INVESTIGATION OF BUBBLES INTERACTION AND COALESCENCEBOILING IN THE BOILING HEAT TRANSFER PROCESS

Zhixin Zhao *, Xudong Xue, Kun Wang

Shaanxi University of Technology, China

* Corresponding author; E-mail: zhixin2000@qq.com

Abstract: *This paper presents a new approach to investigate the bubbles movements and their interaction in the boiling heat transfer proess. Based on the study of the wake flow of single rising bubble, the interaction of bubble pairs, such as interaction and coalescenceboiling, is experimentally and numerically simulated. The validity of the numerical simulation was verified by the experimental pictures captured by high-speed CCD camera. The wakes of two bubbles was revealed by obtatined velocity field, and the interaction between two bubbles was analyzed in fluid dynamics. The simulations shows that the coalescence of bubbles may happen when one bubble enters the wake influence area of the main bubble. The velocity vector field shows that wake flow of the bubble is the major influencing factor determining the interaction between bubbles.*

Key words: *Numerical simulation; High speed photography; Level Set method; boiling heater transfer.*

1. Introduction

In many industrial fields such as energy, power, environment and chemical processes etc, it is always of great interest to investigate the movement of bubbles. In the experimental study of bubbles, although the shape of the interface can be observed accurately, the physical and chemical properties, as well as the transfer of momentum, energy and mass, are powerless. For this reason, many researches use numerical simulation with experimental verification and theoretical analysis to study two-phase flow.

The core technology of numerical simulation of two-phase flow with deformable Multi-Interfaces is the processing of interfaces. It is necessary to consider whether the following properties can be achieved between accuracy, robustness, conservation and ease of operation for different research objects: accurately describing the location of interfaces; meeting the requirements of calculation accuracy; dealing with the topological structure of various interfaces; and reflecting the real objects of fluid-phase interfaces and its real physicochemical properties.

Because Level Set method is mature and representative in interface processing, this paper mainly discusses the numerical simulation of two-phase flow with deformable interface based on this method.The Level Set method generally uses the Continuum Surface Force (CSF) model by Brackbill et al. [1], the method converts fluid surface force into volume force, as the source term of volume force in Navier-Stokes equation, so the complexity of treating the interface force as boundary condition can be avoided. Because of the calculation error of the surface tension term, it will inevitably lead to the "sending" of the interface. Son [2] pointed out that only by improving the accuracy of curvature calculation can the parasitic flow be effectively restrained. Wang et al. [3] proposed a weighted scoring method to improve the accuracy of local curvature. This method set different weighting factors according to the distance from each grid node to the control point, and successfully suppressed the "mailing" caused by the inaccurate calculation of surface tension. Coquerelle and Glockner [4] developed a curvature calculation method with fourth-order accuracy to suppress the parasitic flow. The algorithm consists of two steps: first, calculating the local curvature with fourth-order accuracy based on Level Set function on the element mesh; and then expanding the curvature to the adjacent phase interface accurately according to the nearest point principle.

With the development of bubble dynamics research, level set method has be wildly used in tracking bubble Interface. The recent research work includes: Cho, Sungwook, and G. Son [5] use the Level Set method to research the influence of wall and ambient temperature on the bubble motion in the acoustic droplet vaporization condition; Xu Jinliang [6] studied bubbles migration induced by thermal capillary force under uniform incident heat radiation condition,by use Level Set method

However, most of these studies only consider the motion of a single bubble, and few of them simulate the motion of two bubbles. The fact that whether the bubble pairs can repulse or aggregate due to the interaction will have a great impact on the heat and mass transfer process. In this paper, a 3-D incompressible Navier Stokes solver is proposed for air-water two phases flow in bubbles motion by Level Set method. The effects of the position and size of two bubbles on wake flow field are studied in detail. In addition, some validation experiments have been carried out, and the experimental results have been compared with the numerical simulation results to verify the effectiveness of the method.

