International Journal of Heat and Mass Transfer 53 (2010) 1341–1348

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Modeling and simulation on the mass flow distribution in microchannel heat sinks with non-uniform heat flux conditions

Eun Seok Cho [a], Jong Won Choi [b], Jae Sung Yoon [c], Min Soo Kim [b,*]

[a] Package Development, System LSI division, Samsung Electronics Co. Ltd., Gyunggi-do 446-711, Republic of Korea
[b] School of Mechanical and Aerospace Engineering, Seoul National University, Seoul 151-744, Republic of Korea
[c] Nano Mechanical Systems Research Division, Korea Institute of Machinery and Materials (KIMM), Daejeon 305-343, Republic of Korea

ARTICLE INFO

Article history:
Received 25 March 2009
Received in revised form 19 August 2009
Accepted 28 October 2009
Available online 4 January 2010

Keywords:
Microchannel heat sinks
Evaporation
Two-phase flow
Mass flow distribution
Optimal header design
Non-uniform heat flux

ABSTRACT

This paper describes modeling and numerical simulation on the mass flow distribution in microchannel heat sink, which is a promising device for cooling miniature electronic systems. The microchannel heat sinks in this study consist of headers, multiple fluidic channels and port holes, all of which influence flow distribution in the multiple channels. This study focuses on design of the header with non-uniform heating conditions over the channel area. To investigate the effect of non-uniform heat flux, three different non-uniform heat flux conditions were applied. The simulation work has been carried out to find optimal header geometry for two-phase flow in the microchannel heat sinks. The header geometry was expressed in mathematical terms by defining a geometric parameter of header shape, $\boldsymbol{n}$. For the optimal design of microchannel heat sinks, absolute average deviation and root mean squared deviation of the flow distribution under various header shapes have been calculated as well as pressure drop. The results show that mass flow rate distribution tends to be less changed among microchannels over a certain value of $\boldsymbol{n}$.
© 2009 Elsevier Ltd. All rights reserved.

1. Introduction

When a lot of electro-mechanical systems have been miniaturized and integrated by compact design, thermal management in a small volume should be simultaneously considered. As the devices or systems become smaller, heat flux increases in general. Therefore, an effective cooling strategy for the micro-devices is required especially when the cooling target is made from microfabrication processes with silicon substrates. The microchannel heat sink is one of the most promising devices for cooling down the miniature systems because it can be also made by the microfabrication processes.

Recently, a large number of studies have been carried out to understand the mechanism of the heat transfer and the pressure drop of single-phase flow in the microchannel heat sinks. Kreutz et al. [1] investigated the pressure drop in the water-cooled heat sinks experimentally and theoretically. Fedorov and Viskanta [2] developed three-dimensional model in the microchannel heat sinks using incompressible laminar Navier–Stokes equations. Qu and Mudawar [3] analyzed heat transfer characteristics in a rectangular microchannel heat sink using water as cooling fluid numerically. Zhao and Lu [4] analyzed heat transfer characteristics of forced convection across a microchannel heat sink.

However, most of studies on the microchannel heat sinks were focused on the unit channel system. Numerical investigations on the single-phase flow have generally shown consistent results, because most of the physical phenomena of single-phase flow can be expressed well by the Navier–Stokes equation. However, it is difficult to predict and characterize the pressure drop of the two-phase flow because the flow behavior is too complicated to be expressed in mathematical forms. Therefore, most of the numerical investigations on the two-phase flow in microchannel heat sinks have presented new models to characterize the two-phase flow behavior. A simple one-dimensional model of boiling two-phase flow and heat transfer in a single triangular microchannel has been proposed by Peles and Haber [5]. It has shown that the dry-out length increases as the hydraulic diameter increases, and decreases as the heat flux increases. After their work, three-zone flow boiling model has been introduced to describe evaporation of elongated bubbles in microchannels [6]. This model shows that the heat transfer depends on the bubble generation frequency and liquid film thickness. In multiple channel heat sinks, the mass flow distribution should be considered to analyze the total pressure drop and overall heat transfer. A microchannel heat sink with manifold has been analyzed numerically considering a flow divider in front of microchannel [7]. It has been also investigated that the flow uniformity among branched microchannels depends largely on the geometry of manifolds and the inlet flow rate [8]. An optimum header design has been studied in the microchannels of single-phase flow by varying geometrical

* Corresponding author. Tel.: +82 2 880 8362; fax: +82 2 883 0179.
E-mail address: minskim@snu.ac.kr (M.S. Kim).

