Numerical simulation of the water bubble rising in a liquid column using the combination of level set and moving mesh methods in the collocated grids

Lin Bu a, Jiyun Zhao b, * 

a Energy Research Institute @ NTU (ERIAN), Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Singapore 
b Division of Power Engineering, School of Electrical and Electronic Engineering, 50 Nanyang Avenue, Singapore 639798, Singapore

A R T I C L E   I N F O

Article history:
Received 31 August 2011
Received in revised form 25 February 2012
Accepted 10 April 2012
Available online 14 May 2012

Keywords:
Level set method
Moving mesh method
Collocated grids
Bubble rising

A B S T R A C T

The proper interpretation of bubble rising in a liquid column is critical in the investigation of the water–vapour two phase flow and heat transfer. In this paper, the bubble behaviours are studied using the combination of Level Set Method (LSM) and moving mesh method in a collocated grid. Level set method is used to track the interface of the bubble, which has many advantages over other interface tracking methods such as it can accurately calculate the curvature of the interface and easily expand to three dimensions. But if the uniform grid was adopted, the bubble interface cannot adapt well with the grid which may cause great numerical errors. Hence, moving mesh method is implemented in this paper to increase the numerical accuracy. The numerical model developed in this paper is benchmarked with the experimental data. The results show that the grid distributions used in this paper can catch the interface continuously in space and time. The process of bubble starting from deformation until break into three small bubbles is clearly shown from the numerical results. In addition, the changing of bubble behaviour with the temperature and pressure is also investigated. It is found that as the pressure and temperature increase, the deformation process will slow down and this tendency accelerates.

© 2012 Elsevier Masson SAS. All rights reserved.

1. Introduction

Water either in single phase or water–vapour two phase is widely used as working fluid in the industry. Especially in the nuclear industry, more than 90% power generated by nuclear is from water-cooled nuclear reactors either Pressurized Water Reactor (PWR) or Boiling Water Reactor (BWR). Water boils inside the BWRs. Although water is in subcooled condition in the PWRs, the subcooled boiling occurs inside the reactor which makes the Departure from Nucleate Boiling (DNB) a critical safety concern for the operation of the PWRs. The supercritical water-cooled reactor (SCWR) is one of the six reactor types that are being investigated in the GEN-IV international advanced reactor development program. Although the water is in supercritical condition and no phase change or boiling will occur in the SCWR during the normal operation, the subcritical two phase flow conditions will occur during off-normal operations [1, 2]. Therefore, the proper interpretation of bubble behaviour is critical for investigation of two phase flow and heat transfer, and finally the safety of the nuclear reactors.

There are many studies of the bubble or water droplet behaviours under the atmospheric pressure both experimentally and numerically. Experimental method is one of the most important ways in the study of the bubble or water droplet behaviours. Hepbasli et al. [3] used the experimental method to study the bubble behaviour at the free surface of a large three-dimensional gas fluidized bed. Measurements were carried out to determine the effects of bed height and excess air velocity on the bubble eruption diameter, frequency and bubble fraction. All experiments were performed at freely bubbling mode and the flow characteristics of bubbles were recorded by a video camera. Lee et al. [4] explores experimentally bubble behaviours in a single trapezoid microchannel with a hydraulic diameter of 41.3 μm. Bubble nucleation, growth, departure size, and frequency are observed using a high-speed digital camera and analyzed by the Image-Pro. The size of bubble departure from the microchannel wall is found to be governed by surface tension and drag of bulk flow and may be fairly correlated by a modified form of Levy equation. Wang and Dong [5] designed a suit of visualized experimental device to study the bubble rising in stagnant liquid in a vertical translucent rectangular tank. The high-speed video system combined with digital image process methods was adopted. The movement of bubble rising in...
water was observed. And the behaviours of bubble were summarized with experimental analysing. The behaviour of a single large bubble flowing through a sudden constriction between a cylindrical pipe and a channel of rectangular cross section is studied experimentally by Pucar et al. [6]. Two types of constrictions are considered: an abrupt one and a smooth one. Image analysis displays the deformation of the large bubble generated in the upstream vertical pipe and flowing through each kind of constriction. Bai et al. [7] used a four-point optical fibre probe to study the bubble properties experimentally. The results show that the optical probe underestimates bubble properties, such as, bubble velocity, local void fraction. The presence of the probe in the bubble column influences the local flow conditions. Lucas et al. [8] use wire-mesh sensors to investigate stationary upward air–water flows in a vertical pipe with an inner diameter of 195.3 mm. Due to the combination of the new experimental procedure with the high spatial and temporal resolution of the wire-mesh sensor technology the data have new quality especially regarding their consistency in the evolution with increasing L/D. The wire-mesh sensors were used by Prasser et al. [9] to study the evolution of the bubble size distributions in a vertical two-phase flow. The different behaviours of small and large bubbles in respect to the action of the lift force were observed in a mixture of small and large bubbles. Marco et al. [10] measured the rising velocities of gas bubbles in a still liquid and compared the results with available theories. Bubble size, aspect ratio, detachment frequency, velocity and frequency of shape oscillations were measured by processing of high-speed video images.

