Annals of Nuclear Energy 63 (2014) 680–690

Contents lists available at ScienceDirect

Annals of Nuclear Energy

journal homepage: www.elsevier.com/locate/anucene

Three-dimension simulation of bubble behavior under nonlinear oscillation

Qingyun Zeng [a], Jiejin Cai [a,b,*]

[a] School of Physics and Engineering, Sun Yat-sen University, Guangzhou 510275, China
[b] Sino-French Institute of Nuclear Engineering and Technology, Sun Yat-sen University, Guangzhou 510275, China

ARTICLE INFO

Article history:
Received 3 May 2013
Received in revised form 6 September 2013
Accepted 11 September 2013
Available online 8 October 2013

Keywords:
Bubble behavior
Nonlinear oscillation
Numerical simulation
OpenFOAM

ABSTRACT

An air–water two-phase flow numerical simulation model under nonlinear oscillating condition is established based on the open source code OpenFOAM, and the bubble behavior under oscillating condition is simulated. The three-dimensional oscillating two-phase flow field is simulated using the VOF method and the moving grid of the Arbitrary Lagrangian–Eulerian (ALE) method. At first, the simulation results are compared with the experimental results and are shown to be in reasonable agreement, and the simulation model is validated. The validated model is then used to analyze the bubble motion. Bubble shapes, induced velocity and displacement versus time are investigated under different oscillating conditions. Rising bubbles under oscillation and non-oscillation conditions are also compared. It is found that a bubble, especially a rising bubble, is influenced significantly by oscillation.

1. Introduction

Gas–liquid two-phase flows related to thermal hydraulics are seen widely in various engineering fields, and bubble is the most common gas–liquid flow phenomenon, thus predictions of complicated bubble phenomena are of practical importance.

The analysis of bubble movement is extremely complicated. At first, most studies of bubble movement have been conducted experimentally. Experimental work of Grace et al. (1976) and later more advanced research by Bhaga and Weber (1981) are famous empirical studies of free rising bubble movement in Newtonian viscous fluid. They indicated that three non-dimensional numbers, Reynolds, Eotvos and Morton numbers, are significant in motion description during rising of bubble and determine the final shape and velocity of bubble.

Numerical researches are based on several interface capture methods such as the volume of fluid (VOF) by Hirt and Nichols (1981) and level set method by Osher and Sethian (1988) . After these capture methods were presented, more and more numerical studies are in progress to study bubble under wide variety of conditions.

However, flows in industrial system are often under nonlinear oscillating conditions. For instance, nuclear reactor oscillates under seismic condition and ocean condition, fluid flows in reactor components are externally oscillated and the integrity and coolability would be affected. In recent years, several studies have focused on the thermal hydraulics of a reactor under oscillating condition (Manera et al., 2005; Yan and Yu, 2009; Satou et al., 2010). An oscillating motion will introduce an additional acceleration and flow will be changed a lot because of different responds of gas and fluid, bubble behavior under this condition is still not well known, which affects thermal hydraulics and is meaningful for reactor safety.

In this study, three-dimension bubble motion in a container or channel under nonlinear oscillation is simulated numerically. Mainly two numerical approaches were performed by Watanabe (2012) for simulating reactor thermal hydraulics and bubble behavior under nonlinear oscillation due to seismic condition. One is to take the effect of oscillating container on flow into account an external force induced by the container motion in the momentum equation of fluid. Wei et al. (2011) numerically investigated bubble behaviors in subcooled flow boiling of water under the effect of additional inertial forces due to container motion considering energy and mass transfer during phase change based on this method. The other is to move calculation grid for fluid simulation directly according to the container motion and bubble moves because of its induction by fluid motion. This method has been widely used in sloshing problem or oscillating interface flow (Curadelli et al., 2010; Watanabe, 2011). Watanabe (2012) later simulated two-dimensional bubble motion under oscillating condition using this moving grid method and got the reasonable result. We (Cai et al., 2013; Zeng et al., 2013) also have preliminarily simulated two-dimensional bubble rising and its deformation under nonlinear oscillation with this method.

* Corresponding author. Address: School of Physics and Engineering, Sun Yatsen University, 135 West Xingang Road, Haizhu, Guangzhou 510275, China.
E-mail address: chiven77@hotmail.com (J. Cai).

0306-4549/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.anucene.2013.09.020

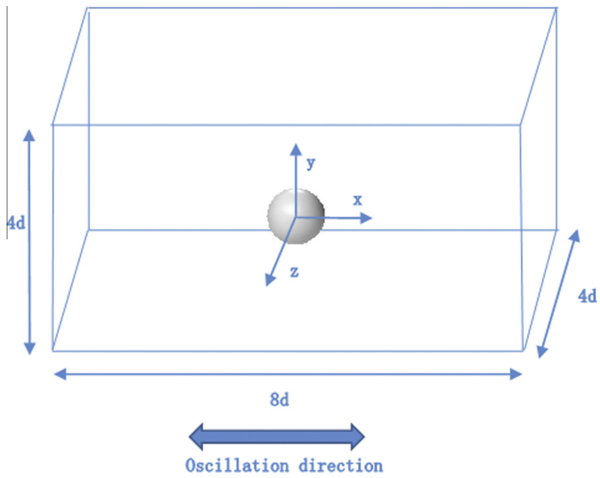

Fig. 1. The three-dimensional simulation domain in mesh resolution dependence tests.