2. Description of numerical model

The numerical simulation of two-phase flow follows the method of solved with the one-fluid incompressible flow approach, Navier–Stokes equations. But for multiphase flow, to deal with the interface between different fluids, the surface tension model is introduced into the equation. So the governing equation can be expressed as follows:

$$\nabla \cdot \vec{V} = 0 \tag{1}$$

$$\rho\left(\frac{\partial \vec{V}}{\partial t} + \nabla \cdot \vec{V}\vec{V}\right) = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{V} + F_\sigma \tag{2}$$

The Eq. (1) is the equations of continuity equation reflecting the Law of Conservation of Mass, and Eq. (2) is the momentum conservation equation. In the equations, $\vec{V}$ is the fluid velocity vector, p is the pressure in the field, ρ and μ are the physical properties of fluids, the fluid density and the fluid dynamic viscosity.

In Eq.(2), the surface tension force F_σ use the Continuum Surface Force model, the force acting on the interface can be replaced by a body force, which can be given by:

$$F_\sigma = \sigma \kappa(\phi)\delta(\phi)\nabla\phi \tag{3}$$

In the equation, σ is the surface tension coefficient of pure water, $\kappa(\varphi)$ caculates the curvature of the boundary profile between two phases and $\delta(\varphi)$ is the Dirac delta function used to describe the distribution of the fluids density, where the level set function ϕ will be described in detail below.

2.1. Level set method for interface between air and water

To solve the problem of boundary tracing, interface between air and the water can be tracked by the level set function. According to the method, the zero level of the function can be treateded as the interface between air and water, so the changes of zero level set function also represents the change of gas-liquid interface.

In mathematics, $\varphi(\vec{x},t)$ is defined as a signed function which is related to the distance from the local to the gas-liquid interface. The movement and deformation of the zero level set should satisfies the equation:

$$\frac{\partial\varphi}{\partial t}+\vec{V}\cdot\nabla\varphi=0, \tag{4}$$

The Eq. (4) gives the action-induced $\varphi(\vec{x},t)$ governed by the fluid velocity. The properties of densityρ, and viscosity μ are function of level set function in the fluid domain too, so the fluid properties ρ and μ in two deferent phases can be represented as:

$$\rho(\varphi)=\rho_g+(\rho_l-\rho_g)H(\varphi) \tag{5}$$

$$\mu(\varphi)=\mu_g+(\mu_l-\mu_g)H(\varphi) \tag{6}$$

The subscripts l and g in the equations are used to distinguish the water and the air. H (ϕ) presents smoothed Heaviside function [7]. With the above mathematical methods, two-phase flow can be treated as single-phase flow.

2.2. Numerical method

In this paper, the three directly dimensional program is carried out to simulate the above model. The governing equations is solved based on the finite volume solver, and the program use the explicit time marching and element-centered numerical techniques of ICE (implicit continuous fluid Euler) [8] to solve equations (1) and (2). And in the advection equation (4), the convection term is calculated by high-order ENO upwind formula, the time step is advanced by second-order TVD Runge-Kutta model [9]. After calculation, level set function is set to be equal to the distance function and the iterative distance redistribution program is kept mass conservation, see Sussman et al for details of the program [7].

2.3. Validation

A single rising air bubble rising in a stationary water was simulated to verify the reliability and accuracy of the program, computational domain was 4×4×8 cm^3. The diameter of the bubble is 0.8cm, and it has been initially placed 0.5cm from the bottom of the computational domain. Pressure boundary conditions (101325Pa) was applied at the top of the computational domain, and non-slip boundary conditions was applied on the remaining five walls.