Owing to the advances in numerical methods and as well as in computational hardware performance, numerical simulations have become increasingly popular in the study of complex multi-fluid systems. There are various numerical methods available in literature to compute bubble behaviour. Nagrath et al. [11] studied the three-dimensional computation of bubble behaviour using a Level set method. The continuum surface force (CSF) model was applied in order to account for surface tension effects. To restrict the interface from moving while re-distancing, an improved re-distancing scheme proposed in the finite-difference context is adapted for finite element discretization. Hence, the accurate computation can be carried out including the computation of the large viscosity difference and large density. A systematic computational study of the behaviour of gas bubbles rising in a viscous liquid is presented by Smolianski et al. [12]. The numerical method presented in this paper can simulate a wide range of flow regimes, accurately capturing the shape of the deforming interface of the bubble and the surface tension effect, while maintaining the mass conversation. Minsner et al. [13] studied the behaviour of a cavitation bubble in a viscous liquid near a solid surface. The effects of the viscosity were taken into account. Gupta et al. [14] used Lattice Boltzmann method (LBM) to study the behaviour of bubble motion and bubble coalescence in liquids. Simulations of bubbles with different sizes and in large periodic domain were studied. A single gas bubble rising in a narrow vertical tube is investigated via a numerical model on a 3-D axisymmetric computational domain by Wang et al. [15]. The transient governing equations are solved by a finite volume scheme with a two-step projection method. The interface between the liquid and gas phase is tracked by a coupled level set and volume-of-fluid (CLSVOF) method. It has been found that the bubble nose retains a relative stable shape while significant oscillations occur at the bubble bottom as it rises through the liquid from the static state.

Many other investigations on the bubble or water droplet behaviours exist in the literature, however, most of them were under the atmospheric pressure [3–15]. In the present paper, Level set method coupled with moving mesh method [16] was adopted to investigate the bubble behaviour in the subcritical water at different higher pressures. Level set method was first proposed by Osher and Sethian [17]. It is used to track the interface of the bubble in this paper. Level set method has the many advantages over other interface tracking methods. It can accurately calculate the curvature of the interface and the continuity of the interface is very well. It also can be easily expand to three dimensions. But if the uniform grid was adopted, the bubble interface cannot adapt very well with the grid, and great errors may appear. Hence, moving mesh method is implemented to increase the numerical accuracy in this paper. The mesh changes continuously in space and time to adapt to the dynamic changes of time-evolving solutions. The numerical simulation is conducted by solving the complete incompressible Navier–Stokes equations in an adaptive grid system, and the surface tension force is modelled by a continuum surface force approximation. During the computation, velocities of the fluids are corrected by using a velocity-correction formula, and the computation proceeds to convergence via a series of correction procedures so as to make the velocity fields satisfy the continuity of the fluids.

2. Numerical model

2.1. Physical model and basic assumptions

In this paper, the rising process of a saturated bubble under the different pressure conditions was studied. The model presented in this paper is for a 2D two-phase flow. The initial bubble is statically located at (0.1, 0.06), in the domain size 0.2 m × 0.2 m, as shown in Fig. 1. The initial radius of the bubble is 0.01 m. The gravitational acceleration is 9.8 m/s². At the initial state, the water and the steam
are in the thermal-equilibrium condition. The velocities on the walls are all set as zero, as shown in Fig. 1.

