

AN ALE SCHEME FOR NUMERICAL SIMULATION OF 2D UNDERWATER EXPLOSION

Ahmadreza Pischevar

Isfahan University of Technology,
Isfahan 84156, Iran
e-mail: apishe@cc.iut.ac.ir

Key words: ALE scheme, Gas-Water Riemann problem, Unstructured moving mesh, Underwater explosion

Abstract. *In the present study an ALE scheme is developed on the unstructured meshes for the simulation of multi-material compressible flows in which a gas phase zone is separated from a compressible liquid zone by an immiscible interface. The method is used for the simulation of 2D underwater explosion near a solid surface.*

1 INTRODUCTION

There are two main techniques for the numerical simulation of multifluid compressible flows. In the first method, the interface is not tracked directly and an Eulerian scheme is used to integrate governing equations on a fixed grid. The drawback of this approach is revealed when two compressible fluids of very different equation of state is mixed within a cell. In such a case the calculation of pressure and speed of sound can be inaccurate¹. To get around this difficulty a model is needed for the equation of state within a multimaterial cell. Unfortunately such a reliable model has not been presented so far. In the second approach, the interface is tracked directly. This can be accomplished either by using an updated Lagrangian method in which the computational grid is fixed on the material or by moving the interface nodes locally with the particle velocity. Since the interface nodes always coincide with the same fluid particle, the interface remains sharp during the computation and therefore, artificial fluid mixing is prevented. However, there are also some difficulties with this approach. After several time steps, due to large velocity gradient, element distortion can occur and as a result of degeneracy of some elements a remeshing stage is inevitable. Unfortunately, the remeshing procedure is usually expensive and data transferring from old mesh to new one in some case can be inaccurate.

In the present study, an Arbitrary Lagrangian Eulerian (ALE) scheme is introduced as a remedy to reduce the number of remeshing stage required in the computation of a multi-phase compressible flow. In this method, the material boundary between two phases is considered as Lagrangian and ALE behavior is only allowed within the materials. At the end, the proposed scheme is used to study an underwater explosion near a solid surface as a test case.

2 GOVERNING EQUATIONS AND NUMERICAL ALGORITHM

We consider the compressible flows involving an immiscible interface separating fluids of different properties (Fig. 1). The fluids of interest consist of two different phases, liquid and gas, and viscous or heat-flux effects can be negligible. Therefore, the governing equations for both phases reduce to the Euler set of equations. We view this problem as a two-field problem where the gas-water interface has its own dynamic. Let $\Omega(t)$ be a control volume confine to the time dependent control surface $\Gamma(t)$ with outward unit vector n , $x(t)$ be the grid point coordinates and $\dot{x}(t)$ be the grid velocity field where time is denoted by t . The ALE formulation of the governing equations in integral form can be written as:

$$\frac{\partial}{\partial t} \int_{\Omega(t)} U d\Omega + \oint_{\Gamma(t)} F(U, n, \dot{x}) d\Gamma = 0 \quad (1)$$

where U is the vector of conservative variables defined as:

$$U^T = [\rho, \rho u, \rho v, \rho E] \quad (2)$$

In Eq. (2), ρ is density, u and v are the Eulerian velocities, p is pressure and E is the total energy. The above set of equations is completed by an equation of state. In our study, we consider only those liquids that behave as a compressible fluid with the 'stiffened gas' equation of state in the form of:

$$e = \frac{1}{N-1} \frac{(p + N\beta)}{\rho} \quad (3)$$

The material parameters for an ideal gas are $N=1.4, \beta=0$ and for water $N=5.5, \beta=4921.15$ bars.

