Development of high resolution methods for analyzing multi-fluid problems

M P Ray¹, B P Puranik² and U V Bhandarkar³

¹ MET’s Institute of Technology, Bhujbal Knowledge City, Adgaon, Nashik 422003, (affiliated to S P Pune University, Pune)
²,³ Mechanical Engineering Department, Indian Institute of Technology Bombay, Powai, Mumbai 400076

E-mail: ¹ milindray@gmail.com, ² puranik@iitb.ac.in, ³ bhandarkar@iitb.ac.in

Abstract. High resolution multi-fluid solvers are developed, using a fifth order weighted essentially non-oscillatory approach for spatial reconstruction and a third order Runge-Kutta scheme for temporal integration. Several flux evaluation schemes, comprising of exact and approximate Riemann solvers available in the literature are incorporated. Multi-fluid simulation capability is achieved using the ghost fluid method. Two-dimensional geometry is considered, and the Euler equations of gas dynamics are used as the basic flow model in view of the fact that the viscosity effects are negligible for most of the time of interest. Relative performance of these solvers is evaluated for several one- and two-dimensional test cases, before simulating the shock-bubble interaction. Three Mach numbers and air-Helium gas combination is studied and the results are presented in this study. Numerical schlieren images are rendered for qualitative study while time evolution of several integral features such as the axial and lateral deformations, the translational velocity of the bubble, the circulation in the flow-field, and the stretching rates of the bubbles are analyzed for quantitative studies. The results are consistent with the strength of baroclinic source term.

1. Introduction

When a shock moving in a stationary medium interacts with a stationary corrugated density interface separating two fluids, it gets diffracted, reflected and refracted in the initially separated fluids. This leads to change in the geometry of the shock and also the thermodynamic properties of the fluids. The fluids that are separated initially, mix due to stretching of the interface, which is caused due to the vortical motions which in turn are a result of the above mentioned interaction. This phenomenon of generation of vortices, stretching of the interface along with simultaneous modifications of the shock wave front, pose a challenging problem to capture numerically. This and similar interactions are seen in several technological and natural phenomena[1, 2].

To solve such complex Shock-bubble interaction (SBI) problems, a finite volume reconstruction-evolution methodology is adopted for developing high-resolution schemes. A fifth order spatial reconstruction is obtained by incorporating weighted essentially non-oscillatory (WENO5) approach, while a third order temporal integration is obtained using a Runge-Kutta scheme (RK3) [3]. The problem under consideration requires handling of two different fluids by the same
solver. One of the most common techniques to handle such two fluid problems is the Level Set Method. In this work, the multi-fluid problem is solved using the ghost fluid method (GFM) [4], that uses the level set function for keeping track of the interface correctly. The cell fluxes are obtained using Riemann solvers. In this work nine high-resolution schemes are developed using nine different Riemann solvers available in the literature. These include exact Riemann solvers, namely Godunov’s exact solver (GODU) and Flux Vector Splitting (FVS) schemes by van Leer (vL) and Steger-Warming (SW). The remaining are approximate Riemann solvers namely, Two Shock Riemann Solver (TSRS), Harten-Lax-van Leer contact (HLLC) solver, Roe-Pike Method (PIKE), Roe Solver with entropy fix (ROEF), Local Lax-Friedrichs (LLF) solver and an Adaptive Riemann Solver Non-iterative type (ASRS) [5]. These schemes are validated for tests from the literature before the SBI is attempted.

2. Scheme for Multi-material Simulation

Euler equations in Cartesian co-ordinates for a two-dimensional unsteady fluid flow are often written as follows.

\[ Q_t + F(Q)_x + G(Q)_{xx} = 0, \]

where \( Q = [\rho, \rho u_1, \rho u_2, E]^T \) are the conserved variables in vector form. These variables are derived from the primitive variables \( q = [\rho, u_1, u_2, P]^T \). The variables have their standard meaning i.e. \( P \) and \( \rho \) represent pressure and density respectively, while \( u_1, u_2 \) represent the velocity components. \( E \) represents the total energy equal to \( \frac{1}{2}(u_1^2 + u_2^2) + P/\rho \times (\gamma - 1) \) where \( \gamma \) is the specific heats ratio of the fluid. Fluxes of conserved variables in the \( x_1 \) and \( x_2 \) directions are \( F(Q) = [\rho u_1, \rho u_1 u_2 + P, \rho u_1, u_1(E + P)]^T \) and \( G(Q) = [\rho u_2, \rho u_2 u_1, \rho u_2^2 + P, u_2(E + P)]^T \) respectively. The subscripts \( t, x_1 \) and \( x_2 \) in Equation (1) represent derivatives with respect to time, \( x_1 \) and \( x_2 \) directions respectively.

The time integration procedure used involves a three step Runge-Kutta scheme and follows the method given in [6], known as TVD-RK3. For obtaining fifth order spatial accuracy, a procedure (WENO5) in [3] is employed. The inter-cell fluxes are calculated by using one of the Riemann solvers. Several one-dimensional and two-dimensional test cases from literature (for a single fluid), are employed for benchmarking of the present schemes. In this work, ten one-dimensional (1-D) and twenty-two two-dimensional (2-D) tests are used for assessment of all the high resolution single fluid solvers.