The model was based on the following assumptions:

1. Since the Reynolds number is very low, the flow can be treated as laminar flow.
2. Since the bubble and water are in thermal-equilibrium and the boundary conditions are adiabatic, the temperature changes are negligible in the whole system. Hence, the heat and mass transfer between the steam and the liquid was ignored in this paper.

The water properties are calculated using the polynomial equations, which are obtained by curve fitting the data calculated using the ASME properties [19]. The detailed process on the water properties calculations can be found in the Appendix of this paper.

2.2. Governing equations

The governing equations include the N–S equations and the Level set equations. All the equations were transformed into the curvilinear coordinates. Geometry coefficients, , , and the Jacobean of transformation, , may be used in the following equations. Therefore, they must be predefined as

\[
\alpha = x^2 + y^2 \\
\beta = x_\xi x_\eta + y_\xi y_\eta \\
\gamma = x_\eta^2 + y_\eta^2 \\
J = x_\xi y_\eta - y_\xi x_\eta
\]

Momentum equations can be transformed as follows [18]:

\[
\rho \left[ \frac{\partial u_\xi}{\partial t} + (u_\xi - \bar{x})u_\eta - (v_\xi - \bar{y})v_\eta \right] + u_\eta \left[ - (u_\xi - \bar{x})y_\eta + (v_\xi - \bar{y})x_\eta \right] \\
= -y_\eta p_\xi + y_\xi p_\eta + \frac{\mu}{\eta} \left( \alpha u_\xi - \beta u_\eta \right) \frac{\partial^2 u_\xi}{\partial \eta^2} + \frac{\mu}{\eta} \left( \gamma u_\eta - \beta u_\xi \right) \frac{\partial^2 u_\xi}{\partial \xi^2} + F_x^w \tag{5}
\]

\[
\rho \left[ \frac{\partial v_\eta}{\partial t} + (u_\xi - \bar{x})v_\eta + (v_\eta - \bar{y})v_\xi \right] + v_\eta \left[ - (u_\xi - \bar{x})y_\eta + (v_\xi - \bar{y})x_\eta \right] \\
= -x_\xi p_\eta + x_\eta p_\xi + \frac{\mu}{\eta} \left( \alpha v_\eta - \beta v_\xi \right) \frac{\partial^2 v_\eta}{\partial \xi^2} + \frac{\mu}{\eta} \left( \gamma v_\xi - \beta v_\eta \right) \frac{\partial^2 v_\eta}{\partial \eta^2} + F_y^w \tag{6}
\]

Where \( \bar{x} \) and \( \bar{y} \) represent the velocities of the grid in \( x \) and \( y \) coordinate, \( F_x^w \) and \( F_y^w \) represent the components of the body force in \( x \) and \( y \) coordinate.

The continuity equation can be transformed as follows:

\[
\rho \left( u_\xi \frac{\partial u_\xi}{\partial x} + u_\eta \frac{\partial u_\eta}{\partial y} \right) = 0 \tag{7}
\]

The Level set equations can be transformed as follows:

\[
\frac{\partial \phi}{\partial t} + \Phi_\xi \left[ (u_\xi - \bar{x})y_\eta - (v_\xi - \bar{y})x_\eta \right] + \Phi_\eta \left[ - (u_\xi - \bar{x})y_\eta + (v_\xi - \bar{y})x_\eta \right] \]

\[
= 0 \tag{8}
\]

\[
\frac{\partial \phi}{\partial t} = \text{sign}(\phi_0) \left[ 1 - J^{-1} \sqrt{\alpha \phi_0^2 + \gamma \phi_0^4 - 2 \beta \phi_0} \right] \tag{9}
\]

2.3. Grid generating technique

The mesh generation method adopted in this article was proposed by Ceniceros and Hou [18]. The grid generating equations can be written as follows:

\[
x_i = \nabla (w \nabla x), y_i = \nabla (w \nabla y) \tag{10}
\]

Where \( w \) is the monitor function. Using the semi-implicit discretization, we can get the following equations:

\[
\frac{x_i^{n+1} - x_i^n}{\Delta t} = a \Delta x_i^{n+1} + \nabla \cdot (w^n \nabla x) - a \Delta x^n \tag{11}
\]

\[
\frac{y_i^{n+1} - y_i^n}{\Delta t} = a \Delta y_i^{n+1} + \nabla \cdot (w^n \nabla y) - a \Delta y^n \tag{12}
\]

Where \( a = \max w^n, \Delta \) and \( \nabla \) are the operators of the Laplacian and the gradient with respect to \( (\xi, \eta) \).

