The interaction of the shock wave with the bubble and the effect of computational grid size on the problem simulation with a fully coupled pressure-based algorithm

Mohammad Pirani, Ariya Rahmani, MohammadReza Ansari*

Faculty of Mechanical Engineering, Tarbiat Modares University, Tehran, Iran

ABSTRACT

When a shock wave propagates through a flow field that has nonlinear thermodynamic properties, different processes occur simultaneously. Wave compression, wave refraction, and vortex generation are examples of these processes that cause the waveform and thermodynamic properties of the fluid to change. The interaction of a shock wave with a cylindrical bubble is an example of a wave-bubble collision problem in which all of the above processes are observed. Due to the high computational cost of density-based algorithms in solving compressible interfacial flow problems such as shock wave interaction with the two-phase flow, using a fully coupled pressure-based algorithm is a good solution that will solve the problem with proper accuracy while reducing computation time. In this paper, using this algorithm, the interaction of the shock wave with the bubble is investigated; while validating the results, the effect of the computational grid size and the method of discretization of the governing equations are determined. It was observed that by increasing the number of computational grids according to the first-order upwind method, the simulation results become more accurate, and the numerical diffusion amount decreases. Also, by changing the discretization method to second-order upwind, the instabilities on the interface of the two phases increase due to spurious fluctuations, and the shape of the interface obtained from the numerical solution moves away from the experimental results.

KEYWORDS

Two-Phase Flow, Compressible Flow, Pressure-Based Algorithm, Shock Wave, Bubbly Flow

---

* Corresponding author's email: mra_1330@modares.ac.ir

1. Introduction

When a shock wave impacts to inhomogeneity flow field, various phenomena such as wave refraction and reflection, vortex generation, and turbulence affect the physics of the problem. These interactions cause complex patterns of shock waves to emerge in the flow field [1]. Shock-accelerated inhomogeneous flows are used in many scientific, industrial, and medical applications. Supersonic combustion systems [2], Shredding kidney stones [3], and the use of high-energy systems such as inertial confinement fusion devices [4] are examples of the application of shock wave interaction with inhomogeneous compressible bubbles. Most simulations of shock wave interaction with Compressible Interfacial Flows have been done by density-based algorithms.[5,6] In density-based algorithms, the governing equations of the problem are solved for mass, momentum, and total energy, and for determining the flux, especially for Interfacial Flows, an exact or approximate Riemann Solver is usually used.[7] While density-based algorithms are naturally suitable for Compressible flows but in small Mach, the dependence of the density value on the pressure is low, and using this algorithm is not recommended.[8] Pressure-based algorithms are used less than other algorithms for simulating compressible flows because of the pressure correction obtained from the continuity equation.[9] But at the lower Mach number, the relationship between velocity and pressure is more vital than density and pressure, so the use of pressure-based algorithms has provided a suitable answer for a wide range of Mach numbers while maintaining the stability of the problem.[10] In this research, we tried to solve the classical problem by using the computational fluids dynamic software (FLUENT version 2021), as well as using the fully Coupled Pressure Based Algorithm, to determine the influence of sizing of the computational mesh and the discretization of advection term in the governing equations on the results.

2. Explain the issue

In this paper, the interaction between the shockwave in the air with R22 and helium bubbles has been simulated as two-dimensional.

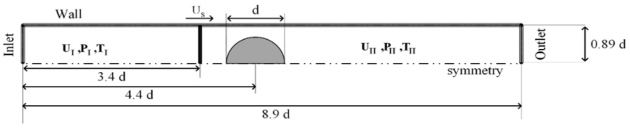

Fig.1. indicates the computational field of the problem of Shock Bubble Interaction. The shadowed area represents the bubble with a 50 mm diameter.

To simulate the problem, the computational field schematic is shown in Fig.1. To be stable in the solution, the time step of the problem is determined by using the acoustic courant number. (Eq. (1))

$$Co = \frac{a_{II\,air}\,\Delta t}{\Delta x} = 0.35 \tag{1}$$

3. Governing equation

The conservation equations governing fluid flow at all velocities for this problem, assuming the absence of viscous stresses and thermal conductivity, are Euler equations, which include:

Mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{2}$$

Momentum:

$$\frac{\partial \rho u_j}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} \tag{3}$$

Energy:

$$\frac{\partial \rho h}{\partial t} + \frac{\partial \rho u_i h}{\partial x_i} = \frac{\partial P}{\partial t} \tag{4}$$

In this study, the volume of fluid method is used to capture the interface between two immiscible phases by using a color function ψ that is presented in Eq. (5).

$$\frac{\partial \psi}{\partial t} + u_i \frac{\partial \psi}{\partial x_i} = 0 \tag{5}$$

4. Results

In this section, the results of the numerical simulation of the shock bubble interaction (SBI) are investigated, and the accuracy of the results obtained from the present study has been confirmed by citing the results of numerical research by Denner et al [11].

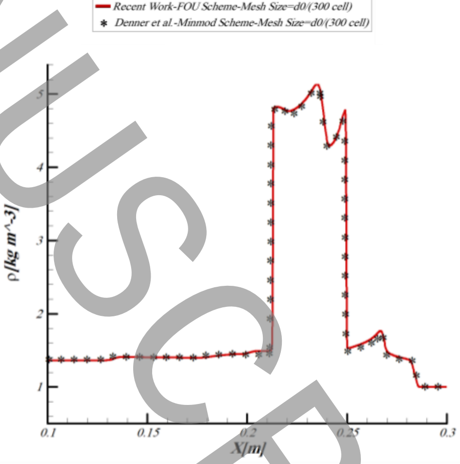

Fig.2. Density diagram resulting from the interaction of the shock wave (M=1.22) with the R22 bubble in 247μs, compared with obtained results from the numerical research of Denner et al [11].