In the numerical simulation, the following specific numerical values are used for the physical properties of the fluid: $\rho_l = 998$ kg/m^3, $\rho_g = 1.1$ kg/m^3, $\sigma = 0.0728$ N/m., $\mu_l = 0.001$ Pa·s, $\mu_g = 1.8\times10^5$ Pa·s,

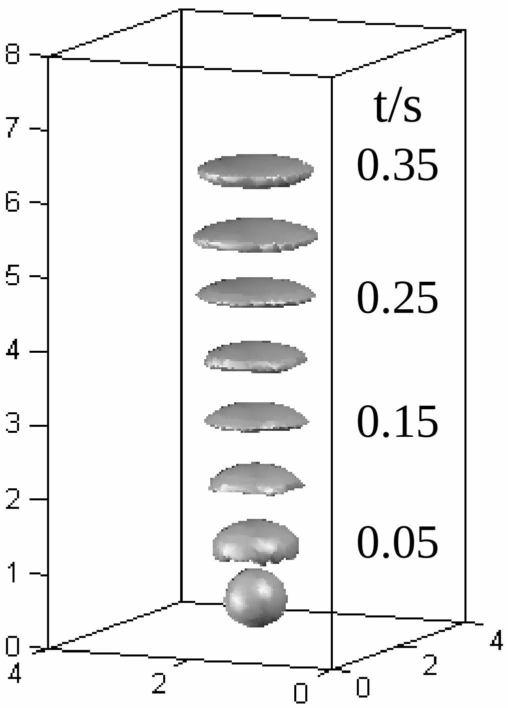


Fig. 1 The profile change of a rising bubble

Figure 1. shows the evolution of bubble shape in its rising process. The initial spherical bubble rises 1cm and becomes flat. The boundary of bubbles is clear and the shape varies periodically between the cover and the oblate ellipsoid. Relatively to the bubble shape model by fan [11], the simulation resultsare in good agreementhwith the given shapes map

The mesh density will have a great influence on the numerical results, so the smaller the mesh, the better, undoubtedly it will increase the amount of calculation. Therefore, in general, when the increase of the mesh number has little effect on the results of calculation, it can be considered that the study of the problem is no longer affected by the mesh density, which is called grid independence.

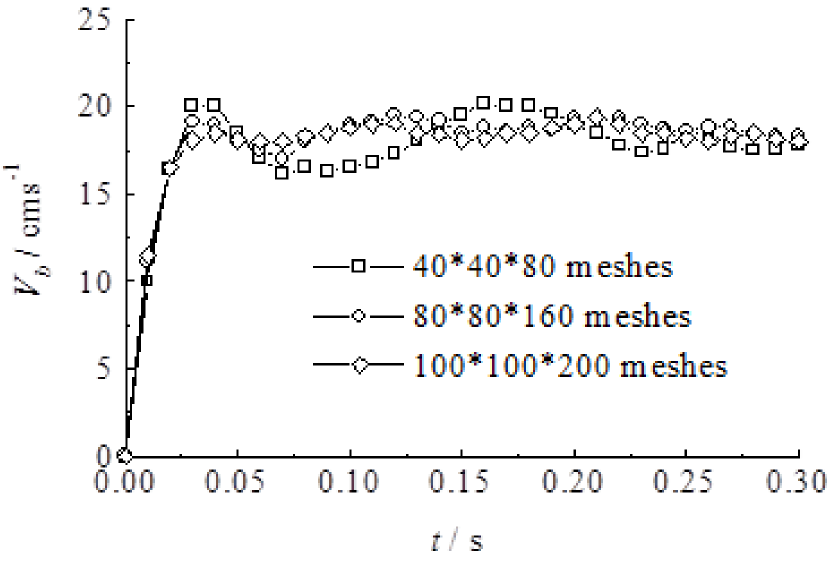


Fig. 2. The velocity of the rising bubble in pure water

Three sizes of grids were used to verify grid independence. Fig. 2. shows the velocities of the rising bubble in three grid sizes. It can be seen that the calculation results no longer depend on the mesh size when the mesh size is less than 0.05 cm in the numerical simulation of bubble rising. Therefore, the mesh size of 0.05 is used in the numerical simulation in this paper.

The result of the simulation fits well with the Collins' theoretical analysis of velocity of bubble rising velocity in a finite space, the velocity can be calculate by $V_b = 0.71\alpha\sqrt{gd_b}$, α reprents the space correction coefficient. We can calculate with this equation that the velocity is 17.85cm/s, the average rising velocity of the bubble in our problem is about 17.6cm/s.

