A FULLY CONSERVATIVE MODEL FOR COMPRESSIBLE TWO-FLUID FLOW

Jeroen Wackers and Barry Koren

CWI
P.O. Box 94079, 1090 GB Amsterdam, The Netherlands
e-mail: Jeroen.Wackers@cwi.nl, Barry.Koren@cwi.nl

Key words: Compressible two-fluid flow, interface capturing, conservation.

Abstract. *A five-equation model for compressible two-fluid flow is proposed, that is based on physical flow equations only. The model is conservative and pressure-oscillation free. Equations for continuous flow and jump conditions for discontinuities are given, as well as a discretisation of the equations and an adaptation of the HLL Riemann solver to two-fluid flow. Numerical tests in 1D and 2D show the accuracy of the method.*

1 INTRODUCTION

Interface-capturing methods for compressible two-fluid flows do not explicitly track the interface between the two fluids, but assume that the flow consists of a mixture of the fluids everywhere. The interface between the fluids appears as a numerically smeared transition from fluid 1 to fluid 2. Most interface-capturing methods are based on bulk-fluid conservation laws, combined with a transport equation for a mixture parameter (like the volume fraction of one fluid) to distinguish between the fluids. This parameter is used in a mixed-fluid equation of state. Abgrall and Karni [1] have shown that many conservative formulations of such models produce large pressure errors. This problem can be solved by using locally non-conservative methods [1] or by solving the full two-phase flow model [2].

Here, an intermediate approach is presented. The current method is fully conservative and pressure-oscillation free, but it is simpler than two-phase methods, because it has a single pressure and velocity for the two fluids. It does not use a mixture-parameter transport equation, instead it is based on physical flow equations and single-fluid equations of state only. The method is an extension of the work by Van Brummelen and Koren [3], it will be described in detail in a future paper. A similar method is derived in a different way by Kapila et al. [4].

The present method has two major advantages. First, the conservative formulation gives good capturing of shocks and interfaces, also for problems with very strong shocks. And second, the model strongly resembles a single-fluid model: it does not require a complex interface-tracking algorithm. It can thus be solved with existing techniques, even on complex, irregular grids.

2 FLOW MODEL

The physical model used here for two-fluid flow is based on a mixture model. However, the fluids are not fully mixed: the 'mixture' may be thought to consist of very small bits of the two pure fluids, in an arbitrary pattern. Each fluid has its own pure-fluid equation of state and the fluids interact only by exerting forces on each other.

In the model, the pressure and the velocity of the fluids are equal, but each fluid has its own density. The volume fraction of fluid 1, α, is used to denote the relative amounts of the two fluids. Thus, in 1D, we have five independent state variables (p, u, ρ_1, ρ_2 and α), so we need five differential equations to solve the flow.

The bulk two-fluid flow satisfies the standard Euler equations:

$$(\rho)_t + (\rho u)_x = 0, \tag{1a}$$

$$(\rho u)_t + (\rho u^2 + p)_x = 0, \tag{1b}$$

$$(\rho E)_t + (\rho E u + p u)_x = 0. \tag{1c}$$

In these equations, the bulk density ρ and bulk total energy E are

$$\rho = \alpha \rho_1 + (1 - \alpha)\rho_2, \qquad \rho E = \alpha \rho_1 E_1 + (1 - \alpha)\rho_2 E_2, \tag{2}$$

2

with the total energy for each fluid $j = 1,\, 2$ defined as

$$E_j = e_j + \tfrac{1}{2}u^2. \tag{3}$$

Here e_j is the internal energy of fluid j.

