Computers & Fluids 56 (2012) 49–60

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

Numerical simulation of bubble generation in a T-junction

S. Arias [a], D. Legendre [b,*], R. González-Cinca [c]

[a] Escola d'Enginyeria de Telecomunicació i Aeroespacial de Castelldefels, Universitat Politècnica de Catalunya, c/ Esteve Terradas 5, 08860 Castelldefels, Barcelona, Spain
[b] CNRS, IMFT, Université de Toulouse, INPT, UPS, Institut de Mécanique des Fluides de Toulouse (IMFT), 1 Allée du Professeur Camille Soula, F-31400 Toulouse, France
[c] Departament de Física Aplicada, Universitat Politècnica de Catalunya, c/ Esteve Terradas 5, 08860 Castelldefels, Barcelona, Spain

ARTICLE INFO

Article history:
Received 19 January 2011
Received in revised form 27 September 2011
Accepted 24 November 2011
Available online 2 December 2011

Keywords:
Two-phase flows
Bubble generation
Microgravity
T-junction
Numerical simulation
Volume of fluid method

ABSTRACT

We present a numerical study of the formation of mini-bubbles in a 2D T-junction by means of the fluid dynamics numerical code JADIM. Numerical simulations were carried out for different flow conditions, giving rise to results on the behavior of bubble velocity, void fraction, bubble generation frequency and length. Numerical results are compared with existing experimental data thanks to non-dimensional analysis.

1. Introduction

In the recent years, a growing interest in the study of gas–liquid flows has arisen as a consequence of their promising technological applications in space [1–3]. Replacing the widely used single-phase for two-phase systems could lead to an improvement in performance as well as to significant reductions in weight in different fields such as power generation and life support. A good understanding of the behavior of the gas–liquid interfaces is the cornerstone of these new technologies.

Bubble generation in low gravity environments is a key issue which requires an accurate control. This implies a good knowledge of the interface geometry and the generation of bubbles in a regular way with the smallest possible size dispersion. In this work we focus on the analysis of the formation of a train of bubbles by means of the cross flow generated in a capillary T-shaped junction [4–7]. In this bubble generator, gas is injected from a capillary into another capillary in a perpendicular direction in which liquid is flowing (see Fig. 1). We consider here the simplest case, in which both capillaries have the same circular cross-section of 1 mm i.d. Bubbles are generated as a result of the competition between the involved forces, being capillary forces predominant over inertia and buoyancy.

In order to explore the behavior of the T-junction bubble generator in a wide range of parameters, it is required a reliable numerical code which can complement experimental results. Different computational fluid dynamics methods have been recently used to study the generation of bubbles and droplets in this type or in similar devices. Qian and Lawal [8] used a commercial CFD package to simulate the bubble formation in the squeezing regime of a T-junction microchannel. Their work was focussed on the study of the effects of pressure, surface tension and shear stress action on the gas thread. Kashid et al. [9] discussed CFD modeling aspects of internal circulations and slug flow generation. The slug flow formation in a 120° Y-junction was simulated and velocity profiles inside the slug were obtained. More recently, De Menech et al. [10] carried out a numerical investigation by means of a phase-field model of the breakup dynamics of streams of immiscible fluids in a microfluidic T-junction. Three regimes of formation of droplets (squeezing, dripping and jetting) were identified and studied. In spite of the promising results obtained in these recent numerical works, important aspects in the flow characterization such as the bubble generation frequency or the void fraction distribution were not addressed.

The numerical code JADIM developed in the *Institut de Mécanique des Fluides de Toulouse* (IMFT) has been applied to a variety of fluid dynamics problems [11–20]. The Volume of Fluid (VoF) module of JADIM is able to perform local analyses of deformable two phase interfaces by resolving the Navier–Stokes equations for incompressible fluids in non-stationary problems. An Eulerian description of each phase is applied on a fixed grid and fluids are supposed to be Newtonian. The interface is calculated by means of the transport equation of the local volume fraction of one phase, being the surface tension constant and uniform along the interface in the absence of thermal exchange.

* Corresponding author.
E-mail address: dominique.legendre@imft.fr (D. Legendre).

0045-7930/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2011.11.013

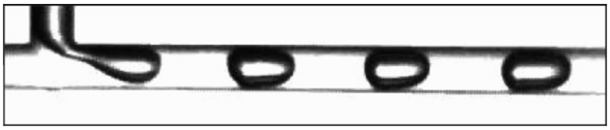

Fig. 1. Detail of the bubble generator. Gas is injected from the top and liquid from the left side.

In this paper we present a numerical study of the generation of millimetric bubbles in a T-junction by means of JADIM. In Section 2 a dimensional analysis of the bubble generation phenomenon is presented. The numerical code is presented in Section 3 and the modeling of the T-junction is presented in Section 4. Numerical results on the characteristics of the generated flows are presented and compared to existing experimental data in Section 5.

