Chemical Engineering Science 65 (2010) 2094–2107

Contents lists available at ScienceDirect

Chemical Engineering Science

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

CFD modelling of flow and heat transfer in the Taylor flow regime

Raghvendra Gupta, David F. Fletcher *, Brian S. Haynes

School of Chemical and Biomolecular Engineering, The University of Sydney, Sydney, NSW, Australia

ARTICLE INFO

Article history:
Received 18 September 2009
Received in revised form
23 November 2009
Accepted 3 December 2009
Available online 11 December 2009

Keywords:
Taylor flow
Microchannel
Multiphase flow
Heat transfer
CFD
Slug flow

ABSTRACT

Transport phenomena in the Taylor flow regime for gas–liquid flows in microchannels have received significant attention in recent years. Whilst the hydrodynamics and mass transfer rate in the Taylor flow regime have been studied extensively using experimental and numerical techniques, studies of heat transfer in Taylor flow have been neglected. In this work, the flow and heat transfer in this regime is studied using the volume of fluid (VOF) and level-set techniques to capture the gas–liquid interface, as implemented in the ANSYS Fluent and TransAT codes, respectively. The results obtained from the two different codes are found to match very closely. Fully-developed flow and heat transfer are studied using the VOF method for a Reynolds number (*Re*) of 280, Capillary number (*Ca*) of 0.006 and homogeneous void fraction (β) of 0.51 for constant wall heat flux (H) and constant wall temperature (T) boundary conditions. The Nusselt numbers obtained for both cases are 2.5 times higher than those for liquid-only flow. The effects of the mixture velocity and the homogeneous void fraction on flow and heat transfer are also studied.

1. Introduction

Two-phase gas–liquid flow in microchannels may result in many different flow patterns, such as bubbly flow, slug flow, slug-annular flow, annular flow and dispersed flow depending upon the gas and liquid flow rates, channel geometry and the properties of the two fluids (Shao et al., 2009). Amongst these flow patterns, slug flow, also known as Taylor flow, plug flow, segmented flow, bolus flow, intermittent flow and bubble-train flow, is a particularly important regime. Slug flow is characterised by the occurrence of regular gas bubbles almost filling the channel, separated by liquid slugs. The liquid slugs are connected by a thin liquid film surrounding the gas bubbles. The large interfacial area obtained in the slug flow regime enhances the gas–liquid mass transfer and the thin liquid film enhances the rates of heat and mass transfer from the channel wall to the liquid slug. The gas bubble and the liquid slugs have internal recirculating flow which promotes mixing and increases transfer rates considerably. Due to these characteristics, slug flow has found applications in a wide range of industries (Günther et al., 2004) and consequently, the transport phenomena occurring in this flow regime in micro-channels are a very active area of research. The hydrodynamics of slug flow has been studied by various researchers experimentally, as well as computationally (Taylor, 1961; Bretherton, 1961; Suo and Griffith, 1964; Thulasidas et al., 1995; Kreutzer et al., 2005; Angeli and Gavriilidis, 2008). Mass transfer and reactions in the slug flow regime in microchannels have also been widely studied (Berčič and Pintar, 1997; Heiszwolf et al., 2001; Kreutzer et al., 2001; van Baten and Krishna, 2004; Kreutzer et al., 2005; van Baten and Krishna, 2005; Onea et al., 2009).

Very few researchers have studied heat transfer without phase change in the gas–liquid slug flow in microchannels using experimental or computational techniques. Prothero and Burton (1961) studied experimentally the heat transfer in air–water slug flow in capillaries and found slug flow to be twice as effective in transferring heat as liquid-only flow. Oliver and Wright (1964) and Oliver and Young Hoon (1968) studied heat transfer in the gas–liquid slug flow in a channel of diameter 6.4 mm for Newtonian and non-Newtonian liquids and reported that the heat transfer enhancement for the two-phase slug flow is up to two and half times that obtained for the liquid-only flow. Monde et al. (1989, 1995) studied the effect on heat transfer of injecting long air bubbles into subcooled water and ethanol in a vertical rectangular channel of cross-section of 2 mm × 20 mm and found the heat transfer coefficient to be five times larger than that without the bubbles for the experimental conditions studied. Bao (1995) and Bao et al. (2000) investigated convective heat transfer experimentally for air–water flow in a circular channel of 1.95 mm diameter for a constant wall heat flux over a large range of gas and liquid flow rates encompassing slug (Taylor) flow, slug-annular flow and annular flow regimes. They found that the heat transfer was enhanced considerably by the presence of the gas, especially at higher gas and liquid velocities. A sharp jump in the heat transfer rate was observed after a particular gas velocity, which the authors believed corresponded to a flow regime transition to slug-annular flow. Hetsroni et al. (2009) studied

* Corresponding author.
E-mail address: d.fletcher@usyd.edu.au (D.F. Fletcher).

0009-2509/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ces.2009.12.008

flow and heat transfer for air–water flow in triangular channels of hydraulic diameter 130 μm and reported that the heat transfer coefficient increases with increasing liquid velocity and decreases with increasing air velocity. However, the values of the Nusselt numbers reported are significantly lower when compared with those for the laminar liquid-only flow, which may relate to an inconsistent definition of the wall and bulk temperatures. Walsh et al. (2009) studied experimentally the heat transfer in laminar slug flow in a channel of 1.5 mm diameter for a constant wall heat flux boundary condition and found that the Nusselt number increases two-fold when the slug length is decreased from 15*d* to 2*d* for the same homogeneous void fraction.

