International Journal of Heat and Mass Transfer 86 (2015) 455–464

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Three dimensional features of convective heat transfer in droplet-based microchannel heat sinks

Zhizhao Che [a,*], Teck Neng Wong [a], Nam-Trung Nguyen [b], Chun Yang [a]

[a] School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, 639798 Singapore, Singapore
[b] Queensland Micro- and Nanotechnology Centre, Griffith University, 170 Kessels Road, Brisbane, QLD 4111, Australia

ARTICLE INFO

Article history:
Received 9 November 2014
Received in revised form 3 March 2015
Accepted 9 March 2015
Available online 27 March 2015

Keywords:
Droplet
Heat transfer
Thermal boundary layer
Multiphase microfluidics

ABSTRACT

Convective heat transfer in droplet-based microchannel heat sinks can be enhanced by the recirculating vortices due to the presence of interfaces. In rectangular microchannels, the three dimensional structures of the vortices and the 'gutters' (i.e., the space between the curved droplet interface and the corner of the microchannel) can significantly affect the heat transfer process. Numerical simulations of the heat transfer process are performed to study the three dimensional features in droplet-based microchannel heat sinks. The finite volume method and the level set method are employed to simulate the flow dynamics, the evolution of the interface, and the heat transfer. The results show that the 'gutters' can hinder the heat transfer process because of its parallel flow, whereas the recirculating flow in droplets and in slug regions between successive droplets can enhance the heat transfer by advecting hot fluid towards the center of the droplets/slugs and advecting fresh fluid towards the wall of the channel. The effects of the length of droplets, the aspect ratio of the channel cross sections, and the Peclet number are analyzed.

1. Introduction

High-efficiency cooling is important in numerous applications, such as in microprocessors with very-large-scale integration and in commercial/military high-power optical/electronic systems. The heat generated by such devices must be removed rapidly to ensure the reliability of the devices. Despite many techniques have been developed for high-efficiency cooling, such as heat pipes [1], spraying [2,3], jet impingement [4], and microchannels [5], it is still attracting researchers' attention to improve the existing cooling techniques and to develop new techniques for the demand of compact high-heat-flux and high-efficiency heat exchangers [6].

Comparing with other cooling techniques, microchannels can be directly embedded closely to heat sources for compact and efficient designs. Heat transfer in microchannels with multiphase flow can benefit from the recirculating flow induced by fluid interfaces [7], which is difficult to achieve in its single phase counterpart due to the low flow speeds and the small Reynolds numbers in microdevices. The recirculating flow in multiphase microfluidics can effectively enhance heat transfer by bringing fresh fluid from the center of the channel to the wall, and transporting heated fluid from the wall to the center of the channel [8]. Many experiments have been performed to measure the heat transfer enhancement in microchannels with droplets, slugs, or plugs [9]. Some experiments used cylindrical microtubes [10–13], while others used rectangular microchannels [14–17]. Their experimental results showed significant enhancement of heat transfer when comparing with single phase flow, and the Nusselt numbers are several times higher than the single phase counterpart.

Numerical simulations can serve as a complimentary way to study convective heat transfer in multiphase microchannels without building complex diagnostic systems, and to provide details of the process which are difficult or impossible to measure directly in experiments, and to allow a clear understanding of the physics. To address the effect of interfaces on heat transfer process, some simulations employed assumed interface shapes for the sake of simplicity [8,18], while others used interface capturing methods to accurately predict interface shapes. The latter category can consider the effect of the complex interface shapes on heat transfer process. Various interface tracking/capturing techniques have been employed to simulate the heat transfer in droplets, slugs, and plugs, such as the volume of fluid method [19], the level set method [20,21], and the phase field method [22]. Regarding the geometries of microchannels, most simulations were performed with cylindrical microcapillaries [18,20,22,21,23,19] or two dimensional (2D) microchannels [8], where the three dimensional (3D) effect on the flow and on the heat transfer process cannot be

---

* Corresponding author.
E-mail address: chez0001@e.ntu.edu.sg (Z. Che).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.03.030
0017-9310/

considered. However, in most microchannel heat exchangers, the microchannels usually have rectangular or other non-circular cross sections [14–17]. When droplets move in microchannels, liquid films often form beneath the droplets. For non-circular microchannels, 'gutters' form in the continuous phase at the corner of the cross sections [24], where 'gutters' refer to the space between the curved interface of the droplet and the corner of the microchannel, as shown in Fig. 1. The thickness of the liquid film and the shape of the liquid 'gutters' depend on the cross-sectional geometry, the liquid properties, and the flow conditions [25–27], and can significantly affect the flow dynamics and the heat transfer. This effect, to the best of our knowledge, has not been studied, and will be the focus of this investigation.

In this paper, we performed 3D numerical simulations on the convective heat transfer in droplet flows in the microchannels with rectangular cross sections, as shown in Fig. 1. The numerical methods are described in Section 2, including the level set method for interface capturing and the finite volume method for fluid flow and for heat transfer. The results are presented and discussed in Section 3, including the droplet dynamics at different flow conditions, the process of the heat transfer, and the effects of the aspect ratio of the channel cross section, the size of the droplet, and the Peclet number. Finally, concluding remarks are drawn in Section 4.

