NUMERICAL INVESTIGATION OF AIR BUBBLES EVOLUTION AND COALESCE FROM SUBMERGED ORIFICES BASED ON OPENFOAM

FENG PAN, YING HE*, LI-ZHONG MU
** School of Energy and Power Engineering, Dalian University of Technology, heying@dlut.edu.cn*

Keywords: *Two-phase flow, Bubble coalescence, Bubble collision, VOF, OpenFOAM*

Summary

With the help of the Volume of Fluid (VOF) model in OpenFOAM framework, the air bubbles evolution and coalescence behaviours from submerged orifices has been investigated. First, the air bubble formation and rising process from a submerged orifice with different air flow rate has been simulated according to Zhang's experiments. Results show that, with the increase of the air flow rate, collision and vertical coalescence occur as a result of the reinforced wake flow. The numerical experiment of the different spacing of the twin orifice shows that, horizontal coalescence only occur in the range when S/D<1.5. The results can provide useful reference for the mechanism of coalescence in boiling process.

1 Introduction

Due to the vast majority of nucleation sites with high randomness on the boiling surface and the complex interactions of the multiple sub-processes, the heat transfer mechanism during boiling process is still challenging. A single bubble on an active nucleation site eliminates the interaction of the surrounding bubbles, thus offering clearer insight into the details of bubble nucleation, growth and departure processes than natural surfaces with many cavities. Shoji and Zhang [1] [2] manufactured isolated cavity on copper and silicon surface, investigating the effect of cavity's shape, size and depth on bubble growth and departure processes. Further investigation was conducted by arranging twin or triple artificial cavities with different spacing and arrangement to make clear the thermal interaction between nucleation sites and the hydrodynamic interaction between vapour bubbles. Three kinds of coalescence were found near the boiling surface: horizontal coalescence, vertical coalescence and declining coalescence. A dimensionless indicator, S/D was introduced as the ratio of the active sites spacing to the single bubble departure diameter. It is shown that the horizontal and declining coalescence only occur in the range when S/D<1.5 in the artificial surface. Similar experiment has been conducted on natural copper surfaces, recently. It is found that in the horizontal coalescence cases, the distances between the adjacent actives sites are less than 1.5 times of the single bubble departure diameter, which is consistent with Shoji's results on artificial surfaces. Besides, it can been also observed that, vertical coalescence only occurs in high heat flux region, with horizontal coalescence and declining coalescence accompanied at the same time.

In fact, the bubble behaviour is not only influenced by the phase change phenomenon on the boiling surface but also the two-phase fluid flow pattern. Zhang [3] investigated successive air bubbles formation, coalescence and departure behaviours from a submerged orifice, whose air flow rate was in the regime 100 cc/min ~ 2000 cc/min. As an effect of the wake flow, the interaction between air bubbles in vertical leads to multiple periods on the growing bubble. In Zhang's work, a force balance model was developed to describe the bubble's evolution in isothermal flow without phase change, including formation, interference, collision and coalescence. In current work, a series of numerical experiments based on OpenFOAM have been carried out to investigate the bubble behaviour from single submerged orifice. Subsequently, the coalescence processes of bubbles from two orifices are simulated. The criteria that horizontal coalescence occurs in isothermal system has been demonstrated. Current work can reveal the mechanism in the horizontal coalescence during boiling process from the perspective of hydrodynamic factor.

2 Numerical Model

interFoam is a standard isothermal two-phase flow solver based on Volume of Fluid (VOF) method in OpenFOAM. It is assumed that the physical properties of single phases are independent with local temperature. The two-phase fluid is treated as a mixture with homogeneous but not constant properties in a certain cell. In the VOF method, fluid volume function α (the volume fraction of the liquid phase in a cell in case of air-liquid two phase flow) is introduced as an indicator to distinguish the two phases and interface. The mixture density is defined as:

$$\rho = \left(1-\alpha_1\right)\rho_g + \alpha_1\rho_l \tag{1}$$

The Navier-Stokes equations of incompressible, Newtonian fluids are given as

$$\frac{\partial\rho}{\partial t} + \nabla\cdot\left(\rho u\right) = 0 \tag{2}$$

$$\frac{\partial\left(\rho u\right)}{\partial t} + \nabla\cdot\left(\rho uu\right) = -\nabla p + \nabla\cdot\tau + \rho g + f_\sigma \tag{3}$$