3. RESULTS AND DISCUSSION

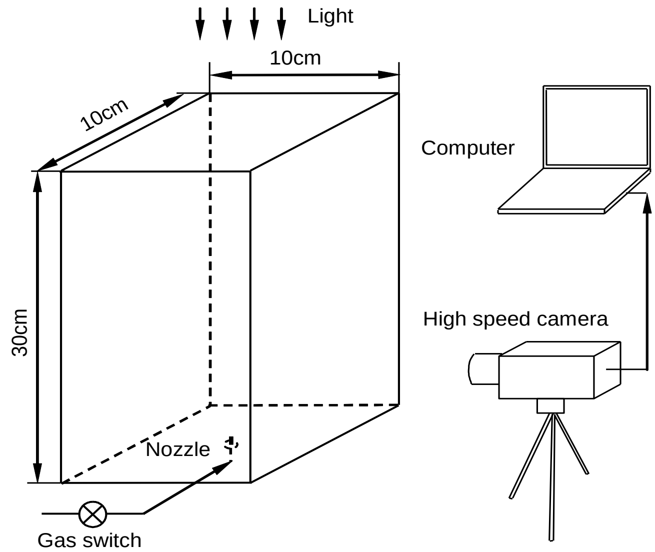


Fig. 3. Schematic diagram of high-speed camera experiment

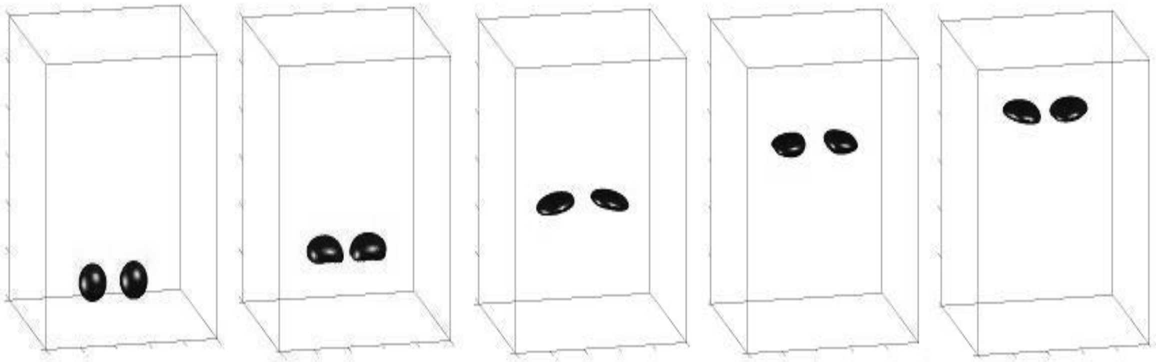
In order to compare with the results of numerical simulation, a high-speed CCD camera (Mega Speed 75K S2) was used to photograph the shape and position of moving bubble in water. Nozzles inner diameter of 0.4 cm was placed at the bottom of a glass cuboid box (10×10×30 cm^3). Air was injected into water through the nozzles. The high-speed CCD camera is placed on the right side of the box for shooting, and the image data was stored in a high-speed memory card. Fig. 3. gives the schematic diagram of this experimental.

3.1. Two horizontally-arranged bubbles

The numerical simulation begins with the simulation of two bubbles of equal size with a horizontal arrangement of 0.8 cm in diameter. From the previous numerical simulation, the calculation domain with the dimension of 5×5×6 cm^3 have little influence of boundary on bubble motion. The two horizontal array bubbles are initially placed at 0.8cm from the bottom, and horizontal distance is 1.2 cm.

The evolution process of two bubbles in different time is given in Fig. 4, and the experimental results are shown in Fig. 5.

.