2. Numerical methods

Simulations of droplet-based heat transfer in microchannels involve the flow field of the fluids, the evolution of the liquid–liquid interface, and the heat transfer process. 3D simulation of these physical processes requires long computation time. To reduce the computation time without sacrificing the resolution with a coarse mesh, we used a frame of reference following each droplet. To do this, the flow fields and the interface shapes of droplets in microchannels are first obtained before implementing them in the heat transfer simulation. This method can significantly reduce the simulation time, and makes it possible to simulate the 3D droplet-based heat transfer in microchannel heat sinks, allowing the systematic parametric investigation of the related effects.

The finite volume method (FVM) and the level set method (LSM) are employed to simulate the heat transfer process of droplets moving in microchannels. The FVM for fluid flow and LSM for interface prediction are described in Ref. [28] and are explained briefly here.

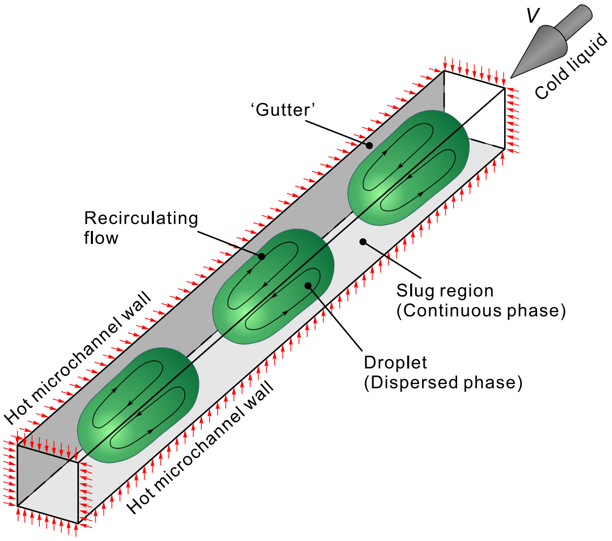

Fig. 1. Schematic diagram of droplet heat transfer in microchannels.

2.1. Finite volume method for flow field

The finite volume method [29] is used to discretize the continuity equation and the momentum equation,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \tag{1}$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \left[\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)\right], \tag{2}$$

where ρ and μ are the density and the viscosity of the fluid, respectively. For multiphase flow, fluid properties ρ and μ in each control volume are calculated using a smoothed Heaviside function H as follows:

$$\rho = H\rho_d + (1 - H)\rho_c, \tag{3}$$

$$\frac{1}{\mu} = \frac{H}{\mu_d} + \frac{1 - H}{\mu_c}, \tag{4}$$

where the subscripts 'd' and 'c' refer to the dispersed and continuous phases, respectively. The exact form of H will be presented in Section 2.2. The ghost fluid method [30,31] is used to take account of the surface tension through the pressure term,

$$[p] = -\kappa\sigma, \tag{5}$$

where $[p]$ indicates the pressure jump across the interface and $\kappa \equiv \nabla \cdot \mathbf{n}$ is the curvature of the interface. The symbol $\mathbf{n}$ denotes the unit direction vector normal to the interface, which will be presented in Eq. (9). No-slip boundary conditions are imposed on the walls. The inlet and outlet are set to be periodic to consider monodisperse droplets in microchannels. Since the frame of reference is following the droplets, the flow can gradually achieve a steady state. Therefore, instead of the whole microchannel, only a period of the flow is used in the simulations, which can significantly reduce the computation time.

In the simulations, the width of the microchannel is set to be 200 μm, the depth is varied from 200 to 800 μm, and the length of the simulation domain is 800 μm, which is used to represent a period of the flow. The fluid properties are set based on the system of water droplets in mineral oil with surfactant Span 80 at a concentration of 2% by weight, which is widely used in droplet-based microfluidic experiments to facilitate the formation of monodisperse droplets and to stabilize the droplets against coalescence [32,33]. The high surfactant concentration can guarantee a complete coverage of the interface by surfactant molecules, and the low speed of droplets can allow rapid replenishment of surfactant molecules from the bulk. Therefore, the surfactant distribution and the interfacial tension can be regarded as uniform. The dynamic viscosities and the densities are 2.39×10^{-2} Pa s and $840\ \mathrm{kg/m^3}$ for the continuous phase, and 8.9×10^{-4} Pa s and $997\ \mathrm{kg/m^3}$ for the dispersed phase. The interfacial tension between the continuous phase and the dispersed phase is 3.65 mN/m.

2.2. Level set method for interface capturing

The evolution of the droplet interface is captured using the level set method [34]. The level set function ϕ is a signed distance from the interface. The level set equation and the re-initialization equation for the distance function ϕ are, respectively,

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = 0, \tag{6}$$

$$\frac{\partial \phi}{\partial \tau} = \mathrm{sign}(\phi)(1 - |\nabla \phi|), \tag{7}$$