Chemical Engineering and Processing 100 (2016) 79–86

Contents lists available at ScienceDirect

Chemical Engineering and Processing: Process Intensification

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

Dynamic characteristics of ventilated bubble moving in micro scale venturi

Xiaolei Li, Xiaowen Ma, Lei Zhang, Huichen Zhang*

Transportation Equipments and Ocean Engineering College, Dalian Maritime University, Liaoning 116026, China

ARTICLE INFO

Article history:
Received 19 August 2015
Received in revised form 27 October 2015
Accepted 16 November 2015
Available online 30 November 2015

PACS:
47.55.dd
47.55.df

Keywords:
Ventilated bubble
Venturi
Flow pattern
Bubble breakup

ABSTRACT

Under different inlet pressures and ventilated velocities, the moving process of ventilated bubble in micro scale venturi was recorded by high speed camera system. The effects of inlet pressure and ventilated velocity on bubble dynamics were analyzed in detail. The results show that the bubble breakup depends on the flow pattern. Bubble breaks up in turbulent flow but not in laminar flow. The intensity of bubble breakup becomes more severe with the increase of turbulent intensity. The main mechanism of bubble breakup in venturi is turbulence fluctuation and shearing-off process, the former happens in the condition of high inlet pressure and the latter exits when the inlet pressure is relatively low.

1. Introduction

Multi-phase flow, especially gas–liquid two phase flow, has been widely applied in many fields, such as chemical reaction catalysis [1,2], wastewater treatment [3], drag reduction [4] and biomedical sciences [5]. In the above applications, the dynamic characteristics of bubble are the key factors to realize these special functions. Therefore, the process of bubble generation and breakup has been widely studied [6–8].

The simplest way to generate bubble is to inject gas into water. In this method, the following elements should be taken into consideration, which are the effects of gas pressure to inject micro-volume gas [9], the structure of submerged orifices [10,11], and the nozzle geometry [12], etc. In addition, when the laser focuses into water, a laser bubble with good spherical shape appears [13]. The spark can also be used to generate bubbles. However, these bubbles' thermal behavior is very complex [14]. Another way to generate bubbles is pulse heating [15], in which the parameters of pulse heating can be adjusted for requiring bubbles.

The dynamics of bubble breakup has also attracted much attention of some researchers. Mukin [16] put forward a new model of bubble collapse in turbulent flows. Ye et al. [17,18] analyzed the bubble breakup in the compressible fluid field. Fu and Zhang [19] studied the bubble collapse in venturi cavitational reactor. Wang et al. [20] conducted researches on the burst of a rising bubble near a free surface theoretically and numerically by the volume-of-fluid method. He et al. [21,22] studied the effects of surface tension and viscosity on bubble breakup. Zhang et al. [23] explored the collapse pressure of bubble cloud in venturi.

Although many works about bubble dynamics have been conducted, its mechanism in micro scale is still unclear, especially in micro venturi, which is widely used in chemical reactors. In this paper, we focused on the movement process of ventilated bubble in micro scale venturi and revealed the mechanisms of bubble breakup, aiming to control and use this gas–liquid flow technology effectively.

2. Experimental techniques and numerical simulation

2.1. Description of the experimental configuration

A set of experiments were carried out at room temperature and under atmospheric pressure with the experimental system shown in Fig. 1. The micro channel included a T-junction where bubbles generated and a venturi assemble in which bubble movement were observed. In the T-junction, a syringe pump injected additional air into the distilled water continuously and the ventilate velocities (Q_g) were spanned between 50 ml/h and 400 ml/h. Then, the

* Corresponding author. Fax: +86 411 84725960.
E-mail address: hczhang@dlmu.edu.cn (H. Zhang).

http://dx.doi.org/10.1016/j.cep.2015.11.009
0255-2701/

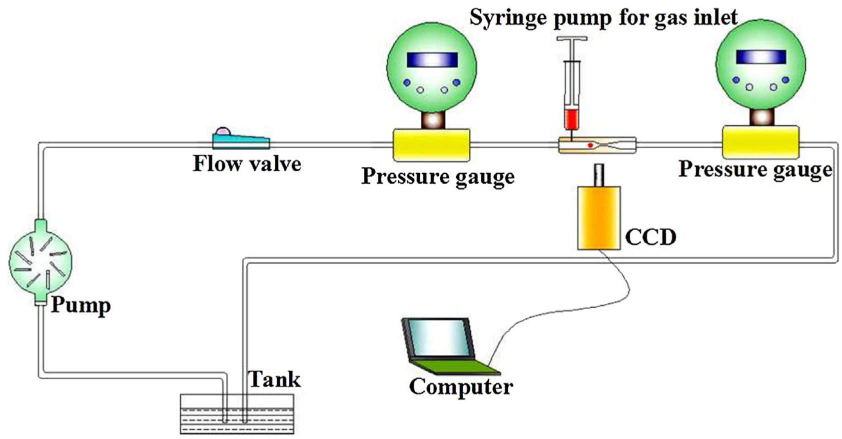

Fig. 1. Experimental system.

