\Omega$ ,  $p$  and  $Q_m$ . However, it is possible to eliminate the pressure variable by introducing the general definition of the celerity of the pressure wave ( $a_g$ ) [e.g., [3]], which relates  $p$  and  $\Omega$ :

$$a_g = \left[ \frac{d(A_f p)}{d\Omega} \right]^{1/2} \quad (3)$$

The pressure-wave celerity for the gas-liquid mixture ( $a_m$ ) can be estimated as ([3]):

$$a_m = \frac{a}{\sqrt{1 + \psi_{ref} \rho_f \rho_{ref} a^2 \frac{p_{ref}^{\frac{1}{\beta}}}{p^{\frac{1+\beta}{\beta}}}}} \quad (4)$$

where  $a$  is the pressure-wave celerity in presence of liquid only,  $p_{ref}$  is a reference pressure for which the density is known ( $\rho_{f,ref}$ ),  $\beta$  is a coefficient equal to 1.0 for isothermal processes and 1.4 for adiabatic conditions, and  $\psi_{ref}$  is the volume fraction of gas at the reference pressure. The water density measured at a temperature of 4 degrees Celsius under atmospheric pressure conditions is 1000 kg/m<sup>3</sup>. Thus, the reference density and pressure when the liquid is water can be taken as 1000 kg/m<sup>3</sup> and 101,325 Pa, respectively.

The relationship between the volume fraction of gas  $\psi$  and pressure for the ‘‘single-equivalent fluid’’ assumptions can be expressed as (e.g., [3]):

$$p\psi^\beta = p_{ref}\psi_{ref}^\beta \quad (5)$$

Substituting Eq. (4) into Eq. (3) and integrating the differentials  $d\Omega$  and  $dp$  ( $A_f$  is assumed to be constant) leads to the following equation that relates  $p$  and  $\Omega$ :

$$\Omega = \Omega_{ref} + \frac{A_f}{a^2} \left[ p - p_{ref} + \left( p_{ref}^{-\frac{1}{\beta}} - p^{-\frac{1}{\beta}} \right) \beta \psi_{ref} \rho_f \rho_{ref} a^2 p_{ref}^{\frac{1}{\beta}} \right] \quad (6)$$

where  $\Omega_{ref} = \rho_{f,ref} A_f$ . The pressure  $p$  in Eq. (6) can be determined by an iterative scheme such as the Newton Raphson method and typically between three and five iterations are needed to ensure convergence. The flow variables used in this paper are  $\Omega$  and  $Q_m$ . However, the engineering community prefers to use the

piezometric head  $h$  and flow discharge  $Q$ . The latter variables can be determined from  $\Omega$  and  $Q_m$  as follows:

$$Q = \frac{Q_m}{\Omega} A_f \quad (7) \quad h = \frac{p - p_{ref}}{\rho_{fref} g} + \frac{d}{2} \quad (8)$$

where  $d$  is the pipe diameter and  $h$  is measured over the conduit bottom. The absolute pressure head ( $H$ ) in meters of water can be obtained as  $H = h + 10.33$ .

### 3. Formulation of Finite Volume Godunov-type schemes

This method is based on writing the governing equations in integral form over an elementary control volume or cell, hence the general term of Finite Volume (FV) method. The computational grid or cell involves discretization of the spatial domain  $x$  into cells of length  $\Delta x_i$  and the temporal domain  $t$  into intervals of duration  $\Delta t$ . The  $i$ th cell is centered at node  $i$  and extends from  $i-1/2$  to  $i+1/2$ . The flow variables ( $A$  and  $Q$ ) are defined at the cell centers  $i$  and represent their average value within each cell. Fluxes, on the other hand are evaluated at the interfaces between cells ( $i-1/2$  and  $i+1/2$ ). For the  $i$ th cell, the updating FV formula for the left side of Eq. (1) is given by (e.g., [6]):

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \frac{\Delta t}{\Delta x_i} (\mathbf{F}_{i+1/2}^{n+1/2} - \mathbf{F}_{i-1/2}^{n+1/2}) \quad (9)$$

where the superscripts  $n$  and  $n+1$  reflect the  $t$  and  $t+\Delta t$  time levels respectively. In Eq. (9), the determination of  $\mathbf{U}$  at the new time step  $n+1$  requires computation of the numerical flux at the cell interfaces at the old time  $n$ . To introduce the source terms [right side of Eq. (1)] into the solution, a time splitting method using a second-order Runge-Kutta discretization is used (e.g., [7]). In the Godunov approach, the flux is obtained by solving the Riemann problem with constant states  $\mathbf{U}_i$  and  $\mathbf{U}_{i+1}$ . This leads to first-order accuracy of the numerical solution. To achieve second-order accuracy in space and time, the MUSCL - Hancock method (e.g., [6]) was used in this paper. Second or higher order schemes are prone to spurious oscillations in the vicinity of discontinuities. To preserve the accuracy of the solution away from discontinuities, while ensuring that the solution is oscillation-free near shock waves and other sharp flow features, the MINMOD pre-processing slope limiter (see [6]) was used.

