International Journal of Heat and Mass Transfer 108 (2017) 1197–1206

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical study on heat transfer and pressure drop in laminar-flow multistage mini-channel heat sink

Yeongseok Kim [a,b], Myungjoon Kim [a], Chisung Ahn [c], Hyeong U. Kim [c], Sang-Woo Kang [b], Taesung Kim [a,c,*]

[a] School of Mechanical Engineering, Sungkyunkwan University, 2066 Seobu-ro, Jangan-gu, Suwon, Gyeonggi 440-746, Republic of Korea
[b] Vacuum Center, Korea Research Institute of Standards and Science, 267 Gajeong-ro, Yuseong-gu, Daejeon 305-340, Republic of Korea
[c] SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, 2066 Seobu-ro, Jangan-gu, Suwon, Gyeonggi 440-746, Republic of Korea

ARTICLE INFO

Article history:
Received 10 February 2016
Received in revised form 7 December 2016
Accepted 10 December 2016

Keywords:
Multistage
Mini-channel
Heat transfer
Pressure drop
Numerical study

ABSTRACT

Mini-channel has been more studied recently than micro-channel to optimize the heat emission and pressure drop by regulating the channel size and length. In this work, a multistage mini-channel heat sink using water coolant was designed to obtain a larger cooling rate in a small area with a lower pressure drop. To confirm the performance of the structure, we conducted numerical simulations under laminar and single-phase conditions. The diameter and length of the channel were 2 and 530 mm, respectively. From the simulation, the local convection coefficient, coolant temperature, channel-wall temperature, effectiveness, and pressure drop were analyzed in relation to the mass flux, heat-source temperature, and number of stage stacks. To obtain valid simulation results on the heat transfer, we used well-matched conventional correlation. The result of the pressure drop was compared with the experimental result to confirm the validity of the hydrodynamic model. The simulation result shows that the maximum cooling rate was 40 W/cm^2 at a pressure drop of 1383 Pa in a quintuple-stage model. However, the triple-stage structure had the best effectiveness of 0.83 under the same simulation conditions. The pressure drop of the multistage structure was higher than that of the single-stage structure. However, the increase of the total pressure drop was small as against the increase of the cooling rate.

1. Introduction

With the advancement in nanotechnology, integrated circuits (ICs) have steadily become smaller to reduce electrical energy consumption and to achieve high calculation performance. In particular, the state-of-the-art IC has gradually become smaller toward 10 nm or less to achieve high density and high performance in large-scale integration (LSI). The downscaling of ICs suffers from various concerns in terms of physical limitation related on the leakage current, leading to heterogeneous three-dimensional (3D) integration from homogeneous two-dimensional (2D) integration [1]. Even though the heterogeneous 3D integration consumes lesser power than the 2D integration, LSIs still require effective cooling applications to discharge the generated heat from the surface of different chips such as semiconductor and photonic device chips.

Liquid cooling process using micro-/mini-channel is one of the efficient candidates for small-scale cooling applications owing to their high heat-transfer performance. As described in [2], Kandlikar et al. classified micro- and mini-channels in terms of the smallest channel dimension and suggested applications depending on the ranges of channel diameters. The dimension range of a micro-channel is within 10–200 μm, and that of a mini-channel is 200–3000 μm. The micro-/mini-channel was suggested by Tuckerman and Pease [3] in 1981 as a novel heat sink for high-power-consumption IC devices. They experimentally obtained heat dissipation of 790 W/cm^2 at a maximum temperature difference of 71 °C where no phase transition occurred between the substrate and input water temperature. Wang and Peng [4] also conducted similar experiments on heat transfer and flow behavior according to the Reynolds number (*Re*). According to their work, a fully developed turbulent regime started at *Re* of from 1000 to 1500. For further study, Peng and Peterson [5] conducted additional experiments on heat transfer and flow in relation to the hydraulic diameter of a micro-channel in which the investigation ranged between 133 and 367 μm. In particular, the effects on the heat transfer and pressure drop according to the shape and size of the micro-/mini-channel were investigated by Morini [6]. According to his work, various conditions such as coolant [7,8] and geometry [3–5,9,10] were used to establish a conventional theory for micro-/mini-channels. And study on various channel shapes was con-

* Corresponding author at: School of Mechanical Engineering, Sungkyunkwan University, 2066 Seobu-ro, Jangan-gu, Suwon, Gyeonggi 440-746, Republic of Korea.
E-mail address: tkim@skku.edu (T. Kim).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.12.025
0017-9310/

ducted by Khan and Kim [11], recently. These results were numerically verified by Dang and Teng [12]. They compared the hydrodynamics and heat-transfer behavior between micro- and mini-channels using numerical methods, which agreed well with the experimental data. Their work demonstrated that a micro-channel heat exchanger could be obtained with 1.2–1.53 times higher effectiveness (defined as the ratio of actual and maximum heat-transfer rates) than the mini-channel.

However, despite their high heat-transfer performance, micro-channels are inappropriate because of their huge pressure drop. Therefore, mini-channel has been more studied recently than the micro-channel. Wang et al. [13] demonstrated that the influence of friction characteristics can be neglected when the hydraulic diameter is larger than 1.0 mm, and their measurements agreed well with the conventional correlations. On the other hand, Reynaud et al. [14] measured the friction and heat-transfer coefficients in mini-channels with diameters of 0.3–1.12 mm. The experimental results on heat transfer agreed well with the classical correlations in a mini-channel. Because the heat-transfer behavior and hydrodynamics in a mini-channel agreed with the conventional correlations, numerical studies were conducted to design a small-size heat sink with simultaneous relatively high heat-transfer performance and low pressure drop. Xie et al. [15,16] conducted numerical simulation on laminar and turbulent mini-channel heat sink whose bottom size was 20 mm × 20 mm to analyze the effect of channel dimensions, channel-wall thickness, bottom thickness, and inlet velocity on the pressure drop, thermal resistance, and maximum allowable heat.