The fluid volume function equation (4) can be derived from (1) and (2), as

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha u) = 0 \tag{4}$$

In the above equations, f_σ is surface tension term. Through the Continuum surface Force (CSF) model embedded in the solver, the surface tension in the two-phase interface is turned into the adjacent control volumes as a body force expressed in equation (5):

$$f_\sigma = \sigma \kappa \nabla \alpha \tag{5}$$

Where κ is the curvature in the two-phase interface. It can be calculated as follows,

$$\kappa = -\nabla \cdot \frac{\nabla \alpha}{|\nabla \alpha|} \tag{6}$$

The governing equations are discretized by finite volume method. At each time step, PIMPLE algorithm embedded in OpenFOAM is called to deal with the pressure–velocity solution loop. The two-phase fluid mixture properties is updated first. Then the fluid volume function equation is solved employing MULES (multidimensional universal limiter with explicit solution) limiter to guarantee the boundedness of fluid volume function. Next, the velocity field and the pressure field are corrected for several iterative steps. Then the turbulence model is updated and next time step will begin.

3 Results and discussions

3.1 Single bubble behaviour

The air bubble formation and rising process from a submerged orifice with different air flow rate has been simulated according to Zhang's experiments [3] at first. The system is supposed at a uniform temperature system and the liquid is water, with the kinematic viscosity of 1.05×10^{6} m^2/s. The calculated area is 3cm×6cm and 100×200 uniform grids has been divided in the case. An air inlet with a diameter of 2mm has been set in the bottom and there is an air space (3cm×1cm) above the mixture. The effect of the air flow rate on bubble departure diameter has been investigated in current simulation. The present prediction obtains a good agreement with Zhang's experiment [3]. As the air flow rate increases from 100cc/min to 2000cc/min, the bubble departure diameter climbs from 5.4mm to 14.0mm, as show in Figure 1.

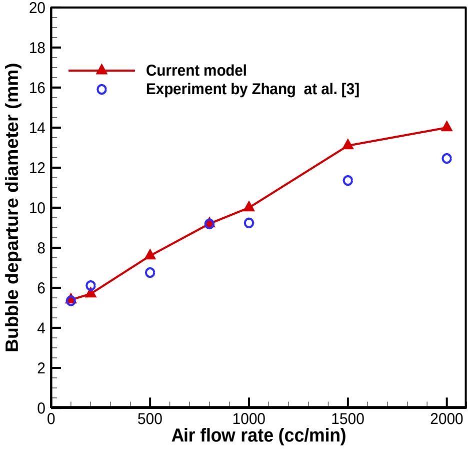


Figure 1: Variation of bubble departure diameter with the increase of air flow rate

Figure 2 shows the formation, departure, and rising process of single bubble when the air flow rate is 100cc/min. The red represents liquid and the blue represents air. There are two stages during the bubble generation. From the initial time to the 100ms, the bubble expands in both vertical and horizontal simultaneously, during which period, the surface tension force keeps balance with the buoyancy. As the bubble grows larger, once the bubble buoyancy is large enough to conquer the surface tension force, the bubble centre raises remarkably and neck forms in the bottom. As can be seen, the bubble departures at 235ms and 460ms respectively, with the same diameter 5.4mm. As the distance between the bubbles is far enough, the influence of the wake flow of the previous bubble seems negligible. In current model, the contact angle between the two-phase flow and the bottom wall is set to 90°, which represents a hydrophobic surface. This may lead to deviation in the simulated bubble growth periods compared with Zhang's experiment at al. [3].