Two more flow equations are needed to close the system. The first one is the conservation of mass for fluid 1: the fluids are not supposed to change into each other. Using the partial density $\rho_1 \alpha$, this equation is

$$(\rho_1 \alpha)_t + (\rho_1 u \alpha)_x = 0. \tag{4a}$$

Together with equation (1a), this equation gives mass conservation for both fluids. For the last equation, the energy balance of fluid 1 is used. This equation has a special property: the fluids exert forces on each other, so they exchange energy. This exchange appears as a source term in the equation:

$$(\rho_1 E_1 \alpha)_t + (\rho_1 E_1 u \alpha + p u \alpha)_x = S. \tag{4b}$$

An expression for this source term is derived in the next section.

To close the system, equations of state (EOS) are needed for the two fluids. A possible EOS is the ideal gas law,

$$p = (\gamma_1 - 1)\rho_1 e_1 = (\gamma_2 - 1)\rho_2 e_2, \tag{5}$$

with constant γ's. For this equation, it is easy to compute the primitive variables p and α from the total energies:

$$p = (\gamma_1 - 1)\left((\rho_1 E_1 \alpha) - \tfrac{1}{2}(\rho_1 \alpha)u^2\right) + (\gamma_2 - 1)\left((\rho E) - (\rho_1 E_1 \alpha) - \tfrac{1}{2}(\rho - (\rho_1 \alpha))u^2\right), \tag{6}$$

and

$$\alpha = \frac{(\gamma_1 - 1)\left((\rho_1 E_1 \alpha) - \tfrac{1}{2}(\rho_1 \alpha)u^2\right)}{p}. \tag{7}$$

3   THE SOURCE TERM

The source term, that appeared in the previous section, is described here for smooth and for discontinuous flow.

3.1   Derivation of the source term

The source term S in equation (4b) models the exchange of energy between fluid 2 and fluid 1. Euler flow has no heat conduction, so the only energy exchanged is the work done by the force between the fluids. This force is found from a momentum analysis.

Consider a small section of a 1D shock tube (see figure 1). This section contains fluid 1 and fluid 2 (the interface is drawn schematically). The force on the entire fluid element

is $p(x) - p(x + dx)$ and its bulk mass is ρdx. The force on fluid 1 in the section is $(p\alpha)(x) - (p\alpha)(x + dx) + S_M dx$. Its mass is $\rho_1 \alpha dx$ and its acceleration is *equal* to the acceleration of the entire element (because both fluids have the same velocity). Therefore

$$\frac{p(x) - p(x + dx)}{\rho dx} = \frac{(p\alpha)(x) - (p\alpha)(x + dx) + S_M dx}{\rho_1 \alpha dx},$$

The force S_M follows from this expression (using the mass fraction $\beta = \frac{\rho_1 \alpha}{\rho}$).

$$S_M = p\alpha_x + (\alpha - \beta)\, p_x. \tag{8}$$

The first term, $p\alpha_x$, expresses the pressure force on the interface: pressure times the projected height of the interface. The second term is friction. There is no friction in normal Euler flow, but the two fluids are mixed so well that friction force *between the fluids* prevents one fluid from moving faster than the other. When fluid 1 is denser than fluid 2 $(\beta > \alpha)$, then it is accelerated less by the pressure force $-p_x$ than the lighter fluid 2. In that case, the friction force passes a part of the pressure force on fluid 2 to fluid 1, such that the velocity of the two fluids remains the same. The energy source term S is the work done by the force S_M:

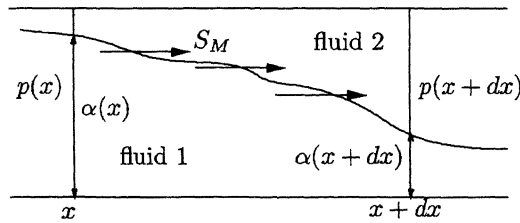$$S = uS_M = pu\alpha_x + (\alpha - \beta)\, up_x. \tag{9}$$

Figure 1: Flow element in 1D smooth flow.

