International Journal of Thermal Sciences 86 (2014) 115–124

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Numerical simulations of heat transfer characteristics of gas–liquid two phase flow in microtubes

Y.S. Lim [a, *], S.C.M. Yu [b]

[a] Division of Aerospace Engineering, School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Singapore
[b] Academic Division, Singapore Institute of Technology, Singapore 179104, Singapore

ARTICLE INFO

Article history:
Received 5 July 2013
Received in revised form
3 July 2014
Accepted 3 July 2014
Available online 28 July 2014

Keywords:
Numerical simulation
Two phase flow
Flow visualization
Heat transfer
Micro-channel

ABSTRACT

Numerical simulations of the heat transfer characteristics in axisymmetric air–water two phase flow have been carried out in microtubes of inner diameter 300 and 500 μm. The two phase flow is achieved by injecting nitrogen gas coaxially through a centrally positioned tube to the continuous liquid phase flow. This arrangement can produce a series of bubbles encapsulated by the continuous water phase. Uniform heat flux is applied on the outer surface of the outer tube. Comparison of the simulated bubbly flow and flow visualization bubbly flow results obtained from experiments show that the difference is within 10%. Subsequent simulation results show that the Nusselt number enhancement can be as high as 200% while the two phase frictional pressure loss for the bubbly flow is about 20% higher than that of the liquid flow alone. The results also show that the heat transfer performance varies with the bubble size, frictional pressure drop and Reynolds number. Analysis of the velocity and temperature profiles near a bubble shows that the bubble obstructs the path of the liquid flow, forcing the redistribution of the axial and radial velocities around the bubble. This redistribution enhances the thermal mixing and is found to be the main reason that enhances the heat transfer performance.

1. Introduction

The changes in human living patterns and working activities have led to the high demand for smaller, faster and improved functionality of the electronic devices. Miniaturization turns out to be one of the solutions for this. However, miniaturizing the electronic components also means that the density of the transistors residing on a compact chip will become much higher and thus, the heat flux generated on that particular spot will increase tremendously. Hence, to avoid overheating, efficient heat removal methods become crucial. Microchannel, which has relatively high interface between fluid and wall for high heat removal capability have drawn lots of attention in the past decade.

Tuckerman and Pease [1] were among the first to conduct experiments on the microchannel heat sink. They reported that the heat flux removed from the heat sink could be as high as 790 W/cm^2 accompanying by pressure loss of about 2×10^5 Pa. Since then, heat transfer enhancement using microchannel has been widely acknowledged and being utilized on many cooling applications [2,3]. Furthermore, various technologies for the fabrication of microchannel heat exchangers have also been explored [4].

Despite the fact that single phase liquid convective flow has successfully achieved high magnitude of heat transfer rate, in order to achieve even higher heat transfer performance, extensive research has also been carried out on multiphase flow such as the flow boiling and condensation in microchannels. Hetsroni et al. [5] found that under certain flow and heat transfer conditions the explosive boiling, where bubble grew rapidly, would occur in the micro-channels and the alternative occurrence of single phase liquid flow and two phase liquid vapor flow in the channels would cause the pressure drop to fluctuate. This pressure fluctuation could potentially damage the device and cause unstable working condition. Zhang et al. [6] experimentally examined the phase change phenomena in silicon channels with internal diameter ranging from 27 to 171 μm. They concluded that the flow boiling mechanism was largely dependent on the channel size. Nucleate boiling was expected to appear in channels larger than 100 μm and explosive boiling would appear in channels smaller than 50 μm. The pressure pulses occurred at the onset of boiling was expected to affect the local boiling and equilibrium conditions. Schilder et al. [7] performed visualization experiment of single phase and two

* Corresponding author.
E-mail addresses: limy0170@e.ntu.edu.sg, yaushiang@hotmail.com (Y.S. Lim).

http://dx.doi.org/10.1016/j.ijthermalsci.2014.07.001
1290-0729/

phase flow boiling on inner diameter 0.6 mm tubes. Their experimental results showed that local Nusselt numbers for flow boiling could be increased by 3–8 times higher than the single phase flow in a micro-channel.

In view of the great complexities and instability of the flow boiling [8], non-boiling two phase segmented flows are still being considered favorably. Xu et al. [9] proposed a design of parallel microchannels which were separated into several sections by intermittent transverse chambers. Fluids in all channels would be converged and remixed when flowing through these chambers. Therefore fluid with uniform temperature would generate a new thermal developing boundary layer while reentering the downstream microchannels.

Although microchannels can effectively enhance the heat transfer performance, an essential parameter which must also be taken into account is the undesired pressure drop. This is because the increment of heat transfer rates could be easily offset by the pumping power required to overcome the pressure loss.

In recent years, researchers have been dedicated to study the heat transfer of microchannel with computational fluid dynamics (CFD). The powerful visualization capabilities enable the evaluation of the flow and heat transfer around the gas bubble. Gupta et al. [10] and Talimi et al. [11] had performed detailed analysis on the heat transfer increase around the gas slug region. In addition, Horvath et al. had conducted the numerical investigation for the two phase flow at different Reynolds numbers. Their results were limited to $\alpha_G = \alpha_L = 0.5$, where α_G and α_L are the void fraction of the gas and liquid phase, respectively. However, the numerical simulation results were not compared with the experimental results especially on the effect of flow patterns to the heat transfer characteristic.

