International Journal of Thermal Sciences 50 (2011) 411–419

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Effects of structural parameters on fluid flow and heat transfer in a microchannel with aligned fan-shaped reentrant cavities

Guodong Xia*, Lei Chai, Mingzheng Zhou, Haiyan Wang

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

A R T I C L E I N F O

Article history:
Received 20 January 2010
Received in revised form
16 August 2010
Accepted 17 August 2010
Available online 29 September 2010

Keywords:
Aligned fan-shaped reentrant cavities
Structural parameters
Numerical simulation

A B S T R A C T

We provide three-dimensional numerical simulations of conjugate heat transfer in the newly proposed microchannels with different structural parameters. The structural parameters include the lengths and widths of the constant cross-section region and the arcuate region. The effects of structural parameters on pressure drop and thermal resistance are presented. For the first part of the effect analysis, we study the fluid flow and heat transfer mechanism of the microchannel with aligned fan-shaped reentrant cavities in detail, which can attribute to the interaction of the increased heat transfer surface area, the redeveloping boundary layers, the jet and throttling effect and the slipping over the reentrant cavities. As to the second part of the present analysis, the effects of two design structural parameters on fluid flow and heat transfer of the new microchannel are individually prescribed and the two suitable ranges of structural parameter are found for the optimum geometric configuration.

1. Introduction

With the rapid development of modern industry, the high performance thermal systems have stimulated research interest in augmentation of heat transfer. The mechanism of conventional heat transfer enhancement techniques can be attributed to the increase of the heat transfer area and/or convective heat transfer coefficient. One of the following ways will result in the rise of the heat transfer coefficient: (a) mixing the main flow and/or the flow in the near-wall region; (b) reducing the flow boundary layer thickness; (c) creating the rotating and/or the secondary flow; (d) raising the turbulence intensity. However, the existing heat transfer enhancement techniques result in the large additional pressure drop associated with heat transfer enhancement, to make which have no or less practical applications [1]. Downscaling of MEMS devices and advances in micro-fabrication processes are helpful to work out various microchannel heat sinks to meet the growing demand for higher dissipation of heat flux. With requirements of the times, the microchannel heat sink with reentrant cavities in sidewall emerges, which can improve heat transfer performance with an acceptable pressure drop. By minimizing the size of the slow-moving fluid boundary layer and increasing the area of contact between the heat sink fins and the fluid, the microchannel heat sink can remove heat more efficiently than conventional methods [2]. Due to the design of fan-shaped reentrant cavities, the newly proposed microchannel can provide increased heat transfer surface area, redevelop boundary layer and result in the jet and throttling effects. Hence, the microchannel heat sink with aligned fan-shaped reentrant cavities can acquire better heat transfer performance. However, at the same time, the slipping over the reentrant cavities impedes heat transfer seriously. Therefore, the structural parameters of the new microchannel heat sink should be optimized. Among the structural parameters, the lengths and widths of the constant cross-section region and the arcuate region play the most important role.

The pressure drop and heat transfer characteristics of a single-phase microchannel heat sink were investigated both experimentally and numerically by Qu and Mudawar [3,4]. They used the finite difference method and the SIMPLE algorithm to solve the conventional Navier–Stokes and energy equations. The measured pressure drop and temperature distributions showed good agreement with the corresponding numerical predictions. The temperature rise along the flow direction in the solid and fluid regions could be approximated as linear. The highest temperature was encountered at the heated base surface of the heat sink immediately above the channel outlet. The heat flux and local Nusselt number had much higher values near the channel inlet and varied around the channel periphery, approaching zero in the corners. These findings demonstrated that the conventional Navier–Stokes and energy equations could adequately predict the fluid flow and heat transfer

* Corresponding author. Tel.: +86 1067392176; fax: +86 1067391983.
E-mail address: xgd@bjut.edu.cn (G. Xia).

1290-0729/$ – see front matter © 2010 Elsevier Masson SAS. All rights reserved.
doi:10.1016/j.ijthermalsci.2010.08.009