International Communications in Heat and Mass Transfer 86 (2017) 215–221

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Numerical investigation of subcooled flow boiling in segmented finned microchannels

Yogesh K. Prajapati[a], Manabendra Pathak[b,*], Mohd. Kaleem Khan[b]

[a] National Institute of Technology Uttarakhand, Srinagar 246174, Uttarakhand, India
[b] Indian Institute of Technology Patna, Bihta, 801103, Bihar, India

ARTICLE INFO

Keywords:
Subcooled flow boiling
Nucleation
Bubble growth
Segmented finned microchannel
Volume of fluid method

ABSTRACT

Bubble growth behavior and heat transfer characteristics during subcooled flow boiling in segmented finned microchannels have been numerically investigated. Simulations have been performed for a single row of segmented finned microchannel and predicted results are compared with experimental investigations. Onset of nucleation, formation of bubbles, their growth and movements have been investigated for different values of applied heat flux. Mechanism of bubble expansion without clogging resulting in enhanced heat transfer in segmented finned microchannels has been explained. Temperature and pressure fluctuations during subcooled flow boiling condition have been investigated. It is observed that at high heat flux, thin liquid film trapped between the bubble and channel wall is evaporated leading to localized heating effect. Predicted flow patterns are similar to experimental results. However, simulations over predict the bubble growth rate and heat transfer coefficient.

1. Introduction

Thermal management of high power density integrated chips has emerged as one of the challenges for electronic industries. Among different cooling techniques, microchannels possess high prospect as heat sinks for efficient heat dissipation from electronic devices. Due to this reason extensive investigations on heat transfer performance of microchannels have been reported for wide ranges of operating parameters and geometric configurations of channels. It has been observed that compared to single phase coolant flow, two-phase flows such as flow boiling involving latent heat, facilitate more heat transfer rate from the channels. However the major problem of flow boiling in microchannels is bubble clogging in the channel passage due to explosive growth of bubble which obstructs the fresh coolant and affect the heat transfer performance. Several efforts have been made to enhance the heat transfer performance during flow boiling in microchannels. Various techniques have been suggested for improving heat transfer performance of the microchannels. Kosar et al. [1]introduced reentrant cavities in the sidewalls of microchannel to enhance the boiling heat transfer. However, no significant improvement was reported. Khanikar et al. [2] used carbon nanotube coating at the walls of rectangular microchannels to enhance heat transfer. They observed that repeated experiments at high coolant mass flux causes the bending of nanotubes which favors the increased heat transfer in the nucleate boiling region. However, critical heat flux (CHF) degraded due to change in morphology of coated surface. Morshed et al. [3] applied the coating of copper nanowires at the surface of microchannels. Coated surface was able to increase heat transfer approximately up to 25% and 56% during single and flow boiling conditions respectively.

Lee and Mudawar [4] observed higher heat transfer rate using nanofluids typically during single phase flow of coolant. However, in flow boiling effect of nanofluids was not appreciable. Among different techniques, geometrical modification of microchannels shows enough scopes for augmentation of heat transfer and reduced instability in flow boiling.

Recently one type of geometrical modification in form of segmented or oblique finned microchannels has been found to perform better than uniform and diverging cross-section microchannels. It has been observed that segmented channel configuration considerably enhances heat transfer rate in single as well as flow boiling conditions. Lee et al. [5] investigated the single phase coolant flow in segmented finned microchannels and reported that high heat flux can be dissipated with a slight penalty of pressure drop [6,7].Recently, Prajapati et al. [8–10] have compared the boiling heat transfer performance in three different microchannel configurations and observed the better performance of segmented finned microchannels compared to uniform and diverging cross-section channels. In addition, it suppresses instabilities and flow reversal during flow boiling. Fan et al. [11] numerically investigated

* Corresponding author.
E-mail address: mpathak@iitp.ac.in (M. Pathak).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2017.06.009

the flow behavior and heat transfer performance during single phase cooling in the segmented channel. It is to be mentioned here that numerical investigation of flow boiling in segmented channel has not been reported so far.

For effective heat dissipation from microchannels, water as coolant possesses several advantages compared to refrigerant as coolant. Water possesses more latent heat compared to most of the refrigerants. Further a refrigerant cycle requires higher working pressure compared to water. Saturation temperature of water at normal condition is 100 °C which is higher than the safe working temperature 85 °C of electronics components. Thus saturated flow boiling may not occur in the microchannels used as heat sinks for electronic cooling applications. Hence subcooled flow boiling in microchannels is more relevant to electronic cooling applications. The objective of the present investigation is to develop a numerical model for analyzing subcooled flow boiling in segmented finned microchannels. Further present numerical results are compared with our own experimental results [8].