On the other hand, the design of the micro-/mini-channels in these works was similar to the 2D model suggested by Tuckerman and Pease. The geometric conditions of this model, such as the channel size and length, could control the heat transfer and pressure drop. To obtain high heat-transfer performance in a small cooling area, the model required a micro-scale channel size that generates a huge pressure drop because a longer channel should occupy a larger area. As a result, alternative designs were suggested by various research groups. Kandlikar and Upadhye [17] suggested split flow arrangement to extend the multiple inlets and outlets as originally recommended by Tuckerman and Pease [3]. This design could reduce the flow length of the fluid stream in the micro-channel by half, and the fluid flow rate through the micro-channels was also halved. This structure could increase the heat transfer coefficient near the multiple channel entrance regions due to the thermally developing flow. Colgan et al. [18] also studied the offset strip-fin arrangement in micro-channels. They obtained the high heat flux dissipation by applying the staggered fin arrangements, although this structure had the high pressure drop caused by high associated friction factors.

In the present work, we designed a long mini-channel that is stacked multiple times, as an alternative heat transfer concept, to maintain high heat-transfer performance without a large increase in the pressure drop. Subsequently, we performed 3D numerical simulation to determine its heat transfer and pressure drop characteristics to analyze the interaction between stages. In the following discussion, the outline of a multistage mini-channel heat sink is introduced, followed by its modeling to perform numerical simulation. Then, simulation results are presented relative to the heat-source temperature, coolant mass flux, and number of stacked stages. Finally, a brief conclusion is presented.

2. Modeling

2.1. Schematics

The concept of a multistage mini-channel heat sink is a 3D model that appears like a folded stacked 2D model (single stage structure), as shown in Fig. 1a. So the channel length of each stages $L_{c,n}$, which belongs to n_{th} stage mini-channel, decreases depending on n. However, this concept can maintain a long channel length L_c and decreases the target cooling area. The heat sink is heated by heat source, which is at the bottom and has uniform temperature. As shown in Fig. 1b, the target cooling area, which is same with the heat source area A_s, is the bottom surface area, and the height of the stage is designated as h. The mini-channel height and width are h_c and w_c, and the distance between channels is w. Similar to the 2D model, each mini-channel does not cross each other and has equal geometrical configuration such as the channel length and size. Water coolant with a constant temperature of 290 K is injected into the inlet located at the top stage, and the hot coolant, heated by internal convection from the channel surface, is ejected at the outlet from the bottom stage exhaust of an aluminum solid body heat sink. This configuration can reduce the temperature gradient at the bottom surface because the heat transfer from the channel wall to the coolant is reduced by the small temperature difference between the channel wall and coolant. To exclude the influence from the surrounding, such as convection and radiation, we assume that the surface of the heat sink is insulated except for A_s. Multistage mini-channel heat sink could be fabricated by assembling the caps and divider as shown in Fig. 1c. Caps are top and bottom units surrounding the divider to generate the insulated channels. Stacking the several dividers fabricates above the triple-stage mini-channel.

2.2. Design

To simulate the water coolant mini-channel heat sink, three types of physical processes were considered. First, the hydrodynamics in the water coolant needs to be defined to simulate the velocity profile and pressure drop in the channel. Then, the conduction in the heat-sink body must be considered. Maranzana et al. [19] demonstrated that the channel-wall temperature becomes largely non-uniform under small Re, which indicates a laminar-flow regime, because most of the heat flux is transferred to the coolant at the entrance of the micro-/mini-channel. Finally, convection between the coolant and channel wall is also a key factor in the modeling. In particular, the conventional correlation agrees well in the laminar-flow regime, as demonstrated in [5–12].

In this work, we designed a partial 3D structure to simulate the hydrodynamics and heat transfer behavior in the coolant and channel wall, as shown in Fig. 2b. This partial structure can be made available as a model that is expected to yield symmetric simulation results, especially in the middle of the heat sink. The geometrical conditions of the mini-channel multistage structure are listed in Table 1. To confirm the effect of the multistage on the heat transfer and pressure drop, we used a long channel length to minimize the local convection decrease and to obtain a sufficient hydrodynamically and thermally developed region which is favorable to perform the CFD on the internal flow, because the condition is robust for calculation. When the hydraulic diameter of the channel ($D_h = h_c \cdot w_c/2\ (h_c + w_c)$) is 2 mm, which is a large but typical diameter of a mini-channel. Generally, the channel length L_c is dozens of times of D_h to develop the flow fully in case of laminar internal flow. So, L_c was determined based on the minimum stage channel length $L_{c,5}$ (106 mm = 53 D_h) of the quintuple stage that had the shortest stage channel length to develop the flow. So, the L_c was defined as 530 mm (= 5 $L_{c,5}$).

In addition, we assumed that the coolant flow is incompressible and laminar to simplify the calculation model. The mean of the injected coolant and the heat-source temperature ($T_{c,m} = (T_{c,i} + T_s)/2$) was used to define the thermophysical properties such as viscosity and density, which are strongly dependent on the temperature. Heat-source temperature T_s of under 370 K was used to