3.2 Analysis of the system for ideal gas

The source term (9) is valid for any EOS. Substitution of the ideal gas law (5) allows a characteristic analysis of the flow equations. This results in five wave speeds,

$$\lambda_1 = u - c, \quad \lambda_{2,3,4} = u, \quad \lambda_5 = u + c, \quad \text{with } c = \sqrt{(\gamma_1 \alpha + \gamma_2(1 - \alpha))p/\rho}. \tag{10}$$

This combination of wave speeds is physically correct. It can be proved that (9) is the only possible source term that gives such a combination.

The analysis shows that the mass fraction β satisfies a transport equation:

$$\beta_t + u\beta_x = 0. \tag{11}$$

4

This is expected, because the two fluids do not move with respect to each other. But the fluid is compressible, so the volume fraction α is not convected:

$$\alpha_t + u\alpha_x = \alpha(1 - \alpha)(\gamma_1 - \gamma_2)u_x. \qquad (12)$$

This equation shows that α is only convected in pure-fluid regions ($\alpha = 0$ or $\alpha = 1$), for iso-atomic fluids ($\gamma_1 = \gamma_2$), or for pure interface flow ($u_x = 0$).

3.3 Source term in discontinuities

To allow weak solutions with discontinuities of the two-fluid flow equations, we need a proper definition of the flow across a discontinuity. The first four equations, (1a) – (1c) and (4a), satisfy the standard Rankine-Hugoniot jump condition $\Delta \mathbf{f} = c_s \Delta \mathbf{q}$, with c_s the speed of the discontinuity. For the fifth equation, (4b), this condition becomes

$$\Delta \mathbf{f} = c_s \Delta \mathbf{q} + \int_{x_L}^{x_R} S dx. \qquad (13)$$

The integral must be evaluated across the discontinuity, which is impossible. However, if we assume that the discontinuity is the limit of a viscous discontinuity and thus has a continuous internal structure (the precise shape is unimportant), then we can write the state variables as continuous functions of p and integrate the source term:

$$\int_{x_L}^{x_R} S dx = \Delta(pu\alpha) + \tfrac{1}{2}\beta_L\rho_L(u_L - c_s)\Delta u^2 + \frac{1}{\rho_L(u_L - c_s)} \int_{p_L}^{p_R} p\alpha dp. \qquad (14)$$

A derivation of this expression will be given in a future paper. The last integral can be evaluated by integration of the fifth equation, (4b), combined with an EOS. So there is a unique jump condition for the present two-fluid model, but, unlike the single-fluid jump condition, it depends on the material properties of the fluids.

4 NUMERICAL METHOD

4.1 Second-order accurate discretisation

The flow equations are discretised with a second-order accurate finite-volume scheme. Fluxes are computed with an improved version of Linde's three-wave HLL approximate Riemann solver [5], combined with a limited reconstruction of the cell interface states. The limiter is applied to the primitive variables ρ, u, p, α and β. Time stepping is done with a two-step scheme [7]:

$$\begin{aligned}
\mathbf{q}_i^{k+1} &= \mathbf{q}_i^k - \tfrac{\Delta t}{\Delta x}\left(\mathbf{f}_{i+\frac{1}{2}}^k - \mathbf{f}_{i-\frac{1}{2}}^k\right) + \tfrac{\Delta t}{\Delta x}\mathbf{s}_i^k, \\
\mathbf{q}_i^{k+2} &= \mathbf{q}_i^k - 2\tfrac{\Delta t}{\Delta x}\left(\mathbf{f}_{i+\frac{1}{2}}^{k+1} - \mathbf{f}_{i-\frac{1}{2}}^{k+1}\right) + 2\tfrac{\Delta t}{\Delta x}\mathbf{s}_i^{k+1}.
\end{aligned} \qquad (15)$$

4.2 Numerical source term

A discretisation of the source term is needed in two places. First, an approximation of the source term in a discontinuity is needed to compute the HLL flux. The HLL solver models a Riemann problem with three discontinuous waves. The easiest way to incorporate the source term in these waves is to compute only one approximate solution of equation (14), using the left and right cell interface state, and to divide this source term proportionally over the three waves. This procedure causes some small inaccuracies, but it is fast and straightforward.