2. Numerical simulation

In present work numerical investigation has been performed for a 2D geometry of a single row of segmented finned microchannelas shown in Fig. 1. In segmented finned channels, besides primary channels along the coolant flow direction, secondary channels are cut at an angle to the primary channels. In present work, secondary channels make an angle of 30° with the primary channels. Due to cutting of both primary and secondary channels, rhomboid elements are formed as shown in Fig. 1.

The width of primary and secondary channels is fixed as 400 μm and secondary channels are made after 1 mm gap. Due to making of primary and secondary channels, rhomboid elements of 1 mm length are formed. In numerical simulation, two primary channels and connected secondary channels are included in computational domain. Simulations have been performed for operating conditions similar to experimental conditions which are discussed in next section.

2.1. Governing equations

Subcooled flow boiling in segmented finned microchannels has been numerically investigated. Liquid coolant has been considered as primary phase and vapor as secondary phase. The formation of vapor bubbles due to evaporation of liquid coolant has been modeled by a source term. The governing equations based on mass (continuity), momentum and energy conservation for two-phase unsteady flow are as follows:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho V) = 0 \tag{1}$$

Momentum equation:

$$\rho\left(\frac{\partial V}{\partial t} + V.\nabla V\right) = -\nabla p + \nabla.[\mu(\nabla V + \nabla V^T)] + S \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho E) + \nabla.[(\rho E + p)V] = \nabla.(k_{eff}\nabla T) + S_h \tag{3}$$

Variable ρ in above equations represents volume averaged density; V is velocity, p is pressure; μ represents the viscosity, and E is enthalpy. Source terms S and S_h are used to model the surface tension force and change in enthalpy in momentum and energy equation respectively. T in above equation represents the temperature and k_{eff} is effective thermal conductivity. Volume of fluid method (VOF) is used to track the secondary phase. In this method the distribution of void fraction α is solved from its transport equation:

$$\frac{\partial \alpha}{\partial t} + V.\nabla\alpha = \frac{S_h}{\rho_2} \tag{4}$$

The density and viscosity are defined as:

$$\rho = \rho_1 + \alpha(\rho_2 - \rho_1) \tag{5}$$

$$\mu = \mu_1 + \alpha(\mu_2 - \mu_1) \tag{6}$$

In above subscript 1 and 2 represent the liquid and vapor phase respectively. Similarly enthalpy term E in energy equation (Eq. (3)) has been calculated based on the mass averaged value of both the phases as follows:

$$E = \frac{\sum_{i=1}^{i=2}\alpha_i\rho_i E_i}{\sum_{i=1}^{2}\alpha_i\rho_i} \tag{7}$$

Source term (S) in momentum represents the surface tension force and it has been calculated by using continuous surface force (CSF) model proposed by Brackbill et al. [12]. It is expressed in terms of interface curvature (κ) as:

$$S = -\sigma\kappa\nabla\alpha \tag{8}$$

Curvature of the bubble interface has been calculated by:

$$\kappa = \nabla.\frac{\nabla\alpha}{|\nabla\alpha|} \tag{9}$$

Source term S_h in energy equation takes care of the changes in energy during phase transformation process. This source term is expressed as the product of latent heat of vaporization and rate of mass transfer.

$$S_h = \dot{m}h_{fg} \tag{10}$$

In above $\dot{m}$ is the rate of mass transfer during the phase change process which depends on the saturation temperature of the coolant. It is defined as:

$$\dot{m} = c(1-\alpha)\rho_1\frac{(T - T_{sat})}{T_{sat}} \text{ when } T \geq T_{sat} \tag{11}$$

$$\dot{m} = c\alpha\rho_2\frac{(T_{sat} - T)}{T_{sat}} \text{ when } T_{sat} \geq T \tag{12}$$

In above T_{sat} is the saturation temperature. The coefficient c is a relaxation parameter responsible for correct trend of mass transfer phenomenon.

2.2. Boundary conditions and numerical methodology

Velocity inlet and pressure outlet boundary conditions have been applied at inlet and outlet of the channel respectively. Water in liquid state enters into the channel. Hence volume fraction of the vapor phase has been put as zero ($\alpha = 0$) at the channel inlet. No-slip condition has been applied at all the channel walls. Uniform heat flux condition has been applied in the simulation. Pressure based solver has been used where both phases have been considered as incompressible. The second order upwind scheme is used to discretize momentum and energy

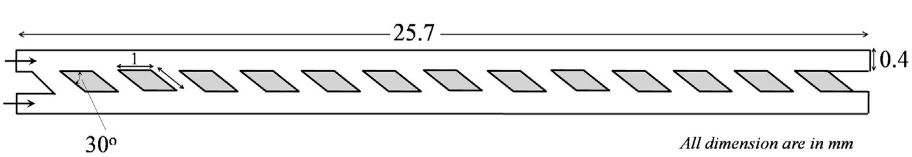

Fig. 1. Schematic diagram of computational domain.