The semi-discrete form of equation (1) for an arbitrary cell is given by:

$$\frac{d}{dt} (\bar{U}\Omega)_i + \tilde{F}_i = 0 \quad (4)$$

where \bar{U}_i is the cell average value and \tilde{F}_i denotes the residuals resulting from the net fluxes cross the closed surface surrounding Ω_i :

$$\tilde{F}_i = \iint_{\Gamma_i(t)} F(U) \cdot \hat{n} d\Gamma = \sum_{j(i)} \Delta\Gamma_{ij} \tilde{F}_{ij}(U_i, U_j, \hat{n}_{ij}, \dot{x}_{ij}) \quad (5)$$

\hat{n} represents the outward normal vector to surface Γ_{ij} , \tilde{F}_{ij} is the numerical flux associated with the control surface Γ_{ij} and \dot{x}_{ij} is the control surface velocity determined at the midpoint of each edge using the end vertices values. Using a simple explicit time integration scheme, Eq (4) can be written as:

$$\Omega_i^{n+1} \bar{U}_i^{n+1} - \Omega_i^n \bar{U}_i^n + \int_{t^n}^{t^{n+1}} \tilde{F}_i dt = 0 \quad (6)$$

The most crucial issue of an ALE method is the proper approximation of the time integral in the above equation. Since the control volume configuration changes in time, we have to decide about the grid geometry on which the time integration is performed. The same question arises for the grid velocity \dot{x} . Some answers to these questions can be found if one applies this scheme to the uniform flow. In such a case the conservation equations reduce to a purely geometric statement for the control volume configurations. This statement is called the geometric conservation law (GCL). For general cell geometry, the conditions for an ALE scheme to satisfy geometric conservation law have been investigated by Zhang et al.², Konga and Guillard³, Lesoinne and Farhat⁴. As shown by the later author, the uniform flows can be recovered by an ALE scheme if the flux is evaluated at the midpoint configuration and the grid velocity is assumed to be constant over time step and given by:

$$\dot{x}^{n+1/2} = \frac{x^{n+1} - x^n}{\Delta t} \quad (7)$$

For our particular problem, the physical system under consideration evolves two medium fluids with several gas-water interfaces, as shown in Fig.1. Each interface is approximated by a piecewise linear curve and represented by an interface mesh consist of nodes and edges. The computational domain is covered by triangle unstructured elements in a way that all elements of interface mesh (nodes and edges) exist in the domain mesh. In a time interval $[t, t + \Delta t]$ the motion of interface nodes is given by

$$x(t + \Delta t) = x(t) + V(x(t), t) \Delta t \quad (8)$$

where $V(X, t)$ is the interface velocity at time t . Determination of velocity at each interface node is accomplished by the resolution of a 1D gas-water Riemann problem solved normal to the interface. The normal unit vector at an interface node is determined by averaging the normal vector to each interface edge containing that node. Then, the right and left state variables to the gas-water interface nodes are reconstructed from the value of cell centers located at the same material and have that node as a vertex. Since the gas-water interface is treated as a Lagrangian surface, the interface node velocity is set to the contact velocity. It should be noted that the tangential contact velocity is not determined by the Riemann problem and we assume that the tangential velocity of the interface is equal to the tangential velocity of the one of the two phases depends on the sign of relative normal velocity to the interface at that node.

Therefore, to preserve the GCL conditions, the new mesh configuration and the midpoint grid velocity are calculated as follows:

- 1- First the velocity of interface grid is obtained by solving a Riemann problem as mentioned above. This velocity is used to update the position of interface.

2- The new position of internal nodes is obtained using a smoothing operator:

$$\nabla^2 x^{n+1} = 0 \quad (9)$$

3-Finally the grid velocity and the midpoint grid position are calculated as:

$$\begin{aligned} \dot{x}^{n+1/2} &= \frac{x^{n+1} - x^n}{\Delta t} \\ x^{n+1/2} &= \frac{x^{n+1} + x^n}{2} \end{aligned} \quad (10)$$

After updating the grid position, some elements, usually near the gas-water interface, can be ultimately distorted due to the high velocity gradients. Therefore, the presented scheme must be associated with a remeshing stage.