In multi-material methods tracking the interface correctly is of utmost importance. GFM does this efficiently at a lesser computational cost. In GFM, the interface is tracked by defining the level set function, as a normal distance function [7]. The interface is always allotted and maintained at zero value of level set function, and fluid 1 with positive value of the level set function, across the interface.

The GFM requires solving the Euler equations twice: once for the fluid 1 and then for the fluid 2 again, for the entire domain. Therefore every cell centre has an allocation of values for two primitive variables: a real value of variables for the real fluid (fluid 1) existing at the point, and a ghost value of variables for fluid 2.

From the above it is seen that the entire procedure for solving a multi-fluid problem consists of (a) Advection of the Level-set function, (b) Extrapolation of variables, (c) Reconstruction of variables, (d) flux evaluation for solving the Riemann problem, (e) First-time-only re-distance step and Re-initialization, (f) time integration.

The solvers are benchmarked with six one-dimensional validation tests [4]. A particular case is
Figure 1. Results from the exact solver compared to those from various schemes for Test 2A, in the form of distributions for: (a) Pressure, (b) Density, (c) Velocity, (d) Entropy.

presented in Fig. 1. From the results, it can be concluded that for the present set of conditions, all the solvers predict correct and consistent results. It is also found that the FVS solvers are slightly diffusive while the approximate Riemann solvers ASRS, HLLC and PIKE predict more accurate results. HLLC appears to be the most robust of all.

3. Benchmarking simulations

Two different gas-combinations are chosen for benchmarking of the present set of multi-material solvers. Each problem has air as the ambient medium with a cylindrical bubble placed in the medium, while a planar shock interacts with the bubble. In the first case, density of the bubble (He) is less than that of air and in the other case, density of the bubble (R22) is more. Both these simulations have experimental [8] and numerical results [9] for a shock with strength $M = 1.22$. For visualization, a numerical technique is used, based on the formulation by Quirk and
Karni [9]. Apart from a Mach number of 1.22, two other Mach numbers (2.0, 3.0) are also used for quantitative comparison.

For both benchmarking cases, experimental results of Haas and Sturtevant [8] compare well with the corresponding results of the present simulation. Similarly, they compare well with the numerical predictions by Marquina and Mulet [10], Quirk and Karni [9], and Shyue [11]. Figure 2 depicts comparison of results for a left moving shock (M = 1.22), interacting with a R22 bubble. While, Fig. 3 depicts comparison of results for the left moving shock (M = 1.22), interacting with a He bubble. Apart from the qualitative comparison, quantitative comparison is made for various parameters such as velocities of different waves generated during the interactions, flow-field circulation and pressure measurement at several points in the shock tube. A particular case for the flow field circulation is presented in Fig. 4. From both the results presented, it is seen that the qualitative and quantitative results compare well with the experimental results as well as the simulation results. The differences observed are due to the Adaptive Mesh Refinement technique used in the earlier simulations.

4. Quantitative Analysis
For quantitative comparison the parameters considered are the axial and lateral deformation of the bubble, flow-field circulation, bubble velocities and the stretching rates etc. These results are presented using suitably defined non-dimensional quantities. Figure 5 presents the quantitative data for the air-He combination.

4.1. Bubble Deformation
Figures 5(a)-(b) present the axial and lateral deformation data respectively. The results are presented for every Mach number separately. From the results, it is concluded that as the Mach number increases, the higher compression increases the axial deformation, whereas there is little effect on the lateral compression. These conclusions are consistent with those reported in the experimental work reported by Layes et al. [12], for a spherical bubble.

4.2. Circulation
Figure 5(c) presents the circulation data for air-He combination. The results are presented for shock strengths of Mach 1.22, 2.0 and 3.0. The circulation value in the flow-field presented in the study, is non-dimensionalised and compared, as a function of non-dimensionalised time for all the shock strengths. It is observed that, at the instance when the travel across the bubble is completed by the faster traveling shock, the circulation in the flow-field attains the peak value. Thus, from the simulation results it can be concluded that increase in the shock strength, results in increase in the baroclinic source term and the flow field circulation.

4.3. Bubble Velocities
The predictions from the model proposed in [13] for bubble velocities are compared with the velocity predictions of the present simulations. The results are not presented here. However, it can be clearly stated that the behaviour is consistent with how a bubble undergoes deformation as a function of the Mach number.