### 4. Riemann solver

In contrast with one-phase water hammer flows, in two-phase flows the pressure-wave celerity may be reduced to very low values, in which case  $u$  is not necessarily negligible compared to  $a_m$ . However,  $u$  is still smaller than  $a_m$  and consequently the characteristics travel in opposite directions, and the star

region (\*), which is an intermediate region between the left and right states, contains the location of the initial discontinuity. Hence, the flow variables in the star region are used to compute the flux. Simple estimates for  $\Omega_*$  and  $Q_{m*}$  that do not require iterations can be obtained by solving the Riemann problem for the linearized hyperbolic system  $\partial U/\partial t + \bar{A} \partial U/\partial x = 0$  where  $\bar{A}$  is the Jacobian matrix of the flux vector. This yields:

$$\Omega_* = \left( \frac{\Omega_L + \Omega_R}{2} \right) \left( 1 + \frac{u_L - u_R}{2\bar{a}_m} \right) \quad (10) \quad Q_{m*} = Q_{mL} + (\bar{u} - \bar{a}_m)(\Omega_* - \Omega_L) \quad (11)$$

where  $\bar{a}_m = (a_{mL} + a_{mR})/2$  and  $\bar{u} = (u_L + u_R)/2$ . By using the estimated values of  $\Omega_*$  and  $Q_{m*}$ , the flux is obtained from Eq. (2).

## 5. Evaluation of the model

The proposed approach is valid for pipes with and without friction. In the considered test case, a frictionless pipe is used only because in such case the physical dissipation is zero, so any dissipation or amplification in the results is solely due to the numerical scheme. The CPU times that are reported in this paper were averaged over three realizations and computed using a HP AMD Athlon (tm) 64 processor 3200 + 997 MHz, 512 MB of Ram notebook.

The test compares the accuracy and numerical efficiency of the proposed scheme against the fixed-grid MOC scheme with space-line interpolation. The two-phase homogeneous mathematical model presented in [2] is solved using the MOC scheme. In this case, if shocks are present, the Rankine-Hugoniot conditions are enforced across the shock. The test considers one horizontal frictionless pipe connected to an upstream reservoir and a downstream valve. The length of the pipe is 10,000 m and its diameter is 1 m, the pressure-wave celerity in presence of water only is 1000 m/s, the upstream reservoir constant head  $h_0$  is 200 m, and the initial steady-state discharge is 2.0 m<sup>3</sup>/s. The void ratio at the reference pressure (101,325 Pa) is assumed to be 0.002 (0.2%). The transient flow is obtained after an instantaneous closure of the downstream valve.

The instantaneous closure of the downstream valve results in the formation of a shock wave at the downstream end of the pipe. This wave propagates upstream until on reflection against the left boundary, it becomes a rarefaction wave. Figure 1 shows the pressure profiles at  $t = 140$  s computed using the MOC scheme and the proposed approach assuming isothermal conditions ( $\beta = 1$ ) for two different number of grids. The ‘‘Near exact’’ profile is also presented in this figure. In this test case, all the simulations were carried out using a maximum Courant number of 0.95 to avoid numerical instability problems. It

should be noted that in two phase flows, the air content and pressure wave celerity are continuously changing [Eqs. (4) and (5)]. When using an explicit scheme (as used here) for simulating these flows, it is possible to exceed  $Cr = 1.0$  if a  $Cr$  close to 1.0 is specified at the beginning of the time step. As shown in Figure 1, for the same number of grids and maximum Courant number, the timing and magnitude of the shock wave simulated by the proposed scheme is in better agreement with the “Near exact” solution than the MOC scheme.

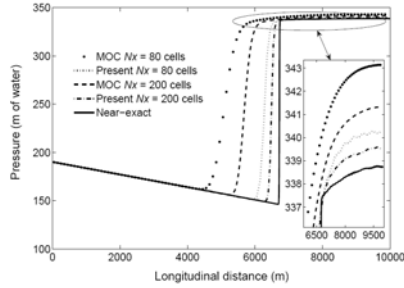


Figure 1 Pressure head versus longitudinal distance ( $\beta = 1$ ,  $t = 140$  s and  $Cr_{max} = 0.95$ ).