Recently, Lim et al. [12] performed the experimental investigation for the two phase flow heat transfer in circular microtube with the size of 300 μm. Their experimental results show that bubbly flow has a better overall heat transfer enhancement over the slug flow yet the optimal bubble size for better heat transfer performance were not fully identified. Furthermore, the liquid velocity profile and bulk fluid mixing mechanism were not discussed. Hence the objectives of this paper are to identify the mechanisms of heat transfer enhancement for air water co-axial flow in microtubes and to provide information for its behavior over various operating parameters.

2. Simulation approach

2.1. Numerical modeling

Fig. 1 shows the schematic configuration of the two co-axial circular mircotubes for the present investigation. The details of the experimental setup have already been described in details by Lim et al. [12]. r_i and w_i are the inner radius and the wall thickness of the center microtube, respectively. Similarly, the radius and the wall thickness of the outer tube are denoted as r_o and w_o. The thickness of the coating film is indicated as w'. Dispersed gas phase is introduced into the continuous liquid phase through the center microtube. Since both microtubes are axisymmetric, the computation domain can be reduced to half of the cross section in the flow direction, as shown in Fig. 2.

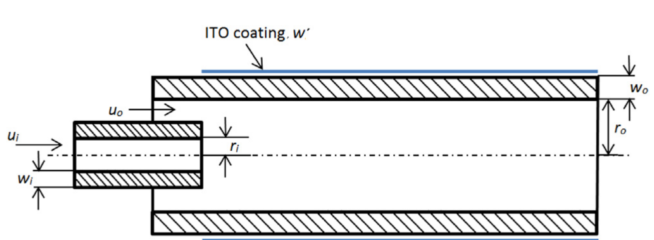

Fig. 1. Schematic of co-axial flow setup.

The effect of the wall thickness of the inner and outer tube have been considered while simplifying the computation model. The influence of the inner tube wall is investigated by comparing the cases of zero and non-zero wall thickness. Ratio of bubble diameter to the distance between the center of two bubbles are compared at different velocity ratios. Fig. 3 shows that the error between the zero and non-zero inner wall thickness is within ±0.03 as long as the inner diameter of the zero and the non-zero wall model are the same. Hence, in Fig. 2, R_i is equivalent to r_i and the inner tube wall thickness w_i is reduced to zero.

The effect of the outer tube wall thickness can be examined by comparing the wall temperature of the zero and non-zero wall model at a given wall heat flux. The wall temperature of zero and non-zero wall model is shown in Fig. 4. The outer tube wall thickness of the non-zero wall is 1/4 of the tube inner diameter and the wall material is aluminum which has high heat conduction coefficient. The difference of the wall temperature between the zero and non-zero wall model is found to be less than 7%. The overall wall temperature for the non-zero wall would be slightly higher than the zero wall model due to the axial conduction along the channel wall. The axial conduction number, $M = q''_{\text{cond}}/q''_{\text{conv}}$, defined by Maranzana et al. [13], has been used to determine the axial conduction effect. For all cases presented in this paper, the M is much less than the criteria of 0.01 if the tube is made of glass. Hence the axial conduction effect in the present paper is neglected. Since our main objective is the overall of the heat transfer enhancement instead of details in some local area, the zero wall thickness model can evaluate the overall Nusselt number as accurate as the model included wall thickness. Therefore, the geometry of computational domain used in the present simulation consists only of two microtubes with zero wall thickness.

The outer microtube length is constructed sufficiently long (60 times of R_o) to ensure that the flow and bubbles are fully developed. Inlet velocity for the two phases are kept to constant and denoted as u_i and u_o. The physical properties employed for the gas and liquid phase are assumed to be constant, i.e., $\rho_L = 1000$ kg/m^3, $\rho_G = 1.225$ kg/m^3, $C_{P,L} = 4182$ J/kg K, $C_{P,G} = 1006.43$ J/kg K, $k_L = 0.6$ W/m K, $k_G = 0.024$ W/m K and $\sigma = 0.072$ N/m, where the subscript G denotes the gas phase and L denotes the liquid phase.

2.2. Governing equations and boundary conditions

The assumptions made for the present simulation are:

- The fluid does not undergo phase change or no phase change will occur
- The flow is unsteady, laminar and incompressible
- Non slip boundary condition is applied at the wall
- Heat transfer at the outer wall is uniform
- Thermo physical properties are constant

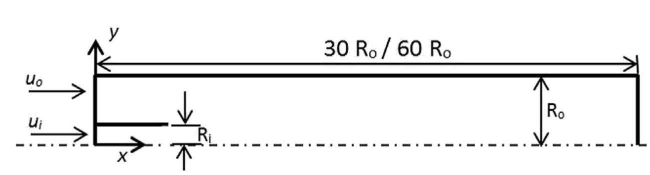

Fig. 2. Computational domain.