4.4. Stretching rates
As mentioned earlier, the interaction starts with no vorticity in the domain. But the subsequent modifications lead to generation of vorticity in the flow field. Later, the interaction dominated by the vorticity results into stretching of the bubble interface. Stretching of the bubble interface is a necessary condition that eventually leads to mixing of the species separated initially. It
Figure 2. Benchmarking results for air-R22 case. M = 1.22, at different times. (a) Haas & Sturtevant [8], (b) Quirk & Karni [9], (c) Shyue [11] and (d) HLLC solver. Time for; 1st row = $115 \times 10^{-6}$s, 2nd row = $187 \times 10^{-6}$s, 3rd row = $417 \times 10^{-6}$s and 4th row = $1020 \times 10^{-6}$s.

is also considered as one of the important parameter for the SBI problem [14]. Stretching of the bubble interface results into intensification of gradients, required for increasing the rate of diffusion of the species. From the results depicted in Fig. 5(d), it is concluded that higher the Mach number, higher is the vorticity, and ultimately the stretching rates.

5. Conclusions
From the results for the present set of conditions, it is concluded that, all the solvers predict correct and consistent results. Also, it is concluded that among the solvers developed in the work, the Exact solvers, GODU and FVS schemes predict diffusive interface. While, the approximate Riemann solvers viz. HLLC, ASRS, PIKE and TSRS interpret accurate interface and shock. Among the approximate solvers, HLLC appears to be the most robust of all.
The parametric investigation performed in the present work indicates that the parameters considered for the study behave consistently with the vorticity generated in the flow field, i.e. as the Mach number increases, the bubble deformation especially the axial deformation increases
Figure 3. Benchmarking results for air-He case. M = 1.22, at different times. (a) Haas & Sturtevant [8], (b) Quirk & Karni [9], (c) Marquina & Mulet [10] and (d) HLLC solver. Time for: 1st row = 52×10^{-6}s, 2nd row = 102×10^{-6}s, 3rd row = 245×10^{-6}s and 4th row = 674×10^{-6}s.

but little effect is seen on the lateral deformation. Other integral features such as the bubble velocities and the flow-field circulation, also increase with increase in the shock strength. The stretching of the bubbles, seen as the prerequisite of mixing (expressed using the specific stretching rates) also show similar trends.

References
[1] M Brouillette. The Richtmyer-Meshkov Instability. Annu. Rev. Fluid Mech., 34:445–468, 2002. https://doi.org/10.1146/annurev.fluid.34.09101.162238.
[2] D Ranjan, J Oakley, and R Bonazza. Shock-bubble interactions. Annual Review Fluid Mech, 43:117–40, 2011. https://doi.org/10.1146/annurev-fluid-122109-164744.
[3] C W Shu. Essentially non-oscillatory and weighted essentially non-oscillatory schemes for hyperbolic conservation laws. ICASE-1997-65, 1997. https://doi.org/.
[4] R P Fedkiw, A Marquina, and B Merriman. An isobaric fix for the overheating problem in multi-material compressible flows. J. Comput. Phys., 148:545–578, 1999. https://doi.org/.
Figure 4. Flow-field circulation comparison; Present simulation predictions with Quirk and Karni [9] (QK predictions). (a) air-He case, (b) air-R22 case.

[5] E F Toro. Riemann Solvers and Numerical Methods for Fluid Dynamics : A Practical Approach. Springer Verlag, Berlin, 1992.
[6] S Osher and C W Shu. High-order essentially non-oscillatory schemes for Hamilton-Jacobi equations. SIAM J. Numer. Anal., 28:907–922, 1991. https://doi.org/.
[7] W Mulder, S Osher, and Sethian J A. Computing interface motion in compressible gas dynamics. J. Comput. Phys., 100:209–228, 1992. https://doi.org/10.1016/0021-9991(92)90229-R.
[8] J F Haas and B Sturtevant. Interactions of Weak Shock Waves with cylindrical and spherical gas inhomogeneities. J. Fluid Mech., 181:41–76, 1987. https://doi.org/10.1017/s0022112087002003.
[9] J J Quirk and S Karni. On the dynamics of shock-bubble interaction. J. Fluid Mech., 318:129–63, 1996. https://doi.org/10.1017/s0022112096007069.
[10] A Marquina and P Mulet. A flux-split algorithm applied to conservative models for multicomponent compressible flows. J. Comput. Phys., 185:120–138, 2003. https://doi.org/10.1016/s0021-9991(02)00050-5.
[11] K.-M. Shyue. A wave-propagation based volume tracking method for compressible multi-component flow in two space dimensions. J. Comput. Phys., 215:219–244, 2006. https://doi.org/.
[12] G Layes, G Jourdon, and L Houas. Experimental study on a plane shock wave accelerating a gas bubble. Phys of Fluids 074102, 21:1–14, 2009. https://doi.org/.
[13] G Rudinger and L M Somers. Behaviour of small regions of different gases carried in accelerated gas flows. J. Fluid Mech., 7:161–76, 1960. https://doi.org/10.1017/s0022112060001419.
[14] J Yang, T Kubota, and E E Zukoski. Applications of shock-induced mixing to supersonic combustion. AIAA J, 31:854–62, 1993. https://doi.org/10.2514/3.11696.
Figure 5. Comparison of quantitative parameters for air-He combination for different Mach numbers. (a) Axial Deformation, (b) Lateral Deformation, (c) Circulation, (d) Stretching Rates.