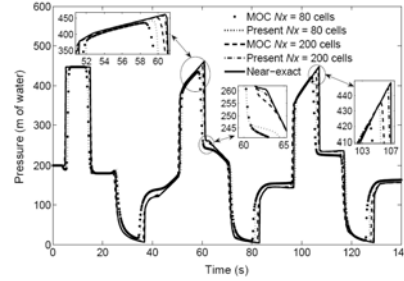


Figure 2 Pressure traces at the middle of the pipe ( $\beta = 1$  and  $Cr_{max} = 0.95$ ).

Figure 2 displays the simulated pressure traces at the middle of the pipe for two different numbers of grids. As can be observed in this figure, the MOC is more dissipative than the proposed scheme. The results presented in Figs. 1 and 2 show that, for the same discretization, the proposed scheme is more accurate than the MOC scheme. A more conclusive comparison requires measurement of the CPU time needed by each scheme to achieve a given level of accuracy. The accuracy of a scheme can be measured using the following error norm (e.g., [8]):

$$ABSERROR = \frac{\sum_{i=1}^{Nx} |e_i|}{\sum_{i=1}^{Nx} |\phi_i^{exact}|} \quad (12)$$

where  $e_i = \phi_i^{numerical} - \phi_i^{exact}$  = difference between the numerical and exact solution at node  $i$ ,  $\phi$  = dependent variable such as the pressure head or flow velocity, ABSERROR = absolute error, and  $Nx$  = number of grids. The absolute error is a measure of the difference between the numerical and exact solution for either the pressure head or flow velocity.

Figure 3 shows the plot of the absolute error for the pressure head against the number of grids on log-log scale. As shown in this figure, to achieve a given level of accuracy, the MOC scheme requires a finer grid than the proposed

scheme, or, for the same number of grids, the proposed scheme is more accurate than the MOC scheme. For comparison of CPU times, four levels of absolute error were selected (0.4% - 10%). The number of grids needed by each of the schemes to achieve the four absolute error levels, were obtained from Figure 3. These numbers of grids in turn were used to compute the CPU times, which results are shown in Figure 4. The CPU time results show that the proposed scheme is about 3 to 130 times faster to execute than the MOC scheme for the same level of accuracy. The numerical efficiency of the proposed scheme compared to the MOC approach increases as the absolute error decreases.

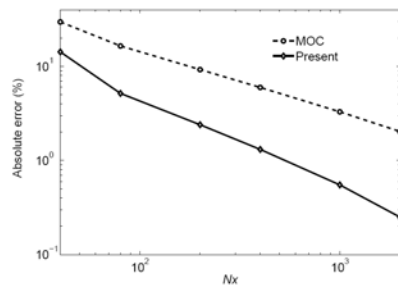


Figure 3 Absolute error for the pressure head versus number of grids ( $\beta = 1$ ,  $t = 140$  s and  $C_{\max} = 0.95$ ).

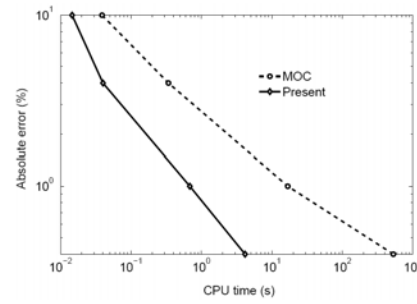


Figure 4 Absolute error for the pressure head versus CPU time ( $\beta = 1$ ,  $t = 140$  s and  $C_{\max} = 0.95$ ).

## References

1. A. S. León, M. S. Ghidaoui, A. R. Schmidt, M. H. García, *Proc., XXXI IAHR Congress*, Seoul, Korea, (2005).
2. C. S. Martin, *Proc., Computer modeling of free-surf. and press. flows*, NATO ASI Series E, **274**, Pullman, WA, U.S.A., 519 (1993).
3. V. Guinot, *Int. J. Numer. Meth. in Eng.*, **50**, 1169 (2001a).
4. M. Padmanabhan, C. S. Martin, *Proc., Int. J. Multiphase Flow*, **4**, 81 (1978).
5. V. Guinot, *Int. J. Numer. Meth. Fluids*, **37**, 341 (2001b).
6. E. F. Toro, *Shock-capturing methods for free-surface shallow flows*, Wiley, LTD, Chichester, U.K. (2001).
7. A. S. León, M. S. Ghidaoui, A. R. Schmidt, M. H. García, *J. Hydraul. Eng.*, **132**(8), 800 (2006).
8. J. H. Liang, M. S. Ghidaoui, J. Q. Deng, W. G. Gray, *J. Hydraul. Research*, Submitted for review (2007).