When the air flow rate increases to 500cc/min, the bubble departures with a diameter of 7.6mm, as shown in Figure 3. The bubble departures at 85ms and rises up into bulk liquid. Meanwhile a new bobble forms, and departures at 155ms in the end. For the sake of the closer distance between the two bubbles, the wake flow induced by the rising bubble is much more intense than lower flow rate case. Therefore, the following bubble elongates obviously and departures with a slimmer shape. Consequently, the rising process of this bubble is accelerated rapidly and collides with the previous bubble at 195ms in the height about 2cm above the orifice. The collision finishes at 215ms and the double bubbles rise as a whole. In the case that the air flow rate is 800cc/min, the collision occurs at a lower position about 1.5cm and there is another bubble collides with the former double bubbles.

When the air flow rate is up to 1500cc/min, vertical coalescence occurs between two successive bubbles. The first bubble leaves the orifice at 65ms. From 65ms, a little bubble begins to grow and at 80ms coalesces with the former bubble during the growth process. Within a very short time interval of 20ms, the following bubble elongates quickly in vertical and is pulled up and absorbed into the upper bubble rapidly.

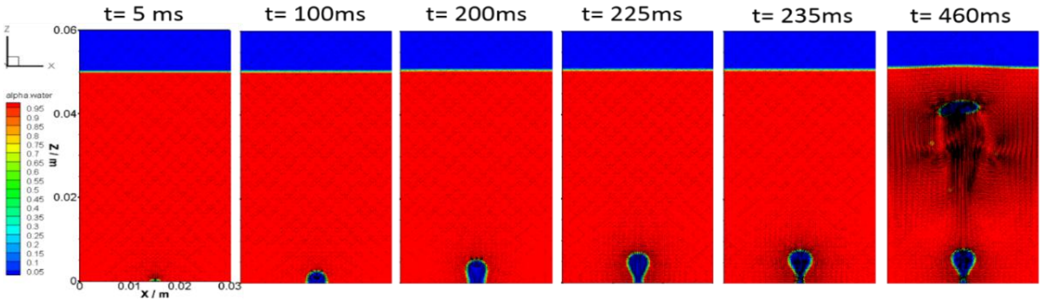


Figure 2: Single bubble behaviour from a submerged orifice when the air flow rate is 100 cc/min

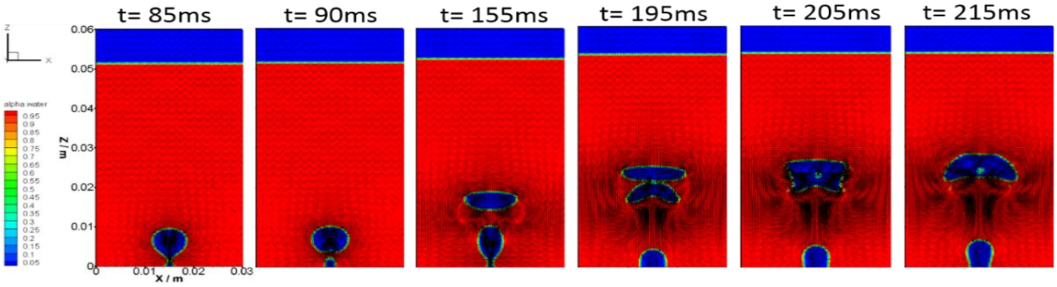


Figure 3: Bubbles collision from a submerged orifice when the air flow rate is 500 cc/min

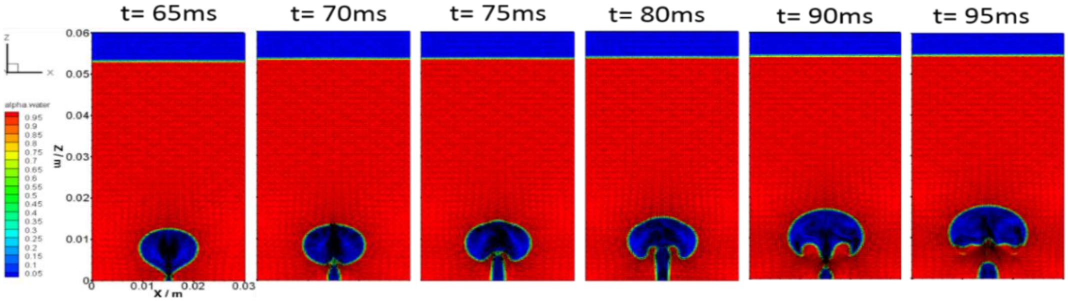


Figure 4: Bubbles coalescence in vertical from a submerged orifice when the air flow rate is 1500 cc/min