The first-order upwind discretization method is a TVD method; this paper uses this method to discretize the advection terms in the governing equations. Due to the low accuracy of this method, numerical diffusion is injected into the problem. So the results obtained are practically the same as those obtained by solving the Navier-Stokes equations. It's noticeable the amount of numerical diffusion can be significantly reduced by reducing the size of the computational grid. In this literature, the size of the computational grid is determined according to the number of cells located in the initial diameter of the bubble. (Eq. (6))

$$\Delta x = \Delta y = \frac{d_0}{cell\ number} \tag{6}$$

According to fig. 3. increasing the number of computational cells to 300 cells (in the initial diameter of the bubble) has a significant effect on the solution results, and by further reducing the size of the computational grid for the first-order upwind discretization method, there is virtually no change in the results.

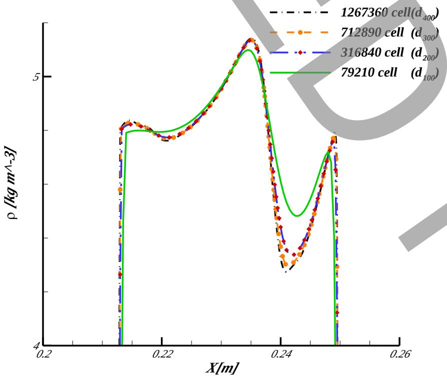

Fig.3. Numerical results of the density of shock wave interaction with the R22 bubble problem at 247μs in different computing mesh sizes.

Also, a comparison of the results of solving the problem of shock interaction with the helium bubble for two methods of first-and second-order upwind discretization is shown in Fig.4.

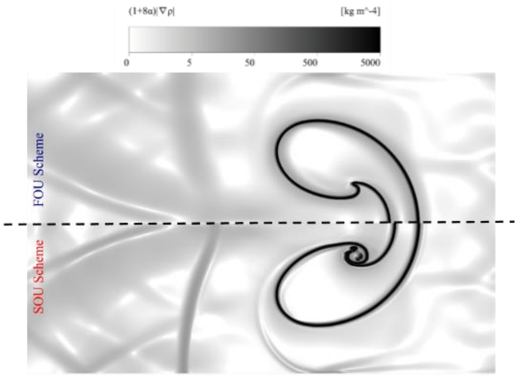

Fig.4. The effect of the discretization method of the governing equations on the results of solving the shock interaction with the helium bubble problem at 427μs.

5. Conclusions

Using the first-order upwind discretization method, the capture of reflective, transmitting, and collision waves in this study are similar to the results obtained from high-precision discretization methods. Still, the waves are thicker due to numerical diffusion. The only advantage of using the second-order upwind discretization method is that in areas far from the interface, it captures the reflected and transmitted waves more accurately without numerical diffusion. So if the discretization method is used in such a way that it has first-order accuracy at the interface of two phases and its accuracy is two or higher in areas far from the interface, while maintaining stability in solving The results obtained from the simulation with such a discretization method will be very consistent with the experimental results.

6. References

[1] D. Ranjan, J. Oakley, R. Bonazza, Shock-bubble interactions, Annual Review of Fluid Mechanics, 43 (2011) 117-140.

[2] F. MARBLE, E. ZUKOSKI, J. Jacobs, G. Hendricks, Shock enhancement and control of hypersonic mixing and combustion, in: 26th Joint Propulsion Conference, 1990, pp. 1981.

[3] M. Delius, F. Ueberle, W. Eisenmenger, Extracorporeal shock waves act by shock wave-gas bubble interaction, Ultrasound in medicine & biology, 24(7) (1998) 1055-1059.

[4] J. Lindl, Development of the indirect-drive approach to inertial confinement fusion and the target physics basis for ignition and gain, Physics of plasmas, 2(11) (1995) 3933-4024.

[5] V. Coralic, T. Colonius, Finite-volume WENO scheme for viscous compressible multicomponent flows, Journal of computational physics, 274 (2014) 95-121.

[6] R. Abgrall, How to prevent pressure oscillations in multicomponent flow calculations: a quasi conservative approach, Journal of Computational Physics, 125(1) (1996) 150-160.

[7] M.R. Baer, J.W. Nunziato, A two-phase mixture theory for the deflagration-to-detonation transition (DDT) in reactive granular materials, International journal of multiphase flow, 12(6) (1986) 861-889.

[8] D.R. van der Heul, C. Vuik, P. Wesseling, A conservative pressure-correction method for flow at all speeds, Computers & Fluids, 32(8) (2003) 1113-1132.

[9] P. Wesseling, Principles of computational fluid dynamics, Springer Science & Business Media, 2009.

[10] F. Moukalled, L. Mangani, M. Darwish, The finite volume method, in: The finite volume method in computational fluid dynamics, Springer, 2016, pp. 103-135.

[11] F. Denner, C.-N. Xiao, B.G. van Wachem, Pressure-based algorithm for compressible interfacial flows with acoustically-conservative interface discretisation, Journal of Computational Physics, 367 (2018) 192-234.