Duda and Vrentas (1971) obtained an analytical solution for unsteady heat transfer to a liquid at low Reynolds number in a cylindrical cavity with recirculating flow induced by relative motion between the fluid and the constant-temperature wall. They showed that the circulation currents can cause substantial heat and mass transfer enhancement if the Péclet number of the system is large enough. This can be thought of as a limiting case of slug flow with flat bubble ends and no liquid film surrounding the bubbles.

Recently, Fukagata et al. (2007) and Lakehal et al. (2008) have shown computationally that the Taylor flow in microchannels can significantly enhance the heat transfer. They used the level-set method to capture the gas–liquid interface. Fukagata et al. (2007) studied the flow and heat transfer (without phase change) in a periodic computational domain for a channel of 20 μm diameter using a constant wall heat flux boundary condition and found the Nusselt number to be up to about twice that for single phase flow for the flow conditions studied. Lakehal et al. (2008) investigated the convective heat transfer for gas–liquid flow in a microchannel in the slug flow regime for a constant wall temperature boundary condition and reported that the presence of gas bubbles increases the heat transfer three to four times above that of the liquid-only flow.

The above review of the existing literature suggests that heat transfer in the slug flow regime is enhanced significantly compared with that for liquid-only flow. The heat transfer mechanism and dependence of Nusselt number on flow parameters in such a flow regime is not well understood and this paper therefore addresses this issue through computational fluid dynamics (CFD) simulations. In previous work (Gupta et al., 2009), a methodology to simulate the hydrodynamics of Taylor flow in microchannels using the VOF method with geometric reconstruction, as implemented in ANSYS Fluent, was developed. Here, we extend this work to include heat transfer with a view to understanding the underlying mechanism of heat transfer. We first model the flow and heat transfer using two different CFD software packages, Fluent and TransAT, which employ different interface capturing methods. A careful comparison between the results obtained from the two CFD codes is made. Then, using Fluent, the modelling is extended to a longer domain so that fully-developed flow is obtained. In the following sections, the effects of the variation of the mixture velocity (the sum of the gas and the liquid superficial velocities) and the homogeneous void fraction (the ratio of the gas superficial velocity to the mixture velocity) on heat transfer are studied.

2. The CFD model

Multiphase flow in microchannels can be modelled using various interface capturing techniques, such as volume of fluid (VOF) (Hirt and Nichols, 1981), level-set (Sussman et al., 1994), marker points method (Tryggvason et al., 2007) or the phase-field method (Jacqmin, 1999). In the present work, the VOF method with geometric reconstruction (Youngs, 1982) and the level-set method have been employed to model two-dimensional, axisymmetric slug flow and heat transfer without phase change in a microchannel. Fluent (ANSYS, 2009), a widely used commercial CFD software is used to perform the VOF simulations, while TransAT (2009) a high-accuracy CFD code is used for the level-set simulations.

2.1. Governing equations

For simulations using the interface-capturing methods, a "single-fluid" formulation is applied and common velocity and temperature fields are shared by all the phases. Therefore, the physical processes of heat and fluid flow can be described by a single set of mass, momentum and energy conservation equations. An additional advection equation for a colour function is solved to capture the gas–liquid interface.

$$\text{Continuity}: \quad \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{v}) = 0 \tag{1}$$

$$\text{Momentum}: \quad \frac{\partial (\rho \boldsymbol{v})}{\partial t} + \nabla \cdot (\rho \boldsymbol{v} \otimes \boldsymbol{v}) = -\nabla P + \nabla \cdot (\mu(\nabla \boldsymbol{v} + \nabla \boldsymbol{v}^T)) + \boldsymbol{F}_{SV} \tag{2}$$

$$\text{Energy}: \quad \frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho \boldsymbol{v} h) = \nabla \cdot (k \nabla T) \tag{3}$$

$$\text{Colour function}: \quad \frac{\partial C}{\partial t} + \boldsymbol{v} \cdot \nabla C = 0 \tag{4}$$

where $\boldsymbol{v}$ denotes the velocity vector, P the pressure, T the temperature, e the internal energy and h the enthalpy of the fluid. C is the colour function, with different functions being used in the different interface-capturing methods. A volume fraction (α) of one of the phases is used as a colour function in the VOF method; a level-set function (ϕ) representing the signed distance from the interface is used as a colour function in the level-set method and the volume fraction (α) is in turn calculated from a smoothed Heaviside function. The bulk properties of the fluid, such as the density (ρ), the thermal conductivity (k) and the dynamic viscosity (μ) are calculated as the volume-fraction-weighted average of the bulk properties of the two fluids. The effect of gravity is neglected here as it plays a minor role in many multiphase flows in microchannels (Triplett et al., 1999a). Liquid evaporation is neglected and it is assumed that the properties of the two fluids remain constant. These assumptions are valid because there is only a small variation in the fluid temperature in the simulations considered and the temperatures are too low for significant evaporation to occur. The surface tension force is approximated as a body force $\boldsymbol{F}_{SV}$ in the vicinity of the interface (Brackbill et al., 1992) and is calculated as follows:

$$\boldsymbol{F}_{SV} = \sigma \kappa \delta (\boldsymbol{r} - \boldsymbol{r}_{int}) \boldsymbol{n} \tag{5}$$

where,

$$\boldsymbol{n} = \frac{\nabla C}{|\nabla C|}; \quad \kappa = \nabla \cdot \boldsymbol{n} \tag{6}$$

2.2. Boundary and initial conditions

At the inlet, the gas enters at the axis of the tube occupying the inlet area in proportion to the homogeneous void fraction and the liquid enters as an annulus around the gas core. A uniform velocity profile for both the phases and a constant temperature of the gas and the liquid are assumed at the inlet. In Fluent, a pressure outlet boundary condition is applied at the exit. In