Table 1: Evolution of bubble behaviours with the increase of air flow rate

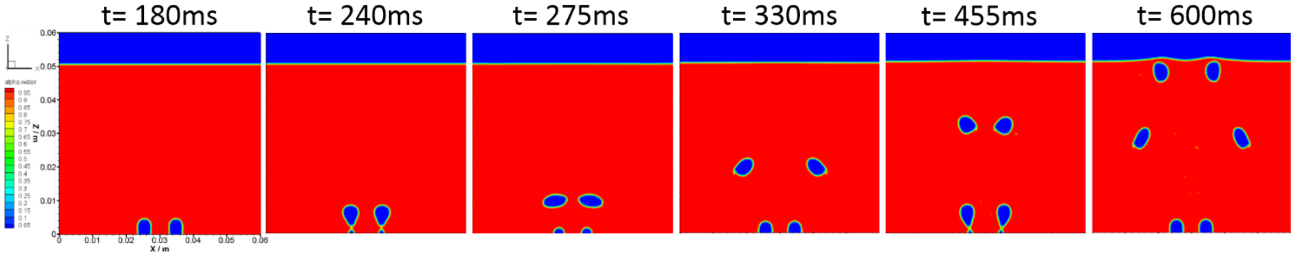
Air flow rate (cc/min)	Without interaction	Bubble collision	Vertical coalescence
100	○	×	×
200	○	×	×
500	○	○	×
800	○	○	×
1000	○	○	×
1500	○	○	○
2000	○	○	○

○: existent in the case; ×: non-existent in the case.

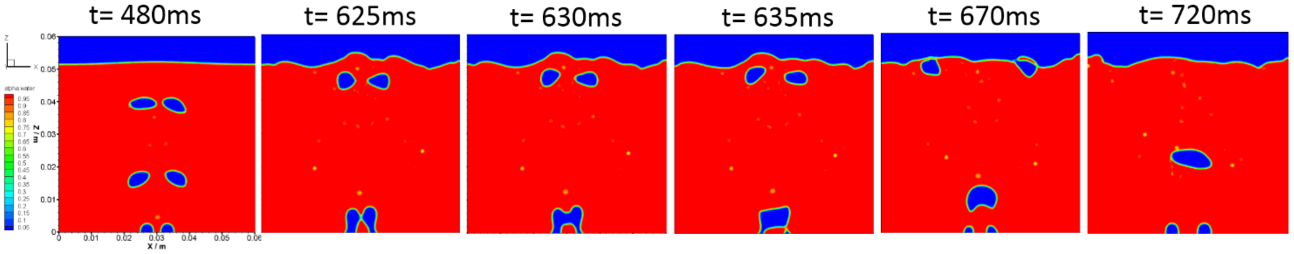
The evolution of bubbles behaviour from single submerged orifice can be summarized in Table.1. From above analysis, conclusion can been drawn that, in lower air flow rate cases (less than 200cc/min), the air bubble grows and rises as a single bubble without interactions with each other. Owing to the strong influence of the wake flow from the rising bubble, when the air flow rate is higher (larger then 500cc/min), bubble collision will occur between two or several successive bubbles. If the air flow is up to 1500cc/min, vertical coalescence will bring about near the bottom orifice. This is analogical with the boiling experiment results that, the vertical coalescence will only occur at high heat flux cases.