(a) 0 ms (b)0.01s (c) 0.15s (d)0.2s (c) 0.24s

Fig. 4. Simulation of two bubble horizontally-arranged

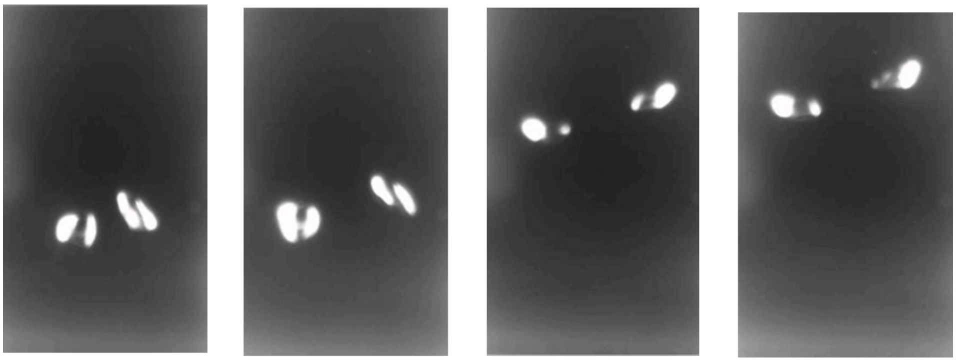
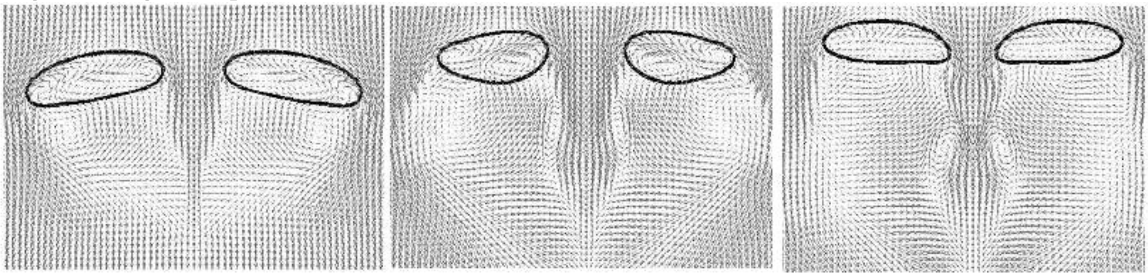


Fig. 5. The experiment photographs

In the numerical simulation, it can be seen that the bubbles quickly become oblate ellipsoids, and the two bubbles show the closure-separation-closure law in the process of rising. The experiment photographs also show that the two bubbles cloud interact under these conditions, but they are not enough to merge or repulse the two bubbles.



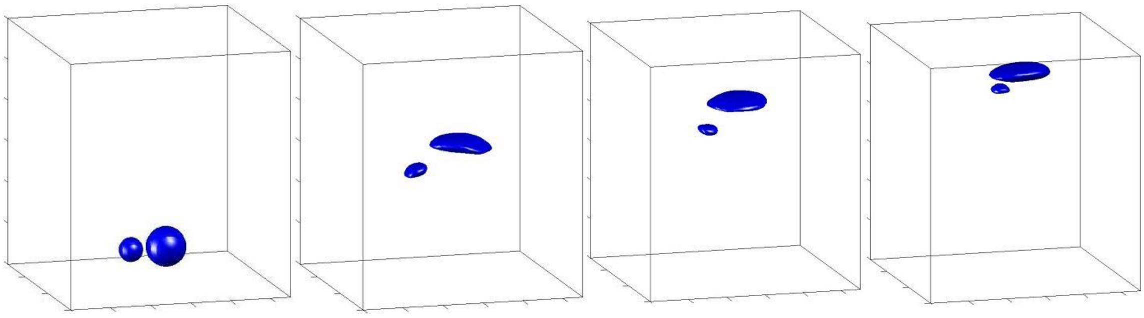
(a) 0.15s (b) 0.2s (c) 0.24s

Fig. 6. The wake flow of the two bubbles

In order to study the wake flow of bubbles, we extract the flow field data of two bubbles during their motion process. Fig. 6. givs the flow fields at the center longitudinal section (x=2.5cm) at different time. To observe the wake of bubbles better, the velocity vector of flow field is subtracted from the average bubble rising velocity calculated previously. Seen in Figure 6. there is a clear eddy region under each bubble, and some separated vortices appear in the wake. Velocity vectors between two bubbles change with the rising bubbles. The wake region becomes longer and stronger, but the two wake regions do not overlap.