In this work the technique used for calculating the fluxes is based on solving a Riemann problem with the constant left and right initial conditions. The second order spatial accuracy is achieved by using a proper gradient function to reconstruct the cell face variables from the cell average values at the center⁵. Then the cell face numerical flux is calculated in the state variables obtained from a classical Riemann solver with U^{R^-}, U^{L^+} constant initial data at either side of each cell interface considering the face velocity:

$$U^k = U^*(U^{L^+}, U^{R^-}, \dot{x}) \quad (11)$$

The medium to the left and right side of each cell face may be gas or liquid. Therefore, three different sets of initial conditions can be defined for the Riemann problem: there is gas on either side of a face, there is water on either side and finally, gas and liquid are located on the left and right sides of an interface. Based on initial conditions, for cases where a vacuum zone can not occur, there are four possible resolutions to the Riemann problem regardless of the type of material located to the left and right of an interface as shown in Fig. 2. However, for the initial conditions that lead to a vacuum zone, the wave pattern is slightly different in each case as shown in Fig.3. In such a case for the water-water Riemann problem we can have two constant states with zero pressure at either side of a vacuum zone that can not occur in the solution of a gas-gas Riemann problem⁶. Similarly for the gas-water Riemann problem a region of constant variable with zero pressure can happen on the water side. On an Eulerian grid dealing with the vacuum zones is not straightforward. As a result, a simplified model for this resolution is used: for the gas-gas case, the interface parameters are set to zero for the pressure, density and velocity while for the water-water case pressure is set to zero and density and velocity are replaced by the averaged left and right contact values. For the gas-water case, since the interface is moved by the contact velocity, only pressure is considered to be zero at the interface.

3 RESULTS AND DISCUSSION

In the first test case, we examine the conservation property of the proposed method by considering the advection of a gas bubble by a uniform water stream with the same pressure and velocity. The computational domain is a square rectangle of 2m length which the bubble is initially located at its left bottom corner. The stream velocity is $200\sqrt{2}$ and in the diagonal direction. Density in the gas and water zone is set to 1.23 and 1000, respectively. The bubble motion is captured by a few snap shots in Fig 4. As one may notice, after several time steps, some elements in the water zone are highly stretched in the windward side of the bubble. The uniform flow condition is recovered intact by the scheme and as shown in Fig 5 the initial bubble mass is not changed during the computation. Therefore, the GCL condition is satisfied exactly by this scheme and conservation is preserved for the uniform flow. Five stages of remeshing were performed for the results shown in this figure. It is interesting to note that by using a more restrictive remeshing criterion, it is not possible to reduce the number of remeshing stage for a given final time.

Next, the capability of the method in predicting the behaviors of a gas-water interface in the presence of strong shock and rarefaction waves is studied. In this test case, we consider the two dimensional flows arisen from an explosion near a solid wall. The radius of the explosive charge is considered to be 1 unit and its center is located 3 units away from a solid boundary. The initial conditions for the density and pressure within the gas bubble are $1270\text{kg}/\text{m}^3$ and 9000atm , respectively, and for the water media are $1000\text{kg}/\text{m}^3$ and 1atm . The computational domain is a rectangular region defined by $x, y \in [(-15,3), (-15,15)]$. It is meshed by a grid of 59929 elements, 1534 of them uniformly distributed inside the bubble at the beginning of computation. The computation is stopped after 13000 time steps. For this simulation only 5 stages of remeshing are required as the bubble area increases by a factor of 4.