Secondly, the source term for the time integration, s_i in equation (15), is computed. It consists of two parts: (i) Sources in the discontinuities at the cell faces. These are summed over all HLL waves, on interfaces $i - \frac{1}{2}$ and $i + \frac{1}{2}$, that actually run into cell i (figure 2a). (ii) Sources in the continuous flow in the cell. These are integrated over the piecewise linear approximations to the primitive variables, that follow from the use of the limiter (figure 2b).
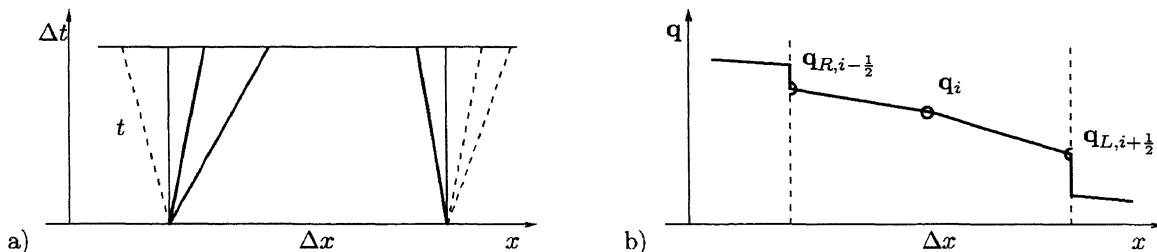

Figure 2: Source term in a cell: from cell faces (a) and internal flow (b).

5 NUMERICAL RESULTS

5.1 Shock tube tests

The method is tested first on two 1D Riemann problems for ideal gases. Results are compared with the exact solutions. The first test is a contact discontinuity (constant pressure and velocity) of water-air density ratio. The solution in figure 3 shows the smeared interface, exactly in the correct location. And, most importantly, the pressure is constant: no pressure oscillations occur.

The second test is a two-fluid variation of Sod's problem, with a ten times higher left pressure and density, giving it a pressure ratio of 1:100. Figure 4 shows that the discontinuities (shock and two-fluid interface) are in the proper locations. Note that the pressure is constant over the contact discontinuity and that the volume fraction is constant over the shock and over the expansion fan. A convergence study for this particular problem, on five grids, shows that the L^1-errors in ρ, u and p converge approximately with the power 0.96 of the mesh width. The volume fraction converges with the power
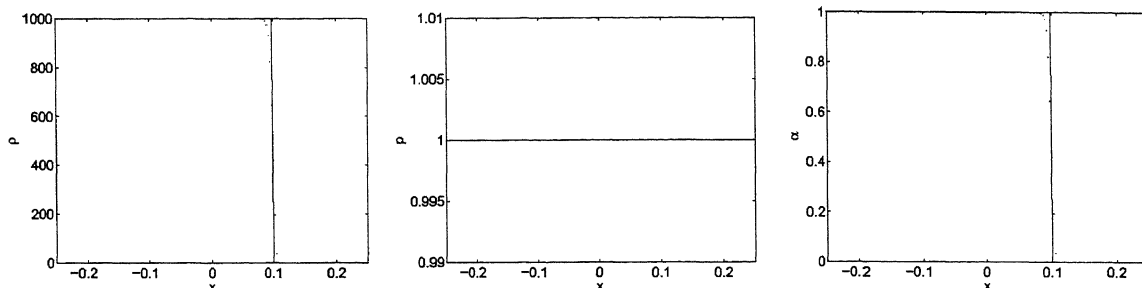
6

Figure 3: 1D contact discontinuity. $p = 1$, $u = 1$, $\rho_L = 1000$, $\rho_R = 1$, $\gamma_L = 1.4$ and $\gamma_R = 1.6$. The grid has 200 cells, 80 time steps and $\Delta t/\Delta x = 0.5$ (CFL $= 0.5$). Solid lines: exact solution.

