Chemical Engineering Science 168 (2017) 1–10

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Experimental study and numerical simulation of periodic bubble formation at submerged micron-sized nozzles with constant gas flow rate

Jian Zhang [a], Yong Yu [a,*], Chen Qu [a], Yu Zhang [b]

[a] School of Aerospace Engineering, Beijing Institute of Technology, Beijing 100081, China
[b] Medical Center, Tsinghua University, Beijing 100084, China

HIGHLIGHTS

- Bubble formation between μm and mm orifices are different under low gas flow rate.
- Under high gas flow rate, coalescence bubbling regimes are observed at μm orifices.
- The CFD can simulate the dynamic characteristics of bubbles at μm orifices well.

ARTICLE INFO

Article history:
Received 26 October 2016
Received in revised form 9 March 2017
Accepted 7 April 2017
Available online 8 April 2017

Keywords:
Bubble formation
Micron orifice
Experimental study
Numerical simulation
Bubble coalescence

ABSTRACT

Visualization experiments and numerical simulations were carried out to investigate the bubble dynamic behavior at the submerged micron orifice. The diameters of the orifices varied from 0.136 mm to 0.204 mm. The bubble formation process was recorded by a high-speed video camera. The detailed bubble characteristics were obtained through image processing and a following Matlab analysis. The outcomes indicate that under a low gas flow rate, the bubble grows and detaches individually, and the bubble formation progress can be differentiated into three stages: nucleation, stable growth, and necking. The differences between micron and millimeter level orifices are obvious at the initial level of bubble formation. At the micron level, the bubble is strongly influenced by capillary pressure and it causes a longer waiting time. It is also found that the bubble shape only depends on the instantaneous bubble volume and has no relation with the gas flow under a high flow rate. We observed that the coalescence bubbling regimes are different from the multi-period formation mechanism at the millimeter level. The final bubble volume demonstrates an index increasing law with the gas flow rate. Additionally, the volume-of-fluid method was used for numerical simulation of the bubble formation process. It indicates that numerical simulations can finely predict the dynamic features of bubbles against the experiments, under low gas flow rate conditions (0.95–4.83 ml/min).

1. Introduction

The formation process of gas bubbles at a submerged orifice has important significance in a variety of gas-liquid reactors and in the equipment widely used in process industries, such as gas-liquid stirred vessels, slurry reactors, bubble columns, separation equipment, and other gas-liquid contacting devices. A variety of parameters like orifice diameter, liquid and gas properties, gas flow rate, and nozzle or orifice materials play an important role in the bubble dynamic process. With regard to this situation, researchers at home and abroad have made considerable efforts to understand the bubble specific behavior.

A number of experiments have been conducted to provide intuitive and more in-depth analysis about bubbles, achieving significant advances. Scientists have been engaged in this area for decades. Buwa et al. (2007) studied the bubble dynamics on an orifice of 2–10 mm, combined with the numerical simulation. In their work, the effects of gas flow rate and orifice diameter on the bubbling regimes, in particular, along the transition from the period-1 to period-2 bubbling regime (with pairing or coalescence at the orifice opening) were investigated. Obtaining the same bubble behaviors, Gerlach et al. (2007) observed the transition from single periodic (SP) to double periodic (DP) in their numerical simulation.

* Corresponding author.
E-mail address: yuyong@bit.edu.cn (Y. Yu).

http://dx.doi.org/10.1016/j.ces.2017.04.012
0009-2509/

The consistency between the simulation results and experimental analysis further supported the reliability of the numerical simulation. Further experiments of various gas flow rates, different viscous liquids, and some other conditions have been carried out (Badam et al., 2007; Tufail and Sartorelli, 2000; Tufail and Sartorelli, 2002; Zhang and Shoji, 2001) to understand the characteristics of the bubble formation process. Hong et al. (1996) and Yang et al. (2007a,b) simulated the time-dependent flow field information around the bubble and the particle, exposing the mechanisms of bubble formation and the interactions among gas, liquid, and solid phases during the bubble-particle collision. Li et al. (2000) numerically studied the single bubble rising characteristics in a bubble column under a high-pressure condition. The result confirms that a higher pressure yields a smaller maximum stable bubble size in a bubble column. Computational fluid dynamics (CFD) simulations have been performed to study various aspects of the adiabatic bubble formation (Quan and Hua, 2008; Das and Das, 2009; Chakraboty et al., 2009; Ohta et al., 2011; Chakraborty et al., 2011) on orifices and nozzles and the results are encouraging. Several numerical simulations of bubble formation and dynamics in liquids have been conducted in recent years, in which a volume of fluid (VOF) model is used, based on the concept of a fractional volume of fluid to treat the complicated gas–liquid interface in the geometry. Zhou et al. (2011) used CFD in conjunction with the VOF method to simulate the processes of gas bubble emergence, growth, deformation, and detachment. This work primarily focused on the effects of wettability of the wall where the bubble emerged. The simulated results showed that the hydrophilic wall facilitated the release of bubbles while the gas coverage ratio increased and the dimensionless flow resistance coefficient decreased for the hydrophobic wall. Chakraborty et al. (2013) used a coupled level-set and VOF method to simulate the bubble floating upward in stagnant liquids and the bursting process at the free surface. Considerable efforts were made in this study for understanding the shape changes, velocity variation, and mechanism of the bubble.