The general features of the computed flow field are shown by a time sequence of pressure contours in Fig. 6. As illustrated in this figure, the process starts with propagation of a cylindrical main shock wave into the water zone along with the propagation of an expansion wave toward the bubble center (6-a). The main shock then reflects from the solid boundary and interacts with the bubble. When the reflected shock hits the bubble, it is partially transmitted into the gas zone and partially reflected back as a regular rarefaction waves into the water zone (6-b). As the interaction proceeds, the angle between the reflected shock and the bubble interface increases resulting in an increase in the pressure of flow behind the incident shock in the liquid region. Moreover, the transmitted shock inside the gas bubble moves much slower than the reflecting shock in the water due to the slower speed of sound in the gas region. As a consequence of this pressure difference, an irregular type of rarefaction wave is generated at the location where the incident shock hit the bubble surface and causes the attenuation of the reflected shock near the interface (6-c). On the other hand, the reflected rarefaction wave reflects back from the solid wall and in turn interacts with the bubble. As shown in Fig. 6-d, as a result of interaction these waves partially are transmitted into the bubble as expansion waves and partially reflected as a bundle of compressive waves into the

water zone. Finally, after reflection of rarefaction waves, pressure ultimately reduces to zero near the center of the wall and the cavitation occurs as shown in Fig. 6-e. As a consequence, the flow moves back to the wall rapidly to compress the cavitation zone. Therefore, a secondary shock is developed from the wall to adjust the velocity of accelerated flow with that needed as a boundary condition at the solid surface. Figure 6-f clearly shows that the process of cavitation creation and collapse can be predicted by this simplified model.

The deviation of mass within the gas bubble from its initial value is depicted in Fig 7 for this test case as well. As expected, conservation is only violated during the remeshing stage. However, the deviation is in the order of accuracy of the interpolating method used for the transferring of state variables from the old mesh to the new one

4 CONCLUSION

In this work an ALE scheme is developed for the computation of compressible multi-fluid flows. The interface is treated purely Lagrangian and the ALE behavior is only allowed within the material. As a result, when the interface undergoes a large deformation the number of remeshing stages is substantially reduced. In this method, the fluxes are calculated by solving a Riemann problem at the gas-gas, gas-water or water-water interface. A simplified model is also proposed for the process of cavitation when pressure reduces to zero in the liquid phase. Finally, the method has been successfully applied to the problem of underwater explosion.

REFERENCES

- [1] R. Abgrall S, Karni, "Computation of compressible multifluids" *J. Comput. Phys.* 2001; **169**: 594-623.
- [2] H. Zhang, M. Reggio, TY. Trepanier, R. Camarero, " Discrete form of the GCL for moving meshes and its implementation in CFD scheme". *Comput. Fluids.* 1993; **22**: 9-23.
- [2] BN Konga, H. Guillard, "Godunov type method on non-structured meshes for three-dimensional moving boundary problems", *Comp. Method Applied Mech. Engrg.* 1994; **1130**: 183-204.
- [4] M. Lesoinne, C. Farhat "Geometric conservation laws for the flow problems with moving boundaries and deformable mesh and their impact on aeroelastic computations" *Comput. Methods Appl. Mech. Engrg.* 1996; **134**: 71-90.
- [5] D. Rausch, JT Batina, H. Yang, "Spatial adaptation procedures on unstructured meshes for accurate unsteady aerodynamic flow computation", *AIAA paper*, 91-1106-cp.
- [6] HS Tang, D. Huang "A second-order accurate capturing scheme for 1D inviscid flows of gas and water with vacuum zones", *J. Comput. Phys.* 1996; **128**: 301-318.