0.78 of the mesh width. This rate of convergence is comparable to that for single-fluid solutions with limited second-order schemes.
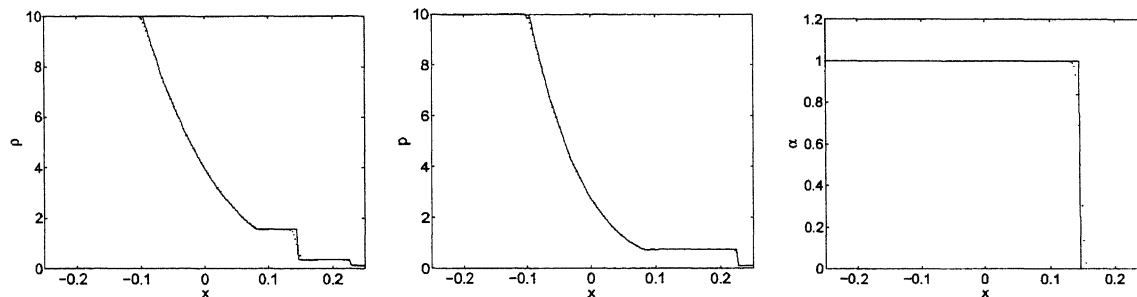

Figure 4: High-pressure, two-fluid Sod problem. $(\rho, u, p)_L = (10, 0, 10)$, $(\rho, u, p)_R = (0.125, 0, 0.1)$, $\gamma_L = 1.4$ and $\gamma_R = 1.6$. The grid has 200 cells, 160 time steps, $\Delta t/\Delta x = 0.2$ (CFL $= 0.56$). Solid lines: exact solution.

5.2 Shock hitting helium bubble

This 2D test case has been taken from literature [6]. It consists of a cylindrical helium bubble in air, which is hit by an incoming shock wave. The problem is solved on a grid of 200×400 cells, with $\Delta t = 1.25 \times 10^{-5}$. Figure 5 shows the solution at two times. The (half) bubble is visible between $x = -0.025$ and $x = 0.025$. The incident shock, coming from the right, can be seen in the air above the bubble, the curved shock in the bubble runs ahead of this shock. The rightmost wave is an expansion wave, reflected into the air behind the shock. At the later time, a complicated λ-shock structure has developed above the bubble. Figure 6 shows the pressure and the volume fraction for this time. Of the waves appearing in the density plot, the shock waves and expansions are visible in the pressure plot only and the interface in the volume fraction plot only, as it should be. The pressure is continuous over the interface.

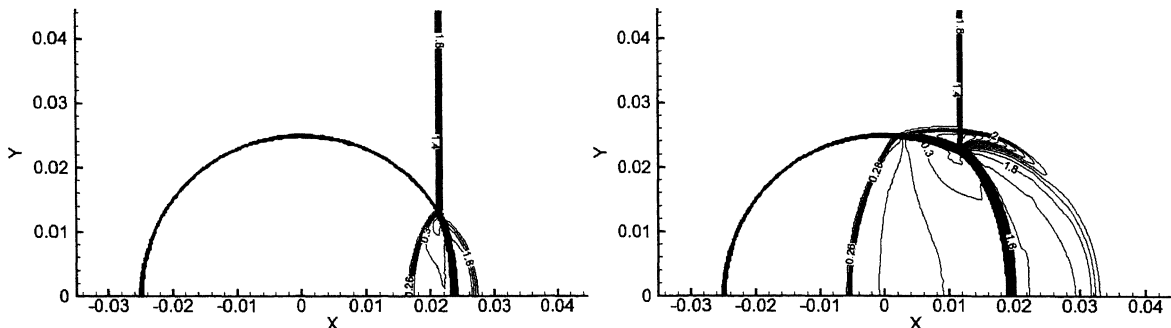The speeds of the shocks and the interface at the centerline ($y = 0$) are compared in