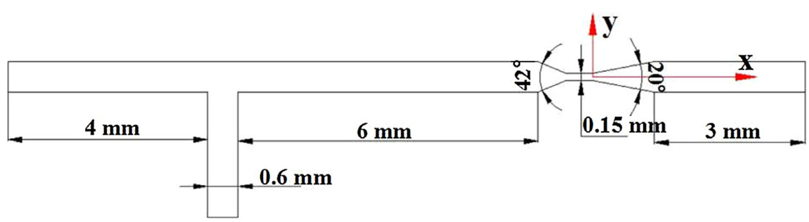

Fig. 2. Schematic of micro channel.

ventilated bubbles generated immediately. Upstream of the T-junction, we used a pressure gauge to test the inlet pressure of micro channel, and a flow valve was used to control the inlet pressure (P_1) which was set between 10 kPa and 40 kPa. Downstream of the venturi assemble, we used another pressure gauge to test the outlet pressure of micro channel. In addition, the volume flow rate of the whole fluid was measured by graduated cylinder and stopwatch in the outlet of channel. The structure of micro channel is shown in Fig. 2.

The size of channel's cross section is 600 μm (width) and 300 μm (depth) and the width of venturi throat is 150 μm. The micro channel was fabricated on a polydimethylsiloxane (PDMA) plate by a precision machine and sealed by another quartz plate. A high-speed camera (CCD, Hotshot 512sc) was placed above the micro channel to record the bubble generation and movement. The frame rate of CCD was set to be 2×10^4 fps and 3×10^4 fps, respectively.

2.2. Numerical model

In order to make a better understanding of the mechanisms of bubble breakup, the processes of a single bubble's movement in venturi part were simulated by Computational Fluid Dynamics (CFD) software Fluent 6.3. Firstly, the 2-dimensional venturi channel was meshed by Gambit which is the pre-processing software of Fluent (Fig. 3), and this region contains 180480 quadrilateral cells (Fig. 3). Secondly, according to the experimental condition, the inlet pressure P_1 was also set between 10 kPa and 40 kPa. The solver chose pressure based solver and unsteady time, the model selected the volume of fluid and k-epsilon. In the process of simulation, the gravity was also taken into consideration. After setting completed, we started the simulation. During the process of simulation, we kept a close eye on the residual curves which would show the stability of the flow field. Thirdly, when the flow field was steady, we would suspend the simulation and patch a bubble into the flow field, and then the simulation was continued. The diameter of patched bubble was d_0, which was set according to the real size of ventilated bubble generated in the experiments.

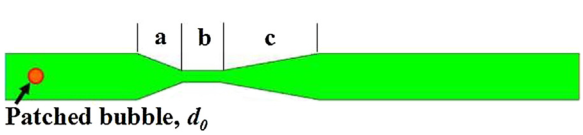

Fig. 3. Schematic of venturi part (a) contraction part; (b) throat part; (c) diffusion part.

3. Results and discussion

3.1. Flow patterns

The Reynolds number can be calculated according to the Reynolds number formula:

$$Re = \frac{\rho v L}{\mu} \quad (1)$$

where v is the average velocity, L is the hydraulic diameter, ρ is the density of liquid, and μ is the viscosity.

Under different inlet pressures, the corresponding volume flow rates and throat Reynolds are presented in Table 1. According to the rule of the engineering application, the critical Reynolds number R_{ecr} = 2000 is used to differentiate the flow pattern. When the inlet pressure (P_1) was 10 kPa, the flow was laminar. While, as

Table 1
Volume flow rates and throat Reynolds under different inlet pressures.

Inlet pressure P_1(kPa)	10	20	30	40
Volume flow rates (ml/s)	0.302	0.468	0.598	0.715
Throat Reynolds	1335	2069	2644	3161