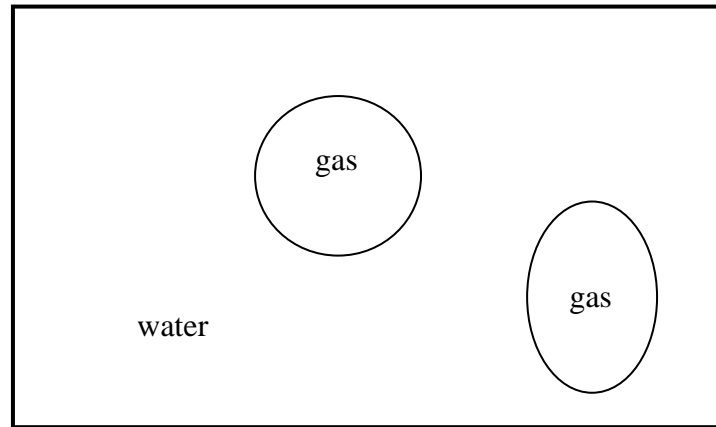


Figure 1: system of two fluids medium with several gas water interfaces

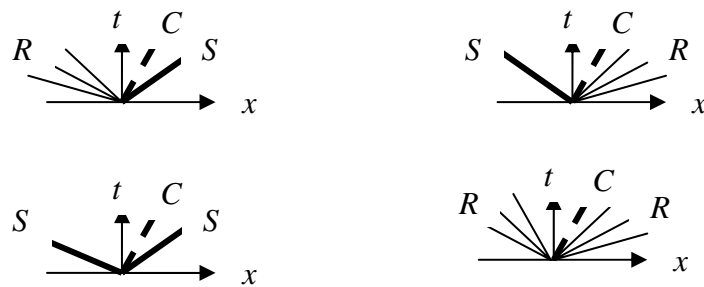


Figure 2: Four possible resolutions of a Riemann problem without a vacuum zone

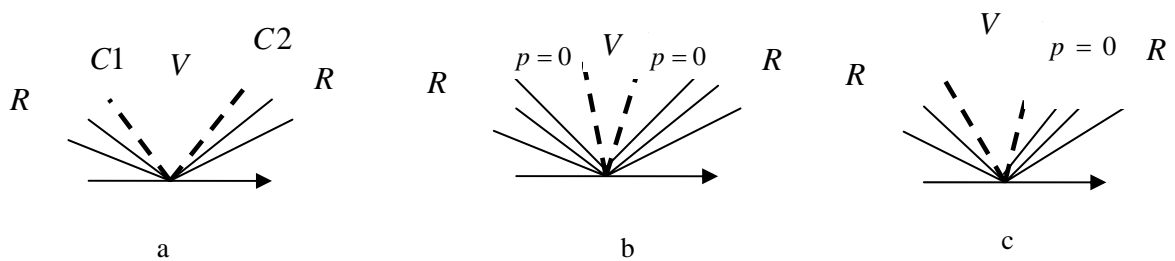


Figure 3: Vacuum occurrence in a Riemann problem, a) gas-gas, b) water-water, c) gas-water