0017-9310/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2009.12.025

Nomenclature

A	cross-sectional area (m^2)
C	Chisholm's C coefficient
D_h	hydraulic diameter (m)
h	convection heat transfer coefficient (W/m^2 K)
L_{ch}	height of channel (m)
L_c	length of channel (m)
L_e	width of channel (m)
$\dot{m}_l$	mass flow rate in the inlet header (kg/s)
$\bar{\dot{m}}_l$	mass flow rate in the channel (kg/s)
$\dot{m}'_{N_c+1-i}$	mass flow rate in the outlet header (kg/s)
n	geometric parameter of header shape
N_c	number of channels
Δp_i	pressure drop of the inlet header (kPa)
$\Delta \bar{p}_i$	pressure drop of the channel (kPa)
$\Delta p'_{N_c+1-i}$	pressure drop of the outlet header (kPa)
q''	heat flux (W/m^2)
u_m	mean velocity (m/s)
W_i	height of inlet header (m)
W'_{N_c+1-i}	height of outlet header (m)
X	Martinelli parameter, $\left\{= ((1-x)^{0.9}/x)^{0.9}(\rho_g/\rho_l)^{0.5}(\mu_l/\mu_g)^{0.1}\right\}$
$\boldsymbol{x}$	quality
x	flowing direction in the header
z	flowing direction in the channel
α	void fraction
μ	viscosity coefficient (Pa s)
ρ	density (kg/m^3)

Superscripts

$'$	inlet header
$^-$	in the channel

Subscripts

amb	ambient
ch	channel
g	gas phase
i	ith channel number
l	liquid phase
ref	refrigerant
s	solid
tp	two phase

dimensions of header [9]. Even though there have been several numerical studies on the mass flow distribution in multiple channels, most of them focused on the single-phase flow. However, the understanding of two-phase flow characteristics in the multiple microchannels is essential for the design of heat sinks. While there have been simulation works to investigate the mal-distribution of the single-phase flow in the multi-channel in previous studies, there have been little studies to calculate or predict the mal-distribution of the two-phase flow for different header shapes and heating conditions. It is very important to predict the mal-distribution of the two-phase flow for designing and deciding the microchannel geometry and the cooling performance in the system using the refrigerant because the cooling capacity is dependent on the pressure drop and mass flow rate. In this study, the one-dimensional simulation work has been carried out for finding the optimum header shape in the microchannel by dividing into three zones.

In the integrated circuits, heat flux (heat generation per unit area) is not uniform in the heat transfer area. Hotspots are usually concentrated in the logic circuit part which operated very actively in the integrated circuits. Hotspots have a great influence on the mass flow distribution and pressure drop in the microchannel heat sinks. Therefore, non-uniform heat flux conditions in the integrated circuits should be taken into consideration as an important design parameter for the microchannel heat sinks. In this study, numerical simulation has been performed to obtain the mass flow distribution and total pressure drop in microchannel heat sinks with non-uniform heat flux condition. This simulation investigates the impact of geometrical parameters on the mass flow distribution in multi-channel heat sinks considering a geometric parameter of header shape, $\boldsymbol{n}$.

2. Modeling and simulation

2.1. Geometric parameter of header

Fig. 1 shows a schematic diagram of the microchannel heat sink for the numerical simulation. The heat sink is divided into three zones: inlet zone, microchannels, and outlet zone. For the development of the simplified two-phase distribution model, the approximate model [10] was modified for the single-phase flow in the inlet header zone. Chisholm's C coefficient method and the homogeneous model were used for the two-phase flow in the microchannel zone and outlet header zone. Table 1 shows the values of the geometric constraints of microchannel heat sink for the simulation work. Total width of microchannel heat sink W_c is fixed as 20 mm and L_c, the total length of microchannel heat sink, is also fixed as 20 mm. The depths of the header and the channels are 300 μm. There should be a geometric constraint for the shape of the microchannel header since there can be an infinite number of inlet header geometries for the uniform mass flow distribution. The shape of the inlet header can be described as the following equation:

$$W_i = W_{N_c} + (W_1 - W_{N_c})\left[1 - \left(\frac{i-1}{N_c-1}\right)^n\right]^{\frac{1}{n}} \tag{1}$$

In given values of W_1 and W_{N_c}, the inlet header shape profile can be obtained by varying the geometric parameter of header shape, $\boldsymbol{n}$. Fig. 2 shows the variation of inlet header shape with respect $\boldsymbol{n}$. In case of $\boldsymbol{n} = 1$, linear header shape is obtained. Convex header shapes can be obtained if $\boldsymbol{n} > 1$, and concave header shapes can be obtained if $\boldsymbol{n} < 1$. To make the simulations simple, the geometry of outlet header is assumed to be identical with that of the inlet header. Since the inlet and outlet zones are located diagonally, the shape of outlet zone is determined as follows:

$$W'_{N_c+1-i} = W_i \tag{2}$$

where prime denotes the outlet zone.

In the simulation, inlet boundary conditions, geometries of microchannel heat sinks, and total mass flow rate are given. Mass qualities at microchannel zone and outlet zone are calculated with the assumed mass flow rate. Then mass flow distribution of the microchannel heat sink is obtained by Tri-Diagonal Matrix Algorithm (TDMA) with the momentum equilibrium condition. The calculation procedure requires iteration until the mass flow rate for each microchannel is converged to meet the constraints, and then the mass flow rates of each channel are obtained. Fig. 3 shows calculation scheme for the modeling and simulation. R-123 has been chosen as the working fluid.

Download English Version:

https://daneshyari.com/en/article/661118

Download Persian Version:

https://daneshyari.com/article/661118

Daneshyari.com