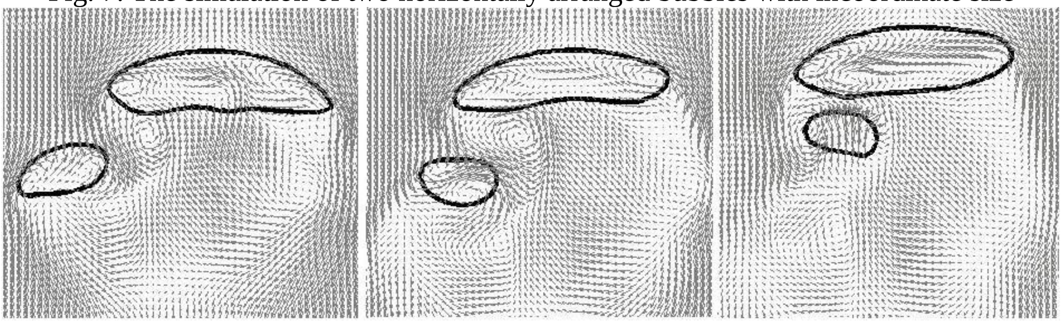
Then the size of one bubble was changed to 0.6cm, and the other one 0.9cm, the simulation is of two bubbles of different sizes, and all other calculation parameters remains unchanged. Fig. 7. shows that the evolution of two bubbles in different sizes in simulation. Fig. 8. Gives the velocity vectors field in the central profile in three different time steps.

It can be seen from Fig. 7. that the big bubble is faster than the small one, but the small bubble is affected by the big bubble. It seems the small bubble is induced by the big one, closely following the motion of the big bubble when the small bubbles get closely to the big one, the two bubbles become to coalesce. Fig. 8. Shows that the small bubble inter the wake flow zone completely, and the small bubble is rising accelerated approach to the big one.



(a) 0 s (b) 0.15s (c) 0.2 s (d) 0.24 s

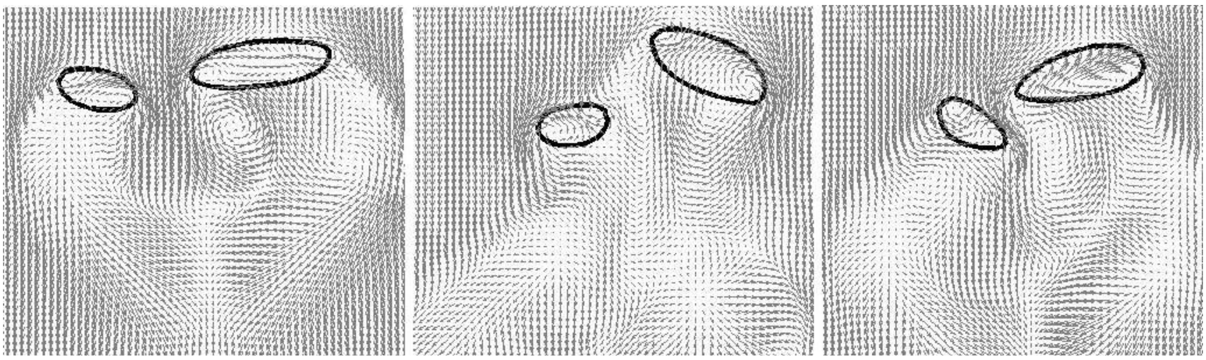
Fig. 7. The simulation of two horizontally arranged bubbles with incoordinate size



(a) 0.15 s (b) 0.2 s (c) 0.24s

Fig. 8. The flow field of two bubbles in incoordinate size(distance=0.9cm)

When the horizontal distance between two bubbles is increased to 1.2 cm, other parameters remain unchanged. The numerical simulation is shown in Fig. 9, and the experimental pictures are shown in Fig. 10.