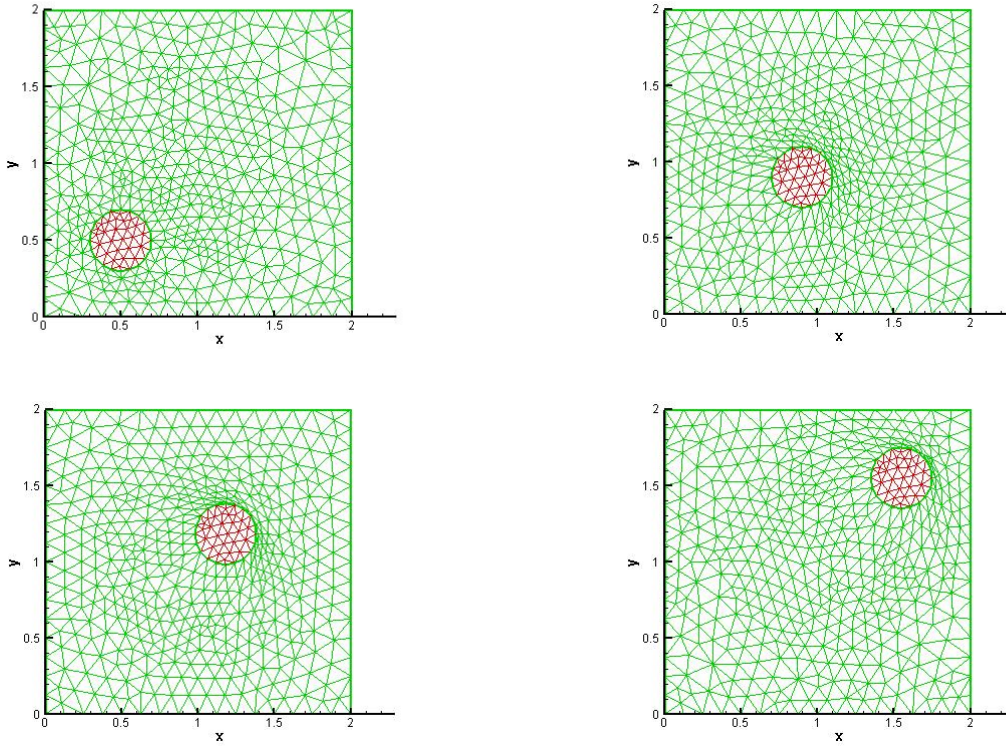


Figure 4: Advection of a gas bubble with a uniform stream

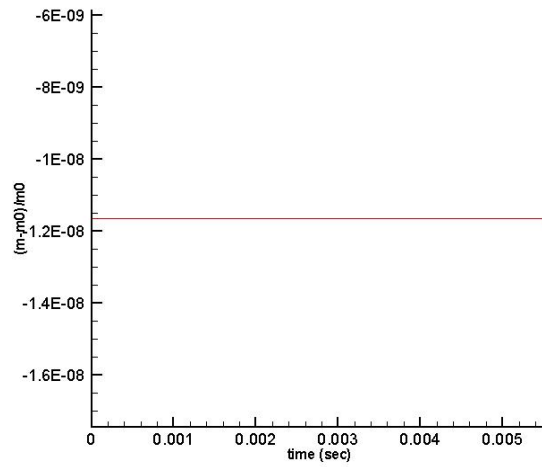


Figure 5: Deviation of gas bubble mass from its initial value

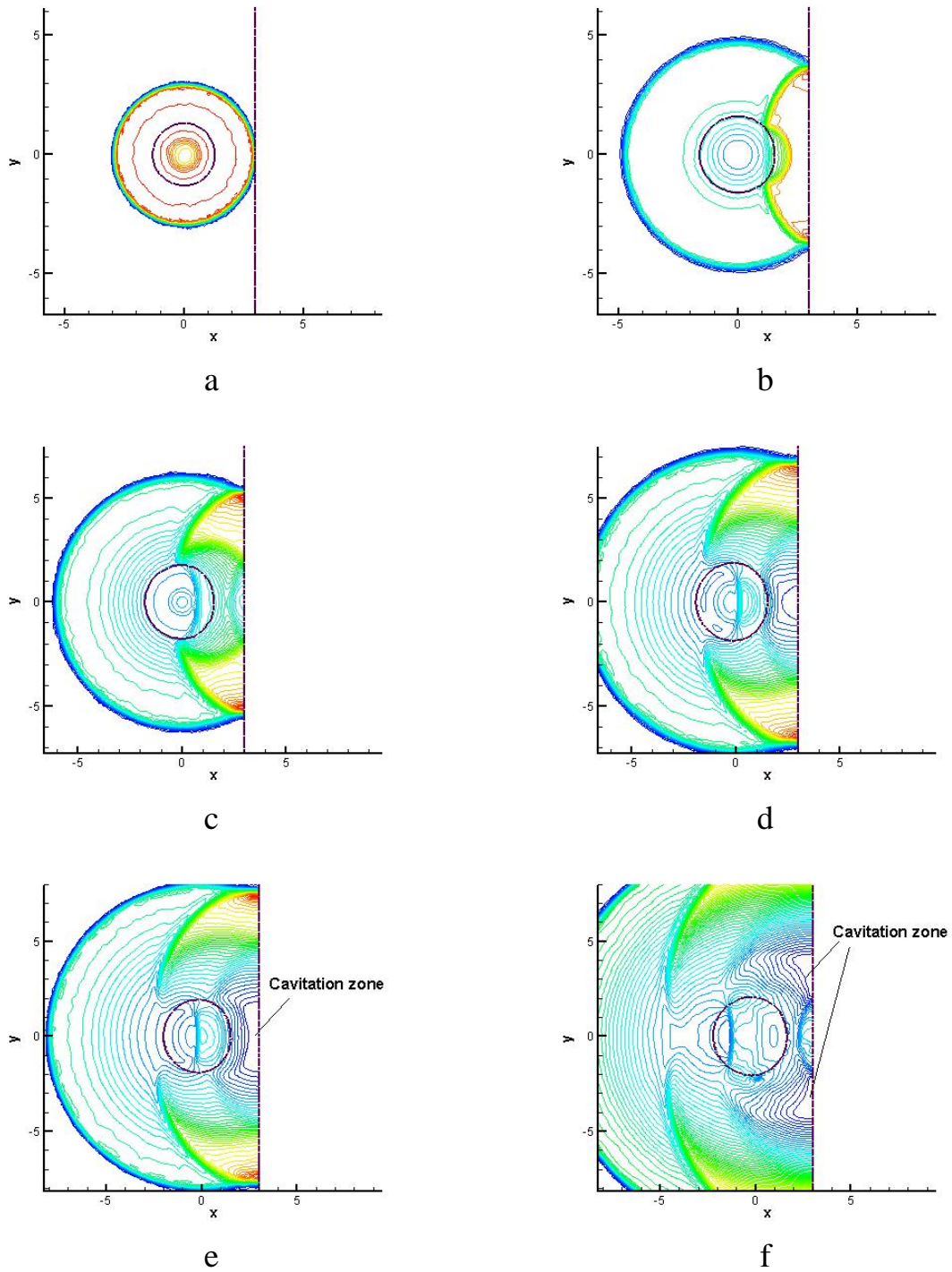