2. Problem statement

We consider a 2D T-junction bubble generator. The connection between the two channels as well as the flow directions are shown in Fig. 1. The problem is described using ten independent parameters, namely the gas and liquid densities (ρ_G and ρ_L, respectively) and viscosities (μ_G and μ_L, respectively), surface tension σ, capillary diameter Φ (the T-junction being formed by the connection of equal size capillaries), contact angle between the capillaries and the gas–liquid interface θ (measured on the internal part of the liquid), gravitational constant g, and gas and liquid superficial velocities (U_{SG} and U_{SL}, respectively), which are obtained from the air and water volumetric flow rates (Q_G and Q_L, respectively):

$$U_{SG} = \frac{Q_G}{A}, \quad U_{SL} = \frac{Q_L}{A}, \tag{1}$$

where A is the cross-sectional area of the capillary. Experiments were conducted at a constant temperature around 20 °C and the system can be assumed adiabatic. According to the Buckingham's π theorem, the system can be described by seven dimensionless parameters. The appropriate dimensionless numbers in our study are:

$$\begin{aligned} &\frac{\rho_L - \rho_G}{\rho_L} \quad Bo = \frac{\Delta\rho g \Phi^2}{\sigma} \\ &Re_{SL} = \frac{\rho_L \Phi U_{SL}}{\mu_L} \quad Re_{SG} = \frac{\rho_G \Phi U_{SG}}{\mu_G} \\ &We_{SL} = \frac{\rho_L \Phi U_{SL}^2}{\sigma} \quad We_{SG} = \frac{\rho_L \Phi U_{SG}^2}{\sigma} \end{aligned} \tag{2}$$

$$\theta$$

Any other dimensionless number should be obtained from the combination of the previous ones. Typically, the Capillary number $Ca = We/Re$ is used to compare viscosity and surface tension effects at the interface.

Experiments carried out in [5,6] are used as reference data for the comparison with the simulations reported here. In these experiments, air and water were mixed in a T-junction of two capillaries with 1 mm of internal diameter. The superficial velocities selected for the comparison with numerical simulations ranged from 0.106 to 0.531 m/s for water and from 0.081 to 0.344 m/s for air. We considered the following values of the physical properties: $\rho_L \simeq 10^3$ kg/m^3, $\rho_G \simeq 1.2$ kg/m^3, $\mu_L \simeq 10^{-3}$ Pa s, $\mu_G \simeq 10^{-5}$ Pa s and $\sigma \simeq 0.072$ N/m. According to these values, we obtain $\Delta\rho/\rho \approx 1$ and $Bo = 0.13$. The values of U_{SL}, U_{SG}, Re_{SL}, Re_{SG}, We_{SL}, We_{SG}, as well as the flow regimes observed in each experiment are shown in Table 1.

Table 1
Superficial velocities, dimensionless numbers and flow regime observed in each experiment.

U_{SL} (m/s)	U_{SG} (m/s)	Re_{SL}	Re_{SG}	We_{SL}	We_{SG}	Regime
0.106	0.242	106	24	0.16	0.81	Slug
0.106	0.344	106	34	0.16	1.64	Slug
0.318	0.081	318	8	1.40	0.09	Bubble-slug transition
0.318	0.242	318	24	1.40	0.81	Slug
0.318	0.337	318	34	1.40	1.58	Slug
0.531	0.068	531	7	3.92	0.06	Bubble
0.531	0.236	531	24	3.92	0.77	Bubble-slug transition

Table 2
Superficial velocities, dimensionless numbers and regime observed in each numerical simulation.

U_{SL} (m/s)	U_{SG} (m/s)	Re_{SL}	Re_{SG}	We_{SL}	We_{SG}	Regime
0.106	0.242	11	2	0.16	0.81	Slug
0.106	0.344	11	3	0.16	1.64	Slug
0.318	0.081	32	1	1.40	0.09	Bubble-slug transition
0.318	0.242	32	2	1.40	0.81	Slug
0.318	0.337	32	3	1.40	1.58	Slug
0.531	0.068	53	1	3.92	0.06	Bubble
0.531	0.236	53	2	3.92	0.77	Bubble-slug transition

In order to carry out the numerical simulations, some changes in the values of two dimensionless parameters (Re_{SL} and Re_{SG}) had to be considered. In case of taking the same values as in the experiments, the method used for the calculation of the surface tension contribution in the momentum equation, the Continuum Surface Force [21], generates the appearance of spurious currents (see next section for a detailed explanation). These currents induce vortices at the interface without any physical meaning, destabilizing the simulations and strongly distorting the interface [20]. Numerical instabilities produced by the spurious currents depend linearly on the ratio σ/μ. For the flow conditions considered here, gas and liquid viscosities had to be increased one order of magnitude in the simulations in order to avoid the spurious currents. Consequently, Re_{SL} and Re_{SG} were decreased one order of magnitude for the simulated flows, although both experiments and simulations were carried out at intermediate Reynolds numbers in the laminar regime.

We considered in the simulations $g = 0$ (thus, $Bo = 0$), while the values of We_{SL} and We_{SG} were the same as in the experiments. We also used the same geometry of the capillaries as well as the same gas and liquid superficial velocities as in the experiments. The latter was possible since the width of the capillary in the 2D simulations corresponds to the hydraulic diameter of the experimental T-junction. Under this assumption, the non-dimensional analysis remains valid and the two-phase flow behavior in the simulations is expected to be similar to the observed in the experiments. The superficial velocities, the values of Re and We, as well as the regime observed in each simulation, are shown in Table 2. The corresponding range of the Capillary number is $Ca_{SL} = 0.015 - 0.074$.

As regards to the contact angle used in the simulations, its value was chosen in agreement with the observations of the experimental videos (see Section 4.4).

3. Numerical code

The implemented VoF method in JADIM consists of an Eulerian description of each phase on a fixed grid, the interface between the two phases being calculated using the transport equation of the local volume fraction of one of the phases. The two fluids are assumed to be Newtonian and incompressible with no phase