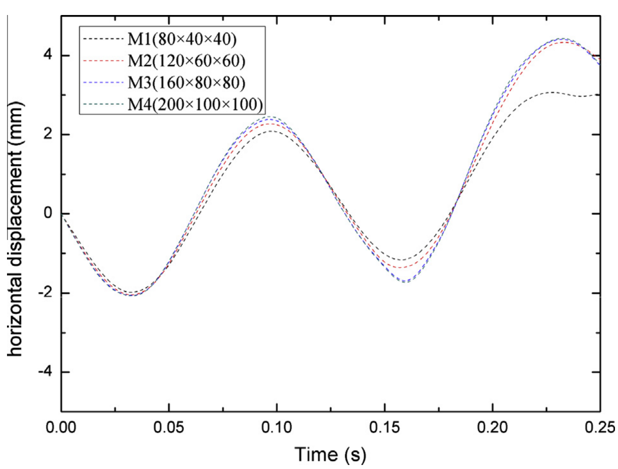

Fig. 2. Bubble displacements on different mesh resolutions.

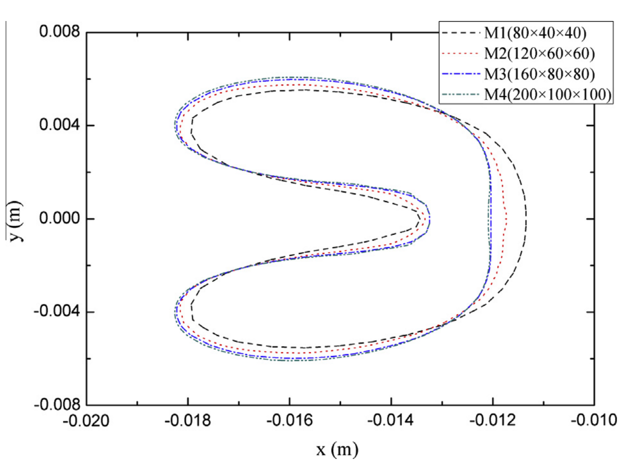

Fig. 3. Bubble shapes on different mesh resolutions.

Fig. 4. The experimental platform.

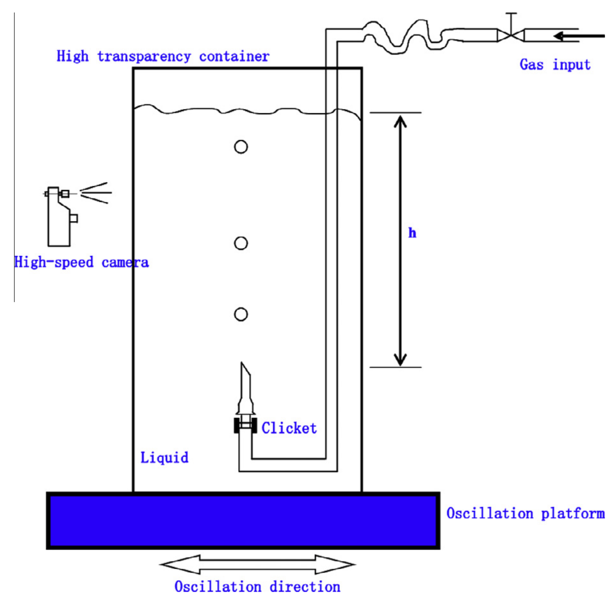

Fig. 5. The experimental sketch.

Though the external force method is easy and simple from the view point of numerical simulation, and could easily be applied for reactor safety analysis, it actually is used under some assumptions, whose validation has not been discussed well (Watanabe, 2012). The moving grid of the Arbitrary Lagrangian–Eulerian (ALE) method (Hirt et al., 1974) is used in this study, in order to learn the oscillating bubble behaviors corresponding to the real phenomenon.

Cases in this study are all simulated using OpenFOAM, an open source CFD package, which is based on the work of Weller et al. (1998). The interface is tracked by the VOF method, bubbles under different oscillating conditions are described and compared.

The remainder of this paper is organized as follows. Section 2 describes the numerical method; Section 3 gives some results and discussions and the conclusions are summarized in Section 4.

2. Numerical method

The two-phase flow is treated as incompressible and immiscible Newtonian fluid. Governing equations for the two-phase flow field are the equation of continuity and the incompressible Navier–Stokes equation for:

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \rho \mathbf{g} + \nabla \cdot \tau + \mathbf{f}_\sigma \tag{2}$$

where ρ, u, p and g, respectively, are the density, velocity, pressure and the gravitational acceleration, f_σ is the surface tension force, τ is