Eqs. (13) and (14) can be discretized as follows:

\[
\left( \frac{2a}{\Delta \xi^2} + \frac{2a}{\Delta \eta^2} + \frac{1}{\Delta \xi} \right) x_{i,j}^{n+1} = \frac{a}{\Delta \xi^2} x_{i+1,j}^{n+1} + \frac{a}{\Delta \eta^2} x_{i,j+1}^{n+1} + \frac{a}{\Delta \eta^2} x_{i,j-1}^{n+1} + \frac{a}{\Delta \xi^2} x_{i-1,j}^{n+1} + \left( \frac{2a - 2w^n}{\Delta \xi^2} + \frac{2a - 2w^n}{\Delta \eta^2} + \frac{1}{\Delta \xi} \right) x_{i,j}^{n+1} \\
+ \left( \frac{-a + w^n}{\Delta \xi^2} + \frac{s_1}{2 \Delta \xi^2} \right) x_{i+1,j}^{n+1} + \left( \frac{-a + w^n}{\Delta \eta^2} - \frac{s_2}{2 \Delta \eta^2} \right) x_{i,j+1}^{n+1} + \left( -a + w^n + \frac{s_1}{2 \Delta \xi^2} \right) x_{i-1,j}^{n+1} + \left( -a + w^n + \frac{s_2}{2 \Delta \eta^2} \right) x_{i,j-1}^{n+1} \tag{13}
\]
2.4. Solving procedures

A collocated grid finite-difference scheme is used in the present study. The scalar parameters are defined on the nodes of the grids and a SIMPLER-based method is adopted to solve the solutions of velocities and the pressure. Cartesian velocity components are used as the independent variables in the momentum equations, while the contravariant velocity components are used as the independent variables in the continuity equation. The computation procedures are shown as follows in Fig. 2:

3. Numerical model benchmarking

In order to verify the accuracy of the numerical model developed in this paper, detailed comparison between the numerical

![Fig. 2. The computational procedure.](image)

![Fig. 3. Computational domain.](image)

![Fig. 4. The collapsing process of the water column.](image)
results and the experimental data are carried out. A typical benchmarking example is used in this paper. The computational domain is: 0.584 m × 0.584 m. The liquid column is static in the initial time. The height of liquid column is 0.292 m, and the width is 0.146 m, as shown in Fig. 3. Driven by the gravity, the liquid column starts to collapse and finally bumped against the wall. Then the liquid rises along the right wall. The properties of the vapour and the liquid are: $\rho_l = 1000$ kg/m$^3$, $\rho_v = 1$ kg/m$^3$, $\mu_l = 0.5$ kg/(m·s), $\mu_v = 0.5 \times 10^{-3}$ kg/(m·s), $\sigma = 0.0755$ (N/m).

The velocity boundary condition specified on walls in contact with the free surface is not well posed due to the contradiction of the moving free surface and a no-slip condition on the bounding wall. For alternative free surface contact wall boundary conditions have been adopted in the current work.

$$\nabla \cdot \mathbf{u} = 0$$  \hspace{1cm} (15)

The contact wall boundary problem has been overcome by adopting a slip boundary condition on the free surface contact walls such that the nonpermeability condition is maintained, whilst the tangential velocity is allowed to take a nonzero value by

![Graph](image1)

**Fig. 5.** The comparison of experimental results [20] and numerical results.

![Grid distributions](image2)

**Fig. 6.** Grid distributions at the different time intervals. (a) $t = 40$ ms (b) $t = 80$ ms (c) $t = 120$ ms.

![Level set profiles](image3)