(a) 0.17 s (b) 0.22 s (c) 0.27 s

Fig. 9. The velocity field of two bubbles in incoordinate size (distance=1.2cm)

Fig. 10. The experiment of two bubbles with incoordinate size

It is seen that the numerical simulations agree reasonable well with the experiment results. Since the distance between two bubbles becomes larger than the previous calculation, the wake flows of the two bubbles do not overlap with each other, and the behavior of this flow is quite similar to that of two horizontally-arranged bubbles with uniform sizes.

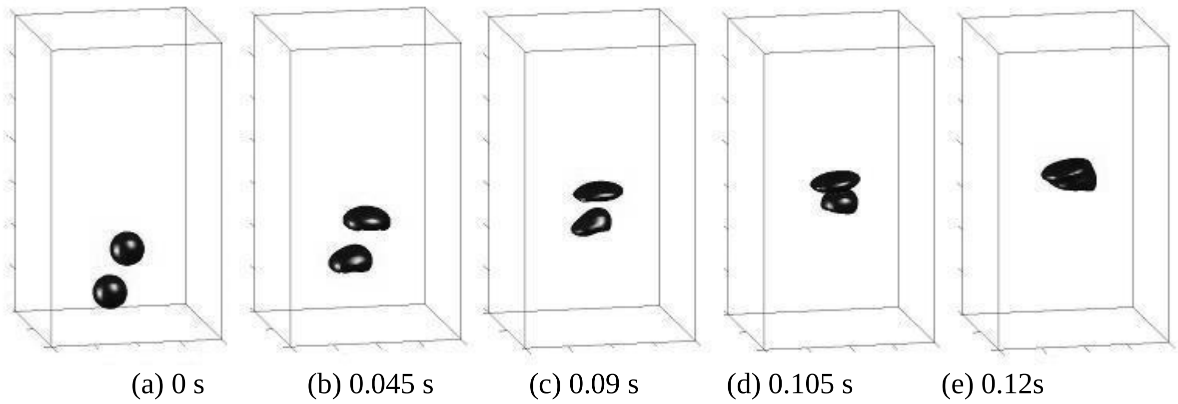
3.2. Two vertically-arranged bubbles

Two vertically-arranged bubbles was simulated to compare the wake flow zone between bubbles. The diameter of the two bubbles has the same size (0.8 cm), the horizontal distance between the two bubbles was 0.4 cm and the vertical distance was 1.0 cm. Fig. 11. displays the process of bubbles floating upward, as shown in Figure 12. The velocity vectors of the central profile at four time steps are shown.

To compare the wake flows, two vertically arranged bubbles is simulated with the same calculation parameters, the diameter of the two bubbles is 0.8cm and the horizontal distance between two bubbles is 0.4cm, and the vertical distance is 1.0cm. The simulation keeps the same parameters as before.

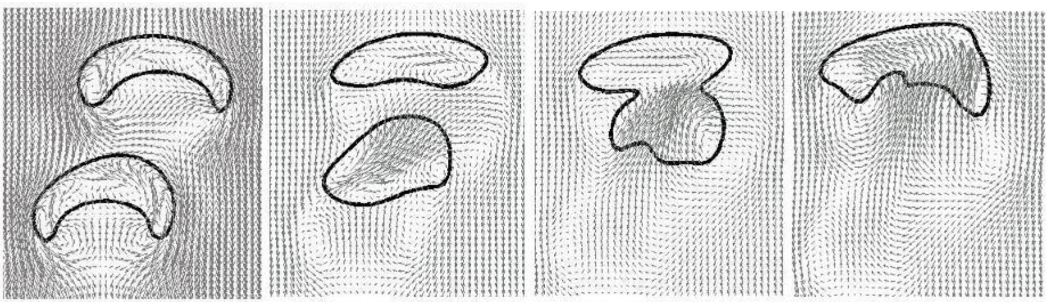
Two bubbles also rise waveringly and display one approach and one separation. When they get close in the second time, the small bubble enters completely into the wake flow of the big bubble. The attraction caused by the wake flow of front bubble accelerates the movement of behind bubble, and the two bubbles merge to a big one.