3.2 Bubble coalescences in horizontal direction

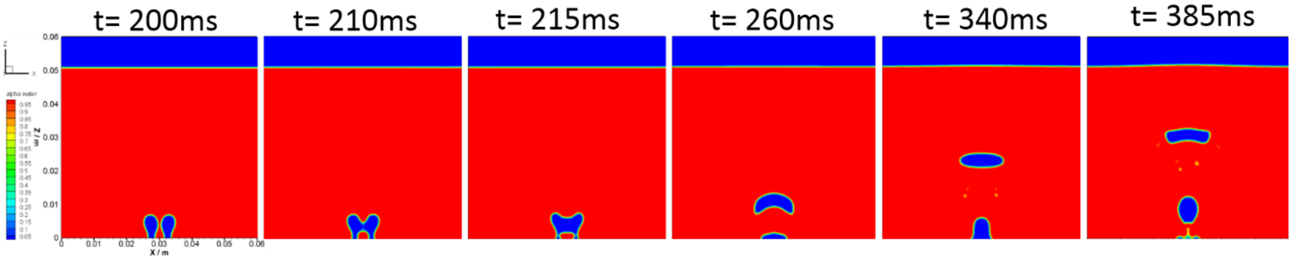
Based on Shoji's boiling experiments, the active sites spacing plays a decisive role in horizontal coalescence process. In current numerical experiments, double orifices has been set with three different spacing as shown in Figure 5. S/D is defined as the distance between the orifices divided by the single bubble departure diameter, as depicts in Figure 1. The bubbles grow and rise as isolated bubbles when S/D=1.67 without coalescence, which are similar to the bubbles in two single orifices, as shown in Figure 5(a). When S/D is 1.48 (Figure 5(b)) and 1.67 (Figure 5(c)), horizontal coalescence occurs at 625ms and 200ms, respectively. The results is consistent with the conclusion at al. [2] that horizontal coalescence couldn't occur if S/D>1.5. As coalescence occurs, two bubbles merged into one bigger bubble and departures form the orifices as a whole. However, it should been noted that, when S/D=1.48, after the coalescence bubble departure at 635ms, new bubble grows from each orifice as shown 670ms and 720ms in Figure 5(b). While in the case of S/D=0.93, coalesced bubble leaves the orifices at 215ms. After that, only one new bubble forms from two orifices as seen in Figure 5(c). The two inlets cooperate and offer air for the single bubble. The bubble grows for 170ms and departures at 385ms. The bubble growth period is less than the former coalesced bubble.



(a): S=9mm, S/D=1.67



(b): S=8mm, S/D=1.48



(c): S=5mm, S/D=0.93

Figure 5: The bubbles behaviour from double submerged orifices with different spacing

4 Conclusions

A numerical simulation has been presented with VOF model based on OpenFOAM to reveal the evolution mechanisms of bubble behaviours. Single bubble behaviours from one orifice with different air flow rate has been investigated first. Results show that, vertical coalescence and collision between successive bubbles only occurs in cases with larger air flow rate, which should been owing to the strong wake flow from the previous bubble. The effect of the spacing between two orifices on the air bubbles behaviour has been investigated subsequently. The conclusion can been drawn that, horizontal coalescence will not occur when S/D >1.5, supporting the results of boiling experiments recently. Although the current research is based on isothermal system without phase change, the results may be helpful to reveal the mechanism of the coalescence during boiling process.

Acknowledgements

This work is financially supported by National Key Technology R&D Program (NO.2013GB11005B).

References

[1] M. Shoji, R. Mosdorf, L. Zhang, Y. Takagi, K. Yasui, and M. Yokota, "Features of Boiling on an Artificial Surface—Bubble Formation, Wall Temperature Fluctuation and Nucleation Site Interaction," *J. Therm. Sci.*, vol. 11, no. 3, pp. 226-234, 2002.

[2] L. Zhang and M. Shoji, "Nucleation site interaction in pool boiling on the artificial surface," *Int. J. Heat Mass Transf.*, vol. 46, no. 3, pp. 513–522, 2003.

[3] L. Zhang and M. Shoji, "Aperiodic bubble formation from a submerged orifice," *Chem. Eng. Sci.*, vol. 56, no. 18, pp. 5371–5381, 2001.