In most of the above literature, the sizes of nozzles or orifices stay at the millimeter level. Few studies pay attention to the detailed characteristics of bubble formation on micron-sized nozzles and orifices, although this is a common phenomenon. Zhu et al. (2014) investigated the dynamic bubbling behavior in an orifice with a diameter of 54 μm. In the experiment, the influence of the gas flow rate on the bubbling period was analyzed using the fast Fourier transform (FFT); a critical gas flow rate (54 ml/min) was found, at which the bubbling state changed from single bubbling to non-interval bubbling. Vafaei et al. (2011, 2010) and Vafaei and Wen (2010) used the Young–Laplace equation to predict the bubble shape. The theoretical methods have been validated by comparing them with bubbles in experiments using a nozzle size from 0.11 mm to 0.84 mm.

This study investigates bubble formation at three micron-scale orifices submerged in liquid using a high-speed video camera to capture the transition of bubbles. The numerical simulation of bubble formation under a low gas flow rate was also carried out using the VOF model.

2. Experimental illustrations and simulation strategies

2.1. Experimental setup

The experimental setup used in this investigation is outlined in Fig. 1. It consists of the following three parts: a nozzle submerged in water, measurement system, and gas supply system. Three different sizes of nozzles are made of standard stainless steel needles and substrates, with the detailed dimensions listed in Table 1. A polishing process is applied at the surface of the nozzle to minimize irregularities and heterogeneities. The nozzle is submerged in a transparent container with a cross section of 200 × 30 mm. The container is made of plexiglass and filled with quiescent deionized water at an elevation of 20 mm and is open to the atmosphere under ambient temperatures. The airflow into the nozzle is supplied by a micro-syringe pump through a connected capillary tube. A high-speed camera and macro lens are used to capture the detailed evolution of the scale and shape of the bubbles. It is found that a shooting speed of 3000 fps and resolution of 512 × 1024 pixels are the ideal set-up for the experiments, which results in an optical magnification of about 0.004 mm/pixel. To improve the quality of the bubble images, a cool lamp is used to reduce the reflections in front of the bubbles and meet the requirement of camera exposure. The lamp, test section, and camera are placed in-line. The images are stored in a PC with the aid of a memory buffer. Finally, the bubble images are processed and examined using our in-house Matlab codes.

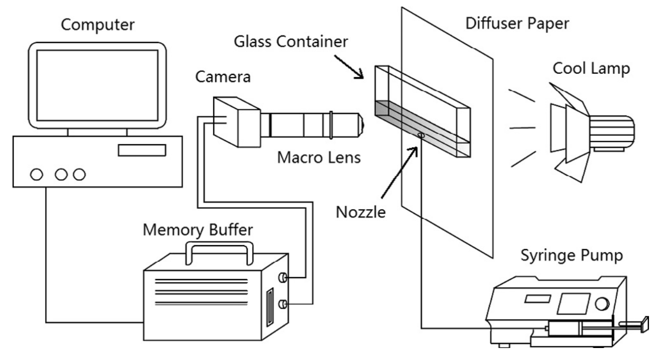

Fig. 1. Schematic of the experimental setup.

Table 1
Dimensions of nozzles.

Type	Inner diameter	Outer diameter
30G	0.136	0.318
28G	0.180	0.358
27G	0.204	0.406

2.2. Numerical simulation models

The Ansys-Fluent CFD software (www.ansys.com) is applied to perform the numerical calculation. The physical model is simplified as two-dimensional axisymmetric. The computational domain is presented in Fig. 2. To save computational time, the domain size

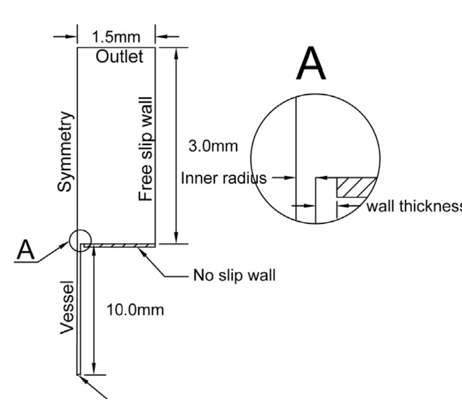

Fig. 2. Computational domain and boundary conditions.