Fig. 11. shows the upward float process of the bubbles, and the Fig. 12. shows the velocity vector field in the central longitudinal section at four different time points.



(a) 0 s (b) 0.045 s (c) 0.09 s (d) 0.105 s (e) 0.12s

Fig. 11. The simulation of two vertically arranged bubbles



(a) 0.045 s (b) 0.09s (c)0.105 s (d) 0.120 s

Fig. 12. The flow field in the central longitudinal section

After a great quantity numerical simulations, we find that there is a certain law of bubble coalescence, that is, when one bubble enters the wake region of another bubble, the two bubbles may merge into a bigger bubble.

4. Conclusion

In our numerical simulations, the Level Set method is adopted to precisely describe the gas and liquid two-phase flow. To overcome the possible instability and ensure the necessary precision, the high precise discrete method is used to solve the interface transportation equation. Using these numerical methodologies, the deformation, attraction and repulsion behavior of two rising bubbles is investigated.

The numerical results show that the wake and interaction of bubbles determine the rising behavior of bubbles. For two bubbles with the same horizontal arrangement, the wake of bubbles is separated by jet, and bubbles will not merge. The two bubbles are wavy in the whole process of rising. They are closed, separated and closed. For two bubbles of different sizes, coalescence may occur when small bubbles completely enter the wake region of bubbles due to different rising rates. For a pair of vertically aligned bubbles, the wake of the upper bubbles cloud produce a smaller pressure region, which causes the lower bubbles to accelerate upward motion. Collisions and mergers occur when the projection area of the lower bubble enters the wake flow field of the upper bubble.

Acknowledgments

The authors wish to acknowledge the financial support by Natural Science Foundation Project of Shaanxi Education Department (No.16JK1146) .

References

[1] J.U. Brackbill, D.B. Kothe, C. Zemach. *A continuum method for modeling surface tension*[J], Journal of Computational Physics 100(1992) , pp.335-354.

[2] Gihun Son. *Efficient implementation of a coupled level-set and volume-of-fluid method for three-dimensional incompressible two-phase flow* [J]. Numerical Heat Transfer, Part B: Fundamentals, 2003, 43(6):17.

[3] Wang J , Yang C , Mao Z . *A Simple Weighted Integration Method for Calculating Surface Tension Force to Suppress Parasitic Flow in the Level Set Approach*[J]. Chinese Journal of Chemical Engineering, 2006, 14(6):740-746.

[4] Coquerelle M , Glockner, Stéphane. *A fourth-order accurate curvature computation in a level set framework for two-phase flows subjected to surface tension forces*[J]. Journal of Computational Physics, 2016, 305:838-876.

[5] Cho, Sungwook, and G. Son. *A level-set method for bubble growth in acoustic droplet vaporization*[J]. International Communications in Heat & Mass Transfer 93(2018), pp83-92.

[6] Xu, Jinliang . *Numerical investigation on spontaneous droplet/bubble migration under thermal radiation*[J]. International Journal of Thermal Sciences 129.6(2018), pp115-123.

[7] M. Sussman, E. Fatemi, P. Smereka, S. Osher． *An improved level set method for incompressible two-phase flows*[J], Computers & Fluids，(27)1998 663 -680．

[8] L. Fan, K. Tsuchiya, Bubble Wake Dynamics in Liquids and Liquid-Solid Suspensions, Bubble Wake Dynamics in Liquids & Liquid–solid Suspensions, Boston, 1990.

[9] B. A. Kashiwa, N. T. Padial, M. Rauenzahnr, W. B. VanderHeyden, A cell centered ICE method for multiphase flow simulation, LA UR 93 3922, Los Alamos National Laboratory, Los Alamos, NM, USA, 1994.