**Fig. 7.** Level set profiles and the velocity fields for different time intervals. (a) $t = 40$ ms (b) $t = 80$ ms (c) $t = 120$ ms (d) $t = 160$ ms (e) $t = 200$ ms (f) $t = 240$ ms.
implementing a Neumann boundary condition at the wall. The boundary conditions were as follows:

\[
\begin{align*}
\mathbf{n} \cdot \mathbf{u} &= 0 \\
\mathbf{n} \cdot \nabla \mathbf{u} &= 0
\end{align*}
\]

(16)

Fig. 4 describes the collapsing process of the water column from the initial state. Martin and Moyce [20] reported the data of the \( x \) versus \( t \) based on the experiment. Fig. 5 shows the comparison of the experimental data and the numerical results obtained from the present paper. It can be seen that the experimental and the numerical results are matched well. Therefore, the numerical model developed in this paper is accurate and valid.

4. Results and discussion

The numerical simulation results of the bubble dynamic process at a temperature of 523.5 K (pressure of 4 MPa) are shown in Figs. 6 and 7. A sensitivity study is also performed for the different temperatures and pressures, and the results are shown in

\[
\begin{array}{cc}
\text{t=0ms} & \text{t=40ms} \\
\text{t=80ms} & \text{t=120ms} \\
\text{t=160ms}
\end{array}
\]

Fig. 8. Level set profiles at different initial temperatures. (a) \( t = 0 \) ms (b) \( t = 40 \) ms (c) \( t = 80 \) ms (d) \( t = 120 \) ms (e) \( t = 160 \) ms.
In all the conditions studied in this paper, the bubble will collapse. The big bubble will finally divide into three bubbles. Fig. 9 shows the bubble collapsing time versus temperature. As the initial saturated temperature increases, the bubble collapsing time will increase, in other words, the bubble needs longer time to collapse.

Fig. 10 shows the peak of the bubble $Y_{g\text{max}}$ versus the temperature. As the temperature increases, the peak of the bubble $Y_{g\text{max}}$ decreases and this tendency accelerates.

5. Conclusions

In order to investigate the bubble behaviour for the water-vapour two phase flow condition, level set coupled with moving mesh method is adopted to develop the numerical model in this paper. The model is benchmarked with the experimental data, which verified the accuracy and validity of the model. The bubble behaviour is then investigated using the model developed. The numerical results show that the grids can catch the bubble profiles precisely. It is shown that as the bubble rising in the liquid, both the left side and the right side of the bubble begin to shrink and the upside of the bubble begins to rise. The process continues as time going on until the big bubble collapses and breaks into three small bubbles. It is also found that as the temperature and pressure of the bubble increasing, the deformation process of the bubble will be slowing down, and the velocities of the bubble rising is decreased. It requires a longer time for bubble to collapse and the tendency accelerates if the temperature and pressure is increased.

Appendix A. The water properties calculations

The properties of the saturated water are calculated by the polynomial equations, which are obtained by curve fitting the data calculated using the ASME properties [19] based on IAPWS-IF 97, shown as follows:

$$
\rho_l = -116403.4569210326 - 1187.2268519176 T + 4.8523870257 T^2 - 0.009850072973 T^3 + 9.9295601755E-6 T^4 - 3.9805490227E-9 T^5
$$

(17)

$$
\rho_v = -107845.8604256314 + 1109.2000537937 T - 4.5297204456 T^2 + 0.0091844859 T^3 - 9.2506529969 E-6 T^4 + 3.7053629736 E-9 T^5
$$

(18)

$$
\mu_l = 0.0259376867 - 2.3823767295 E - 4 T + 8.9076924478 E - 7 T^2 - 1.6792748598 E - 9 T^3 + 1.5903634852 E - 12 T^4 - 6.0421265504 E - 16 T^5
$$

(19)

$$
\mu_v = -0.0088554269 + 9.1292780171 E - 5 T - 3.7214155045 E - 7 T^2 + 7.533699037 E - 10 T^3 - 7.574042816 E - 13 T^4 + 3.0264242733 E - 16 T^5
$$

(20)

The comparison of the properties of the saturated water computed from the equations and the IAPWS-IF 97 are shown in Fig. A1. It can be seen that the two results matched well.
Fig. A1. Comparison of saturated water properties (a) the density of saturated water versus the temperature (b) the density of saturated vapour versus the temperature. (c) the viscosity of saturated water versus the temperature (d) the viscosity of saturated vapour versus the temperature.

References