Figure 5: Shock hitting helium bubble, density at $t = 2.74$ ms and $t = 10.74$ ms.
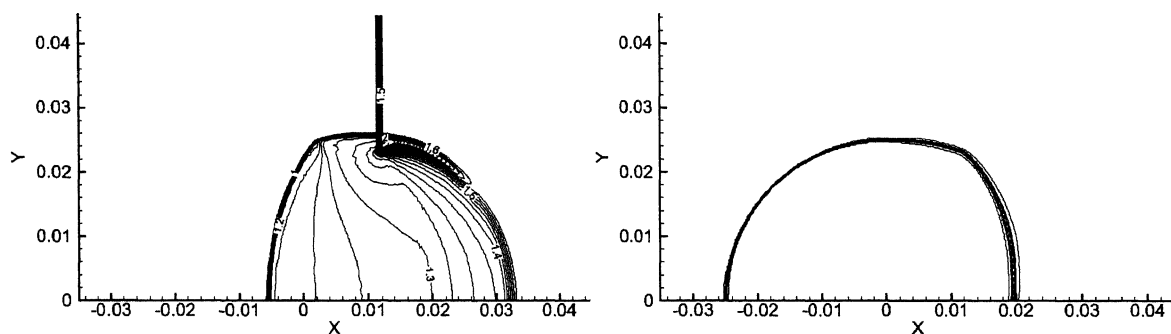

Figure 6: Shock hitting helium bubble, pressure (left) and volume fraction (right) at $t = 10.74$ ms.

table 1 with results from Quirk and Karni [6] (obtained on a very fine, adapted grid). The difference is between 0.7% and 2.8%.

	c_s (m/s)	c_r (m/s)	u_{ri} (m/s)
Quirk & Karni	422	943	178
Present method	419	950	173
Difference (%)	0.7	0.8	2.8

Table 1: Centerline wave speeds in helium bubble test: comparison of computation by Quirk and Karni [6] with present method. c_s: incident shock speed, c_r: shock speed in bubble, u_{ri}: right side of interface speed.

6 CONCLUSION

A model for compressible two-fluid flow is proposed, that is conservative and pressure-oscillation free. 1D tests show that the model resolves contact discontinuities without creating pressure errors and that it accurately handles problems with strong shocks. A 2D test confirms that the method also handles curved shocks and interfaces well.

REFERENCES

[1] R. Abgrall and S. Karni. Computations of Compressible Multifluids. *J. Comp. Phys.*, **169**, 594–623, 2001.

[2] R. Abgrall and R. Saurel. Discrete equations for physical and numerical compressible multiphase mixtures. *J. Comp. Phys.*, **186**, 361–396, 2003.

[3] E.H. van Brummelen and B. Koren. A pressure-invariant conservative Godunov-type method for barotropic two-fluid flows. *J. Comp. Phys.*, **185**, 289–308, 2003.

[4] A.K. Kapila, J.B. Bdzil, R. Menikoff, S.F. Son, and D.S. Steward. Two-phase modelling of DDT in granular materials: reduced equations. *Physics of Fluids*, **13**, 3002–3024, 2001.

[5] T. Linde. A practical, general-purpose, two-state HLL Riemann solver for hyperbolic conservation laws. *Int. J. Numer. Meth. Fluids*, **40**, 391–402, 2002.

[6] J.J. Quirk and S. Karni. On the dynamics of a shock-bubble interaction. ICASE Report 94-75. NASA Langley Research Center, Hampton, VA, 1994.

[7] J. Wackers and B. Koren. A simple and efficient space-time adaptive grid technique for unsteady compressible flows. AIAA Paper 2003-3825. American Institute of Aeronautics and Astronautics, 2003.