International Communications in Heat and Mass Transfer 38 (2011) 577–584

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Numerical simulation of fluid flow and heat transfer in a microchannel heat sink with offset fan-shaped reentrant cavities in sidewall☆

Lei Chai, Guodong Xia *, Mingzheng Zhou, Jian Li

Key Laboratory of Enhanced Heat Transfer and Energy Conservation, Ministry of Education, College of Environmental and Energy Engineering, Beijing University of Technology, Beijing 100124, China

ARTICLE INFO

Available online 8 January 2011

Keywords:
Offset fan-shaped reentrant cavities
Microchannel heat sink
Numerical simulation

ABSTRACT

The paper is focused on the investigation of fluid flow and heat transfer characteristics in a microchannel heat sink with offset fan-shaped reentrant cavities in sidewall. In contrast to the new microchannel heat sink, the corresponding conventional rectangular microchannel heat sink is chosen. The computational fluid dynamics is used to simulate the flow and heat transfer in the heat sinks. The steady, laminar flow and heat transfer equations are solved in a finite-volume method. The SIMPLEX method is used for the computations. The effects of flow rate and heat flux on pressure drop and heat transfer are presented. The results indicate that the microchannel heat sink with offset fan-shaped reentrant cavities in sidewall improved heat transfer performance with an acceptable pressure drop. The fluid flow and heat transfer mechanism of the new microchannel heat sink can attribute to the interaction of the increased heat transfer surface area, the redeveloping of the hydraulic and thermal boundary layers, the jet and throttling effects and the slipping over the reentrant cavities. The increased heat transfer surface area and the periodic thermal developing flow are responsible for the significant heat transfer enhancement. The jet and throttling effects enhance heat transfer, simultaneously increasing pressure drop. The slipping over the reentrant cavities reduces pressure drop, but drastically decreases heat transfer.

1. Introduction

With the rapid development of very large-scale integration technology (VLSI) and Micro-Electro Mechanical Systems (MEMS), the application of microchannel heat sinks is drawing more attention as the most compact and efficient method of transferring heat from a power source to a fluid. These microchannel heat sinks, as one of the basic devices in microfluidic system, can be broadly applied to the cooling electronic devices, automotive heat exchangers, laser process equipments, and aerospace technology. Heat transfer coefficients in microchannels are very high due to their small hydraulic diameters although it is accompanied by a higher pressure drop per unit length. The high pressure gradients have led researchers to employ low flow rates. However, with reduced flow rates, the ability of fluid stream to carry heat away for a given temperature rise becomes limited. In order to improve the overall cooling performance, the following two options should be considered: reduce the flow length of the channels, increase the liquid flow rate [1]. Downscaling of MEMS devices and advances in microfabrication processes have helped to work out various microchannel heat sinks to meet the increasing demand for higher dissipation of heat flux. The microchannel heat sink with reentrant cavities in sidewall emerges with the time requirement.

Tuckerman and Pease [2] first proposed the concept of microchannel heat sink in 1981. After that, intensive investigations were performed experimentally and numerically in the following decades. Xia et al. [3], Peng et al. [4], Harms et al. [5] and Adams et al. [6] did lots of experiments to investigate fluid flow and heat transfer in microchannels. Qu and Mudawar [7,8] used the finite difference method and the SIMPLE algorithm to solve the conventional Navier–Stokes and energy equations. Their numerical results showed good agreement with the corresponding experimental data. Gamrat et al. [9] performed both two- and three-dimensional numerical simulation to analyze the fluence of thermal entrance effects on microchannel convection. They concluded that the continuum model of conventional mass, Navier–Stokes and energy equations were adequately accurate to numerically simulate the fluid flow and heat transfer for their microchannels. Koo and Kleinstreuer [10] studied the effects of viscous dissipation on the temperature field and ultimately on the friction factor using dimensional analysis and experimentally validated computer simulations. They demonstrated that ignoring viscous dissipation could affect accurate flow simulations and measurements in microconduits. Liu et al. [11] presented a theoretical modeling with effects of viscosity and thermal conductivity variations on characteristics of fluid flow and heat transfer to conduct numerical investigation. They confirmed that the heat transfer enhancement due to

☆ Communicated by D. Cheng and W.Q. Tao.

* Corresponding author.
E-mail address: xgd@bjut.edu.cn (G. Xia).

0735-1933/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2010.12.037