Figure 6: Pressure contours of time sequence of an underwater explosion near a solid surface

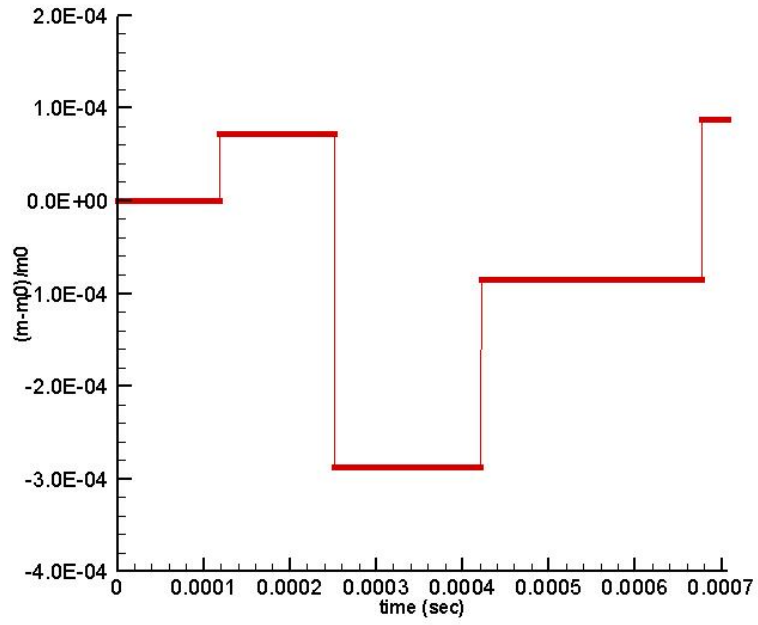


Figure 7: mass conservation assessment for the underwater explosion test case